NI AWR Design Environment

v13

Microwave Office Element Catalog

ni.com/awr

NATIONAL INSTRUMENTS™

Microwave Office Element Catalog

NI AWR Design Environment v13 Edition

1960 E. Grand Avenue, Suite 430
El Segundo, CA 90245
USA
Phone: +1 310.726.3000
Fax: +1 310.726.3005
Website: www.ni.com/awr
`awr.support@ni.com`
U.S. Technical Support phone: 888.349.7610

Table of Contents

Preface

The NI AWR Design Environment™ (NI AWRDE) suite incorporating Microwave Office and Analog Office software is a powerful fully-integrated design and analysis tool for RF, microwave, millimeterwave, analog, and RFIC design.

Microwave Office (MWO) and Analog Office (AO) allow you to create complex circuit designs composed of linear, nonlinear, and EM structures, and generate layout representations of these designs. It allows you to perform fast and accurate analysis of your designs using linear, nonlinear harmonic balance, nonlinear Volterra-series, electromagnetic (EM), and HSPICE simulation engines, and features real-time tuning and optimizing capabilities.

About This Book

The *Microwave Office Element Catalog* provides complete reference information on all of the electrical element models that are part of the Microwave Office and Analog Office database. This extensive database includes linear device models for transmission line systems such as microstrip, stripline, and coplanar waveguide. It also includes nonlinear device models, including such complex models as the BSIM3 MOSFET, PHEMT, Parket-Skellern FET, Fujii, and Triquint TOM3 models.

The element models in this catalog are organized alphabetically within categories such as "Coplanar" and "Lumped Element" (also organized alphabetically). This organization mirrors the hierarchy of the Element Browser in the NI AWRDE, which you use to select the models you want to use in your designs.

For each element model in this guide, the following attributes are described:

Attribute	Description
Symbol	The graphical representation of the element model as it is displayed in the schematic window.
Summary	A short summary of the model that may include its purpose or advantage over other models.
Topology	The physical parameters of the model on a drawing, if applicable.
Equivalent Circuit	An equivalent circuit schematic describing the components used in this model, if applicable.
Parameters, Parameter Details, and Parameter Restrictions and Recommendations	The user-modifiable parameters of the components (for example, capacitors and resistors) of this element model, along with their units, default values, and data types: **I**nteger, **R**eal, **C**omplex, **D**ata Model, **N**ame Element, **S**tring, **E**numeration, and **V**ector. Parameter details, restrictions on the range of acceptable parameter values (either required or recommended), and recommendations may be included if applicable.
Implementation Details	Notes on the implementation of this particular element model, if applicable.
Recommendations for Use	Recommendations, conditions, and cautionary notes (if applicable) for using this model
References	Notes on the origin of the techniques used in the implementation of this model, if applicable.

For a discussion on selecting appropriate models when building your designs, see the User Guide .

This guide assumes that you have a working knowledge of high-frequency electronic design, layout, and analysis.

Additional Documentation

The NI AWRDE includes the following additional documentation:

- *What's New in NI AWRDE 13?* presents the new features, user interface, elements, system blocks, and measurements for this release.
- The *Installation Guide* (available on your Program Disk as *install.pdf* or downloadable from the NI AWR website at www.ni.com/awr under Support) describes how to install the NI AWR Design Environment and configure it for locked or floating licensing options. It also provides licensing configuration troubleshooting tips.
- The *Getting Started Guide* familiarizes you with the NI AWRDE through Microwave Office, Visual System Simulator, Analog Office (AO) , Analyst™, and Monolithic Microwave Integrated Circuit (MMIC) examples.

  MWO example projects show how to design and analyze simple linear, nonlinear, and EM circuits, and how to create layouts. VSS examples show how to design systems and perform simulations using predefined or customized transmitters and receivers. AO examples show how to design circuits composed of schematics and electromagnetic (EM) structures from an extensive electrical model set, and then generate physical layouts of the designs. Analyst examples show how to create and simulate 3D EM structures from MWO, and MMIC examples show MMIC features and designs.

  You can perform simulations using a number of simulators, and then display the output in a wide variety of graphical forms based on your analysis needs. You can also tune or optimize the designs, and your changes are automatically and immediately reflected in the layout.
- The User Guide provides an overview of the NI AWRDE including chapters on the NI AWRDE user interface; using schematics/system diagrams; data files; netlists; graphs, measurements, and output files; and variables and equations in projects.
- The Simulation and Analysis Guide discusses simulation basics such as swept parameter analysis, tuning/optimizing/yield, and simulation filters; and provides simulation details for DC, linear, AC, harmonic balance, transient, and EM simulation/extraction theory and methods.
- The Dialog Box Reference provides a comprehensive reference of all NI AWRDE dialog boxes with dialog box graphics, overviews, option details, and information on how to navigate to each dialog box.
- The Microwave Office Layout Guide contains information on creating and viewing layouts for schematics and EM structures, including use of the Layout Manager, Layout Process File, artwork cell creation/editing/properties, Design Rule Checking, and other topics.
- The VSS System Block Catalog provides complete reference information on all of the system blocks that you use to build systems.
- The Microwave Office Measurement Catalog provides complete reference information on the "measurements" (computed data such as gain, noise, power, or voltage) that you can choose as output for your simulations.
- The VSS Measurement Catalog provides complete reference information on the measurements you can choose as output for your simulations.
- The VSS Modeling Guide contains information on simulation basics, RF modeling capabilities, and noise modeling.
- The API Scripting Guide explains the basic concepts of NI AWRDE scripting and provides coding examples. It also provides information on the most useful objects, properties, and methods for creating scripts in the NI AWR Script Development Environment (NI AWR SDE). In addition, this guide contains the NI AWRDE Component API list.
- The *Quick Reference* document lists keyboard shortcuts, mouse operations, and tips and tricks to optimize your use of the NI AWR Design Environment. This document is available within the NI AWRDE by choosing **Help > Quick Reference**.

- *NI AWR Design Environment Known Issues* lists the known issues for this release. This document is available on your program disk as *KnownIssues.htm*.

Typographical Conventions

This guide uses the following typographical conventions.

Item	Convention
Anything that you select (or click on) in the NI AWRDE, such as menus, submenus, menu items, dialog box options, button names, and tab names.	Shown in a bold alternate type. Nested menu selections are shown with a "**>**" to indicate that you select the first menu item and then select the second menu item from the menu: Choose **File > New Project**.
Text that you enter using the keyboard	Shown in bold type within quotes: Enter "**my_project**" in **Project Name**.
Keys or key combinations that you press	Shown in a bold alternate type with initial capitals. Key combinations using a "+" indicate that you press and hold the first key while pressing the second key: Press **Alt+F1**.
Filenames and directory paths	Shown in italics: See the *default.lpf* file.
Contents of a file, fields within a file, command names, command switches/arguments, or output from a command at the command prompt	Shown in an alternate type: Define this parameter in the `$DEFAULT_VALUES` field.

Getting On-Line Help

NI AWRDE online Help provides information on the windows, menus, and dialog boxes that compose the design environment, as well as on the concepts involved.

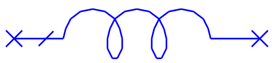
To access context-sensitive Help for any element in the Element Catalog, do one of the following:

- After creating a schematic, choose **Help > Contents and Index** or press the **F1** key and click the **Contents** tab in the left pane of the Help window. Double-click the **Microwave Office Element Catalog** folder to open the Elements Catalog Help.
- Right-click an element in the lower pane of the Element Browser and choose **Element Help**.
- Right-click any element in a schematic and choose **Element Help**.
- Double-click an element to open the Element Options dialog box, then click the **Element Help** button.

3D EM Elements

Air Coil With Centered Axial Leads: AIRCOIL1

Symbol

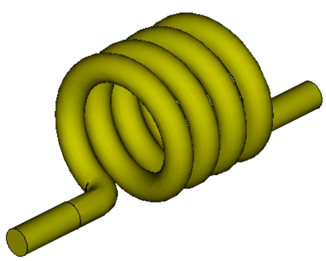


Summary

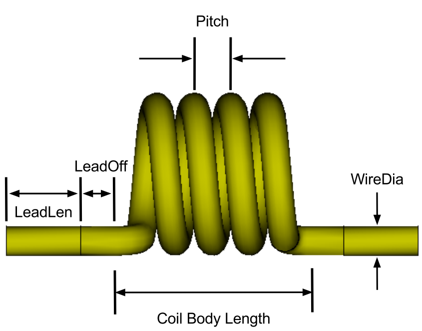
AIRCOIL1 models an air core coil inductor with centered axial leads. Coil and lead dimensions are configurable.

This element is for use in 3D EM documents only. See "Using 3D EM Elements " for details on using 3D pCells with Analyst.

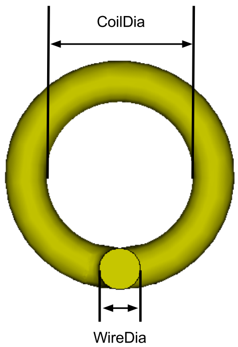
Topology



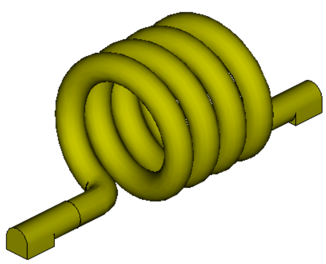
Round Post Lead



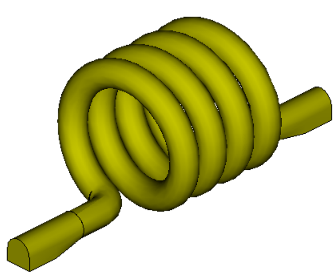
Side



Front



Bottom Flat Tab Lead
TabLenRatio = 0.5



Lofted Flat Tab Lead
TabLenRatio = 0.5

Parameters

Name	Description	Unit Type	Default
ID	Subcircuit ID	Text	S1
NET	Subcircuit name	Text	"AIRCOIL1"
*M	Multiplicity factor - not used for this model		1
NTurns	Number of turns		4
WireDia	Diameter of the wire	Length	1 mm
CoilDia	Coil inner diameter	Length	3.5 mm
Pitch	Distance between turns, measured from wire center-to-center	Length	1.01 mm
LeadLen	Length of lead	Length	1.5 mm
LeadOff	Offset distance between coil body and beginning of lead	Length	1.5 mm
LeadType	Type of lead contact: 0=round post, 1=bottom flat tab, 2=lofted flat tab		0
TabLenRatio	Ratio of tab length to total lead length for LeadType=1 or 2. 0<TabLenRatio<1		0.75
Rho	Bulk resistivity of conductor metal normalized to gold		1

**indicates a secondary parameter*

Parameter Details

Parameters for this model are unitless. Parameters do not scale when changing Global Units.

The coil body length is a function of the pitch and number of turns of the coils:

- Coil Body Length = 2·LeadTranRad + (Nturns - (2·LeadTranRadAngle/360 deg))·Pitch
- LeadTranRad = WireDia·0.75
- LeadTranRadAngle = 2·asin(LeadTranRad/(CoilDia + WireDia))

The lead geometry is configurable. The simplest lead, LeadType=0, is a round post. Next, LeadType=1 is a lead with a bottom flat tab. The length of the flat bottom section is (LeadLen·TabLenRatio), and the length of the round section is (LeadLen·(1-TabLenRatio)). Lastly, LeadType=2 is a lead with flat bottom that gradually transitions to a round cross-section. The length of the flat bottom section is (LeadLen·TabLenRatio), and the length of the lofted (transition) section is (LeadLen·(1-TabLenRatio)).

LeadOff. LeadOff is the offset distance between the coil body and lead. The cross-section of this region is always round. This length typically represents the length of the gap between body of the coil, and the end of the trace which the coil is attached to.

Rho. Rho is the bulk resistivity of conductor metal normalized to gold, where Rho_Gold=2.44 x 10^{-8} Ω*m. So actual bulk resistivity of conductor metal is (Rho·2.44 x 10^{-8})Ω*m.

Layout

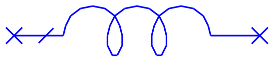
The 2D projection of this element in the x/y plane displays in EM Layout View.

3D EM Layout

This element has a layout cell specifically for Analyst 3D EM layouts. See "Using 3D EM Elements " for details on using 3D pCells with Analyst.

Air Coil With Offset Axial Leads: AIRCOIL2

Symbol



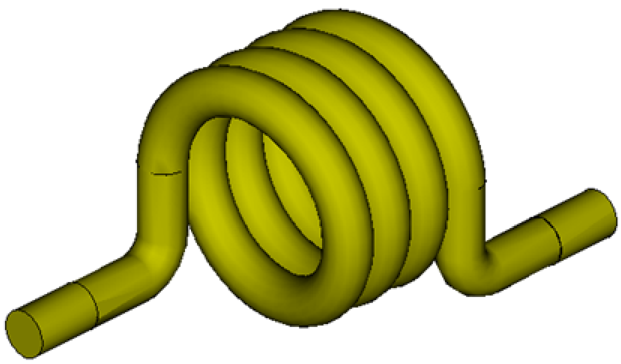
Summary

AIRCOIL2 models an air core coil inductor with offset axial leads. Coil and lead dimensions are configurable.

This element is for use in 3D EM documents only. See “Using 3D EM Elements ” for details on using 3D pCells with Analyst.

Topology

See “Air Coil With Centered Axial Leads: AIRCOIL1” for similar topology details.



Parameters

Name	Description	Unit Type	Default
ID	Subcircuit ID	Text	S1
NET	Subcircuit name	Text	"AIRCOIL2"
*M	Multiplicity factor - not used for this model		1
NTurns	Number of turns		4
WireDia	Diameter of the wire	Length	1 mm
CoilDia	Coil inner diameter	Length	3.5 mm
Pitch	Distance between turns, measured from wire center-to-center	Length	1.01 mm

Name	Description	Unit Type	Default
LeadLen	Length of lead	Length	1.5 mm
LeadOff	Offset distance between coil body and beginning of lead	Length	1.5 mm
LeadType	Type of lead contact: 0=round post, 1=bottom flat tab, 2=lofted flat tab		0
TabLenRatio	Ratio of tab length to total lead length for LeadType=1 or 2. 0<TabLenRatio<1		0.75
Rho	Bulk resistivity of conductor metal normalized to gold		1

** indicates a secondary parameter*

Parameter Details

Parameters for this model are unitless. Parameters do not scale when changing Global Units.

The lead geometry is configurable. The simplest lead, LeadType=0, is a round post. Next, LeadType=1 is a lead with a bottom flat tab. The length of the flat bottom section is (LeadLen·TabLenRatio), and the length of the round section is (LeadLen·(1-TabLenRatio)). Lastly, LeadType=2 is a lead with flat bottom that gradually transitions to a round cross-section. The length of the flat bottom section is (LeadLen·TabLenRatio), and the length of the lofted (transition) section is (LeadLen·(1-TabLenRatio)).

LeadOff. LeadOff is the offset distance between the coil body and lead. The cross-section of this region is always round. This length typically represents the length of the gap between body of the coil, and the end of the trace that the coil is attached to.

Rho. Rho is the bulk resistivity of conductor metal normalized to gold, where Rho_Gold=2.44 x 10^{-8} Ω*m. So actual bulk resistivity of conductor metal is (Rho·2.44 x 10^{-8}) Ω*m.

Layout

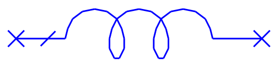
The 2D projection of this element in the x/y plane displays in EM Layout View.

3D EM Layout

This element has a layout cell specifically for Analyst 3D EM layouts. See "Using 3D EM Elements " for details on using 3D pCells with Analyst.

Air Coil With Opposed Leads: AIRCOIL3

Symbol



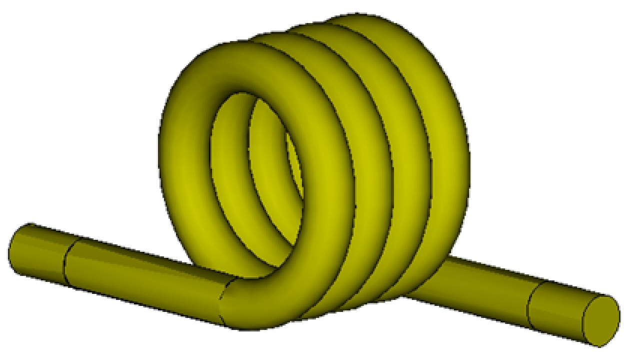
Summary

AIRCOIL3 models an air core coil inductor with opposed leads. Coil and lead dimensions are configurable.

This element is for use in 3D EM documents only. See “Using 3D EM Elements ” for details on using 3D pCells with Analyst.

Topology

See “Air Coil With Centered Axial Leads: AIRCOIL1” for similar topology details.



Parameters

Name	Description	Unit Type	Default
ID	Subcircuit ID	Text	S1
NET	Subcircuit name	Text	"AIRCOIL3"
*M	Multiplicity factor - not used for this model		1
NTurns	Number of turns		4
WireDia	Diameter of the wire	Length	1 mm
CoilDia	Coil inner diameter	Length	3.5 mm
Pitch	Distance between turns, measured from wire center-to-center	Length	1.01 mm

Name	Description	Unit Type	Default
LeadLen	Length of lead	Length	1.5 mm
LeadOff	Offset distance between coil body and beginning of lead	Length	1.5 mm
LeadType	Type of lead contact: 0=round post, 1=bottom flat tab, 2=lofted flat tab		0
TabLenRatio	Ratio of tab length to total lead length for LeadType=1 or 2. 0<TabLenRatio<1		0.75
Rho	Bulk resistivity of conductor metal normalized to gold		1

** indicates a secondary parameter*

Parameter Details

Parameters for this model are unitless. Parameters do not scale when changing Global Units.

The lead geometry is configurable. The simplest lead, LeadType=0, is a round post. Next, LeadType=1 is a lead with a bottom flat tab. The length of the flat bottom section is (LeadLen·TabLenRatio), and the length of the round section is (LeadLen·(1-TabLenRatio)). Lastly, LeadType=2 is a lead with flat bottom that gradually transitions to a round cross-section. The length of the flat bottom section is (LeadLen·TabLenRatio), and the length of the lofted (transition) section is (LeadLen·(1-TabLenRatio)).

LeadOff. LeadOff is the offset distance between the coil body and lead. The cross-section of this region is always round. This length typically represents the length of the gap between body of the coil, and the end of the trace which the coil is attached to.

Rho. Rho is the bulk resistivity of conductor metal normalized to gold, where Rho_Gold=2.44 x 10^{-8} Ω*m. So actual bulk resistivity of conductor metal is (Rho·2.44 x 10^{-8}) Ω*m.

Layout

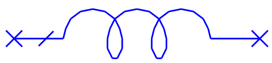
The 2D projection of this element in the x/y plane displays in EM Layout View.

3D EM Layout

This element has a layout cell specifically for Analyst 3D EM layouts. See “Using 3D EM Elements ” for details on using 3D pCells with Analyst.

Air Coil With Radial Leads: AIRCOIL4

Symbol



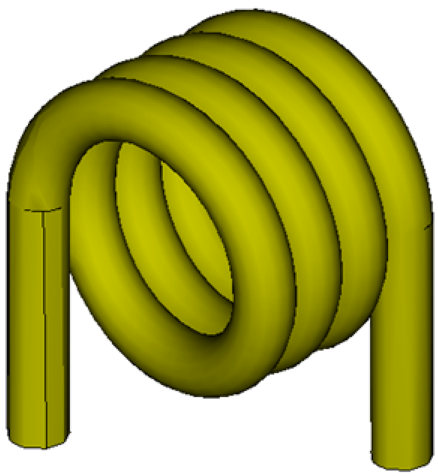
Summary

AIRCOIL4 models an air core coil inductor with radial leads. Coil and lead dimensions are configurable.

This element is for use in 3D EM documents only. See “Using 3D EM Elements ” for details on using 3D pCells with Analyst.

Topology

See “Air Coil With Centered Axial Leads: AIRCOIL1” for similar topology details.



Parameters

Name	Description	Unit Type	Default
ID	Subcircuit ID	Text	S1
NET	Subcircuit name	Text	"AIRCOIL4"
*M	Multiplicity factor - not used for this model		1
NTurns	Number of turns		4
WireDia	Diameter of the wire	Length	1 mm
CoilDia	Coil inner diameter	Length	3.5 mm
Pitch	Distance between turns, measured from wire center-to-center	Length	1.01 mm

Name	**Description**	**Unit Type**	**Default**
LeadLen	Length of lead	Length	1.5 mm
Rho	Bulk resistivity of conductor metal normalized to gold		1

** indicates a secondary parameter*

Parameter Details

Parameters for this model are unitless. Parameters do not scale when changing Global Units.

Rho. Rho is the bulk resistivity of conductor metal normalized to gold, where Rho_Gold=2.44 x 10^{-8} Ω*m. So actual bulk resistivity of conductor metal is (Rho·2.44 x 10^{-8}) Ω*m.

Layout

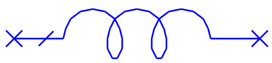
The 2D projection of this element in the x/y plane displays in EM Layout View.

3D EM Layout

This element has a layout cell specifically for Analyst 3D EM layouts. See “Using 3D EM Elements ” for details on using 3D pCells with Analyst.

Air Coil With Centered Radial Leads: AIRCOIL5

Symbol



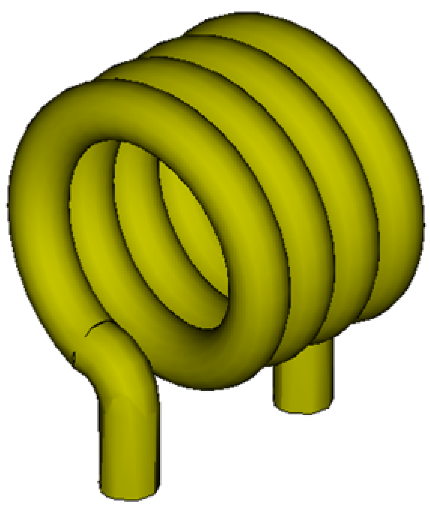
Summary

AIRCOIL4 models an air core coil inductor with centered radial leads. Coil and lead dimensions are configurable.

This element is for use in 3D EM documents only. See "Using 3D EM Elements " for details on using 3D pCells with Analyst.

Topology

See "Air Coil With Centered Axial Leads: AIRCOIL1" for similar topology details.



Parameters

Name	Description	Unit Type	Default
ID	Subcircuit ID	Text	S1
NET	Subcircuit name	Text	"AIRCOIL5"
*M	Multiplicity factor - not used for this model		1
NTurns	Number of turns		4
WireDia	Diameter of the wire	Length	1 mm
CoilDia	Coil inner diameter	Length	3.5 mm
Pitch	Distance between turns, measured from wire center-to-center	Length	1.01 mm

Name	Description	Unit Type	Default
LeadLen	Length of lead	Length	1.5 mm
Rho	Bulk resistivity of conductor metal normalized to gold		1

** indicates a secondary parameter*

Parameter Details

Parameters for this model are unitless. Parameters do not scale when changing Global Units.

Rho. Rho is the bulk resistivity of conductor metal normalized to gold, where Rho_Gold=2.44 x 10^{-8} Ω*m. So actual bulk resistivity of conductor metal is (Rho·2.44 x 10^{-8}) Ω*m.

Layout

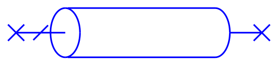
The 2D projection of this element in the x/y plane displays in EM Layout View.

3D EM Layout

This element has a layout cell specifically for Analyst 3D EM layouts. See "Using 3D EM Elements " for details on using 3D pCells with Analyst.

Coaxial Line: COAX3D

Symbol

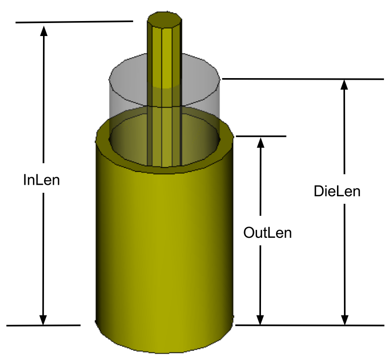


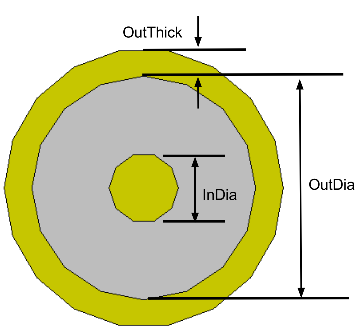
Summary

COAX3D models a coaxial transmission line.

This element is only used in 3D EM documents. See "Using 3D EM Elements" for details on using 3D parts with Analyst.

Topology





Top View

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	S1
NET	Subcircuit name	Text	"COAX3D"

Name	Description	Unit Type	Default
*M	Multiplicity factor - not used for this model		1
InDia	Inner conductor diameter	Length	1.27 mm
OutDia	Outer conductor diameter	Length	4.1 mm
InLen	Inner conductor length	Length	1.27 mm
DieLen	Dielectric insulator length	Length	1.27 mm
OutLen	Outer conductor length	Length	1.27 mm
OutThick	Outer conductor thickness	Length	0.127 mm
Er	Relative dielectric constant of insulator		2.2
TanD	Electric loss tangent of insulator		0.0001
Rho	Bulk resistivity of conductor metal normalized to gold		1

** indicates a secondary parameter*

Parameter Details

Parameters for this model are unitless. Parameters will not scale when changing Global Units.

Er. Er is the permittivity of the insulator material relative to the permittivity of free space, where $\varepsilon_0 = 8.854 \times 10^{-12}$ F/m.

Rho. Rho is the bulk resistivity of conductor metal normalized to gold, where Rho_Gold=2.44 x 10^{-8} Ω*m. So actual bulk resistivity of conductor metal is (Rho·2.44 x 10^{-8}) Ω*m.

Layout

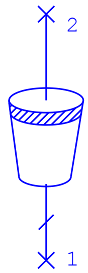
The 2D projection of this element in the x/y plane will display in EM layout view.

3D EM Layout

This element is specifically used for Analyst 3D EM layouts. See “Using 3D EM Elements ” for details on using 3D parts with Analyst.

Copper Pillar Bump with Tin Cap: CU_PILLAR

Symbol

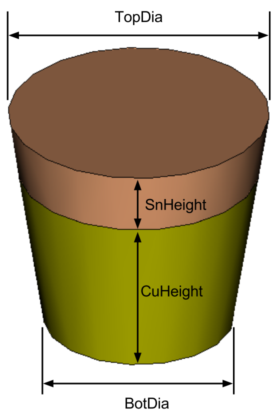


Summary

CU_PILLAR models a copper pillar bump with a tin cap.

This element is for use in 3D EM documents only. See "Using 3D EM Elements " for details on using 3D pCells with Analyst.

Topology



Parameters

Name	Description	Unit Type	Default
ID	Subcircuit ID	Text	S1
NET	Subcircuit name	Text	"CU_PILLAR"
*M	Multiplicity factor - not used for this model		1
CuHeight	Height of the copper pillar base		65 um
SnHeight	Height of the tin cap		25 um
BotDia	Diameter at the bottom of the copper pillar base		36.5 um
TopDia	Diameter at the top of the tin cap		50 um
CuCond	Width of the bondpad mounted on the MMIC substrate		5.74×10^{7} S/m

Name	Description	Unit Type	Default
SnCond	Length of the bondpad mounted on the MMIC substrate		9.17×10^6 S/m

** indicates a secondary parameter*

Restrictions

20 um ≤ CuHeight ≤ 75 um

0 < SnHeight ≤ 40 um

20 um ≤ BotDia ≤ 50 um

20 um ≤ TopDia ≤ 50 um

TopDia ≥ BotDia

TopDia-BotDia ≤ 25 um

Layout

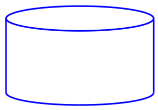
The 2D projection of this element in the x/y plane displays in EM Layout View.

3D EM Layout

This element has a layout cell specifically for Analyst 3D EM layouts. See “Using 3D EM Elements ” for details on using 3D pCells with Analyst.

Dielectric Cylinder Resonator: CYL_RESON

Symbol



Summary

CYL_RESON models a dielectric cylinder resonator.

This element is for use in 3D EM documents only. See "Using 3D EM Elements " for details on using 3D pCells with Analyst.

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	S1
NET	Subcircuit name	Text	"CYL_RESON"
*M	Multiplicity factor - not used for this model		1
InDia	Cylinder inner diameter	Length	6.35 mm
OutDia	Cylinder outer diameter	Length	12.7 mm
Thick	Cylinder height	Length	6.35 mm
Er	Relative dielectric constant		40
TanD	Loss tangent		0.0001

** indicates a secondary parameter*

Parameter Details

Er. Er is the permittivity of the dielectric material relative to the permittivity of free space, where $\varepsilon_0 = 8.854 \times 10^{-12}$ F/m.

TanD. TanD is the electric loss tangent of the dielectric material.

Layout

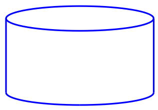
The 2D projection of this element in the x/y plane will display in EM Layout View.

3D EM Layout

This element has a layout cell specifically for Analyst 3D EM layouts. See "Using 3D EM Elements " for details on using 3D pCells with Analyst.

Dielectric Disk Resonator: DISK_RESON

Symbol



Summary

DISK_RESON models a dielectric disk resonator.

This element is for use in 3D EM documents only. See "Using 3D EM Elements " for details on using 3D pCells with Analyst.

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	S1
NET	Subcircuit name	Text	"DISK_RESON"
*M	Multiplicity factor - not used for this model		1
Dia	Disk diameter	Length	12.7 mm
Thick	Disk height	Length	6.35 mm
Er	Relative dielectric constant		40
TanD	Loss tangent		0.0001

** indicates a secondary parameter*

Parameter Details

Er. Er is the permittivity of the dielectric material relative to the permittivity of free space, where $\varepsilon_0 = 8.854 \text{ x } 10^{-12}$ F/m.

TanD. TanD is the electric loss tangent of the dielectric material.

Layout

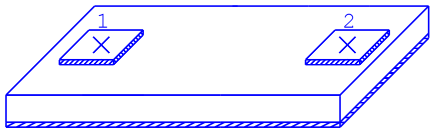
The 2D projection of this element in the x/y plane will display in EM Layout View.

3D EM Layout

This element has a layout cell specifically for Analyst 3D EM layouts. See "Using 3D EM Elements " for details on using 3D pCells with Analyst.

MMIC Substrate with Bond Pads: MMIC_3D

Symbol

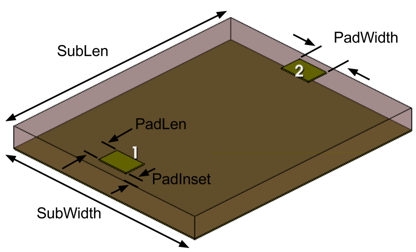


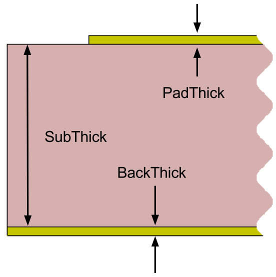
Summary

MMIC_3D models a MMIC substrate containing bondpads and a backside coating for mounting purposes.

This element is for use in 3D EM documents only. See "Using 3D EM Elements " for details on using 3D pCells with Analyst.

Topology





Side View

Parameters

Name	Description	Unit Type	Default
ID	Subcircuit ID	Text	S1

Name	Description	Unit Type	Default
NET	Subcircuit name	Text	"MMIC_3D"
*M	Multiplicity factor - not used for this model		1
SubWidth	Width of the MMIC substrate defined along y-axis	Length	1 mm
SubLen	Length of the MMIC substrate defined along x-axis	Length	1.2 mm
SubThick	Thickness of the MMIC substrate defined along z-axis	Length	0.1 mm
BackThick	Thickness of the backside coating of the MMIC substrate	Length	0.005 mm
PadWidth	Width of the bondpad mounted on the MMIC substrate	Length	0.15 mm
PadLen	Length of the bondpad mounted on the MMIC substrate	Length	0.1 mm
PadThick	Thickness of the bondpad mounted on the MMIC substrate	Length	0.005 mm
PadInset	Distance between the edge of the MMIC substrate and the edge of the bondpad	Length	0.045 mm
Er	Relative dielectric constant of MMIC substrate		12.9
TanD	Loss tangent of MMIC substrate		0.0004
Rho	Bulk resistivity relative to gold of bondpad and backside coating		1

** indicates a secondary parameter*

Parameter Details

Er. Er is the permittivity of the insulator material relative to the permittivity of free space, where $\varepsilon_0 = 8.854 \times 10^{-12}$ F/m.

Rho. Rho is the bulk resistivity of conductor metal normalized to gold, where Rho_Gold=2.44 x 10^{-8} Ω*m. So actual bulk resistivity of conductor metal is (Rho·2.44 x 10^{-8}) Ω*m.

Layout

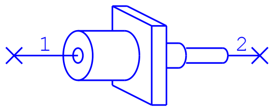
The 2D projection of this element in the x/y plane displays in EM Layout View.

3D EM Layout

This element has a layout cell specifically for Analyst 3D EM layouts. See “Using 3D EM Elements ” for details on using 3D pCells with Analyst.

Configurable SMA Connector: SMACONFIG

Symbol

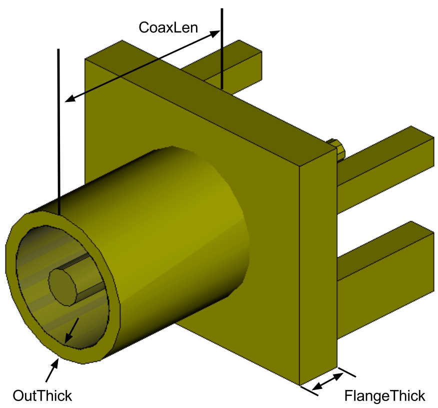


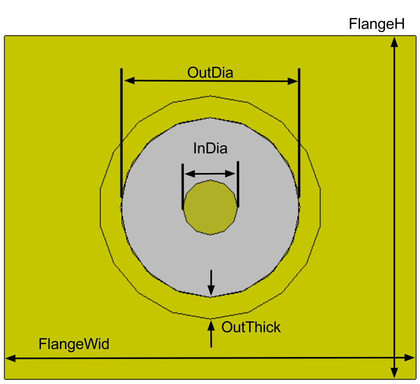
Summary

SMACONFIG is a configurable SMA connector model. Coaxial connector dimensions can be specified, along with ground tab dimensions, and contact pin type.

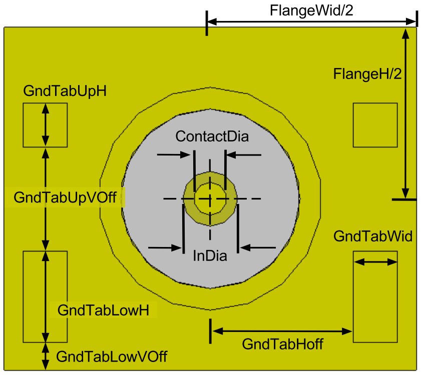
This element is for use in 3D EM documents only. See "Using 3D EM Elements " for details on using 3D pCells with Analyst.

Topology

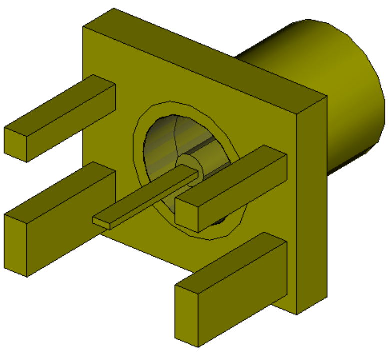




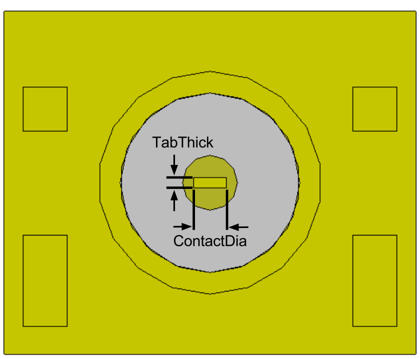
Front View - Coax



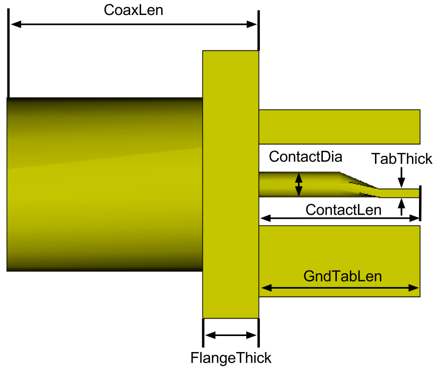
Back View - Round Post Contact



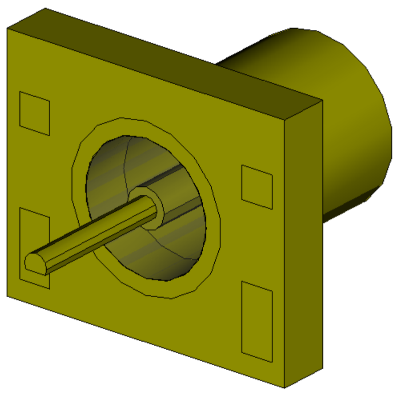
Flat Tab Contact



Back View - Flat Tab Contact



Side View - Round Post Transition
to Flat Tab Contact



Round Post Contact
PostFlatRatio=0.25 and GndTabLen=0

Parameters

Name	Description	Unit Type	Default
ID	Subcircuit ID	Text	S1
NET	Subcircuit name	Text	"SMACONFIG"
*M	Multiplicity factor - not used for this model		1
InDia	Diameter of inner center pin	Length	1.27 mm
OutDia	Diameter of outer conductor (or set OutDia = 0 to use impedance to compute outer diameter)	Length	4.15 mm
Imped	Target impedance of coax used to compute outer radius, if OutDia=0		50 Ohms
OutThick	Thickness of coax outer conductor	Length	0.50 mm
CoaxLen	Length of coax section	Length	7.40 mm
ContactType	Center contact type: 0=round post, 1=round post with spherical cap, 2=flat tab, 3=round post to tab transition		0

Name	Description	Unit Type	Default
ContactDia	Diameter of post-type contacts, and width of tab for tabular contacts	Length	0.76 mm
ContactLen	Length of contact	Length	4.75 mm
PostFlatRatio	Fraction (0 to 1) of pin to remove via horizontal cut (makes mating surface)		0
TabThick	Vertical thickness of tab	Length	0.25 mm
TabLenRatio	Length of the tab on a post-to-tab contact (relative to the overall contact length)		0.25
TabTranRatio	Length of the tab-to-circular post transition section (relative to the overall contact length)		0.25
FlangeThick	Flange thickness	Length	1.650 mm
FlangeH	Flange height	Length	7.92 mm
FlangeWid	Flange width	Length	9.52 mm
GndTabLen	Ground tab length. Set GndTabLen = 0 for no ground tabs	Length	4.75 mm
GndTabWid	Ground tab width	Length	1.02 mm
GndTabUpH	Upper ground tab height	Length	1.02 mm
GndTabLowH	Lower ground tab height	Length	2.11 mm
GndTabHOff	Horizontal offset of ground tabs from center	Length	3.74 mm
GndTabUpVOff	Vertical offset of upper ground tabs from top of lower ground tabs	Length	1.730 mm
GndTabLowVOff	Vertical offset of lower ground tabs from bottom of flange	Length	0 mm
Er	Relative dielectric constant of insulator		2.1
TanD	Electric loss tangent of insulator		0.0004
Rho	Bulk resistivity relative to gold		1

** indicates a secondary parameter*

Parameter Details

Parameters for this model are unitless. Parameters will not scale when changing Global Units.

A common SMA configuration is an edge mount connector with a round post contact, that mounts onto a PCB board. In this case, the distance between the ground tabs is the thickness of the PCB board and the bottom of the upper ground tabs align with the bottom of the center contact. For this configuration, a good set of parameter values to start with are:

- GndTabLowVOff = 0. This value aligns the lower ground tabs with the bottom of the SMA flange.
- GndTabUpVOff = PCB board thickness. This value sets the upper ground tab flush with top of board.
- GndTabLowH = 0.5·(FlangeH - ContactDia) – (PCB board thickness). These values set the lower ground tab height equal to the distance between the bottom of the PCB board and bottom of the flange.

There are two methods for configuring the coaxial side of the connector. The first method is to specify the inner (InDia) and outer (OutDia) diameters of the coaxial line. The impedance parameter (Imped) is ignored. The second method is to only specify the inner diameter (InDia), and set OutDia = 0. Then the outer diameter of the coaxial line will be calculated from the inner diameter (InDia) and impedance (Imped).

Er. Er is the permittivity of the insulator material relative to the permittivity of free space, where $\varepsilon_0 = 8.854 \times 10^{-12}$ F/m.

Rho. Rho is the bulk resistivity of conductor metal normalized to gold, where Rho_Gold=2.44 x 10^{-8} Ω*m. So actual bulk resistivity of conductor metal is (Rho·2.44 x 10^{-8}) Ω*m.

Layout

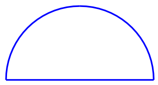
The 2D projection of this element in the x/y plane will display in EM layout view.

3D EM Layout

This element is specifically used for Analyst 3D EM layouts. See "Using 3D EM Elements " for details on using 3D parts with Analyst.

Solder Trace: SOLDER_BLOB

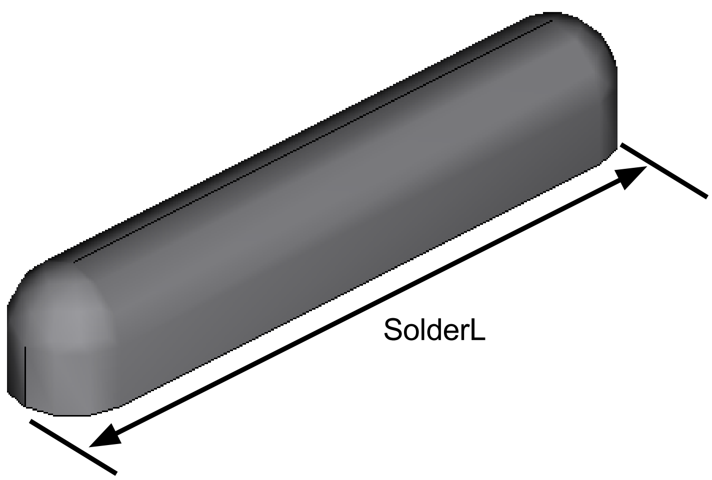
Symbol



Summary

SOLDER_BLOB models a solder trace or solder bridge commonly used for making RF board modifications in the lab. This element is for use in 3D EM documents only. See "Using 3D EM Elements " for details on using 3D pCells with Analyst.

Topology



Parameters

Name	Description	Unit Type	Default
ID	Subcircuit ID	Text	S1
NET	Subcircuit name	Text	SOLDER_BLOB
*M	Multiplicity factor - not used for this model		1
SolderH	Height of Solder Blob	Length	6 mils
SolderL	Length of Solder Body	Length	36 mils
SolderW	Width of the Solder Body	Length	6 mils
SolderRad	Radius of the Edge Fillet	Length	3 mils

Name	Description	Unit Type	Default
Cond	Electric Bulk Conductivity of material for Solder Blob		6900000 S/m

**indicates a secondary parameter*

Restrictions

- 1 mil ≤ SolderH ≤ 1000 mil
- 1 mil < SolderL≤ 5000 mi
- 1 mil ≤ SolderW ≤ 1000 mil
- SolderRad ≤ SolderH/2
- SolderRad ≤ SolderL/2
- SolderRad ≤ SolderW/2
- 10^6 S/m ≤ Cond ≤ 10^8 S/m

Layout

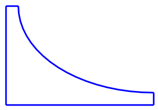
The 2D projection of this element in the x/y plane displays in the EM Layout View.

3D EM Layout

This element has a layout cell specifically for Analyst 3D EM layouts. See “Using 3D EM Elements ” for details on using 3D pCells with Analyst.

Solder Joint Fillet: SOLDER_FILLET

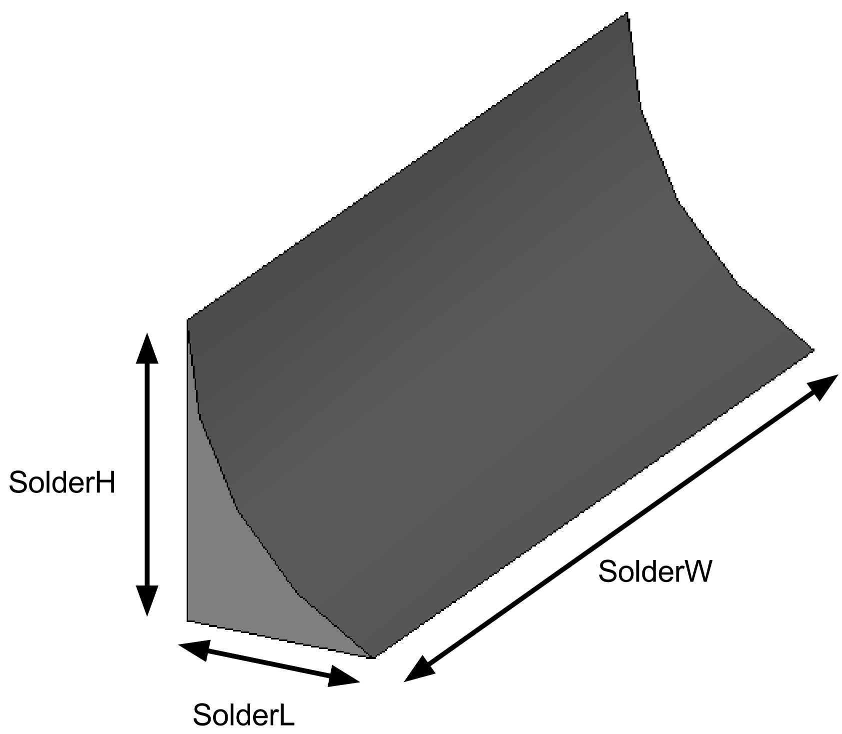
Symbol



Summary

SOLDER_FILLET models a solder joint fillet. The fillet radius is equal to the height (SolderH). This element is for use in 3D EM documents only. See "Using 3D EM Elements " for details on using 3D pCells with Analyst.

Topology



Parameters

Name	Description	Unit Type	Default
ID	Subcircuit ID	Text	S1

Name	Description	Unit Type	Default
*M	Multiplicity factor - not used for this model		
NET	Subcircuit name	Text	SOLDER_FILLET
SolderH	Height of Solder Fillet	Length	45 mils
SolderL	Length of Solder Fillet	Length	27 mils
SolderW	Width of the Solder Fillet	Length	815 mils
Cond	Electric Bulk Conductivity of Solder material		6900000

**indicates a secondary parameter*

Restrictions

- 10 mil ≤ SolderH ≤ 1000 mil
- 10 mil ≤ SolderL≤ 1000 mi
- 10 mil ≤ SolderW ≤ 5000 mil
- 10^6 S/m ≤ Cond ≤ 10^8 S/m

Layout

The 2D projection of this element in the x/y plane displays in the EM Layout View.

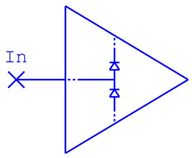
3D EM Layout

This element has a layout cell specifically for Analyst 3D EM layouts. See “Using 3D EM Elements ” for details on using 3D pCells with Analyst.

APLAC

APLAC Ibis: Ibis_AP

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	Ibis1
Type	Buffer type		Input
FILE	IBIS file		""
COMPONENT	The component within the IBIS file	Text	""
PIN	Name of the pin	Text	""
MODEL_NAME	The buffer model name	Text	""
Differential	Differential or single-ended buffer		No
OperatingVoltages	Expose reference voltage nodes		No
DigitalOutput	Expose digital output node		No
*CORNER	Corner (0,-1, or 1 for typ, min, or max) for the I-V curves, C_comp, package model, and reference voltages		
*IV_CORNER	Corner (0,-1, or 1 for typ, min, or max) for the I-V curves, uses CORNER if empty		
*C_COMP_CORNER	Corner (0,-1, or 1 for typ, min, or max) for C_comp, uses CORNER if empty		
*PKG_CORNER	Corner (0,-1, or 1 for typ, min, or max) for the package model, uses CORNER if empty		
*RISING_WF	Use [Ramp] data (0), one V-t table (1), or two V-t tables (2) for rising edge transition		
*FALLING_WF	The same as RISING_WF for a falling edge transition		
*INTERPOL	Interpolation method (LINS, CSPN, or CUCO, for linear, cubic spline or cubic convolution)	Text	""
*NO_PKG	Disable package model		Off
*TEMP	Ambient temperature T [K] of the model. Default: the global ambient temperature defined using Analyze, Prepare, SetParam, or Sweep	Temperature	
*TNOM	Nominal temperature Tnom [K] at which the temperature-dependent model parameters have been specified. Default: the global nominal temperature defined using Analyze, Prepare, SetParam, or Sweep	Temperature	

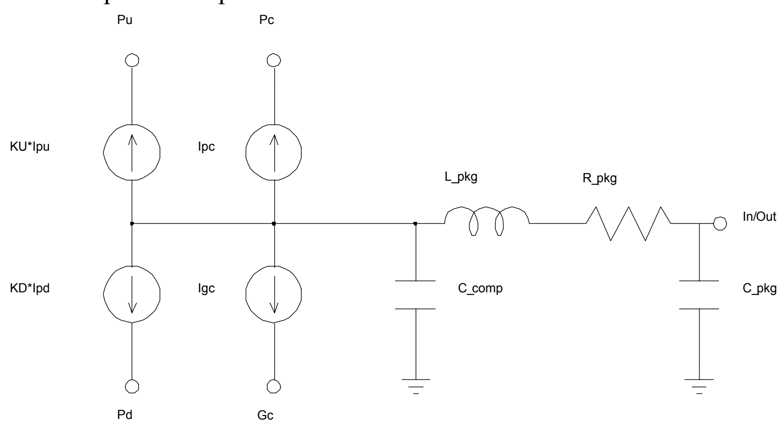
** indicates a secondary parameter*

Implementation Details

Implements a behavioral-level model for an I/O buffer based on the data specified in an IBIS file. IBIS files up to version 6 are allowed, although the simulator does not support all the features described in the IBIS standard.

The IBIS file specified with FILE is read using the "ibischk Golden Parser" developed by the IBIS Open Forum, and the buffer model corresponding to the specified model or pin name is created based on this data. Typically the IBIS file, the component, and the model or pin name are set on the **IBIS** tab of the Element Options dialog box. The error and warning messages generated by the parser are also accessible here.

The internal structure of an IBIS model is shown in the following figure. Node `In/Out` is the analog input (**Input**, **Input_ECL**, or **Terminator** buffer types) or output node of the buffer. The voltage-controlled current sources `KU*Ipu` and `KD*Ipd` model the pullup and pulldown circuitry, respectively, of the buffer. The time-dependent multipliers `KU` and `KD`, which characterize the buffers transitions from one logic state to the other, are determined according to the *V-t* or [Ramp] data specified in the IBIS file. If the *V-t* data is not consistent with the *I-V* data at the first and last time points, the *V-t* data is adjusted to make it consistent and a warning is issued. The voltage-controlled current sources `Ipc` and `Igc` model the power clamp and ground clamp diodes. If any of the keywords [C_comp_pullup], [C_comp_pulldown], [C_comp_power_clamp], or [C_comp_gnd_clamp] are specified in the IBIS file, the capacitor C_comp is not created and the specified capacitors are created between the die node and the corresponding voltage reference node.



The following buffer types are supported:

- **Input** buffers have power clamp and ground clamp diodes, but no pullup or pulldown circuitry. The voltage at the digital output node is either `0 V` or `1 V`, representing the logic state of the buffer.
- **Output** buffers have pullup and pulldown circuitry, but no clamp diodes. The signal at the `Ctrl` node controls the switching between the logic high and low states of the buffer.
- **3-state** buffers have both input and output buffer characteristics. The signal at the `En` node enables or disables the pullup and pulldown circuitry of the buffer.
- **I/O** buffers are similar to 3-state buffers, except that they have the digital output node.
- **Open_drain** and **Open_sink** buffers are similar to Output buffers, except that they do not have pullup circuitry
- **Open_source** buffers are similar to Output buffers, except that they do not have pulldown circuitry.
- **I/O_Open_drain** and **I/O_Open_sink** buffers are similar to I/O buffers, except that they do not have pullup circuitry.
- **I/O_Open_source** buffers are similar to I/O buffers, except that they do not have pulldown circuitry.
- **Input_ECL** buffers are identical to Input buffers, except for the default values of the `Vinl` and `Vinh` sub-parameters.

- **Output_ECL** buffers are similar to Output buffers, except that the pulldown table is referenced to the pullup-reference voltage.
- **3-state_ECL** buffers are similar to 3-state buffers, except that the pulldown table is referenced to the pullup-reference voltage.
- **I/O_ECL** buffers are similar to I/O buffers, except that the pulldown table is referenced to the pullup-reference voltage.
- **Terminator** buffers have power clamp and ground clamp diodes, but no pullup or pulldown circuitry. Additionally, they have resistors [Rpower] and [Rgnd] in parallel with the power clamp and ground clamp diodes, respectively, and a series connection of resistor [Rac] and capacitor [Cac] in parallel with C_comp.

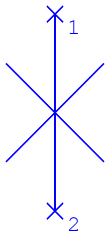
The buffer model is selected using the name of the pin, specified with PIN, or the name of the model, specified with MODEL_NAME. If both PIN and MODEL_NAME are specified, MODEL_NAME overrides the PIN specification. If Differential is set to "Yes", a differential buffer model is assumed. In this case, the non-inverting buffer model is connected to the positive input/output node and the inverting buffer model is connected to the negative input/output node. For a differential pin model, PIN must be specified, and the pin name must match the non-inverting (first) pin name of the [Diff Pin] specification within the IBIS file. The type and the polarity of the buffer are determined by the sub-parameters of the [Model] keyword.

Required and optional nodes for each IBIS buffer type are listed in the following table, with optional nodes in parenthesis. The voltages of the control (Ctrl), enable (En), and digital output (Do) nodes represent digital signals having logic high and logic low states (the value of a digital signal is logic high if it is above `0.5 V`, otherwise it is logic low). The pullup, pulldown, power clamp, and ground clamp reference voltages can be connected to nodes Pu, Pd, Pc, and Gc, respectively. If OperatingVoltages is set to "No", the reference-voltage sources are created internally. In this case, the reference voltages are determined by the keywords [Pullup Reference], [Pulldown Reference], [POWER Clamp Reference], and [GND Clamp Reference] in the IBIS file. If they are missing, the default for the pullup and power clamp references is the voltage specified with the [Voltage Range] keyword and `0 V` for the pulldown and ground clamp references.

Buffer type	Ctrl	En	Do	Pc	Gc	Pu	Pd
Input	□	□	(x)	(x)	(x)	□	□
Output	x	□	□	□	□	(x)	(x)
3-state	x	x	x	(x)	(x)	(x)	(x)
I/O	x	x	(x)	(x)	(x)	(x)	(x)
Open_drain	x	□	□	□	□	□	(x)
Open_sink	x	□	□	□	□	□	(x)
Open_source	x	□	□	□	□	(x)	□
I/O_open_drain	x	x	(x)	(x)	(x)	□	(x)
I/O_open_sink	x	x	(x)	(x)	(x)	□	(x)
I/O_open_source	x	x	(x)	(x)	(x)	(x)	□
Input_ECL	□	□	(x)	(x)	(x)	□	□
Output_ECL	x	□	□	□	□	(x)	□
3-state_ECL	x	x	□	(x)	(x)	(x)	□
I/O_ECL	x	x	(x)	(x)	(x)	(x)	□
Terminator	□	□	□	(x)	(x)	□	□

APLAC Josephson Junction: JJ_AP

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	JJ1
*ALPHA	Amplitude alpha of the cosine term of the junction resistance. ALPHA is meaningful only if RQ is specified.		
*C	Capacitance of the junction C [F]	Capacitance	
*IC	Critical current of the junction Ic [A]	Current	
*K	Scaling factor k of the internal integrator capacitance		
*PHASE	Output phase at t = 0 [rad]. PHASE is equal to the initial voltage of the internal capacitance Cp.	Angle	
*RN	Extra parallel resistance Rn [ohm] of Cp. Typically 1 Gohm resistance is recommended if zero pivot problems occur.	Resistance	
*RQ	Resistance of the junction RQ [ohm]	Resistance	
*TEMP	Ambient temperature T [K] of the model. Default: the global ambient temperature defined using Analyze, Prepare, SetParam, or Sweep	Temperature	
*TNOM	Nominal temperature Tnom [K] at which the temperature-dependent model parameters have been specified. Default: the global nominal temperature defined using Analyze, Prepare, SetParam, or Sweep	Temperature	

**indicates an optional parameter*

Implementation details

JJ_AP is the equivalent circuit of the Josephson Junction (JJ). This model includes the junction resistance and capacitance and can be used, for example, to build a Superconductive QUantum Interference Device (SQUID).

The current of the junction is defined [1]

$$i_1 = I_c \cdot \sin\theta + \frac{\Phi_0}{2\pi}\left[\frac{1 + \alpha \cdot \cos\theta}{R_Q}\frac{d}{dt} + C\frac{d^2}{dt^2}\right]\theta$$

[1]H. Seppa, Analysis of an R-SQUID Noise Thermometer, VTT Publications 54, Technical Research Centre of Finland, 1989.

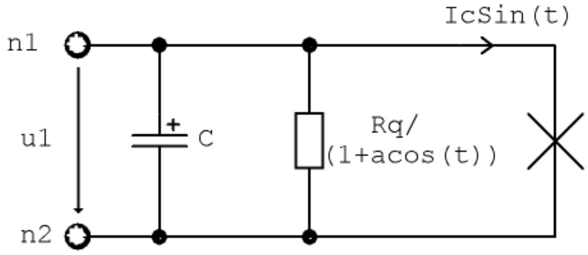
The voltage across the junction u_1 is

$$u_1 = \frac{\Phi_0}{2\pi} \cdot \frac{d\theta}{dt}$$

θ is the phase difference across the junction and Φ_0 is the flux quantum

$$\Phi_0 = \frac{h}{2e} \approx 2.07 \cdot 10^{-15}\,Wb$$

The following figure shows the components of the Josephson junction



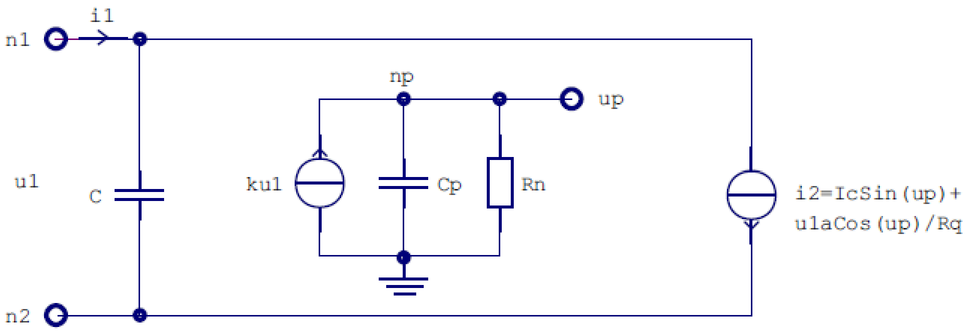
The resistance and the current terms are modeled using a single VCCS controlled by voltages u_1 and u_p (see the following figure). The subcircuit building up voltage u_p (phase difference θ) is defined by the equation (R_n excluded).

$$k \cdot u_1 = C_p \cdot \frac{du_p}{dt}$$

which, together with the equation for the voltage across the junction u_1, yields

$$C_p = \frac{\Phi_0}{2\pi} \cdot k$$

Constant k is selected internally to scale capacitance value C_p. Specifying R_n is recommended if a zero pivot situation occurs. The following figure shows the Josephson junction macro model.



Current i_2 and its derivatives with respect to the controlling voltages are

$$i_2 = I_c \sin u_p + \frac{1 + \alpha \cos u_p}{R_Q} u_1$$

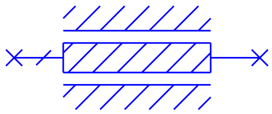
$$\frac{di_2}{du_p} = I_c \cos u_p - \frac{\alpha \sin u_p}{R_Q} u_1$$

$$\frac{di_2}{du_p} = I_c \cos u_p - \frac{\alpha \sin u_p}{R_Q} u_1$$

Coplanar

(Obsolete) Coplanar Line (EM Quasi-Static): CPW1LIN

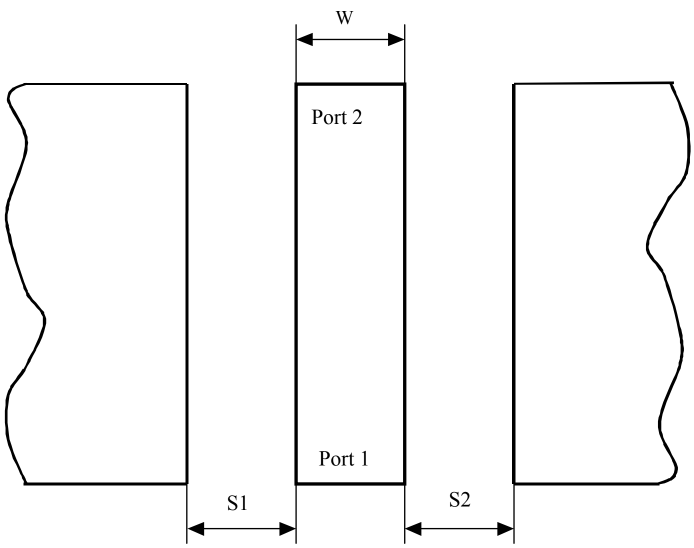
Symbol



Summary

This element is OBSOLETE and is replaced by the Single Conductor Coplanar Line (EM Based Quasi-Static) (CPWLINX), Coplanar Line (EM Quasi-Static) (CPW1LINE), or Coplanar Line (Closed Form) CPWLINE) element.

Topology



Parameters

Name	Description	Unit Type	Default
ID	Name	Text	TL1
W	Conductor width	Length	0 um
S1	Left gap width	Length	0 um
S2	Right gap width	Length	0 um
ShI	Cover shield presence indicator		0
SbI	Backing shield presence indicator		0
L	Conductor length	Length	0 um
Acc	Accuracy parameter		1
CPWSUB	Substrate definition	Text	

Implementation Details

This circuit component models a section of coplanar waveguide on a shielded substrate. The parameters W (conductor width), S1, S2 (S1 - gap between conductor and the left ground halfplane; S2 - gap between conductor and the right ground halfplane) and L (conductor length) are dimensions entered in the default length units. Left and right directions are defined in respect to the line of sight from port 1 to port 2.

The parameter Acc is the accuracy parameter (1<=Acc<=10). The default value for Acc is 1. If Acc is less than 1 or greater than 10 it sets automatically to 2. Larger value of Acc increase density of mesh used in computations. Accuracy of model parameters may gain slightly from increasing Acc at the expense of noticeable growth of computation time. As a rule of thumb, good trade-off between accuracy and computation time is to set Acc to 1.

The parameter CPWSUB specifies the substrate element, which defines additional cross sectional parameters of the transmission line and height of the optional metallic cover. If blank, a default is used.

This component can have metallic grounded shield backing the substrate and metallic cover. Presence of backing shield is controlled by indicator Sbi. If Sbi is set to 0, backing shield is absent; if Sbi is set to 1, backing shield is present. Presence of cover is controlled by indicator Shi. If Shi is set to 0, cover is absent; if Shi is set to 1, cover is present.

This component doesn't impose restrictions on the conductor thickness (thickness may be zero, positive, or negative). Negative thickness means that the conductor is recessed into the substrate.

The component accounts for losses in metal and in substrate dielectric. Dispersion is not included.

NOTE: The implementation of EM Quasi-Static models heavily relies on the involved numerical algorithms. This may lead to the noticeable increase of computation time for some schematics that employ many models of this kind.

Layout

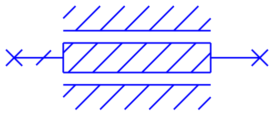
This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

References

M.B. Bazdar, A.R. Djordjevic, R.F. Harrington, and T.K. Sarkar, "Evaluation of quasi-static matrix parameters for multiconductor transmission lines using Galerkin's method," IEEE Trans. Microwave Theory Tech., vol. MTT-42, July 1994, pp. 1223-1228

Coplanar Line (EM Quasi-Static): CPW1LINE

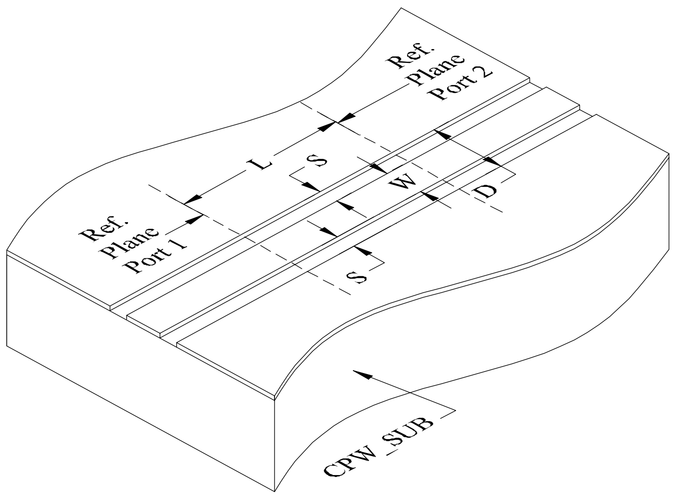
Symbol



Summary

CPW1LINE models a section of symmetric coplanar waveguide (gaps between strip conductor and lateral grounds have equal width) on a dielectric substrate. This model accounts for an optional metallic cover, an optional backing ground plane, and allows for arbitrary metal thickness of the signal conductor and lateral ground planes.

Topology



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	CP1
W	Conductor width	Length	W[a]
S	Gap width	Length	W[a]
L	Conductor length	Length	L[a]
Acc	Accuracy parameter		1
CPW_SUB	Substrate Definition	Text	[b]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory.

[b]If only one CPW_SUB is present in the schematic, this substrate is automatically used. If multiple CPW_SUB substrate definitions are present, the user must specify.

Parameter Details

CPW_SUB. Supplies parameters for dielectric substrate, conductor thickness, conductor metal properties, the presence/absence of metallic cover/backing, and the cover height over the substrate. The Cover and Gnd parameters allow the addition/elimination of infinite metallic plates acting as a cover or grounded backing. The CPW1LINE model does not use the following CPW_SUB parameters: Hab, ER_Nom, H_Nom, Hcov_Nom, Hab_Nom, and T_Nom.

Acc. The accuracy parameter. The default value for Acc is 1. If Acc is less than 1 or greater than 10 it is set automatically to 2.

Parameter Restrictions and Recommendations

1. The Acc parameter A is limited to 1 < Acc < 10. A larger value of Acc increases the density of mesh used in computations. The accuracy of model parameters may improve slightly by increasing Acc, at the expense of a noticeably longer computation time. Generally, a good trade-off between accuracy and computation time is to set Acc to 1.
2. This model does not impose restrictions on the conductor thickness (it may be zero, positive, or negative). Negative thickness means that the conductor is recessed into the substrate.

Implementation Details

1. Model implementation is based on the EM Quasi-Static technique described in [1] [4–6]. It accounts for losses in metal and in the substrate dielectric. Dispersion is partly included.
2. Setting Gnd=0 implies that the substrate is not backed by a perfect conductor, but is bounded by infinite air space. Modeling results are strongly affected by thin substrate heights and might differ substantially from modeling results obtained from models that implement the common conception of a coplanar waveguide (for example, CPWLINE). To model a classic coplanar waveguide featuring an infinitely thick substrate, set Gnd=0 and H >2(W+2S), where Gnd and H are parameters of CPW_SUB.
3. To apply Method of Moments for analysis, a quasi-static model creates 1D mesh covering contours of all conductors. The mesh is made of linear segments (pulses) of varying length. The length of a pulse is relatively big at the conductor center; it decreases toward conductor edges to reveal the charge distribution across conductor. If the conductor width is too large it may cause the pulse size to approach zero for pulses close to edge. In these rare cases the model may display a “Length of pulse #nnn equal to zero” error message.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See “Cell Options Dialog Box: Layout Tab ” for Cell Options dialog box **Layout** tab details.

CPW elements have special configurations for the defined line types. The center conductor geometry draws on all the layers defined in the line type. The spacing to the ground plane is then drawn on negative layers with the same name as all of the layers in the line type. You must then draw the same layers on the positive layer to complete CPW layout. See “Negative Layers ” for more information on setting up processes for positive and negative layers.

See “The Layout Process File (LPF)” for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

Recommendations for Use

NOTE: The implementation of EM Quasi-Static models heavily relies on the involved numerical algorithms. This may lead to a noticeable increase in simulation time for schematics that employ many such models.

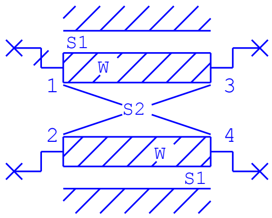
If the thickness of any layer is too small in comparison with the thickness of another layer, simulation time may also noticeably grow.

References

[1] M.B. Bazdar, A.R. Djordjevic, R.F. Harrington, and T.K. Sarkar, "Evaluation of quasi-static matrix parameters for multiconductor transmission lines using Galerkin's method," IEEE Trans. Microwave Theory Tech., vol. MTT-42, July 1994, pp. 1223-1228.

(Obsolete) 2 Coupled Coplanar Lines (EM Quasi-Static): CPW2LIN

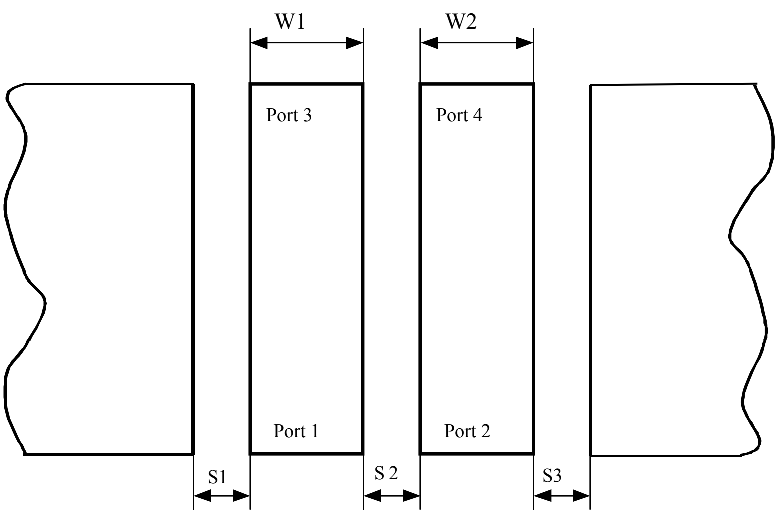
Symbol



Summary

This element is OBSOLETE and is replaced by the 2 Asymmetric Coupled Lines (EM Quasi-Static) (CPW2LINA) element.

Topology



Parameters

Name	Description	Unit Type	Default
ID	Name	Text	TL1
W1	Conductor 1 width - nodes 1&&3	Length	0 um
W2	Conductor 2 width - nodes 2&&4	Length	0 um
S1	Gap Width - Gnd && conductor 1	Length	0 um
S2	Gap Width - conductor 1&& 2	Length	0 um
S3	Gap Width - conductor 2&& Gnd	Length	0 um
Shl	Cover shield presence indicator		0
Sbl	Backing shield presence indicator		0
L	Conductor length	Length	0 um

Name	Description	Unit Type	Default
Acc	Accuracy parameter		1
CPWSUB	Substrate definitions	Text	

Implementation Details

This circuit component models a section of two coupled coplanar waveguides on a shielded substrate. The parameters W1, W2 (conductor widths), S1, S3 (S1 - gap between conductor 1 and the left ground halfplane; S3 - gap between conductor 2 and the right ground halfplane), S2 (gap between conductors), and L (conductor length) are dimensions entered in the default length units. Left and right directions are defined in respect to the line of sight from port 1 to port 3.

The parameter Acc is the accuracy parameter (1<=Acc<=10). The default value for Acc is 1. If Acc is less than 1 or greater than 10 it sets automatically to 2. Larger value of Acc increase density of mesh used in computations. Accuracy of model parameters may gain slightly from increasing Acc at the expense of noticeable growth of computation time. As a rule of thumb, good trade-off between accuracy and computation time is to set Acc to 1.

The parameter CPWSUB specifies the substrate element, which defines additional cross sectional parameters of the transmission line and height of the optional metallic cover. If blank, a default is used.

This component can have metallic grounded shield backing the substrate and metallic cover. Presence of backing shield is controlled by indicator Sbi. If Sbi is set to 0, backing shield is absent; if Sbi is set to 1, backing shield is present. Presence of cover is controlled by indicator Shi. If Shi is set to 0, cover is absent; if Shi is set to 1, cover is present.

This component doesn't impose restrictions on the conductor thickness (thickness may be zero, positive, or negative). Negative thickness means that the conductor is recessed into the substrate.

The component accounts for losses in metal and in substrate dielectric. Dispersion is not included.

NOTE: The implementation of EM Quasi-Static models heavily relies on the involved numerical algorithms. This may lead to the noticeable increase of computation time for some schematics that employ many models of this kind.

Layout

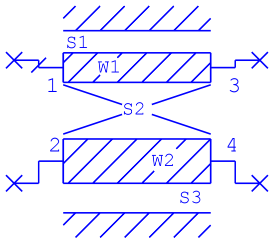
This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

References

[1]M.B. Bazdar, A.R. Djordjevic, R.F. Harrington, and T.K. Sarkar, "Evaluation of quasi-static matrix parameters for multiconductor transmission lines using Galerkin's method," IEEE Trans. Microwave Theory Tech., vol. MTT-42, July 1994, pp. 1223-1228

2 Asymmetric Coupled Lines (EM Quasi-Static): CPW2LINA

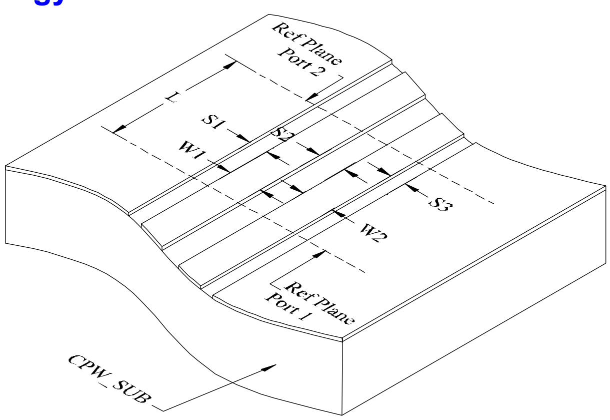
Symbol



Summary

CPW2LINA models a section of two asymmetric coupled coplanar waveguides (strip widths may be unequal and gaps between each strip conductor and closest lateral ground may have unequal width) on dielectric substrate. This model accounts for an optional metallic cover, an optional backing ground plane, and allows for arbitrary metal thickness of the signal conductor and lateral ground planes.

Topology



Parameters

Name	Description	Unit Type	Default
ID	Name	Text	CP1
W1	Conductor 1 width - nodes 1&3	Length	W[a]
W2	Conductor 2 width - nodes 2&4	Length	W[a]
S1	Gap width Gnd & conductor 1	Length	W[a]
S2	Gap width conductor 1&2	Length	W[a]
S3	Gap width conductor 1 & Gnd	Length	W [a]
L	Conductor length	Length	L[a]
Acc	Accuracy parameter		1
CPW_SUB	Substrate definition	Text	[b]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory.

[b] If only one CPW_SUB is present in the schematic, this substrate is automatically used. If multiple CPW_SUB substrate definitions are present, the user must specify.

Parameter Details

CPW_SUB. Supplies parameters for dielectric substrate, conductor thickness, conductor metal properties, the presence/absence of metallic cover/backing, and the cover height over the substrate. The Cover and Gnd parameters allow the addition/elimination of infinite metallic plates acting as a cover or grounded backing. The CPW2LINA model does not use the following CPW_SUB parameters: Hab, ER_Nom, H_Nom, Hcov_Nom, Hab_Nom, and T_Nom.

Acc. The accuracy parameter. The default value for Acc is 1. If Acc is less than 1 or greater than 10 it is set automatically to 2.

Parameter Restrictions and Recommendations

1. The Acc parameter A is limited to 1<Acc>10. A larger value of Acc increases the density of mesh used in computations. The accuracy of model parameters may improve slightly by increasing Acc, at the expense of a noticeably longer computation time. Generally, a good trade-off between accuracy and computation time is to set Acc to 1.
2. This model does not impose restrictions on the conductor thickness (thickness may be zero, positive, or negative). Negative thickness means that the conductor is recessed into the substrate.

Implementation Details

1. Model implementation is based on the EM Quasi-Static technique described in [1] [4–11]. It accounts for losses in metal and in substrate dielectric. Dispersion is partly included.
2. Setting Gnd=0 implies that the substrate is not backed by a perfect conductor, but is bounded by infinite air space. Modeling results are strongly affected by thin substrate heights and might differ substantially from modeling results obtained from models that implement the common conception of coplanar waveguide. To model classic coupled coplanar waveguides featuring infinitely thick substrate, set Gnd=0 and H >2(W1+W2+S1+S2+S3), where Gnd and H are parameters of CPW_SUB.
3. To apply Method of Moments for analysis, a quasi-static model creates 1D mesh covering contours of all conductors. The mesh is made of linear segments (pulses) of varying length. The length of a pulse is relatively big at the conductor center; it decreases toward conductor edges to reveal the charge distribution across conductor. If the conductor width is too large it may cause the pulse size to approach zero for pulses close to edge. In these rare cases the model may display a “Length of pulse #nnn equal to zero” error message.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See “Cell Options Dialog Box: Layout Tab ” for Cell Options dialog box **Layout** tab details.

CPW elements have special configurations for the defined line types. The center conductor geometry draws on all the layers defined in the line type. The spacing to the ground plane is then drawn on negative layers with the same name as

all of the layers in the line type. You must then draw the same layers on the positive layer to complete CPW layout. See "Negative Layers " for more information on setting up processes for positive and negative layers.

See "The Layout Process File (LPF)" for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

Recommendations for Use

NOTE: The implementation of EM Quasi-Static models heavily relies on the involved numerical algorithms. This may lead to a noticeable increase in simulation time for schematics that employ many such models.

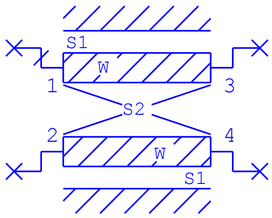
If the thickness of any layer is too small in comparison with the thickness of another layer, simulation time may also noticeably grow.

References

[1] M.B. Bazdar, A.R. Djordjevic, R.F. Harrington, and T.K. Sarkar, "Evaluation of quasi-static matrix parameters for multiconductor transmission lines using Galerkin's method," IEEE Trans. Microwave Theory Tech., vol. MTT-42, July 1994, pp. 1223-1228.

2 Coupled Lines (EM Quasi-Static): CPW2LINE

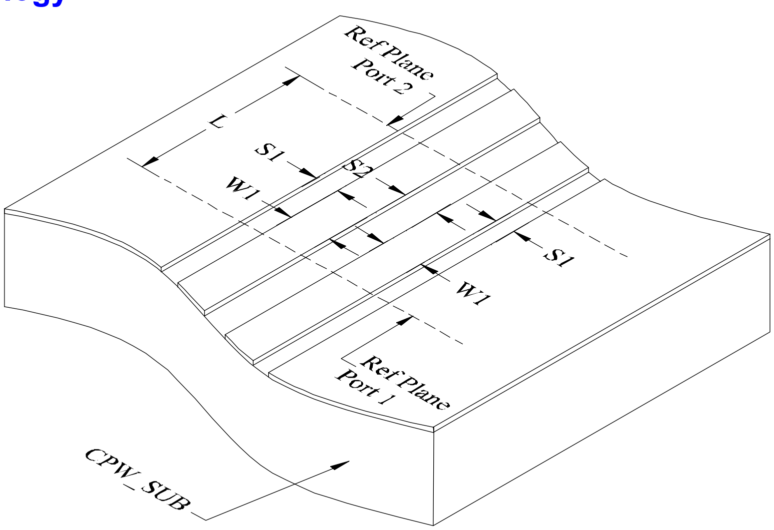
Symbol



Summary

CPW2LINE models a section of two symmetric coupled coplanar waveguides (the strip widths are equal and gaps between each strip conductor and the closest lateral ground have equal width) on dielectric substrate. This model accounts for an optional metallic cover, an optional backing ground plane, and allows for arbitrary metal thickness of the signal conductor and lateral ground planes.

Topology



Parameters

Name	Description	Unit Type	Default
ID	Name	Text	CP1
W1	Conductor widths	Length	W[a]
S1	Gap width to ground	Length	W[a]
S2	Gap width between conductors	Length	W[a]
L	Conductor length	Length	L[a]
Acc	Accuracy parameter		1
CPW_SUB	Substrate definition	Text	[b]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory.

[b]If only one CPW_SUB is present in the schematic, this substrate is automatically used. If multiple CPW_SUB substrate definitions are present, the user must specify.

Parameter Details

CPW_SUB. Supplies parameters for dielectric substrate, conductor thickness, conductor metal properties, the presence/absence of metallic cover/backing, and the cover height over the substrate. The Cover and Gnd parameters allow the addition/elimination of infinite metallic plates acting as a cover or grounded backing. The CPW2LINE model does not use the following CPW_SUB parameters: Hab, ER_Nom, H_Nom, Hcov_Nom, Hab_Nom, and T_Nom.

Acc. The accuracy parameter. The default value for Acc is 1. If Acc is less than 1 or greater than 10 it is set automatically to 2.

Parameter Restrictions and Recommendations

1. The Acc parameter A is limited to 1<Acc>10. A larger value of Acc increases the density of mesh used in computations. The accuracy of model parameters may improve slightly by increasing Acc, at the expense of a noticeably longer computation time. Generally, a good trade-off between accuracy and computation time is to set Acc to 1.
2. This model does not impose restrictions on the conductor thickness (thickness may be zero, positive, or negative). Negative thickness means that the conductor is recessed into the substrate.

Implementation Details

1. Model implementation is based on the EM Quasi-Static technique described in [1] [4–14]. It accounts for losses in metal and in substrate dielectric. Dispersion is partly included.
2. Setting Gnd=0 implies that the substrate is not backed by a perfect conductor, but is bounded by infinite air space. Modeling results are strongly affected by thin substrate heights and might differ substantially from modeling results obtained from models that implement the common conception of coplanar waveguide. To model classic coupled coplanar waveguides featuring infinitely thick substrate, set Gnd=0 and H >2(W1+S1+S2+S3), where Gnd and H are parameters of CPW_SUB.
3. To apply Method of Moments for analysis, a quasi-static model creates 1D mesh covering contours of all conductors. The mesh is made of linear segments (pulses) of varying length. The length of a pulse is relatively big at the conductor center; it decreases toward conductor edges to reveal the charge distribution across conductor. If the conductor width is too large it may cause the pulse size to approach zero for pulses close to edge. In these rare cases the model may display a “Length of pulse #nnn equal to zero” error message.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See “Cell Options Dialog Box: Layout Tab ” for Cell Options dialog box **Layout** tab details.

CPW elements have special configurations for the defined line types. The center conductor geometry draws on all the layers defined in the line type. The spacing to the ground plane is then drawn on negative layers with the same name as

all of the layers in the line type. You must then draw the same layers on the positive layer to complete CPW layout. See "Negative Layers " for more information on setting up processes for positive and negative layers.

See "The Layout Process File (LPF)" for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

Recommendations for Use

NOTE: The implementation of EM Quasi-Static models heavily relies on the involved numerical algorithms. This may lead to a noticeable increase in simulation time for schematics that employ many such models.

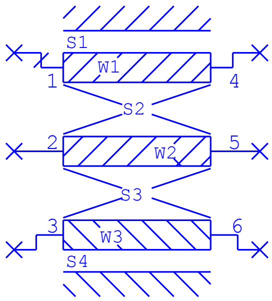
If the thickness of any layer is too small in comparison with the thickness of another layer, simulation time may also noticeably grow.

References

[1] M.B. Bazdar, A.R. Djordjevic, R.F. Harrington, and T.K. Sarkar, "Evaluation of quasi-static matrix parameters for multiconductor transmission lines using Galerkin's method," IEEE Trans. Microwave Theory Tech., vol. MTT-42, July 1994, pp. 1223-1228.

(Obsolete) 3 Coupled Coplanar Lines (EM Quasi-Static): CPW3LIN

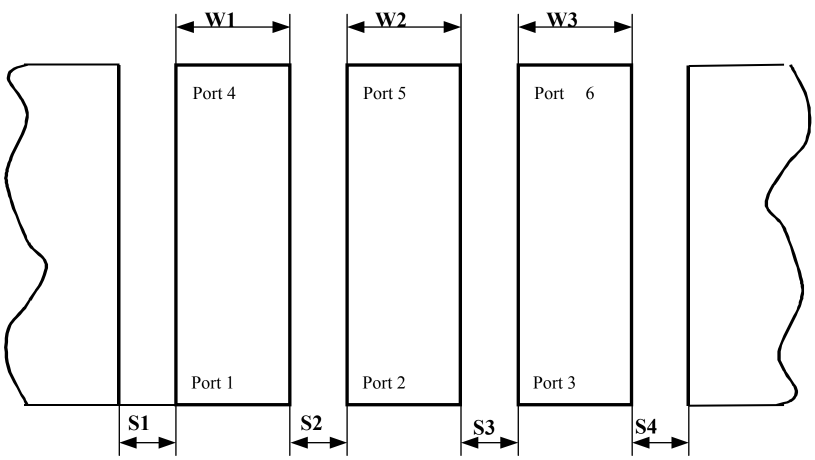
Symbol



Summary

This element is OBSOLETE and is replaced by the 3 Asymmetric Coupled Lines (EM Quasi-Static) (CPW3LINA) element.

Topology



Parameters

Name	Description	Unit Type	Default
ID	Name	Text	TL1
W1	Conductor 1width - nodes 1&4	Length	0 um
W2	Conductor 2 width - nodes 2&5	Length	0 um
W3	Conductor 3 width - nodes 3&6	Length	0 um
S1	Gap Width - Gnd & conductor 1	Length	0 um
S2	Gap Width - conductor 1&2	Length	0 um
S3	Gap Width - conductor 2 &3	Length	0 um
S4	Gap Width - conductor 3&Gnd	Length	0 um
ShI	Cover shield presence indicator		0

Name	Description	Unit Type	Default
SbI	Backing shield presence indicator		0
L	Physical length	Length	0 um
Acc	Accuracy parameter		1
CPWSUB	Substrate definition	Text	

Implementation Details

This circuit component models a section of three coupled coplanar waveguides on a shielded substrate. The parameters W1, W2, W3 (conductors' widths), S1, S4 (S1 - gap between conductor 1 and the left ground halfplane; S4 - gap between conductor 3 and the right ground halfplane), S2 and S3 (gaps between conductors), and L (conductor length) are dimensions entered in the default length units. Left and right directions are defined in respect to the line of sight from port 1 to port 4.

The parameter Acc is the accuracy parameter (1<=Acc<=10). The default value for Acc is 1. If Acc is less than 1 or greater than 10 it sets automatically to 2. Larger value of Acc increase density of mesh used in computations. Accuracy of model parameters may gain slightly from increasing Acc at the expense of noticeable growth of computation time. As a rule of thumb, good trade-off between accuracy and computation time is to set Acc to 1.

The parameter CPWSUB specifies the substrate element, which defines additional cross sectional parameters of the transmission line and height of the optional metallic cover. If blank, a default is used.

This component can have metallic grounded shield backing the substrate and metallic cover. Presence of backing shield is controlled by indicator Sbi. If Sbi is set to 0, backing shield is absent; if Sbi is set to 1, backing shield is present. Presence of cover is controlled by indicator Shi. If Shi is set to 0, cover is absent; if Shi is set to 1, cover is present.

This component doesn't impose restrictions on the conductor thickness (thickness may be zero, positive, or negative). Negative thickness means that the conductor is recessed into the substrate.

The component accounts for losses in metal and in substrate dielectric. Dispersion is not included.

NOTE: The implementation of EM Quasi-Static models heavily relies on the involved numerical algorithms. This may lead to the noticeable increase of computation time for some schematics that employ many models of this kind.

Layout

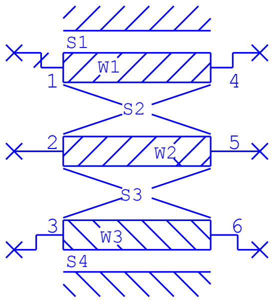
This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

References

M.B. Bazdar, A.R. Djordjevic, R.F. Harrington, and T.K. Sarkar, "Evaluation of quasi-static matrix parameters for multiconductor transmission lines using Galerkin's method," IEEE Trans. Microwave Theory Tech., vol. MTT-42, July 1994, pp. 1223-1228

3 Asymmetric Coupled Lines (EM Quasi-Static): CPW3LINA

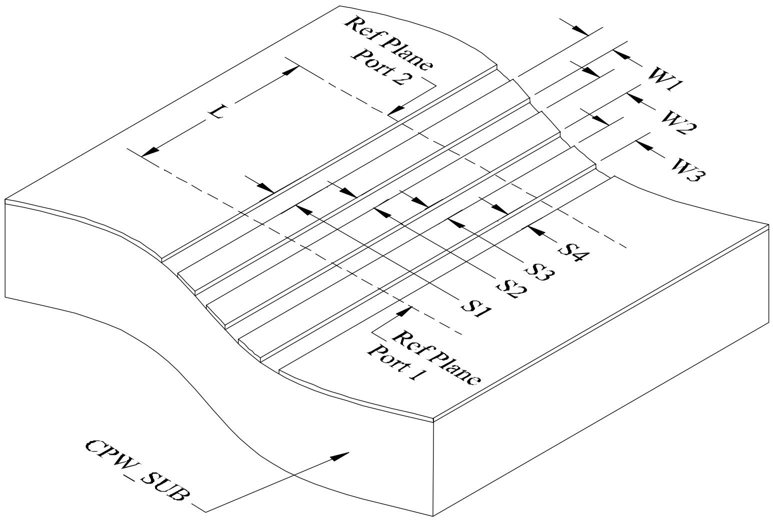
Symbol



Summary

CPW3LINA models a section of three asymmetric coupled coplanar waveguides (the strip widths may be unequal and gaps between each strip conductor and closest lateral ground, as well as gaps between conductors may have unequal width) on dielectric substrate. This model accounts for an optional metallic cover, an optional backing ground plane, and allows for arbitrary metal thickness of the signal conductor and lateral ground planes.

Topology



Parameters

Name	Description	Unit Type	Default
ID	Name	Text	CP1
W1	Conductor 1 width-nodes 1&4	Length	W[a]
W2	Conductor 2 width-nodes 2&5	Length	W[a]
W3	Conductor 3 width-nodes 3&6	Length	W[a]
S1	Gap Width Gnd & conductor 1	Length	W[a]
S2	Gap width conductor 1&2	Length	W[a]

Name	Description	Unit Type	Default
S3	Gap width conductor 2&3	Length	W[a]
S4	Gap width conductor 3 & Gnd	Length	W[a]
L	Conductor length	Length	L[a]
Acc	Accuracy parameter		1
CPW_SUB	Substrate definition	Text	[b]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory.

[b] If only one CPW_SUB is present in the schematic, this substrate is automatically used. If multiple CPW_SUB substrate definitions are present, the user must specify.

Parameter Details

CPW_SUB. Supplies parameters for dielectric substrate, conductor thickness, conductor metal properties, the presence/absence of metallic cover/backing, and the cover height over the substrate. The Cover and Gnd parameters allow the addition/elimination of infinite metallic plates acting as a cover or grounded backing. The CPW3LINA model does not use the following CPW_SUB parameters: Hab, ER_Nom, H_Nom, Hcov_Nom, Hab_Nom, and T_Nom.

Acc. The accuracy parameter. The default value for Acc is 1. If Acc is less than 1 or greater than 10 it is set automatically to 2.

Parameter Restrictions and Recommendations

1. The Acc parameter A is limited to 1<Acc>10. A larger value of Acc increases the density of mesh used in computations. The accuracy of model parameters may improve slightly by increasing Acc, at the expense of a noticeably longer computation time. Generally, a good trade-off between accuracy and computation time is to set Acc to 1.
2. This model does not impose restrictions on the conductor thickness (thickness may be zero, positive, or negative). Negative thickness means that the conductor is recessed into the substrate.

Implementation Details

1. Model implementation is based on the EM Quasi-Static technique described in [1] [4–19]. It accounts for losses in metal and in substrate dielectric. Dispersion is partly included.
2. Setting Gnd=0 implies that the substrate is not backed by a perfect conductor, but is bounded by infinite air space. Modeling results are strongly affected by thin substrate heights and might differ substantially from modeling results obtained from models that implement the common conception of coplanar waveguide. To model classic coupled coplanar waveguides featuring infinitely thick substrate, set Gnd=0 and H >2(W1+S1+S2+S3), where Gnd and H are parameters of CPW_SUB.
3. To apply Method of Moments for analysis, a quasi-static model creates 1D mesh covering contours of all conductors. The mesh is made of linear segments (pulses) of varying length. The length of a pulse is relatively big at the conductor center; it decreases toward conductor edges to reveal the charge distribution across conductor. If the conductor width is too large it may cause the pulse size to approach zero for pulses close to edge. In these rare cases the model may display a “Length of pulse #nnn equal to zero” error message.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.

2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See “Cell Options Dialog Box: Layout Tab ” for Cell Options dialog box **Layout** tab details.

CPW elements have special configurations for the defined line types. The center conductor geometry draws on all the layers defined in the line type. The spacing to the ground plane is then drawn on negative layers with the same name as all of the layers in the line type. You must then draw the same layers on the positive layer to complete CPW layout. See “Negative Layers ” for more information on setting up processes for positive and negative layers.

See “The Layout Process File (LPF)” for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

Recommendations for Use

NOTE: The implementation of EM Quasi-Static models heavily relies on the involved numerical algorithms. This may lead to a noticeable increase in simulation time for schematics that employ many such models.

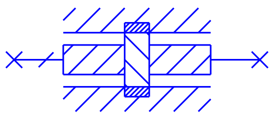
If the thickness of any layer is too small in comparison with the thickness of another layer, simulation time may also noticeably grow.

References

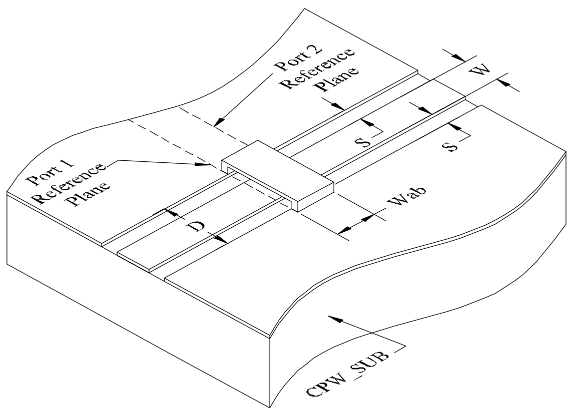
[1] M.B. Bazdar, A.R. Djordjevic, R.F. Harrington, and T.K. Sarkar, "Evaluation of quasi-static matrix parameters for multiconductor transmission lines using Galerkin's method," IEEE Trans. Microwave Theory Tech., vol. MTT-42, July 1994, pp. 1223-1228.

Coplanar Air Bridge (EM-Based): CPWABRGX

Symbol



Topology



Parameters

CPWABRGX$ is a co-planar iCell. See the Microwave Office Layout Guide for more information.

Name	Description	Unit Type	Default
ID	Element ID	Text	CP1
W	Conductor width	Length	W[a]
S	Gap width	Length	W[a]
Wab	AirBridge conductor width	Length	W[a]
CPW_SUB	Substrate definition	Text	[b]
*Autofill	Autofill database if not equal to 0		0

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory.

[b] If only one CPW_SUB is present in the schematic, this substrate is automatically used. If multiple CPW_SUB substrate definitions are present, the user must specify.

** indicates a secondary parameter*

Parameter Restrictions and Recommendations

0.25<=(W+2S)/HSub<=4.0

0.1<=W/(W+2S)<=0.9

0.1<=Wab/(W+2S)<=0.5

Implementation Details

This circuit component models a co-planar waveguide airbridge which connects the ground planes on either side of a transmission line. The purpose of this feature is to suppress the excitation of a slot line mode between the two ground planes. It is highly recommended that airbridges be placed periodically along long lengths of lines and on either side of an unsymmetrical discontinuity. This model is constructed as an X-model (Table Based Interpolation) using results automatically generated by EMSight and the CPWLINX model. For a more detailed discussion of the X-models see " EM-based Models (X-models) " . Importantly, an equivalent circuit model is constructed for the discontinuity based upon the EMSight simulation. As the dispersion of a co-planar waveguide is minimal, values of the equivalent circuit parameters in the model are assumed frequency independent.

The input parameters W,S and Wab as defined in "Topology" are entered in the default length units.

CPW_SUB allows you to specify the co-planar waveguide substrate.

The AutoFill parameter is a hidden input which allows you to specify that the entire interpolation table should be filled automatically at the current fixed values.

Independent Parameters:	W, S, Wab
Scalable Parameters:	Frequency
Fixed Parameters:	CPW_SUB (Cover, Gnd, Er_Nom, H_Nom, Hcov_Nom, Hab_Nom, T_Nom)
Statistical Parameters:	CPW_SUB(Er, Hab, T, Hsub, Hcov)

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See "Cell Options Dialog Box: Layout Tab " for Cell Options dialog box **Layout** tab details.

CPW elements have special configurations for the defined line types. The center conductor geometry draws on all the layers defined in the line type. The spacing to the ground plane is then drawn on negative layers with the same name as all of the layers in the line type. You must then draw the same layers on the positive layer to complete CPW layout. See "Negative Layers " for more information on setting up processes for positive and negative layers.

See "The Layout Process File (LPF)" for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

Recommendations for Use

The CPWABRGX model creates (and uses for modeling) three files with data tables during autofill: Two files with a CPTX prefix for coplanar waveguides, and one file with a CPABG prefix that contains CPWABRGX autofill results.

CPTX files are intermediate data tables for coplanar line that CPWABRGX needs. for work; CPABG files are model final data tables.

CPTX and CPABG files created by the CPWLINX, CPWTEEX, and CPWBENDX models may also qualify for the CPWABRGX model. If the model finds that the CPTX and/or CPABG files were created with the same set of fixed parameters, it does not generate new files, it uses the existing ones for modeling.

Note that all fixed parameters are nominal (prefixed with _Nom) parameters of CPW_SUB. Modification of fixed parameters may instigate autofill of all data files mentioned above (change of T_Nom affects only CPTX autofill).

Usually, for all X-models, if parameter Autofill is set to 0 and X-model finds that it needs to create new data table, X-model issues a request to user to approve this request by changing Autofill to 1. Unlike other X-models, if CPWABRGX parameter Autofill is set to 0 and user changes some fixed parameter than autofill of CPTX files may start automatically. Generally, this calculation does not take much time and upon completion CPWABRGX will request user to set parameter Autofill. By this time, new CPTX files are ready for use and CPWABRGX may proceed with its own autofill.

Example projects for autofilling an EM-based model are provided in the */Examples/Circuit Features/Xmodels* subdirectory of the NI AWR program directory.

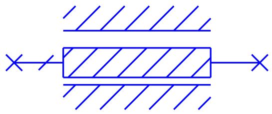
NOTE: You must initiate an autofill of the EM data table after making any changes to nominal parameters of CPW_SUB element.

References

This model was developed under research performed at NI AWR Corporation. The details of the implementation are considered proprietary in nature.

Asymmetric Coplanar Line (EM Quasi-Static): CPWALINE

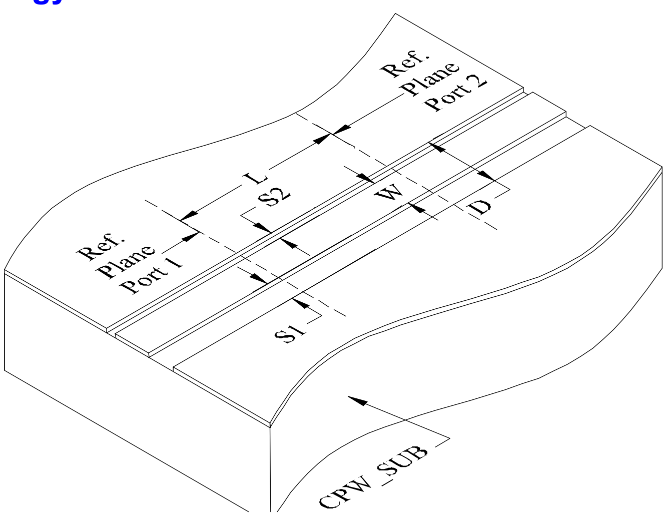
Symbol



Summary

CPWALINE models a section of asymmetric coplanar waveguide (gaps between the strip conductor and lateral grounds may have an unequal width) on a dielectric substrate. This model accounts for an optional metallic cover, an optional backing ground plane, and allows for an arbitrary metal thickness of a signal conductor and lateral ground planes.

Topology



Parameters

Name	Description	Unit Type	Default
ID	Name	Text	CP1
W	Conductor width	Length	W[a]
S1	Left gap width	Length	W[a]
S2	Right gap width	Length	W[a]
L	Conductor length	Length	L[b]
Acc	Accuracy parameter		1
CPW_SUB	Substrate definition	Text	[c]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory.

[b]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory.

[c] If only one CPW_SUB is present in the schematic, this substrate is automatically used. If multiple CPW_SUB substrate definitions are present, the user must specify.

Parameter Details

CPW_SUB. Supplies parameters for dielectric substrate, conductor thickness, conductor metal properties, the presence/absence of metallic cover/backing, and the cover height over the substrate. The Cover and Gnd parameters allow the addition/elimination of infinite metallic plates acting as a cover or grounded backing. The CPW1LINE model does not use the following CPW_SUB parameters: Hab, ER_Nom, H_Nom, Hcov_Nom, Hab_Nom, and T_Nom.

Acc. The accuracy parameter. The default value for Acc is 1. If Acc is less than 1 or greater than 10 it is set automatically to 2.

Parameter Restrictions and Recommendations

1. The Acc parameter A is limited to 1 < Acc > 10. A larger value of Acc increases the density of mesh used in computations. The accuracy of model parameters may improve slightly by increasing Acc, at the expense of a noticeably longer computation time. Generally, a good trade-off between accuracy and computation time is to set Acc to 1.
2. This model does not impose restrictions on the conductor thickness (it may be zero, positive, or negative). Negative thickness means that the conductor is recessed into the substrate.

Implementation Details

1. Model implementation is based on the EM Quasi-Static technique described in [1] [4–25]. It accounts for losses in metal and in the substrate dielectric. Dispersion is partly included.
2. Setting Gnd=0 implies that the substrate is not backed by a perfect conductor, but is bounded by infinite air space. Modeling results are strongly affected by thin substrate heights and might differ substantially from modeling results obtained from models that implement the common conception of a coplanar waveguide (for example, CPWLINE). To model a classic coplanar waveguide featuring an infinitely thick substrate, set Gnd=0 and H >2(W+2S), where Gnd and H are parameters of CPW_SUB.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See "Cell Options Dialog Box: Layout Tab " for Cell Options dialog box **Layout** tab details.

CPW elements have special configurations for the defined line types. The center conductor geometry draws on all the layers defined in the line type. The spacing to the ground plane is then drawn on negative layers with the same name as all of the layers in the line type. You must then draw the same layers on the positive layer to complete CPW layout. See "Negative Layers " for more information on setting up processes for positive and negative layers.

See "The Layout Process File (LPF)" for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

Recommendations for Use

NOTE: The implementation of EM Quasi-Static models heavily relies on the involved numerical algorithms. This may lead to a noticeable increase in simulation time for schematics that employ many such models.

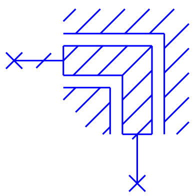
If the thickness of any layer is too small in comparison with the thickness of another layer, simulation time may also noticeably grow.

References

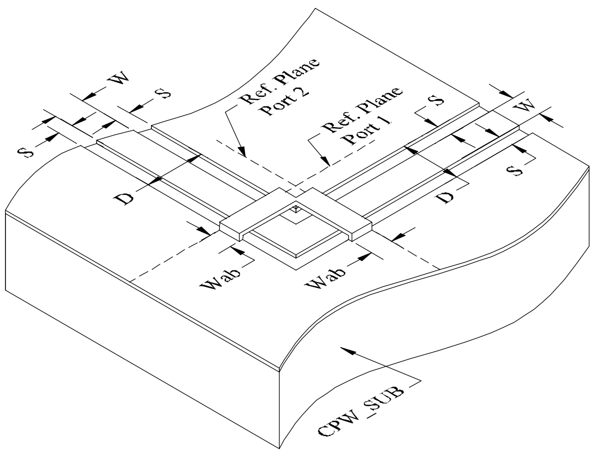
[1] M.B. Bazdar, A.R. Djordjevic, R.F. Harrington, and T.K. Sarkar, "Evaluation of quasi-static matrix parameters for multiconductor transmission lines using Galerkin's method," IEEE Trans. Microwave Theory Tech., vol. MTT-42, July 1994, pp. 1223-1228.

Coplanar 90-Degree Bend (EM Based): CPWBENDX

Symbol



Topology



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	CP1
W	Conductor width	Length	W[a]
S	Gap width	Length	W[a]
Wab	AirBridge conductor width @ port 1&2	Length	W[a]
CPW_SUB	Substrate definition	Text	[b]
*AutoFill	AutoFill data base if not equal to 0		0

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory.

[b]If only one CPW_SUB is present in the schematic, this substrate is automatically used. If multiple CPW_SUB substrates are present, the user must specify.

** indicates a secondary parameter*

Parameter Restrictions and Recommendations

1. 0.25<=(W+2S)/HSub<=4.0
2. 0.1<=W/(W+2S)<=0.9
3. 0.1<=Wab/(W+2S)<=0.5

Implementation Details

This circuit component models a co-planar waveguide 90 deg bend including the effects of airbridges which connect the ground planes on either side of a discontinuity. The purpose of these features is to suppress the excitation of a slot line mode between the two ground planes. This model is constructed as an X-model (Table Based Interpolation) using results automatically generated by EMSight and theCPWLINX,CPWABRG models. For a more detailed discussion of the X-models see " EM-based Models (X-models) ". Importantly, an equivalent circuit model is constructed for the discontinuity based upon the EMSight simulation. As the dispersion of co-planar waveguide is minimal, values of the equivalent circuit parameters in the model are assumed frequency independent.

The input parameters W,S and Wab as defined in "Topology" are entered in the default length units.

CPW_SUB allows you to specify the co-planar waveguide substrate.

The AutoFill parameter is a hidden input which allows you to specify that the entire interpolation table should be filled automatically at the current fixed values.

Independent Parameters:	W, S, Wab
Scalable Parameters:	Frequency
Fixed Parameters:	ACCUR, CPW_SUB(Tand, Rho, Is Grounded, Has Cover)
Statistical Parameters:	CPW_SUB(Er, Hab, T, Hsub, Hcov)

Recommendations for Use

The CPWBENDX model creates (and uses for modeling) four files with data tables during autofill: Two files with a CPTX prefix for coplanar waveguides, one file with a CPABG prefix for air bridge, and one file with a CP90B prefix that contains CPWBENDX autofill results. The CPTX and CPABG files created by the CPWLINX, CPWTEEX, and CPWABRGX models may also qualify for the CPWBENDX model. If a model finds that the CPTX and/or CPABG files were created with the same set of fixed parameters, it does not generate new files, but uses the existing ones for modeling.

Example projects for autofilling an EM-based model are provided in the */Examples/Circuit Features/Xmodels* subdirectory of the NI AWR program directory.

NOTE: You must initiate an autofill of the EM data table after making any changes to the CPW_SUB element.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See "Cell Options Dialog Box: Layout Tab " for Cell Options dialog box **Layout** tab details.

CPW elements have special configurations for the defined line types. The center conductor geometry draws on all the layers defined in the line type. The spacing to the ground plane is then drawn on negative layers with the same name as

all of the layers in the line type. You must then draw the same layers on the positive layer to complete CPW layout. See "Negative Layers " for more information on setting up processes for positive and negative layers.

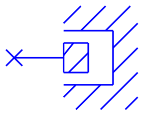
See "The Layout Process File (LPF)" for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

References

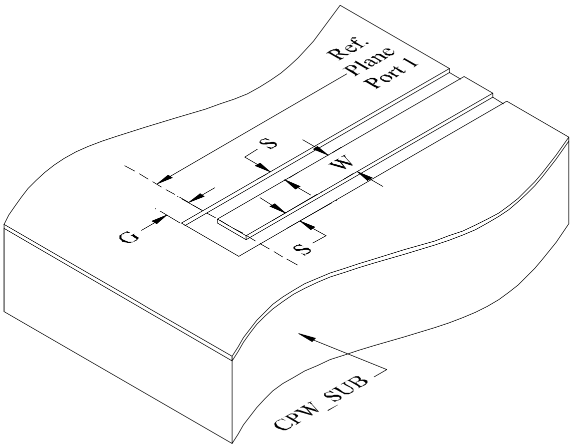
This model was developed under research performed at NI AWR Corporation. The details of the implementation are considered proprietary in nature.

Coplanar Open, with Ground Connected (Closed Form): CPWEG

Symbol



Topology



Parameters

CPWEG$ is a Co-Planar iCell. See the Microwave Office Layout Guide for more information.

Name	Description	Unit Type	Default
ID	Element ID	Text	CP1
W	Conductor width	Length	W[a]
S	Gap width	Length	W[a]
G	Gap width from TxLine end	Length	W[a]
CPW_SUB	Substrate definition	Text	[b]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory.

[b] If only one CPW_SUB is present in the schematic, this substrate is automatically used. If multiple CPW_SUB substrate definitions are present, the user must specify.

Implementation Details

This circuit component models a gap termination of a strip of coplanar waveguide. Model doesn't account for substrate thickness or presence of metallic cover or backing ground. The parameters W (strip width), S (S - spacing between strip and the left/right ground halfplane) and G (gap length) are dimensions entered in the default length units. The parameter CPW_SUB specifies the substrate element, which defines additional cross sectional parameters of the transmission line. If blank, a default is used. Some parameters CPW_SUB, that are Hcover, Cover, and Gnd, are not used by this component. Conductor thickness is assumed zero.

The component doesn't account for losses in metal and in substrate dielectric. Dispersion is not included.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See “Cell Options Dialog Box: Layout Tab ” for Cell Options dialog box **Layout** tab details.

CPW elements have special configurations for the defined line types. The center conductor geometry draws on all the layers defined in the line type. The spacing to the ground plane is then drawn on negative layers with the same name as all of the layers in the line type. You must then draw the same layers on the positive layer to complete CPW layout. See “Negative Layers ” for more information on setting up processes for positive and negative layers.

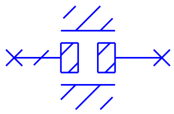
See “The Layout Process File (LPF)” for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

References

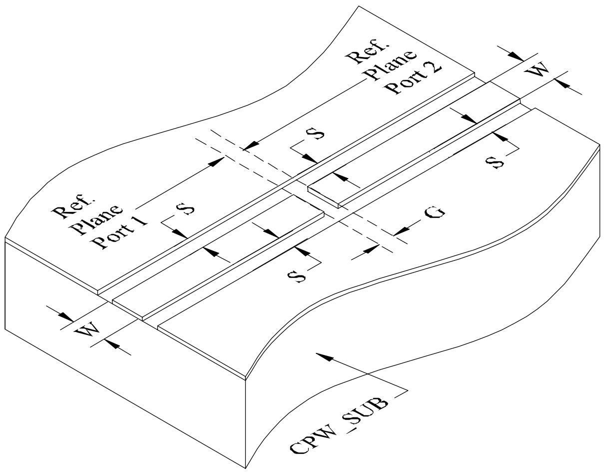
[1] M-H. Mao, R-B.Wu, C-H Chen et al, "Characterization of coplanar waveguide open end capacitance - theory and experiment." IEEE Trans. on MTT, v. 42, No.6, June 1994, pp. 1016-1024.

Coplanar Gap in Center Conductor (Closed Form): CPWGAP

Symbol



Topology



Parameters

CPWGAP$ is a Co-Planar iCell and only has a G parameter. See the Microwave Office Layout Guide for more information.

Name	Description	Unit Type	Default
ID	Element ID	Text	CP1
W	Conductor width	Length	W[a]
S	Spacing between ground and center conductor	Length	W[a]
G	Gap width in center conductor	Length	W[a]
CPW_SUB	Substrate definition	Text	[b]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory.

[b] If only one CPW_SUB is present in the schematic, this substrate is automatically used. If multiple CPW_SUB substrate definitions are present, the user must specify.

Implementation Details

This circuit component models a gap in a strip of coplanar waveguide. Model doesn't account for substrate thickness or presence of metallic cover or backing ground. The parameters W (strip width), S (S - spacing between strip and the left/right ground halfplane) and G (gap length) are dimensions entered in the default length units. Model is represented by capacitive π-circuit. The parameter CPW_SUB specifies the substrate element, which defines additional cross sectional parameters of the transmission line. If blank, a default is used. Some parameters CPW_SUB, that are Hcover, Cover, and Gnd, are not used by this component. Conductor thickness is assumed zero.

The component doesn't account for losses in metal and in substrate dielectric. Dispersion is not included.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See "Cell Options Dialog Box: Layout Tab " for Cell Options dialog box **Layout** tab details.

CPW elements have special configurations for the defined line types. The center conductor geometry draws on all the layers defined in the line type. The spacing to the ground plane is then drawn on negative layers with the same name as all of the layers in the line type. You must then draw the same layers on the positive layer to complete CPW layout. See "Negative Layers " for more information on setting up processes for positive and negative layers.

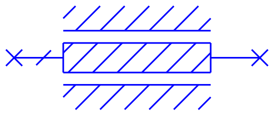
See "The Layout Process File (LPF)" for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

References

[1] S.Gevorgian, A.Deleniv, T.Martinsson et al, "CAD model of a gap in a coplanar waveguide," Int. Journal of Microwave and Millim-Wave CAE, vol. 6, No.5, pp. 369-377.

(Obsolete) Coplanar Waveguide Line (Closed Form): CPWLIN

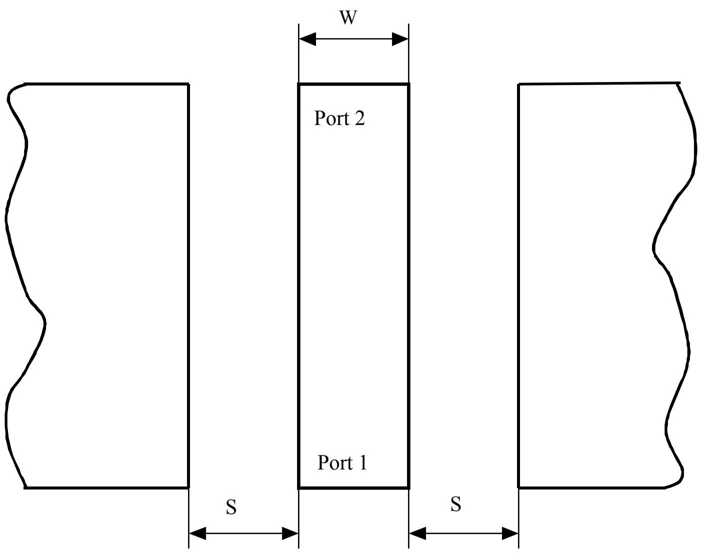
Symbol



Summary

This element is OBSOLETE and is replaced by the Single Conductor Coplanar Line (EM Based Quasi-Static) (CPWLINX), Coplanar Line (EM Quasi-Static) (CPW1LINE), or Coplanar Line (Closed Form) CPWLINE) element.

Topology



Parameters

Name	Description	Unit Type	Default
ID	Name	Text	TL1
W	Conductor width	Length	0 um
S	Gap width	Length	0 um
L	Conductor length	Length	0 um
CPWSUB	Substrate definition	Text	

Implementation Details

This circuit component models a section of coplanar waveguide. The parameters W (strip width), S (S - gap between strip and the ground halfplanes), and L (line length) are dimensions entered in the default length units. The parameter CPWSUB specifies the substrate element, which defines additional cross sectional parameters of the transmission line and height of the optional metallic cover. If blank, a default is used.

This component accounts for dispersion and losses in metal and in substrate dielectric.

NOTE: Parameter HC (cover height above substrate - from CPWSUB substrate element) is not used in this model.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See "Cell Options Dialog Box: Layout Tab " for Cell Options dialog box **Layout** tab details.

CPW elements have special configurations for the defined line types. The center conductor geometry draws on all the layers defined in the line type. The spacing to the ground plane is then drawn on negative layers with the same name as all of the layers in the line type. You must then draw the same layers on the positive layer to complete CPW layout. See "Negative Layers " for more information on setting up processes for positive and negative layers.

See "The Layout Process File (LPF)" for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

References

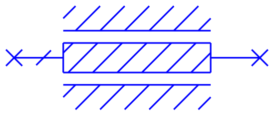
[1] W.Heinrich, "Quazi-TEM description of MMIC Coplanar Lines Including Conductor-Loss Effects," IEEE Trans. Microwave Theory Tech., vol. MTT-41, Jan. 1994, pp. 45-52.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Coplanar Line (Closed Form): CPWLINE

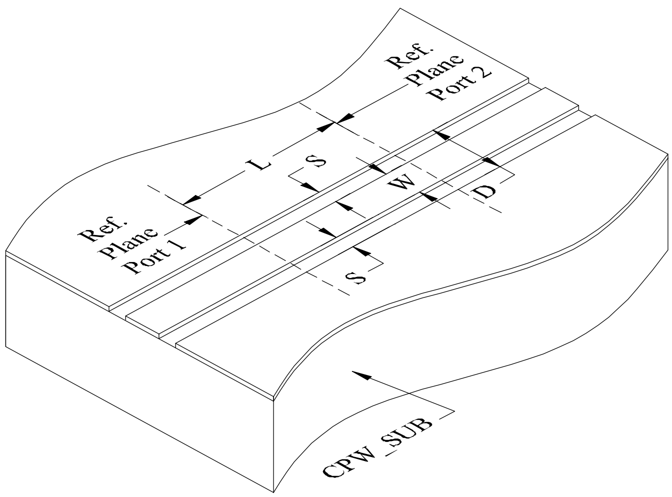
Symbol



Summary

CPWLINE models a section of coplanar waveguide on an infinitely thick dielectric substrate. This model implies that the coplanar waveguide is ungrounded (the backing ground plane, if present, is way below the signal strip and does not affect the modeling results).

Topology



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	CP1
W	Conductor width	Length	W[a]
S	Gap width	Length	W[a]
L	Conductor length	Length	L[a]
CPW_SUB	Substrate Definition	Text	[b]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory.

[b]If only one CPW_SUB is present in the schematic, this substrate is automatically used. If multiple CPW_SUB substrate definitions are present, the user must specify.

Parameter Details

CPW_SUB. Supplies parameters for dielectric substrate, conductor thickness, and conductor metal properties. The CPWLINE model does not use the following CPW_SUB parameters: Hcover, Cover, Gnd, Hab, ER_Nom, H_Nom, Hcov_Nom, Hab_Nom, and T_Nom. The H parameter is used only for validation purposes (see "Parameter Restrictions and Recommendations").

Parameter Restrictions and Recommendations

1. This model issues a warning if the following condition is violated: H<2(W+2S) (H, W, and S are parameters of CPW_SUB.). Actually, this approach implies that H is large compared to W+S, so the warning alerts you that the substrate height is too small, initial model assumptions are not met, and the accuracy may degrade.
2. The CPW1LINE model can be used as a (slower) alternative to CPWLINE, however, note that setting the substrate parameter Gnd=0 for CPW1LINE implies that a substrate is not backed by a perfect conductor, but is bounded by infinite air space. In this case modeling results are strongly affected by thin substrate heights and might differ substantially from the modeling results obtained from CPWLINE, that implements a common conception of coplanar waveguide. To match the results of these two models, set Gnd=0 and H >2(W+2S).

Implementation Details

This model implements a technique developed in [1] [4–36]; this approach implies that H is large, hence, actual modeling results **do not depend** on H.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See “Cell Options Dialog Box: Layout Tab ” for Cell Options dialog box **Layout** tab details.

CPW elements have special configurations for the defined line types. The center conductor geometry draws on all the layers defined in the line type. The spacing to the ground plane is then drawn on negative layers with the same name as all of the layers in the line type. You must then draw the same layers on the positive layer to complete CPW layout. See “Negative Layers ” for more information on setting up processes for positive and negative layers.

See “The Layout Process File (LPF)” for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

Recommendations for Use

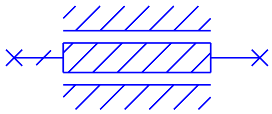
To account for finite H, the impact of a grounded backing plane and/or metallic cover, EM Quasi-Static model CPW1LINE should be used instead of CPWLINE.

References

[1] W. Heinrich, "Quasi-TEM Description of MMIC Coplanar Lines Including Conductor-Loss Effects," IEEE Trans. Microwave Theory Tech., vol. MTT-41, January 1993, pp. 45-52.

Single Conductor Coplanar Line (EM Based Quasi-Static): CPWLINX

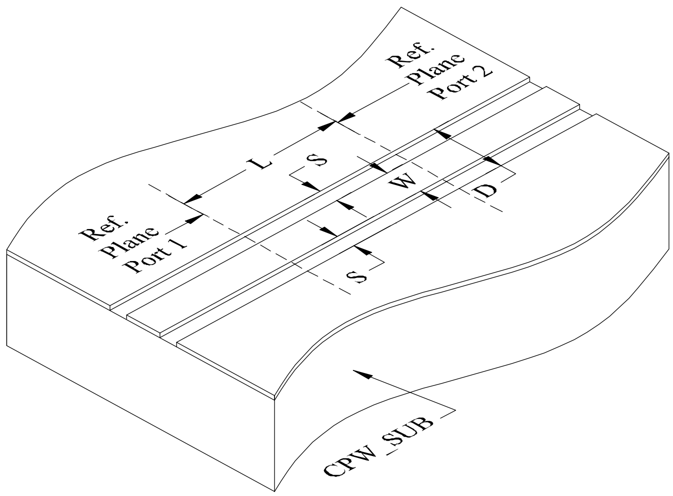
Symbol



Summary

CPWLINX models a section of symmetric coplanar waveguide (gaps between the strip conductor and lateral grounds have equal width) on a dielectric substrate. This model accounts for an optional metallic cover, an optional backing ground plane, and allows for the arbitrary metal thickness of signal conductor and lateral ground planes. This model is constructed as an X-model (Table Based Interpolation) using the results of a quasi-static cross-sectional analysis based upon the Method of Moments. For a more detailed discussion of the X-models see “ EM-based Models (X-models) ”.

Topology



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	CP1
W	Conductor width	Length	W[a]
S	Gap width	Length	W[a]
L	Conductor length	Length	L[a]
Acc	Accuracy parameter		10
CPW_SUB	Substrate definition	Text	[b]
*AutoFill	AutoFill dataBase if not equal to 0		0

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory.

[b] If only one CPW_SUB is present in the schematic, this substrate is automatically used. If multiple CPW_SUB substrate definitions are present, you must specify which to use.

** indicates a secondary parameter*

Parameter Details

CPW_SUB. Supplies parameters for dielectric substrate, conductor thickness, conductor metal properties, the presence/absence of metallic cover/backing, and the cover height over the substrate. The Cover and Gnd parameters allow the addition/elimination of infinite metallic plates acting as a cover or grounded backing. The CPWLINX model does not use the following CPW_SUB parameters: Hab, ER_Nom, H_Nom, Hcov_Nom, Hab_Nom, and T_Nom.

Acc. The accuracy parameter influencing the 2-D Quasi-Static Method of moment analysis. It can range from 1 to 10.

AutoFill. A hidden input which allows you to specify that the entire interpolation table should be filled automatically at the current values. To instigate this filling process, this parameter should be set equal to 1. During normal operation, this parameter should be set equal to zero. To view this parameter click Show Secondary in the **Element Options** dialog box that displays when you double-click this element on a schematic.

Parameter Restrictions and Recommendations

1. The Acc parameter A is limited to 1 < Acc > 10. A larger value of Acc increases the density of mesh used in computations. The accuracy of model parameters may improve slightly by increasing Acc, at the expense of a noticeably longer computation time. This model should be used at the highest ACCUR level of 10, as no additional computational burden is seen once the interpolation table is complete.
2. 0.0325<=(W+2S)/H<=4.0
3. 0.1<=W/(W+S)<=0.9

   NOTE: To implement values outside of these ranges, you can use the CPW1LIN element.
4. This model does not impose restrictions on the conductor thickness (thickness may be zero, positive, or negative). Negative thickness means that the conductor is recessed into the substrate.
5. Model parameters can be broken into groups of independent, scalable, fixed, and statistical parameters in accordance with the detailed discussion of the X-models in " EM-based Models (X-models) " :

Independent Parameters:	W, S, L
Scalable Parameters:	Frequency
Fixed Parameters:	ACCUR, CPW_SUB(Tand, Rho, Is Grounded, Has Cover)
Statistical Parameters:	CPW_SUB(Er, T, Hsub, Hcov)

Implementation Details

1. This 2-D Quasi-Static analysis is the same method as that used in the CPW1LINE model, however, CPWLINX gains large computational speed increases due to the table based interpolation. In exchange for this speed increase, small errors resulting from the interpolation should be expected and the range of the input parameters have been restricted.
2. Setting Gnd=0 implies that the substrate is not backed by a perfect conductor, but is bounded by infinite air space. Modeling results are strongly affected by thin substrate heights and might differ substantially from modeling results obtained from models that implement the common conception of a coplanar waveguide (for example, CPWLINE). To model a classic coplanar waveguide featuring an infinitely thick substrate, set Gnd=0 and H >2(W+2S), where Gnd and H are parameters of CPW_SUB.
3. Example projects for autofilling an EM-based model are provided in the */Examples/Circuit Features/Xmodels* subdirectory of the NI AWR program directory. You must initiate an autofill of the EM data table after making any changes to the CPW_SUB element.

This model was developed under research performed at NI AWR Corporation. The details of the implementation are considered proprietary in nature.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See “Cell Options Dialog Box: Layout Tab ” for Cell Options dialog box **Layout** tab details.

CPW elements have special configurations for the defined line types. The center conductor geometry draws on all the layers defined in the line type. The spacing to the ground plane is then drawn on negative layers with the same name as all of the layers in the line type. You must then draw the same layers on the positive layer to complete CPW layout. See “Negative Layers ” for more information on setting up processes for positive and negative layers.

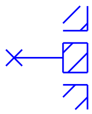
See “The Layout Process File (LPF)” for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

References

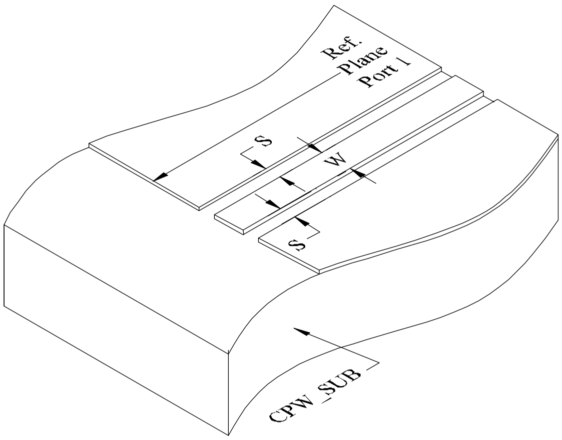
[1] M.B. Bazdar, A.R. Djordjevic, R.F. Harrington, and T.K. Sarkar, "Evaluation of quasi-static matrix parameters for multiconductor transmission lines using Galerkin's method," IEEE Trans. Microwave Theory Tech., vol. MTT-42, July 1994, pp. 1223-1228.

Coplanar Open, with Ground Not Connected (Closed Form): CPWOC

Symbol



Topology



Parameters

CPWOC$ is a co-planar iCell. See the Microwave Office Layout Guide for more information.

Name	Description	Unit Type	Default
ID	Element ID	Text	CP1
W	Conductor width	Length	W[a]
S	Gap width	Length	W[a]
CPW_SUB	Substrate definition	Text	[b]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory.

[b]If only one CPW_SUB is present in the schematic, this substrate is automatically used. If multiple CPW_SUB substrate definitions are present, you must specify which to use.

Implementation Details

This circuit component models an open circuit termination of the coplanar waveguide. Model doesn't account for substrate thickness or presence of metallic cover or backing ground. The parameters W (strip width) and S (S - spacing between strip and the left/right ground halfplane) are dimensions entered in the default length units. This model uses equivalent coaxial waveguide prototype for evaluation open end effect. The parameter CPW_SUB specifies the substrate element, which defines additional cross sectional parameters of the transmission line. If blank, a default is used. Some parameters CPW_SUB, that are Hcover, Cover, and Gnd, are not used by this component. Conductor thickness is assumed zero.

The component doesn't account for losses in metal and in substrate dielectric. Dispersion is not included.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See "Cell Options Dialog Box: Layout Tab " for Cell Options dialog box **Layout** tab details.

CPW elements have special configurations for the defined line types. The center conductor geometry draws on all the layers defined in the line type. The spacing to the ground plane is then drawn on negative layers with the same name as all of the layers in the line type. You must then draw the same layers on the positive layer to complete CPW layout. See "Negative Layers " for more information on setting up processes for positive and negative layers.

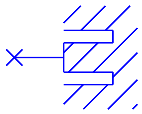
See "The Layout Process File (LPF)" for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

References

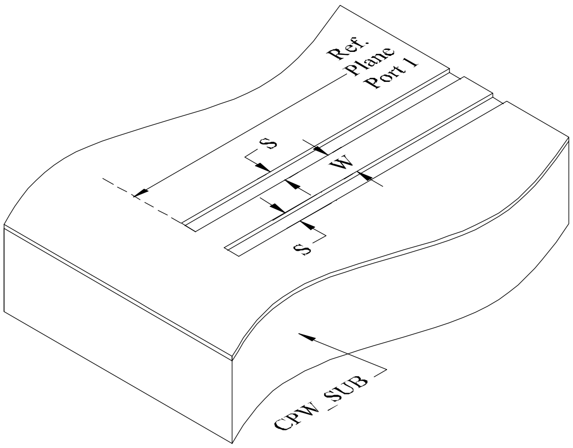
[1] N.Marcuvitz, "Waveguide Handbook", Peter Peregrinus Ltd., 1993 pp.213-216 (section 4-16)

CPW Coplanar Shorted End Effect (Closed Form): CPWSC

Symbol



Topology



Parameters

CPWSC$ is a co-planar iCell. See the Microwave Office Layout Guide for more information.

Name	Description	Unit Type	Default
ID	Element ID	Text	CP1
W	Conductor width	Length	W[a]
S	Gap width	Length	W[a]
CPW_SUB	Substrate definition	Text	[b]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory.

[b] If only one CPW_SUB is present in the schematic, this substrate is automatically used. If multiple CPW_SUB substrate definitions are present, the user must specify.

Implementation Details

This circuit component models a short circuit termination of the coplanar waveguide. Model doesn't account for substrate thickness or presence of metallic cover or backing ground. The parameters W (strip width) and S (S - spacing between strip and the left/right ground halfplane) are dimensions entered in the default length units. The parameter CPW_SUB specifies the substrate element, which defines additional cross sectional parameters of the transmission line. If blank, a default is used. Some parameters CPW_SUB, that are Hcover, Cover, and Gnd, are not used by this component. Conductor thickness is assumed zero.

The component doesn't account for losses in metal and in substrate dielectric. Dispersion is not included.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See “Cell Options Dialog Box: Layout Tab ” for Cell Options dialog box **Layout** tab details.

CPW elements have special configurations for the defined line types. The center conductor geometry draws on all the layers defined in the line type. The spacing to the ground plane is then drawn on negative layers with the same name as all of the layers in the line type. You must then draw the same layers on the positive layer to complete CPW layout. See “Negative Layers ” for more information on setting up processes for positive and negative layers.

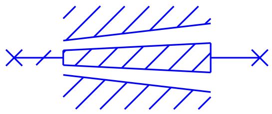
See “The Layout Process File (LPF)” for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

References

[1] W.J.Getsinfer, "End-effects in quasi-TEM transmission lines." IEEE Trans. on MTT, v. 41, No.6, April 1993, pp. 666-672.

Coplanar Tapered Line (Closed Form): CPWTAPER

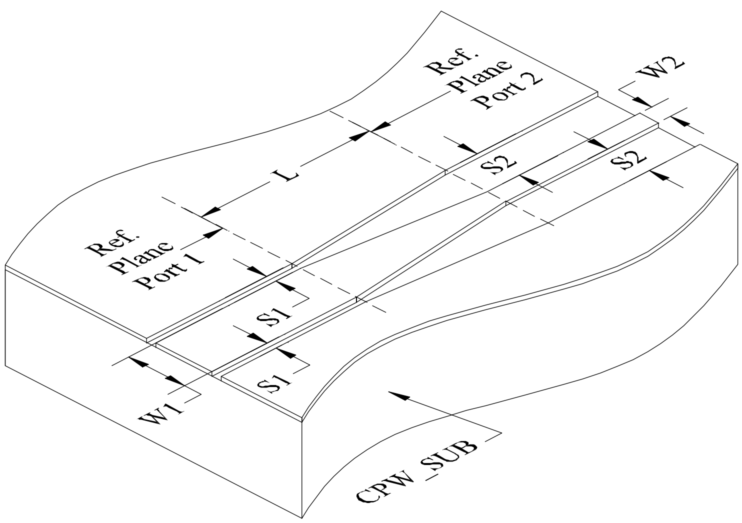
Symbol



Summary

CPWTAPER models a section of coplanar waveguide either on an infinitely thick dielectric substrate or on a finite thickness substrate with optional backing ground and/or optional metallic cover. Signal strip width linearly changes along the section, and the section is symmetric (the local widths of gaps between the strip conductor and lateral grounds are equal). Infinitely thick dielectric substrate implies that a coplanar waveguide is ungrounded -- the backing ground plane (if present) is way below the signal strip and does not affect modeling results. User can switch to the substrate of finite thickness and add optional backing ground and metallic cover.

Topology



Parameters

CPWTAPER$ is a coplanar iCell and only has an L parameter. See the “Intelligent Cells (iCells)” for more information

Name	Description	Unit Type	Default
ID	Name	Text	CP1
W1	Conductor width @ node 1	Length	W[a]
S1	Gap width @ node 1	Length	W[a]
W2	Conductor width @ node 2	Length	W[a]
S2	Gap width @ node 2	Length	W[a]
L	Length of taper	Length	L[b]
CPW_SUB	Substrate definition	Text	[c]

Name	Description	Unit Type	Default
*IsSubInf	Selector: Infinite substrate thickness/Finite substrate thickness	Text	Infinite substrate thickness
*Acc	Accuracy (1 ≤Acc≤10) (effective only if finite substrate thickness selected)		1

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory.

[b]L is user definable in the *default.lpf* file under $DEFAULT_VALUES in the L value.

[c] If only one CPW_SUB is present in the schematic, this substrate is automatically used. If multiple CPW_SUB substrate definitions are present, the user must specify.

** indicates a hidden secondary parameter*

Parameter Details

IsSubInf. IsSubInf is a secondary control parameter-selector that allows user to specify the technique that CPWTAPER uses for simulation. IsSubInf value "Infinite substrate thickness" applies closed form modeling of classic CPW presented at [1] [4–46]. IsSubInf value "Finite substrate thickness" applies method of quasi-static method of moments presented at [2] [4–46]. Default IsSubInf value is "Infinite substrate thickness"

CPW_SUB. CPW_SUB supplies parameters for dielectric substrate, conductor thickness, and conductor metal properties. If IsSubInf = "Infinite substrate thickness" selected then CPWTAPER does not use the following CPW_SUB parameters: Hcover, Cover, Gnd, Hab, ER_Nom, H_Nom, Hcov_Nom, Hab_Nom, T_Nom, Acc. If IsSubInf = "Finite substrate thickness" selected then CPWTAPER still does not use the CPW_SUB parameters Hab, ER_Nom, H_Nom, Hcov_Nom, Hab_Nom, T_Nom, but uses all material parameters as well as parameters H, Hcover, Cover, Gnd, and Acc.

Acc. Parameter Acc is visible and used only if IsSubInf = "Finite substrate thickness". This parameter controls meshing density of method of moments (see [2] [4–46]). Acc default value is 1. Bigger values of Acc may slightly increase simulation time. Good accuracy is provided at 1≤ Acc ≤3.

Parameter Restrictions and Recommendations

If IsSubInf="Infinite substrate thickness" selected than model issues a warning if one of the following conditions is violated: H<2(W+2S1) or H<2(W+2S2). This is because infinite thickness substrate approach [1] [4–46] implies that H is large compared to W+S, so the warning alerts you that the substrate height is too small, initial model assumptions are not met, and accuracy may degrade. If IsSubInf="Finite substrate thickness" than model uses alternative implementation [2] [4–46] and accounts for finite thickness of the substrate.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See "Cell Options Dialog Box: Layout Tab " for Cell Options dialog box **Layout** tab details.

CPW elements have special configurations for the defined line types. The center conductor geometry draws on all the layers defined in the line type. The spacing to the ground plane is then drawn on negative layers with the same name as all of the layers in the line type. You must then draw the same layers on the positive layer to complete CPW layout. See “Negative Layers ” for more information on setting up processes for positive and negative layers.

See “The Layout Process File (LPF)” for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

Implementation Details

This model is constructed out of a cascaded series of constant width coplanar waveguide transmission lines. See the CPWLINE model for details on the model used for these transmission lines. The number of sections used is frequency-dependent and is constant as a function of the length divided by the guided wavelength. The model assumes a quasi-TEM mode of propagation and incorporates the effects of dielectric and conductive losses and dispersion.

Note that if IsSubInf = "Infinite thickness substrate" selected then model implements a technique developed in [1] [4–46]; this approach implies that H is large, hence, actual modeling results do not depend on H. If IsSubInf = "Finite thickness substrate" selected then model implements a technique developed in [2] [4–46]; this approach implies that H is arbitrary and substrate parameters Cover and Gnd may be used to install respectively metallic cover at the height Hcover (substrate parameter) and/or perfect infinite ground at the substrate back.

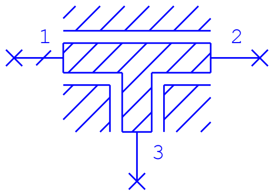
References

[1] W. Heinrich, "Quasi-TEM Description of MMIC Coplanar Lines Including Conductor-Loss Effects," IEEE Trans. Microwave Theory Tech., vol. MTT-41, January 1993, pp. 45-52.

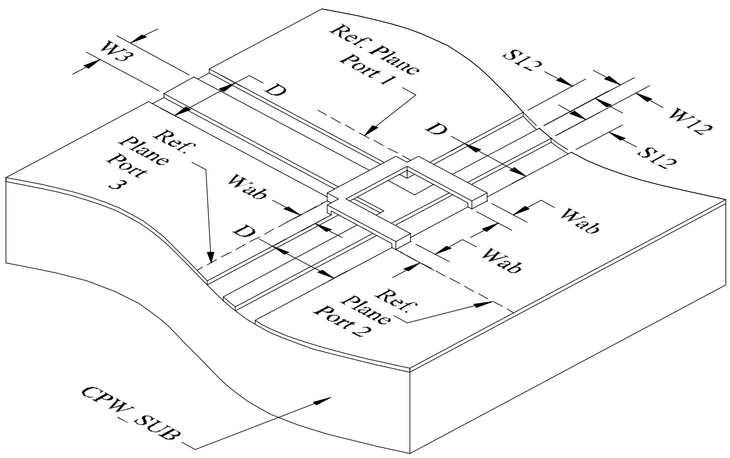
[2] M.B. Bazdar, A.R. Djordjevic, R.F. Harrington, and T.K. Sarkar, "Evaluation of quasi-static matrix parameters for multiconductor transmission lines using Galerkin's method," IEEE Trans. Microwave Theory Tech., vol. MTT-42, July 1994, pp. 1223-1228.

Symmetric Coplanar T-Junction (EM Based): CPWTEEX

Symbol



Topology



Parameters

CPWTEEX$ is a co-planar iCell with no parameters. See the "Intelligent Cells (iCells)" for more information.

Name	Description	Unit Type	Default
ID	Element ID	Text	CP1
W12	Conductor width @ port 1&2	Length	W[a]
S12	Gap width @ port 1&2	Length	W[a]
W3	Conductor width @ port 3	Length	W[a]
Wab	AirBridge conductor widths @ all ports	Length	W[a]
CPW_SUB	Substrate definition	Text	[b]
*AutoFill	AutoFill dataBase if not equal to 0		0

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory.

[b] If only one CPW_SUB is present in the schematic, this substrate is automatically used. If multiple CPW_SUB substrate definitions are present, the user must specify.

** indicates a secondary parameter*

Parameter Restrictions and Recommendations

0.25<=(W12+2*S12)/HSub<=4.0

0.1<=W12/(W12+2*S12)<=0.9

0.1<=W3/(W12+2*S12)<=0.9

0.1<=Wab/(W12+2*S12)<=0.5

Implementation Details

This circuit component models a co-planar waveguide symmetric tee junction including the effects of airbridges which connect the ground planes on each side of a discontinuity. The purpose of these features is to suppress the excitation of a slot line mode between the three ground planes. Note that ports 1 and 2 are assumed symmetric. Further, it is assumed that the span across ports 1 and 2 are equal to the span across port 3. This model is constructed as an X-model (Table Based Interpolation) using results automatically generated by EMSight and theCPWLINX,CPWABRG models. For a more detailed discussion of the X-models see “ EM-based Models (X-models) ”. An equivalent circuit model is constructed for the discontinuity based upon the EMSight simulation. As the dispersion of co-planar waveguide is minimal, values of the equivalent circuit parameters in the model are assumed frequency independent.

The input parameters W12,S12,W3 and Wab as defined in "Topology" are entered in the default length units.

CPW_SUB allows you to specify the co-planar waveguide substrate.

The AutoFill parameter is a hidden input which allows you to specify that the entire interpolation table should be filled automatically at the current fixed values.

Independent Parameters:	W12, S12,W3,Wab
Scalable Parameters:	Frequency
Fixed Parameters:	ACCUR, CPW_SUB(Tand, Rho, Is Grounded, Has Cover)
Statistical Parameters:	CPW_SUB(Er, Hab, T, Hsub, Hcov)

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See “Cell Options Dialog Box: Layout Tab ” for Cell Options dialog box **Layout** tab details.

CPW elements have special configurations for the defined line types. The center conductor geometry draws on all the layers defined in the line type. The spacing to the ground plane is then drawn on negative layers with the same name as all of the layers in the line type. You must then draw the same layers on the positive layer to complete CPW layout. See “Negative Layers ” for more information on setting up processes for positive and negative layers.

See “The Layout Process File (LPF)” for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

Recommendations for Use

The CPWTEEX model creates (and uses for modeling) four files with data tables during autofill: Two files with a CPTX prefix for coplanar waveguides, one file with a CPABG prefix for air bridge, and one file with a CPTA prefix that contains CPWTEEX autofill results. The CPTX and CPABG files created by the CPWLINX, CPWBENDX, and CPWABRGX models may also qualify for the CPWTEEX model. If the model finds that the CPTX and/or CPABG files were created with the same set of fixed parameters, it does not generate new files, but uses the existing ones for modeling.

Example projects for autofilling an EM-based model are provided in the /Examples/Circuit Features/Xmodels subdirectory of the NI AWR program directory.

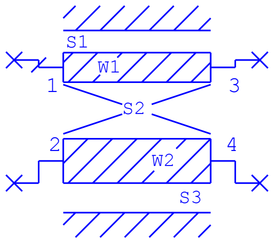
NOTE: You must initiate an autofill of the EM data table after making any changes to the CPW_SUB element.

References

This model was developed under research performed at NI AWR Corporation. The details of the implementation are considered proprietary in nature.

2 Multilayer Asymmetric Coupled Coplanar Lines (EM Quasi-Static): GCPW2LNA

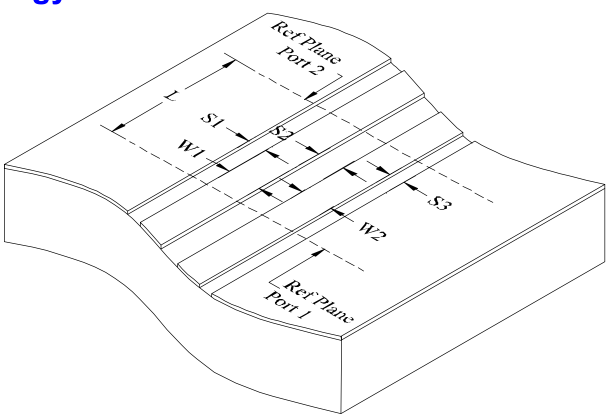
Symbol



Summary

GCPW2LNA models a section of two asymmetric coupled coplanar waveguides (strip widths may be unequal and gaps between each strip conductor and closest lateral ground may have unequal width) on a multilayer dielectric substrate. The substrate can be suspended and/or shielded by the optional metallic cover. This model allows for an arbitrary metal thickness of a signal conductor and lateral ground planes. A backing ground plane is inherent to this model.

Topology



Parameters

Name	Description	Unit Type	Default
ID	Name	Text	TL1
W1	Conductor #1 width (nodes 1,3)	Length	W[a]
W2	Conductor #2 width (nodes 2,4)	Length	W[a]
S1	Gap width between conductor #1 and lateral ground	Length	W[a]
S2	Gap width between conductors #1 and #2	Length	W[a]
S3	Gap width between conductor #2 and lateral ground	Length	W[a]
L	Line length	Length	L[b]
CL	Number of the substrate layer carrying conductors and lateral grounds		1

Name	Description	Unit Type	Default
Acc	Accuracy		1
GMSUB	Substrate definition	Text	[c]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory.

[b]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory.

[c]If only one CMSUB is present in the schematic, this substrate is automatically used. If multiple CMSUB substrate definitions are present, you must specify.

** indicates a secondary parameter*

Parameter Details

GMSUB. Supplies parameters for multilayer dielectric substrate, conductor thickness, conductor metal properties, the presence/absence of metallic cover, and the cover height over the substrate. The GMSUB Cover parameter allows the addition/elimination of infinite metallic plate acting as a cover/shield. Note that the GMSUB T parameter is actually a vector, but GCPW2LNA uses only the first entry so the value of T may be entered as a scalar. T specifies thickness of the signal conductor as well as both lateral grounds.

Acc. The default value for Acc is 1. If Acc is less than 1 or greater than 10 it is auto-set to 2.

CL. Specifies the number of the dielectric layer that carries atop the signal conductor and lateral grounds. Layers are numbered from top to bottom. Note that if Cover is present the dielectric layer adjacent to the cover is excluded from the layer count. If the substrate is suspended, the air layer under the substrate is included in the layer count.

Parameter Restrictions and Recommendations

1. The Acc parameter A is limited to 1 < Acc > 10. A larger value of Acc increases the density of mesh used in computations. The accuracy of model parameters may improve slightly by increasing Acc, at the expense of a noticeably longer computation time. Generally, a good trade-off between accuracy and computation time is to set Acc to 1.
2. This model does not impose restrictions on the conductor thickness (it may be zero, positive, or negative). Negative thickness means that the conductor is recessed into the substrate.
3. Secondary parameter SNAME is for layout only and has no effect on the electrical performance.

Implementation Details

1. Model implementation is based on the EM Quasi-Static technique described in [1] [4–52]. It accounts for losses in metal and in the substrate dielectric. Dispersion is partly included.
2. Modeling results are strongly affected by the substrate height and might differ substantially from modeling results obtained from models that implement the common conception (infinite thickness substrate) of a coplanar waveguide (for example, CPWLINE). A backing ground is inherent to this model.
3. This model uses the GMCLIN model so some warning/error messages may originate from GMCLIN.
4. To apply Method of Moments for analysis, a quasi-static model creates 1D mesh covering cross-sectional contours of all conductors. The mesh is made of linear segments (pulses) of varying length. The length of a pulse is relatively big at the conductor center; it decreases toward conductor edges to reveal the charge distribution across conductor. If the conductor width is too large it may cause the pulse size to approach zero for pulses close to edge. In these rare cases the model may display a “Length of pulse #nnn equal to zero” error message.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See "Cell Options Dialog Box: Layout Tab " for Cell Options dialog box **Layout** tab details.

CPW elements have special configurations for the defined line types. The center conductor geometry draws on all the layers defined in the line type. The spacing to the ground plane is then drawn on negative layers with the same name as all of the layers in the line type. You must then draw the same layers on the positive layer to complete CPW layout. See "Negative Layers " for more information on setting up processes for positive and negative layers.

See "The Layout Process File (LPF)" for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

Recommendations for Use

NOTE: The implementation of EM Quasi-Static models heavily relies on the involved numerical algorithms. This may lead to a noticeable increase in simulation time for schematics that employ many such models.

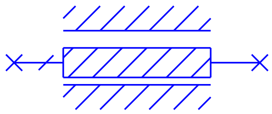
If the thickness of any layer is too small in comparison with the thickness of another layer, simulation time may also noticeably grow.

References

[1] M.B. Bazdar, A.R. Djordjevic, R.F. Harrington, and T.K. Sarkar, "Evaluation of quasi-static matrix parameters for multiconductor transmission lines using Galerkin's method," IEEE Trans. Microwave Theory Tech., vol. MTT-42, July 1994, pp. 1223-1228.

Multilayer Coplanar Asymmetric Line (EM Quasi-Static): GCPWALIN

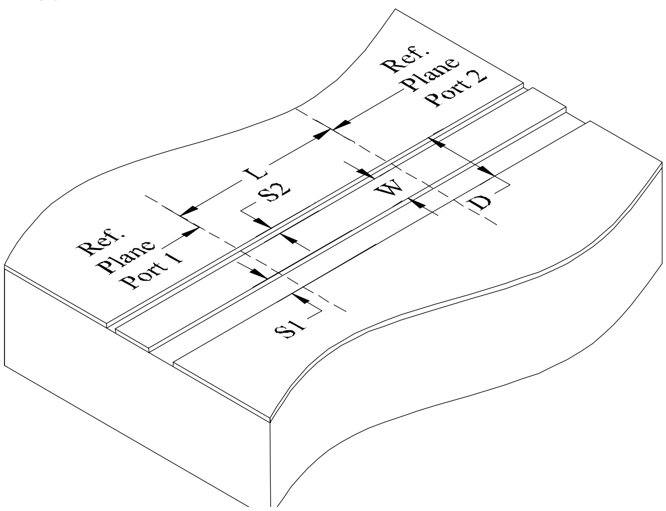
Symbol



Summary

GCPWALIN models a section of asymmetric coplanar waveguide (gaps between the strip conductor and lateral grounds may have an unequal width) on a multilayer dielectric substrate. The substrate can be suspended and/or shielded by the optional metallic cover. This model allows for an arbitrary metal thickness of a signal conductor and lateral ground planes. A backing ground plane is inherent to this model.

Topology



Parameters

Name	Description	Unit Type	Default
ID	Name	Text	TL1
W	Conductor width	Length	W[a]
S1	Gap between conductor and left ground	Length	W[a]
S2	Gap between conductor and right ground	Length	W[a]
L	Line length	Length	L[b]
CL	Number of the substrate layer carrying conductor and lateral grounds		1
Acc	Accuracy		1
GMSUB	Substrate definition	Text	[c]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory.

[b]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory.

[c]If only one CMSUB is present in the schematic, this substrate is automatically used. If multiple CMSUB substrate definitions are present, you must specify.

** indicates a secondary parameter*

Parameter Details

GMSUB. Supplies parameters for multilayer dielectric substrate, conductor thickness, conductor metal properties, the presence/absence of metallic cover, and the cover height over the substrate. The GMSUB Cover parameter allows the addition/elimination of infinite metallic plate acting as a cover/shield. Note that the GMSUB T parameter is actually a vector, but GCPWALIN uses only the first entry so the value of T may be entered as a scalar.

Acc. The default value for Acc is 1. If Acc is less than 1 or greater than 10 it is set automatically to 2.

CL. Specifies the number of the dielectric layer that carries atop the signal conductor and lateral grounds. Layers are numbered from top to bottom. Note that if Cover is present the dielectric layer adjacent to the cover is excluded from the layer count. If the substrate is suspended, the air layer under the substrate is included in the layer count.

Parameter Restrictions and Recommendations

- The Acc parameter A is limited to 1 < Acc > 10. A larger value of Acc increases the density of mesh used in computations. The accuracy of model parameters may improve slightly by increasing Acc, at the expense of a noticeably longer computation time. Generally, a good trade-off between accuracy and computation time is to set Acc to 1.
- This model does not impose restrictions on the conductor thickness (it may be zero, positive, or negative). Negative thickness means that the conductor is recessed into the substrate.

Implementation Details

- Model implementation is based on the EM Quasi-Static technique described in [1] [4–55]. It accounts for losses in metal and in the substrate dielectric. Dispersion is partly included.
- Modeling results are strongly affected by the substrate height and might differ substantially from modeling results obtained from models that implement the common conception (infinite thickness substrate) of a coplanar waveguide (for example, CPWLINE). A backing ground is inherent to this model.
- This model uses the GMCLIN model so some warning/error messages may originate from GMCLIN.
- To apply Method of Moments for analysis, a quasi-static model creates 1D mesh covering cross-sectional contours of all conductors. The mesh is made of linear segments (pulses) of varying length. The length of a pulse is relatively big at the conductor center; it decreases toward conductor edges to reveal the charge distribution across conductor. If the conductor width is too large it may cause the pulse size to approach zero for pulses close to edge. In these rare cases the model may display a “Length of pulse #nnn equal to zero” error message.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See “Cell Options Dialog Box: Layout Tab ” for Cell Options dialog box **Layout** tab details.

CPW elements have special configurations for the defined line types. The center conductor geometry draws on all the layers defined in the line type. The spacing to the ground plane is then drawn on negative layers with the same name as all of the layers in the line type. You must then draw the same layers on the positive layer to complete CPW layout. See “Negative Layers ” for more information on setting up processes for positive and negative layers.

See “The Layout Process File (LPF)” for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

Recommendations for Use

NOTE: The implementation of EM Quasi-Static models heavily relies on the involved numerical algorithms. This may lead to a noticeable increase in simulation time for schematics that employ many such models.

If the thickness of any layer is too small in comparison with the thickness of another layer, simulation time may also noticeably grow.

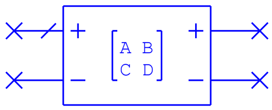
References

[1] M.B. Bazdar, A.R. Djordjevic, R.F. Harrington, and T.K. Sarkar, "Evaluation of quasi-static matrix parameters for multiconductor transmission lines using Galerkin's method," IEEE Trans. Microwave Theory Tech., vol. MTT-42, July 1994, pp. 1223-1228.

General

Two-Port ABCD (Chain) Parameter Block: ABCD_BLK

Symbol



Summary

ABCD_BLK models a generalized two-port network. The network is characterized using ABCD parameters (also known as chain or transmission parameters).

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	X1
A	Reverse voltage gain		1
B	Reverse transresistance	Resistance	0 ohm
C	Reverse transconductance	Conductance	0 S
D	Reverse current gain		1
*ChkPassv	Switch "Do not Check passivity/Check passivity"		No

** indicates a secondary parameter*

Parameter Details

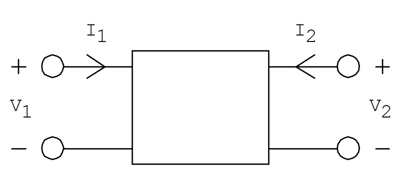
ChkPassv. If set to Yes, tests ABCD_BLK for passivity and displays a warning if there is a passivity violation. The default setting is No (Do not check passivity).

Parameter Restrictions and Recommendations

B must be non-zero.

Implementation Details

The ABCD parameters are defined by the following figure and equation.



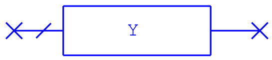
$$\begin{bmatrix} V_1 \\ I_1 \end{bmatrix} = \begin{bmatrix} A & B \\ C & D \end{bmatrix} \begin{bmatrix} V_2 \\ -I_2 \end{bmatrix}$$

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

General Admittance Element (Closed Form): ADMIT

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Name	Text	Y1
G	Real part of admittance	Conductance	0 S
B	Imaginary part of admittance	Conductance	0 S

Implementation Details

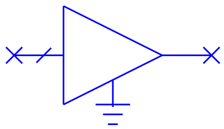
Represents an ideal admittance. G and B represent the real and imaginary parts of the admittance, respectively.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Amplifier Block (Closed Form): AMP

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Name	Text	U1
A	Gain	DB	0 dB
S	Gain slope, dB/octave		6
F	Freq at which gain roll-off begins	Frequency	0 GHz
R	Port resistance	Resistance	50 ohm

Implementation Details

Implements an ideal gain block with real input and output impedances given by R.

$$A(f) = \left(A - \left(n \times 20\log\left(1 + i\frac{f}{F}\right)\right)\right)$$

$$n = \frac{S}{6}$$

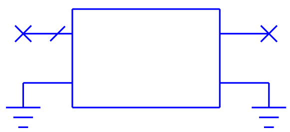
$$S_{12} = 0$$

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Attenuator (Closed Form): ATTEN

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Name	Text	U1
R	Port resistance	Resistance	50 ohm
LOSS	Loss	DB	3 dB

Implementation Details

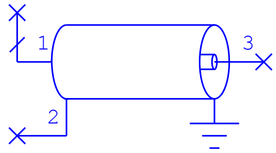
Implements an ideal attenuator with loss specified by LOSS in dB and input and output resistances given by R. LOSS may be entered as either a positive or a negative quantity, but in all cases it is treated as a loss. Input and output VSWR are 1.0.

Layout

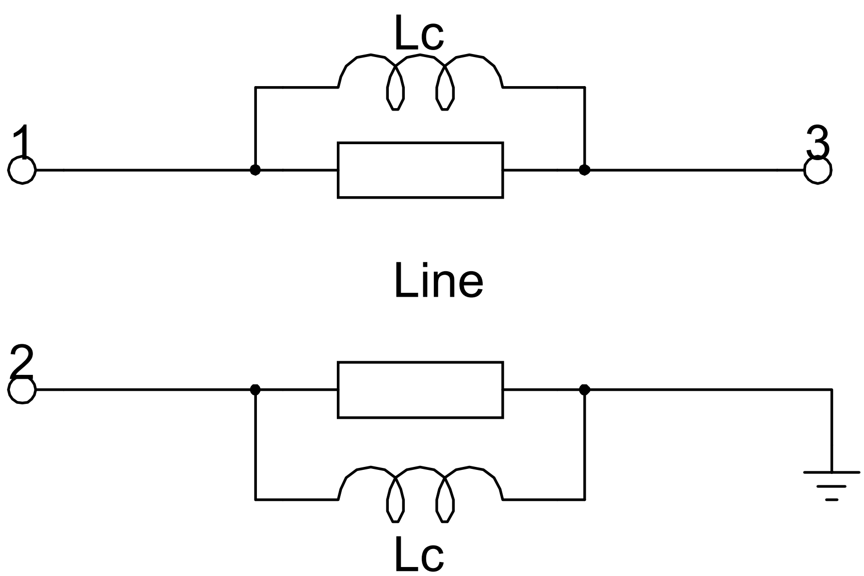
This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Balun, Balanced to Unbalanced Ferrite Core (Closed Form): BALUN1

Symbol



Equivalent Circuit



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	BU#
Zo	Char. imped of Txline	Resistance	50 ohm
L	Physical length of Txline	Length	L1
Er	Effective dielectric constant of Txline		1
A	Attenuation constant of Txline (dB/m)		0
F	Frequency for scaling of Txline attenuation	Frequency	0 GHz
N	Number of turns of line around the ferrite core		1
AL	Inductive index (inductance/turn[a]	Inductance	1 nH

[a]L is user-modifiable. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory.

Implementation Details

This component models an ideal balanced-to-unbalanced transformer that consists of a transmission line wound around a ferrite core. The external balanced line connects to ports 1 and 2; the external unbalanced line to port 3.

The choking inductance, Lc, is

$$L_C = N^2 \cdot \mathrm{AL}$$

The attenuation A(F) has the following frequency dependence:

$$A(F) = \begin{cases} A & F = 0 \\ A \cdot \sqrt{\frac{f}{F}} & F > 0 \end{cases}$$

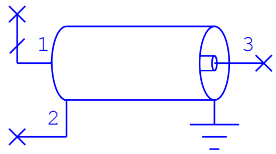
where f is a simulation frequency.

Layout

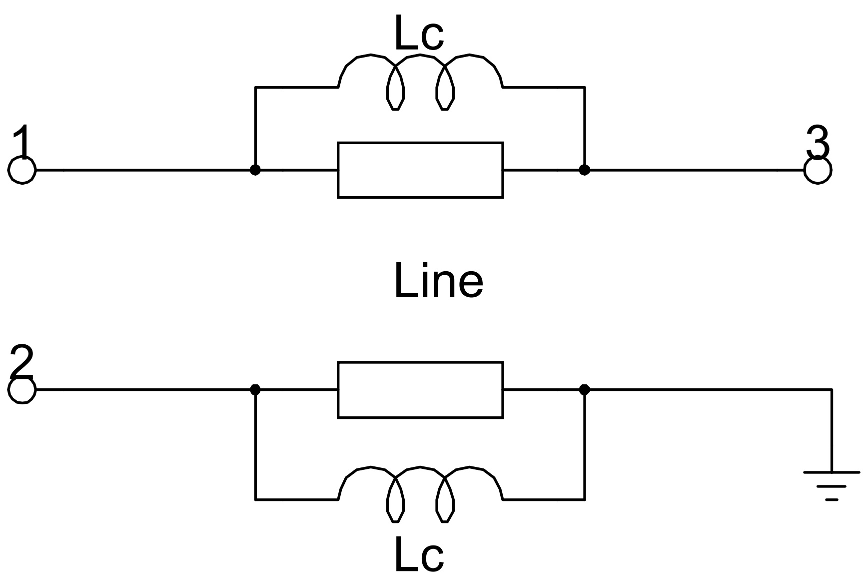
This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Balun, Balanced to Unbalanced Ferrite Sleeve (Closed Form): BALUN2

Symbol



Equivalent Circuit



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	BU#
Zo	Char. Imped of Txline	Resistance	50 ohm
Len	Physical length of Txline	Length	L[a]
Er	Effective dielectric constant of Tx line		1
A	Attenuation const of Txline (dB/m)		0
F	Frequency for scaling of Txline attenuation	Frequency	0 GHz
Mu	Relative permeability of the ferrite sleeve		1
L	Inductive index (inductive units/turns2)	Inductance	1 nH

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory.

Implementation Details

This component models an ideal balanced-to-unbalanced transformer that consists of a transmission line wound around a ferrite core. The external balanced line connects to ports 1 and 2; the external unbalanced line to port 3.

The choking inductance, Lc, is

$$L_c = \mathrm{Mu} \cdot L \cdot \mathrm{LEN}$$

The attenuation A(F) has the following frequency dependence:

$$A(F) = \begin{cases} A & F = 0 \\ A(F) \cdot \sqrt{\dfrac{f}{F}} & F > 0 \end{cases}$$

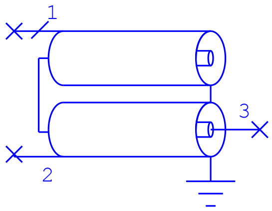
where f is a simulation frequency.

Layout

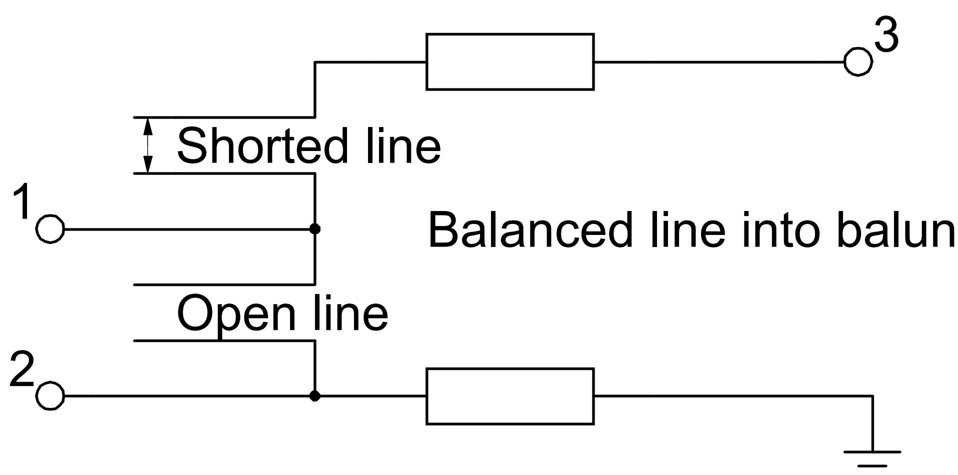
This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

Balun, Marchand Balanced to Unbalanced Transformer: BALUN3

Symbol



Equivalent Circuit



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	BU#
Z_in	Char. imped of Txline into balun	Resistance	65 ohm
Z_o	Char. imped of open Txline	Resistance	75 ohm
Z_s	Char. imped of shorted Txline	Resistance	100 ohm
EL_in	Electrical length of Txline into balun at Fo	Angle	90 Deg
EL_o	Electrical length of open Txline at Fo	Angle	90 Deg
EL_s	Electrical length of shorted Txline at Fo	Angle	90 Deg
Fo	Frequency of electrical lengths	Frequency	1 GHz

Implementation Details

This component models an ideal Marchand balun. The balun consists of balanced, open-circuited and short-circuited transmission line sections. The external balanced line connects to ports 1 and 2; the external unbalanced line connects to port 3.

Layout

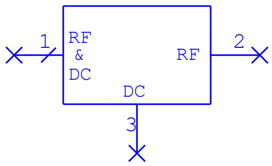
This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

References

[1] R.Mongia, I.Bahl, P.Bhartia, RF and Microwave Coupled-Line Circuits, Norwood, MA: Artech House, 1999. Ch.11.

Ideal Bias Tee (Closed Form): BIASTEE

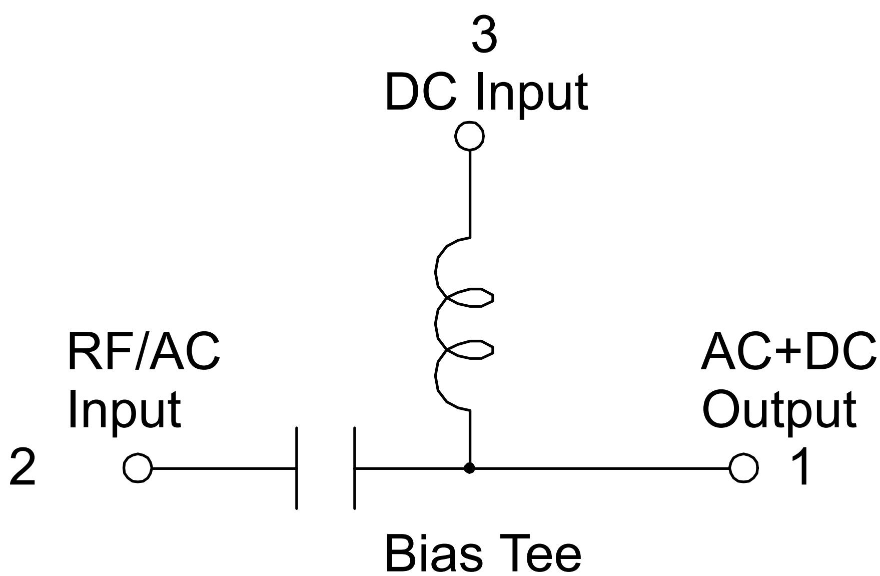
Symbol



Summary

BIASTEE models an ideal bias tee. Bias tees allow a DC current to bias an active device such as a diode. The DC bias current and the AC/RF signal get added together and come out of the AC+DC output port.

Equivalent Circuit



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	X

Implementation Details

Implements an ideal bias tee with the following 3-port S-parameter matrix:

$$S = \begin{bmatrix} 0 & 1 & 0 \\ 1 & 0 & 0 \\ 0 & 0 & 1 \end{bmatrix} \quad \text{for } f \geq 1\text{Hz}$$

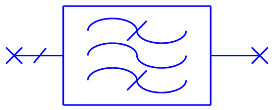
$$S = \begin{bmatrix} 0 & 0 & 1 \\ 0 & 1 & 0 \\ 1 & 0 & 0 \end{bmatrix} \quad \text{for } f < 1\text{Hz}$$

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Butterworth Bandpass Filter: BPFB

Symbol



Summary

BPFB models represent lumped-element Butterworth bandpass filters. They offer simplicity and a compromise between high selectivity and flat group delay. The insertion loss is maximally flat at the passband center frequency and the stopband attenuation increases monotonically.

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	BPFB1
N	Number of reactances in the filter		3
FP1	Lower edge of passband (when Qu is infinite).	Frequency	0.5 GHz
FP2	Upper edge of passband (when Qu is infinite).	Frequency	1.5 GHz
*AP	Maximum passband attenuation (for infinite Qu).	DB	3.0103 dB
*RS	Expected source resistance.	Resistance	50 ohm
*RL	Expected load resistance	Resistance	50 ohm
*QU	Average unloaded Q of the lumped lowpass reactances.		1e12

** indicates a secondary parameter*

Parameter Restrictions and Recommendations

1. 0 < N < 29
2. 0 < FP1
3. 0 < FP2
4. 0 < AP Recommend AP greater than or equal to 0.001 dB.
5. 0 < RS
6. 0 < RL
7. 0 < QU. Recommend QU less than or equal to 10^{12}.

Implementation Details

The model is implemented as a short-circuit admittance matrix, whose equivalent transfer function squared magnitude is that of a Butterworth filter:

$$|S_{21}(s)|^2 = \frac{1}{1+(-1)^N \varepsilon^2 S^{2N}} = \frac{1}{|g(s)|^2} = \frac{1}{1+|h(s)|^2}$$

where

$$\varepsilon^2 = 10^{\mathrm{AP}/10} - 1$$

$$s = \frac{\sqrt{\mathrm{FP2} \times \mathrm{FP1}}}{\mathrm{QU} \times \ |\mathrm{FP2} - \mathrm{FP1}|} + j\omega$$

$$j = \sqrt{-1}$$

and a lowpass-to-bandpass frequency transformation has been applied:

$$\omega = \frac{\_\mathrm{FREQ} - (\mathrm{FP2} \times \mathrm{FP1}) / \_\mathrm{FREQ}}{|\mathrm{FP2} - \mathrm{FP1}|}$$

_FREQ is the variable containing the project frequency, and the admittances are:

$$y_{11} = \left(\frac{1}{\mathrm{RS}}\right)\frac{g(s) + g(-s) - h(s) - h(-s)}{g(s) - g(-s) + h(s) - h(-s)}$$

$$y_{22} = \left(\frac{1}{\mathrm{RL}}\right)\frac{g(s) + g(-s) + h(s) + h(-s)}{g(s) - g(-s) + h(s) - h(-s)}$$

$$y_{12} = y_{21} = \left(\frac{1}{\sqrt{\mathrm{RS} \times \mathrm{RL}}}\right)\frac{2}{g(s) - g(-s) + h(s) - h(-s)}$$

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Recommendations for Use

Note that this model behaves as if it has ideal impedance transformers at its ports, so there is no attenuation due to mismatched source and load impedances. The model expects that the source impedance will equal RS and that the load impedance will equal RL, but RS need not equal RL for ideal transmission (as would normally be the case).

References

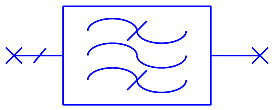
[1] Rolf Schaumann, Mohammed S. Ghausi, and Kenneth R. Laker, Design of Analog Filters: Passive, Active RC, and Switched Capacitor, (Prentice-Hall, 1990), pp. 40-44.

[2] Louis Weinberg, Network Analysis and Synthesis, (Robert E. Krieger Publishing, 1975), pp. 493-498.

[3] Adel S. Sedra and Peter O. Brackett, Filter Theory and Design: Active and Passive, (Matrix Publishers, 1978), pp. 105-111.

Chebyshev Bandpass Filter: BPFC

Symbol



Summary

BPFC models represent lumped-element Chebyshev bandpass filters. The insertion loss ripples between zero and a specified maximum in the passband. The stopband attenuation increases rapidly and monotonically beyond the passband edges. This filter type offers simplicity and good selectivity.

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	BPFC1
N	Number of reactances in the filter		3
FP1	Lower frequency edge of passband (when Qu is infinite).	Frequency	0.5 GHz
FP2	Upper frequency edge of passband (when Qu is infinite).	Frequency	1.5 GHz
AP	Maximum passband attenuation (when Qu is infinite).	DB	0.1 dB
*RS	Expected source resistance.	Resistance	50 ohm
*RL	Expected load resistance	Resistance	50 ohm
*QU	Average unloaded Q of the lumped lowpass reactances.		1e12

** indicates a secondary parameter*

Parameter Restrictions and Recommendations

1. 0 < N < 27
2. 0 < FP1
3. 0 < FP2
4. 0 < AP Recommend AP greater than or equal to 0.001 dB.
5. 0 < RS
6. 0 < RL
7. 0 < QU. Recommend QU less than or equal to 1e12.

Implementation Details

The model is implemented as a short-circuit admittance matrix, whose equivalent transfer function squared magnitude is that of a Chebyshev filter:

$$|S_{21}(s)|^2 = \frac{1}{1+\varepsilon^2 C_n^2\left(\frac{s}{j}\right)} = \frac{1}{|g(s)|^2} = \frac{1}{1+|h(s)|^2}$$

where Cn is the Chebyshev polynomial of the first kind, and

$$\varepsilon^2 = 10^{AP/10} - 1$$

$$C_n(\omega) = \cos(N * \operatorname{acos}(\omega)) \qquad \text{for } 0 \le |\omega| \le 1$$

$$C_n(\omega) = \cosh(N * \operatorname{acosh}(\omega)) \qquad \text{for } 1 < |\omega|$$

$$s = \frac{\sqrt{FP2 \times FP1}}{QU \times |FP2 - FP1|} + j\omega$$

$$\omega = \frac{\_FREQ - (FP2 \times FP1) / \_FREQ}{|FP2 - FP1|}$$

and a lowpass-to-bandpass frequency transformation has been applied:

$$y_{11} = \left(\frac{1}{RS}\right)\frac{g(s) + g(-s) - h(s) - h(-s)}{g(s) - g(-s) + h(s) - h(-s)}$$

_FREQ is the variable containing the project frequency, and the admittances are:

$$y_{22} = \left(\frac{1}{RL}\right)\frac{g(s) + g(-s) + h(s) + h(-s)}{g(s) - g(-s) + h(s) - h(-s)}$$

$$y_{12} = y_{21} = \left(\frac{1}{\sqrt{RS \times RL}}\right)\frac{2}{g(s) - g(-s) + h(s) - h(-s)}$$

$$j = \sqrt{-1}$$

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

Recommendations for Use

Note that this model behaves as if it has ideal impedance transformers at its ports, so there is no attenuation due to mismatched source and load impedances. The model expects that the source impedance will equal RS and that the load impedance will equal RL, but RS need not have any special relationship to RL for ideal transmission (as would normally be the case).

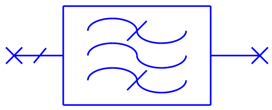
References

[1] Rolf Schaumann, Mohammed S. Ghausi, and Kenneth R. Laker, Design of Analog Filters: Passive, Active RC, and Switched Capacitor, (Prentice-Hall, 1990), pp. 44-48.

[2] Louis Weinberg, Network Analysis and Synthesis, (Robert E. Krieger Publishing, 1975), pp. 507-529.

Generalized Chebyshev Bandpass Filter (Closed Form): BPFCG

Symbol



Summary

BPFCG models represent lumped-element Generalized Chebyshev (or "Quasi-Elliptic") bandpass filters. The insertion loss ripples between zero and a specified maximum in the passband. The stopband attenuation is defined by arbitrarily specified transmission zeros. Real-frequency finite transmission zeros can be specified to improve selectivity at the expense of ultimate stopband attenuation and passband group delay, while complex-plane finite transmission zeros can be specified to provide passband group delay equalization at the expense of selectivity and ultimate stopband attenuation. Generalized Chebyshev filters represents a compromise between the simplicity of Chebyshev filters, the optimum amplitude response of more complicated Elliptic filters, and the phase linearity of Bessel and Gaussian filters. Because this type of filter allows one to make explicit design trade-offs between complexity, selectivity, and group delay equalization, it is often used to meet the demanding requirements of modern communications systems.

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	BPFCG1
N	Order of filter's lowpass prototype		3
FP1	Passband corner frequency (when Qu is infinite).	Frequency	0.5 GHz
FP2	Passband corner frequency (when Qu is infinite).	Frequency	1.5 GHz
*PPD	Passband parameter description:- Maximum Insertion Loss,- Minimum Return Loss, or- Maximum VSWR.	Enumerated	Maximum Insertion Loss
PPV	Passband parameter value (when Qu is infinite)	dB or Scalar	0.1 dB
TZF	Real frequency, finite transmission zeros	(Real) Frequency	{2} GHz
*TZR	Real parts of complex finite transmission zeros	(Imaginary) Frequency	{0} GHz
*RS	source resistance	Resistance	50 ohm
*RL	Load resistance	Resistance	50 ohm
*QU	Average unloaded Q of resonance in the filter.		0

** indicates a secondary parameter*

Parameter Details

N. In mathematical terms, N is defined as the highest exponent of the complex frequency variable s in the transfer function,$S_{21}(s)$, of the filter's normalized lowpass prototype, or, equivalently, half of the highest exponent of s in the transfer function of the bandpass filter. In terms of the number of circuit components, N corresponds to the total number of resonances in single-mode or multi-mode direct-coupled-resonator microwave filters; while, for lumped-element filters, N corresponds to the number of lumped-resonant (LC) circuits that produce zeros of attenuation at finite frequencies.

And, in terms of a measurable electrical characteristic, N corresponds to the number of positive-frequency passband reflection ($S|_{11}|$) zeros.

PPD & PPV. Parameters PPD and PPV work together to specify the characteristic of the filter's passband. PPD is used to indicate what the value of PPV represents. The flexibility these parameters provide eliminates the need to manually convert from the passband specification parameter of one's preference into whatever specific parameter the software was written to accept.

TZF & TZR. List parameters TZF and TZR are used to specify the complex transmission zeros, Z, of the bandpass filter response. Of the N positive-frequency complex transmission zeros, $Z_i = TZR_i + j\ TZF_i$, the model allows up to M -1 to be specified where M = N / 2 if N is even and M = (N + 1)/2 if N is odd. Each consists of a real part, TZR_i, and an imaginary part, the real frequency TZF_i. If TZR_i is not specified, it is assumed to be zero. Note that the user must provide transmission zeros with nonzero real parts in pairs, that is, for each zero A+jB transmission, zero -A+jB must be present. Each unspecified transmission zero is mapped to a normalized lowpass prototype frequency of infinity.

Parameter Restrictions and Recommendations

1. 32 > N > 1.
2. FP1 > 0.

   FP2 > 0.

   FP1 ≠ FP2.
3. If PPD = "Maximum Insertion Loss", then PPV > 0.

   If PPD = "Minimum Return Loss", then PPV > 0.

   If PPD = "Maximum VSWR", then PPV > 1.
4. $TZF_i > 0$.

   If TZR_i is specified, then TZF_i must be specified.

   If TZR_i is zero, then $FP1 > TZF_i > FP2$.

   If $TZR_i \neq 0$, there must be a $TZR_k = -TZR_i$ and a $TZF_k = TZF_i$.
5. RS > 0.

   RL > 0.
6. QU > 0 specifies a finite unloaded Q (recommend $QU < 1e^{12}$).

   QU = 0 specifies an infinite unloaded Q.

Implementation Details

The model is implemented as a short-circuit admittance matrix,

$$\begin{bmatrix} y_{11} & y_{12} \\ y_{21} & y_{22} \end{bmatrix}$$

, whose equivalent normalized lowpass prototype transfer function, $S_{21}(s)$, is [1, 2]:

$$s_{21}^2(s) = \frac{P_N^2(s)}{P_N^2(s) + \varepsilon^2 F_N^2(s)} = \frac{P_N^2(s)}{P_N(s) + j\varepsilon F_N(s))(P_N(s) - j\varepsilon F_N(s))} = \frac{P_N^2(s)}{\varepsilon^2 E_N^2(s)}$$

,

where F_N and E_N are normalized lowpass prototype polynomials of order N, and

$$\exp(-s^2) \approx \sum_{i=0}^{N} \frac{(-1)^i s^{2i}}{i!} = |g(s)|^2 = 1 + |h(s)|^2$$

$$\varepsilon^2 = \frac{1}{10^{PPV/10} - 1} \text{when PPD = " Minimum Return Loss "}$$

$$\varepsilon^2 = \frac{0.25(\mathrm{PPV} - 1)^2}{\mathrm{PPV}} \text{ when PPD="Maximum VSWR"}$$

$$P_N(s) = \prod_{i=1}^{N}\left(1 - \frac{s}{z_i}\right)$$

$$s = \frac{1}{q_u} + j\omega$$

$$j = \sqrt{-1}$$

A specified bandpass transmission zero, Z[i] = TZR[i] + j TZF[i] is mapped to a normalized lowpass prototype transmission zero, z[i], using [5] [5–23]:

$$z[i] = \frac{(Z[i]) + (FP2 \times FP1)}{|\ FP2 - FP1\ |\ Z[i]}$$

And, _FREQ (the variable that represents the project frequency) is mapped to the normalized lowpass prototype radian frequency, ω, using [5] [5–23]:

$$\omega = \frac{\_\mathrm{FREQ} - (FP2 \times FP1) /\ \_\mathrm{FREQ}}{\sqrt{FP2 + FP1}}$$

while the specified bandpass resonator unloaded Q, QU, is converted to an equivalent lowpass prototype element unloaded Q,q_u, using [5] [5–23][6] [5–23]:

$$q_u = \mathrm{QU}\frac{|\ \mathrm{FP2} - \mathrm{FP1}\ |}{\sqrt{\mathrm{FP2} \times \mathrm{FP1}}}$$

Polynomial

$$F_N(s) = F_i|_{i=N}$$

is constructed using a doubly recursive algorithm [1] [5–23][3] [5–23]:

$$G_0 = b_1$$

$$F_1 = a_1$$

$$G_{(i-1)} = a_i G_{(i-2)} + b_i F_{(i-1)}$$

$$F_i = a_i F_{(i-1)} + c_i G_{(i-2)}$$

where i = 2 to N and, employing the normalized lowpass prototype transmission zero, z_k, for k = 1 to N:

$$a_k = -j\left(s + \frac{1}{z_k}\right)$$

$$b_k = \sqrt{1 + \left(\frac{1}{z_k}\right)^2}$$

$$c_k = -b_k(s^2 + 1)$$

And, polynomial $E_N(s)$ is found by applying the "alternating singularity principle" [1] [5–23][2] [5–23][4] [5–23] to the roots of

$$\left(\frac{P_N(s)}{\varepsilon} + \mathrm{jF}_N(s)\right)$$

. Then, E_N and F_N are split into complex-even and complex-odd polynomials [2] [5–23] such that $E_N = E_e + E_0$ and $F_N = F_e + F_0$, where

$$E_e = \mathrm{Re}(e_0) + j \quad \mathrm{Im}(e_1)s + \mathrm{Re}(e_2)s^2 + \cdots, E_o = j \quad \mathrm{Im}(e_o) + \mathrm{Re}(e_1) + j \quad \mathrm{Im}(e_2)s^2 + \cdots$$

$$F_e = \mathrm{Re}(f_0) + j\,\mathrm{Im}(f_1)s + \mathrm{Re}(f_2)s^2 + \cdots, F_o = j\,\mathrm{Im}(f_o) + \mathrm{Re}(f_1) + j\,\mathrm{Im}(f_2)s^2 \cdots$$

and e_1 and f_1 (i = 0 to N) are the complex coefficients of E_N and F_N. Finally [3] [5–23]:

$$y_{11} = \left(\frac{1}{\mathrm{RS}}\right)\left(\frac{E_e - F_e}{E_o + F_o}\right)$$

$$y_{22} = \left(\frac{1}{\mathrm{RL}}\right)\left(\frac{E_e + F_e}{E_o + F_o}\right)$$

$$y_{12} = y_{21} = \left(\frac{1}{\sqrt{\mathrm{RS} \times \mathrm{RL}}}\right)\left(\frac{-P_N/\varepsilon}{E_o + F_o}\right)$$

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

Recommendations for Use

The transmission zeros can be tuned or optimized by assigning variables to the elements of the TZF and/or TZR lists and then tuning or optimizing these variables.

Note that this model behaves as if it has ideal impedance transformers at its ports, so there is no attenuation due to mismatched source and load impedances. The model expects that the source impedance will equal RS and that the load impedance will equal RL, but RS need not have any special relationship to RL for ideal transmission (as would normally be the case).

References

[1] Richard J. Cameron, "Fast generation of Chebyshev filter prototypes with asymmetrically-prescribed transmission zeros," ESA J., vol. 6, pp. 83-95, 1982.

[2] Richard J. Cameron, "General coupling matrix synthesis methods for Chebyshev filtering functions," IEEE Trans. Microwave Theory Tech., vol. 47, no. 4, pp. 433-442, April 1999.

[3] Douglas R. Jachowski, unpublished notes, 1995 and 2002.

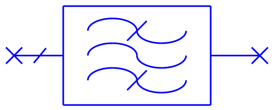
[4] J. D. Rhodes and A. S. Alseyab, "The generalized Chebyshev low pass prototype filter," Int. J. Circuit Theory Applicat., vol. 8, pp. 113-125, 1980.

[5] H. J. Blinchikoff and A. I. Zverev, Filtering in the Time and Frequency Domains, (Robert E. Krieger Publishing Co., 1987), pp. 172, 272.

[6] George L. Matthaei, Leo Young, and E. M. T. Jones, Microwave Filters, Impedance-Matching Networks, and Coupling Structures, (Artech House, 1980), pp. 149-156.

Bessel Bandpass Filter-Maximally Flat Delay: BPFD

Symbol



Summary

BPFD models represent lumped-element Bessel-Thomson bandpass filters. They offer simplicity and maximally flat group delay, but suffer from poor selectivity.

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	BPFD1
N	Number of reactances in the filter		3
FP1	Lower edge of passband (when Qu is infinite).	Frequency	0.5 GHz
FP2	Upper edge of passband (when Qu is infinite).	Frequency	1.5 GHz
AP	Passband corner attenuation (when Qu is infinite).	DB	3.0103 dB
*RS	Expected source resistance.	Resistance	50 ohm
*RL	Expected Load resistance	Resistance	50 ohm
*QU	Average unloaded Q of filter resonators		1e12

* indicates a secondary Parameter

Parameter Restrictions and Recommendations

1. 0 < N < 34
2. 0 < FP1
3. 0 < FP2
4. 0 < AP Recommend AP greater than or equal to 0.001 dB.
5. 0 < RS
6. 0 < RL
7. 0 < QU. Recommend QU less than or equal to 1e12.

Implementation Details

The model is implemented as a short-circuit admittance matrix,

$$\begin{bmatrix} y_{11} & y_{12} \\ y_{21} & y_{22} \end{bmatrix}$$

, whose equivalent transfer function squared magnitude is that of a Bessel-Thomson filter:

$$|S_{21}(s)|^2 = \frac{b_0^2}{|g(s)|^2} = \frac{b_0^2}{1 + |h(s)|^2}$$

where

$$g(s) = \sum_{i=0}^{N} b_i s^i$$

$$b_i = \frac{(2N-i)!}{2^{N-i}(i)!(N-i)!} \text{ for } i = 0, \ldots, N$$

Where b_0 = (2N-1)!!,i.e, the product of all odd integers less than 2N.

$$s = \frac{\sqrt{\mathrm{FP2} \times \mathrm{FP1}}}{\mathrm{QU} \times |\mathrm{FP2} - \mathrm{FP1}|} + j\omega$$

$$j = \sqrt{-1}$$

and a lowpass-to-bandpass frequency transformation has been applied:

$$\omega = \frac{\_\mathrm{FREQ} - (\mathrm{FP2} \times \mathrm{FP1}) / \_\mathrm{FREQ}}{|\mathrm{FP2} - \mathrm{FP1}|}$$

_FREQ is the variable containing the project frequency, and the admittances are:

$$y_{11} = \left(\frac{1}{\mathrm{RS}}\right) \frac{g(s) + g(-s) - h(s) - h(-s)}{g(s) - g(-s) + h(s) - h(-s)}$$

$$y_{22} = \left(\frac{1}{\mathrm{RL}}\right) \frac{g(s) + g(-s) + h(s) + h(-s)}{g(s) - g(-s) + h(s) - h(-s)}$$

$$y_{12} = y_{21} = \left(\frac{1}{\sqrt{RS \times RL}}\right) \frac{(-2)b_0}{g(s) - g(-s) + h(s) - h(-s)}$$

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Recommendations for Use

Note that this model behaves as if it has ideal impedance transformers at its ports, so there is no attenuation due to mismatched source and load impedances. The model expects that the source impedance will equal RS and that the load impedance will equal RL, but RS need not equal RL for ideal transmission (as would normally be the case).

References

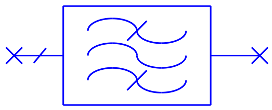
[1] Rolf Schaumann, Mohammed S. Ghausi, and Kenneth R. Laker, Design of Analog Filters: Passive, Active RC, and Switched Capacitor, (Prentice-Hall, 1990), pp. 51-56.

[2] Louis Weinberg, Network Analysis and Synthesis, (Robert E. Krieger Publishing, 1975), pp. 499-506.

[3] Herman J. Blinchikoff and Anatol I. Zverev, Filtering in the Time and Frequency Domains, (Robert E. Krieger Publishing Co., 1987), pp. 124-127.

Elliptic-Function Bandpass Filter (Closed Form): BPFE

Symbol



Summary

BPFE models represent lumped-element elliptic-function (Cauer) bandpass filters. The insertion loss ripples between zero and a specified maximum in the passband. The stopband attenuation increases rapidly between the passband edges and the stopband edges, and ripples between a specified minimum stopband attenuation and infinity. This type of filter offers optimum selectivity at the expense of increased complexity and poor group delay flatness.

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	BPFE1
N	Order of filter's lowpass prototype.		3
FP1	Passband corner frequency (when Qu is infinite).	Frequency	0.5 GHz
FP2	Passband corner frequency (when Qu is infinite).	Frequency	1.5 GHz
AP	Maximum Passband Insertion Loss (when Qu is infinite).	DB	0.1 dB
AS	Minimum Stopband Attenuation (when Qu is infinite).	DB	20 dB
*FS1	Lower stopband corner frequency (when Qu is infinite).	Frequency	0 GHz
*FS2	Upper stopband corner frequency (when Qu is infinite).	Frequency	0 GHz
*DM	Where to put design margin (see below for details).		1
*RS	source resistance.	Resistance	50 ohm
*RL	Load resistance	Resistance	50 ohm
*QU	Average unloaded Q of the bandpass resonators.		0

* indicates a secondary Parameter

Parameter Details

An elliptic-function bandpass filter is completely specified by any three of the following four parameters:

- lowpass prototype filter order, N,
- maximum passband insertion loss, AP,
- minimum stopband attenuation, AS, and
- selectivity.

Filter selectivity, in turn, is completely specified by any three of the following four parameters:

- frequency of the upper edge of the lower stopband, FS1,
- frequency of the lower edge of the passband, FP1,

- frequency of the upper edge of the passband, FP2, and
- frequency of the lower edge of the upper stopband, FS2.

There are eighteen valid sets of these parameters, and these are listed in the table below. Columns 2 through 9 represent individual parameters of the filter, while the rows represent the valid parameter sets - which are numbered in column 1. The last column indicates which parameter is unspecified, or is incompletely specified, in the set. For each specific parameter, the body of the table indicates whether that parameter is specified, unspecified, or ignored in a particular parameter set. "S" means the parameter value is specified (non-zero), "0" means the parameter value has been set to zero (indicating it is intentionally unspecified), and "x" means the parameter value is ignored by the model.

Set	N	AP	AS	FP1	FP2	FS1	FS2	DM	Incompletely Specified Parameter
1	S	S	S	S	S	x	x	x	Selectivity
2	S	S	S	S	0	S	x	x	Selectivity
3	S	S	S	S	0	0	S	x	Selectivity
4	S	S	S	0	S	S	x	x	Selectivity
5	S	S	S	0	S	0	S	x	Selectivity
6	S	S	S	0	0	S	S	x	Selectivity
7	S	S	0	S	S	S	x	x	AS
8	S	S	0	S	S	0	S	x	AS
9	S	S	0	S	0	S	S	x	AS
10	S	S	0	0	S	S	S	x	AS
11	S	0	S	S	S	S	x	x	AP
12	S	0	S	S	S	0	S	x	AP
13	S	0	S	S	0	S	S	x	AP
14	S	0	S	0	S	S	S	x	AP
15	0	S	S	S	S	S	x	S	N
16	0	S	S	S	S	0	S	S	N
17	0	S	S	S	0	S	S	S	N
18	0	S	S	0	S	S	S	S	N

N. In mathematical terms, N is defined as the highest exponent of the complex frequency variable s in the transfer function, S21(s), of the filter's normalized lowpass prototype, or, equivalently, half of the highest exponent of s in the transfer function of the bandpass filter. In terms of the number of circuit components, N corresponds to the total number of resonances in single-mode or multi-mode direct-coupled-resonator microwave filters; while, for lumped-element filters, N corresponds to the number of lumped-resonant (LC) circuits that produce zeros of attenuation at finite frequencies. And, in terms of a measurable electrical characteristic, N corresponds to the number of positive-frequency passband reflection (|S11|) zeros.

DM. If zero is specified for N, the model will determine N. But, since N must be an integer, there will typically be some design margin available, and this margin must be assigned to some parameter other than N. Consequently, after determining N, the model will compute a new value for one or more of the other parameters. The value of DM determines which parameter, or parameters, will have their values recomputed after a value is synthesized for N. For each valid value of

DM, the table below specifies which parameters (X) are effected by the additional design margin and the manner in which they are effected.

DM	AP	AS	FP1	FP2	FS1	FS2	Description
0	X						decrease AP
1		X					increase AS
2			X	X			Decrease FP1 Increase FP2
3			X		X		Decrease FS1 & FP1
4			X			X	Decrease FP1 & FS2
5				X	X		Increase FS1 & FP2
6				X		X	Increase FP2 & FS2
7					X	X	Increase FS1 Decrease FS2
8			X	X	X	X	Increase FS1 & FP2 Decrease FP1 & FS2

Parameter Restrictions and Recommendations

1. If any of the parameters N, AP, AS, FP1, FP2, FS1, or FS2 are set to 0, the model considers them "unspecified." A value of zero for any of these parameters may, or may not, be valid, depending on whether the set of parameters as a whole corresponds to one of the valid parameter sets listed in the table, above.
2. If N is specified, it must fall within the range $0 < N < 27$.

   If N=0, then $0 \leq DM \leq 8$, otherwise DM is ignored.
3. If Ap and AS are both specified, $0 < AP < AS$, otherwise $0 < AP$ or $0 < AS$.

   Recommend AP greater than or equal to 0.001 dB.
4. "Selectivity" is fully specified by any three of FS1, FP1, FP2, and FS2.

   If FS1, FP1 & FP2 are specified, $0 < FS1 < FP1 < FP2$.

   If FS1, FP1 & FS2 are specified, $0 < FS1 < FP1 < FS2$.

   If FS1, FP2 & FS2 are specified, $0 < FS1 < FP2 < FS2$.

   If FP1, FP2 & FS2 are specified, $0 < FP1 < FP2 < FS2$.
5. "Selectivity" is partially specified by any two of FS1, FP1, FP2, and FS2.

   If FS1 & FP1 are specified, $0 < FS1 < FP1$.

   If FS1 & FP2 are specified, $0 < FS1 < FP2$.

If FS1 & FS2 are specified, 0 < FS1 < FS2.

If FP1 & FP2 are specified, 0 < FP1 < FP2.

If FP1 & FS2 are specified, 0 < FP1 < FS2.

If FP2 & FS2 are specified, 0 < FP2 < FS2.

6. 0 < RS.

   0 < RL.

7. QU > 0 specifies a finite unloaded Q (recommend QU ≤ 1e12).

   QU = 0 specifies an infinite unloaded Q.

Implementation Details

The model is implemented as a short-circuit admittance matrix,

$$\begin{bmatrix} y_{11} & y_{12} \\ y_{21} & y_{22} \end{bmatrix}$$

, whose equivalent normalized lowpass prototype transfer function, S21(s), is:

$$|S_{21}(s)|^2 = \frac{1}{1+\varepsilon^2 R_n^2\left(\frac{s}{j}\right)} = \frac{f(s)^2}{|g(s)|^2} = \frac{f(s)^2}{1+|h(s)|^2}$$

where Rn is the Elliptic Rational Function, and

$$\varepsilon^2 = 10^{\mathrm{AP}/10} - 1$$

$$R_n(\omega) = C \times \omega^{\Pi} \times \prod_{i=1}^{\frac{N-\Pi}{2}} \left(\frac{\omega^2 - \omega_i^2}{\omega^2 - \frac{\omega_s^2}{\omega_i^2}} \right)$$

$$C = \prod_{i=1}^{\frac{N-\Pi}{2}} \left(\frac{1 - \frac{\omega_s^2}{\omega_i^2}}{1 - \omega_i^2} \right)$$

$$\omega_i = \mathrm{cd}\left(\frac{2i-1}{N} K\left(\frac{1}{\omega_s^2} \right), \frac{1}{\omega_s^2} \right)$$

$$s = \frac{\sqrt{\mathrm{FP2} \times \mathrm{FP1}}}{\mathrm{QU} \times |\mathrm{FP2} - \mathrm{FP1}|} + j\omega$$

$$\Pi = 0 \text{ if N is even, } \Pi = 1 \text{ if N is odd}$$

$$j = \sqrt{-1}$$

where cd is a Jacobian elliptic function, K is Legendre's complete elliptic integral of the first kind, and a bandpass-to-lowpass frequency transformation has been applied:

$$\omega = \frac{\_\mathrm{FREQ} - (\mathrm{FP2} \times \mathrm{FP1}) / \_\mathrm{FREQ}}{|\ \mathrm{FP2} - \mathrm{FP1}\ |}$$

and

$$\omega_s = \frac{\mathrm{FS2} - (\mathrm{FP2} \times \mathrm{FP1}) / \mathrm{FS2}}{|\ \mathrm{FP2} - \mathrm{FP1}\ |}$$

Note that _FREQ is the variable containing the project frequency and the frequency parameters FP1, FP2, FS1, and FS2 are related according to: FP1xFP2=FS1xFS2

And the seven parameters of the elliptic-function bandpass filter (N, AP, AS, FP1, FP2, FS1, and FS2) are related by "the degree equation":

$$N = \left(\frac{K(m)}{K(1-m)}\right)\left(\frac{K(1-m_o)}{K(m_o)}\right)$$

where

$$m_o = \frac{10^{\mathrm{AP}/10} - 1}{10^{\mathrm{AS}/10} - 1}$$

and

$$m = \left|\ \frac{|\ \mathrm{FP2} - \mathrm{FP1}}{\mathrm{FS2} - (\mathrm{FP2} \times \mathrm{FP1}) / \mathrm{FS2}}\ \right|^2$$

The admittances are given by:

$$y_{11} = \left(\frac{1}{\mathrm{RS}}\right)\frac{g(s) + g(-s) - h(s) - h(-s)}{g(s) - g(-s) + h(s) - h(-s)}$$

$$y_{22} = \left(\frac{1}{\mathrm{RL}}\right)\frac{g(s) + g(-s) + h(s) + h(-s)}{g(s) - g(-s) + h(s) - h(-s)}$$

$$y_{12} = y_{21} = \left(\frac{1}{\sqrt{\mathrm{RS} \times \mathrm{RL}}}\right)\frac{2 \times f(s)}{g(s) - g(-s) + h(s) - h(-s)}$$

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

Recommendations for Use

Note that this model behaves as if it has ideal impedance transformers at its ports, so there is no attenuation due to mismatched source and load impedances. The model expects that the source impedance will equal RS and that the load impedance will equal RL, but RS need not have any special relationship to RL for ideal transmission (as would normally be the case).

References

[1] Miroslav D. Lutovac, Dejan V. Tosic, and Brian L. Evans, Filter Design For Signal Processing Using MATLAB and Mathematica, (Prentice Hall, 2001), Chapters 6, 12, and 13.

[2] Alexander J. Grossman, "Synthesis of Tchebycheff parameter symmetrical filters," Proceedings of the IRE, pp. 545-473, April 1957.

[3] Dante Youla, "A tutorial exposition of some key network-theoretic ideas underlying classical insertion-loss filter design," Proc. IEEE, vol. 59, no. 5, pp. 760-799, May 1971.

[4] Miroslav Vleck and Rolf Unbehauen, "Degree, ripple, and transition width of elliptic filters," IEEE Trans. Circuits Syst., vol. 36, no. 3, pp. 469-472, March 1989.

[5] H. J. Orchard and Alan N. Willson, Jr., "Elliptic functions for filter design," IEEE Trans. Circuits Syst., I, vol. 44, no. 4, pp. 273-287, April 1997.

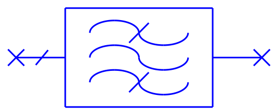
[6] Rolf Schaumann, Mohammed S. Ghausi, and Kenneth R. Laker, Design of Analog Filters: Passive, Active RC, and Switched Capacitor, (Prentice-Hall, 1990), pp. 49-51.

[7] Kendall L. Su, Handbook of Tables for Elliptic-Function Filters, (Kluwer Academic, 1990).

[8] Handbook of Mathematical Functions With Formulas, Graphs, and Mathematical Tables, edited by Milton Abramowitz and Irene A. Stegun, (U. S. National Bureau of Standards, 1964), Chapters 16 and 17 by L. M. Milne-Thomson.

Gaussian Bandpass Filter (Closed Form): BPFG

Symbol



Summary

BPFG models represent lumped-element Gaussian bandpass filters. They approximate the ideal Gaussian magnitude response and offer simplicity, relatively flat group delay, and good time domain performance, but suffer from poor frequency selectivity. Although similar to Bessel-Thomson filters, Gaussian filters offer faster rise times and lower transient overshoots, but have slightly less stopband attenuation and less group delay flatness.

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	BPFG1
N	Number of reactances in the filter		3
FP1	Passband lower band-edge frequency (when Qu is infinite).	Frequency	0.5 GHz
FP2	Passband upper band-edge frequency (when Qu is infinite).	Frequency	1.5 GHz
AP	Maximum Passband corner attenuation (when Qu is infinite).	DB	3.0103 dB
*RS	Expected source resistance.	Resistance	50 ohm
*RL	Expected Load resistance	Resistance	50 ohm
*QU	Average unloaded Q of the filter resonators		1e12

** indicates a secondary parameter*

Parameter Restrictions and Recommendations

1. 0 < N < 51
2. 0 < FP1
3. 0 < FP2
4. 0 < AP Recommend AP greater than or equal to 0.001 dB.
5. 0 < RS
6. 0 < RL
7. 0 < QU.
8. Recommend QU less than or equal to 1e12.

Implementation Details

The model is implemented as a short-circuit admittance matrix,

$$\begin{bmatrix} y_{11} & y_{12} \\ y_{21} & y_{22} \end{bmatrix}$$

, whose equivalent transfer function squared magnitude is that of an ideal Gaussian filter. The ideal Gaussian squared magnitude characteristic is:

$$|S_{21}|^2 = \frac{1}{\exp(-s^2)}$$

In the model, the denominator of this ideal Gaussian characteristic is approximated by a truncated Maclaurin series:

$$\exp(-s^2) \approx \sum_{i=0}^{N} \frac{(-1)^i s^{2i}}{i!} = |g(s)|^2 = 1 + |h(s)|^2$$

where

$$s = \frac{\sqrt{\text{FP2} \times \text{FP1}}}{\text{QU} \times |\text{FP2} - \text{FP1}|} + j\omega$$

$$j = \sqrt{-1}$$

and a lowpass-to-bandpass frequency transformation has been applied:

$$\omega = \frac{\text{\_FREQ} - (\text{FP2} \times \text{FP1}) / \text{\_FREQ}}{|\text{FP2} - \text{FP1}|}$$

_FREQ is the variable containing the project frequency, and the admittances are:

$$y_{11} = \left(\frac{1}{\text{RS}}\right)\frac{g(s) + g(-s) - h(s) - h(-s)}{g(s) - g(-s) + h(s) - h(-s)}$$

$$y_{22} = \left(\frac{1}{\text{RL}}\right)\frac{g(s) + g(-s) + h(s) + h(-s)}{g(s) - g(-s) + h(s) - h(-s)}$$

$$y_{12} = y_{21} = \left(\frac{1}{\sqrt{\text{RS} \times \text{RL}}}\right)\frac{(-2)}{g(s) - g(-s) + h(s) - h(-s)}$$

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Recommendations for Use

Note that this model behaves as if it has ideal impedance transformers at its ports, so there is no attenuation due to mismatched source and load impedances. The model expects that the source impedance will equal RS and that the load impedance will equal RL, but RS need not equal RL for ideal transmission (as would normally be the case).

References

[1] Milton Dishal, "Gaussian-Response Filter Design," Electrical Communication, vol. 36, March 1959, pp. 3-26.

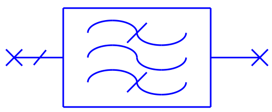
[2] Anatol I. Zverev, Handbook of Filter Synthesis, (John Wiley & Sons, 1967), pp. 67, 70, 71, 73, 74, 90, 91.

[3] DeVerl. S. Humpherys, The Analysis, Design, and Synthesis of Electrical Filters, (Prentice-Hall, 1970), pp. 413-417.

[4] Herman J. Blinchikoff and Anatol I. Zverev, Filtering in the Time and Frequency Domains, (Robert E. Krieger Publishing Co., 1987), pp. 130-132.

Raised-Cosine Bandpass Filter (Closed Form): BPFRC

Symbol



Summary

BPFRC models represent Raised-Cosine Bandpass filters. Raised-Cosine filters are applied in digital transmission system theory to represent ideal (intersymbol interference free) Nyquist filtering of impulse and pulse data transmissions. Hardware designers attempt to design channel filters which approximate these characteristics and have the smallest possible bandwidth.

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	BPFRC1
FP1	Lower frequency edge of passband	Frequency	0.5 GHz
FP2	Upper frequency edge of passband	Frequency	1.5 GHz
A	Roll-off factor		0.5
TYP	Transmission type	Vector Text(pull-down)	Impulse
E	Exponent of the raised-cosine response		1.
*RS	Expected source resistance	Resistance	50 ohm
*RL	Expected load resistance	Resistance	50 ohm

* indicates a secondary Parameter

Parameter Restrictions and Recommendations

1. $0 < FP1$
2. $0 < FP2$
3. $0 \le A \le 1$.

   A minimum bandwidth (A=0) filter would require an infinite number of filter sections. Approximations of a 30% excess Nyquist bandwidth (A=0.3) raised-cosine filter are considered feasible with present-day technology.
4. TYP is either $\{0,1\}$.

   TYP = 0 (Impulse) specifies the ideal raised-cosine response, the theoretical filter model for infinitesimally narrow impulses. TYP = 1 (Pulse) specifies a sinc-normalized raised-cosine response, the filter model for rectangular pulse transmission.
5. $0 < E \le 1$.

The ideal raised-cosine response is raised to the exponent, E. For cascaded identical transmit and receive filters, you would typically specify E=0.5 for each filter. The composite channel response of the cascaded filters would be represented by specifying E=1.

6. 0 < RS.

   By definition, the model matches RS at its input port.

7. 0 < RL.

   By definition, the model matches RL at its output port.

Implementation Details

The model is implemented as a short-circuit admittance matrix,

$$\begin{bmatrix} y_{11} & y_{12} \\ y_{21} & y_{22} \end{bmatrix}$$

, whose equivalent amplitude functions S21 and S12 implement the raised-cosine response for impulse or pulse data transmission, while S11 and S22 are, by definition, constant and matched to the filter port termination resistances (i.e., equal to 1). For impulses,

$$S_{21}(j\omega) = 1 \text{ for } [0 \le \omega \le (1 - A)]$$

$$S_{21}(j\omega) = \left(\cos^2\left(\left(\frac{\pi}{4}\right)\left(\frac{\omega + A - 1}{A}\right)\right)\right)^E \text{ for } [(1 - A) \le \omega \le (1 + A)]$$

$$S_{21}(j\omega) = 0 \text{ for } [(1 + A) \le \omega]$$

For pulses,

$$S_{21}(j\omega) = 1 \text{ for } [0 \le \omega \le (1 - A)]$$

$$S_{21}(j\omega) = \frac{\left(\cos^2\left(\left(\frac{\pi}{4}\right)\left(\frac{\omega + A - 1}{A}\right)\right)\right)^E}{\operatorname{sinc}\left(\frac{\pi}{2}\omega\right)} \text{ for } [(1 - A) \le \omega \le (1 + A)]$$

$$S_{21}(j\omega) = 0 \text{ for } [(1 + A) \le \omega]$$

where

$$\operatorname{sinc}(x) = \frac{\sin(x)}{x}$$

$$j = \sqrt{-1}$$

and a lowpass-to-bandpass frequency transformation has been applied:

$$\omega = \frac{\_FREQ - (FP2 \times FP1)/\_FREQ}{|\ FP2 - FP1\ |}$$

_FREQ is the variable containing the project frequency, and the admittances are:

$$y_{11} = \left(\frac{1}{\mathrm{RS}}\right)\left(\frac{-S_{21}^2(j\omega)}{S_{21}^2(j\omega)-4}\right)$$

$$y_{22} = \left(\frac{1}{\mathrm{RL}}\right)\left(\frac{-S_{21}^2(j\omega)}{S_{21}^2(j\omega)-4}\right)$$

$$y_{22} = \left(\frac{1}{\sqrt{\mathrm{RS}\times\mathrm{RL}}}\right)\left(\frac{2S_{21}(j\omega)}{S_{21}^2(j\omega)-4}\right)$$

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Recommendations for Use

The model expects that the source impedance will equal RS and that the load impedance will equal RL, but RS need not equal RL for ideal transmission.

References

[1] Kamilo Feher, Digital Communications: Microwave Applications, (Prentice-Hall, 1981), pp. 46-51.

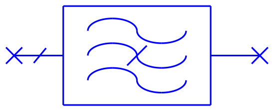
[2] John G. Proakis, Digital Communications, 2nd Ed., (McGraw-Hill, 1989), pp. 532-536.

[3] John Bellamy, Digital Telephony, 2nd Ed., (John Wiley & Sons, 1991), pp. 523-528.

[4] E. A. Lee and D. G. Messerschmitt, Digital Communications, 2nd Ed., (Kluwer Academic Publishers, 1994), pp. 188-191, 226-228.

Butterworth BandStop Filter: BSFB

Symbol



Summary

BSFB models represent lumped-element Butterworth bandstop filters. They offer simplicity and a compromise between high selectivity and flat group delay.

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	BSFB1
N	Number of resonators in the filter		3
FP1	Lower frequency edge of passband (when Qu is infinite).	Frequency	0.5 GHz
FP2	Upper frequency edge of passband (when Qu is infinite).	Frequency	1.5 GHz
*AP	Maximum passband attenuation (when Qu is infinite).	DB	3.0103 dB
*RS	Expected source resistance.	Resistance	50 Ohm
*RL	Expected load resistance	Resistance	50 Ohm
*QU	Average unloaded Q of filter resonators		1e12

** indicates a secondary parameter*

Parameter Restrictions and Recommendations

1. 0 < N < 29
2. 0 < FP1
3. 0 < FP2
4. 0 < AP Recommend AP greater than or equal to 0.001 dB.
5. 0 < RS
6. 0 < RL
7. 0 < QU. Recommend QU less than or equal to 1012.

Implementation Details

The model is implemented as a short-circuit admittance matrix, whose equivalent transfer function squared magnitude is that of a Butterworth filter:

$$|S_{21}(s)|^2 = \frac{1}{1+(-1)^N\varepsilon^2 S^{2N}} = \frac{1}{|g(s)|^2} = \frac{1}{1+|h(s)|^2}$$

where

$$\varepsilon^2 = 10^{\mathrm{AP}/10} - 1$$

$$s = \frac{\sqrt{\mathrm{FP2} \times \mathrm{FP1}}}{\mathrm{QU} \times \ |\ \mathrm{FP2} - \mathrm{FP1}\ |} + j\omega$$

$$j = \sqrt{-1}$$

and a lowpass-to-bandpass (not lowpass-to-bandstop) frequency transformation has been applied:

$$\omega = \frac{\_\mathrm{FREQ} - (\mathrm{FP2} \times \mathrm{FP1}) / \_\mathrm{FREQ}}{|\ \mathrm{FP2} - \mathrm{FP1}\ |}$$

_FREQ is the variable containing the project frequency, and the admittances are:

$$y_{11} = \left(\frac{1}{\mathrm{RS}}\right)\frac{g(s) + g(-s) - h(s) - h(-s)}{g(s) - g(-s) + h(s) - h(-s)}$$

$$y_{22} = \left(\frac{1}{\mathrm{RL}}\right)\frac{g(s) + g(-s) + h(s) + h(-s)}{g(s) - g(-s) + h(s) - h(-s)}$$

$$y_{12} = y_{21} = \left(\frac{1}{\sqrt{\mathrm{RS} \times \mathrm{RL}}}\right)\frac{2}{g(s) - g(-s) + h(s) - h(-s)}$$

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

Recommendations for Use

Note that this model behaves as if it has ideal impedance transformers at its ports, so there is no attenuation due to mismatched source and load impedances. The model expects that the source impedance will equal RS and that the load impedance will equal RL, but RS need not equal RL for ideal transmission (as would normally be the case).

References

[1] Rolf Schaumann, Mohammed S. Ghausi, and Kenneth R. Laker, Design of Analog Filters: Passive, Active RC, and Switched Capacitor, (Prentice-Hall, 1990), pp. 40-44.

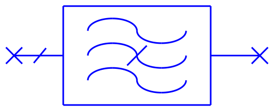
[2] Louis Weinberg, Network Analysis and Synthesis, (Robert E. Krieger Publishing, 1975), pp. 493-498.

[3] Adel S. Sedra and Peter O. Brackett, Filter Theory and Design: Active and Passive, (Matrix Publishers, 1978), pp. 105-111.

[4] George L. Matthaei, Leo Young, and E. M. T. Jones, Microwave Filters, Impedance-Matching Networks, and Coupling Structures, (Artech House, 1980), pp. 149-161.

Chebyshev Bandstop Filter: BSFC

Symbol



Summary

BSFC models represent lumped-element Chebyshev bandstop filters. The insertion loss ripples between zero and a specified maximum in the passbands. The stopband attenuation increases rapidly and monotonically beyond the passband edges. This filter type offers simplicity and good selectivity.

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	BSFC1
N	Number of resonators in the filter		3
FP1	Lower frequency edge of passband (when Qu is infinite).	Frequency	0.5 GHz
FP2	Upper frequency edge of passband (when Qu is infinite).	Frequency	1.5 GHz
AP	Maximum Passband attenuation (when Qu is infinite).	DB	0.1 dB
*RS	Expected source resistance.	Resistance	50 ohm
*RL	Expected load resistance	Resistance	50 ohm
*QU	Average unloaded Q of filter resonators		1e12

* indicates a secondary Parameter

Parameter Restrictions and Recommendations

1. 0 < N < 27
2. 0 < FP1
3. 0 < FP2
4. 0 < AP Recommend AP greater than or equal to 0.001 dB.
5. 0 < RS
6. 0 < RL
7. 0 < QU. Recommend QU less than or equal to 1e12.

Implementation Details

The model is implemented as a short-circuit admittance matrix, whose equivalent transfer function squared magnitude is that of a Chebyshev filter:

$$|S_{21}(s)|^2 = \frac{1}{1+\varepsilon^2 C_n^2\left(\frac{s}{j}\right)} = \frac{1}{|g(s)|^2} = \frac{1}{1+|h(s)|^2}$$

where Cn is the Chebyshev polynomial of the first kind, and

$$\varepsilon^2 = 10^{AP/10} - 1$$

$$C_n(\omega) = \cos(N * \mathrm{acos}(\omega)) \qquad \text{for } 0 \leq |\omega| \leq 1$$

$$C_n(\omega) = \cosh(N * \mathrm{acosh}(\omega)) \qquad \text{for } 1 < |\omega|$$

$$s = \frac{\sqrt{\mathrm{FP2} \times \mathrm{FP1}}}{\mathrm{QU} \times |\mathrm{FP2} - \mathrm{FP1}|} + j\omega$$

$$\omega = \frac{\_\mathrm{FREQ} - (\mathrm{FP2} \times \mathrm{FP1}) / \_\mathrm{FREQ}}{|\mathrm{FP2} - \mathrm{FP1}|}$$

and a lowpass-to-bandpass (not lowpass-to-bandstop) frequency transformation has been applied:

$$y_{11} = \left(\frac{1}{\mathrm{RS}}\right)\frac{g(s) + g(-s) - h(s) - h(-s)}{g(s) - g(-s) + h(s) - h(-s)}$$

_FREQ is the variable containing the project frequency, and the admittances are:

$$y_{22} = \left(\frac{1}{\mathrm{RL}}\right)\frac{g(s) + g(-s) + h(s) + h(-s)}{g(s) - g(-s) + h(s) - h(-s)}$$

$$y_{12} = y_{21} = \left(\frac{1}{\sqrt{\mathrm{RS} \times \mathrm{RL}}}\right)\frac{2}{g(s) - g(-s) + h(s) - h(-s)}$$

$$j = \sqrt{-1}$$

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Recommendations for Use

Note that this model behaves as if it has ideal impedance transformers at its ports, so there is no attenuation due to mismatched source and load impedances. The model expects that the source impedance will equal RS and that the load impedance will equal RL, but RS need not have any special relationship to RL for ideal transmission (as would normally be the case).

References

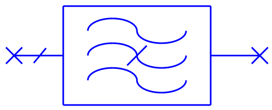
[1] Rolf Schaumann, Mohammed S. Ghausi, and Kenneth R. Laker, Design of Analog Filters: Passive, Active RC, and Switched Capacitor, (Prentice-Hall, 1990), pp. 44-48.

[2] Louis Weinberg, Network Analysis and Synthesis, (Robert E. Krieger Publishing, 1975), pp. 507-529.

[3] George L. Matthaei, Leo Young, and E. M. T. Jones, Microwave Filters, Impedance-Matching Networks, and Coupling Structures, (Artech House, 1980), pp. 149-161.

Generalized Chebyshev Bandstop Filter (Closed Form): BSFCG

Symbol



Summary

BSFCG models represent lumped-element Generalized Chebyshev (or "Quasi-Elliptic") bandstop filters. The insertion loss ripples between zero and a specified maximum in the passband. The stopband attenuation is defined by arbitrarily specified transmission zeros. Real-frequency finite transmission zeros can be specified to improve selectivity at the expense of ultimate stopband attenuation and passband group delay, while complex-plane finite transmission zeros can be specified to provide passband group delay equalization at the expense of selectivity and ultimate stopband attenuation. Generalized Chebyshev filters represents a compromise between the simplicity of Chebyshev filters, the optimum amplitude response of more complicated Elliptic filters, and the phase linearity of Bessel and Gaussian filters. Because this type of filter allows one to make explicit design trade-offs between complexity, selectivity, and group delay equalization, it is often used to meet the demanding requirements of modern communications systems.

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	BSFCG1
N	Order of filter's lowpass prototype		3
FP1	Passband corner frequency (when Qu is infinite).	Frequency	0.5 GHz
FP2	Passband corner frequency (when Qu is infinite).	Frequency	1.5 GHz
*PPD	Passband parameter description:- Maximum Insertion Loss,- Minimum Return Loss, or- VSWR.	Enumerated	Maximum Insertion Loss
PPV	Passband parameter value (when Qu is infinite)	dB or Scalar	0.1 dB
TZF	Real frequency, finite transmission zeros	(Real) Frequency	{1} GHz
*TZR	Real parts of complex finite transmission zeros	(Imaginary) Frequency	{0} GHz
*RS	source resistance	Resistance	50 ohm
*RL	Load resistance	Resistance	50 ohm
*QU	Average unloaded Q of resonances in the filter.		1e12

* indicates a secondary Parameter

Parameter Details

N. In mathematical terms, N is defined as the highest exponent of the complex frequency variable s in the transfer function,$S_{21}(s)$, of the filter's normalized lowpass prototype, or, equivalently, half of the highest exponent of s in the transfer function of the bandpass filter. In terms of the number of circuit components, N corresponds to the total number of resonances in single-mode or multi-mode direct-coupled-resonator microwave filters; while, for lumped-element filters, N corresponds to the number of lumped-resonant (LC) circuits that produce zeros of attenuation at finite frequencies.

And, in terms of a measurable electrical characteristic, N corresponds to the number of positive-frequency passband reflection ($S|_{11}|$) zeros.

PPD & PPV. Parameters PPD and PPV work together to specify the characteristic of the filter's passband. PPD is used to indicate what the value of PPV represents. The flexibility these parameters provide eliminates the need to manually convert from the passband specification parameter of one's preference into whatever specific parameter the software was written to accept.

PTZF & TZR. List parameters TZF and TZR are used to specify the complex transmission zeros, Z, of the bandpass filter response. Of the N positive-frequency complex transmission zeros, $Z_i = TZR_i + jTZF_i$, the model allows up to(N-1) to be specified. Each consists of a real part, TZR_i, and an imaginary part, the real frequency TZF_i. If TZR_i is not specified, it is assumed to be zero. And, each unspecified transmission zero is mapped to a normalized lowpass prototype frequency of infinity (i.e., the geometric center of the bandstop filter's stopband).

Parameter Restrictions and Recommendations

1. 32 > N > 1.
2. FP1 > 0.

   FP2 > 0.

   FP1 ≠ FP2.
3. If PPD = "Maximum Insertion Loss", then PPV > 0.

   If PPD = "Minimum Return Loss", then PPV > 0.

   If PPD = "Maximum VSWR", then PPV > 1.
4. $TZF_i > 0$.

   If TZR_i is specified, then TZF_i must be specified.

   If TZR_i is zero, then $FP1 < TZF_i < FP2$.

   If $TZR_i \neq 0$, there must be a $TZR_k = -TZR_i$ and a $TZF_k = TZF_i$.
5. RS > 0.

   RL > 0.
6. QU > 0 specifies a finite unloaded Q (recommend $QU < 1e^{12}$).

   QU = 0 specifies an infinite unloaded Q.

Implementation Details

The model is implemented as a short-circuit admittance matrix,

$$\begin{bmatrix} y_{11} & y_{12} \\ y_{21} & y_{22} \end{bmatrix}$$

, whose equivalent normalized lowpass prototype transfer function, $S_{21}(s)$, is [1, 2]:

$$s_{21}^2(s) = \frac{P_N^2(s)}{P_N^2(s) + \varepsilon^2 F_N^2(s)} = \frac{P_N^2(s)}{P_N(s) + j\varepsilon F_N(s))(P_N(s) - j\varepsilon F_N(s))} = \frac{P_N^2(s)}{\varepsilon^2 E_N^2(s)}$$

where F_N and E_N are normalized lowpass prototype polynomials of order N, and

$$\exp(-s^2) \approx \sum_{i=0}^{N} \frac{(-1)^i s^{2i}}{i!} = |g(s)|^2 = 1 + |h(s)|^2$$

$$\varepsilon^2 = \frac{1}{10^{PPV/10} - 1} \text{when PPD = " Minimum Return Loss "}$$

$$\varepsilon^2 = \frac{0.25(\text{PPV} - 1)^2}{\text{PPV}} \text{ when PPD="Maximum VSWR"}$$

$$P_N(s) = \prod_{i=1}^{N}\left(1 - \frac{s}{z_i}\right)$$

$$s = \frac{1}{q_u} + j\omega$$

$$j = \sqrt{-1}$$

A specified bandstop transmission zero, Z[i] = TZR[i] + jTZF[i] is mapped to a normalized lowpass prototype transmission zero, z[i], using [5] [5–48]:

$$Z[i] = \frac{|\text{FP2} - \text{FP1}|\, Z[i]}{(Z[i]^2 + (\text{FP1} \times \text{FP2})}$$

Loss is introduced into the bandstop response through the highpass prototype (rather than the lowpass prototype) complex frequency variable. Then, a highpass-to-lowpass transformation is employed to correctly map the loss and real frequency to the lowpass prototype [3] [5–48][5] [5–48][6] [5–48]:

$$s = 1 / \left(\frac{1}{q_u} + j\omega\right)$$

.

And, _FREQ (the variable that represents the project frequency) is mapped to the normalized highpass prototype radian frequency, ω, using [5] [5–48]:

$$\omega = \frac{\text{\_FREQ} - (FP2 \times FP1) / \text{\_FREQ}}{\sqrt{FP2 + FP1}}$$

while the specified bandstop resonator unloaded Q, QU, is converted to an equivalent highpass prototype element unloaded Q, q_u, using [5] [5–48][6] [5–48]:

$$q_u = \mathrm{QU}\frac{|\ \mathrm{FP2} - \mathrm{FP1}\ |}{\sqrt{\mathrm{FP2} \times \mathrm{FP1}}}$$

.

Polynomial

$$F_N(s) = F_i|_{i=N}$$

is constructed using a doubly recursive algorithm [1] [5–48][3] [5–48]:

$$G_0 = b_1$$

$$F_1 = a_1$$

$$G_{(i-1)} = a_i G_{(i-2)} + b_i F_{(i-1)}$$

$$F_i = a_i F_{(i-1)} + c_i G_{(i-2)}$$

where i = 2 to N and, employing the normalized lowpass prototype transmission zero,z_k, for k = 1 to N:

$$a_k = -j\left(s + \frac{1}{z_k}\right)$$

$$b_k = \sqrt{1 + \left(\frac{1}{z_k}\right)^2}$$

$$c_k = -b_k\left(s^2 + 1\right)$$

And, polynomial $E_s(s)$ is found by applying the "alternating singularity principle" [1] [5–48][2] [5–48][4] [5–48] to the roots of

$$\left(\frac{P_N(s)}{\varepsilon} + \mathrm{jF}_N(s)\right)$$

. Then, E_N and F_N are split into complex-even and complex-odd polynomials [2] [5–48] such that E_N= E_e+$E_0$and F_N= F_e+F_0, where

$$E_e = \mathrm{Re}(e_0) + j \quad \mathrm{Im}(e_1)s + \mathrm{Re}(e_2)s^2 + \ \cdots\ ,E_o = j \quad \mathrm{Im}(e_o) + \mathrm{Re}(e_1) + j \quad \mathrm{Im}(e_2)s^2 + \ \cdots$$

$$F_e = \mathrm{Re}(f_0) + j\ \mathrm{Im}(f_1)s + \mathrm{Re}(f_2)s^2 + \ \cdots\ ,F_o = j\ \mathrm{Im}(f_o) + \mathrm{Re}(f_1) + j\ \mathrm{Im}(f_2)s^2\ \cdots$$

andand e_1 and f_1 (i = 0 to N) are the complex coefficients of E_N and F_N. Finally [3]:

$$y_{11} = \left(\frac{1}{\mathrm{RS}}\right)\left(\frac{E_e - F_e}{E_o + F_o}\right)$$

$$y_{22} = \left(\frac{1}{\mathrm{RL}}\right)\left(\frac{E_e + F_e}{E_o + F_o}\right)$$

$$y_{12} = y_{21} = \left(\frac{1}{\sqrt{\mathrm{RS} \times \mathrm{RL}}}\right)\left(\frac{-P_N / \varepsilon}{E_o + F_o}\right)$$

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

Recommendations for Use

The transmission zeros can be tuned or optimized by assigning variables to the elements of the TZF and/or TZR lists and then tuning or optimizing these variables.

Note that this model behaves as if it has ideal impedance transformers at its ports, so there is no attenuation due to mismatched source and load impedances. The model expects that the source impedance will equal RS and that the load impedance will equal RL, but RS need not have any special relationship to RL for ideal transmission (as would normally be the case).

References

[1] Richard J. Cameron, "Fast generation of Chebyshev filter prototypes with asymmetrically-prescribed transmission zeros," ESA J., vol. 6, pp. 83-95, 1982.

[2] Richard J. Cameron, "General coupling matrix synthesis methods for Chebyshev filtering functions," IEEE Trans. Microwave Theory Tech., vol. 47, no. 4, pp. 433-442, April 1999.

[3] Douglas R. Jachowski, unpublished notes, 1995 and 2002.

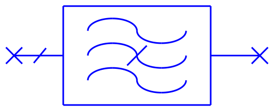
[4] J. D. Rhodes and A. S. Alseyab, "The generalized Chebyshev low pass prototype filter," Int. J. Circuit Theory Applicat., vol. 8, pp. 113-125, 1980.

[5] H. J. Blinchikoff and A. I. Zverev, Filtering in the Time and Frequency Domains, (Robert E. Krieger Publishing Co., 1987), pp. 205-209, 248-249.

[6] George L. Matthaei, Leo Young, and E. M. T. Jones, Microwave Filters, Impedance-Matching Networks, and Coupling Structures, (Artech House, 1980), pp. 149-161.

Bessel Bandstop Filter, Maximally Flat Delay: BSFD

Symbol



Summary

BSFD models represent lumped-element Bessel-Thomson bandstop filters. They offer simplicity and maximally flat group delay, but suffer from poor selectivity.

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	BSFD1
N	Number of resonators in the filter		3
FP1	Passband lower band-edge frequency (when Qu is infinite).	Frequency	0.5 GHz
FP2	Passband upper band-edge frequency (when Qu is infinite).	Frequency	1.5 GHz
AP	Maximum passband attenuation (when Qu is infinite).	DB	3.0103 dB
*RS	Expected source resistance.	Resistance	50 ohm
*RL	Expected Load resistance	Resistance	50 ohm
*QU	Average unloaded Q of filter resonators		1e12

* indicates a secondary Parameter

Parameter Restrictions and Recommendations

1. 0 < N < 34
2. 0 < FP1
3. 0 < FP2
4. 0 < AP Recommend AP greater than or equal to 0.001 dB.
5. 0 < RS
6. 0 < RL
7. 0 < QU. Recommend QU less than or equal to 1e12.

Implementation Details

The model is implemented as a short-circuit admittance matrix,

$$\begin{bmatrix} y_{11} & y_{12} \\ y_{21} & y_{22} \end{bmatrix}$$

, whose equivalent transfer function squared magnitude is that of a Bessel-Thomson filter:

$$|S_{21}(s)|^2 = \frac{b_0^2}{|g(s)|^2} = \frac{b_0^2}{1 + |h(s)|^2}$$

where

$$g(s) = \sum_{i=0}^{N} b_i s^i$$

$$b_i = \frac{(2N-i)!}{2^{N-i}(i)!(N-i)!} \text{ for } i = 0, \ldots, N$$

Where b_0 = (2N-1)!!,i.e, the product of all odd integers less than 2N.

$$|S_{21}(s)|^2 = \frac{b_0^2}{|g(s)|^2} = \frac{b_0^2}{1 + |h(s)|^2}$$

where

$$j = \sqrt{-1}$$

and a lowpass-to-bandpass (not lowpass-to-bandstop) frequency transformation has been applied:

$$\omega = \frac{\_FREQ - (FP2 \times FP1)/\_FREQ}{|FP2 - FP1|}$$

_FREQ is the variable containing the project frequency, and the admittances are:

$$y_{11} = \left(\frac{1}{RS}\right)\frac{g(s) + g(-s) - h(s) - h(-s)}{g(s) - g(-s) + h(s) - h(-s)}$$

$$y_{22} = \left(\frac{1}{RL}\right)\frac{g(s) + g(-s) + h(s) + h(-s)}{g(s) - g(-s) + h(s) - h(-s)}$$

$$y_{12} = y_{21} = \left(\frac{1}{\sqrt{RS \times RL}}\right)\frac{2}{g(s) - g(-s) + h(s) - h(-s)}$$

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Recommendations for Use

Note that this model behaves as if it has ideal impedance transformers at its ports, so there is no attenuation due to mismatched source and load impedances. The model expects that the source impedance will equal RS and that the load impedance will equal RL, but RS need not equal RL for ideal transmission (as would normally be the case).

References

[1] Rolf Schaumann, Mohammed S. Ghausi, and Kenneth R. Laker, Design of Analog Filters: Passive, Active RC, and Switched Capacitor, (Prentice-Hall, 1990), pp. 51-56.

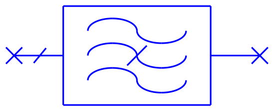
[2] Louis Weinberg, Network Analysis and Synthesis, (Robert E. Krieger Publishing, 1975), pp. 499-506.

[3] Herman J. Blinchikoff and Anatol I. Zverev, Filtering in the Time and Frequency Domains, (Robert E. Krieger Publishing Co., 1987), pp. 124-127.

[4] George L. Matthaei, Leo Young, and E. M. T. Jones, Microwave Filters, Impedance-Matching Networks, and Coupling Structures, (Artech House, 1980), pp. 149-161.

Elliptic-Function Bandstop Filter (Closed Form): BSFE

Symbol



Summary

BSFE models represent lumped-element elliptic-function (Cauer) bandstop filters. The insertion loss ripples between zero and a specified maximum in the passband. The stopband attenuation increases rapidly between the passband edges and the stopband edges, and ripples between a specified minimum stopband attenuation and infinity. This type of filter offers optimum selectivity at the expense of increased complexity and poor group delay flatness.

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	BSFE1
N	Order of filter's lowpass prototype		3
FP1	Passband corner frequency (when Qu is infinite).	Frequency	0.5 GHz
FP2	Passband corner frequency (when Qu is infinite).	Frequency	1.5 GHz
AP	Maximum Passband Insertion Loss (when Qu is infinite).	DB	0.1 dB
AS	Minimum Stopband Attenuation (when Qu is infinite).	DB	20 dB
*FS1	Stopband corner frequency (when Qu is infinite).	Frequency	0 GHz
*FS2	Stopband corner frequency (when Qu is infinite).	Frequency	0 GHz
*DM	Where to put design margin (see below for details).		1
*RS	Source resistance.	Resistance	50 ohm
*RL	Load resistance	Resistance	50 ohm
*QU	Average unloaded Q of the bandstop resonators.		1e12

* indicates a secondary Parameter

Parameter Details

An elliptic-function bandstop filter is completely specified by any three of the following four parameters:

- lowpass prototype filter order, N,
- maximum passband insertion loss, AP,
- minimum stopband attenuation, AS, and
- selectivity.

Filter selectivity, in turn, is completely specified by any three of the following four parameters:

- frequency of the lower edge of the stopband, FS1,
- frequency of the upper edge of the lower passband, FP1,

- frequency of the lower edge of the upper passband, FP2, and
- frequency of the upper edge of the stopband, FS2.

There are eighteen valid sets of these parameters, and these are listed in the table below. Columns 2 through 9 represent individual parameters of the filter, while the rows represent the valid parameter sets - which are numbered in column 1. The last column indicates which parameter is unspecified, or is incompletely specified, in the set. For each specific parameter, the body of the table indicates whether that parameter is specified, unspecified, or ignored in a particular parameter set. "S" means the parameter value is specified (non-zero), "0" means it has been set to zero (indicating it is intentionally unspecified), and "x" means it is ignored by the model.

Set	N	AP	AS	FP1	FP2	FS1	FS2	DM	Incompletely Specified Parameter
1	S	S	S	S	S	x	x	x	Selectivity
2	S	S	S	S	0	S	x	x	Selectivity
3	S	S	S	S	0	0	S	x	Selectivity
4	S	S	S	0	S	S	x	x	Selectivity
5	S	S	S	0	S	0	S	x	Selectivity
6	S	S	S	0	0	S	S	x	Selectivity
7	S	S	0	S	S	S	x	x	AS
8	S	S	0	S	S	0	S	x	AS
9	S	S	0	S	0	S	S	x	AS
10	S	S	0	0	S	S	S	x	AS
11	S	0	S	S	S	S	x	x	AP
12	S	0	S	S	S	0	S	x	AP
13	S	0	S	S	0	S	S	x	AP
14	S	0	S	0	S	S	S	x	AP
15	0	S	S	S	S	S	x	S	N
16	0	S	S	S	S	0	S	S	N
17	0	S	S	S	0	S	S	S	N
18	0	S	S	0	S	S	S	S	N

N. In mathematical terms, N is defined as the highest exponent of the complex frequency variable s in the transfer function, $S_{21}(s)$, of the filter's normalized lowpass prototype, or, equivalently, half of the highest exponent of s in the transfer function of the bandstop filter. And, in terms of a measurable electrical characteristic, N corresponds to the number of positive-frequency passband reflection ($|S_{11}|$) zeros.

DM. If zero is specified for N, the model will determine N. But, since N must be an integer, there will typically be some design margin available, and this margin must be assigned to some parameter other than N. Consequently, after determining N, the model will compute a new value for one or more of the other parameters. The value of DM determines which parameter, or parameters, will have their values recomputed after a value is synthesized for N. For each valid value of DM, the table below specifies which parameters (X) are effected by the additional design margin and the manner in which they are effected.

DM	AP	AS	FP1	FP2	FS1	FS2	Description
0	X						decrease AP
1		X					increase AS
2			X	X			Increase FP1 Decrease FP2
3			X		X		Decrease FS1 & FP1
4			X			X	Increase FP1 & FS2
5				X	X		Decrease FS1 & FP2
6				X		X	Increase FP2 & FS2
7					X	X	Decrease FS1 Increase FS2
8			X	X	X	X	Decrease FS1 & FP2 Increase FP1 & FS2

Parameter Restrictions and Recommendations

1. If any of the parameters N, AP, AS, FP1, FP2, FS1, or FS2 are set to 0, the model considers them "unspecified." A value of zero for any of these parameters may, or may not, be valid, depending on whether the set of parameters as a whole corresponds to one of the valid parameter sets listed in the table, above.
2. If N is specified, it must fall within the range $0 < N < 27$.

   If N=0, then $0 \leq DM \leq 8$, otherwise DM is ignored.
3. If Ap and AS are both specified, $0 < AP < AS$, otherwise $0 < AP$ or $0 < AS$.

   Recommend AP greater than or equal to 0.001 dB.
4. "Selectivity" is fully specified by any three of FS1, FP1, FP2, and FS2.

   If FS1, FP1 & FP2 are specified, $0 < FS1 < FP1 < FP2$.

   If FS1, FP1 & FS2 are specified, $0 < FS1 < FP1 < FS2$.

   If FS1, FP2 & FS2 are specified, $0 < FS1 < FP2 < FS2$.

   If FP1, FP2 & FS2 are specified, $0 < FP1 < FP2 < FS2$.
5. "Selectivity" is partially specified by any two of FS1, FP1, FP2, and FS2.

   If FS1 & FP1 are specified, $0 < FS1 < FP1$.

   If FS1 & FP2 are specified, $0 < FS1 < FP2$.

   If FS1 & FS2 are specified, $0 < FS1 < FS2$.

If FP1 & FP2 are specified, 0 < FP1 < FP2.

If FP1 & FS2 are specified, 0 < FP1 < FS2.

If FP2 & FS2 are specified, 0 < FP2 < FS2.

6. 0 < RS.

   0 < RL.

7. QU > 0 specifies a finite unloaded Q (recommend QU ≤ 1e12).

   QU = 0 specifies an infinite unloaded Q.

Implementation Details

The model is implemented as a short-circuit admittance matrix,

$$\begin{bmatrix} y_{11} & y_{12} \\ y_{21} & y_{22} \end{bmatrix}$$

, whose equivalent normalized lowpass prototype transfer function, S21(s), is:

$$|\, S_{21}(s)|^2 = \frac{1}{1+\varepsilon^2 R_n^2\left(\frac{s}{j}\right)} = \frac{f(s)^2}{|\, g(s)|^2} = \frac{f(s)^2}{1+\ |\, h(s)|^2}$$

where Rn is the Elliptic Rational Function, and

$$\varepsilon^2 = 10^{\mathrm{AP}/10} - 1$$

$$R_n(\omega) = C \times \omega^{\Pi} \times \prod_{i=1}^{\frac{N-\Pi}{2}} \left(\frac{\omega^2 - \omega_i^2}{\omega^2 - \frac{\omega_s^2}{\omega_i^2}} \right)$$

$$C = \prod_{i=1}^{\frac{N-\Pi}{2}} \left(\frac{1 - \frac{\omega_s^2}{\omega_i^2}}{1 - \omega_i^2} \right)$$

$$\omega_i = \mathrm{cd}\left(\frac{2i-1}{N} K\left(\frac{1}{\omega_s^2}\right), \frac{1}{\omega_s^2} \right)$$

$$s = 1 / \left(\frac{\sqrt{\mathrm{FP2} \times \mathrm{FP1}}}{\mathrm{QU} \times\ |\ \mathrm{FP2} - \mathrm{FP1}\ |} + j\left(\frac{-1}{\omega}\right) \right)$$

$$\Pi =\ 0 \text{ if N is even}, \Pi =\ 1 \text{ if N is odd}$$

$$j = \sqrt{-1}$$

where cd is a Jacobian elliptic function, K is Legendre's complete elliptic integral of the first kind, and a bandstop-to-lowpass frequency transformation has been applied:

$$\omega = \frac{-\mid \mathrm{FP2} - \mathrm{FP1} \mid}{\_\mathrm{FREQ} - (\mathrm{FP2} \times \mathrm{FP1}) / \_\mathrm{FREQ}}$$

and

$$\omega_s = \frac{-\mid \mathrm{FP2} - \mathrm{FP1} \mid}{\mathrm{FS2} - (\mathrm{FP2} \times \mathrm{FP1}) / \mathrm{FS2}}$$

Note that _FREQ is the variable containing the project frequency and the frequency parameters FP1, FP2, FS1, and FS2 are related according to:

$$FP1 \times FP2 = FS1 \times FS2$$

And the seven parameters of the elliptic-function bandstop filter (N, AP, AS, FP1, FP2, FS1, and FS2) are related by "the degree equation":FP1xFP2=FS1xFS2,

$$N = \left(\frac{K(m)}{K(1-m)}\right)\left(\frac{K(1-m_o)}{K(m_o)}\right)$$

where

$$m_o = \frac{10^{\mathrm{AP}/10} - 1}{10^{\mathrm{AS}/10} - 1}$$

and

$$m = \left| \frac{\mathrm{FS2} - (\mathrm{FP2} \times \mathrm{FP1}) / \mathrm{FS2}}{\mid \mathrm{FP2} - \mathrm{FP1} \mid} \right|^2$$

The admittances are given by:

$$y_{11} = \left(\frac{1}{\mathrm{RS}}\right)\frac{g(s) + g(-s) - h(s) - h(-s)}{g(s) - g(-s) + h(s) - h(-s)}$$

$$y_{22} = \left(\frac{1}{\mathrm{RL}}\right)\frac{g(s) + g(-s) + h(s) + h(-s)}{g(s) - g(-s) + h(s) - h(-s)}$$

$$y_{12} = y_{21} = \left(\frac{1}{\sqrt{\mathrm{RS} \times \mathrm{RL}}}\right)\frac{2 \times f(s)}{g(s) - g(-s) + h(s) - h(-s)}$$

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

Recommendations for Use

Note that this model behaves as if it has ideal impedance transformers at its ports, so there is no attenuation due to mismatched source and load impedances. The model expects that the source impedance will equal RS and that the load impedance will equal RL, but RS need not have any special relationship to RL for ideal transmission (as would normally be the case).

References

[1] Miroslav D. Lutovac, Dejan V. Tosic, and Brian L. Evans, Filter Design For Signal Processing Using MATLAB and Mathematica, (Prentice Hall, 2001), Chapters 6, 12, and 13.

[2] Alexander J. Grossman, "Synthesis of Tchebycheff parameter symmetrical filters," Proceedings of the IRE, pp. 545-473, April 1957.

[3] Dante Youla, "A tutorial exposition of some key network-theoretic ideas underlying classical insertion-loss filter design," Proc. IEEE, vol. 59, no. 5, pp. 760-799, May 1971.

[4] Miroslav Vleck and Rolf Unbehauen, "Degree, ripple, and transition width of elliptic filters," IEEE Trans. Circuits Syst., vol. 36, no. 3, pp. 469-472, March 1989.

[5] H. J. Orchard and Alan N. Willson, Jr., "Elliptic functions for filter design," IEEE Trans. Circuits Syst., I, vol. 44, no. 4, pp. 273-287, April 1997.

[6] Rolf Schaumann, Mohammed S. Ghausi, and Kenneth R. Laker, Design of Analog Filters: Passive, Active RC, and Switched Capacitor, (Prentice-Hall, 1990), pp. 49-51.

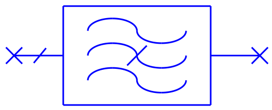
[7] Kendall L. Su, Handbook of Tables for Elliptic-Function Filters, (Kluwer Academic, 1990).

[8] Handbook of Mathematical Functions With Formulas, Graphs, and Mathematical Tables, edited by Milton Abramowitz and Irene A. Stegun, (U. S. National Bureau of Standards, 1964), Chapters 16 and 17 by L. M. Milne-Thomson.

[9] George L. Matthaei, Leo Young, and E. M. T. Jones, Microwave Filters, Impedance-Matching Networks, and Coupling Structures, (Artech House, 1980), pp. 149-161.

Gaussian Bandstop Filter (Closed Form): BSFG

Symbol



Summary

BSFG models represent lumped-element Gaussian bandstop filters. They approximate the ideal Gaussian magnitude response and offer simplicity, relatively flat group delay, and good time domain performance, but suffer from poor frequency selectivity. Although similar to Bessel-Thomson filters, Gaussian filters offer faster rise times and lower transient overshoots, but have slightly less stopband attenuation and less group delay flatness.

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	BSFG1
N	Number of resonators in the filter		3
FP1	Passband lower band-edge frequency (when Qu is infinite).	Frequency	0.5 GHz
FP2	Passband upper band-edge frequency (when Qu is infinite).	Frequency	1.5 GHz
AP	Maximum passband attenuation (when Qu is infinite).	DB	3.0103 dB
*RS	Expected source resistance.	Resistance	50 ohm
*RL	Expected Load resistance	Resistance	50 ohm
*QU	Average unloaded Q of the lumped lowpass reactances.		1e12

* indicates a secondary Parameter

Parameter Restrictions and Recommendations

1. 0 < N < 51
2. 0 < FP1
3. 0 < FP2
4. 0 < AP Recommend AP greater than or equal to 0.001 dB.
5. 0 < RS
6. 0 < RL
7. 0 < QU. Recommend QU less than or equal to 1e12.

Implementation Details

The model is implemented as a short-circuit admittance matrix,

$$\begin{bmatrix} y_{11} & y_{12} \\ y_{21} & y_{22} \end{bmatrix}$$

, whose equivalent transfer function squared magnitude is that of an ideal Gaussian filter. The ideal Gaussian squared magnitude characteristic is:

$$|S_{21}|^2 = \frac{1}{\exp(-s^2)}$$

In the model, the denominator of this ideal Gaussian characteristic is approximated by a truncated Maclaurin series:

$$\exp(-s^2) \approx \sum_{i=0}^{N} \frac{(-1)^i s^{2i}}{i!} = |g(s)|^2 = 1 + |h(s)|^2$$

where

$$|S_{21}(s)|^2 = \frac{b_0^2}{|g(s)|^2} = \frac{b_0^2}{1 + |h(s)|^2}$$

$$j = \sqrt{-1}$$

and a lowpass-to-bandpass (not lowpass-to-bandstop) frequency transformation has been applied:

$$\omega = \frac{\_FREQ - (FP2 \times FP1)/\_FREQ}{|FP2 - FP1|}$$

_FREQ is the variable containing the project frequency, and the admittances are:

$$y_{11} = \left(\frac{1}{RS}\right)\frac{g(s) + g(-s) - h(s) - h(-s)}{g(s) - g(-s) + h(s) - h(-s)}$$

$$y_{22} = \left(\frac{1}{RL}\right)\frac{g(s) + g(-s) + h(s) + h(-s)}{g(s) - g(-s) + h(s) - h(-s)}$$

$$y_{12} = y_{21} = \left(\frac{1}{\sqrt{RS \times RL}}\right)\frac{(-2)}{g(s) - g(-s) + h(s) - h(-s)}$$

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

Recommendations for Use

This model behaves as if it has ideal impedance transformers at its ports, so there is no attenuation due to mismatched source and load impedances. The model expects that the source impedance will equal RS and that the load impedance will equal RL, but RS need not equal RL for ideal transmission (as would normally be the case).

References

[1] Milton Dishal, "Gaussian-Response Filter Design," Electrical Communication, vol. 36, March 1959, pp. 3-26.

[2] Anatol I. Zverev, Handbook of Filter Synthesis, (John Wiley & Sons, 1967), pp. 67, 70, 71, 73, 74, 90, 91.

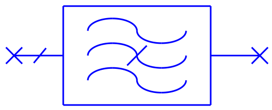
[3] DeVerl. S. Humpherys, The Analysis, Design, and Synthesis of Electrical Filters, (Prentice-Hall, 1970), pp. 413-417.

[4] Herman J. Blinchikoff and Anatol I. Zverev, Filtering in the Time and Frequency Domains, (Robert E. Krieger Publishing Co., 1987), pp. 130-132.

[5] George L. Matthaei, Leo Young, and E. M. T. Jones, Microwave Filters, Impedance-Matching Networks, and Coupling Structures, (Artech House, 1980), pp. 149-161.

Raised-Cosine Bandstop Filter (Closed Form): BSFRC

Symbol



Summary

BSFRC models represent Raised-Cosine Bandstop filters. Raised-Cosine filters are applied in digital transmission system theory to represent ideal (intersymbol interference free) Nyquist filtering of impulse and pulse data transmissions. Hardware designers attempt to design channel filters which approximate these characteristics and have the smallest possible bandwidth.

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	BSFRC1
FP1	Lower frequency edge of passband	Frequency	0.5 GHz
FP2	Upper frequency edge of passband	Frequency	1.5 GHz
A	Roll-off factor (between o and 1)		0.5
TYP	Transmission type	Vector Text(pull-down)	Impulse
E	Exponent of the raised-cosine response		1.
*RS	Expected source resistance	Resistance	50 ohm
*RL	Expected load resistance	Resistance	50 ohm

* indicates a secondary Parameter

Parameter Restrictions and Recommendations

1. $0 < FP1$
2. $0 < FP2$
3. $0 \leq A \leq 1$.

   A minimum bandwidth (A=0) filter would require an infinite number of filter sections. Approximations of a 30% excess Nyquist bandwidth (A=0.3) raised-cosine filter are considered feasible with present-day technology.
4. TYP is either $\{0,1\}$.

   TYP = 0 (Impulse) specifies the ideal raised-cosine response, the theoretical filter model for infinitesimally narrow impulses. TYP = 1 (Pulse) specifies a sinc-normalized raised-cosine response, the filter model for rectangular pulse transmission.
5. $0 < E \leq 1$.

The ideal raised-cosine response is raised to the exponent, E. For cascaded identical transmit and receive filters, you would typically specify E=0.5 for each filter. The composite channel response of the cascaded filters would be represented by specifying E=1.

6. 0 < RS.

   By definition, the model matches RS at its input port.

7. 0 < RL.

   By definition, the model matches RL at its output port.

Implementation Details

This model is implemented as a short-circuit admittance matrix,

$$\begin{bmatrix} y_{11} & y_{12} \\ y_{21} & y_{22} \end{bmatrix}$$

, whose equivalent amplitude functions S21 and S12 implement the raised-cosine response for impulse or pulse data transmission, while S11 and S22 are, by definition, constant and matched to the filter port termination resistances (i.e., equal to 1). For impulses,

$$S_{21}(j\omega) = 1 \text{ for } [0 \le \omega \le (1 - A)]$$

$$S_{21}(j\omega) = \left(\cos^2\left(\left(\frac{\pi}{4}\right)\left(\frac{\omega + A - 1}{A}\right)\right)\right)^E \text{ for } [(1 - A) \le \omega \le (1 + A)]$$

$$S_{21}(j\omega) = 0 \text{ for } [(1 + A) \le \omega]$$

For pulses,

$$S_{21}(j\omega) = 1 \text{ for } [0 \le \omega \le (1 - A)]$$

$$S_{21}(j\omega) = \frac{\left(\cos^2\left(\left(\frac{\pi}{4}\right)\left(\frac{\omega + A - 1}{A}\right)\right)\right)^E}{\operatorname{sinc}\left(\frac{\pi}{2}\omega\right)} \text{ for } [(1 - A) \le \omega \le (1 + A)]$$

$$S_{21}(j\omega) = 0 \text{ for } [(1 + A) \le \omega]$$

where

$$\operatorname{sinc}(x) = \frac{\sin(x)}{x}$$

$$j = \sqrt{-1}$$

and a lowpass-to-bandstop frequency transformation has been applied:

$$\omega = \frac{-\mid \text{FP2} - \text{FP1} \mid}{\text{\_FREQ} - (\text{FP2} \times \text{FP1}) / \text{\_FREQ}}$$

_FREQ is the variable containing the project frequency, and the admittances are:

$$y_{11} = \left(\frac{1}{\mathrm{RS}}\right)\left(\frac{-S_{21}^2(j\omega)}{S_{21}^2(j\omega) - 4}\right)$$

$$y_{22} = \left(\frac{1}{\mathrm{RL}}\right)\left(\frac{-S_{21}^2(j\omega)}{S_{21}^2(j\omega) - 4}\right)$$

$$y_{22} = \left(\frac{1}{\sqrt{\mathrm{RS} \times \mathrm{RL}}}\right)\left(\frac{2S_{21}(j\omega)}{S_{21}^2(j\omega) - 4}\right)$$

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Recommendations for Use

The model expects that the source impedance will equal RS and that the load impedance will equal RL, but RS need not equal RL for ideal transmission.

References

[1] Kamilo Feher, Digital Communications: Microwave Applications, (Prentice-Hall, 1981), pp. 46-51.

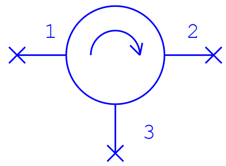
[2] John G. Proakis, Digital Communications, 2nd Ed., (McGraw-Hill, 1989), pp. 532-536.

[3] John Bellamy, Digital Telephony, 2nd Ed., (John Wiley & Sons, 1991), pp. 523-528.

[4] E. A. Lee and D. G. Messerschmitt, Digital Communications, 2nd Ed., (Kluwer Academic Publishers, 1994), pp. 188-191, 226-228.

Circulator (Closed Form): CIRC

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Name	Text	U1
R	Port resistance	Resistance	50 ohm
LOSS	Loss	DB	0 dB
ISOL	Isolation	DB	30 dB

Implementation Details

Implements an ideal 3-port circulator in which all ports have the impedance R. Forward insertion loss (with clockwise circulation) is given by **LOSS** and reverse isolation is given by **ISOL**. Both LOSS and ISOL can be entered as positive or negative quantities, but they are always interpreted as losses.

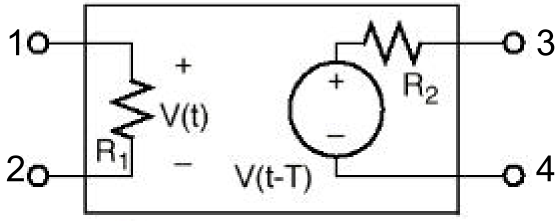
Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Linear Delay Element (Closed Form): DELAY

Symbol

Topology



Parameters

Name	Description	Unit Type	Default
ID	Name	Text	U1
DELAY	Delay	Time	0 ns
R1	Input resistance	Resistance	0 ohm
R2	Output resistance	Resistance	1 ohm

Implementation Details

For a voltage applied between ports 1 and 2, the output voltage between ports 3 and 4 will be delayed by the amount specified in DELAY.

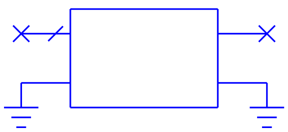
R1 is the resistance between ports 1 and 2, and R2 is the resistance between ports 3 and 4. If parameter R1 = 0, then R1 =∞.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Linear Delay Element 2 (Closed Form): DELAY2

Symbol



Summary

DELAY2 implements an ideal linear delay.

Parameters

Name	Description	Unit Type	Default
ID	Name		U1
Delay	Delay		0 ns
CONJ	S12=(S21)* for nonzero		0
M1	Input refl coef mag		0
M2	Output refl coef mag		0

Parameter Restrictions and Recommendations

1. Delay ≥ 0
2. 0 ≥ M1 ≥ 1
3. 0 ≥ M2 ≥ 1

Implementation Details

$$S_{11} = M_1 \angle 0^o$$

$$S_{22} = M_2 \angle 0^o$$

$$S_{21} = 1 \angle \theta^o \text{ Where } \theta = -2 * \pi * \_\text{FREQ} * \text{Delay}$$

if CONJ = 0

S12 = S21

else

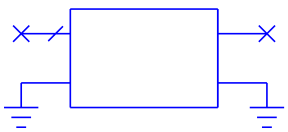
S12 = S 21*

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Digital Attenuator (Closed Form): DGATTEN

Symbol



Summary

DGATTEN models an ideal digital attenuator.

Parameters

Name	Description	Unit Type	Default
ID	Name	Text	U
STEP	Loss per state in dB	dB	0 dB
STATE	State (0 is no loss)		0
Zo	Reference impedance	Resistance	50 ohms

Parameter Details

STEP. The loss per state is specified in positive dB units. For example, STEP = 3 means 3 dB of loss per state.

Parameter Restrictions and Recommendations

1. STEP must be greater than or equal to 0 (0 indicates no loss).
2. STATE must be greater than or equal to 0 (0 indicates no loss).
3. Zo must be greater than or equal to 0.1 ohm.

Implementation Details

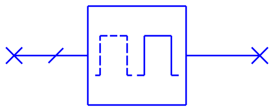
The total loss is calculated as Loss(db)=STEPxSTATE. To specify no loss, set STEP or STATE equal to 0. In all cases, $S_{11} = S_{22} = 0.0$.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Ideal Digital Time Delay, Reciprocal (Closed Form): DGDELAY

Symbol



Summary

DGDELAY models an ideal, linear, frequency-dependent, digital time delay element.

Parameters

Name	Description	Unit Type	Default
ID	Name	Text	U1
STEP	Delay step	Time	0 ns
STATE	Current state		0
Zo	Reference impedance	Resistance	50 ohms

Parameter Restrictions and Recommendations

1. STEP must be greater than or equal to 0 (0 indicates no delay).
2. STATE must be greater than or equal to 0 (0 indicates no delay).
3. Zo must be greater than or equal to 0.1 ohm.

Implementation Details

Implements an ideal, linear, digital delay according to the following S-parameter definitions normalized to Zo:

$$S_{11} = S_{22} = 0$$

$$S_{21} = S_{12} = e^{j*2*\pi*f*\mathrm{STEP}*\mathrm{STATE}}$$

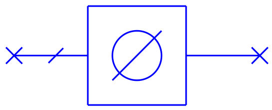
where f is the simulation frequency.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

Digital Phase Shift Element, Non-Reciprocal (Closed Form): DGPHASE

Symbol



Summary

DGPHASE models an ideal, frequency-independent, reciprocal, digital phase shift element.

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	P1
NUMSTATES	Total number of states		1
STATE	Current state		0
Zo	Reference impedance	Resistance	50 ohms

Parameter Restrictions and Recommendations

1. NUMSTATES must be greater than or equal to 1.
2. 0 ≤STATE ≤ NUMSTATES.
3. Zo must be greater than or equal to 0.1 ohm.

Implementation Details

Applies a phase shift according to the equations below, normalized to Zo.

$$S_{11} = S_{22} = 0$$

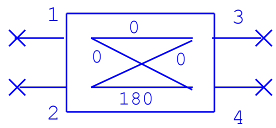
$$S_{21} = S_{12} = e^{j*2*\pi*\frac{\text{STATE}}{\text{NUMSTATES}}}$$

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

180-Degree Hybrid (Closed Form): DHYB

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Name	Text	U1
R	Port resistance	Resistance	50 ohm
Loss	Excess loss	DB	0 dB

Implementation Details

Implements the following 4-port S parameter matrix normalized to the resistance R.

$$S = \begin{bmatrix} 0 & 0 & S_t & S_t \\ 0 & 0 & S_t & -S_t \\ S_t & S_t & 0 & 0 \\ S_t & -S_t & 0 & 0 \end{bmatrix}$$

where

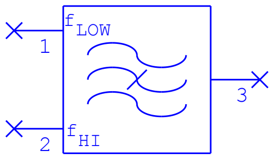
$$S_t = \frac{1}{\sqrt{2}} 10^{\frac{-|\text{Loss}|}{20}}$$

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

Ideal Frequency Diplexer (Closed Form): DIPLEXF

Symbol



Summary

The DIPLEXF models and ideal, passive frequency diplexer. Two frequency ranges (low and high) are specified.

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	DP
F1L	Lower frequency cut-off for low band	Frequency	0 GHz
F1H	Upper frequency cut-off for low band	Frequency	10 GHz
F2L	Lower frequency cut-off for high band	Frequency	10 GHz
F2H	Upper frequency cut-off for high band	Frequency	100 GHz

Parameter Restrictions and Recommendations

1. $F1L \le F1H < F2L \le F2H$
2. $F1L, F1H, F2L, F2H \le 0$

Implementation Details

Implements an ideal frequency diplexer with the following 3-port S-parameter matrix:

$$S = \begin{bmatrix} 0 & 0 & 1 \\ 0 & 1 & 0 \\ 1 & 0 & 0 \end{bmatrix} \quad \text{for} \quad F1L \le f \le F1H$$

$$S = \begin{bmatrix} 1 & 0 & 0 \\ 0 & 0 & 1 \\ 0 & 1 & 0 \end{bmatrix} \quad \text{for} \quad F2L \le f \le F2H$$

$$S = \begin{bmatrix} 1 & 0 & 0 \\ 0 & 1 & 0 \\ 0 & 0 & 1 \end{bmatrix} \quad \text{for} \quad \text{Else}$$

Also:

If F1L = F1H, then S31 = S13 = 0 and S11 = S33 = 1.

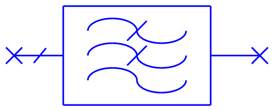
If F2L = F2H, then S32 = S23 = 0 and S22 = S33 = 1.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

Butterworth Distributed Lowpass Filter: DLPFB

Symbol



Summary

DLPFB models represent distributed-element Butterworth lowpass filters. They offer simplicity and a compromise between high selectivity and flat group delay. They exhibit an initial lowpass characteristic, followed by alternating stopbands and passbands at higher frequencies. The filter's attenuation characteristic is symmetrical about the commensurate frequency, FC. Insertion loss is maximally flat at zero frequency and at the center of each of the repeating bandpass characteristics.

Parameters

Name	Description	Unit Type	Default
ID	Filter ID	Text	DLPFB1
N	Number of elements in the filter		3
FP	Passband corner frequency (when QU is infinite)	Frequency	1 GHz
FC	Commensurate frequency	Frequency	2 GHz
*AP	Passband corner attenuation (when QU is infinite)	DB	3.0103 dB
*RS	Expected source resistance	Resistance	50 ohm
*RL	Expected load resistance	Resistance	50 ohm
*QU	Uniform unloaded Q of elements		1e12

** indicates a secondary parameter*

Parameter Restrictions and Recommendations

1. 0 < N < 29
2. 0 < FP < FC
3. 0 < AP Recommend AP greater than or equal to 0.001 dB.
4. 0 < RS
5. 0 < RL
6. 0 < QU. Recommend QU less than or equal to 1e12.

Implementation Details

The model is implemented as a short-circuit admittance matrix, whose equivalent transfer function squared magnitude is that of a Butterworth filter:

$$|S_{21}(s)|^2 = \frac{1}{1+(-1)^N \varepsilon^2 S^{2N}} = \frac{1}{|g(s)|^2} = \frac{1}{1+|h(s)|^2}$$

where

$$\varepsilon^2 = 10^{AP/10} - 1$$

$$s = \frac{1}{QU} + j\omega$$

$$j = \sqrt{-1}$$

and Richard's transformation has been applied to the frequency variable:

$$\omega = \tan\left(\left(\frac{\pi}{2}\right)\frac{\_FREQ}{FC}\right) / \left(\tan\left(\left(\frac{\pi}{2}\right)\frac{FP}{FC}\right)\right)$$

_FREQ is the variable containing the project frequency, and the admittances are:

$$y_{11} = \left(\frac{1}{RS}\right)\frac{g(s) + g(-s) - h(s) - h(-s)}{g(s) - g(-s) + h(s) - h(-s)}$$

$$y_{22} = \left(\frac{1}{RL}\right)\frac{g(s) + g(-s) + h(s) + h(-s)}{g(s) - g(-s) + h(s) - h(-s)}$$

$$y_{12} = y_{21} = \left(\frac{1}{\sqrt{RS \times RL}}\right)\frac{2}{g(s) - g(-s) + h(s) - h(-s)}$$

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Recommendations for Use

Note that this model behaves as if it has ideal impedance transformers at its ports, so there is no attenuation due to mismatched source and load impedances. The model expects that the source impedance will equal RS and that the load impedance will equal RL, but RS need not equal RL for ideal transmission (as would normally be the case).

References

[1] H. J. Horton and R. J. Wenzel, "Optimum quarter-wave TEM filters," IRE Trans. on MTT, Vol. MTT-13, May 1965, pp. 316-327.

[2] P. I. Richards, "Resistor-transmission-line circuits," Proc. IRE, vol. 36, February 1948, pp. 217-220.

[3] Joseph Helszajn, Synthesis of Lumped Element, Distributed and Planar Filters, (McGraw-Hill, 1990), pp. 160-163, 284-286.

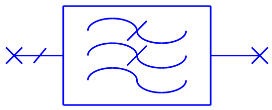
[4] Rolf Schaumann, Mohammed S. Ghausi, and Kenneth R. Laker, Design of Analog Filters: Passive, Active RC, and Switched Capacitor, (Prentice-Hall, 1990), pp. 40-44.

[5] Louis Weinberg, Network Analysis and Synthesis, (Robert E. Krieger Publishing, 1975), pp.493-498, 585-597.

[6] Adel S. Sedra and Peter O. Brackett, Filter Theory and Design: Active and Passive, (Matrix Publishers, 1978), pp.105-111.

Chebyshev Distributed Lowpass Filter: DLPFC

Symbol



Summary

DLPFC models represent distributed-element Chebyshev lowpass filters. They offer simplicity and good selectivity. They exhibit an initial lowpass characteristic, followed by alternating stopbands and passbands at higher frequencies. The insertion loss ripples between zero and a specified maximum in the passband. The stopband attenuation increases rapidly beyond the passband edge towards FC. The attenuation characteristic is symmetrical about the commensurate frequency, FC.

Parameters

Name	Description	Unit Type	Default
ID	Filter ID	Text	DLPFC1
N	Number of elements in the filter		3
FP	Passband corner frequency (when QU is infinite)	Frequency	1 GHz
FC	Commensurate frequency (	Frequency	2 GHz
*AP	Maximum passband insertion loss (when QU is infinite)	DB	0.1 dB
*RS	Expected source resistance	Resistance	50 ohm
*RL	Expected load resistance	Resistance	50 ohm
*QU	Uniform unloaded Q of elements		1e12

* indicates a secondary Parameter

Parameter Restrictions and Recommendations

1. 0 < N < 27
2. 0 < FP < FC
3. 0 < AP Recommend AP greater than or equal to 0.001 dB.
4. 0 < RS
5. 0 < RL
6. 0 < QU. Recommend QU less than or equal to 1e12.

Implementation Details

The model is implemented as a short-circuit admittance matrix, whose equivalent transfer function squared magnitude is that of a Chebyshev filter:

$$|S_{21}(s)|^2 = \frac{1}{1+\varepsilon^2 C_n^2\left(\frac{s}{j}\right)} = \frac{1}{|g(s)|^2} = \frac{1}{1+|h(s)|^2}$$

where Cn is the Chebyshev polynomial of the first kind, and

$$\varepsilon^2 = 10^{AP/10} - 1$$

$$C_n(\omega) = \cos(N * \mathrm{acos}(\omega)) \qquad \text{for } 0 \le |\omega| \le 1$$

$$C_n(\omega) = \cosh(N * \mathrm{acosh}(\omega)) \qquad \text{for } 1 < |\omega|$$

$$s = \frac{1}{\mathrm{QU}} + j\omega$$

$$j = \sqrt{-1}$$

and Richard's transformation has been applied to the frequency variable:

$$\omega = \tan\left(\left(\frac{\pi}{2}\right)\frac{\_\mathrm{FREQ}}{\mathrm{FC}}\right) / \left(\tan\left(\left(\frac{\pi}{2}\right)\frac{\mathrm{FP}}{\mathrm{FC}}\right)\right)$$

_FREQ is the variable containing the project frequency, and the admittances are:

$$y_{11} = \left(\frac{1}{\mathrm{RS}}\right)\frac{g(s) + g(-s) - h(s) - h(-s)}{g(s) - g(-s) + h(s) - h(-s)}$$

$$y_{22} = \left(\frac{1}{\mathrm{RL}}\right)\frac{g(s) + g(-s) + h(s) + h(-s)}{g(s) - g(-s) + h(s) - h(-s)}$$

$$y_{12} = y_{21} = \left(\frac{1}{\sqrt{\mathrm{RS} \times \mathrm{RL}}}\right)\frac{2}{g(s) - g(-s) + h(s) - h(-s)}$$

The model only takes distributed series inductances and distributed shunt capacitances into account - it does not account for unit elements.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

Recommendations for Use

Note that this model behaves as if it has ideal impedance transformers at its ports, so there is no attenuation due to mismatched source and load impedances. The model expects that the source impedance will equal RS and that the load impedance will equal RL, but RS need not equal RL for ideal transmission (as would normally be the case for odd-order Chebyshev filters).

References

[1] H. J. Horton and R. J. Wenzel, "Optimum quarter-wave TEM filters," IRE Trans. on MTT, Vol. MTT-13, pp. 316-327, May 1965.

[2] P. I. Richards, "Resistor-transmission-line circuits," Proc. IRE, vol. 36, February 1948, pp. 217-220.

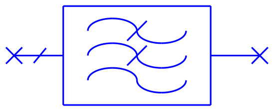
[3] Joseph Helszajn, Synthesis of Lumped Element, Distributed and Planar Filters, (McGraw-Hill, 1990), pp. 160-163, 284-288.

[4] Rolf Schaumann, Mohammed S. Ghausi, and Kenneth R. Laker, Design of Analog Filters: Passive, Active RC, and Switched Capacitor, (Prentice-Hall, 1990), pp. 44-48.

[5] Louis Weinberg, Network Analysis and Synthesis, (Robert E. Krieger Publishing, 1975), pp. 507-529.

Bessel Distributed Lowpass Filter: DLPFD

Symbol



Summary

DLPFD models represent distributed-element Bessel-Thomson lowpass filters. They offer simplicity and maximally flat group delay, but suffer from poor selectivity. They exhibit an initial lowpass characteristic, followed by alternating stopbands and passbands at higher frequencies. The filter's attenuation characteristic is symmetrical about the commensurate frequency, FC. The group delay is maximally flat at zero frequency and at the center of each of the repeating bandpass characteristics.

Parameters

Name	Description	Unit Type	Default
ID	Filter ID	Text	DLPFD1
N	Number of ements.in the filter		3
FP	Passband corner frequency (when QU is infinite)	Frequency	1 GHz
FC	Commensurate frequency	Frequency	2 GHz
*AP	Passband corner attenuation (when QU is infinite)	DB	3.0103 dB
*RS	Expected Source Resistance	Resistance	50 ohm
*RL	Expected Load Resistance	Resistance	50 ohm
*QU	Uniform unloaded Q of elements		1e12

* indicates a secondary Parameter

Parameter Restrictions and Recommendations

1. 0 < N < 27
2. 0 < FP < FC
3. 0 < AP Recommend AP greater than or equal to 0.001 dB.
4. 0 < RS
5. 0 < RL
6. 0 < QU. Recommend QU less than or equal to 1e12.

Implementation Details

The model is implemented as a short-circuit admittance matrix,

$$\begin{bmatrix} y_{11} & y_{12} \\ y_{21} & y_{22} \end{bmatrix}$$

, whose equivalent transfer function squared magnitude is that of a Bessel-Thomson filter:

$$|S_{21}(s)|^2 = \frac{b_0^2}{|g(s)|^2} = \frac{b_0^2}{1 + |h(s)|^2}$$

where

$$g(s) = \sum_{i=0}^{N} b_i s^i$$

$$b_i = \frac{(2N-i)!}{2^{N-i}(i)!(N-i)!} \text{ for } i = 0, \ldots, N$$

Where b_0 = (2N-1)!!,i.e, the product of all odd integers less than 2N.

$$s = \frac{1}{\text{QU}} + j\omega$$

$$j = \sqrt{-1}$$

and Richard's transformation has been applied to the frequency variable:

$$\omega = \tan\left(\left(\frac{\pi}{2}\right)\frac{\_\text{FREQ}}{\text{FC}}\right) / \left(\tan\left(\left(\frac{\pi}{2}\right)\frac{\text{FP}}{\text{FC}}\right)\right)$$

_FREQ is the variable containing the project frequency, and the admittances are:

$$y_{11} = \left(\frac{1}{\text{RS}}\right)\frac{g(s) + g(-s) - h(s) - h(-s)}{g(s) - g(-s) + h(s) - h(-s)}$$

$$y_{22} = \left(\frac{1}{\text{RL}}\right)\frac{g(s) + g(-s) + h(s) + h(-s)}{g(s) - g(-s) + h(s) - h(-s)}$$

$$y_{12} = y_{21} = \left(\frac{1}{\sqrt{\text{RS} \times \text{RL}}}\right)\frac{2}{g(s) - g(-s) + h(s) - h(-s)}$$

The model only takes distributed series inductances and distributed shunt capacitances into account - it does not account for unit elements.

Recommendations for Use

Note that this model behaves as if it has ideal impedance transformers at its ports, so there is no attenuation due to mismatched source and load impedances. The model expects that the source impedance will equal RS and that the load impedance will equal RL, but RS need not equal RL for ideal transmission (as would normally be the case).

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

References

[1] H. J. Horton and R. J. Wenzel, "Optimum quarter-wave TEM filters," IRE Trans. on MTT, Vol. MTT-13, May 1965, pp. 316-327.

[2] P. I. Richards, "Resistor-transmission-line circuits," Proc. IRE, vol. 36, February 1948, pp. 217-220.

[3] Joseph Helszajn, Synthesis of Lumped Element, Distributed and Planar Filters, (McGraw-Hill, 1990), pp. 160-163, 284-288.

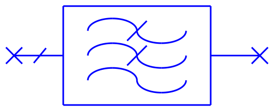
[4] Rolf Schaumann, Mohammed S. Ghausi, and Kenneth R. Laker, Design of Analog Filters: Passive, Active RC, and Switched Capacitor, (Prentice-Hall, 1990), pp. 51-56.

[5] Louis Weinberg, Network Analysis and Synthesis, (Robert E. Krieger Publishing, 1975), pp. 499-506.

[6] Herman J. Blinchikoff and Anatol I. Zverev, Filtering in the Time and Frequency Domains, (Robert E. Krieger Publishing Co., 1987), pp. 124-127.

Gaussian Distributed Lowpass Filter (Closed Form): DLPFG

Symbol



Summary

DLPFG models represent distributed-element Gaussian lowpass filters. They approximate the ideal Gaussian magnitude response and offer simplicity, relatively flat group delay, and good time domain performance, but suffer from poor frequency selectivity. Although similar to Bessel-Thomson filters, Gaussian filters offer faster rise times and lower transient overshoots, but have slightly less stopband attenuation and less group delay flatness. The DLPFG models exhibit an initial lowpass characteristic, followed by alternating stopbands and passbands. The filter's attenuation is symmetrical about the commensurate frequency, FC.

Parameters

Name	Description	Unit Type	Default
ID	Filter ID	Text	DLPFG1
N	Number of elements in the filter.		3
FP	Passband corner frequency (when QU is infinite)	Frequency	1 GHz
FC	Commensurate frequency	Frequency	2 GHz
*AP	Passband corner attenuation (when QU is infinite)	DB	3.0103 dB
*RS	Expected Source Resistance	Resistance	50 ohm
*RL	Expected Load Resistance	Resistance	50 ohm
*QU	Uniform unloaded Q of elements		1e12

* indicates a secondary Parameter

Parameter Restrictions and Recommendations

1. 0 < N < 51
2. 0 < FP < FC
3. 0 < AP Recommend AP greater than or equal to 0.001 dB.
4. 0 < RS
5. 0 < RL
6. 0 < QU. Recommend QU less than or equal to 1e12.

Implementation Details

The model is implemented as a short-circuit admittance matrix,

$$\begin{bmatrix} y_{11} & y_{12} \\ y_{21} & y_{22} \end{bmatrix}$$

, whose equivalent transfer function squared magnitude is that of an ideal Gaussian filter. The ideal Gaussian squared magnitude characteristic is:

$$|S_{21}|^2 = \frac{1}{\exp(-s^2)}$$

In the model, the denominator of this ideal Gaussian characteristic is approximated by a truncated Maclaurin series:

$$\exp(-s^2) \approx \sum_{i=0}^{N} \frac{(-1)^i s^{2i}}{i!} = |g(s)|^2 = 1 + |h(s)|^2$$

where

$$s = \frac{1}{\mathrm{QU}} + j\omega$$

$$j = \sqrt{-1}$$

and Richard's transformation has been applied to the frequency variable:

$$\omega = \tan\left(\left(\frac{\pi}{2}\right)\frac{\mathrm{\_FREQ}}{\mathrm{FC}}\right) / \left(\tan\left(\left(\frac{\pi}{2}\right)\frac{\mathrm{FP}}{\mathrm{FC}}\right)\right)$$

_FREQ is the variable containing the project frequency, and the admittances are:

$$y_{11} = \left(\frac{1}{\mathrm{RS}}\right)\frac{g(s) + g(-s) - h(s) - h(-s)}{g(s) - g(-s) + h(s) - h(-s)}$$

$$y_{22} = \left(\frac{1}{\mathrm{RL}}\right)\frac{g(s) + g(-s) + h(s) + h(-s)}{g(s) - g(-s) + h(s) - h(-s)}$$

$$y_{12} = y_{21} = \left(\frac{1}{\sqrt{\mathrm{RS} \times \mathrm{RL}}}\right)\frac{(-2)}{g(s) - g(-s) + h(s) - h(-s)}$$

The model only takes distributed series inductances and distributed shunt capacitances into account - it does not account for unit elements.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

Recommendations for Use

This model behaves as if it has ideal impedance transformers at its ports, so there is no attenuation due to mismatched source and load impedances. The model expects that the source impedance will equal RS and that the load impedance will equal RL, but RS need not equal RL for ideal transmission (as would normally be the case).

References

[1] H. J. Horton and R. J. Wenzel, "Optimum quarter-wave TEM filters," IRE Trans. on MTT, Vol. MTT-13, May 1965, pp. 316-327.

[2] P. I. Richards, "Resistor-transmission-line circuits," Proc. IRE, vol. 36, February 1948, pp. 217-220.

[3] Joseph Helszajn, Synthesis of Lumped Element, Distributed and Planar Filters, (McGraw-Hill, 1990), pp. 160-163, 284-288.

[4] Milton Dishal, "Gaussian-Response Filter Design," Electrical Communication, vol. 36, March 1959, pp. 3-26.

[5] Anatol I. Zverev, Handbook of Filter Synthesis, (John Wiley & Sons, 1967), pp. 67, 70, 71, 73, 74, 90, 91.

[6] DeVerl. S. Humpherys, The Analysis, Design, and Synthesis of Electrical Filters, (Prentice-Hall, 1970), pp. 413-417.

[7] Herman J. Blinchikoff and Anatol I. Zverev, Filtering in the Time and Frequency Domains, (Robert E. Krieger Publishing Co., 1987), pp. 130-132.

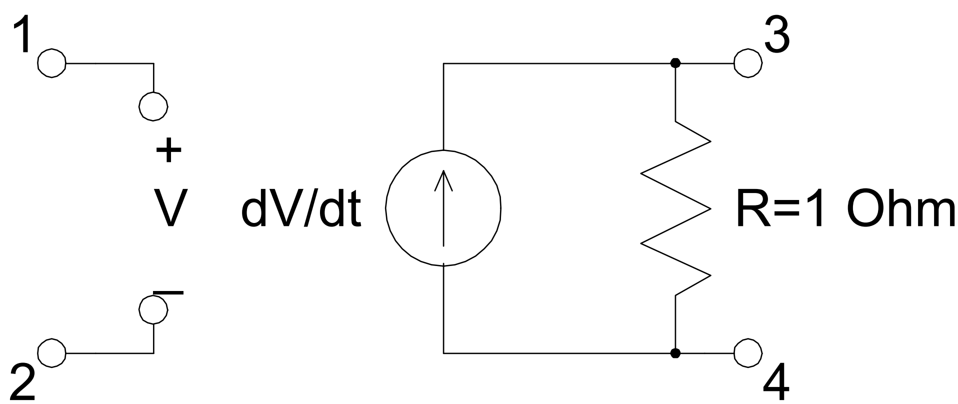
(Obsolete) Differentiator (Closed Form): DVDT

Symbol

Summary

This element is OBSOLETE and is replaced by the Differentiator with Gain and Output Resistance (Closed Form) (DVDTB) element.

Equivalent Circuit



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	DV1

Implementation Details

This element is an ideal differentiator. The short circuit output current is the time derivative of the input voltage. The output resistance is equal to 1 ohm.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

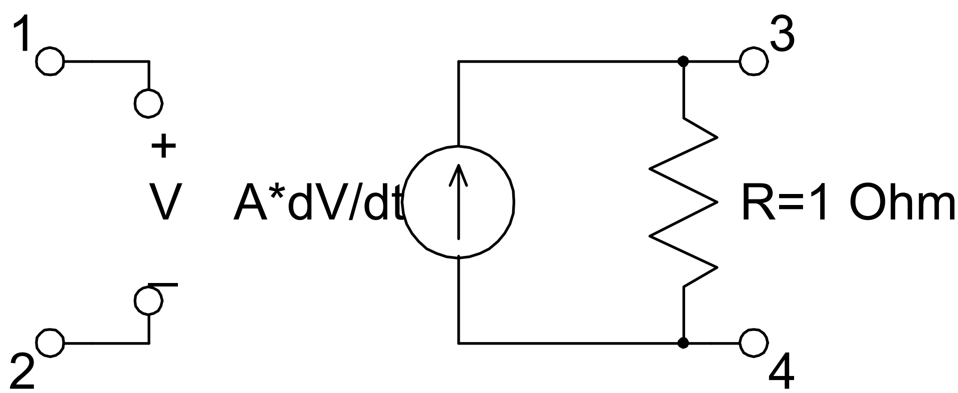
(Obsolete) Differentiator With Gain (Closed Form): DVDTA

Symbol

Summary

This element is OBSOLETE and is replaced by the Differentiator with Gain and Output Resistance (Closed Form) (DVDTB) element.

Equivalent Circuit



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	DV1
A	Gain		1

Implementation Details

This is an ideal differentiator. The short circuit output current is the time derivative of the input voltage.

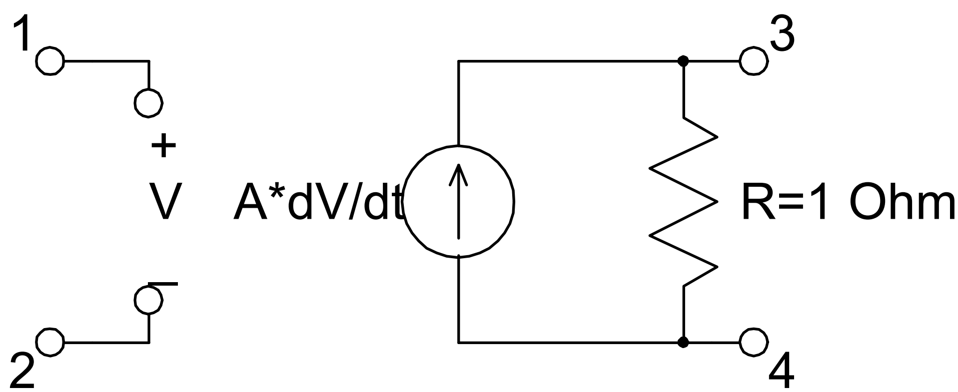
Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Differentiator with Gain and Output Resistance (Closed Form): DVDTB

Symbol

Equivalent Circuit



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	DV1
A	Gain		1
RO	Output resistance (0 = infinite, empty = 1 ohm)	Resistance	

Implementation Details

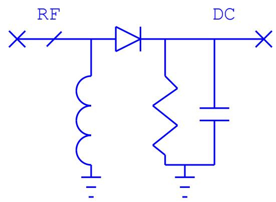
This is an ideal differentiator. The short circuit output current is the time derivative of the input voltage.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Ideal Envelope Detector: ENVDET

Symbol



Summary

ENVDET models an ideal envelope detector. The detector can be configured as a linear detector, in which the output voltage is proportional to the magnitude of the RF input voltage, or as a square-law detector, in which the output voltage is proportional to RF input power. You can also select the post-detection time constant.

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	E1
*MODE	Operating mode (Linear or Square Law)	Vector Text (drop-down)	Linear
*POLARITY	Polarity (Negative or Positive)	Vector Text (drop-down)	Positive
*SQSNS	Square law sensitivity V/W (MODE=Linear only)		1000.0
*Z0	Input impedance	Resistance	50.0 ohm
*TC	Envelope detector video time constant	Time	1e-3 Sec
*ROUT	Video output resistance	Resistance	1000.0 ohm
*SCALE	Output level scale factor		1.0

** indicates a secondary parameter*

Parameter Restrictions and Recommendations

Accuracy is best between -60 dBm and 30 dBm for Z_0=50Ω. Above that level, the square-law response (MODE=Linear) may degrade slightly, and convergence may be more difficult. At low levels, it may be necessary to decrease the Absolute current-error parameter in the harmonic-balance simulator to achieve good accuracy.

Implementation Details

SCALING - When MODE=Square Law, the output-scaling parameter SCALE multiplies SQSNS. When the MODE=Linear, the default scaling gives a video output voltage equal to the input RF envelope-voltage amplitude, and SQSNS is the only means to scale the output.

CIRCUIT - The circuit shown in the symbol is somewhat stylized and does not represent the equivalent circuit of the model. The output impedance, for example, is linear, and the input impedance is only very weakly nonlinear. In contrast to the behavior of many real detectors, the video time constant is the same for positive- and negative-going pulses.

Layout

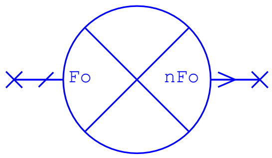
This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

References

This model was developed under research performed at NI AWR Corporation. The full set of details of the implementation are considered proprietary in nature.

Frequency Multiplier (System): FMULT

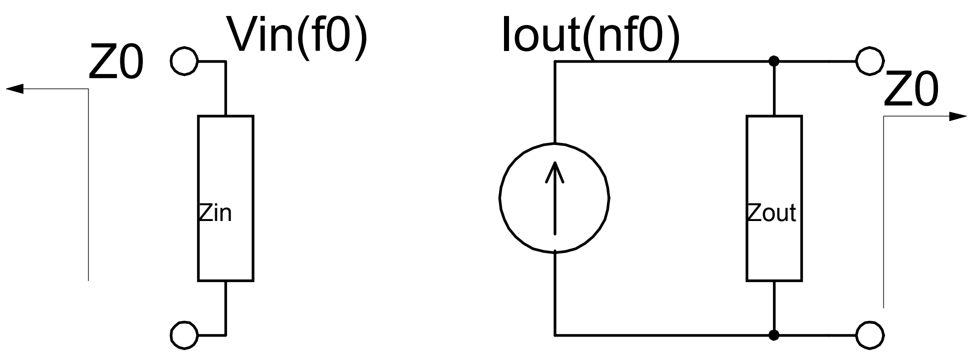
Symbol



Summary

FMULT is a behavioral representation of a frequency multiplier. The model converts a signal at frequency F_0 to a signal at frequency NF_0. The conversion gain is user-specified.

Equivalent Circuit



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	
ID	Element ID		FM1
N	Frequency multiplication factor		2
G	Conversion gain	DB	-3
*F0	Input center frequency	Frequency	_FREQH1
*BW	Signal bandwidth as a % of input center frequency		25
*S11MAG	Input reflection coefficient (magnitude)		0
*S11ANG	Input reflection coefficient (phase)		0
*S22MAG	Output reflection coefficient (magnitude)		0
*S22ANG	Output reflection coefficient (phase)		0
Z0	Port impedance	Resistance	50 ohm
Pth	Input power threshold	Power (dB)	-10

** indicates a secondary parameter*

Parameter Details

Parameter BW sets the model's passband for multitone operation. Set the parameter high enough to cover the bandwidth of the input signal.

The specified input and output impedances are valid for all frequencies. The input and output ports are AC-coupled.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

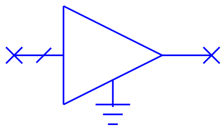
Recommendations for Use

Parameter $P_{th}$is extremely important for proper operation of the model. It should be set approximately 20 dB below the expected available input power. Values higher than 10 dB below input will result in incorrect conversion gain; values lower than 30 dB below input will result in slower convergence.

The model operates as a frequency mixer, with the local oscillator at frequency (N-1)F_0 generated from the input signal by a clipping operation. The lower the value of P_{th} the harder the clipping and the slower the convergence. Conversely, the higher the value of P_{th} the softer the clipping, the smaller the amplitude of the local oscillator and the less accurate the conversion gain estimate

Gain Block (Closed Form): GAIN

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Name	Text	U1
A	Gain, dB	dB	0 dB
SL	Positive (lower) gain slope, dB/octave		6
SH	Negative (upper) gain slope, dB/octave		6
FL	Lower break frequency for positive gain slope	Frequency	0 GHz
FH	Upper break frequency for negative gain slope	Frequency	0 GHz
R	Port resistance	Resistance	50 ohms

Implementation Details

Implements an ideal gain block with input and output impedances given by R. Gain is frequency-dependent with positive gain slope, SL, stopping at frequency FL and negative gain slope, SH, starting at frequency FH. The gain is constant between FL and FH, inclusive.

$$A(f) = A + \mathrm{SL} * \log_2\left(\frac{f}{\mathrm{FL}}\right) \qquad \text{for } f < \mathrm{FL},\ \mathrm{SL} > 0, \text{ and } \mathrm{FL} > 0$$

$$A(f) = A - \mathrm{SH} * \log_2\left(\frac{f}{\mathrm{FH}}\right) \qquad \text{for } f > \mathrm{FH},\ \mathrm{SH} > 0, \text{ and } \mathrm{FH} > 0$$

$$A(f) = A \text{ when } \mathrm{FL} \le f \le \mathrm{FH}, \text{ or } \mathrm{FL} = 0 \text{ and } \mathrm{FH} = 0, \text{ or } \mathrm{SL} = 0 \text{ and } \mathrm{SH} = 0$$

$$S_{21} = 10^{(A(f))/20}$$

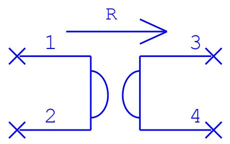
$$S_{12} = \frac{1}{S_{21}}$$

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Gyrator (Closed Form): GYR8R

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Name	Text	U1
R	Gyrational resistance	Resistance	50 ohm

Implementation Details

The gyrator is defined by either Y or Z parameters:

$$Y = \begin{bmatrix} 0 & \frac{1}{R} \\ \frac{-1}{R} & 0 \end{bmatrix}$$

$$Z = \begin{bmatrix} 0 & -R \\ R & 0 \end{bmatrix}$$

or by the defining equations:

V_1=-RI_2

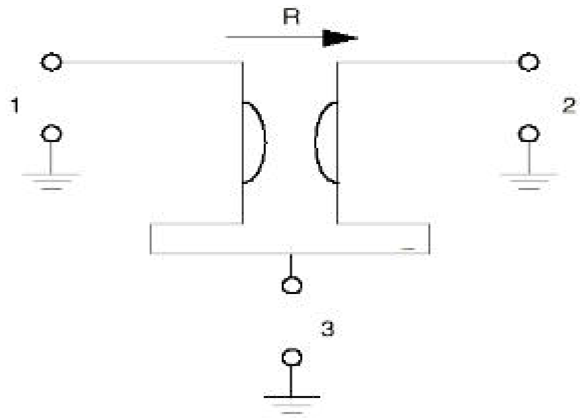
V_2=RI_1

A gyrator is a noiseless element. Although the gyrator is a relatively useless element by itself, it can be used to realize elements such as ideal transformers, voltage-controlled voltage sources, and current-controlled sources, which do not have nodal representations and therefore cannot be represented directly within the program.

These elements can be represented as a cascade of one or more gyrators and VCCS elements. The forms are as shown below.

Element	Form	Transfer function
Transformer	Gyr 1 * Gyr 2	n : 1 = R 1 / R 2
CCCS	Gyr * VCCS	I 2 / I 1 = R * G m
VCVS	VCCS * Gyr	V 2 / V 1 = - G m * R
CCVS	Gyr 1 * VCCS * Gyr 2	V 2 / I 1 = - G m * R 1 * R 2

A gyrator can also realize an ideal circulator. The most practical form (of several possible) is shown below; R is equal to the port impedance and the direction of circulation is from port 1 to port 2 to port 3. An ideal isolator can be realized by connecting a resistor of resistance R to port 3.



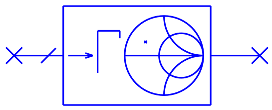
NOTE: It is possible for very simple circuits using gyrators to have no nodal representation; such circuits may generate error messages. Because it is nearly impossible to trap all such errors, the user must assume the primary responsibility for preventing them.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Harmonic Balance Lossless Tuner (Closed Form): HBTUNER

Symbol



Summary

The HBTUNER element is a frequency-dependent, lossless two-port network which transforms the impedance Z_0 seen at port 2 to a user-defined impedance seen at port 1. Specifying the magnitude and angle of the reflection coefficient as well as the system reference impedance sets the user-defined impedance. It is assumed that port 2 of this element is loaded in the same Z_0 as specified for the element.

You can use this model in load pull simulations using NI AWR's Load Pull Wizard.

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	TU1
Mag1	Magnitude of gamma looking into port 1 @ Fo		0.5
Ang1	Angle of gamma looking into port 1 @ Fo	Angle	0 Deg
Mag2	Magnitude of gamma looking into port 1 @ 2Fo		0.5
Ang2	Angle of gamma looking into port 1 @ 2Fo	Angle	0 Deg
Mag3	Magnitude of gamma looking into port 1 @ 3Fo		0.5
Ang3	Angle of gamma looking into port 1 @ 3Fo	Angle	0 Deg
Fo	Fundamental frequency	Frequency	10 GHz
Zo	System impedance	Resistance	50 ohms

Parameter Details

Fo. The specified fundamental frequency determines the impedance seen at port 1. This impedance, Z, is represented as:

$$\mathrm{Zin}_{\mathrm{port1}} = Z_0 * \frac{1 + \mathrm{Mag1} * e^{j*\mathrm{Ang1}}}{1 - \mathrm{Mag1} * e^{j*\mathrm{Ang1}}} \qquad \text{for } 1\mathrm{Hz} \le f \le 1.5F_0$$

$$\mathrm{Zin}_{\mathrm{port1}} = Z_0 * \frac{1 + \mathrm{Mag2} * e^{j*\mathrm{Ang2}}}{1 - \mathrm{Mag2} * e^{j*\mathrm{Ang2}}} \qquad \text{for } 1.5F_0 \le f \le 2.5F_0$$

$$\mathrm{Zin}_{\mathrm{port1}} = Z_0 * \frac{1 + \mathrm{Mag3} * e^{j*\mathrm{Ang3}}}{1 - \mathrm{Mag3} * e^{j*\mathrm{Ang3}}} \qquad \text{for } f \ge 2.5F_0$$

where f is the current simulation frequency.

Parameter Restrictions and Recommendations

1. -1 < Mag1, Mag2, Mag3 < 1. If Mag1, Mag2, or Mag3 is less than or equal to -1, it will be set to -0.99999. If Mag1, Mag2, or Mag3 is greater or equal to 1, it will be set to 0.99999.
2. Zo must be greater than or equal to 0.1 ohm.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

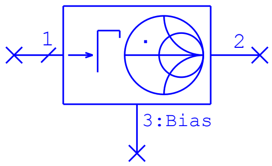
Recommendations for Use

NOTES: The impedance seen at port 1 of the element is a lossless transformation of an impedance Zo seen at port 2, to the specified reflection coefficient. If port 2 is not loaded with an impedance equal to Zo, the reflection seen at port 1 will not be as specified.

This element can be used to replace a lossless matching network. However, when comparing S-Parameters to an actual realization of this matching network, the phase of S12,S21and S22 may not agree, due to the incomplete specification of the network by only defining only the reflection at port 1. A finite length of ideal transmission line (impedance = Zo) placed at port 2 of the tuner may be required to make the make S-Parameters agree.

Harmonic Balance Lossless Tuner with Bias Tee (Closed Form): HBTUNER2

Symbol



Summary

The HBTUNER2 element is a frequency-dependent, lossless network which transforms the impedance Z0 seen at port 2 to a user-defined impedance seen at port 1. Specifying the magnitude and angle of the reflection coefficient as well as the system reference impedance sets the user-defined impedance. It is assumed that port 2 of this element is loaded in the same Z0 as specified for the element. A bias voltage can be applied to port 3 to bias a device connected to port 1, eliminating the need for a BIASTEE element.

You can use this model in load pull simulations using NI AWR's Load Pull Wizard.

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	TU1
Mag1	Magnitude of gamma looking into port 1 @ Fo		0.5
Ang1	Angle of gamma looking into port 1 @ Fo	Angle	0 Deg
Mag2	Magnitude of gamma looking into port 1 @ 2Fo		0.5
Ang2	Angle of gamma looking into port 1 @ 2Fo	Angle	0 Deg
Mag3	Magnitude of gamma looking into port 1 @ 3Fo		0.5
Ang3	Angle of gamma looking into port 1 @ 3Fo	Angle	0 Deg
Fo	Fundamental Frequency	Frequency	10 GHz
Zo	System Impedance	Resistance	50 ohms

Parameter Details

Fo. The specified fundamental frequency is used to determine the impedance seen at port 1. This impedance, Z, is represented as:

$$\mathrm{Zin}_{\mathrm{port1}} = Z_0 * \frac{1 + \mathrm{Mag1} * e^{j*\mathrm{Ang1}}}{1 - \mathrm{Mag1} * e^{j*\mathrm{Ang1}}} \qquad \text{for } 1\mathrm{Hz} \le f \le 1.5F_0$$

$$\mathrm{Zin}_{\mathrm{port1}} = Z_0 * \frac{1 + \mathrm{Mag2} * e^{j*\mathrm{Ang2}}}{1 - \mathrm{Mag2} * e^{j*\mathrm{Ang2}}} \qquad \text{for } 1.5F_0 \le f \le 2.5F_0$$

$$\mathrm{Zin}_{\mathrm{port1}} = Z_0 * \frac{1 + \mathrm{Mag3} * e^{j*\mathrm{Ang3}}}{1 - \mathrm{Mag3} * e^{j*\mathrm{Ang3}}} \qquad \text{for } f \geq 2.5F_0$$

where f is the current simulation frequency.

Parameter Restrictions and Recommendations

1. -1 < Mag1, Mag2, Mag3 < 1. If Mag1, Mag2, or Mag3 is less than or equal to -1, it will be set to -0.99999. If Mag1, Mag2, or Mag3 is greater or equal to 1, it will be set to 0.99999.
2. Zo must be greater than or equal to 0.1 ohm.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

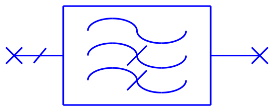
Recommendations for Use

NOTES: The impedance seen at port 1 of the element is a lossless transformation of an impedance Zo seen at port 2, to the specified reflection coefficient. If port 2 is not loaded with an impedance equal to Zo, the reflection seen at port 1 will not be as specified.

This element can be used to replace a lossless matching network. However, when comparing S-Parameters to an actual realization of this matching network, the phase of S12,S21and S22 may not agree, due to the incomplete specification of the network by only defining only the reflection at port 1. A finite length of ideal transmission line (impedance = Zo) placed at port 2 of the tuner may be required to make the make S-Parameters agree.

Butterworth Highpass Filter: HPFB

Symbol



Summary

HPFB models represent lumped-element Butterworth highpass filters. They offer simplicity and a compromise between high selectivity and flat group delay. The insertion loss is maximally flat at infinite frequency and the stopband attenuation increases monotonically with decreasing frequency at an asymptotic rate of 6N dB per octave.

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	HPFB1
N	Number of elements in the filter		3
FP	Passband corner frequency (when Qu is infinite).	Frequency	1 GHz
*AP	Passband corner attenuation (when Qu is infinite).	DB	3.0103 dB
*RS	Source resistance.	Resistance	50 ohm
*RL	Load resistance	Resistance	50 ohm
*QU	Uniform unloaded Q of elements		1e12

** indicates a secondary parameter*

Parameter Restrictions and Recommendations

1. 0 < N < 29
2. 0 < FP
3. 0 < AP Recommend AP greater than or equal to 0.001 dB.
4. 0 < RS
5. 0 < RL
6. 0 < QU. Recommend QU less than or equal to 1012.

Implementation Details

The model is implemented as a short-circuit admittance matrix, whose equivalent transfer function squared magnitude is that of a Butterworth filter:

$$|S_{21}(s)|^2 = \frac{1}{1+(-1)^N\varepsilon^2 S^{2N}} = \frac{1}{|g(s)|^2} = \frac{1}{1+|h(s)|^2}$$

where

$$\varepsilon^2 = 10^{\mathrm{AP}/10} - 1$$

$$s = \frac{1}{\mathrm{QU}} + j\omega$$

$$j = \sqrt{-1}$$

and a lowpass-to-highpass frequency transformation has been applied:

$$\omega = \frac{\_\mathrm{FREQ} - (\mathrm{FP2} \times \mathrm{FP1}) / \_\mathrm{FREQ}}{|\ \mathrm{FP2} - \mathrm{FP1}\ |}$$

_FREQ is the variable containing the project frequency, and the admittances are:

$$\omega = -\frac{\mathrm{FP}}{\_\mathrm{FREQ}}$$

$$y_{22} = \left(\frac{1}{\mathrm{RL}}\right)\frac{g(s) + g(-s) + h(s) + h(-s)}{g(s) - g(-s) + h(s) - h(-s)}$$

$$y_{12} = y_{21} = \left(\frac{1}{\sqrt{\mathrm{RS} \times \mathrm{RL}}}\right)\frac{2}{g(s) - g(-s) + h(s) - h(-s)}$$

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

Recommendations for Use

This model behaves as if it has ideal impedance transformers at its ports, so there is no attenuation due to mismatched source and load impedances. The model expects that the source impedance will equal RS and that the load impedance will equal RL, but RS need not equal RL for ideal transmission (as would normally be the case).

References

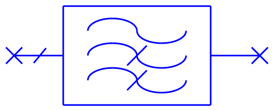
[1] Rolf Schaumann, Mohammed S. Ghausi, and Kenneth R. Laker, Design of Analog Filters: Passive, Active RC, and Switched Capacitor, (Prentice-Hall, 1990), pp. 40-44.

[2] Louis Weinberg, Network Analysis and Synthesis, (Robert E. Krieger Publishing, 1975), pp. 493-498.

[1] Adel S. Sedra and Peter O. Brackett, Filter Theory and Design: Active and Passive, (Matrix Publishers, 1978), pp. 105-111.

Chebyshev Highpass Filter: HPFC

Symbol



Summary

HPFC models represent lumped-element Chebyshev highpass filters. The insertion loss ripples between zero and a specified maximum in the passband. The stopband attenuation increases rapidly beyond the passband edge, is monotonic, and is maximally flat at zero frequency. This type of filter is popular because it offers both simplicity and good selectivity.

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	HPFC1
N	Number of reactive elements in the filter		3
FP	Passband corner frequency (when Qu is infinite).	Frequency	1 GHz
AP	Passband corner attenuation (when Qu is infinite).	DB	0.1 dB
*RS	Expected source resistance.	Resistance	50 ohm
*RL	Expected load resistance	Resistance	50 ohm
*QU	Average unloaded Q of filter reactive elements.		1e12

* indicates a secondary Parameter

Parameter Restrictions and Recommendations

1. 0 < N < 27
2. 0 < FP
3. 0 < AP Recommend AP greater than or equal to 0.001 dB.
4. 0 < RS
5. 0 < RL
6. 0 < QU. Recommend QU less than or equal to 1e12.

Implementation Details

The model is implemented as a short-circuit admittance matrix, whose equivalent transfer function squared magnitude is that of a Chebyshev filter:

$$|S_{21}(s)|^2 = \frac{1}{1+\varepsilon^2 C_n^2\left(\frac{s}{j}\right)} = \frac{1}{|g(s)|^2} = \frac{1}{1+|h(s)|^2}$$

where Cn is the Chebyshev polynomial of the first kind, and

$$\varepsilon^2 = 10^{AP/10} - 1$$

$$C_n(\omega) = \cos(N * \text{acos}(\omega)) \qquad \text{for } 0 \le |\omega| \le 1$$

$$C_n(\omega) = \cosh(N * \text{acosh}(\omega)) \qquad \text{for } 1 < |\omega|$$

$$s = \frac{1}{QU} + j\omega$$

$$j = \sqrt{-1}$$

and a lowpass-to-highpass frequency transformation has been applied:

$$\omega = -\frac{FP}{\_FREQ}$$

_FREQ is the variable containing the project frequency, and the admittances are:

$$y_{11} = \left(\frac{1}{RS}\right)\frac{g(s) + g(-s) - h(s) - h(-s)}{g(s) - g(-s) + h(s) - h(-s)}$$

$$y_{22} = \left(\frac{1}{RL}\right)\frac{g(s) + g(-s) + h(s) + h(-s)}{g(s) - g(-s) + h(s) - h(-s)}$$

$$y_{12} = y_{21} = \left(\frac{1}{\sqrt{RS \times RL}}\right)\frac{2}{g(s) - g(-s) + h(s) - h(-s)}$$

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

Recommendations for Use

This model behaves as if it has ideal impedance transformers at its ports, so there is no attenuation due to mismatched source and load impedances. The model expects that the source impedance will equal RS and that the load impedance will equal RL, but RS need not have any special relationship to RL for ideal transmission (as would normally be the case).

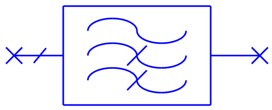
References

[1] Rolf Schaumann, Mohammed S. Ghausi, and Kenneth R. Laker, Design of Analog Filters: Passive, Active RC, and Switched Capacitor, (Prentice-Hall, 1990), pp. 44-48.

[2] Louis Weinberg, Network Analysis and Synthesis, (Robert E. Krieger Publishing, 1975), pp. 507-529.

Generalized Chebyshev Highpass Filter: HPFCG

Symbol



Summary

HPFCG models represent lumped-element Generalized Chebyshev (or "Quasi-Elliptic") highpass filters. The insertion loss ripples between zero and a specified maximum in the passband. The stopband attenuation is defined by arbitrarily specified transmission zeros. Real-frequency finite transmission zeros can be specified to improve selectivity at the expense of ultimate stopband attenuation and passband group delay, while complex-plane finite transmission zeros can be specified to provide passband group delay equalization at the expense of selectivity and ultimate stopband attenuation. Generalized Chebyshev filters represents a compromise between the simplicity of Chebyshev filters, the optimum amplitude response of more complicated Elliptic filters, and the phase linearity of Bessel and Gaussian filters. Because this type of filter allows one to make explicit design trade-offs between complexity, selectivity, and group delay equalization, it is often used to meet the demanding requirements of modern communications systems.

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	HPFCG1
N	Order of the filter		3
FP	Passband corner frequency (when Qu is infinite).	Frequency	1 GHz
*PPD	Passband parameter description:- Maximum Insertion Loss,- Minimum Return Loss, or- VSWR.	Enumerated	Maximum Insertion Loss
PPV	Passband parameter value (when Qu is infinite)	dB or Scalar	0.1 dB
TZF	Real frequency, finite transmission zeros	(Real) Frequency	{0.5} GHz
*TZR	Real parts of complex finite transmission zeros	(Imaginary) Frequency	{0} GHz
*RS	Source resistance	Resistance	50 ohm
*RL	Load resistance	Resistance	50 ohm
*QU	Average unloaded Q of reactive elements in the filter.		1e12

* indicates a secondary Parameter

Parameter Details

N. In mathematical terms, N is defined as the highest exponent of the complex frequency variable s in the transfer function, $S_{21}(s)$, of the filter's normalized lowpass prototype, or, equivalently, the highest exponent of s in the transfer function of the highpass filter. And, in terms of a measurable electrical characteristic, the number of positive-frequency passband reflection ($S|_{11}|$) zeros corresponds to N/2 for N even and (N+1)/2 for N odd.

PPD & PPV. Parameters PPD and PPV work together to specify the characteristic of the filter's passband. PPD is used to indicate what the value of PPV represents. The flexibility these parameters provide eliminates the need to manually convert from the passband specification parameter of one's preference into whatever specific parameter the software was written to accept.

PTZF & TZR. List parameters TZF and TZR are used to specify the complex transmission zeros, Z, of the highpass filter response. Up to (N-1)/2 complex transmission zeros, $Z_i = TZR_i + jTZF_i$, can be specified. Each consists of a real part, TZR_i, and an imaginary part, the real frequency TZF_i. If TZR_i is not specified, it is assumed to be zero. And, each unspecified transmission zero is mapped to a normalized lowpass prototype frequency of infinity.

Parameter Restrictions and Recommendations

1. 32 > N > 1.
2. FP > 0.
3. If PPD = "Maximum Insertion Loss", then PPV > 0.

   If PPD = "Minimum Return Loss", then PPV > 0.

   If PPD = "Maximum VSWR", then PPV > 1.
4. $TZF_i > 0$.

   If TZR_i is specified, then TZF_i must be specified.

   If TZR_i is zero, then $FP > TZF_i$.

   If $TZR_i \neq 0$, there must be a $TZR_k = -TZR_i$ and a $TZF_k = TZF_i$.
5. RS > 0.

   RL > 0.
6. QU > 0 specifies a finite unloaded Q (recommend $QU < 1e^{12}$).

   QU = 0 specifies an infinite unloaded Q.

Implementation Details

The model is implemented as a short-circuit admittance matrix,

$$\begin{bmatrix} y_{11} & y_{12} \\ y_{21} & y_{22} \end{bmatrix}$$

, whose equivalent normalized lowpass prototype transfer function, $S_{21}(s)$, is [1, 2]:

$$s_{21}^2(s) = \frac{P_N^2(s)}{P_N^2(s) + \varepsilon^2 F_N^2(s)} = \frac{P_N^2(s)}{P_N(s) + j\varepsilon F_N(s))(P_N(s) - j\varepsilon F_N(s))} = \frac{P_N^2(s)}{\varepsilon^2 E_N^2(s)}$$

where F_N and E_N are polynomials of order N, and

$$\exp(-s^2) \approx \sum_{i=0}^{N} \frac{(-1)^i s^{2i}}{i!} = |g(s)|^2 = 1 + |h(s)|^2$$

$$\varepsilon^2 = \frac{1}{10^{PPV/10} - 1} \text{when PPD = " Minimum Return Loss "}$$

$$\varepsilon^2 = \frac{0.25(\text{PPV} - 1)^2}{\text{PPV}} \text{ when PPD="Maximum VSWR"}$$

$$P_N(s) = \prod_{i=1}^{N}\left(1 - \frac{s}{z_i}\right)$$

$$s = \frac{1}{q_u} + j\omega$$

$$j = \sqrt{-1}$$

A specified highpass transmission zero, Z[i] = TZR[i] + jTZF[i] is mapped to a normalized lowpass prototype transmission zero, z[i], using [5] [5–107]:

$$z[i] = \frac{\text{FP}}{Z[i]}$$

And, _FREQ (the variable that represents the project frequency) is mapped to the normalized lowpass prototype radian frequency,ω, using [5] [5–107]:

$$\omega = \frac{-\text{FP}}{\text{\_FREQ}}$$

Polynomial

$$F_N(s) = F_i|_{i=N}$$

is constructed using a doubly recursive algorithm [1] [5–106][3] [5–107]:

$$G_0 = b_1$$

$$F_1 = a_1$$

$$G_{(i-1)} = a_i G_{(i-2)} + b_i F_{(i-1)}$$

$$F_i = a_i F_{(i-1)} + c_i G_{(i-2)}$$

where i = 2 to N and, employing the normalized lowpass prototype transmission zero, z_k, for k = 1 to N:

$$a_k = -j\left(s + \frac{1}{z_k}\right)$$

$$b_k = \sqrt{1 + \left(\frac{1}{z_k}\right)^2}$$

$$c_k = -b_k\left(s^2+1\right)$$

Polynomial $E_s(s)$ is found by applying the "alternating singularity principle" [1] [5–106][2] [5–106][4] [5–107] to the roots of

$$\left(\frac{P_N(s)}{\varepsilon} + \mathrm{jF}_N(s)\right)$$

. Then, E_N and F_N are split into complex-even and complex-odd polynomials [2] [5–106] such that $E_N= E_e+E_0$and $F_N= F_e+F_0$, where

$$E_e = \mathrm{Re}(e_0) + j \quad \mathrm{Im}(e_1)s + \mathrm{Re}(e_2)s^2 + \cdots, E_o = j \quad \mathrm{Im}(e_o) + \mathrm{Re}(e_1) + j \quad \mathrm{Im}(e_2)s^2 + \cdots$$

$$F_e = \mathrm{Re}(f_0) + j\ \mathrm{Im}(f_1)s + \mathrm{Re}(f_2)s^2 + \cdots, F_o = j\ \mathrm{Im}(f_o) + \mathrm{Re}(f_1) + j\ \mathrm{Im}(f_2)s^2 \cdots$$

and and e_1 and f_1 (i = 0 to N) are the complex coefficients of E_N and F_N. Finally [3]:

$$y_{11} = \left(\frac{1}{\mathrm{RS}}\right)\left(\frac{E_e - F_e}{E_o + F_o}\right)$$

$$y_{22} = \left(\frac{1}{\mathrm{RL}}\right)\left(\frac{E_e + F_e}{E_o + F_o}\right)$$

$$y_{12} = y_{21} = \left(\frac{1}{\sqrt{\mathrm{RS} \times \mathrm{RL}}}\right)\left(\frac{-P_N/\varepsilon}{E_o + F_o}\right)$$

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

Recommendations for Use

The transmission zeros can be tuned or optimized by assigning variables to the elements of the TZF and/or TZR lists and then tuning or optimizing these variables.

Note that this model behaves as if it has ideal impedance transformers at its ports, so there is no attenuation due to mismatched source and load impedances. The model expects that the source impedance will equal RS and that the load impedance will equal RL, but RS need not have any special relationship to RL for ideal transmission (as would normally be the case).

References

[1] Richard J. Cameron, "Fast generation of Chebyshev filter prototypes with asymmetrically-prescribed transmission zeros," ESA J., vol. 6, pp. 83-95, 1982.

[2] Richard J. Cameron, "General coupling matrix synthesis methods for Chebyshev filtering functions," IEEE Trans. Microwave Theory Tech., vol. 47, no. 4, pp. 433-442, April 1999.

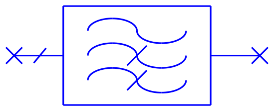
[3] Douglas R. Jachowski, unpublished notes, 1995 and 2002.

[4] J. D. Rhodes and A. S. Alseyab, "The generalized Chebyshev low pass prototype filter," Int. J. Circuit Theory Applicat., vol. 8, pp. 113-125, 1980.

[5] H. J. Blinchikoff and A. I. Zverev, Filtering in the Time and Frequency Domains, (Robert E. Krieger Publishing Co., 1987), pp. 157, 158, 163

Bessel Highpass Filter: HPFD

Symbol



Summary

HPFD models represent lumped-element Bessel-Thomson highpass filters. They offer simplicity and maximally flat group delay, but suffer from poor selectivity.

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	HPFD1
N	Number of reactive elements in the filter		3
FP	Passband corner frequency (when Qu is infinite).	Frequency	1 GHz
AP	Passband corner attenuation (when Qu is infinite).	DB	3.0103 dB
*RS	Expected source resistance.	Resistance	50 ohm
*RL	Expected Load resistance	Resistance	50 ohm
*QU	Average unloaded Q of reactive elements in filter		1e12

* indicates a secondary Parameter

Parameter Restrictions and Recommendations

1. 0 < N < 32
2. 0 < FP
3. 0 < AP Recommend AP greater than or equal to 0.001 dB.
4. 0 < RS
5. 0 < RL
6. 0 < QU. Recommend QU less than or equal to 1e12.

Implementation Details

The model is implemented as a short-circuit admittance matrix,

$$\begin{bmatrix} y_{11} & y_{12} \\ y_{21} & y_{22} \end{bmatrix}$$

, whose equivalent transfer function squared magnitude is that of a Bessel-Thomson filter:

$$|S_{21}(s)|^2 = \frac{b_0^2}{|g(s)|^2} = \frac{b_0^2}{1 + |h(s)|^2}$$

where

$$g(s) = \sum_{i=0}^{N} b_i s^i$$

$$b_i = \frac{(2N-i)!}{2^{N-i}(i)!(N-i)!} \text{ for } i = 0, \ldots, N$$

Where b_0 = (2N-1)!!,i.e, the product of all odd integers less than 2N.

$$s = \frac{1}{\mathrm{QU}} + j\omega$$

$$j = \sqrt{-1}$$

and a lowpass-to-highpass frequency transformation has been applied:

$$\omega = -\frac{\mathrm{FP}}{\mathrm{\_FREQ}}$$

_FREQ is the variable containing the project frequency, and the admittances are:

$$y_{11} = \left(\frac{1}{\mathrm{RS}}\right)\frac{g(s)+g(-s)-h(s)-h(-s)}{g(s)-g(-s)+h(s)-h(-s)}$$

$$y_{22} = \left(\frac{1}{\mathrm{RL}}\right)\frac{g(s)+g(-s)+h(s)+h(-s)}{g(s)-g(-s)+h(s)-h(-s)}$$

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

Recommendations for Use

This model behaves as if it has ideal impedance transformers at its ports, so there is no attenuation due to mismatched source and load impedances. The model expects that the source impedance will equal RS and that the load impedance will equal RL, but RS need not equal RL for ideal transmission (as would normally be the case).

References

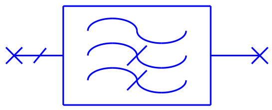
[1] Rolf Schaumann, Mohammed S. Ghausi, and Kenneth R. Laker, Design of Analog Filters: Passive, Active RC, and Switched Capacitor, (Prentice-Hall, 1990), pp. 51-56.

[2] Louis Weinberg, Network Analysis and Synthesis, (Robert E. Krieger Publishing, 1975), pp. 499-506.

[3] Herman J. Blinchikoff and Anatol I. Zverev, Filtering in the Time and Frequency Domains, (Robert E. Krieger Publishing Co., 1987), pp. 124-127.

Elliptic-Function Highpass Filter (Closed Form): HPFE

Symbol



Summary

HPFE models represent lumped-element elliptic-function (Cauer) highpass filters. The insertion loss ripples between zero and a specified maximum in the passband. The stopband attenuation increases rapidly between the passband edge and the stopband edge, and ripples between a specified minimum stopband attenuation and infinite attenuation. This type of filter offers optimum selectivity at the expense of increased complexity and poor group delay flatness.

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	HPFE1
N	Order of the filter.		3
FP	Passband corner frequency (when Qu is infinite).	Frequency	1 GHz
AP	Maximum Passband Insertion Loss (when Qu is infinite).	DB	0.1 dB
AS	Minimum Stopband Attenuation(when Qu is infinite).	DB	20 dB
FS	Stopband corner frequency (when Qu is infinite).	Frequency	0 GHz
DM	Where to put design margin:0=AP, 1=AS, 2=FP, 3=FS.		1
RS	Source resistance.	Resistance	50 ohm
RL	Load resistance	Resistance	50 ohm
QU	Average unloaded Q of reactive elements in the filter.		1e12

Parameter Details

An elliptic function highpass prototype filter is completely specified by any four of the five parameters: N, FP, FS, AP, and AS. That is, the value of any parameter is dependent on the value of the other four parameters.

- If zero is specified for any one of these five parameters, then the model computes its value from the value of the other four parameters.
- If a value is specified for all five parameters, then the specified value of the last parameter (AS) is ignored and is computed by the model from the values of the other four parameters.
- It is an error to specify zero for more than one of these five parameters.

N. In mathematical terms, N is defined as the highest exponent of the complex frequency variable s in the transfer function, S21(s), of the filter's normalized lowpass prototype, or, equivalently, the highest exponent of s in the transfer function of the highpass filter. And, in terms of a measurable electrical characteristic, the number of positive-frequency passband reflection (|S11|) zeros corresponds to N/2 for N even and (N+1)/2 for N odd.

DM. If zero is specified for N, the model will determine N. But, since N must be an integer, there will typically be some design margin available, and this margin must be assigned to some parameter other than N. The value of DM determines where the model will assign this design margin.

- DM=0. After determining N, the model will recompute AP, such that 0 < new AP ≤ AP
- DM=1. After determining N, the model will recompute AS, such that AS ≤ new AS < ∞
- DM=2. After determining N, the model will recompute FP, such that FS < new FP ≤ FP
- DM=3. After determining N, the model will recompute FS, such that FS ≤ new FS < FP

Parameter Restrictions and Recommendations

1. N can be zero if FP, FS, AP, and AS are not, otherwise 0 < N < 27.
2. FP can be zero if N, FS, AP, and AS are not, otherwise 0 < FP, and, if FS is not zero, then FS < FP.

   FS can be zero if N, FP, AP, and AS are not, otherwise 0 < FS, and, if FP is not zero, then FS < FP.
3. AP can be zero if N, FP, FS, and AS are not, otherwise 0 < AP, and, if AS is not zero, then AP < AS.

   Recommend AP greater than or equal to 0.001 dB.
4. AS can be zero if N, FP, FS, and AP are not, otherwise AP < AS.
5. 0 < RS.

   0 < RL
6. QU > 0 specifies a finite unloaded Q (recommend QU ≤ 1e12).

   QU = 0 specifies an infinite unloaded Q.
7. 0 ≤ DM < 4

Implementation Details

The model is implemented as a short-circuit admittance matrix,

$$\begin{bmatrix} y_{11} & y_{12} \\ y_{21} & y_{22} \end{bmatrix}$$

, whose equivalent normalized lowpass prototype transfer function, S21(s), is:

$$|\,S_{21}(s)|^2 = \frac{1}{1+\varepsilon^2 R_n^2\left(\frac{s}{j}\right)} = \frac{f(s)^2}{|\,g(s)|^2} = \frac{f(s)^2}{1+\,|\,h(s)|^2}$$

where Rn is the Elliptic Rational Function, and

$$\varepsilon^2 = 10^{\mathrm{AP}/10} - 1$$

$$R_n(\omega) = C \times \omega^{\Pi} \times \prod_{i=1}^{\frac{N-\Pi}{2}} \left(\frac{\omega^2 - \omega_i^2}{\omega^2 - \frac{\omega_s^2}{\omega_i^2}}\right)$$

$$C = \prod_{i=1}^{\frac{N-\Pi}{2}} \left(\frac{1 - \frac{\omega_s^2}{\omega_i^2}}{1 - \omega_i^2} \right)$$

$$\omega_i = \mathrm{cd}\left(\frac{2i-1}{N} K\left(\frac{1}{\omega_s^2} \right), \frac{1}{\omega_s^2} \right)$$

$$s = \frac{1}{\mathrm{QU}} + j\omega$$

$$v = 0 \; if N \; is \; even, \;\; v = 1 \; if N \; is \; odd,$$

$$j = \sqrt{-1}$$

where cd is a Jacobian elliptic function, K is Legendre's complete elliptic integral of the first kind, and a highpass-to-lowpass frequency transformation has been applied:

$$\omega = -\frac{\mathrm{FP}}{\mathrm{\_FREQ}}$$

and

$$\omega_s = -\frac{\mathrm{FP}}{\mathrm{FS}}$$

_FREQ is the variable containing the project frequency. The parameters of the elliptic-function filter (N, FP, FS, AP, AS) are related by "the degree equation":

$$N = \left(\frac{K(m)}{K(1-m)} \right) \left(\frac{K(1-m_o)}{K(m_o)} \right)$$

where

$$m_o = \frac{10^{\mathrm{AP}/10} - 1}{10^{\mathrm{AS}/10} - 1}$$

and

$$m = \left(\frac{FS}{FP} \right)^2$$

The admittances are given by:

$$y_{11} = \left(\frac{1}{\mathrm{RS}} \right) \frac{g(s) + g(-s) - h(s) - h(-s)}{g(s) - g(-s) + h(s) - h(-s)}$$

$$y_{22} = \left(\frac{1}{\mathrm{RL}} \right) \frac{g(s) + g(-s) + h(s) + h(-s)}{g(s) - g(-s) + h(s) - h(-s)}$$

$$y_{12} = y_{21} = \left(\frac{1}{\sqrt{\mathrm{RS} \times \mathrm{RL}}}\right)\frac{2 \times f(s)}{g(s) - g(-s) + h(s) - h(-s)}$$

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Recommendations for Use

This model behaves as if it has ideal impedance transformers at its ports, so there is no attenuation due to mismatched source and load impedances. The model expects that the source impedance will equal RS and that the load impedance will equal RL, but RS need not have any special relationship to RL for ideal transmission (as would normally be the case).

References

[1] Miroslav D. Lutovac, Dejan V. Tosic, and Brian L. Evans, Filter Design For Signal Processing Using MATLAB and Mathematica, (Prentice Hall, 2001), Chapters 6, 12, and 13.

[2] Alexander J. Grossman, "Synthesis of Tchebycheff parameter symmetrical filters," Proceedings of the IRE, pp. 545-473, April 1957.

[3] Dante Youla, "A tutorial exposition of some key network-theoretic ideas underlying classical insertion-loss filter design," Proc. IEEE, vol. 59, no. 5, pp. 760-799, May 1971.

[4] Miroslav Vleck and Rolf Unbehauen, "Degree, ripple, and transition width of elliptic filters," IEEE Trans. Circuits Syst., vol. 36, no. 3, pp. 469-472, March 1989.

[5] H. J. Orchard and Alan N. Willson, Jr., "Elliptic functions for filter design," IEEE Trans. Circuits Syst., I, vol. 44, no. 4, pp. 273-287, April 1997.

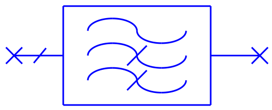
[6] Rolf Schaumann, Mohammed S. Ghausi, and Kenneth R. Laker, Design of Analog Filters: Passive, Active RC, and Switched Capacitor, (Prentice-Hall, 1990), pp. 49-51.

[7] Kendall L. Su, Handbook of Tables for Elliptic-Function Filters, (Kluwer Academic, 1990).

[8] Handbook of Mathematical Functions With Formulas, Graphs, and Mathematical Tables, edited by Milton Abramowitz and Irene A. Stegun, (U. S. National Bureau of Standards, 1964), Chapters 16 and 17 by L. M. Milne-Thomson.

Gaussian Highpass Filter (Closed Form): HPFG

Symbol



Summary

HPFG models represent lumped-element Gaussian highpass filters. They approximate the ideal Gaussian magnitude response and offer simplicity, relatively flat group delay, and good time domain performance, but suffer from poor frequency selectivity. Although similar to Bessel-Thomson filters, Gaussian filters offer faster rise times and lower transient overshoots, but have slightly less stopband attenuation and less group delay flatness.

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	HPFG1
N	Number of reactances in the filter		3
FP	Passband corner frequency (when Qu is infinite).	Frequency	1 GHz
AP	Passband corner attenuation (when Qu is infinite).	DB	3.0103 dB
*RS	Expected source resistance.	Resistance	50 ohm
*RL	Expected Load resistance	Resistance	50 ohm
*QU	Average unloaded Q of reactive elements in filter		1e12

* indicates a secondary Parameter

Parameter Restrictions and Recommendations

1. $0 < N < 51$
2. $0 < FP$
3. $0 < AP$ Recommend AP greater than or equal to 0.001 dB.
4. $0 < RS$
5. $0 < RL$
6. $0 < QU$. Recommend QU less than or equal to 1e12.

Implementation Details

The model is implemented as a short-circuit admittance matrix,

$$\begin{bmatrix} y_{11} & y_{12} \\ y_{21} & y_{22} \end{bmatrix}$$

, whose equivalent transfer function squared magnitude is that of an ideal Gaussian filter. The ideal Gaussian squared magnitude characteristic is:

$$|S_{21}|^2 = \frac{1}{\exp(-s^2)}$$

In the model, the denominator of this ideal Gaussian characteristic is approximated by a truncated Maclaurin series:

$$\exp(-s^2) \approx \sum_{i=0}^{N} \frac{(-1)^i s^{2i}}{i!} = |g(s)|^2 = 1 + |h(s)|^2$$

where

$$s = \frac{1}{\mathrm{QU}} + j\omega$$

$$j = \sqrt{-1}$$

and a lowpass-to-highpass frequency transformation has been applied:

$$\omega = -\frac{\mathrm{FP}}{\mathrm{\_FREQ}}$$

_FREQ is the variable containing the project frequency, and the admittances are:

$$y_{11} = \left(\frac{1}{\mathrm{RS}}\right)\frac{g(s) + g(-s) - h(s) - h(-s)}{g(s) - g(-s) + h(s) - h(-s)}$$

$$y_{22} = \left(\frac{1}{\mathrm{RL}}\right)\frac{g(s) + g(-s) + h(s) + h(-s)}{g(s) - g(-s) + h(s) - h(-s)}$$

$$y_{12} = y_{21} = \left(\frac{1}{\sqrt{\mathrm{RS} \times \mathrm{RL}}}\right)\frac{(-2)}{g(s) - g(-s) + h(s) - h(-s)}$$

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Recommendations for Use

This model behaves as if it has ideal impedance transformers at its ports, so there is no attenuation due to mismatched source and load impedances. The model expects that the source impedance will equal RS and that the load impedance will equal RL, but RS need not equal RL for ideal transmission (as would normally be the case).

References

[1] Milton Dishal, "Gaussian-Response Filter Design," Electrical Communication, vol. 36, March 1959, pp. 3-26.

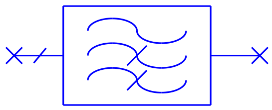
[2] Anatol I. Zverev, Handbook of Filter Synthesis, (John Wiley & Sons, 1967), pp. 67, 70, 71, 73, 74, 90, 91.

[3] DeVerl. S. Humpherys, The Analysis, Design, and Synthesis of Electrical Filters, (Prentice-Hall, 1970), pp. 413-417.

[4] Herman J. Blinchikoff and Anatol I. Zverev, Filtering in the Time and Frequency Domains, (Robert E. Krieger Publishing Co., 1987), pp. 130-132.

Raised-Cosine Highpass Filter (Closed Form): HPFRC

Symbol



Summary

HPFRC models represent Raised-Cosine Highpass filters. Raised-Cosine filters are applied in digital transmission system theory to represent ideal (intersymbol interference free) Nyquist filtering of impulse and pulse data transmissions. Hardware designers attempt to design channel filters which approximate these characteristics and have the smallest possible bandwidth.

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	HPFRC1
FP	Passband corner frequency	Frequency	1 GHz
A	Roll-off factor (between 0 and 1)		0.5
TYP	Transmission type	Vector Text(pull-down)	Impulse
E	Exponent of the raised-cosine response		1.
*RS	Source resistance	Resistance	50 ohm
*RL	Load resistance	Resistance	50 ohm

* indicates a secondary Parameter

Parameter Restrictions and Recommendations

1. $0 < FP$
2. $0 \leq A \leq 1$.

   A minimum bandwidth (A=0) filter would require an infinite number of filter sections. Approximations of a 30% excess Nyquist bandwidth (A=0.3) raised-cosine filter are considered feasible with present-day technology.
3. TYP is either $\{0,1\}$.

   TYP = 0 (Impulse) specifies the ideal raised-cosine response, the theoretical filter model for infinitesimally narrow impulses. TYP = 1 (Pulse) specifies a sinc-normalized raised-cosine response, the filter model for rectangular pulse transmission.
4. $0<E\leq 1$.

   The ideal raised-cosine response is raised to the exponent, E. For cascaded identical transmit and receive filters, you would typically specify E=0.5 for each filter. The composite channel response of the cascaded filters would be represented by specifying E=1.
5. $0 < RS$.

By definition, the model matches RS at its input port.

6. 0 < RL.

   By definition, the model matches RL at its output port.

Implementation Details

The model is implemented as a short-circuit admittance matrix,

$$\begin{bmatrix} y_{11} & y_{12} \\ y_{21} & y_{22} \end{bmatrix}$$

, whose equivalent amplitude functions S21 and S12 implement the raised-cosine response for impulse or pulse data transmission, while S11 and S22 are, by definition, constant and matched to the filter port termination resistances (i.e., equal to 1). For impulses,

$$S_{21}(j\omega) = 1 \text{ for } [0 \le \omega \le (1-A)]$$

$$S_{21}(j\omega) = \left(\cos^2\left(\left(\frac{\pi}{4}\right)\left(\frac{\omega + A - 1}{A}\right)\right)\right)^E \text{ for } [(1-A) \le \omega \le (1+A)]$$

$$S_{21}(j\omega) = 0 \text{ for } [(1+A) \le \omega]$$

For pulses,

$$S_{21}(j\omega) = 1 \text{ for } [0 \le \omega \le (1-A)]$$

$$S_{21}(j\omega) = \frac{\left(\cos^2\left(\left(\frac{\pi}{4}\right)\left(\frac{\omega + A - 1}{A}\right)\right)\right)^E}{\operatorname{sinc}\left(\frac{\pi}{2}\omega\right)} \text{ for } [(1-A) \le \omega \le (1+A)]$$

$$S_{21}(j\omega) = 0 \text{ for } [(1+A) \le \omega]$$

where

$$\operatorname{sinc}(x) = \frac{\sin(x)}{x}$$

$$j = \sqrt{-1}$$

and a lowpass-to-highpass frequency transformation has been applied:

$$\omega = -\frac{\text{FP}}{\text{\_FREQ}}$$

_FREQ is the variable containing the project frequency, and the admittances are:

$$y_{11} = \left(\frac{1}{\text{RS}}\right)\left(\frac{-S_{21}^2(j\omega)}{S_{21}^2(j\omega) - 4}\right)$$

$$y_{22} = \left(\frac{1}{\mathrm{RL}}\right)\left(\frac{-S_{21}^2(j\omega)}{S_{21}^2(j\omega) - 4}\right)$$

$$y_{22} = \left(\frac{1}{\sqrt{\mathrm{RS} \times \mathrm{RL}}}\right)\left(\frac{2S_{21}(j\omega)}{S_{21}^2(j\omega) - 4}\right)$$

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Recommendations for Use

This model expects that the source impedance equals RS and that the load impedance equals RL, but RS need not equal RL for ideal transmission.

References

[1] Kamilo Feher, Digital Communications: Microwave Applications, (Prentice-Hall, 1981), pp. 46-51.

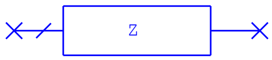
[2] John G. Proakis, Digital Communications, 2nd Ed., (McGraw-Hill, 1989), pp. 532-536.

[3] John Bellamy, Digital Telephony, 2nd Ed., (John Wiley & Sons, 1991), pp. 523-528.

[4] E. A. Lee and D. G. Messerschmitt, Digital Communications, 2nd Ed., (Kluwer Academic Publishers, 1994), pp. 188-191, 226-228.

General Impedance Element (Closed Form): IMPED

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Name	Text	Z1
R	Real part of impedance	Resistance	0 ohm
X	Imaginary part of impedance	Resistance	0 ohm

Implementation Details

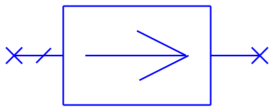
Represents an ideal impedance. R and X represent the real and imaginary parts of the impedance, respectively. The impedance is independent of frequency.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Isolator (Closed Form): ISOL8R

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Name	Text	U1
R	Port resistance	Resistance	50 ohm
LOSS	Fwd loss	DB	0 dB
ISOL	Rev loss	DB	30 dB

Implementation Details

Implements an ideal isolator with forward insertion loss given by **LOSS** and reverse isolation given by **ISOL**. Both LOSS and ISOL can be entered as either positive or negative quantities, but they are always interpreted as losses.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Load Pull model (APLAC only): LOADPULL

Symbol

Summary

LOADPULL models the nonlinear behavior of the device based on measured load-pull data.

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	LP1
NET	Name of the GMDIF document		
*INTERP	Interpolation method		Linear
*EXTRAP	Extrapolation method		Constant
*EXTRAP_WARNING	Give a warning if extrapolation is used for converged point		Yes
*K1	Gain reduction parameter for fundamental frequency gamma extrapolation		0.0
*K2	Gain reduction parameter for 2nd harmonic gamma extrapolation		0.0
*K3	Gain reduction parameter for 3rd harmonic gamma extrapolation		0.0

** indicates a secondary parameter*

Parameter Details

NET. The model is defined by the measured a- and b-waves stored in a GMDIF file.

INTERP. Specifies the method used to interpolate the data to frequencies between those specified in the file. Choices are "Linear", "Cubic Spline", "Cubic", "Floor", and "Ceiling".

EXTRAP. Specifies the method used to extrapolate the data. When set to "Interp Method", extrapolation uses the method specified by INTERP. When set to "Constant", the first data point is used for any simulation data below the range specified in the file, and the last data point is used for any simulation above it.

EXTRAP_WARNING. Specifies if the model gives a warning if the model has extrapolated when the analysis result is accepted. Based on the GMDIF file data the model monitors DC voltages or currents as well as a- and b-waves. If the model has evaluated itself so that it has extrapolated any of these, it issues a warning if that analysis point has been accepted. The warning is not given when the model extrapolates during iteration.

K1. Activates gain reduction extrapolation. The model reduces its gain if the load gamma values are exceeded (load gamma needs extrapolation). K1 defines the weight for the reduction caused by the fundamental gamma extrapolation. The gain reduction is not used by default (K1=K2=K3=0).

K2. Gain reduction parameter for weighting the reduction caused by the 2nd harmonic gamma extrapolation.

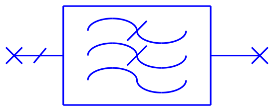
K3. Gain reduction parameter for weighting the reduction caused by the 3rd harmonic gamma extrapolation.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Butterworth Lowpass Filter: LPFB

Symbol



Summary

LPFB models represent lumped-element Butterworth lowpass filters. They offer simplicity and a compromise between high selectivity and flat group delay. The insertion loss is maximally flat at zero frequency and the stopband attenuation increases monotonically with increasing frequency at an asymptotic rate of 6N dB per octave.

Parameters

Name	Description	Unit Type	Default
ID	Filter ID	Text	LPFB1
N	Number of reactances in the filter		3
FP	Passband corner frequency (when Qu is infinite).	Frequency	1 GHz
*AP	Passband corner attenuation (when Qu is infinite).	DB	3.0103 dB
*RS	Expected source resistance.	Resistance	50 ohm
*RL	Expected load resistance	Resistance	50 ohm
*QU	Uniform unloaded Q of lowpass reactances		1e12

** indicates a secondary parameter*

Parameter Restrictions and Recommendations

1. $0 < N < 29$
2. $0 < FP$
3. $0 < AP$ Recommend AP greater than or equal to 0.001 dB.
4. $0 < RS$
5. $0 < RL$
6. $0 < QU$. Recommend QU less than or equal to 1e12.

Implementation Details

The model is implemented as a short-circuit admittance matrix, whose equivalent transfer function squared magnitude is that of a Butterworth filter:

$$|S_{21}(s)|^2 = \frac{1}{1 + (-1)^N \varepsilon^2 S^{2N}} = \frac{1}{|g(s)|^2} = \frac{1}{1 + |h(s)|^2}$$

where

$$\varepsilon^2 = 10^{\mathrm{AP}/10} - 1$$

$$s = \frac{1}{\mathrm{QU}} + j\omega$$

$$j = \sqrt{-1}$$

and a lowpass-to-lowpass frequency transformation has been applied:

$$\omega = -\frac{\_FREQ}{FP}$$

_FREQ is the variable containing the project frequency, and the admittances are:

$$y_{11} = \left(\frac{1}{\mathrm{RS}}\right)\frac{g(s) + g(-s) - h(s) - h(-s)}{g(s) - g(-s) + h(s) - h(-s)}$$

$$y_{22} = \left(\frac{1}{\mathrm{RL}}\right)\frac{g(s) + g(-s) + h(s) + h(-s)}{g(s) - g(-s) + h(s) - h(-s)}$$

$$y_{12} = y_{21} = \left(\frac{1}{\sqrt{\mathrm{RS} \times \mathrm{RL}}}\right)\frac{2}{g(s) - g(-s) + h(s) - h(-s)}$$

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

Recommendations for Use

Note that this model behaves as if it has ideal impedance transformers at its ports, so there is no attenuation due to mismatched source and load impedances. The model expects that the source impedance equals RS and that the load impedance equals RL, but RS need not equal RL for ideal transmission (as would normally be the case).

References

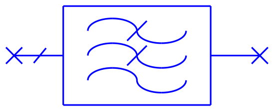
[1] Rolf Schaumann, Mohammed S. Ghausi, and Kenneth R. Laker, Design of Analog Filters: Passive, Active RC, and Switched Capacitor, (Prentice-Hall, 1990), pp. 40-44.

[2] Louis Weinberg, Network Analysis and Synthesis, (Robert E. Krieger Publishing, 1975), pp. 493-498.

[3] Adel S. Sedra and Peter O. Brackett, Filter Theory and Design: Active and Passive, (Matrix Publishers, 1978), pp. 105-111.

Chebyshev Lowpass Filter (Closed Form): LPFC

Symbol



Summary

LPFC models represent lumped-element Chebyshev lowpass filters. The insertion loss ripples between zero and a specified maximum in the passband. The stopband attenuation increases rapidly beyond the passband edge, is monotonic, and is maximally flat at infinite frequency. This filter type offers simplicity and good selectivity.

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	LPFC1
N	Number of reactive elements in the filter		3
FP	Passband corner frequency (when Qu is infinite).	Frequency	1 GHz
AP	Passband corner attenuation (when Qu is infinite).	DB	0.1 dB
*RS	Source resistance.	Resistance	50 ohm
*RL	Load resistance	Resistance	50 ohm
*QU	Average unloaded Q of reactive elements in the filter		1e12

** indicates a secondary parameter*

Parameter Restrictions and Recommendations

1. 0 < N < 27
2. 0 < FP
3. 0 < AP Recommend AP greater than or equal to 0.001 dB.
4. 0 < RS
5. 0 < RL
6. 0 < QU. Recommend QU less than or equal to 1e12.

Implementation Details

The model is implemented as a short-circuit admittance matrix, whose equivalent transfer function squared magnitude is that of a Chebyshev filter:

$$|S_{21}(s)|^2 = \frac{1}{1+\varepsilon^2 C_n^2\left(\frac{s}{j}\right)} = \frac{1}{|g(s)|^2} = \frac{1}{1+|h(s)|^2}$$

where Cn is the Chebyshev polynomial of the first kind, and

$$\varepsilon^2 = 10^{\mathrm{AP}/10} - 1$$

$$C_n(\omega) = \cos(N * \operatorname{acos}(\omega)) \qquad \text{for } 0 \le |\omega| \le 1$$

$$C_n(\omega) = \cosh(N * \operatorname{acosh}(\omega)) \qquad \text{for } 1 < |\omega|$$

$$s = \frac{1}{\mathrm{QU}} + j\omega$$

$$j = \sqrt{-1}$$

and a lowpass-to-lowpass frequency transformation has been applied:

$$\omega = -\frac{\_FREQ}{FP}$$

_FREQ is the variable containing the project frequency, and the admittances are:

$$y_{11} = \left(\frac{1}{\mathrm{RS}}\right)\frac{g(s) + g(-s) - h(s) - h(-s)}{g(s) - g(-s) + h(s) - h(-s)}$$

$$y_{22} = \left(\frac{1}{\mathrm{RL}}\right)\frac{g(s) + g(-s) + h(s) + h(-s)}{g(s) - g(-s) + h(s) - h(-s)}$$

$$y_{12} = y_{21} = \left(\frac{1}{\sqrt{\mathrm{RS} \times \mathrm{RL}}}\right)\frac{2}{g(s) - g(-s) + h(s) - h(-s)}$$

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Recommendations for Use

Note that this model behaves as if it has ideal impedance transformers at its ports, so there is no attenuation due to mismatched source and load impedances. The model expects that the source impedance equals RS and that the load impedance equals RL, but RS need not have any special relationship to RL for ideal transmission (as would normally be the case).

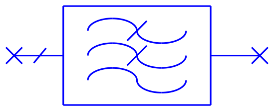
References

[1] Rolf Schaumann, Mohammed S. Ghausi, and Kenneth R. Laker, Design of Analog Filters: Passive, Active RC, and Switched Capacitor, (Prentice-Hall, 1990), pp. 44-48.

[2] Louis Weinberg, Network Analysis and Synthesis, (Robert E. Krieger Publishing, 1975), pp. 507-529.

Generalized Chebyshev Lowpass Filter (Closed Form): LPFCG

Symbol



Summary

LPFCG models represent lumped-element Generalized Chebyshev (or "Quasi-Elliptic") lowpass filters. The insertion loss ripples between zero and a specified maximum in the passband. The stopband attenuation is defined by arbitrarily specified transmission zeros. Real-frequency finite transmission zeros can be specified to improve selectivity at the expense of ultimate stopband attenuation and passband group delay, while complex-plane finite transmission zeros can be specified to provide passband group delay equalization at the expense of selectivity and ultimate stopband attenuation. Generalized Chebyshev filters represents a compromise between the simplicity of Chebyshev filters, the optimum amplitude response of more complicated Elliptic filters, and the phase linearity of Bessel and Gaussian filters. Because this type of filter allows one to make explicit design trade-offs between complexity, selectivity, and group delay equalization, it is often used to meet the demanding requirements of modern communications systems.

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	LPFCG1
N	Order of the filter		3
FP	Passband corner frequency (when Qu is infinite).	Frequency	1 GHz
*PPD	Passband parameter description:- Maximum Insertion Loss,- Minimum Return Loss, or- VSWR.	Enumerated	Maximum Insertion Loss
PPV	Passband parameter value (when Qu is infinite)	dB or Scalar	0.1 dB
TZF	Real frequency, finite transmission zeros	(Real) Frequency	{2} GHz
*TZR	Real parts of complex finite transmission zeros	(Imaginary) Frequency	{0} GHz
*RS	Source resistance	Resistance	50 ohm
*RL	Load resistance	Resistance	50 ohm
*QU	Average unloaded Q of reactive element in the filter.		0

** indicates a secondary parameter*

Parameter Details

N. In mathematical terms, N is defined as the highest exponent of the complex frequency variable s in the transfer function, $S_{21}(s)$, of the filter's normalized lowpass prototype, or, equivalently, the highest exponent of s in the transfer function of the lowpass filter. And, in terms of a measurable electrical characteristic, the number of positive-frequency passband reflection $(S_{|11}|)$ zeros corresponds to N/2 for N even and (N+1)/2 for N odd.

PPD & PPV. Parameters PPD and PPV work together to specify the characteristic of the filter's passband. PPD is used to indicate what the value of PPV represents. The flexibility these parameters provide eliminates the need to manually convert from the passband specification parameter of one's preference into whatever specific parameter the software was written to accept.

PTZF & TZR. List parameters TZF and TZR are used to specify the complex transmission zeros, Z, of the highpass filter response. Up to (N-1)/2 complex transmission zeros, $Z_i = TZR_i + jTZF_i$, can be specified. Each consists of a real part, TZR_i, and an imaginary part, the real frequency TZF_i. If TZR_i is not specified, it is assumed to be zero. And, each unspecified transmission zero is mapped to a normalized lowpass prototype frequency of infinity.

Parameter Restrictions and Recommendations

1. 32 > N > 1.
2. FP > 0.
3. If PPD = "Maximum Insertion Loss", then PPV > 0.

   If PPD = "Minimum Return Loss", then PPV > 0.

   If PPD = "Maximum VSWR", then PPV > 1.
4. $TZF_i > 0$.

   If TZR_i is specified, then TZF_i must be specified.

   If TZR_i is zero, then FP $< TZF_i$.

   If $TZR_i \neq 0$, there must be a $TZR_k = -TZR_i$ and a $TZF_k = TZF_i$.
5. RS > 0.

   RL > 0.
6. QU > 0 specifies a finite unloaded Q (recommend QU<$1e^{12}$).

   QU = 0 specifies an infinite unloaded Q.

Implementation Details

The model is implemented as a short-circuit admittance matrix,

$$\begin{bmatrix} y_{11} & y_{12} \\ y_{21} & y_{22} \end{bmatrix}$$

, whose equivalent normalized lowpass prototype transfer function, $S_{21}(s)$, is[1, 2]:

$$s_{21}^2(s) = \frac{P_N^2(s)}{P_N^2(s) + \varepsilon^2 F_N^2(s)} = \frac{P_N^2(s)}{P_N(s) + j\varepsilon F_N(s))(P_N(s) - j\varepsilon F_N(s))} = \frac{P_N^2(s)}{\varepsilon^2 E_N^2(s)}$$

where F_N and E_N are polynomials of order N, and

$$\exp(-s^2) \approx \sum_{i=0}^{N} \frac{(-1)^i s^{2i}}{i!} = |g(s)|^2 = 1 + |h(s)|^2$$

$$\varepsilon^2 = \frac{1}{10^{PPV/10} - 1} \text{when PPD = " Minimum Return Loss "}$$

$$\varepsilon^2 = \frac{0.25(\text{PPV} - 1)^2}{\text{PPV}} \text{ when PPD="Maximum VSWR"}$$

$$P_N(s) = \prod_{i=1}^{N}\left(1 - \frac{s}{z_i}\right)$$

$$s = \frac{1}{q_u} + j\omega$$

$$j = \sqrt{-1}$$

A specified lowpass transmission zero, Z[i] = TZR[i] + jTZF[i] is mapped to a normalized lowpass prototype transmission zero, z[i], using [5] [5–132]:

$$z[i] = \frac{Z[i]}{\text{FP}}$$

And, _FREQ (the variable that represents the project frequency) is mapped to the normalized lowpass prototype radian frequency, ω, using [5] [5–132]:

$$\omega = \frac{\text{\_FREQ}}{\text{FP}}$$

Polynomial

$$F_N(s) = F_i\big|_{i=N}$$

is constructed using a doubly recursive algorithm [1] [5–131][3] [5–132]:

$$G_0 = b_1$$

$$F_1 = a_1$$

$$G_{(i-1)} = a_i G_{(i-2)} + b_i F_{(i-1)}$$

$$F_i = a_i F_{(i-1)} + c_i G_{(i-2)}$$

where i = 2 to N and, employing the normalized lowpass prototype transmission zero, z_k, for k = 1 to N:

$$a_k = -j\left(s + \frac{1}{z_k}\right)$$

$$b_k = \sqrt{1 + \left(\frac{1}{z_k}\right)^2}$$

$$c_k = -b_k(s^2+1)$$

Polynomial $E_s(s)$ is found by applying the "alternating singularity principle" [1] [5–131][2] [5–131][4] [5–132] to the roots of

$$\left(\frac{P_N(s)}{\varepsilon} + \mathrm{jF}_N(s)\right)$$

. Then, E_N and F_N are split into complex-even and complex-odd polynomials [2] [5–131] such that $E_N = E_e + E_0$ and $F_N = F_e + F_0$, where

$$E_e = \mathrm{Re}(e_0) + j \quad \mathrm{Im}(e_1)s + \mathrm{Re}(e_2)s^2 + \cdots, E_o = j \quad \mathrm{Im}(e_o) + \mathrm{Re}(e_1) + j \quad \mathrm{Im}(e_2)s^2 + \cdots$$

$$F_e = \mathrm{Re}(f_0) + j\,\mathrm{Im}(f_1)s + \mathrm{Re}(f_2)s^2 + \cdots, F_o = j\,\mathrm{Im}(f_o) + \mathrm{Re}(f_1) + j\,\mathrm{Im}(f_2)s^2 \cdots$$

and and e_1 and f_1 (i = 0 to N) are the complex coefficients of E_N and F_N. Finally [3]:

$$y_{11} = \left(\frac{1}{\mathrm{RS}}\right)\left(\frac{E_e - F_e}{E_o + F_o}\right)$$

$$y_{22} = \left(\frac{1}{\mathrm{RL}}\right)\left(\frac{E_e + F_e}{E_o + F_o}\right)$$

$$y_{12} = y_{21} = \left(\frac{1}{\sqrt{\mathrm{RS} \times \mathrm{RL}}}\right)\left(\frac{-P_N/\varepsilon}{E_o + F_o}\right)$$

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Recommendations for Use

The transmission zeros can be tuned or optimized by assigning variables to the elements of the TZF and/or TZR lists and then tuning or optimizing these variables.

Note that this model behaves as if it has ideal impedance transformers at its ports, so there is no attenuation due to mismatched source and load impedances. The model expects that the source impedance equals RS and that the load impedance equals RL, but RS need not have any special relationship to RL for ideal transmission (as would normally be the case).

References

[1] Richard J. Cameron, "Fast generation of Chebyshev filter prototypes with asymmetrically-prescribed transmission zeros," ESA J., vol. 6, pp. 83-95, 1982.

[2] Richard J. Cameron, "General coupling matrix synthesis methods for Chebyshev filtering functions," IEEE Trans. Microwave Theory Tech., vol. 47, no. 4, pp. 433-442, April 1999.

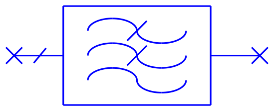
[3] Douglas R. Jachowski, unpublished notes, 1995 and 2002.

[4] J. D. Rhodes and A. S. Alseyab, "The generalized Chebyshev low pass prototype filter," Int. J. Circuit Theory Applicat., vol. 8, pp. 113-125, 1980.

[5] H. J. Blinchikoff and A. I. Zverev, Filtering in the Time and Frequency Domains, (Robert E. Krieger Publishing Co., 1987), pp. 154-155

Bessel Lowpass Filter: LPFD

Symbol



Summary

LPFD models represent lumped-element Bessel-Thomson lowpass filters. They offer simplicity and maximally flat group delay, but suffer from poor selectivity.

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	LPFD1
N	Number of reactive elements in the filter		3
FP	Passband corner frequency (when Qu is infinite).	Frequency	1 GHz
AP	Passband corner attenuation (when Qu is infinite).	DB	3.0103 dB
*RS	Expected source resistance.	Resistance	50 ohm
*RL	Expected Load resistance	Resistance	50 ohm
*QU	Average unloaded Q of reactive elements in the filter.		1e12

** indicates a secondary parameter*

Parameter Restrictions and Recommendations

1. 0 < N < 34
2. 0 < FP
3. 0 < AP Recommend AP greater than or equal to 0.001 dB.
4. 0 < RS
5. 0 < RL
6. 0 < QU. Recommend QU less than or equal to 1e12.

Implementation Details

The model is implemented as a short-circuit admittance matrix,

$$\begin{bmatrix} y_{11} & y_{12} \\ y_{21} & y_{22} \end{bmatrix}$$

, whose equivalent transfer function squared magnitude is that of a Bessel-Thomson filter:

$$|S_{21}(s)|^2 = \frac{b_0^2}{|g(s)|^2} = \frac{b_0^2}{1 + |h(s)|^2}$$

where

$$g(s) = \sum_{i=0}^{N} b_i s^i$$

$$b_i = \frac{(2N-i)!}{2^{N-i}(i)!(N-i)!} \text{ for } i = 0,\ \ldots,\ N$$

Where b_0 = (2N-1)!!,i.e, the product of all odd integers less than 2N.

$$s = \frac{1}{\mathrm{QU}} + j\omega$$

$$j = \sqrt{-1}$$

and a lowpass-to-lowpass frequency transformation has been applied:

$$\omega = \frac{\_\mathrm{FREQ}}{\mathrm{FP}}$$

_FREQ is the variable containing the project frequency, and the admittances are:

$$y_{11} = \left(\frac{1}{\mathrm{RS}}\right)\frac{g(s) + g(-s) - h(s) - h(-s)}{g(s) - g(-s) + h(s) - h(-s)}$$

$$y_{22} = \left(\frac{1}{\mathrm{RL}}\right)\frac{g(s) + g(-s) + h(s) + h(-s)}{g(s) - g(-s) + h(s) - h(-s)}$$

$$y_{12} = y_{21} = \left(\frac{1}{\sqrt{RS \times RL}}\right)\frac{(-2)b_0}{g(s) - g(-s) + h(s) - h(-s)}$$

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Recommendations for Use

Note that this model behaves as if it has ideal impedance transformers at its ports, so there is no attenuation due to mismatched source and load impedances. The model expects that the source impedance equals RS and that the load impedance equals RL, but RS need not equal RL for ideal transmission (as would normally be the case).

References

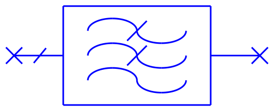
[1] Rolf Schaumann, Mohammed S. Ghausi, and Kenneth R. Laker, Design of Analog Filters: Passive, Active RC, and Switched Capacitor, (Prentice-Hall, 1990), pp. 51-56.

[2] Louis Weinberg, Network Analysis and Synthesis, (Robert E. Krieger Publishing, 1975), pp. 499-506.

[3] Herman J. Blinchikoff and Anatol I. Zverev, Filtering in the Time and Frequency Domains, (Robert E. Krieger Publishing Co., 1987), pp. 124-127.

Elliptic-Function Lowpass Filter (Closed Form): LPFE

Symbol



Summary

LPFE models represent lumped-element elliptic-function (Cauer) lowpass filters. The insertion loss ripples between zero and a specified maximum in the passband. The stopband attenuation increases rapidly between the passband edge and the stopband edge, and ripples between a specified minimum stopband attenuation and infinite attenuation. This type of filter offers optimum selectivity at the expense of increased complexity and poor group delay flatness.

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	LPFE1
N	Order of the filter.		3
FP	Passband corner frequency (when Qu is infinite).	Frequency	1 GHz
AP	Maximum Passband Insertion Loss (when Qu is infinite).	DB	0.1 dB
AS	Minimum Stopband Attenuation(when Qu is infinite).	DB	20 dB
*FS	Stopband corner frequency (when Qu is infinite).	Frequency	0 GHz
*DM	Where to put design margin:0=AP, 1=AS, 2=FP, 3=FS.		1
*RS	Source resistance.	Resistance	50 ohm
*RL	Load resistance	Resistance	50 ohm
*QU	Average unloaded Q of reactive elements in the filter.		1e12

** indicates a secondary parameter*

Parameter Details

An elliptic function lowpass prototype filter is completely specified by any four of the five parameters: N, FP, FS, AP, and AS. That is, the value of any parameter is dependent on the value of the other four parameters.

- If zero is specified for any one of these five parameters, then the model computes its value from the value of the other four parameters.
- If a value is specified for all five parameters, then the specified value of the last parameter (AS) is ignored and is computed by the model from the values of the other four parameters.
- It is an error to specify zero for more than one of these five parameters.

N. In mathematical terms, N is defined as the highest exponent of the complex frequency variable s in the transfer function, S21(s0), of the filter's normalized lowpass prototype, or, equivalently, the highest exponent of s in the transfer function of the lowpass filter. And, in terms of a measurable electrical characteristic, the number of positive-frequency passband reflection (|S11|) zeros corresponds to N/2 for N even and (N+1)/2 for N odd.

DM. If zero is specified for N, the model will determine N. But, since N must be an integer, there will typically be some design margin available, and this margin must be assigned to some parameter other than N. The value of DM determines where the model will assign this design margin.

- DM=0. After determining N, the model will recompute AP, such that $0 < \text{new AP} \le \text{AP}$
- DM=1. After determining N, the model will recompute AS, such that $\text{AS} \le \text{new AS} < \infty$
- DM=2. After determining N, the model will recompute FP, such that $\text{FS} < \text{new FP} \le \text{FP}$
- DM=3. After determining N, the model will recompute FS, such that $\text{FS} \le \text{new FS} < \text{FP}$

Parameter Restrictions and Recommendations

1. N can be zero if FP, FS, AP, and AS are not, otherwise $0 < N < 27$.
2. FP can be zero if N, FS, AP, and AS are not, otherwise $0 < \text{FP}$, and, if FS is not zero, then $\text{FS} < \text{FP}$.

   FS can be zero if N, FP, AP, and AS are not, otherwise $0 < \text{FS}$, and, if FP is not zero, then $\text{FS} < \text{FP}$.
3. AP can be zero if N, FP, FS, and AS are not, otherwise $0 < \text{AP}$, and, if AS is not zero, then $\text{AP} < \text{AS}$.

   Recommend AP greater than or equal to 0.001 dB.
4. AS can be zero if N, FP, FS, and AP are not, otherwise $\text{AP} < \text{AS}$.
5. If N is set to 0, then $0 \le \text{DM} < 4$
6. $0 < \text{RS}$.

   $0 < \text{RL}$
7. $\text{QU} > 0$ specifies a finite unloaded Q (recommend $\text{QU} \le 1\text{e}12$).

   $\text{QU} = 0$ specifies an infinite unloaded Q.
8. $0 \le \text{DM} < 4$

Implementation Details

The model is implemented as a short-circuit admittance matrix,

$$\begin{bmatrix} y_{11} & y_{12} \\ y_{21} & y_{22} \end{bmatrix}$$

, whose equivalent normalized lowpass prototype transfer function, S21(s), is:

$$|S_{21}(s)|^2 = \frac{1}{1 + \varepsilon^2 R_n^2\left(\frac{s}{j}\right)} = \frac{f(s)^2}{|g(s)|^2} = \frac{f(s)^2}{1 + |h(s)|^2}$$

where Rn is the Elliptic Rational Function, and

$$|S_{21}(s)|^2 = \frac{1}{1 + \varepsilon^2 R_n^2\left(\frac{s}{j}\right)} = \frac{f(s)^2}{|g(s)|^2} = \frac{f(s)^2}{1 + |h(s)|^2}$$

where Rn is the Elliptic Rational Function, and

$$\varepsilon^2 = 10^{\mathrm{AP}/10} - 1$$

$$R_n(\omega) = C \times \omega^{\Pi} \times \prod_{i=1}^{\frac{N-\Pi}{2}} \left(\frac{\omega^2 - \omega_i^2}{\omega^2 - \frac{\omega_s^2}{\omega_i^2}} \right)$$

$$C = \prod_{i=1}^{\frac{N-\Pi}{2}} \left(\frac{1 - \frac{\omega_s^2}{\omega_i^2}}{1 - \omega_i^2} \right)$$

$$\omega_i = \mathrm{cd}\left(\frac{2i-1}{N} K\left(\frac{1}{\omega_s^2} \right), \frac{1}{\omega_s^2} \right)$$

$$s = 1 / \left(\frac{\sqrt{\mathrm{FP2} \times \mathrm{FP1}}}{\mathrm{QU} \times \ |\ \mathrm{FP2} - \mathrm{FP1}\ |} + j\left(\frac{-1}{\omega} \right) \right)$$

$$s = \frac{1}{\mathrm{QU}} + j\omega$$

$$j = \sqrt{-1}$$

where cd is a Jacobian elliptic function, K is Legendre's complete elliptic integral of the first kind, and a lowpass-to-lowpass frequency transformation has been applied:

$$\omega = \frac{\_\mathrm{FREQ}}{\mathrm{FP}}$$

and

$$\omega_s = \frac{\mathrm{FS}}{\mathrm{FP}}$$

_FREQ is the variable containing the project frequency. The parameters of the elliptic-function filter (N, FP, FS, AP, AS) are related by "the degree equation":

$$N = \left(\frac{K(m)}{K(1-m)} \right) \left(\frac{K(1-m_o)}{K(m_o)} \right)$$

where

$$m_o = \frac{10^{\mathrm{AP}/10} - 1}{10^{\mathrm{AS}/10} - 1}$$

and

$$m = \left(\frac{\text{FP}}{\text{FS}}\right)^2$$

The admittances are given by:

$$y_{11} = \left(\frac{1}{\text{RS}}\right)\frac{g(s) + g(-s) - h(s) - h(-s)}{g(s) - g(-s) + h(s) - h(-s)}$$

$$y_{22} = \left(\frac{1}{\text{RL}}\right)\frac{g(s) + g(-s) + h(s) + h(-s)}{g(s) - g(-s) + h(s) - h(-s)}$$

$$m = \left(\frac{\text{FS}}{\text{FP}}\right)^2$$

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

Recommendations for Use

Note that this model behaves as if it has ideal impedance transformers at its ports, so there is no attenuation due to mismatched source and load impedances. The model expects that the source impedance equals RS and that the load impedance equals RL, but RS need not have any special relationship to RL for ideal transmission (as would normally be the case).

References

[1] Miroslav D. Lutovac, Dejan V. Tosic, and Brian L. Evans, Filter Design For Signal Processing Using MATLAB and Mathematica, (Prentice Hall, 2001), Chapters 6, 12, and 13.

[2] Alexander J. Grossman, "Synthesis of Tchebycheff parameter symmetrical filters," Proceedings of the IRE, pp. 545-473, April 1957.

[3] Dante Youla, "A tutorial exposition of some key network-theoretic ideas underlying classical insertion-loss filter design," Proc. IEEE, vol. 59, no. 5, pp. 760-799, May 1971.

[4] Miroslav Vleck and Rolf Unbehauen, "Degree, ripple, and transition width of elliptic filters," IEEE Trans. Circuits Syst., vol. 36, no. 3, pp. 469-472, March 1989.

[5] H. J. Orchard and Alan N. Willson, Jr., "Elliptic functions for filter design," IEEE Trans. Circuits Syst., I, vol. 44, no. 4, pp. 273-287, April 1997.

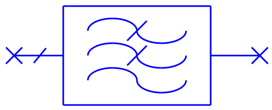
[6] Rolf Schaumann, Mohammed S. Ghausi, and Kenneth R. Laker, Design of Analog Filters: Passive, Active RC, and Switched Capacitor, (Prentice-Hall, 1990), pp. 49-51.

[7] Kendall L. Su, Handbook of Tables for Elliptic-Function Filters, (Kluwer Academic, 1990).

[8] Handbook of Mathematical Functions With Formulas, Graphs, and Mathematical Tables, edited by Milton Abramowitz and Irene A. Stegun, (U. S. National Bureau of Standards, 1964), Chapters 16 and 17 by L. M. Milne-Thomson.

Gaussian Lowpass Filter (Closed Form): LPFG

Symbol



Summary

LPFG models represent lumped-element Gaussian lowpass filters. They approximate the ideal Gaussian magnitude response and offer simplicity, relatively flat group delay, and good time domain performance, but suffer from poor frequency selectivity. Although similar to Bessel-Thomson filters, Gaussian filters offer faster rise times and lower transient overshoots, but have slightly less stopband attenuation and less group delay flatness.

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	LPFG1
N	Number of reactive elements in the filter		3
FP	Passband corner frequency (when Qu is infinite).	Frequency	1 GHz
AP	Passband corner attenuation (when Qu is infinite).	DB	3.0103 dB
*RS	Expected source resistance.	Resistance	50 ohm
*RL	Expected Load resistance	Resistance	50 ohm
*QU	Average unloaded Q of reactive elements in the filter.		1e12

** indicates a secondary parameter*

Parameter Restrictions and Recommendations

1. 0 < N < 51
2. 0 < FP
3. 0 < AP Recommend AP greater than or equal to 0.001 dB.
4. 0 < RS
5. 0 < RL
6. 0 < QU. Recommend QU less than or equal to 1e12.

Implementation Details

This model is implemented as a short-circuit admittance matrix,

$$\begin{bmatrix} y_{11} & y_{12} \\ y_{21} & y_{22} \end{bmatrix}$$

, whose equivalent transfer function squared magnitude approximates that of an ideal Gaussian filter. The ideal Gaussian squared magnitude characteristic is:

$$|S_{21}|^2 = \frac{1}{\exp(-s^2)}$$

In the model, the denominator of this ideal Gaussian characteristic is approximated by a truncated Maclaurin series:

$$\exp(-s^2) \approx \sum_{i=0}^{N} \frac{(-1)^i s^{2i}}{i!} = |g(s)|^2 = 1 + |h(s)|^2$$

where

$$s = \frac{1}{\mathrm{QU}} + j\omega$$

$$j = \sqrt{-1}$$

and a lowpass-to-lowpass frequency transformation has been applied:

$$\omega = \frac{\mathrm{\_FREQ}}{\mathrm{FP}}$$

_FREQ is the variable containing the project frequency, and the admittances are:

$$y_{11} = \left(\frac{1}{\mathrm{RS}}\right)\frac{g(s)+g(-s)-h(s)-h(-s)}{g(s)-g(-s)+h(s)-h(-s)}$$

$$y_{22} = \left(\frac{1}{\mathrm{RL}}\right)\frac{g(s)+g(-s)+h(s)+h(-s)}{g(s)-g(-s)+h(s)-h(-s)}$$

$$y_{12} = y_{21} = \left(\frac{1}{\sqrt{\mathrm{RS}\times\mathrm{RL}}}\right)\frac{(-2)}{g(s)-g(-s)+h(s)-h(-s)}$$

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Recommendations for Use

This model behaves as if it has ideal impedance transformers at its ports, so there is no attenuation due to mismatched source and load impedances. The model expects that the source impedance equals RS and that the load impedance equals RL, but RS need not equal RL for ideal transmission (as would normally be the case).

References

[1] Milton Dishal, "Gaussian-Response Filter Design," Electrical Communication, vol. 36, March 1959, pp. 3-26.

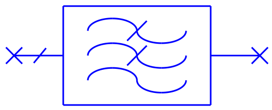
[2] Anatol I. Zverev, Handbook of Filter Synthesis, (John Wiley & Sons, 1967), pp. 67, 70, 71, 73, 74, 90, 91.

[3] DeVerl. S. Humpherys, The Analysis, Design, and Synthesis of Electrical Filters, (Prentice-Hall, 1970), pp. 413-417.

[4] Herman J. Blinchikoff and Anatol I. Zverev, Filtering in the Time and Frequency Domains, (Robert E. Krieger Publishing Co., 1987), pp. 130-132.

Raised-Cosine Lowpass Filter (Closed Form): LPFRC

Symbol



Summary

LPFRC models represent Raised-Cosine lowpass filters. Raised-Cosine filters are applied in digital transmission system theory to represent ideal (intersymbol interference free) Nyquist filtering of impulse and pulse data transmissions. Hardware designers attempt to design channel filters which approximate these characteristics and have the smallest possible bandwidth.

Parameters

Name	Description	Unit Type	Default
ID	Filter ID	Text	LPFRC1
FP	Passband corner frequency	Frequency	1 GHz
A	Roll-off factor (between 0 and 1)		0.5
TYP	Transmission type	Vector Text(pull-down)	Impulse
E	Exponent of the raised-cosine response		1.
*RS	Expected source resistance	Resistance	50 ohm
*RL	Expected load resistance	Resistance	50 ohm

** indicates a secondary parameter*

Parameter Restrictions and Recommendations

1. $0 < FP$
2. $0 \leq A \leq 1$.

   A minimum bandwidth (A=0) filter would require an infinite number of filter sections. Approximations of a 30% excess Nyquist bandwidth (A=0.3) raised-cosine filter are considered feasible with present-day technology.
3. TYP is either $\{0,1\}$.

   TYP = 0 (Impulse) specifies the ideal raised-cosine response, the theoretical filter model for infinitesimally narrow impulses. TYP = 1 (Pulse) specifies a sinc-normalized raised-cosine response, the filter model for rectangular pulse transmission.
4. $0 < E \leq 1$.

   The ideal raised-cosine response is raised to the exponent, E. For cascaded identical transmit and receive filters, you would typically specify E=0.5 for each filter. The composite channel response of the cascaded filters would be represented by specifying E=1.
5. $0 < RS$.

By definition, the model matches RS at its input port.

6. 0 < RL.

   By definition, the model matches RL at its output port.

Implementation Details

The model is implemented as a short-circuit admittance matrix,

$$\begin{bmatrix} y_{11} & y_{12} \\ y_{21} & y_{22} \end{bmatrix}$$

, whose equivalent amplitude functions S21 and S12 implement the raised-cosine response for impulse or pulse data transmission, while S11 and S22 are, by definition, constant and matched to the filter port termination resistances (i.e., equal to 1). For impulses,

$$S_{21}(j\omega) = 1 \text{ for } [0 \le \omega \le (1-A)]$$

$$S_{21}(j\omega) = \left(\cos^2\left(\left(\frac{\pi}{4}\right)\left(\frac{\omega + A - 1}{A}\right)\right)\right)^E \text{ for } [(1-A) \le \omega \le (1+A)]$$

$$S_{21}(j\omega) = 0 \text{ for } [(1+A) \le \omega]$$

For pulses,

$$S_{21}(j\omega) = 1 \text{ for } [0 \le \omega \le (1-A)]$$

$$S_{21}(j\omega) = \frac{\left(\cos^2\left(\left(\frac{\pi}{4}\right)\left(\frac{\omega + A - 1}{A}\right)\right)\right)^E}{\operatorname{sinc}\left(\frac{\pi}{2}\omega\right)} \text{ for } [(1-A) \le \omega \le (1+A)]$$

$$S_{21}(j\omega) = 0 \text{ for } [(1+A) \le \omega]$$

where

$$\operatorname{sinc}(x) = \frac{\sin(x)}{x}$$

$$j = \sqrt{-1}$$

and a lowpass-to-lowpass frequency transformation has been applied:

$$\omega = \frac{\_FREQ}{FP}$$

_FREQ is the variable containing the project frequency, and the admittances are:

$$y_{11} = \left(\frac{1}{RS}\right)\left(\frac{-S_{21}^2(j\omega)}{S_{21}^2(j\omega) - 4}\right)$$

$$y_{22} = \left(\frac{1}{\mathrm{RL}}\right)\left(\frac{-S_{21}^2(j\omega)}{S_{21}^2(j\omega)-4}\right)$$

$$y_{22} = \left(\frac{1}{\sqrt{\mathrm{RS} \times \mathrm{RL}}}\right)\left(\frac{2S_{21}(j\omega)}{S_{21}^2(j\omega)-4}\right)$$

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Recommendations for Use

This model expects that the source impedance equals RS and that the load impedance equals RL, but RS need not equal RL for ideal transmission.

References

[1] Kamilo Feher, Digital Communications: Microwave Applications, (Prentice-Hall, 1981), pp. 46-51.

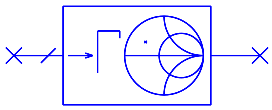
[2] John G. Proakis, Digital Communications, 2nd Ed., (McGraw-Hill, 1989), pp. 532-536.

[3] John Bellamy, Digital Telephony, 2nd Ed., (John Wiley & Sons, 1991), pp. 523-528.

[4] E. A. Lee and D. G. Messerschmitt, Digital Communications, 2nd Ed., (Kluwer Academic Publishers, 1994), pp. 188-191, 226-228.

Lossless Multi-Frequency Tuner (Closed Form): LPTUNER

Symbol



Summary

The LPTUNER element is a frequency-dependent, lossless two-port network which transforms the impedance Z_0 seen at port 2 to a user-defined impedance seen at port 1. Specifying the magnitude and angle arrays of the reflection coefficient as well as the system reference impedance sets the user-defined impedance. It is assumed that port 2 of this element is loaded in the same Z_0 as specified for the element. This element is a vectorized version of the HBTUNER or LTUNER elements and allows the specification of the reflection coefficient at as many frequencies as the user desires.

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	TU1
Mag	Array of magnitudes of gamma looking into port 1		{0.5}
Ang	Array of angles of gamma looking into port 1	Angle	{0} degs
Fo	Array of frequencies at which mag and ang are specified	Frequency	{10} GHz
Zo	System impedance	Resistance	50 ohms

Parameter Details

Mag, Ang, & Fo. These parameters are arrays which must contain the same number of elements. Array elements are enclosed in curly brackets and are comma deliminated.

Parameter Restrictions and Recommendations

1. -1 < Mag1, Mag2, Mag3 < 1. If Mag1, Mag2, or Mag3 is less than or equal to -1, it will be set to -0.99999. If Mag1, Mag2, or Mag3 is greater or equal to 1, it will be set to 0.99999.
2. Zo must be greater than or equal to 0.1 ohm
3. The Mag, Ang and Fo array's must have the same number of elements

Implementation Details

It is assumed that S22 is equal to -S11 in this implementation. This assumption fully specifies the S-Parameters of a lossless reciprocal network.

$$\mathrm{Zin}_{\mathrm{port1}} = Z_0 * \frac{1 + \mathrm{Mag}[1] * e^{j * \mathrm{Ang}[1]}}{1 - \mathrm{Mag}[1] * e^{j * \mathrm{Ang}[1]}} \quad \text{for } 1\mathrm{Hz} < f < \frac{F_0[1] + F_0[2]}{2}$$

$$\text{Zin}_{\text{port1}} = Z_0 * \frac{1 + \text{Mag}[2] * e^{j*\text{Ang}[2]}}{1 - \text{Mag}[2] * e^{j*\text{Ang}[2]}} \quad \text{for} \quad \frac{F_0[1] + F_0[2]}{2} \le f < \frac{F_0[2] + F_0[3]}{2}$$

$$\text{Zin}_{\text{port1}} = Z_0 * \frac{1 + \text{Mag}[n] * e^{j*\text{Ang}[n]}}{1 - \text{Mag}[n] * e^{j*\text{Ang}[n]}} \quad \text{for} \quad f \ge \frac{F_0[n-1] + F_0[n]}{2}$$

where f is the current simulation frequency, and n is the number of elements in the Fo array.

Layout

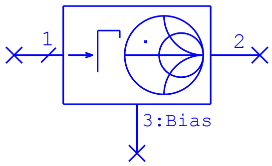
This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Recommendations for Use

NOTE: The impedance seen at port 1 of the element is a lossless transformation of an impedance Zo seen at port 2, to the specified reflection coefficent. If port 2 is not loaded with an impedance equal to Zo, the reflection seen at port 1 is not as specified.

Lossless Multi-Frequency Tuner (Closed Form): LPTUNER2

Symbol



Summary

The LPTUNER2 element is a frequency-dependent, lossless two-port network which transforms the impedance Z_0 seen at port 2 to a user-defined impedance seen at port 1. Specifying the magnitude and angle arrays of the reflection coefficient as well as the system reference impedance sets the user-defined impedance. It is assumed that port 2 of this element is loaded in the same Z_0 as specified for the element. A bias voltage can be applied to port 3 to bias a device connected to port 1, eliminating the need for a BIASTEE element. This element is a vectorized version of the HBTUNER2 or LTUNER2 elements and allows the specification of the reflection coefficient at as many frequencies as the user desires.

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	TU1
Mag	Array of magnitudes of gamma looking into port 1		{0.5}
Ang	Array of angles of gamma looking into port 1	Angle	{0} degs
Fo	Array of frequencies at which mag and ang are specified	Frequency	{10} GHz
Zo	System impedance	Resistance	50 ohms

Parameter Details

Mag, Ang, & Fo. These parameters are arrays which must contain the same number of elements. Array elements are enclosed in curly brackets and are comma deliminated.

Parameter Restrictions and Recommendations

1. -1 < Mag1, Mag2, Mag3 < 1. If Mag1, Mag2, or Mag3 is less than or equal to -1, it will be set to -0.99999. If Mag1, Mag2, or Mag3 is greater or equal to 1, it will be set to 0.99999.
2. Zo must be greater than or equal to 0.1 ohm
3. The Mag, Ang and Fo array's must have the same number of elements
4. The frequencies in the Fo array must be order sequentially

Implementation Details

It is assumed that S22 is equal to -S11 in this implementation. This assumption fully specifies the S-Parameters of a lossless reciprocal network.

$$\text{Zin}_{\text{port1}} = Z_0 * \frac{1 + \text{Mag}[1] * e^{j*\text{Ang}[1]}}{1 - \text{Mag}[1] * e^{j*\text{Ang}[1]}} \quad \text{for } 1\text{Hz} < f < \frac{F_0[1] + F_0[2]}{2}$$

$$\text{Zin}_{\text{port1}} = Z_0 * \frac{1 + \text{Mag}[2] * e^{j*\text{Ang}[2]}}{1 - \text{Mag}[2] * e^{j*\text{Ang}[2]}} \quad \text{for} \quad \frac{F_0[1] + F_0[2]}{2} \le f < \frac{F_0[2] + F_0[3]}{2}$$

$$\text{Zin}_{\text{port1}} = Z_0 * \frac{1 + \text{Mag}[n] * e^{j*\text{Ang}[n]}}{1 - \text{Mag}[n] * e^{j*\text{Ang}[n]}} \quad \text{for} \quad f \ge \frac{F_0[n-1] + F_0[n]}{2}$$

where f is the current simulation frequency, and n is the number of elements in the Fo array.

Layout

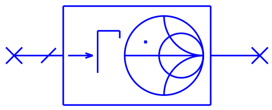
This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Recommendations for Use

NOTE: The impedance seen at port 1 of the element is a lossless transformation of an impedance Zo seen at port 2, to the specified reflection coefficient. If port 2 is not loaded with an impedance equal to Zo, the reflection seen at port 1 is not as specified.

Simple Lossless Tuner LTUNER (Closed Form): LTUNER

Symbol



Summary

The LTUNER element is a frequency-independent, lossless two-port network which transforms the impedance Z_0 seen at port 2 to a user-defined impedance seen at port 1. Specifying the magnitude and angle of the reflection coefficient as well as the system reference impedance sets the user-defined impedance. It is assumed that port 2 of this element is loaded in the same Z_0 as specified for the element.

You can use this model in load pull simulations using NI AWR's Load Pull Wizard.

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	TU1
Mag	Magnitude of gamma looking into port 1		0.5
Ang	Angle of gamma looking into port 1	Angle	0 Deg
Zo	System impedance	Resistance	50 ohms

Parameter Details

The input impedance, Z, at port 1 is calculated as:

$$Z = Z_0 * \frac{1 + \mathrm{Mag} * e^{j*\mathrm{Ang}}}{1 - \mathrm{Mag} * e^{j*\mathrm{Ang}}}$$

for f>1Hz else Z=Zo

Parameter Restrictions and Recommendations

1. -1 < Mag < 1. If Mag is less than or equal to -1, it will be set to -0.99999. If Mag is greater or equal to 1, it will be set to 0.99999.
2. Zo must be greater than or equal to 0.1 ohm.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

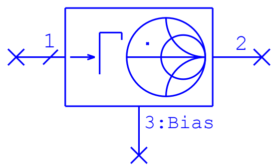
Recommendations for Use

NOTES: The impedance seen at port 1 of the element is a lossless transformation of an impedance Zo seen at port 2, to the specified reflection coefficient. If port 2 is not loaded with an impedance equal to Zo, the reflection seen at port 1 will not be as specified.

This element can be used to replace a lossless matching network. However, when comparing S-parameters to an actual realization of this matching network, the phase of S12,S21and S22 may not agree, due to the incomplete specification of the network by only defining only the reflection at port 1. A finite length of ideal transmission line (impedance = Zo) placed at port 2 of the tuner may be required to make the make S-parameters agree.

Simple Lossless Tuner with Bias Tee (Closed Form): LTUNER2

Symbol



Summary

The LTUNER2 element is a frequency-independent, lossless network which transforms the impedance Z_0 seen at port 2 to a user-defined impedance seen at port 1. Specifying the magnitude and angle of the reflection coefficient as well as the system reference impedance sets the user-defined impedance. It is assumed that port 2 of this element is loaded in the same Z_0 as specified for the element. A bias voltage can be applied to port 3 to bias a device connected to port 1, eliminating the need for a BIASTEE element.

You can use this model in load pull simulations using NI AWR's Load Pull Wizard.

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	TU1
Mag	Magnitude of gamma looking into port 1		0.5
Ang	Angle of gamma looking into port 1	Angle	0 Deg
Zo	System impedance	Resistance	50 ohms

Parameter Details

The input impedance, Z, at port 1 is calculated as:

$$Z = Z_0 * \frac{1 + \mathrm{Mag} * e^{j * \mathrm{Ang}}}{1 - \mathrm{Mag} * e^{j * \mathrm{Ang}}}$$

for f>1Hz else Bias

Parameter Restrictions and Recommendations

1. -1 < Mag < 1. If Mag is less than or equal to -1, it will be set to -0.99999. If Mag is greater or equal to 1, it will be set to 0.99999.
2. Zo must be greater than or equal to 0.1 ohm.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

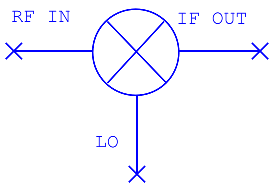
Recommendations for Use

NOTES: The impedance seen at port 1 of the element is a lossless transformation of an impedance Zo seen at port 2, to the specified reflection coefficient. If port 2 is not loaded with an impedance equal to Zo, the reflection seen at port 1 will not be as specified.

This element can be used to replace a lossless matching network. However, when comparing S-parameters to an actual realization of this matching network, the phase of S12,S21and S22 may not agree, due to the incomplete specification of the network by only defining only the reflection at port 1. A finite length of ideal transmission line (impedance = Zo) placed at port 2 of the tuner may be required to make the make S-parameters agree.

System Model for a Mixer (Closed Form): MIXER

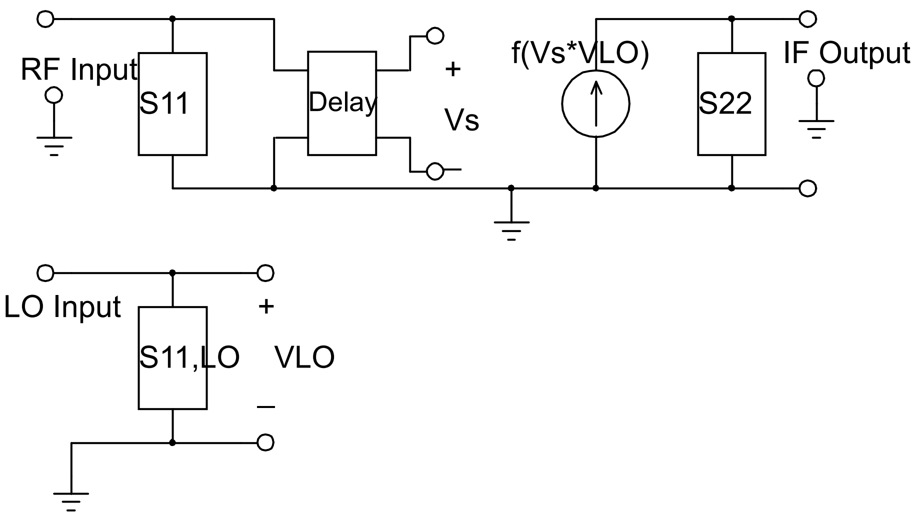
Symbol



Summary

MIXER is a generic mixer model used to represent a mixer in RF subcircuit simulation. It includes intermodulation distortion, nonzero input and output reflection coefficient, LO saturation, group delay, and port-to-port isolations.

Equivalent Circuit



Parameters

Name	Description	Unit Type	Default
ID	Element ID		MX#
GCONV	Conversion efficiency, dB, at rated LO power PLO (negative for loss)	dB	-6
P1DB_IN	1-dB compression point at input	dB units	10
IP3_IN	Third-order IP, 2f2-f1, at input; dB units	dB units	20
*RF2IF	RF to IF isolation (neg. for loss)	dB	-20
*IF2RF	IF to RF isolation (neg. for loss)	dB	-20
*LO2IF	LO to IF isolation (neg. for loss)	dB	-20

Name	Description	Unit Type	Default
*LO2RF	LO to RF isolation (neg. for loss)	dB	-20
*IF2LO	IF to LO isolation (neg. for loss)	dB	-90
*RF2LO	RF to LO isolation (neg. for loss)	dB	-90
PLO	Available LO power (dB units) for specified conversion gain	dB units	10
*S11RFMAG	RF input reflection coefficient magnitude		0.0
*S11RFANG	RF input reflection coefficient angle	Angle	0.0
*S22IFMAG	IF output reflection coefficient magnitude		0.0
*S22IFANG	IF output reflection coefficient angle	Angle	0.0
*S11LOMAG	LO input reflection coefficient magnitude		0.0
*S11LOANG	LO input reflection coefficient phase angle	Angle	0.0
*Z0	Port reference impedance	Resistance	50Ω
*TDLY	Group delay	Time	0.0
NF	Noise figure, dB	dB	0.0

** indicates a secondary parameter*

Parameter Details

DB UNITS. Units listed as dB can be in dBm or dBW, depending on your specified project units.

GCONV. The conversion loss (or gain) at the specified LO power level, GCONV, increases slightly if LO power is increased beyond the specified value. At low LO levels, the IF output is proportional to the LO level.

IP3_IN, P1DB_IN, and Isolations. These are independent of the LO level. They are modeled in the same manner as NL_AMP2.

Implementation Details

The mixer model is fundamentally a multiplier with some additional characteristics that make it behave as an RF or microwave mixer. Note that this element models mixer behavior in a very general manner, and may not precisely model any particular mixer type.

The conversion efficiency saturates much like most types of mixers. At the specified LO level, the conversion efficiency is approximately 1 dB below its maximum value. Saturation is accomplished by clipping the LO waveform; this process generates spurious mixing products involving the fundamental RF and LO harmonics. You can use output filters to adjust the levels of these products.

The IF is unfiltered, so both sum and difference frequencies are produced. You can include an IF output filter to eliminate the undesired product.

The input and output reflection coefficients are constant with frequency.

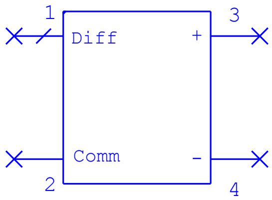
Many of the comments regarding NL_AMP2 apply to MIXER. In particular, the gain, IP3, and 1-dB compression level are those measured with the specified S_{11} and S_{22}. Thus, they do not change when S_{11} or S_{22} are modified, as long as the source and load impedance are equal to Z_0.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Mixed-Mode Converter (Closed Form): MMCONV

Symbol



Summary

MMCONV (mixed-mode converter) separates the signals at ports 3 and 4 into a common (also referred to as sum or even) mode at port 2 and a differential (also referred to as delta or odd) mode at port 1.

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	MM#

Implementation Details

In terms of voltages and currents at the ports, this model can be represented as follows:

$$V_1 = V_3 - V_4$$

$$V_2 = \frac{V_3}{2} + \frac{V_4}{2}$$

$$I_1 = \frac{I_3}{2} - \frac{I_4}{2}$$

$$I_2 = I_3 + I_4$$

or

$$V_3 = \frac{V_1}{2} + V_2$$

$$V_4 = -\frac{V_1}{2} + V_2$$

$$I_3 = I_1 + \frac{I_2}{2}$$

$$I_4 = -I_1 + \frac{I_2}{2}$$

If ports 3 and 4 are both terminated in an impedance, Z, then port 1 must be terminated in 2Z and port 2 in Z/2 in order to be matched.

Layout

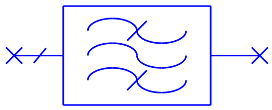
This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

Recommendations for Use

This model is commonly used for differential circuit analysis, but can also be used as an ideal power combiner/splitter (with an impedance transform) or magic-tee (with an impedance transform).

Butterworth Narrowband Bandpass Filter: NBPFB

Symbol



Summary

NBPFB models represent lumped-element Butterworth bandpass filters. The applicability of this filter type is not limited to narrow bandwidths, as the name would appear to imply. The group delay is flatter than that of a "regular" Butterworth bandpass filter of the same bandwidth, especially for wideband filters. And the passband magnitude displays arithmetic, rather than geometric, symmetry. The Butterworth characteristic offers simplicity and a compromise between high selectivity and flat group delay. The insertion loss is maximally flat at the passband's arithmetic center frequency and the stopband attenuation increases monotonically.

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	NBPFB1
N	Number of resonators in the filter		3
FP1	Lower frequency edge of passband (when Qu is infinite).	Frequency	0.5 GHz
FP2	Upper frequency edge of passband (when Qu is infinite).	Frequency	1.5 GHz
*AP	Maximum passband attenuation (when Qu is infinite).	DB	3.0103 dB
*RS	Expected source resistance.	Resistance	50 ohm
*RL	Expected load resistance	Resistance	50 ohm
*QU	Average unloaded Q of filter resonators.		1e12

** indicates a secondary parameter*

Parameter Restrictions and Recommendations

1. 0 < N < 29
2. 0 < FP1
3. 0 < FP2
4. 0 < AP Recommend AP greater than or equal to 0.001 dB.
5. 0 < RS
6. 0 < RL
7. 0 < QU. Recommend QU less than or equal to 1e12.

Implementation Details

The model is implemented as a short-circuit admittance matrix, whose equivalent transfer function squared magnitude is that of a Butterworth filter:

$$|S_{21}(s)|^2 = \frac{1}{1+(-1)^N\varepsilon^2 S^{2N}} = \frac{1}{|g(s)|^2} = \frac{1}{1+|h(s)|^2}$$

where

$$\varepsilon^2 = 10^{\mathrm{AP}/10} - 1$$

$$s = \frac{\mathrm{FP2} + \mathrm{FP1}}{2 \times \mathrm{QU} \times |\mathrm{FP2} - \mathrm{FP1}|} + j\omega$$

$$j = \sqrt{-1}$$

and a lowpass-to-narrowband-bandpass frequency transformation has been applied:

$$\omega = \frac{2 \times \_\mathrm{FREQ} - (\mathrm{FP2} + \mathrm{FP1})}{|\mathrm{FP2} - \mathrm{FP1}|}$$

This frequency transformation has good delay preserving properties for wide band filters and produces passband amplitudes with arithmetic symmetry. _FREQ is the variable containing the project frequency, and the admittances are:

$$y_{11} = \left(\frac{1}{\mathrm{RS}}\right)\frac{g(s)+g(-s)-h(s)-h(-s)}{g(s)-g(-s)+h(s)-h(-s)}$$

$$y_{22} = \left(\frac{1}{\mathrm{RL}}\right)\frac{g(s)+g(-s)+h(s)+h(-s)}{g(s)-g(-s)+h(s)-h(-s)}$$

$$y_{12} = y_{21} = \left(\frac{1}{\sqrt{\mathrm{RS} \times \mathrm{RL}}}\right)\frac{2}{g(s)-g(-s)+h(s)-h(-s)}$$

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

Recommendations for Use

Note that this model behaves as if it has ideal impedance transformers at its ports, so there is no attenuation due to mismatched source and load impedances. The model expects that the source impedance will equal RS and that the load impedance will equal RL, but RS need not equal RL for ideal transmission (as would normally be the case).

References

[1] Rolf Schaumann, Mohammed S. Ghausi, and Kenneth R. Laker, Design of Analog Filters: Passive, Active RC, and Switched Capacitor, (Prentice-Hall, 1990), pp. 40-44.

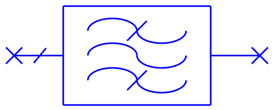
[2] Louis Weinberg, Network Analysis and Synthesis, (Robert E. Krieger Publishing, 1975), pp. 493-498.

[3] Adel S. Sedra and Peter O. Brackett, Filter Theory and Design: Active and Passive, (Matrix Publishers, 1978), pp. 105-111.

[4] H. Blinchikoff, "A note on wide-band group delay," IEEE Trans. Circuit Theory, pp. 577-578, Sept. 1971.

Chebyshev Narrowband Bandpass Filter (Closed Form): NBPFC

Symbol



Summary

NBPFC models represent lumped-element Chebyshev bandpass filters. The applicability of this filter type is not limited to narrow bandwidths, as the name would appear to imply. The group delay is flatter than that of a "regular" Chebychev bandpass filter of the same bandwidth, especially for wideband filters. And the passband magnitude displays arithmetic, rather than geometric, symmetry. The insertion loss ripples between zero and a specified maximum in the passband. The stopband attenuation increases rapidly and monotonically beyond the passband edges. This filter type offers simplicity and good selectivity.

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	NBPFC1
N	Number of resonators in the filter		3
FP1	Lower frequency edge of passband (when Qu is infinite).	Frequency	0.5 GHz
FP2	Upper frequency edge of passband (when Qu is infinite).	Frequency	1.5 GHz
AP	Maximum passband attenuation (when Qu is infinite).	DB	0.1 dB
*RS	Expected source resistance.	Resistance	50 ohm
*RL	Expected load resistance	Resistance	50 ohm
*QU	Average unloaded Q of filter resonators		1e12

** indicates a secondary parameter*

Parameter Restrictions and Recommendations

1. 0 < N < 27
2. 0 < FP1
3. 0 < FP2
4. 0 < AP Recommend AP greater than or equal to 0.001 dB.
5. 0 < RS
6. 0 < RL
7. 0 < QU. Recommend QU less than or equal to 1e12.

Implementation Details

The model is implemented as a short-circuit admittance matrix, whose equivalent transfer function squared magnitude is that of a Chebyshev filter:

$$|S_{21}(s)|^2 = \frac{1}{1+\varepsilon^2 C_n^2\left(\frac{s}{j}\right)} = \frac{1}{|g(s)|^2} = \frac{1}{1+|h(s)|^2}$$

where Cn is the Chebyshev polynomial of the first kind, and

$$\varepsilon^2 = 10^{\mathrm{AP}/10} - 1$$

$$C_n(\omega) = \cos(N * \mathrm{acos}(\omega)) \qquad \text{for } 0 \le |\omega| \le 1$$

$$C_n(\omega) = \cosh(N * \mathrm{acosh}(\omega)) \qquad \text{for } 1 < |\omega|$$

$$s = \frac{\mathrm{FP2} + \mathrm{FP1}}{2 \times \mathrm{QU} \times |\mathrm{FP2} - \mathrm{FP1}|} + j\omega$$

$$j = \sqrt{-1}$$

and a lowpass-to-narrowband-bandpass frequency transformation has been applied:

$$\omega = \frac{2 \times \_\mathrm{FREQ} - (\mathrm{FP2} + \mathrm{FP1})}{|\mathrm{FP2} - \mathrm{FP1}|}$$

This frequency transformation has good delay preserving properties for wide band filters and produces passband amplitudes with arithmetic symmetry. _FREQ is the variable containing the project frequency, and the admittances are:

$$y_{22} = \left(\frac{1}{\mathrm{RL}}\right)\frac{g(s)+g(-s)+h(s)+h(-s)}{g(s)-g(-s)+h(s)-h(-s)}$$

$$y_{12} = y_{21} = \left(\frac{1}{\sqrt{\mathrm{RS} \times \mathrm{RL}}}\right)\frac{2}{g(s)-g(-s)+h(s)-h(-s)}$$

$$y_{12} = y_{21} = \left(\frac{1}{\sqrt{RS \times RL}}\right)\frac{(-2)b_0}{g(s)-g(-s)+h(s)-h(-s)}$$

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

Recommendations for Use

Note that this model behaves as if it has ideal impedance transformers at its ports, so there is no attenuation due to mismatched source and load impedances. The model expects that the source impedance will equal RS and that the load impedance will equal RL, but RS need not have any special relationship to RL for ideal transmission (as would normally be the case).

References

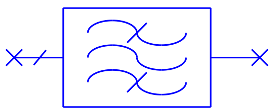
[1] Rolf Schaumann, Mohammed S. Ghausi, and Kenneth R. Laker, Design of Analog Filters: Passive, Active RC, and Switched Capacitor, (Prentice-Hall, 1990), pp. 44-48.

[2] Louis Weinberg, Network Analysis and Synthesis, (Robert E. Krieger Publishing, 1975), pp. 507-529.

[3] H. Blinchikoff, "A note on wide-band group delay," IEEE Trans. Circuit Theory, pp. 577-578, Sept. 1971.

Generalized Chebyshev Narrowband Bandpass Filter (Closed Form): NBPFCG

Symbol



Summary

NBPFCG models represent lumped-element Generalized Chebyshev (or "Quasi-Elliptic") narrowband bandpass filters. The applicability of this filter type is not limited to narrow (i.e., less than 5%) bandwidths, as the name would appear to imply. The group delay is flatter than that of a "regular" Generalized Chebyshev bandpass filter of the same bandwidth, especially for wideband filters. And the passband magnitude displays arithmetic, rather than geometric, symmetry. The insertion loss ripples between zero and a specified maximum in the passband. The stopband attenuation is defined by arbitrarily specified transmission zeros. Real-frequency finite transmission zeros can be specified to improve selectivity at the expense of ultimate stopband attenuation and passband group delay, while complex-plane finite transmission zeros can be specified to provide passband group delay equalization at the expense of selectivity and ultimate stopband attenuation. Generalized Chebyshev filters represents a compromise between the simplicity of Chebyshev filters, the optimum amplitude response of more complicated Elliptic filters, and the phase linearity of Bessel and Gaussian filters. Because this type of filter allows one to make explicit design trade-offs between complexity, selectivity, and group delay equalization, it is often used to meet the demanding requirements of modern communications systems.

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	NBPFCG1
N	Order of filter's lowpass prototype		3
FP1	Passband corner frequency (when Qu is infinite).	Frequency	0.5 GHz
FP2	Passband corner frequency (when Qu is infinite).	Frequency	1.5 GHz
*PPD	Passband parameter description:- Maximum Insertion Loss,- Minimum Return Loss, or- Maximum VSWR.	Enumerated	Maximum Insertion Loss
PPV	Passband parameter value (when Qu is infinite)	dB or Scalar	0.1 dB
TZF	Real frequency, finite transmission zeros	(Real) Frequency	{2} GHz
*TZR	Real parts of complex finite transmission zeros	(Imaginary) Frequency	{0} GHz
*RS	Source resistance	Resistance	50 ohm
*RL	Load resistance	Resistance	50 ohm
*QU	Average unloaded Q of the lumped bandpass resonators.		0

** indicates a secondary parameter*

Parameter Details

N. In mathematical terms, N is defined as the highest exponent of the complex frequency variable s in the transfer function,S_{21}(s), of the filter's normalized lowpass prototype, or, equivalently, half of the highest exponent of s in the transfer function of the bandpass filter. In terms of the number of circuit components, N corresponds to the total number of resonances in single-mode or multi-mode direct-coupled-resonator microwave filters; while, for lumped-element filters, N corresponds to the number of lumped-resonant (LC) circuits that produce zeros of attenuation at finite frequencies. And, in terms of a measurable electrical characteristic, N corresponds to the number of positive-frequency passband reflection ($S|_{11}|$) zeros.

PPD & PPV. Parameters PPD and PPV work together to specify the characteristic of the filter's passband. PPD is used to indicate what the value of PPV represents. The flexibility these parameters provide eliminates the need to manually convert from the passband specification parameter of one's preference into whatever specific parameter the software was written to accept.

TZF & TZR. List parameters TZF and TZR are used to specify the complex transmission zeros, Z, of the bandpass filter response. Of the N positive-frequency complex transmission zeros, $Z_i = TZR_i + jTZF_i$, the model allows up to (N-1) to be specified. Each consists of a real part, TZR_i, and an imaginary part, the real frequency TZF_i. If TZR_i is not specified, it is assumed to be zero. You must provide transmission zeros with nonzero real parts in pairs; that is, for each zero A+jB transmission, zero -A+jB must be present. Each unspecified transmission zero is mapped to a normalized lowpass prototype frequency of infinity. Each unspecified transmission zero is mapped to a normalized lowpass prototype frequency of infinity.

Parameter Restrictions and Recommendations

1. 32 > N > 1.
2. FP1 > 0.

   FP2 > 0.

   FP1 ≠ FP2.
3. If PPD = "Maximum Insertion Loss", then PPV > 0.

   If PPD = "Minimum Return Loss", then PPV > 0.

   If PPD = "Maximum VSWR", then PPV > 1.
4. $TZF_i > 0$.

   If TZR_i is specified, then TZF_i must be specified.

   If TZR_i is zero, then $FP1 > TZF_i > FP2$.

   If $TZR_i \neq 0$, there must be a $TZR_k = -TZR_i$ and a $TZF_k = TZF_i$.
5. RS > 0.

   RL > 0.
6. QU > 0 specifies a finite unloaded Q (recommend $QU<1e^{12}$).

   QU = 0 specifies an infinite unloaded Q.

Implementation Details

The model is implemented as a short-circuit admittance matrix,

$$\begin{bmatrix} y_{11} & y_{12} \\ y_{21} & y_{22} \end{bmatrix}$$

, whose equivalent normalized lowpass prototype transfer function, $S_{21}(s)$, is [1, 2]:

$$s_{21}^2(s) = \frac{P_N^2(s)}{P_N^2(s) + \varepsilon^2 F_N^2(s)} = \frac{P_N^2(s)}{P_N(s) + j\varepsilon F_N(s))(P_N(s) - j\varepsilon F_N(s))} = \frac{P_N^2(s)}{\varepsilon^2 E_N^2(s)}$$

where F_N and E_N are normalized lowpass prototype polynomials of order N, and

$$\exp(-s^2) \approx \sum_{i=0}^{N} \frac{(-1)^i s^{2i}}{i!} = |g(s)|^2 = 1 + |h(s)|^2$$

$$\varepsilon^2 = \frac{1}{10^{PPV/10} - 1} \text{when PPD = " Minimum Return Loss "}$$

$$\varepsilon^2 = \frac{0.25(\text{PPV} - 1)^2}{\text{PPV}} \text{ when PPD="Maximum VSWR"}$$

$$P_N(s) = \prod_{i=1}^{N}\left(1 - \frac{s}{z_i}\right)$$

$$s = \frac{1}{q_u} + j\omega$$

$$j = \sqrt{-1}$$

A specified narrowband bandpass transmission zero, Z[i] = TZR[i] + jTZF[i] is mapped to a normalized lowpass prototype transmission zero, z[i], using [5] [5–169]:

$$Z[i] = \frac{2Z[i]^2 - j(\text{FP2} + \text{FP1})}{|\ \text{FP2} - \text{FP1}\ |}$$

And, _FREQ (the variable that represents the project frequency) is mapped to the normalized lowpass prototype radian frequency,ω, using [5] [5–169]:

$$\omega = \frac{2(\text{\_FREQ}) - (\text{FP2} + \text{FP1})}{|\ \text{FP2} - \text{FP1}\ |}$$

while the specified narrowband bandpass resonator unloaded Q, QU, is converted to an equivalent lowpass prototype element unloaded Q, q_u, using [5] [5–169][6] [5–169]:

$$q_u = \text{QU}\frac{2\ |\ \text{FP2} - \text{FP1}\ |}{\text{FP2} + \text{FP1}}$$

Polynomial

$$F_N(s) = F_i|_{i=N}$$

is constructed using a doubly recursive algorithm [1] [5–169][3] [5–169]:

$$G_0 = b_1$$

$$F_1 = a_1$$

$$G_{(i-1)} = a_i G_{(i-2)} + b_i F_{(i-1)}$$

$$F_i = a_i F_{(i-1)} + c_i G_{(i-2)}$$

where i = 2 to N and, employing the normalized lowpass prototype transmission zero, z_k, for k = 1 to N:

$$a_k = -j\left(s + \frac{1}{z_k}\right)$$

$$b_k = \sqrt{1 + \left(\frac{1}{z_k}\right)^2}$$

$$c_k = -b_k\left(s^2 + 1\right)$$

And, polynomial $E_N(s)$ is found by applying the "alternating singularity principle" [1] [5–169][2] [5–169][4] [5–169] to the roots of

$$\left(\frac{P_N(s)}{\varepsilon} + \mathrm{jF}_N(s)\right)$$

. Then, E_N and F_N are split into complex-even and complex-odd polynomials [2] [5–169] such that $E_N = E_e + E_0$ and $F_N = F_e + F_0$, where

$$E_e = \mathrm{Re}(e_0) + j \quad \mathrm{Im}(e_1)s + \mathrm{Re}(e_2)s^2 + \cdots, E_o = j \quad \mathrm{Im}(e_o) + \mathrm{Re}(e_1) + j \quad \mathrm{Im}(e_2)s^2 + \cdots$$

$$F_e = \mathrm{Re}(f_0) + j\,\mathrm{Im}(f_1)s + \mathrm{Re}(f_2)s^2 + \cdots, F_o = j\,\mathrm{Im}(f_o) + \mathrm{Re}(f_1) + j\,\mathrm{Im}(f_2)s^2 \cdots$$

and e_i and f_i (i = 0 to N) are the complex coefficients of E_N and F_N. Finally [3]:

$$y_{11} = \left(\frac{1}{\mathrm{RS}}\right)\left(\frac{E_e - F_e}{E_o + F_o}\right)$$

$$y_{22} = \left(\frac{1}{\mathrm{RL}}\right)\left(\frac{E_e + F_e}{E_o + F_o}\right)$$

$$y_{12} = y_{21} = \left(\frac{1}{\sqrt{\mathrm{RS} \times \mathrm{RL}}}\right)\left(\frac{-P_N / \varepsilon}{E_o + F_o}\right)$$

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Recommendations for Use

The transmission zeros can be tuned or optimized by assigning variables to the elements of the TZF and/or TZR lists and then tuning or optimizing these variables.

Note that this model behaves as if it has ideal impedance transformers at its ports, so there is no attenuation due to mismatched source and load impedances. The model expects that the source impedance will equal RS and that the load impedance will equal RL, but RS need not have any special relationship to RL for ideal transmission (as would normally be the case).

This filter model is non-causal and not usable in transient simulations. An error message is issued if a transient simulation of a circuit containing this model is attempted. (Causality is defined as the response of a circuit *following* a stimulus--not *preceding* a stimulus. Non-causal models do not correspond to a physically realizable device.)

References

[1] Richard J. Cameron, "Fast generation of Chebyshev filter prototypes with asymmetrically-prescribed transmission zeros," ESA J., vol. 6, pp. 83-95, 1982.

[2] Richard J. Cameron, "General coupling matrix synthesis methods for Chebyshev filtering functions," IEEE Trans. Microwave Theory Tech., vol. 47, no. 4, pp. 433-442, April 1999.

[3] Douglas R. Jachowski, unpublished notes, 1995 and 2002.

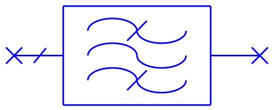
[4] J. D. Rhodes and A. S. Alseyab, "The generalized Chebyshev low pass prototype filter," Int. J. Circuit Theory Applicat., vol. 8, pp. 113-125, 1980.

[5] H. J. Blinchikoff and A. I. Zverev, Filtering in the Time and Frequency Domains, (Robert E. Krieger Publishing Co., 1987), pp. 178-186, 272.

[6] George L. Matthaei, Leo Young, and E. M. T. Jones, Microwave Filters, Impedance-Matching Networks, and Coupling Structures, (Artech House, 1980), pp. 149-156.

Bessel Narrowhead Bandpass Filter: NBPFD

Symbol



Summary

NBPFD models represent lumped-element Bessel-Thomson bandpass filters. The applicability of this filter type is not limited to narrow bandwidths, as the name would appear to imply. The group delay is flatter than that of a "regular" Bessel bandpass filter of the same bandwidth, especially for wideband filters. And the passband magnitude displays arithmetic, rather than geometric, symmetry. The Bessel-Thomson characteristic offers simplicity and maximally flat group delay, but suffers from poor selectivity.

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	NBPFD1
N	Number of resonators in the filter		3
FP1	Passband lower band-edge frequency (when Qu is infinite).	Frequency	0.5 GHz
FP2	Passband upper band-edge frequency (when Qu is infinite).	Frequency	1.5 GHz
AP	Maximum Passband corner attenuation (when Qu is infinite).	DB	3.0103 dB
*RS	Expected source resistance.	Resistance	50 ohm
*RL	Expected Load resistance	Resistance	50 ohm
*QU	Average unloaded Q of filter resonators.		1e12

** indicates a secondary parameter*

Parameter Restrictions and Recommendations

1. 0 < N < 32
2. 0 < FP1
3. 0 < FP2
4. 0 < AP Recommend AP greater than or equal to 0.001 dB.
5. 0 < RS
6. 0 < RL
7. 0 < QU. Recommend QU less than or equal to 1e12.

Implementation Details

The model is implemented as a short-circuit admittance matrix,

$$\begin{bmatrix} y_{11} & y_{12} \\ y_{21} & y_{22} \end{bmatrix}$$

, whose equivalent transfer function squared magnitude is that of a Bessel-Thomson filter:

$$|S_{21}(s)|^2 = \frac{b_0^2}{|g(s)|^2} = \frac{b_0^2}{1 + |h(s)|^2}$$

where

$$g(s) = \sum_{i=0}^{N} b_i s^i$$

$$b_i = \frac{(2N-i)!}{2^{N-i}(i)!(N-i)!} \text{ for } i = 0, \ldots, N$$

Where b_0 = (2N-1)!!,i.e, the product of all odd integers less than 2N.

$$s = \frac{\mathrm{FP2} + \mathrm{FP1}}{2 \times \mathrm{QU} \times |\mathrm{FP2} - \mathrm{FP1}|} + j\omega$$

$$j = \sqrt{-1}$$

and a lowpass-to-narrowband-bandpass frequency transformation has been applied:

$$\omega = \frac{2 \times \_\mathrm{FREQ} - (\mathrm{FP2} + \mathrm{FP1})}{|\mathrm{FP2} - \mathrm{FP1}|}$$

This frequency transformation has good delay preserving properties for wide band filters and produces passband amplitudes with arithmetic symmetry. _FREQ is the variable containing the project frequency, and the admittances are:

$$y_{11} = \left(\frac{1}{\mathrm{RS}}\right)\frac{g(s) + g(-s) - h(s) - h(-s)}{g(s) - g(-s) + h(s) - h(-s)}$$

$$y_{22} = \left(\frac{1}{\mathrm{RL}}\right)\frac{g(s) + g(-s) + h(s) + h(-s)}{g(s) - g(-s) + h(s) - h(-s)}$$

$$y_{12} = y_{21} = \left(\frac{1}{\sqrt{\mathrm{RS} \times \mathrm{RL}}}\right)\frac{2}{g(s) - g(-s) + h(s) - h(-s)}$$

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

Recommendations for Use

Note that this model behaves as if it has ideal impedance transformers at its ports, so there is no attenuation due to mismatched source and load impedances. The model expects that the source impedance will equal RS and that the load impedance will equal RL, but RS need not equal RL for ideal transmission (as would normally be the case).

References

[1] Rolf Schaumann, Mohammed S. Ghausi, and Kenneth R. Laker, Design of Analog Filters: Passive, Active RC, and Switched Capacitor, (Prentice-Hall, 1990), pp. 51-56.

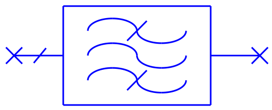
[2]Louis Weinberg, Network Analysis and Synthesis, (Robert E. Krieger Publishing, 1975), pp. 499-506.

[3] Herman J. Blinchikoff and Anatol I. Zverev, Filtering in the Time and Frequency Domains, (Robert E. Krieger Publishing Co., 1987), pp. 124-127.

[4] H. Blinchikoff, "A note on wide-band group delay," IEEE Trans. Circuit Theory, pp. 577-578, Sept. 1971.

Elliptic-Function Narrowband Bandpass Filter (Closed Form): NBPFE

Symbol



Summary

NBPFE models represent lumped-element elliptic-function (Cauer) bandpass filters. The applicability of this filter type is not limited to narrow bandwidths, as the name would appear to imply. The group delay is flatter than that of a "regular" elliptic-function bandpass filter of the same bandwidth, especially for wideband filters. And the passband magnitude displays arithmetic, rather than geometric, symmetry. The insertion loss ripples between zero and a specified maximum in the passband. The stopband attenuation increases rapidly between the passband edges and the stopband edges, and ripples between a specified minimum stopband attenuation and infinity. This type of filter offers optimum selectivity at the expense of increased complexity and poor group delay flatness.

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	NBPFE1
N	Order of filter's lowpass prototype.		3
FP1	Passband corner frequency (when Qu is infinite).	Frequency	0.5 GHz
FP2	Passband corner frequency (when Qu is infinite).	Frequency	1.5 GHz
AP	Maximum Passband Insertion Loss (when Qu is infinite).	DB	0.1 dB
AS	Minimum Stopband Attenuation (when Qu is infinite).	DB	20 dB
*FS1	Stopband corner frequency (when Qu is infinite).	Frequency	0 GHz
*FS2	Stopband corner frequency (when Qu is infinite).	Frequency	0 GHz
*DM	Where to put design margin (see below for details).		1
*RS	Source resistance.	Resistance	50 ohm
*RL	Load resistance	Resistance	50 ohm
*QU	Average unloaded Q of the bandpass resonators.		0

** indicates a secondary parameter*

Parameter Details

An elliptic-function narrowband bandpass filter is completely specified by any three of the following four parameters:

- lowpass prototype filter order, N,
- maximum passband insertion loss, AP,
- minimum stopband attenuation, AS, and
- selectivity.

Filter selectivity, in turn, is completely specified by any three of the following four parameters:

- frequency of the upper edge of the lower stopband, FS1,
- frequency of the lower edge of the passband, FP1,
- frequency of the upper edge of the passband, FP2, and
- frequency of the lower edge of the upper stopband, FS2.

There are eighteen valid sets of these parameters, and these are listed in the table below. Columns 2 through 9 represent individual parameters of the filter, while the rows represent the valid parameter sets - which are numbered in column 1. The last column indicates which parameter is unspecified, or is incompletely specified, in the set. For each specific parameter, the body of the table indicates whether that parameter is specified, unspecified, or ignored in a particular parameter set. "S" means the parameter value is specified (non-zero), "0" means it has been set to zero (indicating it is intentionally unspecified), and "x" means it is ignored by the model.

Set	N	AP	AS	FP1	FP2	FS1	FS2	DM	Incompletely Specified Parameter
1	S	S	S	S	S	x	x	x	Selectivity
2	S	S	S	S	0	S	x	x	Selectivity
3	S	S	S	S	0	0	S	x	Selectivity
4	S	S	S	0	S	S	x	x	Selectivity
5	S	S	S	0	S	0	S	x	Selectivity
6	S	S	S	0	0	S	S	x	Selectivity
7	S	S	0	S	S	S	x	x	AS
8	S	S	0	S	S	0	S	x	AS
9	S	S	0	S	0	S	S	x	AS
10	S	S	0	0	S	S	S	x	AS
11	S	0	S	S	S	S	x	x	AP
12	S	0	S	S	S	0	S	x	AP
13	S	0	S	S	0	S	S	x	AP
14	S	0	S	0	S	S	S	x	AP
15	0	S	S	S	S	S	x	S	N
16	0	S	S	S	S	0	S	S	N
17	0	S	S	S	0	S	S	S	N
18	0	S	S	0	S	S	S	S	N

N. In mathematical terms, N is defined as the highest exponent of the complex frequency variable s in the transfer function, S21(s), of the filter's normalized lowpass prototype, or, equivalently, half of the highest exponent of s in the transfer function of the narrowband bandpass filter. In terms of the number of circuit components, N corresponds to the total number of resonances in single-mode or multi-mode direct-coupled-resonator microwave filters; while, for lumped-element filters, N corresponds to the number of lumped-resonant (LC) circuits that produce zeros of attenuation at finite frequencies. And, in terms of a measurable electrical characteristic, N corresponds to the number of positive-frequency passband reflection (|S11|) zeros.

DM. If zero is specified for N, the model will determine N. But, since N must be an integer, there will typically be some design margin available, and this margin must be assigned to some parameter other than N. Consequently, after determining

N, the model will compute a new value for one or more of the other parameters. The value of DM determines which parameter, or parameters, will have their values recomputed after a value is synthesized for N. For each valid value of DM, the table below specifies which parameters (X) are effected by the additional design margin and the manner in which they are effected.

DM	AP	AS	FP1	FP2	FS1	FS2	Description
0	X						decrease AP
1		X					increase AS
2			X	X			Decrease FP1 Increase FP2
3			X		X		Decrease FS1 & FP1
4			X			X	Decrease FP1 & FS2
5				X	X		Increase FS1 & FP2
6				X		X	Increase FP2 & FS2
7					X	X	Increase FS1 Decrease FS2
8			X	X	X	X	Increase FS1 & FP2 Decrease FP1 & FS2

Parameter Restrictions and Recommendations

1. If any of the parameters N, AP, AS, FP1, FP2, FS1, or FS2 are set to 0, the model considers them "unspecified." A value of zero for any of these parameters may, or may not, be valid, depending on whether the set of parameters as a whole corresponds to one of the valid parameter sets listed in the table, above.
2. If N is specified, it must fall within the range $0 < N < 27$.

   If N=0, then $0 \leq DM \leq 8$, otherwise DM is ignored.
3. If Ap and AS are both specified, $0 < AP < AS$, otherwise $0 < AP$ or $0 < AS$.

   Recommend AP greater than or equal to 0.001 dB.
4. "Selectivity" is fully specified by any three of FS1, FP1, FP2, and FS2.

   If FS1, FP1 & FP2 are specified, $0 < FS1 < FP1 < FP2$.

   If FS1, FP1 & FS2 are specified, $0 < FS1 < FP1 < FS2$.

   If FS1, FP2 & FS2 are specified, $0 < FS1 < FP2 < FS2$.

   If FP1, FP2 & FS2 are specified, $0 < FP1 < FP2 < FS2$.
5. "Selectivity" is partially specified by any two of FS1, FP1, FP2, and FS2.

If FS1 & FP1 are specified, 0 < FS1 < FP1.

If FS1 & FP2 are specified, 0 < FS1 < FP2.

If FS1 & FS2 are specified, 0 < FS1 < FS2.

If FP1 & FP2 are specified, 0 < FP1 < FP2.

If FP1 & FS2 are specified, 0 < FP1 < FS2.

If FP2 & FS2 are specified, 0 < FP2 < FS2.

6. 0 < RS.

   0 < RL.

7. QU > 0 specifies a finite unloaded Q (recommend QU ≤ 1e12).

   QU = 0 specifies an infinite unloaded Q.

Implementation Details

The model is implemented as a short-circuit admittance matrix,

$$\begin{bmatrix} y_{11} & y_{12} \\ y_{21} & y_{22} \end{bmatrix}$$

, whose equivalent normalized lowpass prototype transfer function, S21(s), is:

$$|S_{21}(s)|^2 = \frac{1}{1 + \varepsilon^2 R_n^2\left(\frac{s}{j}\right)} = \frac{f(s)^2}{|g(s)|^2} = \frac{f(s)^2}{1 + |h(s)|^2}$$

where Rn is the Elliptic Rational Function, and

$$\varepsilon^2 = 10^{\mathrm{AP}/10} - 1$$

$$R_n(\omega) = C \times \omega^{\Pi} \times \prod_{i=1}^{\frac{N-\Pi}{2}} \left(\frac{\omega^2 - \omega_i^2}{\omega^2 - \frac{\omega_s^2}{\omega_i^2}} \right)$$

$$C = \prod_{i=1}^{\frac{N-\Pi}{2}} \left(\frac{1 - \frac{\omega_s^2}{\omega_i^2}}{1 - \omega_i^2} \right)$$

$$\omega_i = \mathrm{cd}\left(\frac{2i-1}{N} K\left(\frac{1}{\omega_s^2} \right), \frac{1}{\omega_s^2} \right)$$

$$s = \frac{\mathrm{FP2} + \mathrm{FP1}}{2 \times \mathrm{QU} \times |\mathrm{FP2} - \mathrm{FP1}|} + j\omega$$

$$\Pi = 0 \text{ if N is even}, \Pi = 1 \text{ if N is odd}$$

$$j = \sqrt{-1}$$

where cd is a Jacobian elliptic function, K is Legendre's complete elliptic integral of the first kind, and a narrowband-bandpass-to-lowpass frequency transformation has been applied:

$$\omega = \frac{2 \times \text{\_FREQ} - (\text{FP2} + \text{FP1})}{|\ \text{FP2} - \text{FP1}\ |}$$

and

$$\omega_s = \frac{2 \times \text{FS2} - (\text{FP2} + \text{FP1})}{|\ \text{FP2} - \text{FP1}\ |}$$

This frequency transformation has good delay preserving properties for wide band filters and produces passband amplitudes with arithmetic symmetry. Note that _FREQ is the variable containing the project frequency, and the frequency parameters FP1, FP2, FS1, and FS2 are related according to: FP1xFP2=FS1xFS2

The seven parameters of the elliptic-function bandpass filter (N, AP, AS, FP1, FP2, FS1, and FS2) are related by "the degree equation":

$$N = \left(\frac{K(m)}{K(1-m)}\right)\left(\frac{K(1-m_o)}{K(m_o)}\right)$$

where

$$m_o = \frac{10^{\text{AP}/10} - 1}{10^{\text{AS}/10} - 1}$$

and

$$m = \left| \frac{|\ \text{FP2} - \text{FP1}}{2 \times \text{FS2} - (\text{FP2} + \text{FP1})} \right|^2$$

The admittances are given by:

$$y_{11} = \left(\frac{1}{\text{RS}}\right)\frac{g(s) + g(-s) - h(s) - h(-s)}{g(s) - g(-s) + h(s) - h(-s)}$$

$$y_{22} = \left(\frac{1}{\text{RL}}\right)\frac{g(s) + g(-s) + h(s) + h(-s)}{g(s) - g(-s) + h(s) - h(-s)}$$

$$y_{12} = y_{21} = \left(\frac{1}{\sqrt{\text{RS} \times \text{RL}}}\right)\frac{2 \times f(s)}{g(s) - g(-s) + h(s) - h(-s)}$$

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Recommendations for Use

Note that this model behaves as if it has ideal impedance transformers at its ports, so there is no attenuation due to mismatched source and load impedances. The model expects that the source impedance will equal RS and that the load impedance will equal RL, but RS need not have any special relationship to RL for ideal transmission (as would normally be the case).

References

[1] Miroslav D. Lutovac, Dejan V. Tosic, and Brian L. Evans, Filter Design For Signal Processing Using MATLAB and Mathematica, (Prentice Hall, 2001), Chapters 6, 12, and 13.

[2] Alexander J. Grossman, "Synthesis of Tchebycheff parameter symmetrical filters," Proceedings of the IRE, pp. 545-473, April 1957.

[3] Dante Youla, "A tutorial exposition of some key network-theoretic ideas underlying classical insertion-loss filter design," Proc. IEEE, vol. 59, no. 5, pp. 760-799, May 1971.

[4] Miroslav Vleck and Rolf Unbehauen, "Degree, ripple, and transition width of elliptic filters," IEEE Trans. Circuits Syst., vol. 36, no. 3, pp. 469-472, March 1989.

[5] H. J. Orchard and Alan N. Willson, Jr., "Elliptic functions for filter design," IEEE Trans. Circuits Syst., I, vol. 44, no. 4, pp. 273-287, April 1997.

[6] Rolf Schaumann, Mohammed S. Ghausi, and Kenneth R. Laker, Design of Analog Filters: Passive, Active RC, and Switched Capacitor, (Prentice-Hall, 1990), pp. 49-51.

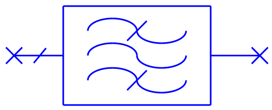
[7] Kendall L. Su, Handbook of Tables for Elliptic-Function Filters, (Kluwer Academic, 1990).

[8] Handbook of Mathematical Functions With Formulas, Graphs, and Mathematical Tables, edited by Milton Abramowitz and Irene A. Stegun, (U. S. National Bureau of Standards, 1964), Chapters 16 and 17 by L. M. Milne-Thomson.

[9] H. Blinchikoff, "A note on wide-band group delay," IEEE Trans. Circuit Theory, pp. 577-578, Sept. 1971.

Gaussian Narrowband Bandpass Filter (Closed Form): NBPFG

Symbol



Summary

NBPFG models represent lumped-element Gaussian bandpass filters. The applicability of this filter type is not limited to narrow bandwidths, as the name seems to imply. The group delay is flatter than that of a "regular" Gaussian bandpass filter of the same bandwidth, especially for wideband filters. Also, the passband magnitude displays arithmetic, rather than geometric, symmetry. NBPFG models approximate the ideal Gaussian magnitude response and offer simplicity, relatively flat group delay, and good time domain performance, but suffer from poor frequency selectivity. Although similar to Bessel-Thomson filters, Gaussian filters offer faster rise times and lower transient overshoots, but have slightly less stopband attenuation and less group delay flatness.

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	NBPFG1
N	Number of resonators in the filter		3
FP1	Passband lower band-edge frequency (when Qu is infinite).	Frequency	0.5 GHz
FP2	Passband upper band-edge frequency (when Qu is infinite).	Frequency	1.5 GHz
AP	Maximum Passband corner attenuation (when Qu is infinite).	DB	3.0103 dB
*RS	Expected source resistance.	Resistance	50 ohm
*RL	Expected Load resistance	Resistance	50 ohm
*QU	Average unloaded Q of filter resonators		1e12

** indicates a secondary parameter*

Parameter Restrictions and Recommendations

1. 0 < N < 51
2. 0 < FP1
3. 0 < FP2
4. 0 < AP Recommend AP greater than or equal to 0.001 dB.
5. 0 < RS
6. 0 < RL
7. 0 < QU. Recommend QU less than or equal to 1e12.

Implementation Details

This model is implemented as a short-circuit admittance matrix,

$$\begin{bmatrix} y_{11} & y_{12} \\ y_{21} & y_{22} \end{bmatrix}$$

, whose equivalent transfer function squared magnitude is that of an ideal Gaussian filter. The ideal Gaussian squared magnitude characteristic is:

$$|S_{21}|^2 = \frac{1}{\exp(-s^2)}$$

In the model, the denominator of this ideal Gaussian characteristic is approximated by a truncated Maclaurin series:

$$\exp(-s^2) \approx \sum_{i=0}^{N} \frac{(-1)^i s^{2i}}{i!} = |g(s)|^2 = 1 + |h(s)|^2$$

where

$$s = \frac{\mathrm{FP2} + \mathrm{FP1}}{2 \times \mathrm{QU} \times |\mathrm{FP2} - \mathrm{FP1}|} + j\omega$$

$$j = \sqrt{-1}$$

and a lowpass-to-narrowband-bandpass frequency transformation has been applied:

$$\omega = \frac{2 \times \_\mathrm{FREQ} - (\mathrm{FP2} + \mathrm{FP1})}{|\mathrm{FP2} - \mathrm{FP1}|}$$

This frequency transformation has good delay preserving properties for wide band filters and produces passband amplitudes with arithmetic symmetry. _FREQ is the variable containing the project frequency, and the admittances are:

$$y_{11} = \left(\frac{1}{\mathrm{RS}}\right)\frac{g(s) + g(-s) - h(s) - h(-s)}{g(s) - g(-s) + h(s) - h(-s)}$$

$$y_{22} = \left(\frac{1}{\mathrm{RL}}\right)\frac{g(s) + g(-s) + h(s) + h(-s)}{g(s) - g(-s) + h(s) - h(-s)}$$

$$y_{12} = y_{21} = \left(\frac{1}{\sqrt{\mathrm{RS} \times \mathrm{RL}}}\right)\frac{(-2)}{g(s) - g(-s) + h(s) - h(-s)}$$

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Recommendations for Use

This model behaves as if it has ideal impedance transformers at its ports, so there is no attenuation due to mismatched source and load impedances. The model expects that the source impedance will equal RS and that the load impedance will equal RL, but RS need not equal RL for ideal transmission (as would normally be the case).

This filter model is non-causal and not usable in transient simulations. An error message is issued if a transient simulation of a circuit containing this model is attempted. (Causality is defined as the response of a circuit *following* a stimulus--not *preceding* a stimulus. Non-causal models do not correspond to a physically realizable device.)

References

[1] Milton Dishal, "Gaussian-Response Filter Design," Electrical Communication, vol. 36, March 1959, pp. 3-26.

[2] Anatol I. Zverev, Handbook of Filter Synthesis, (John Wiley & Sons, 1967), pp. 67, 70, 71, 73, 74, 90, 91.

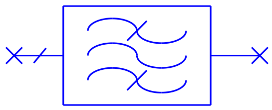
[3] DeVerl. S. Humpherys, The Analysis, Design, and Synthesis of Electrical Filters, (Prentice-Hall, 1970), pp. 413-417.

[4] Herman J. Blinchikoff and Anatol I. Zverev, Filtering in the Time and Frequency Domains, (Robert E. Krieger Publishing Co., 1987), pp. 130-132.

[5] H. Blinchikoff, "A note on wide-band group delay," IEEE Trans. Circuit Theory, pp. 577-578, Sept. 1971.

Raised-Cosine Narrowband Bandpass Filter (Closed Form): NBPFRC

Symbol



Summary

NBPFRC models represent Raised-Cosine Bandpass filters. The applicability of this filter type is not limited to narrow bandwidths, as the name seems to imply. The passband magnitude displays arithmetic, rather than geometric, symmetry. Raised-Cosine filters are applied in digital transmission system theory to represent ideal (intersymbol interference free) Nyquist filtering of impulse and pulse data transmissions. Hardware designers attempt to design channel filters which approximate these characteristics and have the smallest possible bandwidth.

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	NBPFRC1
FP1	Lower frequency edge of passband	Frequency	0.5 GHz
FP2	Upper frequency edge of passband	Frequency	1.5 GHz
A	Roll-off factor (between 0 and 1)		0.5
TYP	Transmission type	Vector Text(pull-down)	Impulse
E	Exponent of the raised-cosine response		1.
*RL	Expected source resistance	Resistance	50 ohm
*RL	Expected load resistance	Resistance	50 ohm

** indicates a secondary parameter*

Parameter Restrictions and Recommendations

1. 0 < FP1
2. 0 < FP2
3. $0 \leq A \leq 1$.

   A minimum bandwidth (A=0) filter would require an infinite number of filter sections. Approximations of a 30% excess Nyquist bandwidth (A=0.3) raised-cosine filter are considered feasible with present-day technology.
4. TYP is either {0,1} .

   TYP = 0 (Impulse) specifies the ideal raised-cosine response, the theoretical filter model for infinitesimally narrow impulses. TYP = 1 (Pulse) specifies a sinc-normalized raised-cosine response, the filter model for rectangular pulse transmission.
5. $0<E\leq 1$.

The ideal raised-cosine response is raised to the exponent, E. For cascaded identical transmit and receive filters, you would typically specify E=0.5 for each filter. The composite channel response of the cascaded filters would be represented by specifying E=1.

6. 0 < RS.

   By definition, the model matches RS at its input port.

7. 0 < RL.

   By definition, the model matches RL at its output port.

Implementation Details

The model is implemented as a short-circuit admittance matrix,

$$\begin{bmatrix} y_{11} & y_{12} \\ y_{21} & y_{22} \end{bmatrix}$$

, whose equivalent amplitude functions S21 and S12 implement the raised-cosine response for impulse or pulse data transmission, while S11 and S22 are, by definition, constant and matched to the filter port termination resistances (i.e., equal to 1). For impulses,

$$S_{21}(j\omega) = 1 \text{ for } [0 \le \omega \le (1 - A)]$$

$$S_{21}(j\omega) = \left(\cos^2\left(\left(\frac{\pi}{4}\right)\left(\frac{\omega + A - 1}{A}\right)\right)\right)^E \text{ for } [(1 - A) \le \omega \le (1 + A)]$$

$$S_{21}(j\omega) = 0 \text{ for } [(1 + A) \le \omega]$$

For pulses,

$$S_{21}(j\omega) = 1 \text{ for } [0 \le \omega \le (1 - A)]$$

$$S_{21}(j\omega) = \frac{\left(\cos^2\left(\left(\frac{\pi}{4}\right)\left(\frac{\omega + A - 1}{A}\right)\right)\right)^E}{\mathrm{sinc}\left(\frac{\pi}{2}\omega\right)} \text{ for } [(1 - A) \le \omega \le (1 + A)]$$

$$S_{21}(j\omega) = 0 \text{ for } [(1 + A) \le \omega]$$

where

$$\mathrm{sinc}(x) = \frac{\sin(x)}{x}$$

$$j = \sqrt{-1}$$

and a lowpass-to-narrowband-bandpass frequency transformation has been applied:

$$\omega = \frac{2 \times \_\mathrm{FREQ} - (\mathrm{FP2} + \mathrm{FP1})}{|\ \mathrm{FP2} - \mathrm{FP1}\ |}$$

This frequency transformation produces passband amplitudes with arithmetic symmetry. _FREQ is the variable containing the project frequency, and the admittances are:

$$y_{11} = \left(\frac{1}{\mathrm{RS}}\right)\left(\frac{-S_{21}^2(j\omega)}{S_{21}^2(j\omega)-4}\right)$$

$$y_{22} = \left(\frac{1}{\mathrm{RL}}\right)\left(\frac{-S_{21}^2(j\omega)}{S_{21}^2(j\omega)-4}\right)$$

$$y_{22} = \left(\frac{1}{\sqrt{\mathrm{RS} \times \mathrm{RL}}}\right)\left(\frac{2S_{21}(j\omega)}{S_{21}^2(j\omega)-4}\right)$$

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Recommendations for Use

This model expects that the source impedance will equal RS and that the load impedance will equal RL, but RS need not equal RL for ideal transmission.

This filter model is non-causal and not usable in transient simulations. An error message is issued if a transient simulation of a circuit containing this model is attempted. (Causality is defined as the response of a circuit *following* a stimulus--not *preceding* a stimulus. Non-causal models do not correspond to a physically realizable device.)

References

[1] Kamilo Feher, Digital Communications: Microwave Applications, (Prentice-Hall, 1981), pp. 46-51.

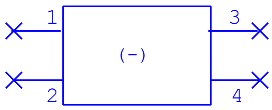
[2] John G. Proakis, Digital Communications, 2nd Ed., (McGraw-Hill, 1989), pp. 532-536.

[3] John Bellamy, Digital Telephony, 2nd Ed., (John Wiley & Sons, 1991), pp. 523-528.

[4] E. A. Lee and D. G. Messerschmitt, Digital Communications, 2nd Ed., (Kluwer Academic Publishers, 1994), pp. 188-191, 226-228.

N Port Negation Element (Closed Form): NEG

Symbol



Summary

This element replaces the obsolete NEG2 and NEG4 elements.

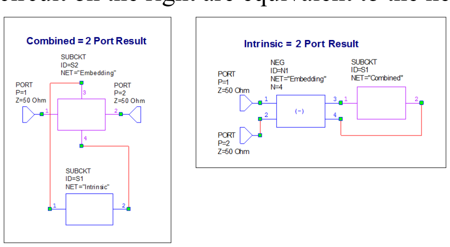
Parameters

Name	Description	Unit Type	Default
ID	Name	Text	N1
NET	Network used for negation	Text	"name"
N	Number of ports in network (even)		4

Implementation Details

NET specifies the name of an N-port embedding network whose electrical characteristics are to de-embed the effects of the embedding network. When NEG is connected in series with a bilateral network the result is a short circuit. When connected in parallel with a symmetric network the result is an open circuit. A simple example illustrates the series use of the model.

In the following figure the circuit on the left shows a network constructed by connecting an Embedding network with an Intrinsic network. This connected circuit is named “Combined”. The circuit on the right shows how to recover the Intrinsic network parameters from the Combined circuit using the Embedding network. The network parameters for the circuit on the right are equivalent to the network parameters for the Intrinsic network.

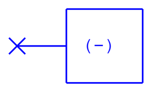


Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

1 Port Negation Element (Closed Form): NEG1

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Name	Text	N1
NET	Network used for negation	Text	name

Implementation Details

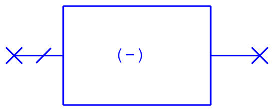
NET specifies the name of a 1-port network whose electrical characteristics are to be negated. Used for deembedding the network from a circuit. When connected in series with the network the result is a short circuit. When connected in parallel with the network the result is an open circuit.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

(Obsolete) 2-Port Negation Element (Closed Form): NEG2

Symbol



Summary

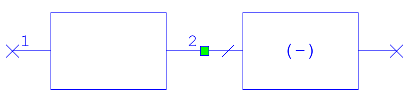
This element is OBSOLETE and is replaced by the N Port Negation Element (Closed Form) NEG element.

Parameters

Name	Description	Unit Type	Default
ID	Name	Text	N1
NET	Network used for negation	Text	name

Implementation Details

NET specifies the name of a 2-port, bilateral network whose electrical characteristics are to be negated. It is used for de-embedding the network from a circuit. When connected in series with the network, the result is a short circuit. When connected in parallel with the network, the result is an open circuit (provided that the network is symmetric). The following figure shows NEG2 connected in series with the network.



The following equations show the admittance for the network in series with NEG2:

$$
\begin{aligned}
\Delta &= Y_{22} + Y_{11}^{NEG2} \\
Y_{11}^{total} &= Y_{11} - \frac{Y_{12}Y_{21}}{\Delta} \\
Y_{22}^{total} &= Y_{22}^{NEG2} - \frac{Y_{12}^{NEG2}Y_{21}^{NEG2}}{\Delta} \\
Y_{12}^{total} &= -\frac{Y_{12}Y_{12}^{NEG2}}{\Delta} \\
Y_{21}^{total} &= -\frac{Y_{21}Y_{21}^{NEG2}}{\Delta}
\end{aligned}
\tag{1}
$$

By definition for NEG2 you have:

$$\begin{aligned} Y_{11}^{NEG2} &= -Y_{22} \\ Y_{22}^{NEG2} &= -Y_{11} \\ Y_{12}^{NEG2} &= -Y_{12} \\ Y_{21}^{NEG2} &= -Y_{21} \end{aligned} \tag{2}$$

Substituting equation (2) to (1) (Δ is actually zero but the circuit solver sets delta to a small value):

$$\begin{aligned} \Delta &= 10^{-9} \\ Y_{11}^{total} &= Y_{11} - \frac{Y_{12}Y_{21}}{\Delta} \\ Y_{22}^{total} &= -Y_{11} - \frac{Y_{12}Y_{21}}{\Delta} \\ Y_{12}^{total} &= \frac{(Y_{12})^2}{\Delta} \\ Y_{21}^{total} &= \frac{(Y_{21})^2}{\Delta} \end{aligned} \tag{3}$$

Note that in this case 1/Δ aims to infinity, so you can ignore the Y11 terms. Also, because we require the network to be reciprocal

$$Y_{12} = Y_{21} \tag{4}$$

and if you define

$$Y = -\frac{(Y_{12})^2}{\Delta} \tag{5}$$

the equations become:

$$\begin{aligned} Y_{11}^{total} &= Y \\ Y_{22}^{total} &= Y \\ Y_{12}^{total} &= -Y \\ Y_{21}^{total} &= -Y \end{aligned} \tag{6}$$

You can see that (6) describes the Y-matrix of a small series resistor, almost a short, with only the requirement that the network to be negated by NEG2 must be reciprocal.

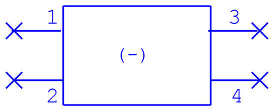
Note that simulating only the NEG2 block in series with the network you want to negate represents a short circuit. This is similar to simply connecting two ports together with a wire and simulating. This setup can cause a problem with Y-matrix calculations. If you want to analyze just the NEG2 block in series with the network you want to negate, you should add a 0 ohm resistor before the NEG2 block.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

(Obsolete) 4 Port Negation Element (Closed Form): NEG4

Symbol



Summary

This element is OBSOLETE and is replaced by the N Port Negation Element (Closed Form) NEG element.

Parameters

Name	Description	Unit Type	Default
ID	Name	Text	N1
NET	Network used for negation	Text	name

Implementation Details

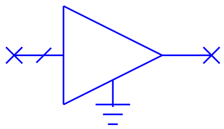
NET specifies the name of a 4-port, bilateral network whose electrical characteristics are to be negated. It is used for de-embedding the network from a circuit. When connected in series with the network, the result is a short circuit. When connected in parallel with the network, the result is an open circuit (provided that the network is symmetric).

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Nonlinear Amplifier System Model (Closed Form): NL_AMP

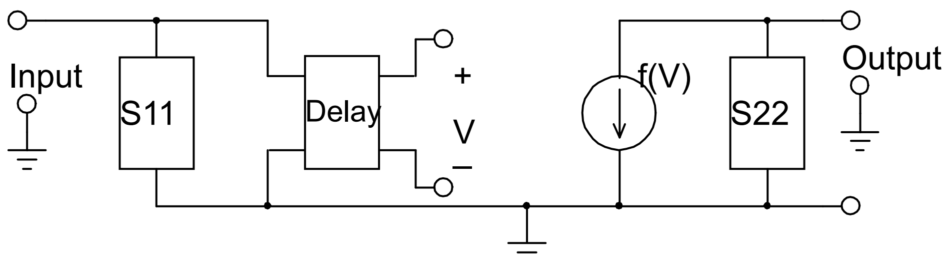
Symbol



Summary

This is a sophisticated model, and some aspects of its behavior may be different than expected. You should read this document entirely before using this component.

Equivalent Circuit



Parameters

Name	Description	Unit Type	Default
ID	Element ID		AM1
GAIN	Mid-band transducer gain	dB	10
NF	Noise figure	dB	0
IP2H	Mid-band output IP2 (harmonic)	dB Power	40
IP3	Mid-band output IP3	dB Power	30
P1DB	Output 1-db compression point	dB Power	10
*S11MAG	Input reflection coefficient magnitude		
*S11ANG	Input reflection coefficient phase angle	Angle	
*S22MAG	Output reflection coefficient magnitude		
*S22ANG	Output reflection coefficient phase angle	Angle	
*Z0	Port Impedance	Resistance	50
*TDLY	Group delay	Time	0

Linear Behavior

The linear behavior of the amplifier is modeled by a controlled current source (the linear part of f(v) in the equivalent circuit) and input and output impedances. The parameters of the controlled current source are derived from the user-specified gain and impedances, so the transducer gain is always the value specified. The gain, therefore, is the gain with the specified values of S_{11} or S_{22}. Changing S_{11} or S_{22} does not change the transducer gain.

Reverse transmission is assumed to be negligible; i.e., $S_{12} = 0$.

Nonlinear Behavior

IP_3 and Compression

It is important to recognize that IP_3 and the 1-dB compression point are not independent. If compression is caused by the small-signal nonlinearities of the device, expressed in Eq. , below, the 1 dB compression point must be approximately 10 dB below IP_3. However, if compression is caused by clipping of the large-signal drain or collector waveforms, saturation can occur at a lower level, and need not be related to IP_3. This is why amplifiers that are highly linear, in terms of IP_3, often do not obey the "10 dB" rule. For this reason, you can specify the 1-dB compression point and IP_3 independently, and the model automatically models distortion and saturation by an appropriate combination of clipping and small-signal effects.

The Nonlinear Model

Nonlinearities are modeled by a polynomial and a clipping function, providing the correct saturation and intermodulation characteristics, regardless of the relative values of IP_3 and P1DB. The controlled current source f(v) is modeled by a polynomial:

$$f(v) = a_1 v + a_2 v^2 + a_3 v^3$$

This polynomial models intermodulation distortion through third order. The values of the coefficients are derived from the specified intercept points.

The one-dB compression point is more problematic to model. One cause of compression in an amplifier is clipping of the waveforms when dc bias power is inadequate to provide output. This can happen, in theory, even if the amplifier is perfectly linear for small signals; that is, $a_2 = 0$ and $a_3 = 0$ in the polynomial. Compression can also be caused by the inherent small-signal nonlinearities in f(v). In this case, a cubic polynomial is not adequate to model compression, and unless other means are used, the model becomes very poor above the 1-dB compression point.

To avoid these difficulties, the amplifier model calculates the 1 dB compression point according to both criteria and uses the one that represents the lower of the two compression levels. If the amplifier's compression is caused by clipping, a clipping function is used with the value set appropriately. However, if compression is caused by the nonlinearities in f(v), these are allowed to provide compression. The clipping level is then set somewhat higher, to provide the correct behavior in hard saturation.

The transition between these two conditions is approximately 10 dB below the third-order intercept point, IP_3. Therefore, if $P1DB < IP_3 - 10$, the amplifier saturates on clipping, while, for higher values, the nonlinearities of f(v) dominate.

The clipping function is symmetrical, so it affects only third-order intermodulation. The second-order IM level saturates gracefully, but does not exhibit the sudden increase in level that can be observed in the third-order.

As with linear characteristics, the calculation of the coefficients of the polynomial includes the effect of S_{11} and S_{22}. Therefore, if the value of load and source resistance is Z_0, changing S_{11} or S_{22} does not affect the calculated IM levels.

Layout

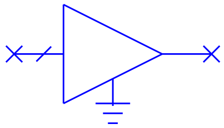
This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Reference

This model was developed under research performed at NI AWR Corporation. The full set of details of the implementation are considered proprietary in nature.

Nonlinear Amplifier System Model (Closed Form): NL_AMP2

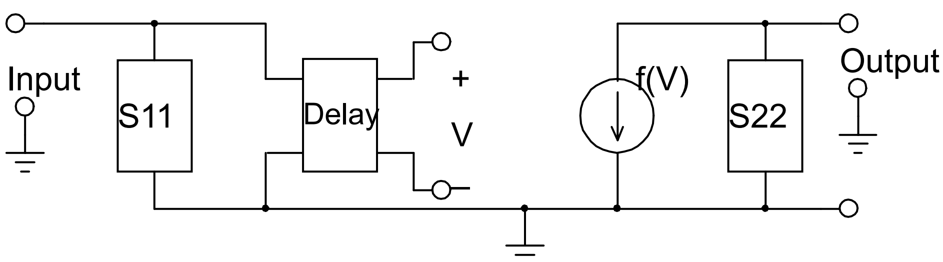
Symbol



Summary

NL_AMP2 is a sophisticated model, and some aspects of its behavior may be different from expected. Therefore, you should read this entire section before using this model. NL_AMP2 offers improved accuracy of nonlinear characteristics and full noise and S parameters over NL_AMP.

Equivalent Circuit



Parameters

Name	Description	Unit Type	Default
ID	Element ID		AM1
NFMIN	Minimum noise figure	dB	3.0
RN_NORM	Normalized noise resistance		0.5
GOPT_MAG	Source reflection coefficient magnitude at optimum NF		0.0
GOPT_ANG	Source reflection coefficient angle at optimum NF	Angle	0.0
IP2H	Mid-band output IP2 (harmonic)	dB Power	40
IP3	Mid-band output IP3	dB Power	20
P1DB	Output 1-db compression point	dB Power	10
*S11MAG	Input reflection coefficient magnitude		0.0
*S11ANG	Input reflection coefficient phase angle	Angle	0.0
*S21MAG	Transmission coefficient magnitude		3.1623
*S21ANG	Transmission coefficient phase angle	Angle	180.0
*S12MAG	Reverse transmission coefficient magnitude		0.0
*S12ANG	Reverse transmission coefficient phase angle	Angle	0.0
*S22MAG	Output reflection coefficient magnitude		0.0

Name	Description	Unit Type	Default
*S22ANG	Output reflection coefficient phase angle	Angle	0.0
*Z0	Port Impedance	Resistance	50.0

** indicates a secondary parameter*

Linear Behavior

The linear behavior of the amplifier is modeled by a controlled current source (the linear part of f(v) in the equivalent circuit) and input and output impedances. The parameters of the controlled current source are derived from the user-specified gain and impedances, so the transducer gain is always the value specified. The gain, therefore, is the gain with the specified values of S11 or S22. Changing S11 or S22 does not change the transducer gain, as long as the source and load impedances remain at Z0.

The gain is entered by S21. Note that the transducer power gain, when the source and load impedances equal Z0, is $S21MAG^2$.

Nonlinear Behavior

IP3 and Compression

It is important to recognize that IP3 and the 1-dB compression point are not independent. If compression is caused by the small-signal nonlinearities of the device, expressed in the equation below, the 1 dB compression point must be approximately 10 dB below IP3. However, if compression is caused by clipping the large-signal drain or collector waveforms, saturation can occur at a lower level, and need not be related to IP3. This is why amplifiers that are highly linear, in terms of IP3, often do not obey the "10 dB" rule.

The Nonlinear Model

Nonlinearities are modeled by a polynomial and a clipping function. This provides the correct saturation and intermodulation characteristics, regardless of the relative values of IP3 and P1DB. The controlled current source f(v) is modeled by a polynomial:

$$f(v) = a_1 v + a_2 v^2 + a_3 v^3$$

This polynomial models intermodulation distortion through third order. The values of the coefficients are derived from the specified intercept points.

The one-dB compression point is more problematic to model. One cause of compression in an amplifier is clipping of the waveforms when dc bias power is inadequate to provide output. This can happen, in theory, even if the amplifier is perfectly linear for small signals; that is, when $a_2 = a_3 = 0$ in the polynomial. Compression can also be caused by the inherent small-signal nonlinearities in f(v). In this case, a cubic polynomial is not adequate to model compression, and unless other means are used, the model becomes very poor above the 1-dB compression point.

To avoid these difficulties, the amplifier model calculates the 1 dB compression point according to both criteria and uses the one that represents the lower of the two compression levels. If the amplifier's compression is caused by clipping, a clipping function is used with the value set appropriately. However, if compression is caused by the nonlinearities in f(v), these are allowed to provide compression. The clipping level is then set somewhat higher, to provide the correct behavior in hard saturation.

The transition between these two conditions is approximately 10 dB below the third-order intercept point, IP3. Therefore, if P1DB < IP3 - 10, the amplifier saturates on clipping, while, for higher values, the nonlinearities of f(v) dominate.

The clipping function is symmetrical, so it affects only third-order intermodulation. The second-order IM level saturates gracefully, but does not exhibit the sudden increase in level that is observable in the third-order.

As with linear characteristics, the calculation of the coefficients of the polynomial includes the effect of S11 and S22. Therefore, if the value of load and source resistance is Z0, changing S11 or S22 does not affect the calculated IM levels.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

Reference

This model was developed under research performed at NI AWR Corporation. The full set of details of the implementation are proprietary.

N-Port S-Parameter Block (from file): NPORT_F

Symbol

Summary

NPORT_F models a generalized N-port S-parameter network, with the S-parameters read from a Touchstone format file. It includes an option to expose an explicit ground terminal.

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	X1
N	Number of ports		2
FILE	File Name		""
*INTERP	Interpolation method		Linear
*INTERPDOM	Interpolation domain		Rectangular
*EXTRAP	Extrapolation method		Interp Method
*DIR	Directory to search for file		Project
*GND	Expose Explicit Ground		No
*TEMP	Temperature for noise calculation	Temperature	16.85 DegC
*FORCEPASSIVE	Consider data passive for noise and transient simulations		Default
*SHOWPORTNAMES	Display port names on symbol		Yes

** indicates a secondary parameter*

Parameter Details

FILE. May contain a relative or absolute file path. Relative paths are relative to the directory specified by the DIR parameter.

INTERP. Specifies the method used to interpolate the data to frequencies between those specified in the file. Choices are "Linear", "Cubic Spline", "Cubic", "Floor", and "Ceiling".

INTERPDOM. Specifies the form in which the data is interpolated. Choices are "Rectangular", "Polar", and "DB". When set to "Rectangular" the real and imaginary components of the data are interpolated separately. When set to "Polar" the interpolation is applied to the magnitude and angle of the data. Setting to "DB" is the same as "Polar", except that the log of the magnitude values is used during interpolation. Note that this parameter has no effect when INTERP is set to "Floor" or "Ceiling".

EXTRAP. Specifies the method used to extrapolate the data to frequencies below the first or after the last frequency specified in the file. Choices are "Interp Method" and "Constant". When the parameter is set to "Interp Method", extrapolation uses the method specified by INTERP. When set to "Constant", the first frequency point is used for any simulation frequencies below the range specified in the file, and the last frequency point is used for any simulation above it.

DIR. When FILE has a relative path, this parameter specifies to which directory the path is relative. Choices are "Project", "Library", and "EM Models". The "Project" directory stores the project file. The "Library" directory is specified in the Environment Options dialog box (choose **Options > Environment Options**), and the "EM Models" directory is the directory displayed in the Directories dialog box (choose **Help > Show Files/Directories**).

GND. Specifies whether or not to use **Explicit ground node** as the grounding type. To learn more about grounding types see “Grounding Types”.

SHOWPORTNAMES. Specifies whether port names on the terminals display or are hidden for Touchstone data files that define the port names.

Parameter Restrictions and Recommendations

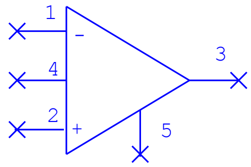
N must be greater than zero.

Layout

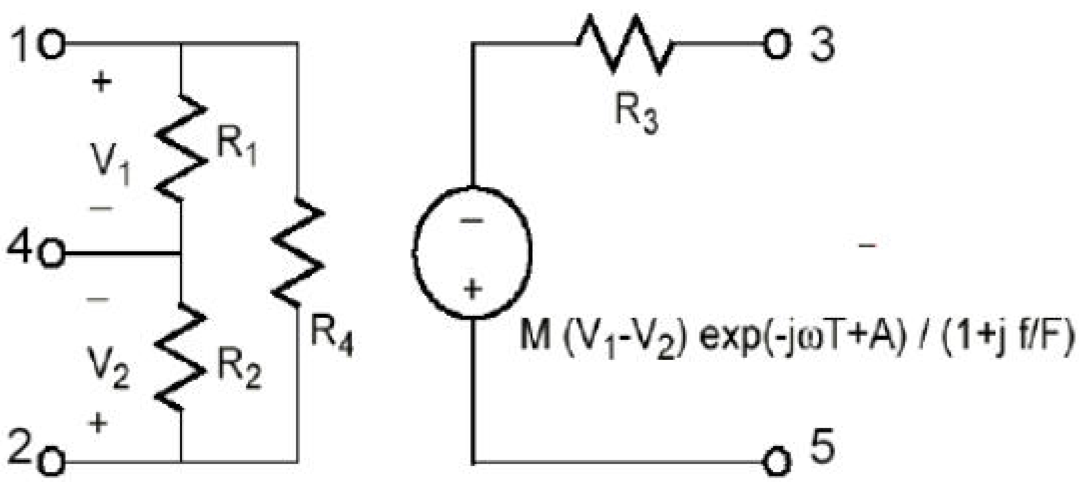
This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

Opamp (Closed Form): OPAMP

Symbol



Equivalent Circuit



Parameters

Name	Description	Unit Type	Default
ID	Name	Text	U1
M	DC voltage gain (factor)		1e+06
A	Phase offset	Angle	0 Deg
R1	(-) terminal input resistance	Resistance	0 ohm
R2	(+) terminal input resistance	Resistance	0 ohm
R3	Output resistance	Resistance	100 ohm
R4	(+) to (-) terminal resistance	Resistance	0 ohm
F	3-dB freq	Frequency	100 GHz
T	Time delay	Time	0 ns

Implementation Details

Implements an ideal operational amplifier with output voltage given by

$$V_{out} = M(V_1 - V_2)\frac{(-j\omega T + A)}{1 + j\frac{f}{F}}$$

NOTES::

R1, R2, R4, resistance value of 0 means infinite impedance.

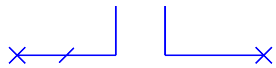
If F = 0, then gain has no frequency dependence.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Open-circuited Stub Resonator: OST

Symbol



Summary

OST models a resonator made up of an open-circuited segment of ideal transmission line.

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	TL1
Z0	Characteristic impedance of transmission line	Resistance	50 ohm
EL	Electrical length (phase length) at F0	Angle	90 degrees
F0	Frequency used to specify EL	Frequency	1 GHz

Parameter Details

EL and **F0**. These parameters determine the frequency-dependence of the electrical length of the line defined as: electrical length in radians = $EL \cdot freq/Fo \cdot \pi/180$ where freq is the evaluation frequency.

Layout

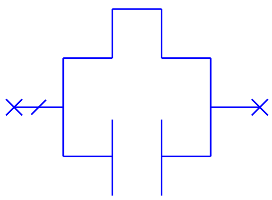
This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

Recommendations for Use

This element is often used for filter synthesis.

Short-circuited Stub Resonator in Parallel with Capacitor: PCS

Symbol



Summary

PCS models a parallel connection of a capacitor and resonator made up of short-circuited segments of ideal transmission line.

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	TL1
C	Capacitance of shunt capacitor	Capacitance	1 pF
Z0	Characteristic impedance of short-circuited transmission line	Resistance	50 ohm
EL	Electrical length (phase length) at F0	Angle	90 degrees
F0	Frequency used to specify EL	Frequency	1 GHz

Parameter Details

EL and **F0**. These parameters determine the frequency-dependence of the electrical length of the line defined as: electrical length in radians = EL· freq/Fo ·π/180 where freq is the evaluation frequency.

Layout

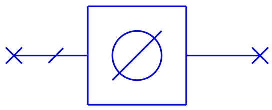
This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Recommendations for Use

This element is often used for filter synthesis.

Phase Shift, NonReciprocal (Closed Form): PHASE

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	P1
A	Constant phase for frequency < F	Angle	0 Deg
S	Phase slope per frequency octave	Angle	0 Deg
F	Frequency for start of phase slope	Frequency	0 GHz
Zo	Reference impedance		50 ohm

Implementation Details

Applies a non-reciprocal phase shift per the following equations. For a reciprocal phase shift use the PHASE2 element. Frequency dependence is defined by

then

$$\Theta(f) = \begin{cases} -A & f \le F \\ A + S \times \log_2\left(\frac{f}{F}\right) & f \ge F \end{cases} .$$

$$S_{11} = S_{22} = 0$$

$$S_{21} = e^{j\Theta(f)}$$

$$S_{12} = e^{-j\Theta(f)}$$

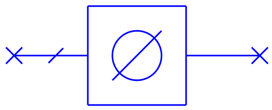
NOTE: if F = 0, the phase slope is 0 regardless of the value of S.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

Phase Shift, Reciprocal (Closed Form): PHASE2

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	P1
A	Constant phase for frequency < F	Angle	0 Deg
S	Phase slope per frequency octave	Angle	0 Deg
F	Frequency for start of phase slope	Frequency	0 GHz
Zo	Reference impedance		50 ohm

Implementation Details

This is the reciprocal version of the PHASE element. The two elements are identical except for the S parameter definition.

$$S_{11} = S_{22} = 0$$

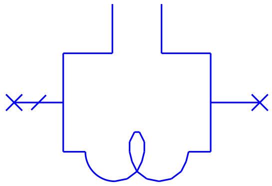
$$S_{21} = S_{12} = e^{j\Theta(f)}$$

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Open-circuited Stub Resonator in Parallel with Inductor: PLO

Symbol



Summary

PLO models a parallel connection of an inductor and resonator made up of open-circuited segments of ideal transmission line.

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	TL1
L	Inductance of shunt inductor	Inductance	1 nH
Z0	Characteristic impedance of open-circuited transmission line	Resistance	50 ohm
EL	Electrical length (phase length) at F0	Angle	90 degrees
F0	Frequency used to specify EL	Frequency	1 GHz

Parameter Details

EL and **F0**. These parameters determine the frequency-dependence of the electrical length of the line defined as: electrical length in radians = EL· freq/Fo ·π/180 where freq is the evaluation frequency.

Layout

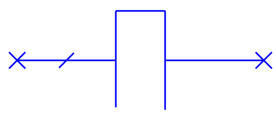
This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Recommendations for Use

This element is often used for filter synthesis.

Open-circuited and Short-circuited Stub Resonators Connected in Parallel: PSR

Symbol



Summary

PSR models a pair of parallel resonators made up of open-circuited and short-circuited segments of ideal transmission line.

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	TL1
ZSC	Characteristic impedance of short-circuited transmission line	Resistance	50 ohm
ZOC	Characteristic impedance of open-circuited transmission line	Resistance	50 ohm
EL	Electrical length (phase length) at F0	Angle	90 degrees
F0	Frequency used to specify EL	Frequency	1 GHz

Parameter Details

EL and **F0**. These parameters determine the frequency-dependence of the electrical length of the line defined as: electrical length in radians = $EL \cdot freq/Fo \cdot \pi/180$ where freq is the evaluation frequency. EL and F0 are common for both resonators.

Layout

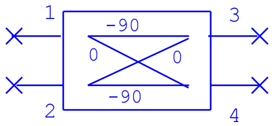
This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Recommendations for Use

This element is often used for filter synthesis.

Quadrature Hybrid (Closed Form): QHYB

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Name	Text	U1
R	Port resistance	Resistance	50 ohm
COUPL	Coupling	DB	3 dB
LOSS	Excess loss	DB	0 dB

Implementation Details

Implements an ideal quadrature coupler with the following 4-port S-parameter matrix, where all ports have the resistance R.

$$S = \begin{bmatrix} 0 & 0 & T & C \\ 0 & 0 & C & T \\ T & C & 0 & 0 \\ C & T & 0 & 0 \end{bmatrix}$$

the transmission term T is given by

$$T = \text{-j}(\text{LX}\sqrt{1-\text{CF}^2})$$

and the coupling term C is given by

$$C = \text{LX} \cdot \text{CF}$$

where

$$\text{LX} = 10^{\frac{-|\text{LOSS}|}{20}}$$

$$\text{CF} = 10^{\frac{-|\text{COUPL}|}{20}}$$

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

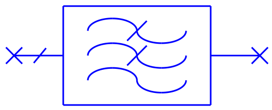
Restrictions

R ≥ 0.01 Required

COUPL ≥ 0.001 Required

Butterworth Quasi-Lowpass Filter: QLPFB

Symbol



Summary

QLPFB models represent lumped-element Butterworth quasi-lowpass filters. They offer simplicity and a compromise between high selectivity and flat group delay. The insertion loss is maximally flat at the passband's geometric center and the upper stopband attenuation increases monotonically.

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	QLPFB1
HN	Half the number of reactances.		3
FL	Lower frequency edge of passband (when Qu is infinite).	Frequency	0.5 GHz
FH	Upper frequency edge of passband (when Qu is infinite).	Frequency	1.5 GHz
*AP	Maximum passband attenuation (when Qu is infinite).	DB	3.0103 dB
*RS	Source resistance.	Resistance	50 ohm
*RL	Load resistance	Resistance	50 ohm
*QU	Uniform unloaded Q reactive element in the filter		1e12

** indicates a secondary parameter*

Parameter Restrictions and Recommendations

1. 0 < HN < 29
2. 0 < FL
3. 0 < FH
4. 0 < AP Recommend AP greater than or equal to 0.001 dB.
5. 0 < RS
6. 0 < RL
7. 0 < QU. Recommend QU less than or equal to 1e12.

Implementation Details

The model is implemented as a short-circuit admittance matrix, whose equivalent transfer function squared magnitude is that of a Butterworth filter:

$$|S_{21}(s)|^2 = \frac{1}{1+(-1)^{\mathrm{HN}}\varepsilon^2 s^{2\mathrm{HN}}} = \frac{1}{|g(s)|^2} = \frac{1}{1+|h(s)|^2}$$

where

$$\varepsilon^2 = 10^{AP/10} - 1$$

where

$$s = \frac{1}{QU} + j\omega$$

where

$$j = \sqrt{-1}$$

where

and a lowpass-to-quasi-lowpass frequency transformation has been applied:

$$\omega = \frac{2 \times (\_FREQ)^2 - FL^2 - FH^2}{| FH^2 - FL^2 |}$$

_FREQ is the variable containing the project frequency, and the admittances are:

$$y_{11} = \left(\frac{1}{RS}\right)\frac{g(s) + g(-s) - h(s) - h(-s)}{g(s) - g(-s) + h(s) - h(-s)}$$

$$y_{22} = \left(\frac{1}{RL}\right)\frac{g(s) + g(-s) + h(s) + h(-s)}{g(s) - g(-s) + h(s) - h(-s)}$$

$$y_{12} = y_{21} = \left(\frac{1}{\sqrt{RS \times RL}}\right)\frac{2}{g(s) - g(-s) + h(s) - h(-s)}$$

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Recommendations for Use

Note that this model behaves as if it has ideal impedance transformers at its ports, so there is no attenuation due to mismatched source and load impedances. The model expects that the source impedance will equal RS and that the load impedance will equal RL, but RS need not have any special relationship to RL for ideal transmission (as would normally be the case).

References

[1] Rolf Schaumann, Mohammed S. Ghausi, and Kenneth R. Laker, Design of Analog Filters: Passive, Active RC, and Switched Capacitor, (Prentice-Hall, 1990), pp. 40-44.

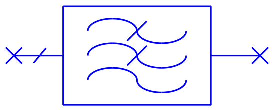
[2] Louis Weinberg, Network Analysis and Synthesis, (Robert E. Krieger Publishing, 1975), pp. 493-498.

[3] Adel S. Sedra and Peter O. Brackett, Filter Theory and Design: Active and Passive, (Matrix Publishers, 1978), pp. 105-111.

[4] Max W. Medley, Jr., Microwave and RF Circuits: Analysis, Synthesis and Design, (Artech House, 1993), pp. 312-317.

Chebyshev Quasi-Lowpass Filter: QLPFC

Symbol



Summary

QLPFC models represent lumped-element Chebyshev quasi-lowpass filters. The insertion loss ripples between zero and a specified maximum in the passband. The stopband attenuation increases rapidly beyond the upper passband edge, is monotonic, and is maximally flat at infinite frequency. This type of filter offers both simplicity and good selectivity.

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	LPFC1
HN	Half the number of reactive elements		3
FL	Lower frequency edge of passband (when Qu is infinite).	Frequency	0.5 GHz
FH	Upper frequency edge of passband (when Qu is infinite).	Frequency	1.5 GHz
AP	Maximum passband attenuation (when Qu is infinite).	DB	0.1 dB
*RS	Expected source resistance (at port 1)	Resistance	50 ohm
*RL	Expected load resistance (at port 2)	Resistance	50 ohm
*QU	Average unloaded Q of reactive elements in the filter		1e12

** indicates a secondary parameter*

Parameter Restrictions and Recommendations

1. 0 < HN < 27
2. 0 < FL
3. 0 < FH
4. 0 < AP Recommend AP greater than or equal to 0.001 dB.
5. 0 < RS
6. 0 < RL
7. 0 < QU. Recommend QU less than or equal to 1e12.

Implementation Details

The model is implemented as a short-circuit admittance matrix, whose equivalent transfer function squared magnitude is that of a Chebyshev filter:

$$|S_{21}(s)|^2 = \frac{1}{1+\varepsilon^2 C_n^2\left(\frac{s}{j}\right)} = \frac{1}{|g(s)|^2} = \frac{1}{1+|h(s)|^2}$$

where Cn is the Chebyshev polynomial of the first kind, and

$$\varepsilon^2 = 10^{\mathrm{AP}/10} - 1$$

$$C_n(\omega) = \cos(\mathrm{HN} * \mathrm{acos}(\omega)) \ \mathrm{for} 0 \leq \ |\ \omega\ |\ \leq 1$$

$$C_n(\omega) = \cosh(\mathrm{HN} * \mathrm{acosh}(\omega)) \ \mathrm{for} 1 < \ |\ \omega\ |$$

$$s = \frac{1}{\mathrm{QU}} + j\omega$$

$$j = \sqrt{-1}$$

and a lowpass-to-quasi-lowpass frequency transformation has been applied:

$$\omega = \frac{2 \times (\_\mathrm{FREQ})^2 - \mathrm{FL}^2 - \mathrm{FH}^2}{|\ \mathrm{FH}^2 - \mathrm{FL}^2\ |}$$

_FREQ is the variable containing the project frequency, and the admittances are:

$$y_{22} = \left(\frac{1}{\mathrm{RL}}\right)\frac{g(s) + g(-s) + h(s) + h(-s)}{g(s) - g(-s) + h(s) - h(-s)}$$

$$y_{12} = y_{21} = \left(\frac{1}{\sqrt{\mathrm{RS} \times \mathrm{RL}}}\right)\frac{2}{g(s) - g(-s) + h(s) - h(-s)}$$

$$j = \sqrt{-1}$$

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Recommendations for Use

Note that this model behaves as if it has ideal impedance transformers at its ports, so there is no attenuation due to mismatched source and load impedances. The model expects that the source impedance will equal RS and that the load impedance will equal RL, but RS need not have any special relationship to RL for ideal transmission (as would normally be the case).

References

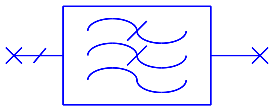
[1] Rolf Schaumann, Mohammed S. Ghausi, and Kenneth R. Laker, Design of Analog Filters: Passive, Active RC, and Switched Capacitor, (Prentice-Hall, 1990), pp. 44-48.

[2] Louis Weinberg, Network Analysis and Synthesis, (Robert E. Krieger Publishing, 1975), pp. 507-529.

[3] Max W. Medley, Jr., Microwave and RF Circuits: Analysis, Synthesis and Design, (Artech House, 1993), pp. 312-317.

Bessel Quasi-Lowpass Filter: QLPFD

Symbol



Summary

QLPFD models represent lumped-element Bessel-Thomson quasi-lowpass filters. They offer simplicity and maximally flat group delay, but suffer from poor selectivity.

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	QLPFD1
HN	Half the number of reactances		3
FL	Lower passband edge (when Qu is infinite).	Frequency	0.5 GHz
FH	Upper passband edge (when Qu is infinite).	Frequency	1.5 GHz
AP	Maximum passband attenuation (when Qu is infinite).	DB	3.0103 dB
*RS	Expected source resistance.	Resistance	50 ohm
*RL	Expected Load resistance	Resistance	50 ohm
*QU	Average unloaded Q of reactive elements in filter.		1e12

* indicates a secondary Parameter

Parameter Restrictions and Recommendations

1. 0 < HN < 34
2. 0 < FL
3. 0 < FH
4. 0 < AP Recommend AP greater than or equal to 0.001 dB.
5. 0 < RS
6. 0 < RL
7. 0 < QU. Recommend QU less than or equal to 1e12.

Implementation Details

The model is implemented as a short-circuit admittance matrix,

$$\begin{bmatrix} y_{11} & y_{12} \\ y_{21} & y_{22} \end{bmatrix}$$

, whose equivalent transfer function squared magnitude is that of a Bessel-Thomson filter:

$$|S_{21}(s)|^2 = \frac{b_0^2}{|g(s)|^2} = \frac{b_0^2}{1 + |h(s)|^2}$$

where

$$g(s) = \sum_{i=0}^{N} b_i s^i$$

$$b_i = \frac{(2\mathrm{HN}-1)!}{2^{\mathrm{HN}-i}(i)!(\mathrm{HN}-1)!} \text{ for } i = 0, ..., \mathrm{HN}$$

Where b_0 = (2HN-1)!!,i.e, the product of all odd integers less than 2HN.

$$s = \frac{1}{\mathrm{QU}} + j\omega$$

$$j = \sqrt{-1}$$

and a lowpass-to-quasi-lowpass frequency transformation has been applied:

$$\omega = \frac{2 \times (\_\mathrm{FREQ})^2 - \mathrm{FL}^2 - \mathrm{FH}^2}{|\mathrm{FH}^2 - \mathrm{FL}^2|}$$

_FREQ is the variable containing the project frequency, and the admittances are:

$$y_{11} = \left(\frac{1}{\mathrm{RS}}\right)\frac{g(s) + g(-s) - h(s) - h(-s)}{g(s) - g(-s) + h(s) - h(-s)}$$

$$y_{22} = \left(\frac{1}{\mathrm{RL}}\right)\frac{g(s) + g(-s) + h(s) + h(-s)}{g(s) - g(-s) + h(s) - h(-s)}$$

$$y_{12} = y_{21} = \left(\frac{1}{\sqrt{RS \times RL}}\right)\frac{(-2)b_0}{g(s) - g(-s) + h(s) - h(-s)}$$

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Recommendations for Use

Note that this model behaves as if it has ideal impedance transformers at its ports, so there is no attenuation due to mismatched source and load impedances. The model expects that the source impedance will equal RS and that the load impedance will equal RL, but RS need not have any special relationship to RL for ideal transmission (as would normally be the case).

References

[1] Rolf Schaumann, Mohammed S. Ghausi, and Kenneth R. Laker, Design of Analog Filters: Passive, Active RC, and Switched Capacitor, (Prentice-Hall, 1990), pp. 51-56.

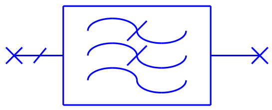
[2] Louis Weinberg, Network Analysis and Synthesis, (Robert E. Krieger Publishing, 1975), pp. 499-506.

[3] Herman J. Blinchikoff and Anatol I. Zverev, Filtering in the Time and Frequency Domains, (Robert E. Krieger Publishing Co., 1987), pp. 124-127.

[4] Max W. Medley, Jr., Microwave and RF Circuits: Analysis, Synthesis and Design, (Artech House, 1993), pp. 312-317.

Gaussian Quasi-Lowpass Filter (Closed Form): QLPFG

Symbol



Summary

QLPFG models represent lumped-element Gaussian quasi-lowpass filters. They offer simplicity, relatively flat group delay, and good time domain performance, but suffer from poor frequency selectivity. Although similar to Bessel-Thomson filters, Gaussian filters offer faster rise times and lower transient overshoots, but have slightly less stopband attenuation and less group delay flatness.

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	QLPFG1
HN	Half the number of reactances		3
FL	Lower passband edge (when Qu is infinite).	Frequency	0.5 GHz
FH	Upper passband edge (when Qu is infinite).	Frequency	1.5 GHz
AP	Maximum passband attenuation (when Qu is infinite).	DB	3.0103 dB
*RS	Expected source resistance.	Resistance	50 ohm
*RL	Expected Load resistance	Resistance	50 ohm
*QU	Average unloaded Q of reactive elements in filter.		1e12

* indicates a secondary Parameter

Parameter Restrictions and Recommendations

1. 0 < HN < 51
2. 0 < FL
3. 0 < FH
4. 0 < AP Recommend AP greater than or equal to 0.001 dB.
5. 0 < RS
6. 0 < RL
7. 0 < QU. Recommend QU less than or equal to 1e12.

Implementation Details

The model is implemented as a short-circuit admittance matrix,

$$\begin{bmatrix} y_{11} & y_{12} \\ y_{21} & y_{22} \end{bmatrix}$$

, whose equivalent transfer function squared magnitude is that of an ideal Gaussian filter. The ideal Gaussian squared magnitude characteristic is:

$$|S_{21}|^2 = \frac{1}{\exp(-s^2)}$$

In the model, the denominator of this ideal Gaussian characteristic is approximated by a truncated Maclaurin series:

$$\exp(-s^2) \approx \sum_{i=0}^{N} \frac{(-1)^i s^{2i}}{i!} = |g(s)|^2 = 1 + |h(s)|^2$$

where

$$s = \frac{1}{\mathrm{QU}} + j\omega$$

$$j = \sqrt{-1}$$

and a lowpass-to-quasi-lowpass frequency transformation has been applied:

$$\omega = \frac{2 \times (\_\mathrm{FREQ})^2 - \mathrm{FL}^2 - \mathrm{FH}^2}{|\mathrm{FH}^2 - \mathrm{FL}^2|}$$

_FREQ is the variable containing the project frequency, and the admittances are:

$$y_{11} = \left(\frac{1}{\mathrm{RS}}\right)\frac{g(s) + g(-s) - h(s) - h(-s)}{g(s) - g(-s) + h(s) - h(-s)}$$

$$y_{22} = \left(\frac{1}{\mathrm{RL}}\right)\frac{g(s) + g(-s) + h(s) + h(-s)}{g(s) - g(-s) + h(s) - h(-s)}$$

$$y_{12} = y_{21} = \left(\frac{1}{\sqrt{\mathrm{RS} \times \mathrm{RL}}}\right)\frac{(-2)}{g(s) - g(-s) + h(s) - h(-s)}$$

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Recommendations for Use

This model behaves as if it has ideal impedance transformers at its ports, so there is no attenuation due to mismatched source and load impedances. The model expects that the source impedance will equal RS and that the load impedance will equal RL, but RS need not have any special relationship to RL for ideal transmission (as would normally be the case).

This filter model is non-causal and not usable in transient simulations. An error message is issued if a transient simulation of a circuit containing this model is attempted. (Causality is defined as the response of a circuit *following* a stimulus--not *preceding* a stimulus. Non-causal models do not correspond to a physically realizable device.)

References

[1] Milton Dishal, "Gaussian-Response Filter Design," Electrical Communication, vol. 36, March 1959, pp. 3-26.

[2] Anatol I. Zverev, Handbook of Filter Synthesis, (John Wiley & Sons, 1967), pp. 67, 70, 71, 73, 74, 90, 91.

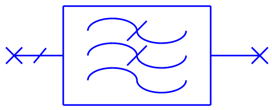
[3] DeVerl. S. Humpherys, The Analysis, Design, and Synthesis of Electrical Filters, (Prentice-Hall, 1970), pp. 413-417.

[4] Herman J. Blinchikoff and Anatol I. Zverev, Filtering in the Time and Frequency Domains, (Robert E. Krieger Publishing Co., 1987), pp. 130-132.

[5]Max W. Medley, Jr., Microwave and RF Circuits: Analysis, Synthesis and Design, (Artech House, 1993), pp. 312-317.

Raised-Cosine Quasi-Lowpass Filter (Closed Form): QLPFRC

Symbol



Summary

QLPFRC models represent Raised-Cosine Quasi-Lowpass filters. Raised-Cosine filters are applied in digital transmission system theory to represent ideal (intersymbol interference free) Nyquist filtering of impulse and pulse data transmissions. Hardware designers attempt to design channel filters which approximate these characteristics and have the smallest possible bandwidth.

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	QLPFRC1
FL	Lower frequency edge of passband	Frequency	0.5 GHz
FH	Upper frequency edge of passband	Frequency	1.5 GHz
A	Roll-off factor (between 0 and 1)		0.5
TYP	Transmission type	Vector Text(pull-down)	Impulse
E	Exponent of the raised-cosine response		1.
*RS	Source resistance	Resistance	50 ohm
*RL	Load resistance	Resistance	50 ohm

* indicates a secondary Parameter

Parameter Restrictions and Recommendations

1. $0 < FL$
2. $0 < FH$
3. $0 \leq A \leq 1$.

   A minimum bandwidth (A=0) filter would require an infinite number of filter sections. Approximations of a 30% excess Nyquist bandwidth (A=0.3) raised-cosine filter are considered feasible with present-day technology.
4. TYP is either $\{0,1\}$.

   TYP = 0 (Impulse) specifies the ideal raised-cosine response, the theoretical filter model for infinitesimally narrow impulses. TYP = 1 (Pulse) specifies a sinc-normalized raised-cosine response, the filter model for rectangular pulse transmission.
5. $0<E\leq 1$.

The ideal raised-cosine response is raised to the exponent, E. For cascaded identical transmit and receive filters, you would typically specify E=0.5 for each filter. The composite channel response of the cascaded filters would be represented by specifying E=1.

6. 0 < RS.

   By definition, the model matches RS at its input port.

7. 0 < RL.

   By definition, the model matches RL at its output port.

Implementation Details

The model is implemented as a short-circuit admittance matrix,

$$\begin{bmatrix} y_{11} & y_{12} \\ y_{21} & y_{22} \end{bmatrix}$$

, whose equivalent amplitude functions S21 and S12 implement the raised-cosine response for impulse or pulse data transmission, while S11 and S22 are, by definition, constant and matched to the filter port termination resistances (i.e., equal to 1). For impulses,

$$S_{21}(j\omega) = 1 \text{ for } [0 \le \omega \le (1-A)]$$

$$S_{21}(j\omega) = \left(\cos^2\left(\left(\frac{\pi}{4}\right)\left(\frac{\omega + A - 1}{A}\right)\right)\right)^E \text{ for } [(1-A) \le \omega \le (1+A)]$$

$$S_{21}(j\omega) = 0 \text{ for } [(1+A) \le \omega]$$

For pulses,

$$S_{21}(j\omega) = 1 \text{ for } [0 \le \omega \le (1-A)]$$

$$S_{21}(j\omega) = \frac{\left(\cos^2\left(\left(\frac{\pi}{4}\right)\left(\frac{\omega + A - 1}{A}\right)\right)\right)^E}{\operatorname{sinc}\left(\frac{\pi}{2}\omega\right)} \text{ for } [(1-A) \le \omega \le (1+A)]$$

$$S_{21}(j\omega) = 0 \text{ for } [(1+A) \le \omega]$$

where

$$\operatorname{sinc}(x) = \frac{\sin(x)}{x}$$

$$j = \sqrt{-1}$$

and a lowpass-to-quasi-lowpass frequency transformation has been applied:

$$\omega = \frac{2 \times (\_\mathrm{FREQ})^2 - \mathrm{FL}^2 - \mathrm{FH}^2}{|\ \mathrm{FH}^2 - \mathrm{FL}^2\ |}$$

_FREQ is the variable containing the project frequency, and the admittances are:

$$y_{11} = \left(\frac{1}{\mathrm{RS}}\right)\left(\frac{-S_{21}^2(j\omega)}{S_{21}^2(j\omega)-4}\right)$$

$$y_{22} = \left(\frac{1}{\mathrm{RL}}\right)\left(\frac{-S_{21}^2(j\omega)}{S_{21}^2(j\omega)-4}\right)$$

$$y_{22} = \left(\frac{1}{\sqrt{\mathrm{RS}\times\mathrm{RL}}}\right)\left(\frac{2S_{21}(j\omega)}{S_{21}^2(j\omega)-4}\right)$$

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Recommendations for Use

This model expects that the source impedance will equal RS and that the load impedance will equal RL, but RS need not equal RL for ideal transmission.

This filter model is non-causal and not usable in transient simulations. An error message is issued if a transient simulation of a circuit containing this model is attempted. (Causality is defined as the response of a circuit *following* a stimulus--not *preceding* a stimulus. Non-causal models do not correspond to a physically realizable device.)

References

[1] Kamilo Feher, Digital Communications: Microwave Applications, (Prentice-Hall, 1981), pp. 46-51.

[2] John G. Proakis, Digital Communications, 2nd Ed., (McGraw-Hill, 1989), pp. 532-536.

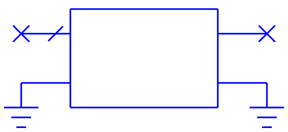
[3] John Bellamy, Digital Telephony, 2nd Ed., (John Wiley & Sons, 1991), pp. 523-528.

[4] E. A. Lee and D. G. Messerschmitt, Digital Communications, 2nd Ed., (Kluwer Academic Publishers, 1994), pp. 188-191, 226-228.

[5] Max W. Medley, Jr., Microwave and RF Circuits: Analysis, Synthesis and Design, (Artech House, 1993), pp. 312-317.

Two-Port S-Parameter Block: S2P_BLK

Symbol



Summary

S2P_BLK models a generalized two-port, S-parameter network. The full two-port S-parameters are specified. Noise is also included.

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	X1
S11	S(1,1)	Scalar (Complex)	0.0 + j*0.0
S21	S(2,1)	Scalar (Complex)	0.0 + j*0.0
S12	S(1,2)	Scalar (Complex)	0.0 + j*0.0
S22	S(2,2)	Scalar (Complex)	0.0 + j*0.0
*Z1	Port 1 impedance	Scalar (Complex)	50.0 ohm
*Z2	Port 2 impedance	Scalar (Complex)	50.0 ohm
*NFMIN	Minimum noise figure in dB	dB	0.0
*GOPT	Optimum noise match	Scalar (Complex)	0.0 + j*0.0
*RN	Noise resistance	Resistance	0.0 ohm
*ChkPassv	Switch "Do not Check passivity/Check passivity"		No

** indicates a secondary parameter*

Parameter Details

ChkPassv. If set to Yes, tests S2P_BLK for passivity and displays a warning if there is a passivity violation. The default setting is No (Do not check passivity).

Parameter Restrictions and Recommendations

1. Z1 and Z2 must be greater than zero.
2. RN must be greater than the minimum noise resistance, which is calculated from the NFMIN and GOPT parameters using the following formula:

$$RN_{min} = \frac{10^{NF_{min}/10} - 1}{4.0 * \mathrm{Re}(Y_{OPT})}$$

   where YOPT is the optimum noise match admittance calculated from GOPT. If RN is less than RNmin, RN is set equal to RNmin and a warning message is displayed to the user.

Layout

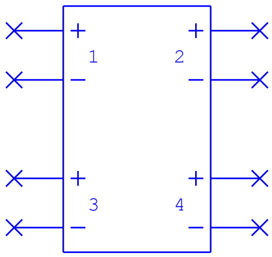
This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Recommendations for Use

The S2P_BLK model can be used to model any general two-port network (passive or active) by specifying the S-parameters and the noise parameters.

Four-Port S-Parameter Block: S4P_BLK

Symbol



Summary

S4P_BLK models a generalized four-port, S-parameter network. The full four-port S-parameters are specified.

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	X1
S11	S(1,1)	Scalar (Complex)	0.0 + j*0.0
S21	S(2,1)	Scalar (Complex)	0.0 + j*0.0
S31	S(3,1)	Scalar (Complex)	0.0 + j*0.0
S41	S(4,1)	Scalar (Complex)	0.0 + j*0.0
S12	S(1,2)	Scalar (Complex)	0.0 + j*0.0
S22	S(2,2)	Scalar (Complex)	0.0 + j*0.0
S32	S(3,2)	Scalar (Complex)	0.0 + j*0.0
S42	S(4,2)	Scalar (Complex)	0.0 + j*0.0
S13	S(1,3)	Scalar (Complex)	0.0 + j*0.0
S23	S(2,3)	Scalar (Complex)	0.0 + j*0.0
S33	S(3,3)	Scalar (Complex)	0.0 + j*0.0
S43	S(4,3)	Scalar (Complex)	0.0 + j*0.0
S14	S(1,4)	Scalar (Complex)	0.0 + j*0.0
S24	S(2,4)	Scalar (Complex)	0.0 + j*0.0
S34	S(3,4)	Scalar (Complex)	0.0 + j*0.0
S44	S(4,4)	Scalar (Complex)	0.0 + j*0.0
*Z1	Port 1 impedance	Scalar (Complex)	50.0 ohm
*Z2	Port 2 impedance	Scalar (Complex)	50.0 ohm
*Z3	Port 3 impedance	Scalar (Complex)	50.0 ohm
*Z4	Port 4 impedance	Scalar (Complex)	50.0 ohm
*ChkPassv	Switch "Do not Check passivity/Check passivity"		No

** indicates a secondary parameter*

Parameter Details

ChkPassv. If set to Yes, tests S4P_BLK for passivity and displays a warning if there is a passivity violation. The default setting is No (Do not check passivity).

Parameter Restrictions and Recommendations

Z1, Z2, Z3, and Z4 must not be equal to zero.

Layout

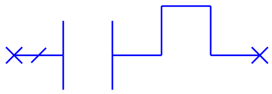
This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Recommendations for Use

The S4P_BLK model can be used to model any general four-port network (passive or active) by specifying the S-parameters.

Short-circuited Stub Resonator in Series with Capacitor: SCS

Symbol



Summary

SCS models a series connection of a capacitor and resonator made up of short-circuited segments of ideal transmission line.

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	TL1
C	Capacitance of shunt capacitor	Capacitance	1 pF
Z0	Characteristic impedance of short-circuited transmission line	Resistance	50 ohm
EL	Electrical length (phase length) at F0	Angle	90 degrees
F0	Frequency used to specify EL	Frequency	1 GHz

Parameter Details

EL and **F0**. These parameters determine the frequency-dependence of the electrical length of the line defined as: electrical length in radians = EL· freq/Fo ·π/180 where freq is the evaluation frequency.

Layout

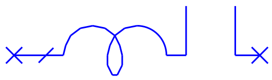
This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

Recommendations for Use

This element is often used for filter synthesis.

Open-circuited Stub Resonator in Series with Inductor: SLO

Symbol



Summary

SLO models a series connection of an inductor and resonator made up of open-circuited segments of ideal transmission line.

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	TL1
L	Inductance of series inductor	Inductance	1 nH
Z0	Characteristic impedance of open-circuited transmission line	Resistance	50 ohm
EL	Electrical length (phase length) at F0	Angle	90 degrees
F0	Frequency used to specify EL	Frequency	1 GHz

Parameter Details

EL and **F0**. These parameters determine the frequency-dependence of the electrical length of the line defined as: electrical length in radians = EL· freq/Fo ·π/180 where freq is the evaluation frequency.

Layout

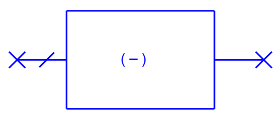
This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Recommendations for Use

This element is often used for filter synthesis.

2-Port Negation Based on Short-Open Calibration for De-embedding (Closed Form): SOC_DEMB

Symbol



Summary

SOC_DEMB allows you to manually de-embed parasitics from an EM simulation by simulating the structure with no de-embedding for the ports of interest. After this step, a second structure is created as a standard to de-embed port parasitics. You can then use a schematic with the SOC_DEMB element to properly remove the parasitics.

Parameters

Name	Description	Unit Type	Default
ID	Name	Text	N1
NET	Symmetric 3-port network used for negation	Text	"name"

Layout

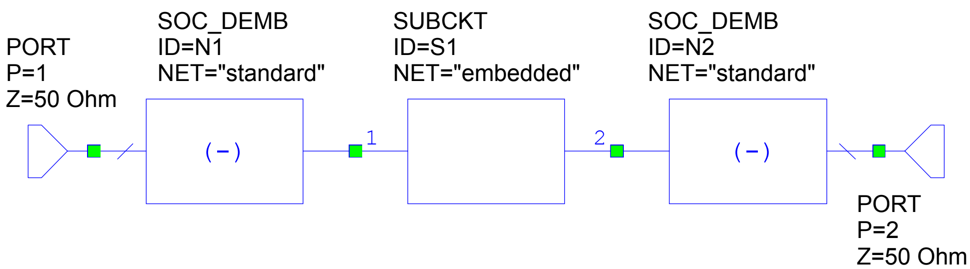
This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Implementation Details

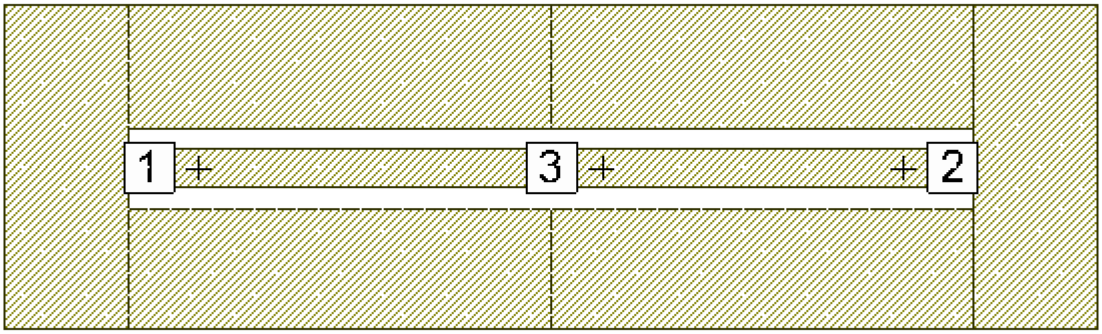
See the NI AWR Knowledge Base for examples on how to use these models for specific configurations of ports when using AXIEM.

When using the SOC_DEMB element, note the following limitations:

- When building the de-embedding schematic, you should connect port1 of the SOC_DEMB (the side with the "/" next to the node) to the port in the schematic, NOT to the subcircuit containing the embedded structure. The following figure shows the proper configuration.



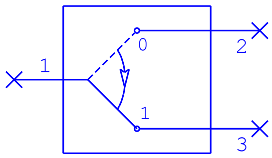
- When generating your standard for EM structures, the internal port should be the middle of the structure, and the positive side of the internal port should be on the same side as port 2 of the structure. The following figure shows the proper port configuration.



- The overall length of the structure should be 4x (2x on either side of the internal port) the line width or the substrate height, whichever is greater.
- For any ports de-embedded this way, make sure de-embedding is turned off for all ports in the de-embedding standard and those ports in the simulated structure.

Ideal Single Pole Double Throw Switch (Closed Form): SPDT

Symbol



Summary

The SPDT model allows you to implement an ideal single pole double throw switch whose position is determined by the STATE parameter. The BIT parameter allows you to control multiple switches using one STATE parameter by specifying the significant binary bit on which the switch operates.

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	S
STATE	Switch state		0
BIT	Defines On/Off bit of state		0

Parameter Details

STATE. An integer greater than or equal to zero which is converted to a binary number. For example, if STATE is equal to 14 in base 10, it is converted into 001110 in binary, or base 2.

BIT. An integer greater than or equal to zero which determines the significant bit the switch operates on 2^{BIT}. For example, if STATE is equal to 14 as given above, and BIT is equal to zero, then the switch is in the OFF state. However, if BIT is equal to 1, the switch is in the ON state.

Parameter Restrictions and Recommendations

1. STATE ≥ 0
2. $0 \leq$BIT ≤ 30

Implementation Details

$$S_{\mathrm{OFF}} = \begin{bmatrix} 0 & 1 & 0 \\ 1 & 0 & 0 \\ 0 & 0 & 1 \end{bmatrix}$$

$$S_{\mathrm{ON}} = \begin{bmatrix} 0 & 0 & 1 \\ 0 & 1 & 0 \\ 1 & 0 & 0 \end{bmatrix}$$

4-Bit Truth Table

STATEBase 10	STATEBASE 2	BIT=3	BIT=2	BIT=1	BIT=0
0	0000	OFF	OFF	OFF	OFF
1	0001	OFF	OFF	OFF	ON
2	0010	OFF	OFF	ON	OFF
3	0011	OFF	OFF	ON	ON
4	0100	OFF	ON	OFF	OFF
5	0101	OFF	ON	OFF	ON
6	0110	OFF	ON	ON	OFF
7	0111	OFF	ON	ON	ON
8	1000	ON	OFF	OFF	OFF
9	1001	ON	OFF	OFF	ON
10	1010	ON	OFF	ON	OFF
11	1011	ON	OFF	ON	ON
12	1100	ON	ON	OFF	OFF
13	1101	ON	ON	OFF	ON
14	1110	ON	ON	ON	OFF
15	1111	ON	ON	ON	ON

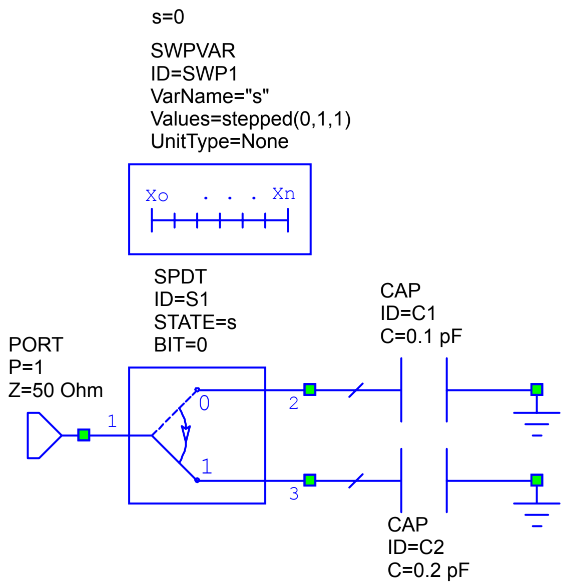
Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

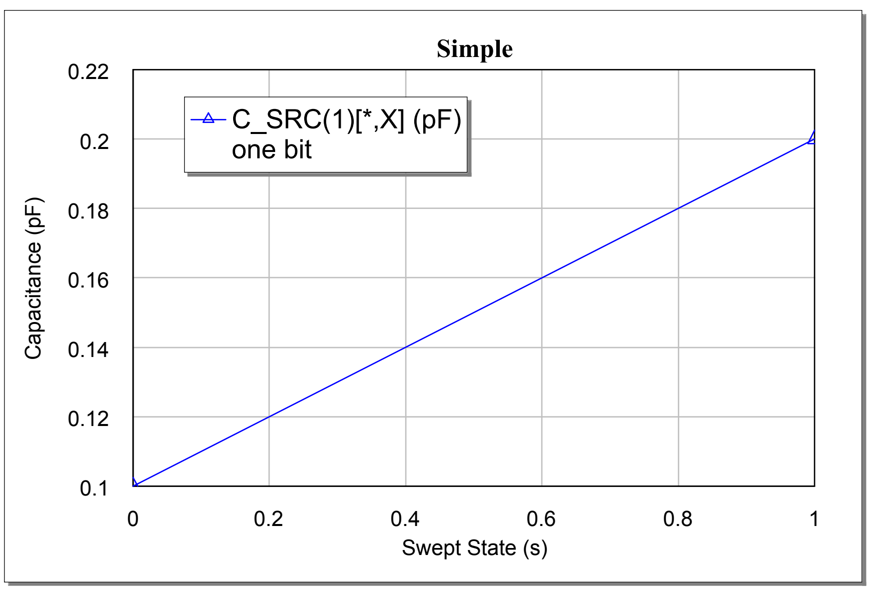
Recommendations for Use

You can use SPDT to create ideal digital components such as attenuators or phase shifters. See the phase shifter examples for how these are implemented in this type of design.

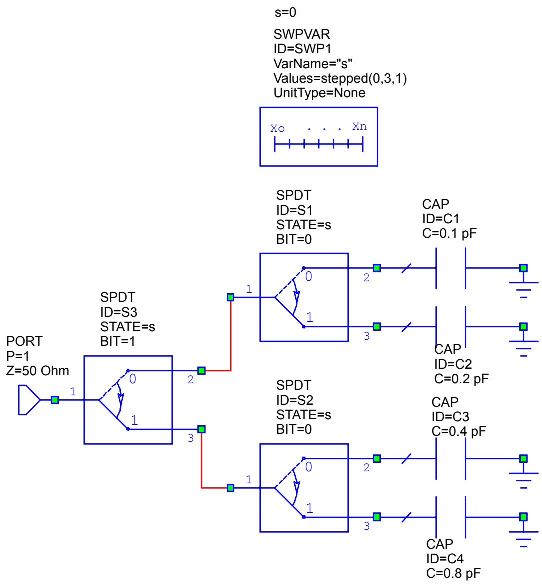
Several examples demonstrate how to properly use this element. The simplest case is where you want to do a simple switch between two paths. The following example shows switching between two different capacitor values.



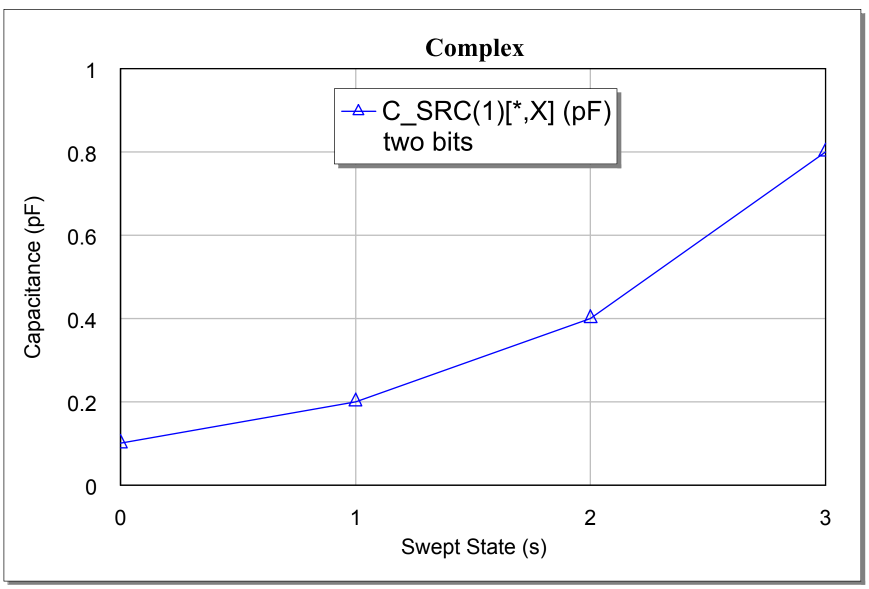
A swept variable is used to sweep the two possible states. Note that BIT is set to 0. Setting up the graph to use the swept value on the x-axis produces a graph similar to the following.



You can get more complex with the switches by using the BIT setting, for example, switching between four different capacitor values.

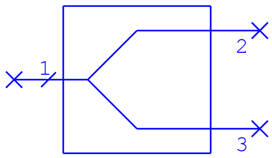


Note that the left-most switch has BIT set to 1 and the two right switches have BIT set to 0. Sweeping four different states easily produces the four possible paths through the switch.



2-Way Power Splitter (Closed Form): SPLIT2

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	P1
L21	Insertion loss from port 1 to port 2	dB	3 dB
L31	Insertion loss from port 1 to port 3	dB	3 dB
Z0	Port impedance	Resistance	50 ohm
*ChkPassv	Switch "Do not Check passivity/Check passivity"		No

** indicates a secondary parameter*

Parameter Details

ChkPassv. If set to Yes, tests SPLIT2 for passivity and displays a warning if there is a passivity violation. The default setting is No (Do not check passivity).

Implementation Details

A signal applied to port 1 is split between ports 2 and 3 with insertion losses given by L21 and L31, respectively. The losses are bi-directional and ports 2 and 3 are isolated. You can enter either positive or negative quantities for L21 and L31, but they are interpreted, in all cases, as losses. The following equations implement the behavior of this model:

$$\varepsilon = 1e-8$$

$$l_2 = 10^{-|L21|/20}$$

$$l_3 = 10^{-|L31|/20}$$

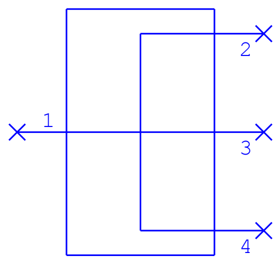
$$S = \begin{bmatrix} \varepsilon & l_2 & l_3 \\ l_2 & \varepsilon & \varepsilon \\ l_3 & \varepsilon & \varepsilon \end{bmatrix}$$

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

3-Way Power Splitter (Closed Form): SPLIT3

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	P1
L21	Insertion loss from port 1 to port 2	DB	4.77 dB
L31	Insertion loss from port 1 to port 3	DB	4.77 dB
L41	Insertion loss from port 1 to port 4	DB	4.77 dB
Z0	Port impedance	Resistance	50 ohm
*ChkPassv	Switch "Do not Check passivity/Check passivity"		No

** indicates a secondary parameter*

Parameter Details

ChkPassv. If set to Yes, tests SPLIT2 for passivity and displays a warning if there is a passivity violation. The default setting is No (Do not check passivity).

Implementation Details

A signal applied to port 1 is split between ports 2, 3, and 4 with insertion losses given by L21, L31, and L41, respectively. The losses are bi-directional and ports 2, 3 and 4 are isolated. You can enter either positive or negative quantities for L21, L31 and L41, but they are interpreted, in all cases, as losses. The following equations implement the behavior of this model:

$$\varepsilon = 1e-8$$

$$l_2 = 10^{-|L21|/20}$$

$$l_3 = 10^{-|L31|/20}$$

$$l_4 = 10^{-|L41|/20}$$

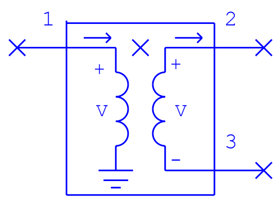
$$S = \begin{bmatrix} \varepsilon & l_2 & l_3 & l_4 \\ l_2 & \varepsilon & \varepsilon & \varepsilon \\ l_3 & \varepsilon & \varepsilon & \varepsilon \\ l_4 & \varepsilon & \varepsilon & \varepsilon \end{bmatrix}$$

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

Ideal Transformer for Converting a Single-ended Source to a Two-terminal Source: SRC_CONV

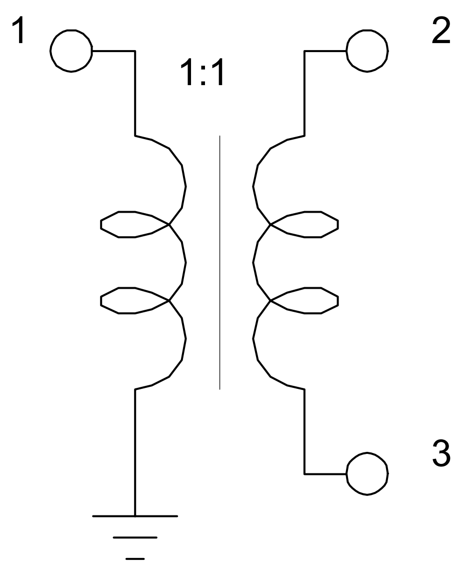
Symbol



Summary

SRC_CONV represents a single-ended to two-terminal source conversion element.

Equivalent Circuit



Implementation Details

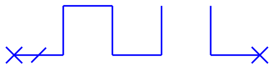
This element is equivalent to an ideal transformer with the one-input terminal grounded and 1:1 turn ratio.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Open-circuited and Short-circuited Stub Resonators Connected in Series: SSR

Symbol



Summary

SSR models a pair of series-connected resonators made up of open-circuited and short-circuited segments of ideal transmission line.

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	TL1
ZSC	Characteristic impedance of short-circuited transmission line	Resistance	50 ohm
ZOC	Characteristic impedance of open-circuited transmission line	Resistance	50 ohm
EL	Electrical length (phase length) at F0	Angle	90 degrees
F0	Frequency used to specify EL	Frequency	1 GHz

Parameter Details

EL and **F0**. These parameters determine the frequency-dependence of the electrical length of the line defined as: electrical length in radians = EL· freq/Fo ·π/180 where freq is the evaluation frequency. EL and F0 are common for both resonators.

Layout

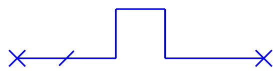
This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Recommendations for Use

This element is often used for filter synthesis.

Short-circuited Stub Resonator: SST

Symbol



Summary

SST models a resonator made up of a short-circuited segment of ideal transmission line.

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	TL1
Z0	Characteristic impedance of transmission line	Resistance	50 ohm
EL	Electrical length (phase length) at F0	Angle	90 degrees
F0	Frequency used to specify EL	Frequency	1 GHz

Parameter Details

EL and **F0**. These parameters determine the frequency-dependence of the electrical length of the line defined as: electrical length in radians = EL· freq/Fo ·π/180 where freq is the evaluation frequency.

Layout

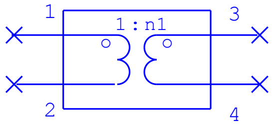
This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Recommendations for Use

This element is often used for filter synthesis.

Ideal Transformer Using Ymatrix Implementation: XFMR

Symbol



Parameters

Name	Description	Unit Type	Default
ID	ID	Text	X1
N	Sec: pri turns ratio		1

Implementation Details

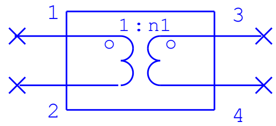
Implements an ideal transformer with primary coil between nodes 1 and 2 and secondary coil between nodes 3 and 4. The ratio of secondary to primary turns is given by N. "Ideal" means that DC is also transformed with this element. If you do not want to transform DC, you need to add capacitors on either side of the model to block DC.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Physical Transformer: XFMRP

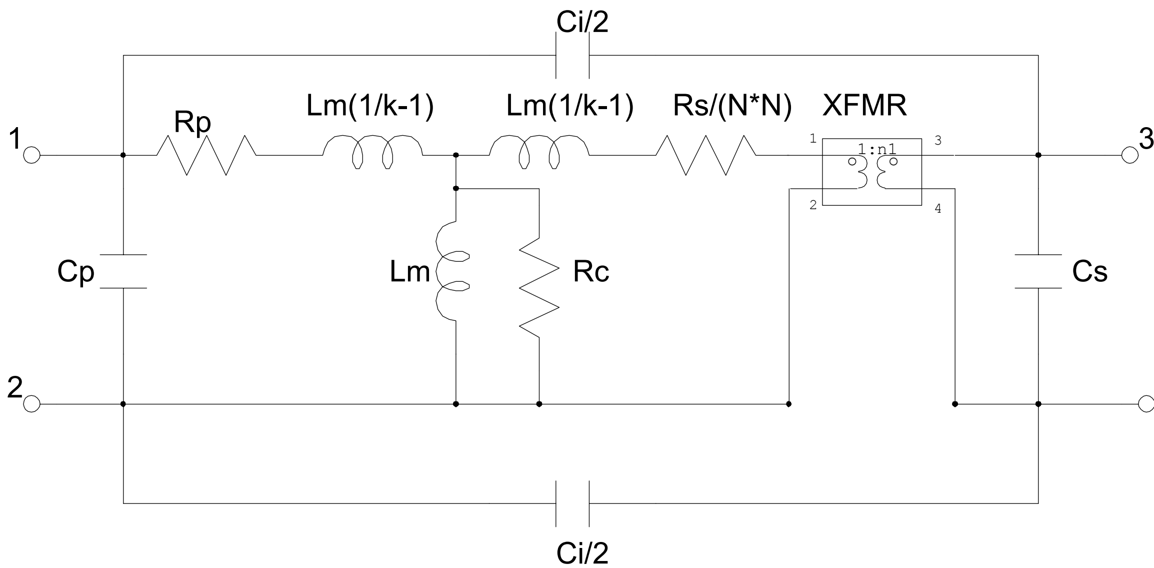
Symbol



Summary

XFMRP implements linear model of physical transformer. Model accounts for core and winding resistive losses as well as parasitic winding capacitances.

Topology



Parameters

Name	Description	Unit Type	Default
ID	ID	Text	X1
N	Turns ratio Secondary:Primary		1
Lm	Magnetizing inductance	Inductance	300 nH
Rc	Core loss resistance	Resistance	1000 Ohm
k	Coupling coefficient		0.9
Rp	Primary loss resistance	Resistance	10 Ohm
Rs	Secondary loss resistance	Resistance	10 Ohm
Cp	Primary capacitance	Capacitance	30 pF
Cs	Secondary capacitance	Capacitance	30 pF
Ci	Interwinding capacitance	Capacitance	10 pF

Parameter Details

N. N is the ratio of secondary to primary turns.

Lm. Magnetizing inductance of primary winding is defined as a product of primary open-circuit inductance Lp and coupling coefficient k: *Lm= Lp*k*.

Lm(1/k-1), Rs/(N*N) - See Topology. Series inductances in T-circuit are leakage inductances. Series resistance in right shoulder of T-circuit is equivalent resistance of secondary winding. See [1] for transformer equivalent circuit details.

Implementation Details

Implements physical transformer with primary coil between nodes 1 and 2 and secondary coil between nodes 3 and 4. Equivalent circuit of XMRFP is implemented after [1] with resistive losses and parasitic capacitances added. XMRFP incorporates model of ideal transformer XFMR so similar to XFMR DC is also transformed with this element. If you do not want to transform DC, you need to add capacitors on either side of the model to block DC.

Layout

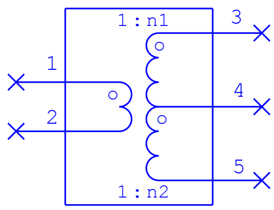
This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

References

[1] E. Brenner, M. Javid, Analysis of Electric Circuits, 2nd ed.: McGraw-Hill Book Company, New York, 1967, Ch. 15.

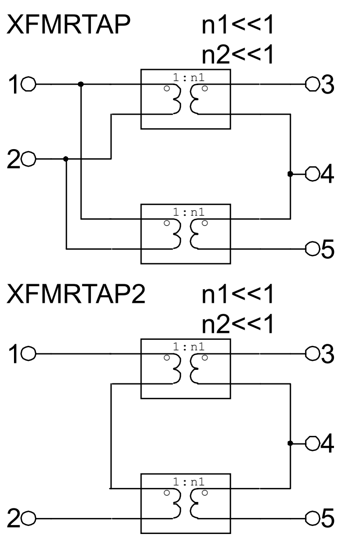
Ideal Transformer with Center Tap (Closed Form): XFMRTAP; XFMRTAP2

Symbol



Equivalent Circuits

The following figures display the difference between the XFMRTAP and XFMRTAP2 models.



The major difference between the XFMRTAP and XFMRTAP2 models is the manner in which the primary coil affects the secondary coils. In the XFMRTAP model, the entire primary coil is seen by both of the secondary coils, so changing the turn's ratio for one coil does not affect the voltage at the other coil. In this case, it is equivalent to three coils in parallel, not really a center tap at the output coil. In the XFMRTAP2 model, a fraction of the input coil is seen by the two secondary coils, so changing the turn's ratio for one coil affects the output voltage of both coils.

You can change the turn's ratios and observe what happens in both scenarios, then use the transformer that is appropriate for your application.

Parameters

Name	Description	Unit Type	Default
ID	ID	Text	X1
N1	Sec1:pri turns ratio		1
N2	Sec2:pri turns ratio		1

Implementation Details

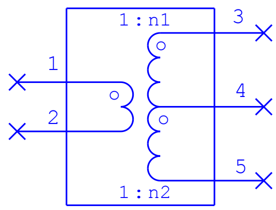
Implements an ideal transformer with primary coil between nodes 1 and 2 and two secondary coils between nodes 3 and 4 and between nodes 4 and 5. The ratio of secondary to primary turns is given by N1 and N2. "Ideal" means that DC is also transformed with this element. If you do not want to transform DC, you need to add capacitors on either side of the model to block DC.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Ideal Transformer with Center Tap (with Finite Inductance and Leakage): XFMRTAP3

Symbol



Summary

XFMRTAP3 models a transformer with a tap at the secondary winding. XFMRTAP3 differs from XFMRTAP in accounting for inductance of the primary winding and magnetic flux leakage.

Parameters

Name	Description	Unit Type	Default
ID	ID	Text	X1
N1	Turns ratio Secondary winding 1 : Primary		1
N2	Turns ratio Secondary winding 2 : Primary		1
Lp	Inductance of primary winding (open-circuit)	Inductance	300 nH
k	Coupling coefficient		0.9

Implementation Details

XFMRTAP3 implements an ideal transformer with a primary coil between nodes 1 and 2 and two secondary coils between nodes 3 and 4 and between nodes 4 and 5. The ratio of secondary to primary turns is given by N1 and N2. This model implies that windings have finite inductances and all winding are magnetically coupled with a coupling coefficient between each pair of windings equal to the value of parameter k.

DC is also transformed with this element. If you do not want to transform DC, you need to add capacitors on either side of the model to block DC.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Nonlinear Behavioral Model Compatible with Agilent's X-parameters® (from file): XPARAM

Symbol

Summary

XPARAM models nonlinear behavioral models compatible with Agilent's X-parameters.

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	X1
N	Number of ports		2
FILE	File name	Text	
*INTERP	Interpolation method		Linear
*INTERPDOM	Interpolation domain		RI
*EXTRAP	Extrapolation method		Interp method
DIR	Directory to search for file		Project

** indicates a secondary parameter*

Parameter Details

FILE. May contain a relative or absolute file path. Relative paths are relative to the directory specified by the DIR parameter.

INTERP. Specifies the method used to interpolate the data specified in the file.

INTERPDOM. Specifies the form in which the data is interpolated. When set to "Rectangular" the real and imaginary components of the data are interpolated separately. When set to "Polar" the interpolation is applied to the magnitude and angle of the data.

EXTRAP. Specifies the method used to extrapolate the data specified in the file. When set to "Interp Method", extrapolation uses the method specified by INTERP. When set to "Constant" the lowest value data point is used for any value below the range specified in the file, and the highest value data point is used for any value above it.

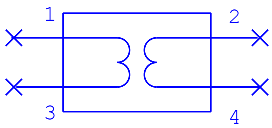
DIR. When FILE has a relative path, this parameter specifies to which directory the path is relative. The "Project" directory is where the project file is stored. The "Library" directory is specified in the Environment Options dialog box (choose **Options > Environment Options**). The "EM Models" directory location displays in the Directories dialog box (choose **Help > Show Files/Directories**).

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Transmission Line Transformer on Ferrite Core (Closed Form): XFERTL1

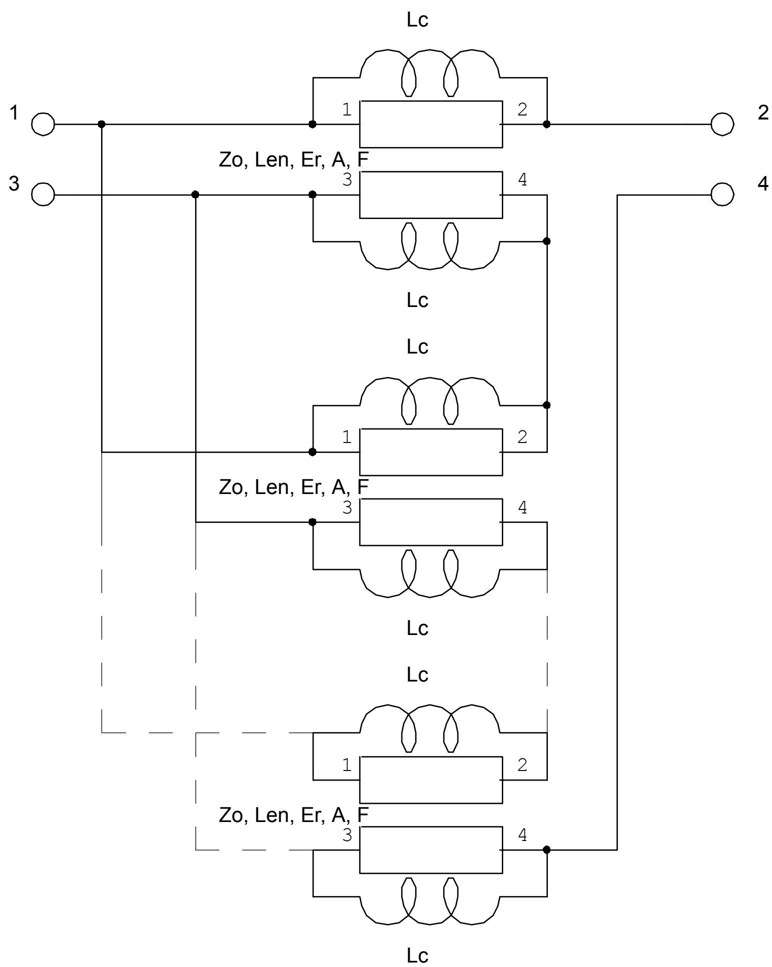
Symbol



Summary

XFERTL1 is a behavioral model of a Guanella Balun transformer. This transformer consists of several Tx lines wound on ferrite core. Inputs of all lines are connected in parallel, and outputs are connected in series. This Tx line transformer provides highly efficient impedance matching within a relatively wide frequency bandpass.

Equivalent Circuit



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	XU#
Zo	Char. imped of Tx line	Resistance	50 ohm
Len	Physical length of Tx line	Length	L1
Er	Effective dielectric constant of Tx line		1
A	Attenuation constant of Tx line (dB/m)		0
F	Frequency for scaling of Tx line attenuation	Frequency	0 GHz
N	Number of turns of line around the ferrite core		1
AL	Inductive index (inductance/turn)[a])	Inductance	1 nH
Order	Number of Tx lines		1

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory.

Implementation Details

XFERTL1 is comprised of several four-port two-conductor Tx lines. The number of lines is equal to Order. Inputs of all lines are connected in parallel, outputs are connected in series. Characteristic parameters of each line are specified via XFERTL1 parameters. Conductors of each Tx line have choke inductances Lc attached in parallel (see "Equivalent Circuit"). These inductances actually account for the presence of ferrite core. To define Lc, you must specify the number of turns (of Tx line winding on ferrite core) and inductance per turn. The resulting transformer provides transformation ratio N = $Order^2$:1.

The choking inductance, Lc, is

$$L_C = N^2 \cdot \mathrm{AL}$$

The attenuation A(F) has the following frequency dependence:

$$A(F) = \begin{cases} A & F = 0 \\ A \cdot \sqrt{\frac{f}{F}} & F > 0 \end{cases}$$

where f is a simulation frequency.

Layout

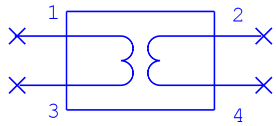
This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

References

[1] Jerry Sevick, *Transmission Line Transformers*, 4th Ed., SciTech Publishing, 2001.

Transmission Line Transformer in Ferrite Sleeve (Closed Form): XFERTL2

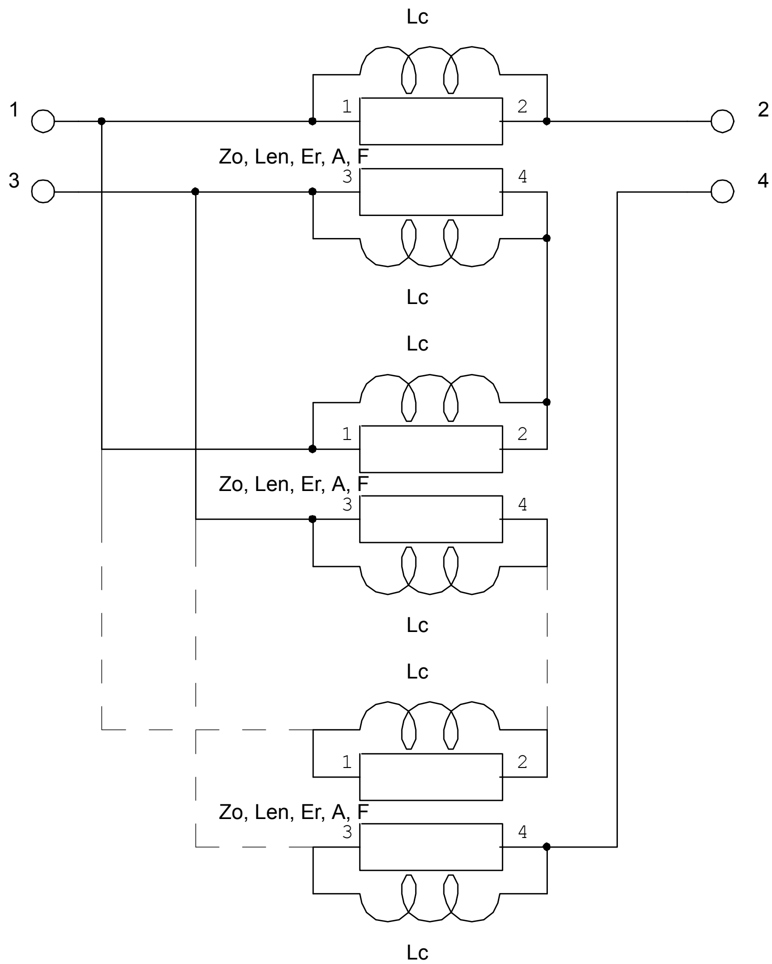
Symbol



Summary

XFERTL2 is a behavioral model of a Guanella Balun transformer. This transformer consists of several Tx lines wound in ferrite sleeve. Inputs of all lines are connected in parallel, and outputs are connected in series. This Tx line transformer provides highly efficient impedance matching within a relatively wide frequency bandpass.

Equivalent Circuit



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	XU#
Zo	Char. imped of Tx line	Resistance	50 ohm
Len	Physical length of Tx line	Length	L1
Er	Effective dielectric constant of Tx line		1
A	Attenuation constant of Tx line (dB/m)		0
F	Frequency for scaling of Tx line attenuation	Frequency	0 GHz
Mu	Relative permeability of ferrite sleeve		1
L	Inductance per unit length of Tx line without sleeve[a])	Inductance	1 nH
Order	Number of Tx lines		1

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory.

Implementation Details

XFERTL2 is comprised of several four-port two-conductor Tx lines. The number of lines is equal to Order. Inputs of all lines are connected in parallel, outputs are connected in series. Characteristic parameters of each line are specified via XFERTL2 parameters. Conductors of each Tx line have choke inductances Lc attached in parallel (see "Equivalent Circuit"). These inductances actually account for the presence of ferrite sleeve. To define Lc, you must define the relative permeability of the ferrite sleeve, Tx line length, and Tx line inductance (without sleeve) per unit length. The resulting transformer provides transformation ratio N = Order^2:1.

The choking inductance, Lc, is

$$L_c = \text{Mu} \cdot L \cdot \text{LEN}$$

The attenuation A(F) has the following frequency dependence:

$$A(F) = \begin{cases} A & F = 0 \\ A(F) \cdot \sqrt{\dfrac{f}{F}} & F > 0 \end{cases}$$

where f is a simulation frequency.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

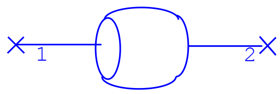
References

[1] Jerry Sevick, *Transmission Line Transformers*, 4th Ed., SciTech Publishing, 2001

Interconnects

Three-Layer Model of Ball Grid Array (BGA): BGA

Symbol



Summary

The Ball Grid Array package (BGA) is modeled as a rectangular grid of electromagnetically coupled vias arranged inside a three-layer dielectric stack. Each via connects at its bottom end to a solder ball landed onto the printed circuit board (PCB). All vias and all solder balls are identical. The upper two layers (signal layer and power layer) are backed by perfectly conducting ground planes perforated by round apertures (antipads) that allow vias to go through. The bottom dielectric layer (solder ball layer) is a composite layer comprised of the PCB dielectric layer, the layer above the PCB containing solder balls, and a junction layer that contains a transition between via and solder ball. The PCB layer is backed by a perfect ground plane.

This model accounts for via-to-via and ball-to-ball electromagnetic coupling. The modeling technique assumes that layers are laterally unlimited, namely, the effect of lateral boundaries is not included. Via analysis is based on the theory of long cylindrical antenna excited by a frill of magnetic current radiating in a parallel plate waveguide and on the theory of monopole, radiating into a parallel plate waveguide. This model also implements quasi-static FEM analysis for accurate evaluation of capacitances of complex conducting configurations in a multilayered dielectric.

Input (odd numbered) ports are located at the centers of via capture pads, and output (even numbered) ports are located at the centers of the bottom ends of solder balls.

BGA is a dynamic model, that is, its symbol redraws and its number of nodes changes as one changes the number of package pins.

Topology

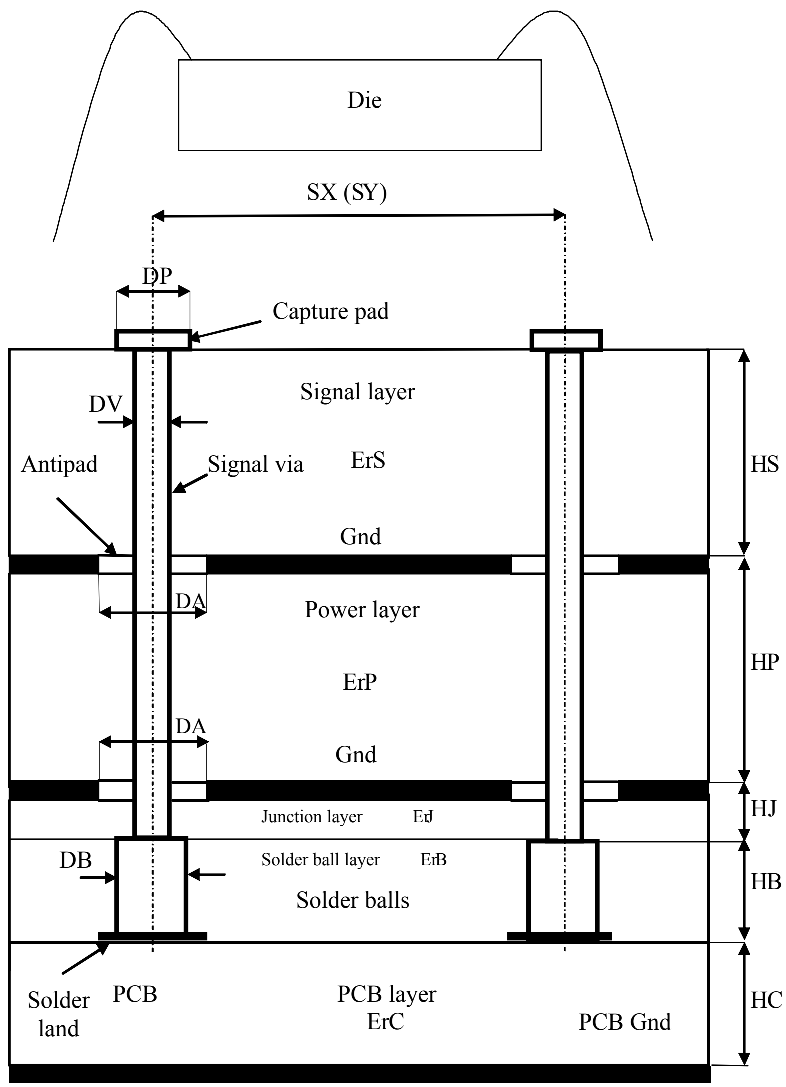


Figure 1. Cross-section

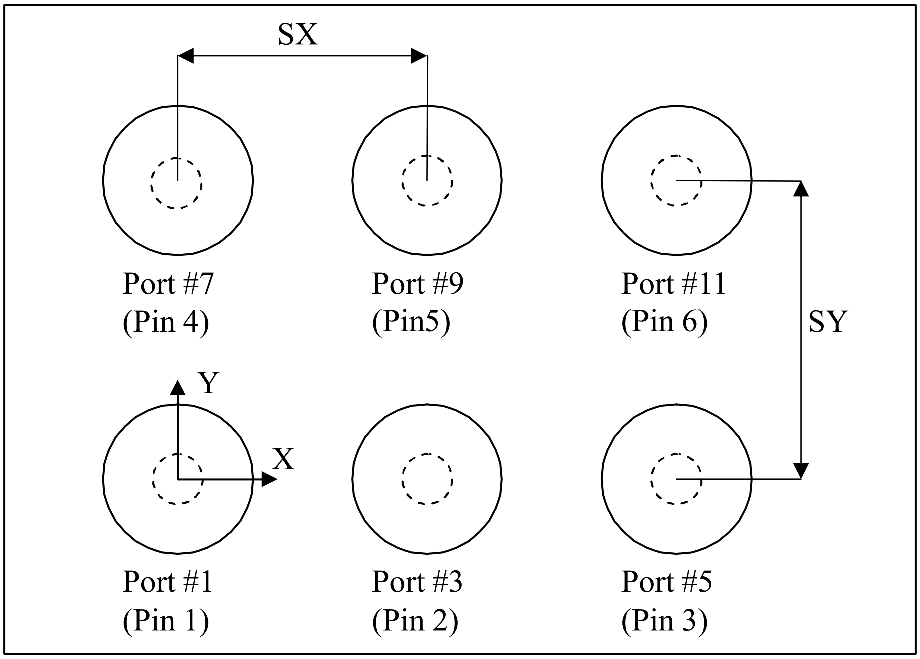


Figure 2. Top

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	PKG1
NX	Number of pins alongside X-side of package (number of columns)		1
NY	Number of pins alongside Y-side of package (number of rows)		1
SX	Pin pitch along X-side of package	Length	0.9 mm
SY	Pin pitch along Y-side of package	Length	0.9 mm
DP	Diameter of capture pad	Length	0.4 mm
TP	Thickness of capture pad	Length	0.01 mm
DA	Diameter of antipad	Length	0.6 mm
DV	Diameter of via	Length	0.2 mm
DB	Diameter of solder ball	Length	0.8 mm
HS	Height of signal layer	Length	0.4 mm
HP	Height of power layer	Length	0.4 mm
HJ	Height of junction layer	Length	0.4 mm
HB	Height of solder ball	Length	0.8mm
HC	Height of PCB layer	Length	0.4 mm

Name	Description	Unit Type	Default
Ers	Relative dielectric constant of signal layer		1
Erp	Relative dielectric constant of power layer		1
Erj	Relative dielectric constant of junction layer		1
Erb	Relative dielectric constant of solder ball layer		1
Erc	Relative dielectric constant of PCB layer		1
HP	Height of power layer	Length	0.4 mm
Rho	Via metal bulk resistance relative to gold		1

Parameter Details

SX, SY. These parameters are pitches of solder balls/vias measured between respective centers. (see Figure 1. Top in "Topology" section).

Parameter Restrictions and Recommendations

1. This model implies that package pins fully fill an equidistant rectangular grid, thus, packages that have a pin layout with a void central area (for instance, cavity down BGA) cannot be successfully modeled by this model version.
2. This model assumes that only dominant mode TMo propagates in all layers. Heights HP and HJ+HB+HC are checked for normal operation below the cutoff frequency of a higher mode propagating in respective parallel-plate waveguide. If the frequency exceeds the cutoff threshold then this model generates an error and reports the highest sweep frequency allowed at the specified distance between ground planes

Implementation Details

Vias are modeled as a set of segments contained within respective layers. Segments are electromagnetically coupled in each layer that supports single propagating TMo mode. Each segment is modeled as a short dipole radiating between perfectly conducting parallel plates. Port reference planes are located at the centers of capture pads and at the centers of the bottom tips of solder balls.

Recommendations for Use

This model assigns the port numbers to pins counting from the left bottom corner and moving from the left to the right along the rows. After reaching the last pin in a row the counting moves up to the next row and starts from the leftmost pin. See Figure 1. Top in "Topology" section.

This model uses a disk cache. This means that if a project contains multiple instances of identical BGA only one instance simulates, saves results to cache and all identical (that is, having the same parameter set) BGA simply fetch these results from cache. Note that the cache keeps saved data and any project on the same computer that contains BGA with the same set of parameters will reuse the cached data.

Layout

The layout for this cell has hard-coded model layers. When you first use this layout cell, a layer named "BGA" is added to your drawing layer and model layer list (if not already there). Using the model layer mapping, you can assign these layers to draw on any drawing layer. .

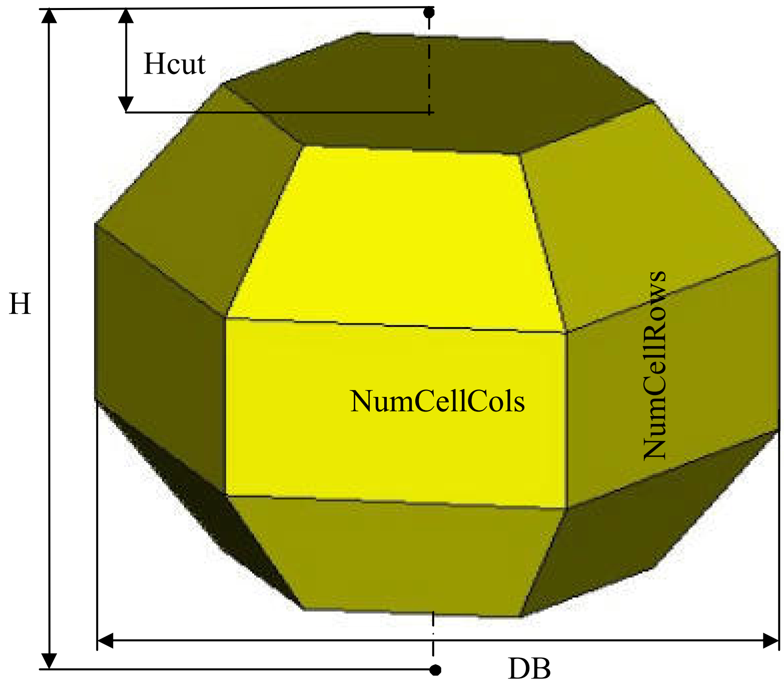
3D EM Layout

This element has a layout cell specifically for Analyst 3D EM layouts. See "Using 3D EM Elements " for details on using 3D pCells with Analyst.

This model has several layout only parameters that control how the 3D pCell draws. You access these parameters by selecting the item in the layout, right-clicking and choosing **Shape Properties** to display the Cell Options dialog box, then clicking the **Parameters** tab.

Name	Value	Units	Description
CapCutRatio	0.15	Real	Ratio of ball cut (from top and bottom) to vertical semi-axis, maximum = 0.8
OblSpheroidRatio	1	Real	Oblate spheroid shape ratio of vertical axis to horizontal axis, maximum = 1
NumCellRows	3	Integer	Number of azimuthal flat cell rows, minimum = 3
NumCellCols	6	Integer	Number of meridional flat cell columns, minimum = 6

3D EM Layout Parameter Details



The BGA solder ball is approximated by a polyhedron inscribed into an oblate spheroid (see the previous figure). On this figure, black points above and below the structure represent the North and South poles of the spheroid. By default, the approximating polyhedron is a regular polyhedron inscribed in a sphere. The shape of the oblate spheroid is controlled by the OblSpheroidRatio=H/DB parameter. This ratio must be equal to or less than 1 and greater than 0.01; the default value is 1 (sphere).

The spheroid can be horizontally cut from the top and bottom (simultaneously). The size of this cut is controlled by the CapCutRatio=Hcut/(0.5H) parameter. This ratio must be equal to or less than 0.8 and greater than 0.01; the default value is 0.15.

The approximating polyhedron is controlled by the NumCellRows and NumCellCols parameters. NumCellRows defines the number of azimuthal rows of flat cells and NumCellCols defines the number of meridional columns of flat cells. NumCellRows must be equal to or greater than 3; the default value is 3. NumCellCols must be equal to or greater than 6; the default value is 6. To provide better spheroid approximation use NumCellRows and NumCellCols in 10-20 range.

Note that only small selection of model parameters (and each one of layout shape parameters) effect 3D EM Layout. Height of ball H is not specified as parameter: Model calculates it as OblSpheroidRatio*DB. Same about Hcut: It is evaluated as 0.5*CapCutRatio*H.

Model parameters that effect 3D EM layout

NX, NY, SX, SY, DB, Rho

Layout shape parameters

CapCutRatio, OblSpheroidRatio, NumCellRows, NumCellCols

References

[1] Qizheng Gu, Y. Eric Yang, and M.Ali Tassoudji , "Modeling and Analysis of Vias in Multilayered Integrated Circuits," *IEEE Trans. on Microwave Theory and Tech.*, vol. 41, February 1993, pp. 206-214

[2] Qizheng Gu, M.Ali Tassoudji et al, "Coupled Noise Analysis for Adjacent Vias in Multilayered Digital Circuits," *IEEE Trans. on Circ. and Syst.*, vol. 41, December 1994, pp. 796-804

[3] Ryosuke Ito, R.W. Jackson, T. Hongsmatip, "Modeling of Interconnects and Isolation Within a Multilayered Ball Grid Array Package," *IEEE Trans. on Microwave Theory and Tech.*, vol. 47, September 1999, pp. 1819-1825

[4] B.Tomasic and A. Hessel, "Linear Array of Coaxially Fed Monopole Elements in a Parallel Plate Waveguide," *IEEE Trans. on Antennas and Prop.*, vol. 36, April 1988, pp. 449-462

[5] M. Goldfarb and R. Pucel, "Modeling Via Hole in Microstrip," *IEEE Microwave and Guided Wave Lett.*, vol. 1, June 1991, pp. 135-137

[6] David Meeker home page: http://femm.foster-miller.net/~dmeeker/ Follow this link for information: http://femm.foster-miller.net/Archives/readme.htm FEMM Manual: http://femm.foster-miller.net/Archives/doc/manual.pdf

Bond Wire Model (EIA/JEDEC Standard No. 59): BWIRES

Symbol

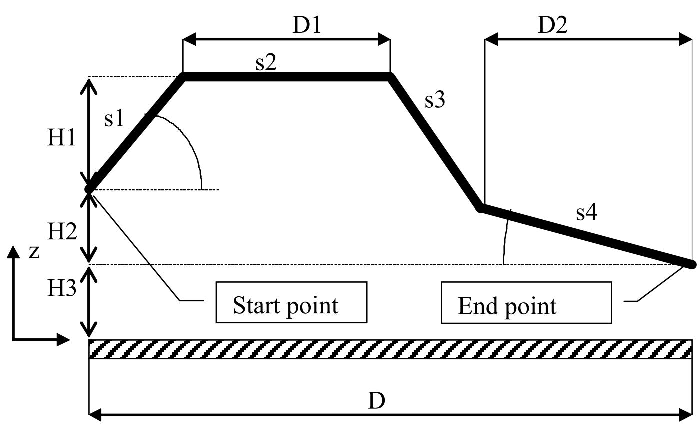
Summary

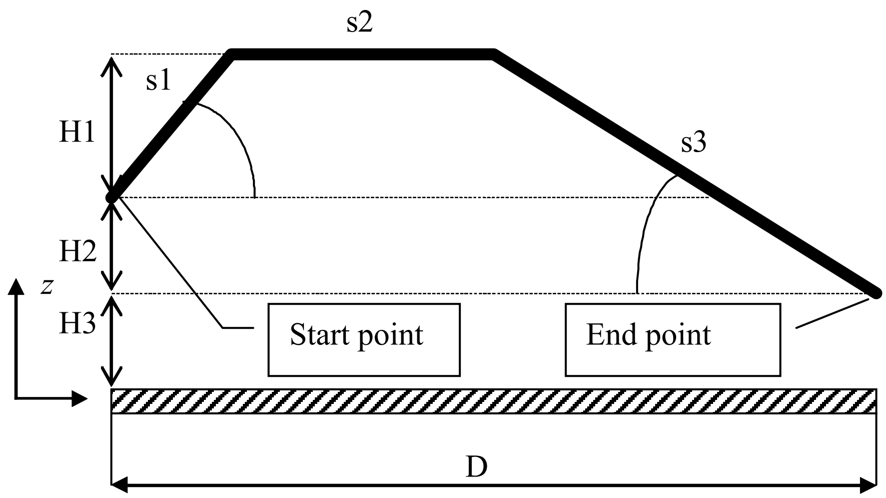
BWIRES models a collection of metallic round bonding wires (bond wires) made of linear segments each, and arranged above a conducting ground plane. Segments that comprise a wire are coplanar and the plane they belong to is assumed to be orthogonal to the ground plane. The actual smooth contour of a bonding wire is approximated by three or four linear segments and parameters that define orientation of segments are selected in compliance with standard EIA/JEDEC Standard No.59.

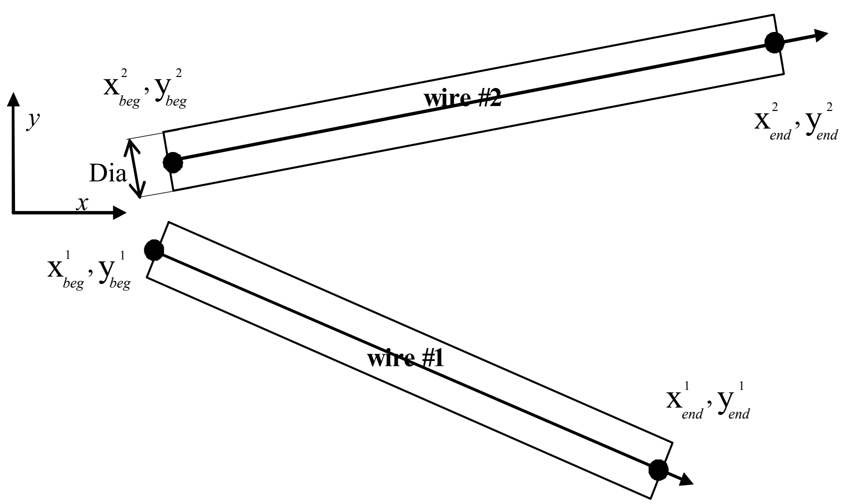
BWIRES implies that all bond wires share same shape and elevation above the ground plane. Multi-level and multi-shape arrangement is provided by model BWIRES2 .

BWIRES is a dynamic model, that is, its symbol redraws and its number of nodes changes as you change the number of wires.

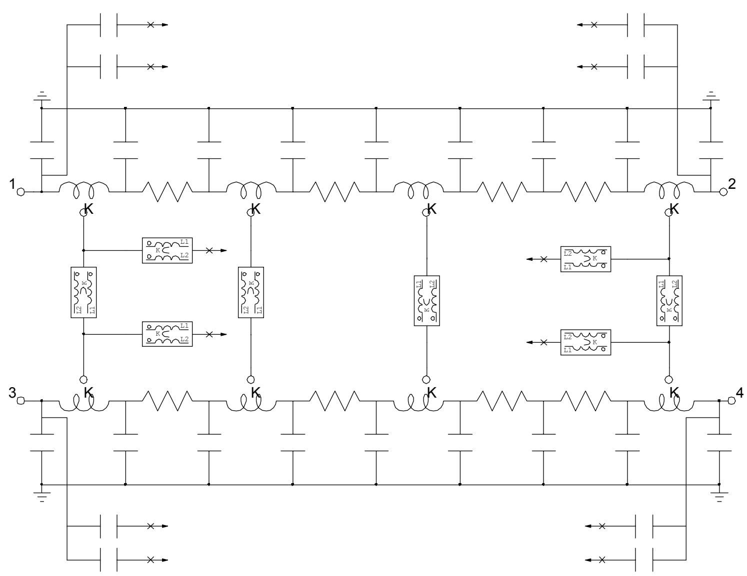
Topology




s2
s1
H1
s3
H2
z
H3
Start point
End point
D


x^2_{beg}, y^2_{beg}
wire #2
y
x^2_{end}, y^2_{end}
Dia
x
x^1_{beg}, y^1_{beg}
wire #1
x^1_{end}, y^1_{end}

Equivalent Circuit



Only part of a full equivalent circuit for N=2 is presented due to the complex nature of multiple interconnects. The actual equivalent circuit includes inductive and capacitive coupling between all segments of each wire.

Parameters

Name	Description	Unit Type	Default
ID	Element ID		IC1
N	Number of wires		1
Dia	Wire diameter	Length	%T[a]
Rho	Metal resistivity relative to gold		1
*WModel	Mode switch: 4 segments/3 segments		4 segments
*Alpha	Angle from first segment (s1) to the ground plane	Degree	90
*Beta	Angle from last segment (s4/s3) to the ground plane	Degree	45
*H1	Height from beginning point to top	Length	%W[a]
*H2	Height from end point to beginning point	Length	%W[a]
*H3	Height of end point over the ground plane	Length	%W[a]
*D1_Ratio	Ratio of segment #2 length to total wire ground footprint		0.125
*D2_Ratio	Ratio of segment #4 ground footprint to total wire ground footprint		0.5
*P0	X,Y coordinates of beginning points (vector of length N)	Length	{0,0,0,1e-6}
*P1	X,Y coordinates of end points (vector of length N)	Length	{5e-6,0,5e-6,1e-6}
*IndModel	Switch: RLC/RL model		RLC

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory.

** indicates a secondary parameter*

Parameter Details

Alpha, Beta, H1, H2, H3, D1_Ratio, D2_Ratio. These parameters are "shape parameters" that define the shape of the bond wire. All wires defined by BWIRES share the same shape parameters.

D1_Ratio, D2_Ratio. Parameters are unitless. They are defined as ratios D1/D and D2/D correspondingly (see the "Topology" section).

WModel. This switch allows switching between two modes: "4 segment" (default) and "3 segment" (see the "Topology" section). In "4 segment" mode, the model accepts, checks and uses all model shape parameters. In "3 segment" mode, the model does not check and does not use parameters D1_Ratio, and D2_Ratio. The latter parameters are excessive because shape parameters Alpha, Beta, H1, H2, and H3 unambiguously define the shape of a 3-segment bond wire.

P0, P1. These parameters are vectors. P0 contains pairs of (x, y) coordinates of the start point of each wire; P1 contains pairs of (x,y) coordinates of the end point of each wire. For example, the order of entries in vector P0 for N=2 is the following: {x1,y1,x2,y2} where x1 is x-coordinate of the start point of wire #1, y1 is y-coordinate of the start point of wire #1, x2 is x-coordinate of the start point of wire #2, and y2 is y-coordinate of the start point of wire #2.

The end coordinates of vector P1 are not required to be coordinates on wire end points - they can be located anywhere on the line that specifies azimuthal position of the respective wire plane.

Note that BWIRES expects the number of P0, P1 entries to be equal to 2N, and issues an error message if these numbers differ.

IndModel. This switch allows switching between two representations of the model equivalent circuit: "RLC" (Resistor-Inductor-Capacitor) and "RL" (Resistor-Inductor). Setting IndModel to "RL" **significantly reduces computation time** excluding capacitances from the equivalent circuit. At relatively low frequencies (say, below 10 GHz) both values of IndModel yield close modeling results. The difference between modeling results depends on the proximity of the evaluation frequency to the self-resonance frequency (SRF) of tested wire. A model at the "RL" setting cannot predict the location of SRF, while a model at the "RLC" setting (default) is "SRF aware" and predicts SRF reasonably accurately (see the "Recommendations for Use" section).

Parameter Restrictions and Recommendations

1. This model implies that the number of bond wires is less than or equal to 512.
2. The height of the end point over the ground H3 may be set equal to zero, allowing bond wire to touch ground.
3. When possible, wire should retain the recommended convex upward profile. You should not attempt to create straight, particularly horizontal wires. For straight wires with 3D EM pCells, use the BWIRES3 model with the BWShape parameter set to 2D and the shape parameter OrtTip set to 1.
4. If WModel = "3 segment", parameters D1_Ratio and D2_Ratio are not used and not checked. Wire shape is fully defined by H1, H2, Alpha, and Beta.
5. If WModel = "4 segment", ensure that D1_Ratio+D2_Ratio<=1. If this condition is not met, the wire contour gets ragged, and in certain configurations non-adjacent segments may even cross each other.
6. Alpha <135 ^{o} , Beta<=90 ^{o}. If WModel = "3 segment", the layout cell sets caps on parameters H1, Alpha, and Beta to prevent creation of twisted/self-crossing wire profiles.
7. This model subjects shape parameters (see the "Parameter Details" section) and locations of wire end points (P0, P1) to extended design rule control that prevents segments within separate wires and wires themselves from touching and intersecting (i.e., from getting closer than Dia). Model also provide control of wire shape and tries to disallow spikes,

that is, too small acute angles between adjacent segments. For example, model checks if length of any segment exceeds three wire footprints: it means that some segments are long and almost vertical. Models also checks if junction of segments #3 and #4 is elevated too high above segment #2: This means that angle between segments #3 and #4 is acute and very small. User can constantly monitor actual shape of wire on layout 3D view window.

8. Dia>0.1 micron

Implementation Details

Wires are considered a collection of infinitely thin, arbitrarily oriented conducting segmented filaments situated above infinite perfect ground. The actual diameter and material of the wire are accounted for at the evaluation of self-characteristics of each segment; mutual parameters neglect diameter and material. This model does not solve for distribution of charges/currents along/over wires; the approach is based on the assumption that the method of static-averaged potentials is applicable in this case.

This model accounts for internal and external inductance of each wire segment; its evaluation of resistances is based on a skin-effect approach and the radiation resistance of a small dipole above ground. Evaluation of capacitances implies that the surrounding media is a vacuum and neglects the electrical properties of adjacent dielectric objects.

Recommendations for Use

User discretion is advised if the following three factors join together:

- The evaluation frequency is very high (say, above 50 GHz)
- The wires are relatively long (above 1 mm)
- The low profile wires span a substrate of relatively high permittivity (~8 and above). Even in this case, SRF is usually predicted well within 5-7% error; however, the error in phases of "crosstalk" scattering coefficients might be large due to additional unaccounted substrate coupling.

Setting the IndMode parameter to "RL" substantially reduces calculation time (this may be advantageous if the number of bond wires exceeds 20), but excludes effects of capacitances coupling (these effects are responsible for self-resonances at higher frequencies). "RL" leaves in place frequency-dependence of self-inductances; radiation resistance also remains added to the self-resistance of each wire.

User discretion is also advised when the total length/footprint of wire is small. Evidently, short wires have even shorter segments. At the same time, the model assumes that each segment is represented by infinitely thin filament, which means Lseg >> Dia. The model issues a warning if Lseg<6*Dia.

We have compared our Bondwire model to other commercially available software which accounts for all loss and radiation, and found that we have a good match using 48 wires up to 1mm (roughly 40mils) long and up to 100GHz. A safe rule of thumb to follow is Length of wire / Wavelength ratio should be less than 1/6.

Layout

The layout for this cell has hard-coded model layers. When you first use this layout cell, a layer named "BondWires" is added to your drawing layer and model layer list (if they are not already there). Using the model layer mapping, you can assign these layers to draw on any drawing layer. The via is drawn the size of the "D1" parameter.

This model has several layout-only parameters that control how the pad draws. You access these parameters by selecting the item in the layout, right-clicking and choosing **Shape Properties** to display the Cell Options dialog box, then clicking the **Parameters** tab.

Name	Value	Units	Description
ShowNodeNum	0	Integer	Layout displays node numbers only if this value is non-zero
PointFaces	0	Integer	Faces are points, not line segments, only if this value is non-zero

3D EM Layout

This element has a layout cell specifically for Analyst 3D EM layouts. See "Using 3D EM Elements " for details on using 3D pCells with Analyst.

References

[1] Grover, F.W., Inductance Calculations, Mineola, New York: Dover Publishing, 2004

[2] Bahl, I.J., Lumped Elements for RF and Microwave Circuits, Norwood, MA: Artech House, 2003

[3] Balanis C.A., Antenna Theory: analysis and design, 2nd edition: Wiley, 1997

[4] Bond Wire Modeling Standard. EIA/JEDEC Standard EIA/JESD59: Electronics Industries Association, June 1997.

Multilevel Bond Wire Model (EIA/JEDEC Standard No. 59): BWIRES2

Symbol

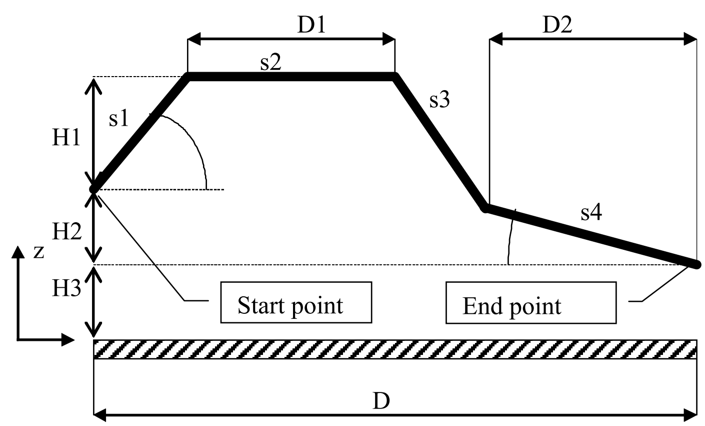
Summary

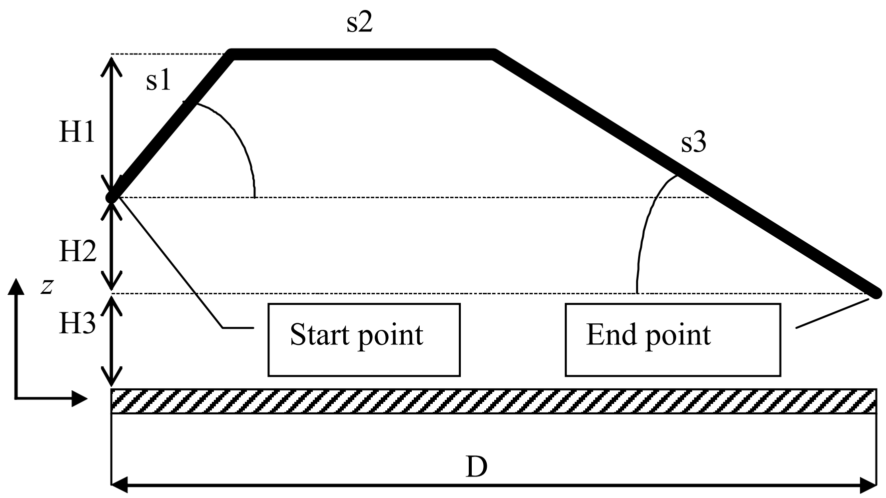
BWIRES2 models a collection of metallic round bonding wires (bond wires) made of linear segments each, and arranged arbitrarily above a conducting ground plane. Segments that comprise a wire are coplanar, and the plane they belong to is assumed to be orthogonal to the ground plane. The actual smooth contour of a bonding wire is approximated by three or four linear segments, and parameters that define orientation of segments are selected in compliance with standard EIA/JEDEC Standard No.59.

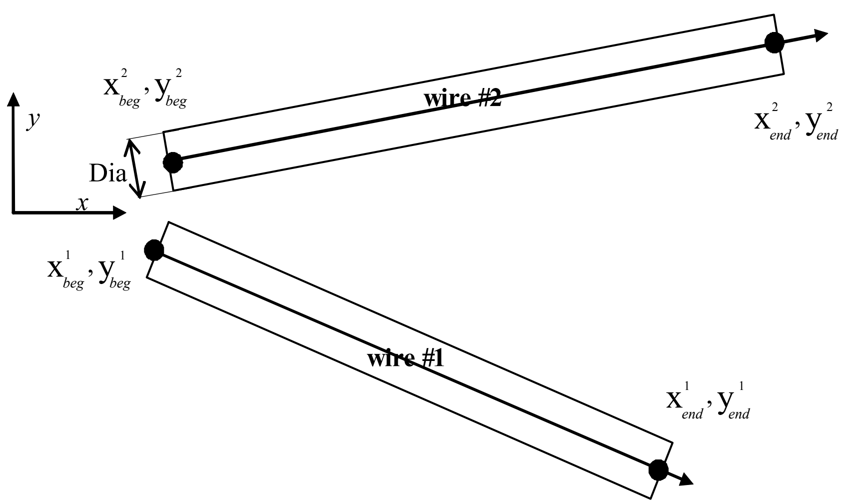
BWIRES2 allows you to assign to each wire individual shape and elevation above the ground plane. The BWIRES model provides single-shape and single-level arrangement.

BWIRES2 is a dynamic model, that is, its symbol redraws and its number of nodes changes as you change the number of wires.

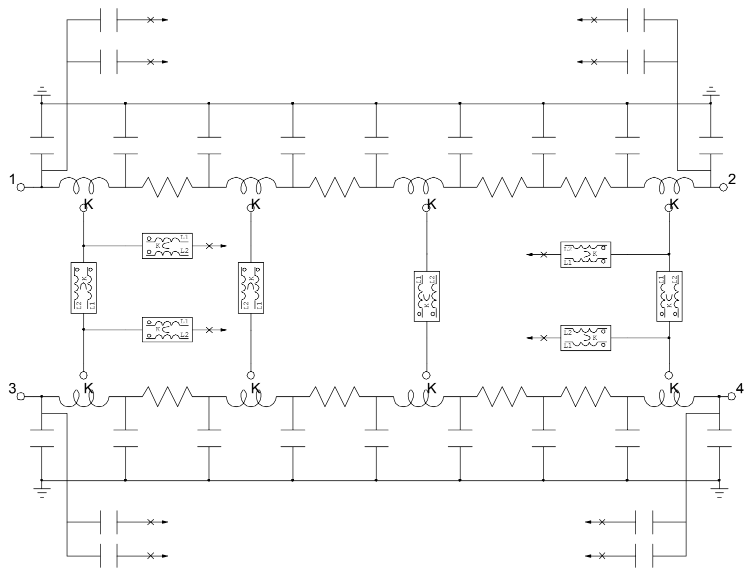
Topology




s2
s1
H1
s3
H2
z
H3
Start point
End point
D


x^2_{beg}, y^2_{beg}
y
wire #2
x^2_{end}, y^2_{end}
Dia
x
x^1_{beg}, y^1_{beg}
wire #1
x^1_{end}, y^1_{end}

Equivalent Circuit



Only part of a full equivalent circuit for N=2 is presented due to the complex nature of multiple interconnects. The actual equivalent circuit includes inductive and capacitive coupling between all segments of each wire.

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	IC1
N	Number of wires		1
Dia	Wire diameter	Length	%T[a]
Rho	Metal resistivity relative to gold		1
*WModel	Mode switch: 4 segments/3 segments		4 segments
*Alpha	Angle from first segment (s1) to the ground plane (vector of length N)	Degree	90
*Beta	Angle from last segment (s4/s3) to the ground plane (vector of length N)	Degree	45
*H1	Height from beginning point to top (vector of length N)	Length	%W[a]
*H2	Height from end point to beginning point (vector of length N)	Length	%W[a]
*H3	Height of end point over the ground plane (vector of length N)	Length	%W[a]
*D1_Ratio	Ratio of segment #2 length to total wire ground footprint (vector of length N)		0.125
*D2_Ratio	Ratio of segment #4 ground footprint to total wire ground footprint (vector of length N)		0.5

Name	Description	Unit Type	Default
*P0	Coordinates of beginning points (vector of length N)	Length	{0,0,0,1e-6}
*P1	Coordinates of end points (vector of length N)	Length	{5e-6,0,5e-6,1e-6}
*IndModel	Switch: RLC/RL model		RLC
*Er	Relative dielectric constant of optional encasing material		1
*Tand	Loss tangent of optional encasing material		0
*ShapeCnt	Strict shape control/No strict shape control switch		Strict shape control

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory.

** indicates a secondary parameter*

Parameter Details

Alpha, Beta, H1, H2, H3, D1_Ratio, D2_Ratio. These parameters are "shape parameters" that define the shape of the bond wire. Each parameter is a vector, so shape parameters pertaining to wire #I must take the I-th position within each vector. Vector values must be put in braces.

D1_Ratio, D2_Ratio. These parameters are unitless. They are defined as ratios D1/D and D2/D correspondingly (see the "Topology" section).

WModel. This switch allows switching between two modes: "4 segment" (default) and "3 segment" (see the "Topology" section). In "4 segment" mode, the model accepts, checks, and uses all model shape parameters. In "3 segment" mode, the model does not check and does not use parameters D1_Ratio, and D2_Ratio. The latter parameters are excessive because shape parameters Alpha, Beta, H1, H2, and H3 unambiguously define the shape of a 3-segment bond wire.

P0, P1. These parameters are vectors. P0 contains pairs of (x, y) coordinates of the start point of each wire; P1 contains pairs of (x, y) coordinates of the end point of each wire. For example, the order of entries in vector P0 for N=2 is the following: {x1,y1,x2,y2} where x1 is the x-coordinate of the start point of wire #1, y1 is the y-coordinate of the start point of wire #1, x2 is the x-coordinate of the start point of wire #2, and y2 is the y-coordinate of the start point of wire #2.

End coordinates of vector P1 are not required to be coordinates on wire end points - they can be located anywhere on the line that specifies azimuthal position of the respective wire plane.

This model expects the number of P0,P1 entries to be equal to 2N, and issues an error message if these numbers differ.

IndModel. This switch allows switching between two representations of the model equivalent circuit: "RLC" (Resistor-Inductor-Capacitor) and "RL" (Resistor-Inductor). Setting IndModel to "RL" *significantly reduces computation time*, excluding capacitances from the equivalent circuit. At relatively low frequencies (for example, below 10 GHz) both values of IndModel yield close modeling results. The difference between modeling results depends on the proximity of the evaluation frequency to the self-resonance frequency (SRF) of tested wire. A model at the "RL" setting cannot predict the location of SRF, while a model at the "RLC" setting (default) is "SRF aware" and predicts SRF reasonably accurately (see the "Recommendations for Use" section).

Er, Tand. Parameters of material used for optional encasing. The defaults are Er=1, Tand=0.

ShapeCnt. This switch allows you to disable certain wire shape checking. This shape checking (if not disabled) forbids combinations of dimensions and angles that create steep acute angled "bumps" in the wire outline.

Parameter Restrictions and Recommendations

1. This model implies that the number of bond wires is less than or equal to 512.
2. The height of the end point over the ground H3 may be set equal to zero, allowing bond wire to touch ground.
3. If WModel = 4, ensure that D1_Ratio+D2_Ratio<=1. If this condition is not met, the wire contour gets ragged, and in certain configurations non-adjacent segments may even cross each other.
4. When possible, wire should retain the recommended convex upward profile. You should not attempt to create straight, particularly horizontal wires. For straight wires with 3D EM pCells, use the BWIRES3 model with the BWShape parameter set to 2D and the shape parameter OrtTip set to 1.
5. If WModel = "3 segment", parameters D1_Ratio and D2_Ratio are not used and not checked. Wire shape is fully defined by H1, H2, Alpha, and Beta.
6. Alpha <135 ^{o}, Beta<=90 ^{o}. If WModel = "3 segment", the layout cell sets caps on parameters H1, Alpha, and Beta to prevent creation of twisted/self-crossing wire profiles.
7. This model subjects shape parameters (see the "Parameter Details" section) and locations of wire end points (P0, P1) to extended design rule control that prevents segments within separate wires and wires themselves from touching and intersecting (from getting closer than Dia). BWIRES2 also provides control of the wire shape and tries to disallow spikes (acute angles that are too small, between adjacent segments). For example, BWIRES2 checks if the length of any segment exceeds three wire footprints; this means that some segments are long and almost vertical. This model also checks if the junction of segments #3 and #4 is elevated too high above segment #2; this means that the angle between segments #3 and #4 is acute and very small. You can constantly monitor the actual shape of wire in the layout 3D view window.
8. Dia>0.1 micron
9. Er and Tand affect modeling results only if IndModel=RLC.

Implementation Details

Wires are considered a collection of infinitely thin, arbitrarily oriented, conducting, segmented filaments situated above infinite perfect ground. The actual diameter and material of the wire are accounted for at the evaluation of self-characteristics of each segment; mutual parameters neglect diameter and material. This model does not solve for distribution of charges/currents along or over wires; the approach is based on the assumption that the method of static-averaged potentials is applicable in this case.

This model accounts for internal and external inductance of each wire segment; its evaluation of resistances is based on a skin-effect approach and the radiation resistance of a small dipole above ground. Evaluation of capacitances implies that the surrounding media is infinite over perfect ground plane and uniformly filled with optional encasing material (the default is vacuum); electrical properties of adjacent dielectric objects are neglected. A value of Er other than 1 changes all capacitances in the equivalent circuit; nonzero Tand adds a shunt resistor to each capacitance throughout all equivalent circuits (see the "Equivalent Circuit" section).

Recommendations for Use

User discretion is advised if the following three factors are combined:

- The evaluation frequency is very high (for example, above 50 GHz)
- The wires are relatively long (above 1 mm)
- The low profile wires span a substrate of relatively high permittivity (~8 and above). Even in this case, SRF is usually predicted well within 5-7% error; however, the error in phases of "crosstalk" scattering coefficients might be large due to additional unaccounted substrate coupling.

Setting the IndMode parameter to "RL" *substantially reduces calculation time* (this may be advantageous if the number of bond wires exceeds 20) but excludes effects of capacitances coupling (these effects are responsible for self-resonances at higher frequencies). "RL" leaves in place frequency-dependence of self-inductances; radiation resistance also remains added to the self-resistance of each wire.

User discretion is also advised when the total length/footprint of wire is small. Evidently, short wires have even shorter segments. At the same time, the model assumes that each segment is represented by infinitely thin filament, which means Lseg >> Dia. The model issues a warning if Lseg<6*Dia.

We have compared our Bondwire model to other commercially available software which accounts for all loss and radiation, and found that we have a good match using 48 wires up to 1mm (roughly 40mils) long and up to 100GHz. A safe rule of thumb to follow is Length of wire / Wavelength ratio should be less than 1/6.

Layout

The layout for this cell has hard-coded model layers. When you first use this layout cell, a layer named "BondWires" is added to your drawing layer and model layer list (if they are not already there). Using the model layer mapping, you can assign these layers to draw on any drawing layer. The via is drawn the size of the "D1" parameter.

This model has several layout-only parameters that control how the pad draws. You access these parameters by selecting the item in the layout, right-clicking and choosing **Shape Properties** to display the Cell Options dialog box, then clicking the **Parameters** tab.

Name	Value	Units	Description
ShowNodeNum	0	Integer	Layout displays node numbers only if this value is non-zero
PointFaces	0	Integer	Faces are points, not line segments, only if this value is non-zero

References

[1] Grover, F.W., Inductance Calculations, Mineola, New York: Dover Publishing, 2004

[2] Bahl, I.J., Lumped Elements for RF and Microwave Circuits, Norwood, MA: Artech House, 2003

[3] Balanis C.A., Antenna Theory: analysis and design, 2nd edition: Wiley, 1997

[4] Bond Wire Modeling Standard. EIA/JEDEC Standard EIA/JESD59: Electronics Industries Association, June 1997.

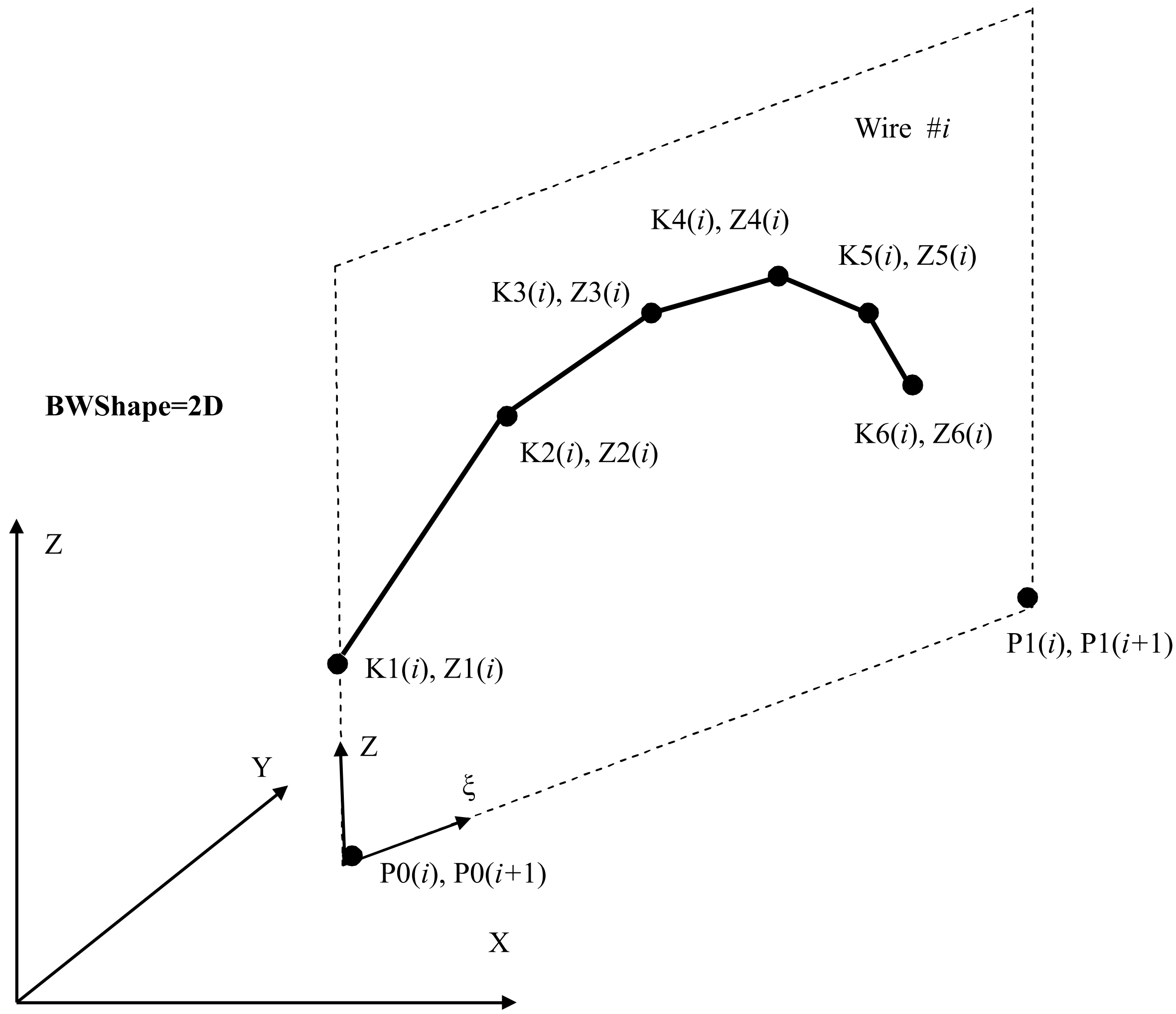
Multilevel Bond Wire Model (User Defined Shape): BWIRES3

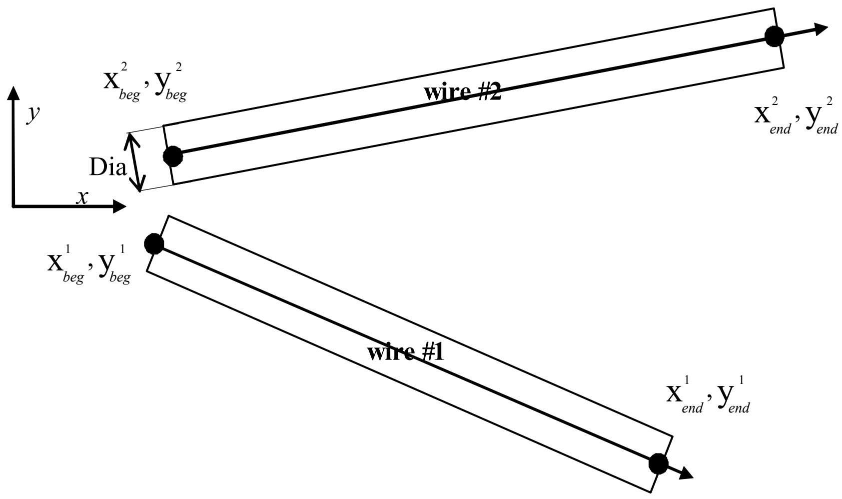
Symbol

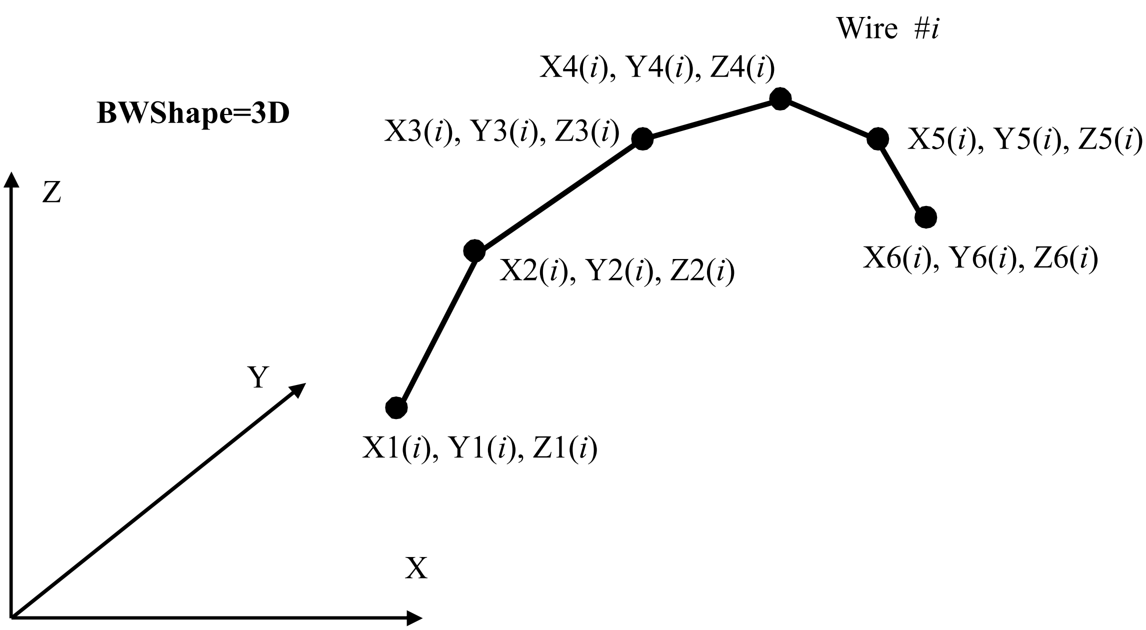
Summary

BWIRES3 models a collection of metallic round bonding wires (bond wires) made of linear segments each and placed above a conducting ground plane. The actual smooth contour of a bonding wire is approximated by a user-specified number of linear segments, and parameters that define orientation of segments are Cartesian coordinates of the segment ends. Segments that comprise a wire can be either coplanar and fit in a plane orthogonal to the ground plane (2D wire shape), or arranged arbitrarily in space (3D wire shape). Each wire can be shaped individually and placed at a selected elevation above the ground plane. **Unlike BWIRES2, BWIRES3 allows you to shape each wire in an arbitrary manner.** BWIRES3 is a dynamic model, that is, its symbol redraws and its number of nodes change as you change the number of wires.

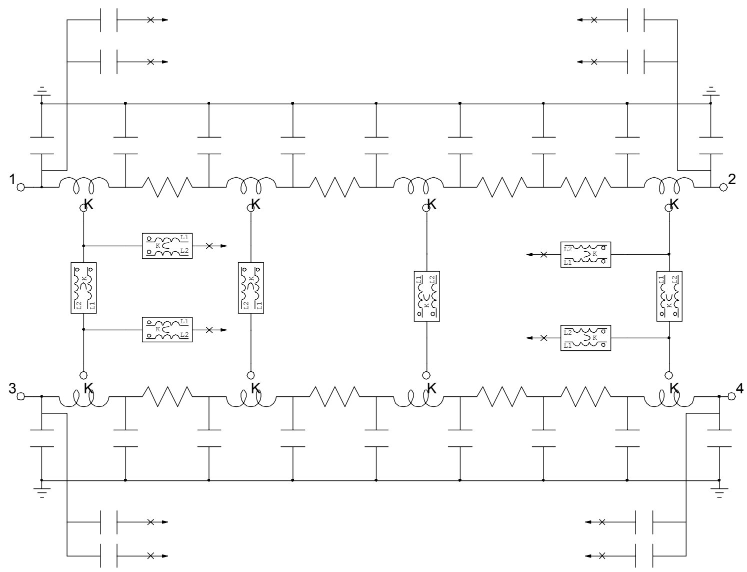
Topology


Wire #i
K4(i), Z4(i)
K5(i), Z5(i)
K3(i), Z3(i)
BWShape=2D
K2(i), Z2(i)
K6(i), Z6(i)
Z
P1(i), P1(i+1)
K1(i), Z1(i)
Z
Y
ξ
P0(i), P0(i+1)
X


x^2_{beg}, y^2_{beg}
y
wire #2
x^2_{end}, y^2_{end}
Dia
x
x^1_{beg}, y^1_{beg}
wire #1
x^1_{end}, y^1_{end}


Wire #i
X4(i), Y4(i), Z4(i)
BWShape=3D
X3(i), Y3(i), Z3(i)
X5(i), Y5(i), Z5(i)
Z
X2(i), Y2(i), Z2(i)
X6(i), Y6(i), Z6(i)
Y
X1(i), Y1(i), Z1(i)
X

Equivalent Circuit



Only part of a full equivalent circuit for N=2 is presented due to the complex nature of multiple interconnects. The actual equivalent circuit includes inductive and capacitive coupling between all segments of each wire.

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	IC1
N	Number of wires		1
Nseg	Number of segments		5
Dia	Wire diameter	Length	25.4 micron
Rho	Metal resistivity relative to gold		1
*IndModel	Switch: RLC/RL model		"RLC"
*Er	Relative dielectric constant of optional encasing material		1
*Tand	Loss tangent of optional encasing material		0
BWShape	Switch: 3D/2D wire shape		3D
	The following parameters display if BWShape=2D		
K1	Ksi coordinate of wire point #1(vector of length N)	Length	0 microns
Z1	Z coordinate of wire point #1(vector of length N)	Length	50 microns
Kn	Ksi coordinate of wire point #n(vector of length N)	Length	Depends on Nseg
Zn	Z coordinate of wire point #n(vector of length N)	Length	Depends on Nseg
P0	X,Y coordinates of beginning points (vector of length N)	Length	{0,0} microns
P1	X,Y coordinates of end points (vector of length N)	Length	{500,0} microns

Name	Description	Unit Type	Default
	The following parameters display if BWShape=3D		
X1	X coordinate of wire point #1(vector of length N)	Length	0 microns
Y1	Y coordinate of wire point #1(vector of length N)	Length	0 microns
Z1	Z coordinate of wire point #1(vector of length N)	Length	50 microns
Xn	X coordinate of wire point #n(vector of length N)	Length	Depends on Nseg
Yn	Y coordinate of wire point #n(vector of length N)	Length	Depends on Nseg
Zn	Z coordinate of wire point #n(vector of length N)	Length	Depends on Nseg

** indicates a secondary parameter*

Parameter Details

Nseg. Specifies the number of linear segments that make each wire. The number of segments is limited (see the "Parameter Restrictions and Recommendations" section).

BWShape. This switch parameter defines the interface to wire specification. If BWShape=3D, then you can arrange each wire arbitrarily in 3D space. If BWShape=2D, then each wire is confined to a single plane orthogonal to the ground plane (or X-Y plane). The azimuthal position of a wire plane is specified by vector parameters P0 and P1, (see the "Topology" section). The default value of BWShape=3D.

K1, Z1,... Kn, Zn. These vector parameters display only if BWShape=2D. They are vector "shape parameters" (Cartesian coordinates Ksi and Z - see the "Topology" section), that specify positions of all segments on the Ksi-Z plane and define the shape of the bond wire. The ground plane is located at Z=0. The number of these Kn,Zn pairs is Nseg+1. Each parameter is a vector, so shape parameters pertaining to wire #I must take the I-th position within each vector. Vector values must be in braces. Note that this model silently forces each wire to start at Ksi=0 even if you specify nonzero K1(n). The start position of each wire in the X-Y plane is specified by the respective pair of values: Xbeg=P0(n), Ybeg=P1(n) (see the "Topology" section). This model expects the sizes of all Kn and Zn to be equal to N and issues an error message if these sizes differ.

P0, P1. These vector parameters display only if BWShape=2D. P0 contains pairs of (x, y) coordinates of the start point of each wire; P1 contains pairs of (x, y) coordinates of the end point of each wire. For example, the order of entries in vector P0 for N=2 is the following: {x1,y1,x2,y2} where x1 is the x-coordinate of the start point of wire #1, y1 is the y-coordinate of the start point of wire #1, x2 is the x-coordinate of the start point of wire #2, and y2 is the y-coordinate of the start point of wire #2.

End coordinates of vector P1 are not required to be coordinates on wire end points - they can be located anywhere on the line that specifies azimuthal position of the respective wire plane.

This model expects the number of P0,P1 entries to be equal to 2N and issues an error message if these numbers differ.

X1, Y1, Z1,... Xn, Yn, Zn. These vector parameters display only if BWShape=3D.They are vector "shape parameters" (Cartesian coordinates X, Y, and Z - see the "Topology" section), that specify positions of all segments above the ground plane (located at Z=0) and define the shape of the bond wire. The number of these vector triplets is Nseg+1. Each parameter is a vector, so shape parameters pertaining to wire #I must take the I-th position within each vector. Vector values must be in braces. This model expects the sizes of all Xn, Yn, and Zn to be equal to N and issues an error message if these sizes differ.

IndModel. This switch allows switching between two representations of the model equivalent circuit: "RLC" (Resistor-Inductor-Capacitor) and "RL" (Resistor-Inductor). Setting IndModel to "RL" *significantly reduces computation* *time*, excluding capacitances from the equivalent circuit. At relatively low frequencies (for example, below 10 GHz) both values of IndModel yield close modeling results. The difference between modeling results depends on the proximity of the evaluation frequency to the self-resonance frequency (SRF) of tested wire. A model at the "RL" setting cannot predict the location of SRF, while a model at the "RLC" setting (default) is "SRF aware" and predicts SRF reasonably accurately (see the "Recommendations for Use" section).

Er, Tand. Parameters of material used for optional encasing. The defaults are Er=1, Tand=0.

Parameter Restrictions and Recommendations

1. Number of segments is limited: 1≤Nseg≤10
2. This model implies that the number of bond wires is less than or equal to 512.
3. Heights of the start and end point over the ground (Z1(n) and ZNseg(n)) may be set equal to zero, allowing the bond wire to touch ground. Z coordinates of intermediate points cannot be set to zero because BWIRES3 does not allow joints of segments to touch ground.
4. BWIRES3 subjects shape parameters (see the "Parameter Details" section) and locations of wire end points (P0, P1) to extended design rule control that prevents segments within separate wires and wires themselves from touching and intersecting (from getting closer than Dia). Also, the length of any segment cannot be less than Dia. You can constantly monitor the actual shape of wire in the Layout 3D View window.
5. Dia>0.1 micron
6. Er and Tand affect modeling results only if IndModel=RLC.

Implementation Details

Wires are considered a collection of infinitely thin, arbitrarily oriented, conducting, segmented filaments situated above infinite perfect ground. The actual diameter and material of the wire are accounted for at the evaluation of self-characteristics of each segment; mutual parameters neglect diameter and material. This model does not solve for distribution of charges/currents along or over wires; the approach is based on the assumption that the method of static-averaged potentials is applicable in this case.

This model accounts for internal and external inductance of each wire segment; its evaluation of resistances is based on a skin-effect approach and the radiation resistance of a small dipole above ground. Evaluation of capacitances implies that the surrounding media is infinite over a perfect ground plane and uniformly filled with optional encasing material (the default is vacuum); electrical properties of adjacent dielectric objects are neglected. A value of Er other than 1 changes all capacitances in the equivalent circuit; nonzero Tand adds a shunt resistor to each capacitance throughout all equivalent circuits (see the "Equivalent Circuit" section).

Recommendations for Use

The recommended range of number of segments Nseg is 3≤Nseg≤6. Note that the number of segments may affect the position of calculated series resonance frequency (SRF) due to adopted approximations of constant distribution of charge along the segments.

User discretion is advised if the following three factors are combined:

- The evaluation frequency is very high (for example, above 50 GHz)
- The wires are relatively long (above 1 mm)

- The low profile wires span a substrate of relatively high permittivity (~8 and above). Even in this case, SRF is usually predicted well within 5-7% error; however, the error in phases of "crosstalk" scattering coefficients might be large due to additional unaccounted substrate coupling.

Setting the IndMode parameter to "RL" *substantially reduces calculation time* (this may be advantageous if the number of bond wires exceeds 20) but excludes effects of capacitances coupling (these effects are responsible for self-resonances at higher frequencies). "RL" leaves in place frequency-dependence of self-inductances; radiation resistance also remains added to the self-resistance of each wire.

User discretion is also advised when the total length/footprint of wire is small. Evidently, short wires have even shorter segments. At the same time, the model assumes that each segment is represented by infinitely thin filament, which means Lseg >> Dia. The model issues a warning if Lseg<6*Dia.

Note that BWIRES3 hides two different models under one hood. One model is selected by setting BWShape=3D; this allows you to position the bond wire arbitrarily in 3D space. Another model is selected by setting BWShape=2D; this confines each bond wire to its own vertical plane. Because these models differ significantly in their interfaces, parameters are not inherited when switching from one model to another.

Also note that initially placing BWIRES3 on a schematic provides a fully operational instance at the default settings N=1 and BWShape=3D. Changing N only automatically creates a set of shifted identical wires; however, this is done only for reference. You must fully specify all vector parameters and provide all N entries for each parameter in accordance with your target design.

Switching BWShape from 3D to 2D on an existing instance of BWIRES3 may cause a change (decrease) in the Dia parameter. You should check the value of Dia on your schematic after switching BWShape from 3D to 2D and reset Dia in accordance with your design.

We have compared our bondwire model to other commercially available software which accounts for all loss and radiation, and found that we have a good match using 48 wires up to 1mm (roughly 40mils) long and up to 100GHz. A safe rule of thumb to follow is length of wire / wavelength ratio should be less than 1/6.

Layout

The layout for this cell has hard-coded model layers. When you first use this layout cell, a layer named "BondWires" is added to your drawing layer and model layer list (if they are not already there). Using the model layer mapping, you can assign these layers to draw on any drawing layer. The via is drawn the size of the "D1" parameter.

This model has several layout-only parameters that control how the pad draws. You access these parameters by selecting the item in the layout, right-clicking and choosing **Shape Properties** to display the Cell Options dialog box, then clicking the **Parameters** tab.

Name	Value	Units	Description
ShowNodeNum	0	Integer	Layout displays node numbers only if this value is non-zero
PointFaces	0	Integer	Faces are points, not line segments, only if this value is non-zero

Name	Value	Units	Description
OrtTip	0	Integer	If zero, the wire tip cut is horizontal, otherwise the tip cut is orthogonal to the end segment axis

Note that option BWShape=2D forbids implementation of strictly vertical wires and wires with coinciding X-Y projections of start and end ponts. Attempt to create these wires in 2D causes layout interruption and is accompanied with error text messages outputted into layout window. These configurations can be created after selection BWShape=3D.

3D EM Layout

This model includes OrtTip, a layout-only parameter that controls how the tip of a wire is cut. If OrtTip=0 (default), the tip is cut horizontally (this provides a smooth transition to some on-layer metal patch or strip). If OrtTip≠0, the wire tip is cut orthogonally to the last/first segment axis. An orthogonal cut improves the transition to metal connectors if the last/first segments are horizontal or almost horizontal. To access this parameter, select the item in the layout, right-click and choose **Shape Properties** to display the Cell Options dialog box, then click the **Parameters** tab.

Note that implementation of 3D EM pcells strictly preserve undistorted cross-sectional view of a wire only if all wire segments belong to the same vertical plane. Off-plane deviation of segments may cause distortion of displaced wires.

References

[1] Grover, F.W., Inductance Calculations, Mineola, New York: Dover Publishing, 2004

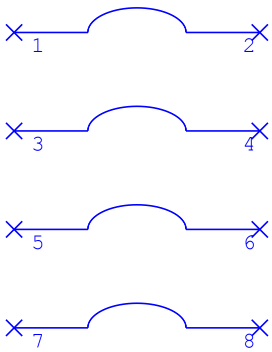
[2] Bahl, I.J., Lumped Elements for RF and Microwave Circuits, Norwood, MA: Artech House, 2003

[3] Balanis C.A., Antenna Theory: analysis and design, 2nd edition: Wiley, 1997

[4] Bond Wire Modeling Standard. EIA/JEDEC Standard EIA/JESD59: Electronics Industries Association, June 1997.

Multilevel Bond Wire Model (Arrays of User-defined N-segment Shapes): BWIRES3ARRAY

Symbol



Summary

BWIRES3ARRAY models a collection of metallic round bonding wires (bond wires) each made of linear segments and placed above a conducting ground plane. The actual smooth contour of a bonding wire is approximated by a user-specified number of linear segments, and parameters that define orientation of segments are Cartesian coordinates of the segment ends.

BWIRES3ARRAY provides a simpler way to specify arrays of bond wires for the BWIRES3 model.

This model allows specifying the X/Z cross section of one or more bonding profiles with the XProfile1, ZProfile1, XProfile2, ZProfile2... parameters. This is useful when modeling multiple couple bond wires with mixed bonding profiles, heights, and spacings.

Implementation Details

Internally, this model uses the BWIRES3 element. The BWIRES3ARRAY element provides a simpler interface for defining arrays of bond wires. See BWIRES3 for topology, equivalent circuit, and recommended usage.

Parameters

Name	Description	Unit Type	Default
ID	Element ID		IC1
N	Number of wires		4
Nseg	Number of segments		5
Dia	Wire diameter	Length	25.4
Rho	Metal bulk resistivity relative to gold		1
*Er	Relative dielectric constant of encasing material		1
*Tand	Loss tangent of encasing material		0
*ZOffset	Global Z Offset	Length	0
*YOffsetBegin	Y Offset at the start of the bond array	Length	0
*YOffsetEnd	Y Offset at the end of the bond array	Length	0

Name	Description	Unit Type	Default
YSpacBegin	Spacing between wires at start of bond. Can be a single value or a vector (repeats until N wires if vector).	Length	{100}
YSpacEnd	Spacing between wires at end of bond. Can be a single value or a vector (repeats until N wires if vector).	Length	{100}
NProfiles	Number of bond wire profiles		2
Pattern	Pattern to repeat bond wire profiles		{1,2,2,1}
XProfile1	X coordinates of wire bond profile 1	Length	{0,200,400,600,800,1000}
ZProfile1	Z coordinates of wire bond profile 1	Length	[illegible]
XProfile2	X coordinates of wire bond profile 2	Length	{0,40,80,280,640,1000}
ZProfile2	Z coordinates of wire bond profile 2	Length	[illegible]

** indicates a secondary parameter*

Parameter Details

N. Specifies the total number of bond wires in the array. The maximum is 512.

Nseg. Specifies the number of linear segments that make each wire. The number of segments is limited to 10. The recommended range is $3 \le Nseg \le 6$.

NProfiles. Specifies the number of different bonding profiles that are used in the array. This parameter changes the number of XProfile_, ZProfile_ parameters.

XProfile1,ZProfile1,...XProfileN,ZProfileN. Each of these parameters is a vector of NSeg+1 length. Each pair of vectors defines one bonding profile. By default, XProfile1,ZProfile1 define a symmetric 'sinc' shape typical of a stitch-stitch bonding. XProfile2,ZProfile2 default to an asymmetric profile typical of a ball-stitch bonder.

Note that you can quickly modify the arrays with equations:

- Adding/subtracting a value to the XProfile shifts the bond profile, for example, "100 + {0, 100, 200, 300, 400, 500}"
- Adding/subtracting a value to the ZProfile raises and lower bond profile, for example, "100 + {0, 100, 200, 300, 200, 100}"
- Multiplying the XProfile by a value stretches the length of the bond profile, for example, "1.5 * {0, 100, 200, 300, 400, 500}"
- Multiplying the ZProfile by a value stretches the height of the bond profile, for example, "0.7*{0, 100, 200, 300, 200, 100}"

Pattern A vector that defines the order in which the N bond wires use the NProfiles specified profiles. This vector can be any length; the specified pattern is repeated out to N bond wires. For example:

- There are N=10 bond wires, and NProfiles=2 profiles. Pattern={1,2,2,1}. Starting from the first wire, the profiles used are 1,2,2,1,1,2,2,1,1,2.
- N=10 bond wires, and NProfiles=2. Pattern={1}. Starting from the first wire, the profiles used are 1,1,1,1,1,1,1,1,1,1,1 (not all profiles are required to be used.)
- N=10 bond wires, and NProfiles=2. Pattern={0,1,2,3}. Starting from the first wire, the profiles used are 1,1,2,2,1,1,2,2,1,1 (profile #'s are automatically limited between 1 and NProfiles.)

YSpacBegin,YSpacEnd These parameters define the spacing between bond wires at the start and end of the bond. They can be scalar or vector parameters. Like the Profile parameter, the vectors can be any length, and the specified spacing is repeated out to N bond wires. For example:

- YSpacBegin = {50,100,150}. YSpacEnd = {40,60}. If N = 10, the Y spacing between the start of adjacent wires is {50,100,150,50,100,150,50,100,150,50}. The spacing at the end of the wires is {40,60,40,60,40,60,40,60,40,60}.
- YSpacBegin = {50,100,150}. YSpacEnd = {40,60}. If N = 10, the Y spacing between the start of adjacent wires is {50,100,150,50,100,150,50,100,150,50}. The spacing at the end of the wires is {40,60,40,60,40,60,40,60,40,60}.

YOffsetBegin,YOffsetEnd These scalar parameters add a constant YOffset to the beginning and end of the wires, respectively.

Er, Tand. Material used for optional encasing. The defaults are Er=1 and Tand=0.

Parameter Restrictions and Recommendations

1. Number of segments is limited: 1≤Nseg≤10
2. This model implies that the number of bond wires is less than or equal to 512.
3. Dia>0.1 micron

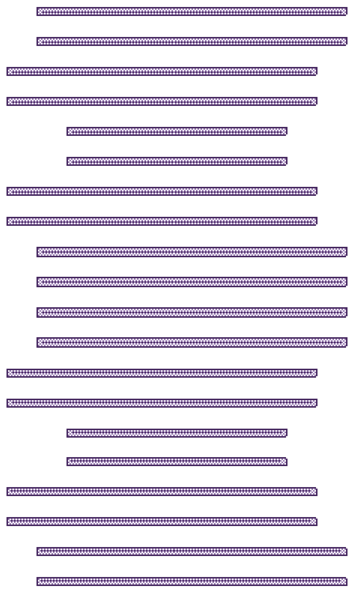
We have compared our Bondwire model to other commercially available software which accounts for all loss and radiation, and found that we have a good match using 48 wires up to 1mm (roughly 40mils) long and up to 100GHz. A safe guideline to follow is Length of wire / Wavelength ratio should be less than 1/6.

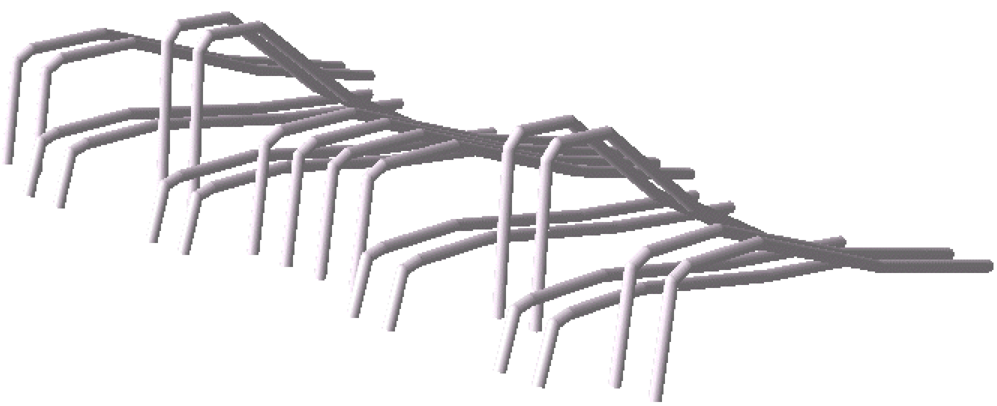
Usage Examples

Example #1:

Three different bond profiles are used. The Pattern is 10 items long, so it repeats twice for N=20 bond wires. Note that the three profiles have the same vector values, but they are modified by adding and multiplying different constants to XProfileN, ZProfileN.

```
BWIRES3ARRAY
ID=IC1
N=20
Nseg=5
Dia=25.4 um
Rho=1
YSpacBegin={100} um
YSpacEnd={100} um
NProfiles=3
Pattern={1,1,2,2,3,3,2,2,1,1}
XProfile1=100 +{ 0,40,80,280,640,1000 } um
ZProfile1={ 0,223.7847225,242.4828375,235.5260375,94.4133,0 } um
XProfile2={ 0,40,80,280,640,1000 } um
ZProfile2=0.5*{ 0,223.7847225,242.4828375,235.5260375,94.4133,0 } um
XProfile3=200 + 0.7*{ 0,40,80,280,640,1000 } um
ZProfile3=1.5*{ 0,223.7847225,242.4828375,235.5260375,94.4133,0 } um
```

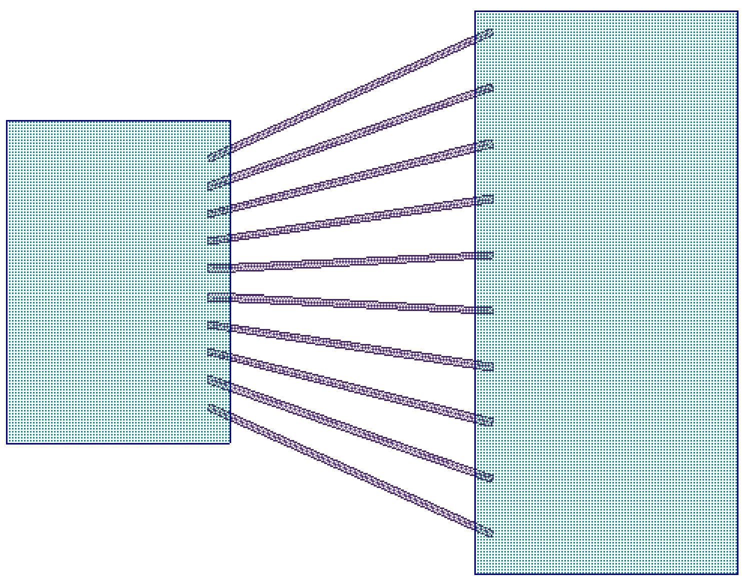


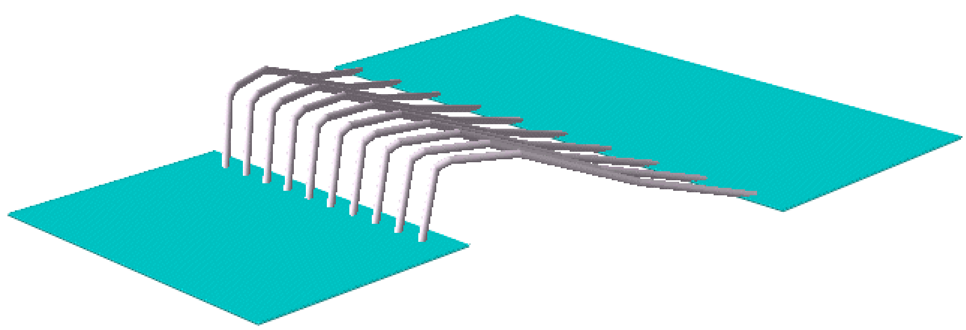


Example #2:

One bond profile is used with different YSpacBegin and YSpacEnd. YOffsetBegin is adjusted so that the wires are symmetrical about the center wires.

BWIRES3ARRAY
ID=IC1
N=10
Nseg=5
Dia=25.4 um
Rho=1
YOffsetBegin=450 um
YOffsetEnd=0 um
YSpacBegin={100} um
YSpacEnd={200} um
NProfiles=1
Pattern=1
XProfile1={ 0,40,80,280,640,1000 } um
ZProfile1={ 0,223.7847225,242.4828375,235.5260375,94.4133,0 } um

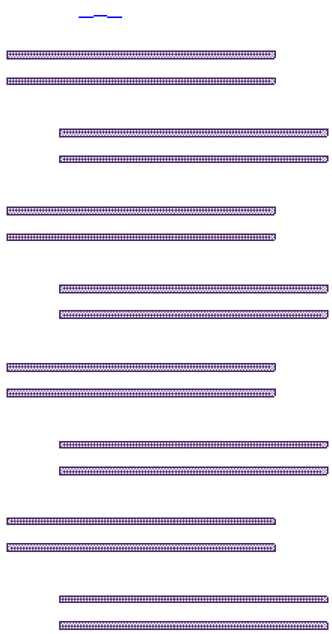


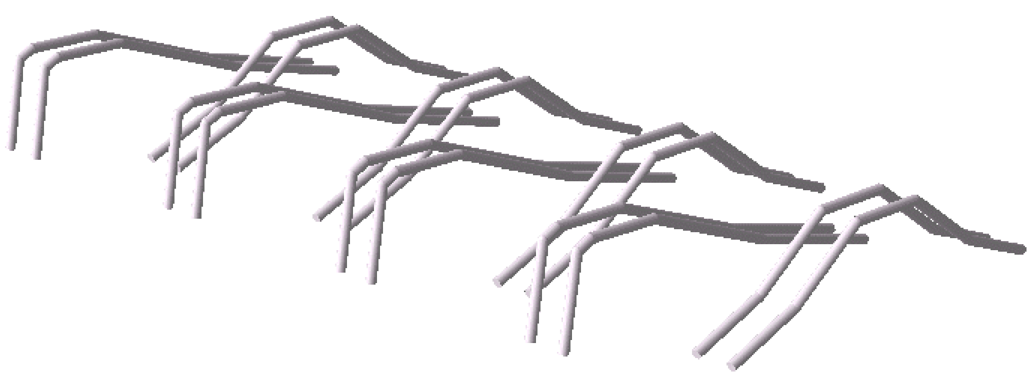


Example #3:

Two bond profiles are used with a pattern of {1,1,2,2}. The YSpacBegin, YSpacEnd parameters are also vectors, so that similar bonds are spaced closer together.

BWIRES3ARRAY
ID=IC1
N=16
Nseg=5
Dia=25.4 um
Rho=1
YSpacBegin={100,200,100,200} um
YSpacEnd={100,200,100,200} um
NProfiles=2
Pattern={1,1,2,2}
XProfile1=200 + { 0,200,400,600,800,1000 } um
ZProfile1={ 0,77.131705,223.38425,223.38425,77.131705,0 } um
XProfile2={ 0,40,80,280,640,1000 } um
ZProfile2={ 0,223.7847225,242.4828375,235.5260375,94.4133,0 } um





Layout

The layout for this cell has hard-coded model layers. When you first use this layout cell, a layer named "BondWires" is added to your drawing layer and model layer list (if they do not already exist). Using the model layer mapping, you can assign these layers to draw on any drawing layer.

This model has several layout-only parameters that control how the pad draws. You access these parameters by selecting the item in the layout, right-clicking and choosing **Shape Properties** to display the Cell Options dialog box, then clicking the **Parameters** tab.

Name	Value	Units	Description
ShowNodeNum	0	Integer	Show/hide node number
PointFaces	0	Integer	Faces are points, not line segments
OrtTip	0	Integer	0 - Tip cut is horizontal, !=0 Tip cut orthogonal to segment axis

3D EM Layout

This model includes OrtTip, a layout-only parameter that controls how the tip of a wire is cut. If OrtTip=0 (default) then the tip is cut horizontally (this provides a smooth transition to some on-layer metal patch or strip); if OrtTip≠0 then the wire tip is cut orthogonally to the last/first segment axis. An orthogonal cut improves the transition to metal connectors if the last/first segments are horizontal or almost horizontal. To access this parameter, select the item in the layout, right-click and choose **Shape Properties** to display the Cell Options dialog box, then click the **Parameters** tab.

References

[1] Grover, F.W., Inductance Calculations, Mineola, New York: Dover Publishing, 2004

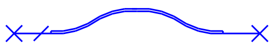
[2] Bahl, I.J., Lumped Elements for RF and Microwave Circuits, Norwood, MA: Artech House, 2003

[3] Balanis C.A., Antenna Theory: analysis and design, 2nd edition: Wiley, 1997

[4] Bond Wire Modeling Standard. EIA/JEDEC Standard EIA/JESD59: Electronics Industries Association, June 1997.

Multilevel Bond Wire Model (User-defined Multisegment Shapes with Varying Number of Segments): BWIRES4

Symbol



Summary

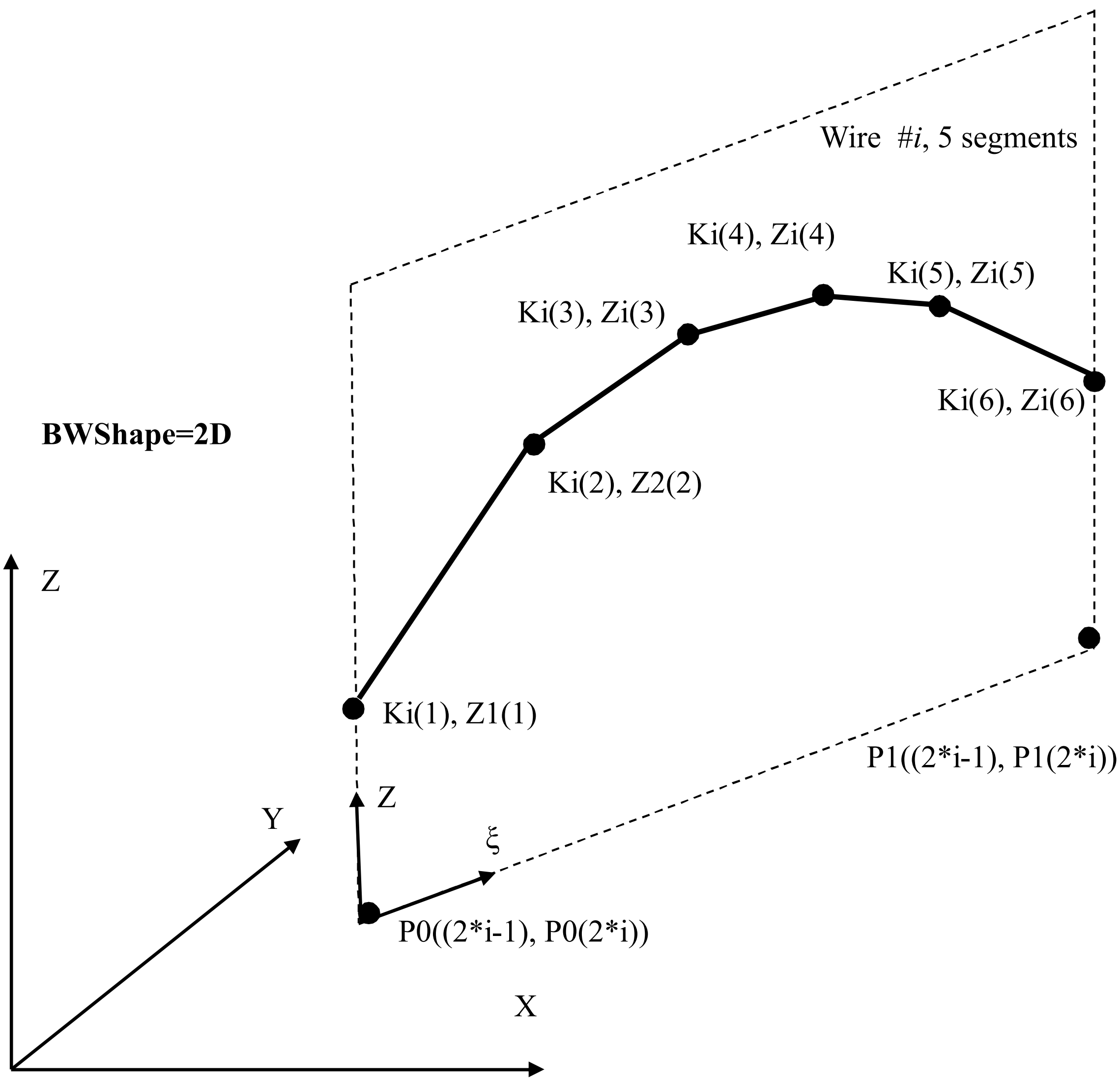
BWIRES4 models a collection of metallic round bonding wires (bond wires) made of linear segments each and placed above a conducting ground plane. The actual smooth contour of a bonding wire is approximated by a user-specified number of linear segments, and parameters that define orientation of segments are Cartesian coordinates of the segment ends. Segments that comprise a wire can be either coplanar and fit in a plane orthogonal to the ground plane (2D wire shape), or arranged arbitrarily in space (3D wire shape). Each wire can be shaped individually and placed at a selected elevation above the ground plane.

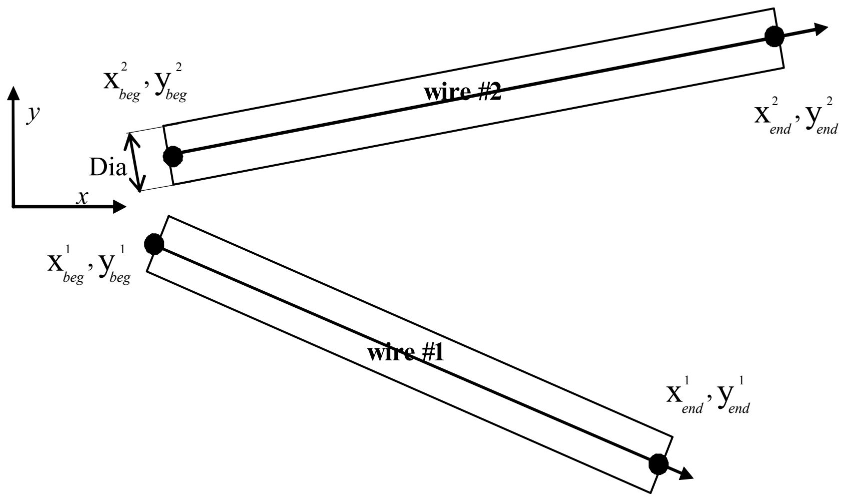
Unlike BWIRES2, BWIRES4 allows you to shape each wire in an arbitrary manner.

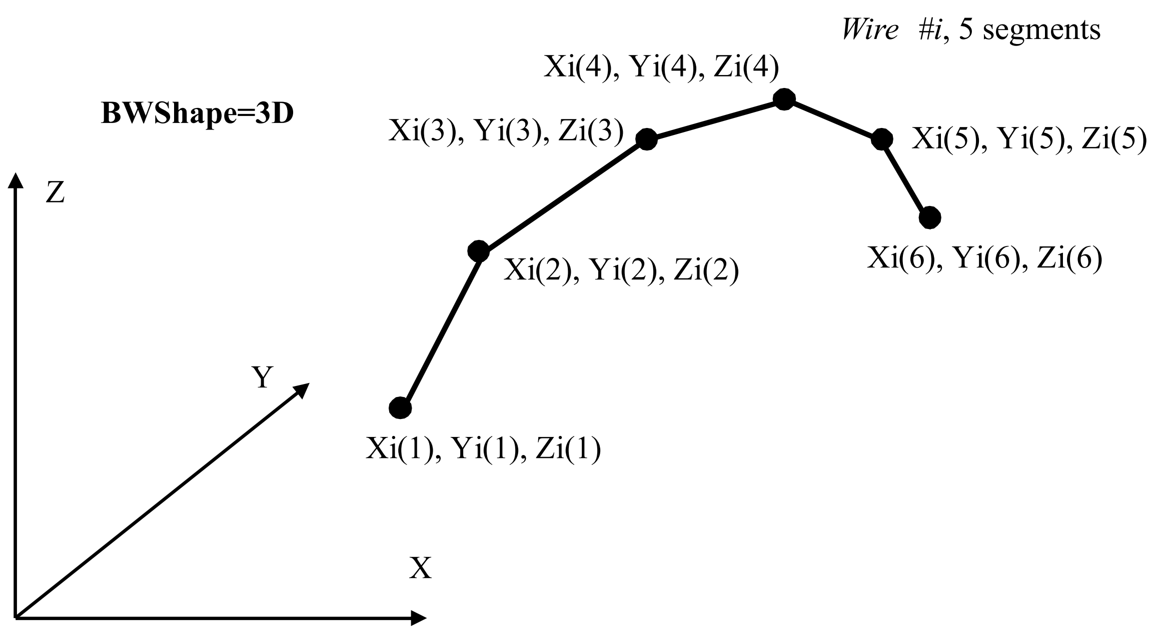
Unlike BWIRES2 and BWIRES3, BWIRES4 allows you to set an arbitrary number of segments for each wire.

BWIRES4 is a dynamic model; its symbol redraws and its number of nodes change as you change the number of wires.

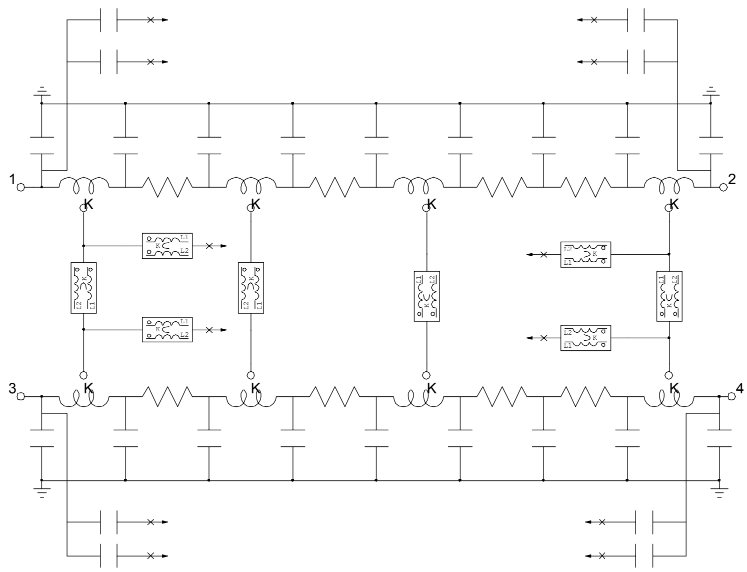
Topology


Wire #i, 5 segments
Ki(4), Zi(4)
Ki(5), Zi(5)
Ki(3), Zi(3)
Ki(6), Zi(6)
BWShape=2D
Ki(2), Z2(2)
Z
Ki(1), Z1(1)
P1((2*i-1), P1(2*i))
Z
Y
ξ
P0((2*i-1), P0(2*i))
X


x^2_{beg}, y^2_{beg}
wire #2
x^2_{end}, y^2_{end}
y
Dia
x
x^1_{beg}, y^1_{beg}
wire #1
x^1_{end}, y^1_{end}


Wire #i, 5 segments
BWShape=3D
Xi(4), Yi(4), Zi(4)
Xi(3), Yi(3), Zi(3)
Xi(5), Yi(5), Zi(5)
Z
Xi(2), Yi(2), Zi(2)
Xi(6), Yi(6), Zi(6)
Y
Xi(1), Yi(1), Zi(1)
X

Equivalent Circuit



Only part of a full equivalent circuit for N=2 is presented due to the complex nature of multiple interconnects. The actual equivalent circuit includes inductive and capacitive coupling between all segments of each wire.

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	IC1
N	Number of wires		1
Dia	Wire diameter	Length	25.4 micron
Rho	Metal resistivity relative to gold		1
*IndModel	Switch: RLC/RL model		"RLC"
*Er	Relative dielectric constant of optional encasing material		1
*Tand	Loss tangent of optional encasing material		0
BWShape	Switch: 3D/2D wire shape		3D
ZOffset	Global Z-offset applied to all wires	Length	0
	The following parameters display if BWShape=2D		
K1	Ksi coordinates of all wire #1 points (vector of user-defined size)	Length	
Z1	Z coordinates of wire #1 points (vector of user-defined size)	Length	
Kn	Ksi coordinates of wire #n points (vector of user-defined size)	Length	
Zn	Z coordinates of wire #n points (vector of user-defined size)	Length	
P0	X,Y coordinates of wire start points (vector of length 2*N)	Length	

Name	Description	Unit Type	Default
P1	X,Y coordinates of wire end points (vector of length 2*N)	Length	
	The following parameters display if BWShape=3D		
X1	X coordinates of all wire #1 points (vector of user-defined size)	Length	
Y1	Y coordinates of all wire #1 points (vector of user-defined size)	Length	
Z1	Z coordinates of all wire #1 points (vector of user-defined size)	Length	
Xn	X coordinates of wire #n points (vector of user-defined size)	Length	
Yn	Y coordinates of wire #n points (vector of user-defined size)	Length	
Zn	Z coordinates of wire #n points (vector of user-defined size)	Length	

** indicates a secondary parameter*

Parameter Details

N. Specifies the number of wires. The number of segments on each wire is user-defined and can vary from wire to wire.

BWShape. This switch parameter defines the interface to wire specification. If BWShape=3D, then you can arrange each wire arbitrarily in 3D space. If BWShape=2D, then each wire is confined to a single plane orthogonal to the ground plane (or X-Y plane). **The K(start) and K(end) coordinates of 2D wires as well as azimuthal orientation of each 2D wire plane are governed by vector parameters P0 and P1 (see the "Topology" and "Parameter Restrictions and Recommendations" sections)**. The default = 3D.

K1, Z1,... Kn, Zn. These vector parameters display only if BWShape=2D. They are vector "shape parameters" (Cartesian coordinates Ksi and Z - see the "Topology" section), that specify the positions of all segments on the Ksi-Z plane and define the shape of the bond wire. The ground plane is located at Z=0. The number of these Ki, Zi pairs is user-defined. Each parameter is a vector, so shape parameters pertaining to wire #i should be placed into respective vectors Ki and Zi. Vector values must be in braces. The first and last values in each K vector (Ki(1), Ki(N) and Zi(1), Zi(N)) are ignored because they are defined by vectors P0 and P1. The recommended design technique is to start and end each Ki vector with a zero entry. The actual start position of each i-th wire in the X-Y plane is defined by the pair of values: Xbeg = P0(2*(i-1)+1), Ybeg=P0(2*(i-1)+2), and the actual end position of each i-th wire in the X-Y plane is defined by the pair of values: Xbeg = P1(2*(i-1)+1), Ybeg=P1(2*(i-1)+2) (see the "Topology" section). This model expects the sizes of all Ki and Zi to be equal to each other and issues an error message if they differ.

P0, P1. These vector parameters display only if BWShape=2D. P0 contains pairs of (x, y) coordinates of the start point of each wire; P1 contains pairs of (x, y) coordinates of the end point of each wire. For example, the order of entries in vector P0 for N=2 is the following: {x1,y1,x2,y2} where x1 is the x-coordinate of the start point of wire #1, y1 is the y-coordinate of the start point of wire #1, x2 is the x-coordinate of the start point of wire #2, and y2 is the y-coordinate of the start point of wire #2. Also, the order of entries in vector P1 for N=2 is the following: {x1,y1,x2,y2} where x1 is the x-coordinate of the end point of wire #1, y1 is the y-coordinate of the end point of wire #1, x2 is the x-coordinate of the end point of wire #2, and y2 is the y-coordinate of the end point of wire #2

End coordinates of vector P1 fully define end Ksi coordinates of each wire as well as azimuthal position of the respective wire plane. See "Recommendations for use" for selection of start and end Ksi coordinates.

This model expects the number of P0,P1 entries to be equal to 2*N and issues an error message if they differ.

X1, Y1, Z1,... Xn, Yn, Zn. These vector parameters display only if BWShape=3D. They are vector "shape parameters" (Cartesian coordinates X, Y, and Z - see the "Topology" section), that specify the positions of each wire segment above the ground plane (located at Z=0) and define the shape of the bond wire. The number of these vector triplets is N. Vector values must be in braces. This model expects the sizes of each vector triplet (Xi, Yi and Zi) to be equal to each other, and auto-corrects their sizes.

IndModel. This switch allows switching between two representations of the model equivalent circuit: "RLC" (Resistor-Inductor-Capacitor) and "RL" (Resistor-Inductor). Setting IndModel to "RL" *significantly reduces computation time*, excluding capacitances from the equivalent circuit. At relatively low frequencies (for example, below 10 GHz) both values of IndModel yield close modeling results. The difference between modeling results depends on the proximity of the evaluation frequency to the self-resonance frequency (SRF) of tested wire. A model at the "RL" setting cannot predict the location of SRF, while a model at the "RLC" setting (default) is "SRF aware" and predicts SRF reasonably accurately (see the "Recommendations for Use" section).

Er, Tand. Parameters of material used for optional encasing. The defaults are Er=1, Tand=0.

Zoffset. Global vertical (Z) offset defines a value that adds to all z-coordinates of all points of each wire. The default ZOffset=0

Parameter Restrictions and Recommendations

1. The number of segments on each wire is user-defined but limited: It should be greater than or equal to 1 but less than or equal to 10.
2. This model implies that the number of bond wires is less than or equal to 512.
3. Heights of the start and end points of any wire over the ground may be set equal to zero, allowing the bond wire to touch ground. Z coordinates of intermediate points cannot be set to zero because BWIRES4 does not allow joints of segments to touch ground.
4. BWIRES4 subjects shape parameters (see the "Parameter Details" section) and locations of wire end points (P0, P1) to extended design rule control that prevents segments within separate wires and wires themselves from touching and intersecting (from getting closer than Dia). Also, the length of any segment cannot be less than Dia. You can constantly monitor the actual shape of wire in the Layout 3D View window.
5. Dia>0.1 micron
6. Er and Tand affect modeling results only if IndModel=RLC.

Implementation Details

Wires are considered a collection of infinitely thin, arbitrarily oriented, conducting, segmented filaments situated above infinite perfect ground. The actual diameter and material of the wire are accounted for at the evaluation of self-characteristics of each segment; mutual parameters neglect diameter and material. This model does not solve for distribution of charges/currents along or over wires; the approach is based on the assumption that the method of static-averaged potentials is applicable.

This model accounts for internal and external inductance of each wire segment; its evaluation of resistances is based on a skin-effect approach and the radiation resistance of a small dipole above ground. Evaluation of capacitances implies that the surrounding media is infinite over a perfect ground plane and uniformly filled with optional encasing material (the default is vacuum); electrical properties of adjacent dielectric objects are neglected. A value of Er other than 1 changes all capacitances in the equivalent circuit; nonzero Tand adds a shunt resistor to each capacitance throughout all equivalent circuits (see the "Equivalent Circuit" section).

Recommendations for Use

1. **User discretion is advised when the total length of wire is small.** Typically, short wires have even shorter segments. At the same time, the model assumes that each segment is represented by infinitely thin filament, which means Lseg >> Dia. The model issues a warning about possible loss of accuracy if Lseg<6*Dia.

   BWIRES4, unlike BWIRES3 and BWIRES2 allows you to assign to each wire its own number of linear segments. By doing so you can avoid the "accuracy may reduce" warnings for shorter wires.
2. If the following three factors are combined, loss of accuracy may occur:
   - The evaluation frequency is very high (for example, above 50 GHz)
   - The wires are relatively long (above 1 mm)
   - The low profile wires span a substrate of relatively high permittivity (~8 and above). Even in this case, SRF is usually predicted well within 5-7% error; however, the error in phases of "crosstalk" scattering coefficients might be large due to additional unaccounted substrate coupling.
3. The recommended range of number of segments is 3≤Nseg≤6. Note that the number of segments may affect the position of calculated series resonance frequency (SRF) due to adopted approximations of constant distribution of charge along the segments.
4. Setting the IndMode parameter to "RL" *substantially reduces calculation time* (this may be advantageous if the number of bond wires exceeds 20) but excludes effects of capacitances coupling (these effects are responsible for self-resonances at higher frequencies). "RL" leaves in place frequency-dependence of self-inductances ([4]); radiation resistance ([3]) also remains added to the self-resistance of each wire.
5. Note that BWIRES4 can comprise two models. One model is selected by setting BWShape=3D to allow you to position the bond wire arbitrarily in 3D space. Another model is selected by setting BWShape=2D to confine each bond wire to its own vertical plane. Because these models differ significantly in their interfaces, parameters are not inherited when switching from one model to another.
6. Also note that initially placing BWIRES4 on a schematic provides a fully operational instance at the default settings N=1 and BWShape=3D. Changing N only automatically creates a set of shifted identical wires; however, this is done only for reference. You must fully specify all vector parameters and provide all needed entries for each parameter in accordance with your target design.
7. Modifying the size of any coordinate vector parameter causes BWIRES4 to turn on an auto-correction mechanism to keep vectors-siblings of the affected parameter at the same size. For example, if you decrease the number of segments of wire #5 from 5 segments to 3 segments and start from vector X5 (3D option), the number of X5 entries is reduced from 6 to 4. When modification of X5 is completed, auto-correction applies to vector parameters Y5 and Z5. Both Y5 and Z5 are truncated from size 6 to 4. Alternatively, increasing the number of segments (adding new X5 entries) causes automatic extension of Y5 and Z5 with a default step of 50 microns between new entries.
8. BWShape=2D: About K-coordinates of the first and last points of a wire. BWIRES4 shifts all wires to the origin of the K axis, so you may want to design wire so the first point has Ki(1)=0. Also, you can set Ki(n) (where n is the last point entry number) to a zero value because this model automatically recalculates Ki(n) from respective X and Y entries of vector P1 and redisplays this value as the last entry of vector Ki.

Layout

The layout for this cell has hard-coded model layers. When you first use this layout cell, a layer named "BondWires2" is added to your drawing layer and model layer list if they are not already there. Using the model layer mapping, you can assign these layers to draw on any drawing layer. The via is drawn the size of the "D1" parameter.

This model has several layout-only parameters that control how the pad draws. To access these parameters, select the item in the layout, right-click and choose **Shape Properties** to display the Cell Options dialog box, then click the **Parameters** tab.

Name	Value	Units	Description
ShowNodeNum	0	Integer	Layout displays node numbers only if this value is non-zero
PointFaces	0	Integer	Faces are points, not line segments, only if this value is non-zero
OrtTip	0	Integer	If zero, the wire tip cut is horizontal, otherwise the tip cut is orthogonal to the end segment axis

Note that option BWShape=2D forbids implementation of strictly vertical wires and wires with coinciding X-Y projections of start and end points. Attempt to create these wires in 2D causes layout interruption and is accompanied with error text messages outputted into layout window. These configurations can be created after selection BWShape=3D.

3D EM Layout

This model includes OrtTip, a layout-only parameter that controls how the tip of a wire is cut. If OrtTip=0 (default), the tip is cut horizontally (this provides a smooth transition to some on-layer metal patch or strip). If OrtTip≠0, the wire tip is cut orthogonally to the last/first segment axis. An orthogonal cut improves the transition to metal connectors if the last/first segments are horizontal or almost horizontal. To access this parameter, select the item in the layout, right-click and choose **Shape Properties** to display the Cell Options dialog box, then click the **Parameters** tab.

Note that implementation of 3D EM pcells strictly preserve undistorted cross-sectional view of a wire only if all wire segments belong to the same vertical plane. Off-plane deviation of segments may cause distortion of displaced wires.

References

[1] Grover, F.W., Inductance Calculations, Mineola, New York: Dover Publishing, 2004

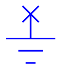
[2] Bahl, I.J., Lumped Elements for RF and Microwave Circuits, Norwood, MA: Artech House, 2003

[3] Balanis C.A., Antenna Theory: analysis and design, 2nd edition: Wiley, 1997

[4] S.Ramo and J.R.Whinnery, Fields and Waves in Modern Radio, 1st ed., General Electric Advanced Engineering Program, New York, NY; London, England: J.Wiley and Sons, Inc.; Chapman and Hall, 1944

Ground (Closed Form): GND

Symbol



Implementation Details

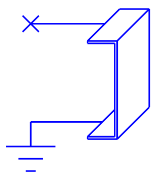
Implements an ideal ground.

Layout

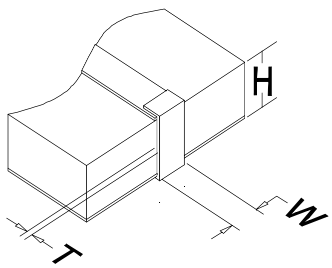
This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Ribbon Wire Ground Strap (Closed Form): Gnd_Strap

Symbol



Structure



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	IC1
W	Ribbon width	Length	0 um
T	Ribbon thickness	Length	0 um
H	Substrate thickness	Length	0 um
Rho	Resistivity relative to gold		0

Implementation Details

This circuit component models a ground conductor having a rectangular cross section. The model is implemented as a lumped element series inductor and resistor whose values are derived from the dimensions and resistivity of the ribbon.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Restrictions

Recommended	$0.005 \leq W/T \leq 200$ Required
$0 \leq Rho \leq 100$ Recommended	$L/\lambda_o \leq 0.2$ Recommended

Recommendations for Use

Discontinuity models function most accurately when attached to lines that match their corresponding edges. Directly connecting discontinuity models to one another reduces their accuracy.

References

[1] Greenhouse, H.M, "Design of Planar Rectangular Microelectronic Inductors" IEEE Trans. on Parts, Hybrids and Packaging Vol PHP-10 No.2, June 1974 pp. 101-109

[2] U.S. Department of Commerce, Circular of the National Bureau of Standards C74, Radio Instruments and Measurements, "Calculation of Inductance", Jan 1, 1937

[3] Grover, F.W., "Inductance Calculations", Van Nostrand, Princeton, NJ 1946.

[4] Caulton, M., "Hybrid Integrated Lumped-Element Microwave Amplifiers" IEEE Trans on Electron Devices, Vol ED-15, July 1968, pp.459-466

[5] Wadell, Brian C. "Transmission Line Design Handbook" Artech 1991

Flat Bent Package Lead "Gull-Wing" (Closed Form): GWLEAD

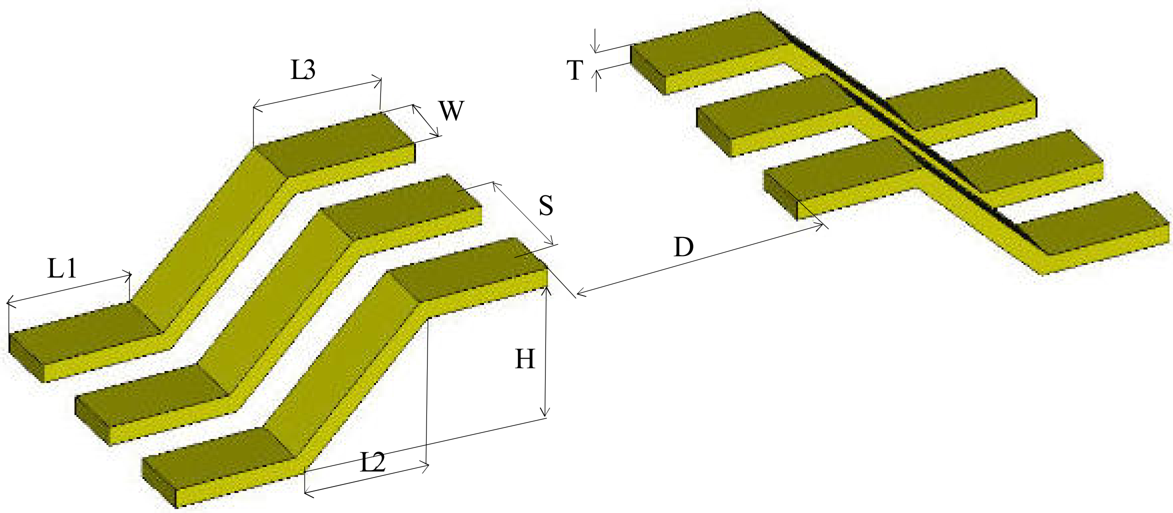
Symbol

Summary

GWLEAD is a dynamic model of flat bent package lead known as "Gull-Wing". You can specify the number of leads, lead pitch, and place leads on one or two parallel sides of a package.

This element is only for use in 3D EM documents. It is intended to create a 3D parameterized cell for Gull-Wing lead(s) for 3D EM analysis. See "Using 3D EM Elements " for details on using 3D pCells with Analyst.

Topology



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	IC1
N	Number of leads		1
W	Lead width	Length	W[a]
T	Lead thickness	Length	W[a]
L1	Length of lower section	Length	L[a]
L2	Length of footprint of intermediate section	Length	L[a]
L3	Length of upper section	Length	L[a]
S	Lead pitch	Length	L[a]
D	Distance between lead rows	Length	0 um
Rho	Lead metal bulk resistivity relative to gold		Rho[a]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See the Microwave Office Layout Guide for details.

Parameter Details

N. Parameter N specifies the number of leads at one side of a package. If D>0 and a mirrored set of leads is used, the total number of leads is 2N.

S. Parameter S specifies lead pitch equal to the distance between the central lines of adjacent leads (see the "Topology" section).

D. When greater than 0, parameter D adds a second set of N leads, mirroring the first set, and specifies the distance between the ends of upper sections on opposite leads (see the "Topology" section). When D is less than or equal to 0, there are only N leads.”.

Parameter Restrictions and Recommendations

1. The distance between lead rows D can be positive, negative, or zero. If D is negative or zero, the 3D cell draws only one row of leads; if D is positive, the 3D cell draws two rows of leads separated by distance D.
2. Parameter L2 can be either positive or zero.
3. Parameter T has a lower limit of 0.1 micron.

Layout

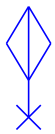
The layout for this cell has hard-coded model layers. When you first use this layout cell, a layer named "GWlead" is added to your drawing layer and model layer list (if they are not already there). Using the model layer mapping, you can assign these layers to draw on any drawing layer. This element is not visible in any 3D layout view other than that of a 3D EM structure.

3D EM Layout

This element has a layout cell specifically for Analyst 3D EM layouts. See “Using 3D EM Elements ” for details on using 3D pCells with Analyst.

Named Connector: NCONN

Symbol



Summary

NCONN is used to create schematic connectivity by name instead of a wire. See “Element Connection by Name” for details.

Parameters

Name	Description	Unit Type	Default
*ID	Network ID	Text	NC1
Name	Connector Name	Text	CONN

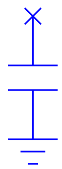
** indicates a secondary parameter*

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

Open Circuit (Closed Form): OPEN

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	J1

Implementation Details

Implements an ideal open circuit.

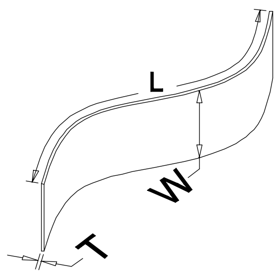
Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Flat Wire Ribbon (Closed Form): Ribbon

Symbol

Topology



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	IC1
L	Ribbon length	Length	0 um
W	Ribbon width	Length	0 um
T	Ribbon thickness	Length	0 um
Rho	Metal resistivity relative to gold		0

Implementation Details

This circuit component models a length of conductor having a rectangular cross section. The model is implemented as a lumped element series inductor and resistor whose values are derived from the dimensions and resistivity of the wire.

Restrictions

$0.01 \geq W/T \geq 100$ Recommended	Rho > 0 Recommended
$0.005 \geq W/T \geq 200$ Required	$L/\lambda_o \leq 0.2$ Recommended

Limitations on 3D EM pcell parameters RotAng and ShiftZ: Both RotAng and ShiftZ must be greater than or equal to zero. This pCell automatically changes the sign of negative RotAng and ShiftZ. Do not use high RotAng values that make the upper tip of the ribbon almost horizontal-- this may cause visual distortion. NI AWR recommends keeping RotAng below 60-65 degrees.

Recommendations for Use

Discontinuity models function most accurately when attached to lines that match their corresponding edges. Directly connecting discontinuity models to one another reduces their accuracy.

Layout

The layout for this cell has hard-coded model layers. When you first use this layout cell, a layer named "Ribbon" is added to your drawing layer and model layer list (if they are not already there). Using the model layer mapping, you can assign these layers to draw on any drawing layer. The via is drawn the size of the "D1" parameter.

3D EM Layout

This element has a layout cell specifically for Analyst 3D EM layouts. See "Using 3D EM Elements " for details on using 3D pCells with Analyst.

This model has several layout-only parameters that control how the 3D pCell draws. You access these parameters by selecting the item in the layout, right-clicking and choosing **Shape Properties** to display the Cell Options dialog box, then clicking the **Parameters** tab.

Name	Value	Units	Description
H	0	Layout	If H <=0 then the height of the midpoint of the ribbon is L/2, otherwise this height is the value entered.
ShiftZ	0	Layout	Defines z-displacement of wire above substrate (ShiftZ≥0)
RotAng	0	Layout	Defines angle of wire rotation (RotAng≥0)
Nd	7	Layout	Number of drawing points

References

[1] Greenhouse, H.M, "Design of Planar Rectangular Microelectronic Inductors" IEEE Trans. on Parts, Hybrids and Packaging Vol PHP-10 No.2, June 1974 pp. 101-109

[2] U.S. Department of Commerce, Circular of the National Bureau of Standards C74, Radio Instruments and Measurements, "Calculation of Inductance", Jan 1, 1937

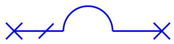
[3] Grover, F.W., "Inductance Calculations", Van Nostrand, Princeton, NJ 1946.

[4] Caulton, M., "Hybrid Integrated Lumped-Element Microwave Amplifiers" IEEE Trans on Electronic Devices, Vol ED-15, July 1968, pp.459-466

[5] Wadell, Brian C. Transmission Line Design Handbook, Artech 1991

2-Node Short Circuit (Closed Form): SHORT

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	J1

Implementation Details

Implements an ideal short circuit.

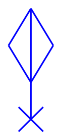
Layout

This element uses a special layout cell for a short circuit. The layout cell allows the elements connected on either side of the element to look through this element. For example, when a MLIN is hooked to a TFCM (capacitor) model and the project has bridge code configured, the line draws the proper interconnect between the line and the cap. If this element is placed between the line and the cap, the layout still draws the same.

You typically do not assign artwork cells to these items.

Thermal Node Identifier: TNODE

Symbol



Summary

TNODE is used to identify any nodes of models that are considered thermal nodes of that model. It is primarily used when calculating measurements for the model, such as the power dissipated for the model. This is an important quantity when using a thermal simulator. In these models, thermal quantities are represented as current and voltage and should not be included when calculating the electrical quantities of the model. This element on a node tells the software to skip those nodes when calculating electrical properties. If using a thermal solver, you should place this element on any thermal node of models you are using.

Parameters

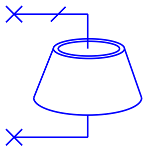
Name	Description	Unit Type	Default
ID	Network ID	Text	NC1

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Tapered Via Hole (Closed Form): TVIA

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	V1
D1	Hole diameter at node 1	Length	W[a]
D2	Hole diameter at node 2	Length	W
H	Substrate thickness	Length	H[a]
T	Metal thickness	Length	T[a]
RHO	Metal bulk resistivity normalized to gold		Rho[a]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory.

Implementation Details

This component models a tapered cylindrical plated-through via hole. The model is implemented as a lumped element series inductor and resistor whose values are derived from the dimensions and resistivity of the wire.

Recommendations for Use

Discontinuity models function most accurately when attached to lines that match their corresponding edges. Directly connecting discontinuity models to one another reduces their accuracy.

Layout

The layout for this cell has hard-coded model layers. When you first use this layout cell, a layer named "TVIA" is added to your drawing layer and model layer list (if they are not already there). Using the model layer mapping, you can assign these layers to draw on any drawing layer. The via is drawn the size of the "D1" parameter.

3D EM Layout

This element has a layout cell specifically for Analyst 3D EM layouts. See "Using 3D EM Elements " for details on using 3D pCells with Analyst.

This model has several layout-only parameters that control how the 3D pCell draws. You access these parameters by selecting the item in the layout, right-clicking and choosing **Shape Properties** to display the Cell Options dialog box, then clicking the **Parameters** tab.

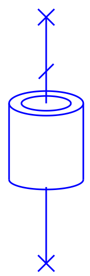
Name	Value	Units	Description
NumSideCell	6	Integer	Number of sides used to draw the cell, minimum = 4

References

[1] M.Goldfarb and R.Pucel, "Modeling Via Hole Grounds in Microstrip," IEEE Microwave and Guided Wave Letters., vol. 1, No.6, pp. 135-137.

Cylindrical Via (Closed Form): VIA

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	V1
D	External diameter of via (external diameter of via cylinder)	Length	L[a]
H	Substrate thickness	Length	H[a]
T	Thickness of metal (thickness of via cylinder wall)	Length	T[a]
RHO	Metal bulk resistivity normalized to gold		Rho[a]
*SNAME	Via drawing definition		Default

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See the Microwave Office Layout Guide for details.

** indicates a secondary parameter*

Implementation Details

This model is implemented as a lumped element series inductor and resistor whose values are derived from the dimensions and resistivity of the hollow cylindrical conductor.

RHO is a ratio of actual metal bulk resistivity to gold bulk resistivity, that is, the increase in RHO results in an increase of series resistor. This model allows a perfectly conducting via at RHO=0.

Recommendations for Use

Discontinuity models function most accurately when attached to lines that match their corresponding edges. Directly connecting discontinuity models to one another reduces their accuracy.

Layout

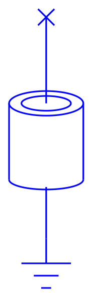
VIA models draw according to the SNAME parameter setting on the via, as configured in the LPF. If this parameter is left blank the via definition in the LPF is used to define the via layers. See "Via Definitions" for details. This typically works well if there is only one via type in the design. If you need to have different via layers, the best approach is to create structures in the LPF for each type, then set the SNAME parameter to match the structure name in the LPF. See "Structure Type Definitions" for more information.

References

[1] Goldfarb M.E., Pucel R.A., "Modeling Via Hole Grounds in Microstrip" IEEE Microwave and Guided Wave Letters, Vol. 1, No.6, June 1991, pp. 135-137

Cylindrical One Port Via (Closed Form): VIA1P

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	V1
D	External diameter of via (external diameter of via cylinder)	Length	L[a]
H	Substrate thickness	Length	H[a]
T	Thickness of metal (thickness of via cylinder wall)	Length	T[a]
RHO	Metal bulk resistivity normalized to gold		Rho[a]
*SNAME	Via drawing definition		Default

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See the Microwave Office Layout Guide for details.

** indicates a secondary parameter*

Implementation Details

Parameters are the same as the VIA element.

Recommendations for Use

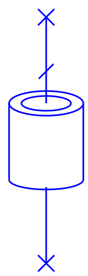
Discontinuity models function most accurately when attached to lines that match their corresponding edges. Directly connecting discontinuity models to one another reduces their accuracy.

Layout

VIA models draw according to the SNAME parameter setting on the via, as configured in the LPF. If this parameter is left blank the via definition in the LPF is used to define the via layers. See "Via Definitions" for details. This typically works well if there is only one via type in the design. If you need to have different via layers, the best approach is to create structures in the LPF for each type, then set the SNAME parameter to match the structure name in the LPF. See "Structure Type Definitions" for more information.

Single Via in Multilayer Board: VIAM

Symbol



Summary

VIAM models an isolated via in a multilayer printed circuit board. This model assumes that all dielectric layers crossed by the via are separated by infinite perfect conducting grounded planes. Each ground plane has an antipad and the via passes through the center of each antipad. Capture pads are located at via ends. The environment of each capture pad is user-controlled via model parameters. These parameters allow implementation of all typical via configurations: through, buried, and blind. You can take a pad out from one or both via ends and implement a series connection of several vias.

It is imperative that the via cross at least one ground plane with antipad.

Multilayer via analysis is based on the theory of long cylindrical antenna excited by a frill of magnetic current radiating in a parallel plate waveguide, and on the theory of monopole radiating into a parallel plate waveguide. This model also uses quasi-static FEM analysis for accurate evaluation of capacitances of complex conducting configurations in a multilayered dielectric.

Topology

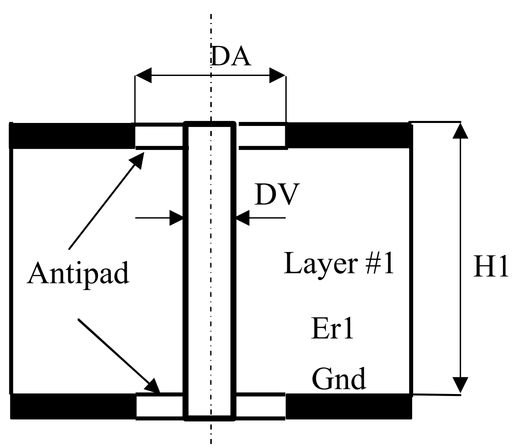


Figure 1a. Basic stackup (N=1)

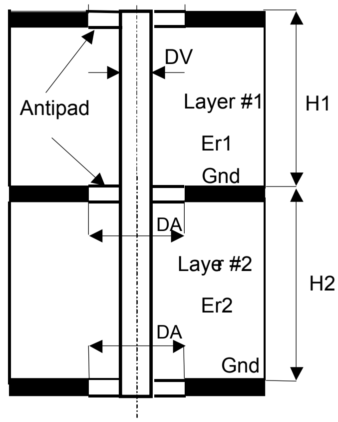


Figure 1b. Basic stackup (N=2)

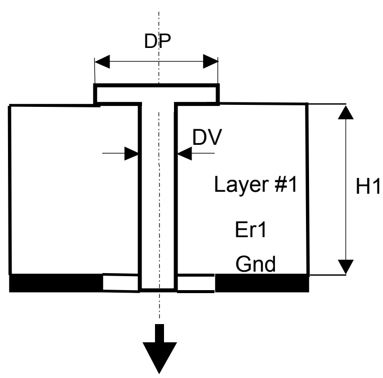


Figure 2. Option "Board Top"

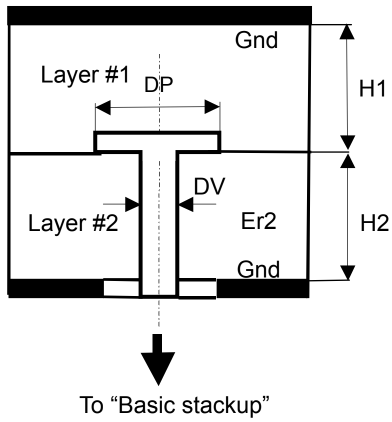


Figure 3. Option "Inside Board with Pad"

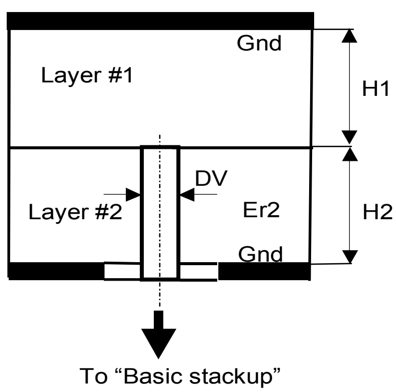


Figure 4 .Option "Inside Board no Pad"

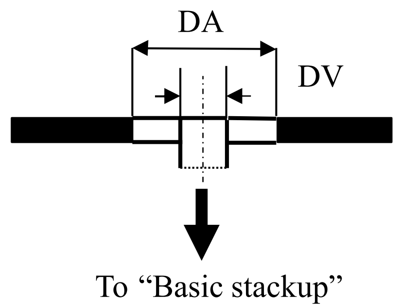


Figure 4_1 .Option "Inside Antipad no Pad"

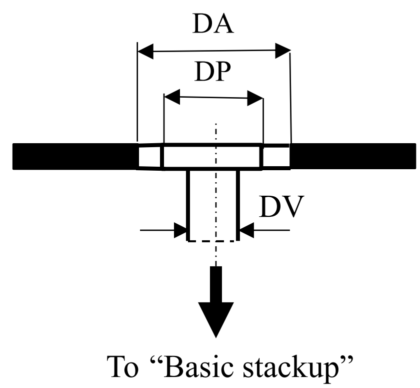


Figure 4_2 .Option "Inside Antipad with Pad"

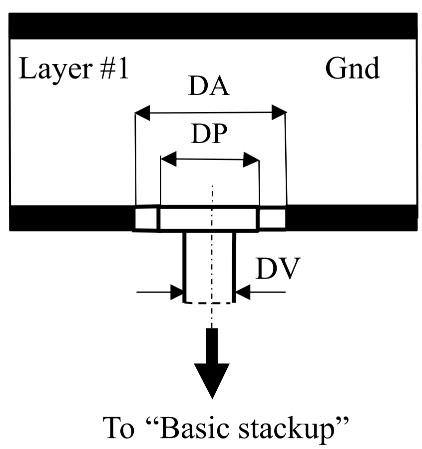


Figure 4_3 .Option "Inside Board & Antipad with Pad"

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	V1
N	Number of dielectric layers		1
DV	External diameter of via hole	Length	W[a]
DA	Diameter of antipad	Length	L[a]
DPT	Diameter of top capture pad	Length	L[a]
DPB	Diameter of bottom capture pad	Length	L[a]
TP	Thickness of capture pad	Length	T[a]
H	Heights of dielectric layers (vector)	Length	{0.1 mm}
Er	Relative dielectric constants of dielectric layers (vector)		{1}
Rho	Via metal bulk resistance relative to gold		1
TopCap	Switch "Board Top"/"Inside Board with Pad/Inside Board No Pad/Inside Antipad No Pad"/"Inside Antipad with Pad"/"Inside Board & Antipad with Pad"	Length	"Board Top"
BotCap	Switch "Board Top"/"Inside Board with Pad/Inside Board No Pad/Inside Antipad No Pad"/"Inside Antipad with Pad"/"Inside Board & Antipad with Pad"	Length	"Board Top"
*Acc	Switch "Default/High"		"Default"

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See "Default Values" for details.

** indicates a secondary parameter*

Parameter Details

N. Defines the number of dielectric layers in a multilayered board separated by ground planes (see the "Basic stackup" figures in the "Topology" section). It is important to note that *only layers from the "Basic stackup" are counted toward N*. The actual number of layers VIAM uses depends on the options selected with the TopCap and BotCap parameters. Note that VIAM allows a zero value of N (see "Parameter Restrictions and Recommendations".)

DPT, DPB. The diameters of the capture pads attached to the via top end (Port #1) and to the via bottom end (Port #2). Certain TopCap and BotCap parameter options imply that the respective pad is missing.

H, Er. These vector parameters define the height and dielectric constant of every dielectric layer in the board stackup. Note that these vectors always include heights and dielectric constants of layers defined by the TopCap and BotCap parameters. TopCap adds values to the vector head while BotCaps adds values to the vector tail. You must provide correct values for each layer the model uses.

TopCap, BotCap. Each of these switching parameters has six user-selectable options. These options allow the *addition* of via caps of various configurations to each via end of the basic multilayer configuration presented in the "Basic stackup" figures in the "Topology" section.

TopCap defines the configuration of the via cap at Port #1 and BotCap defines the configuration of the via cap at Port #2.

All configurations of TopCap and BotCap (except "Inside Board no Pad" and "Inside Antipad no Pad") assume that a capture pad is present.

Note that TopCap and BotCap can be set independently to different options. The size of vectors H and Er are *defined by both selected options*.

All available configuration/options are presented in the "Topology" section in figures with corresponding captions. The "Board Top" option requires the addition of *one more value* to vectors H and Er in addition to the layer values needed for "Basic stackup"; the "Inside Board with Pad " and "Inside Board no Pad" options require the addition of *two more values* to vectors H and Er; the "Inside Antipad no Pad" option *does not add any values* to H and Er because it actually leaves the via end "as is" in "Basic stackup. The "Inside Antipad with Pad" and "Inside Board & Antipad with Pad" options insert a capture pad inside the top or bottom antipad; "Inside Antipad with Pad" (as well as "Inside Antipad no Pad") *do not add any values* to H and Er. The "Inside Board & Antipad with Pad" option requires the addition of *one more value* to vectors H and Er in addition to the layer values needed for "Basic stackup".

Acc. The default value of the Acc parameter excludes the contribution of higher modes into the via model. Setting Acc=High adds the evaluation of series representing the contribution of higher modes and slightly increases simulation time. In most cases Acc=Default provides sufficient accuracy, but for very long vias crossing many dielectric layers Acc=High might be beneficial.

Parameter Restrictions and Recommendations

1. The number of layers N in Basic Stackup must be 0<=N<=30. If N=0, then "Basic stackup" is basically a *single ground plane with an antipad* that separates any of two configurations selected by the TopCap and Bottom cap parameter options. For example, selecting N=0, TopCap="Board Top", and BotCap="Board Top" implements a *through via in a two-layer board* with a ground plane (containing an antipad) separating adjacent dielectric layers. The lengths of vectors H and Er amount to 2, and you must specify two heights and two dielectric constants. Selecting N=0, TopCap="Board Top", BotCap="Inside Board with Pad" implements a *blind via in a three-layer board* with a ground plane (containing an antipad) separating the first and second (counting from top) dielectric layers. The length of vectors H and Er amount to 3, and you must specify three heights and three dielectric constants.
2. This model assumes that only dominant mode TMo propagates in each layer over all frequency sweep. The heights of all layers are checked for normal operation below the cutoff frequency of the higher mode propagating in respective parallel-plate waveguide. If the evaluation frequency exceeds the cutoff threshold, the model generates an error and reports the highest sweep frequency allowed for the specified separation between ground planes.

Implementation Details

The via is modeled as a set of segments contained within ground-separated dielectric layers. Each segment is modeled as a cylindrical antenna/monopole radiating between perfectly conducting parallel plates. Modeling techniques used are based on publications [1]-[4] [6–71]. Via cap capacitance is treated by means of a quasi-static FEM technique based on [5] [6–72]. Port reference planes are located at the centers of capture pads. Port #1 is associated with the top end of the via and port #2 is associated with the bottom end of the via. The schematic symbol pin marked with a "slash" is associated with Port #1.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

Recommendations for Use

The following are examples of various via implementations:

1. Implementation of through via, N=2

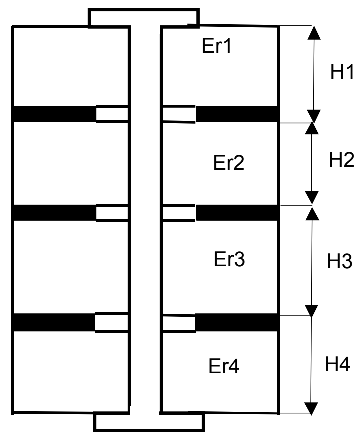


Figure 5. Via Configuration

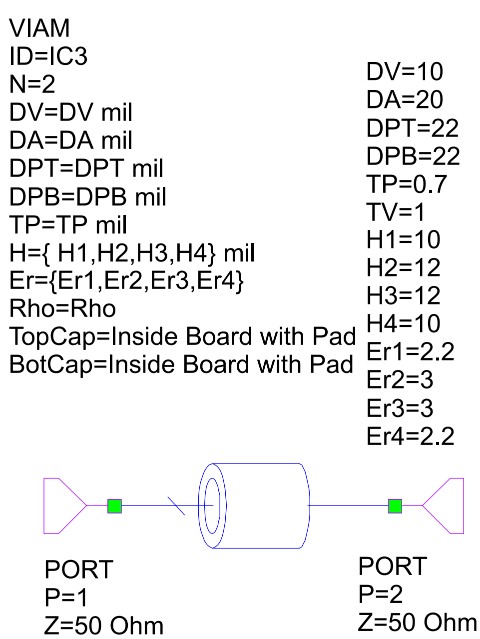


Figure 6. Schematic

2. Implementation of blind via, N=2

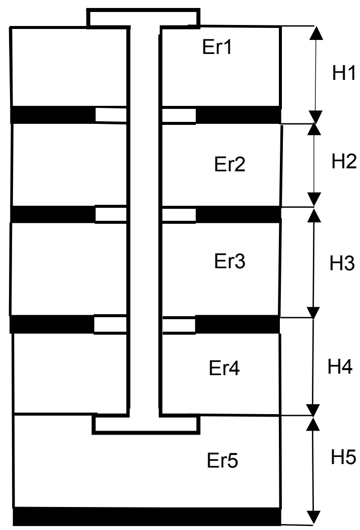


Figure 7. Via Configuration

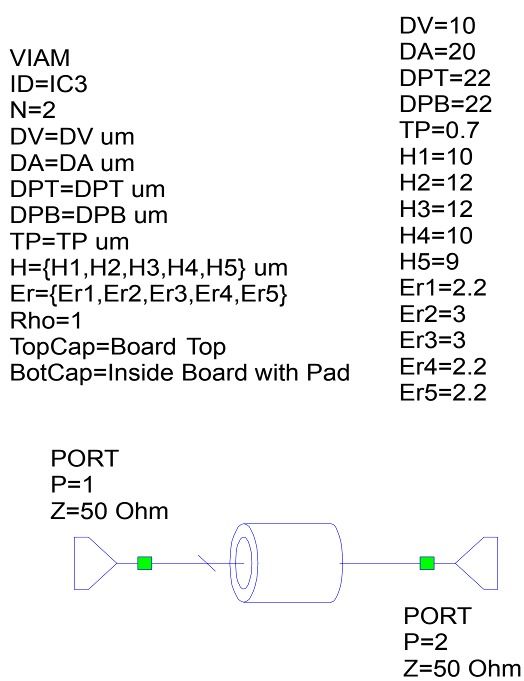


Figure 8. Schematic

3. Implementation of buried via, N=2

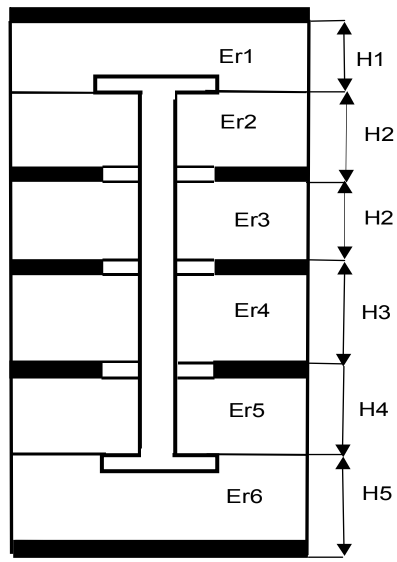


Figure 9. Via Configuration

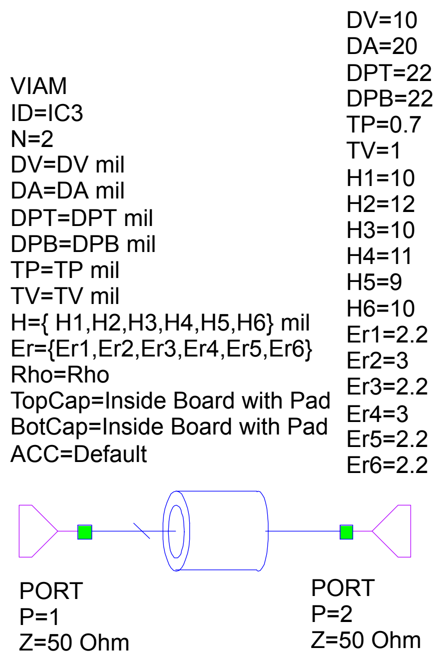


Figure 10. Schematic

4. Implementation of through via, N=0

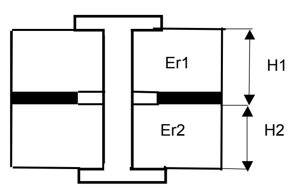


Figure 11. Via Configuration

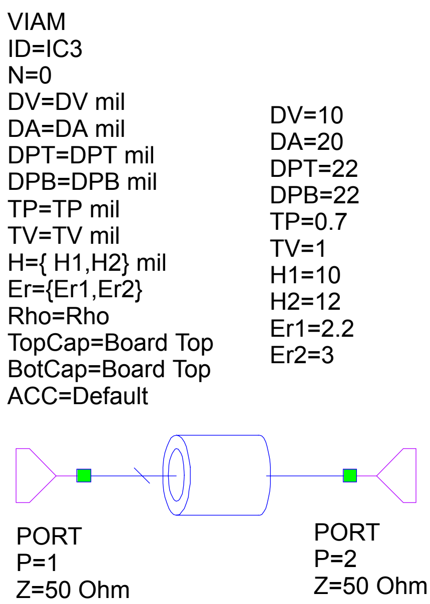


Figure 12. Schematic

5. Implementation of buried via, N=0

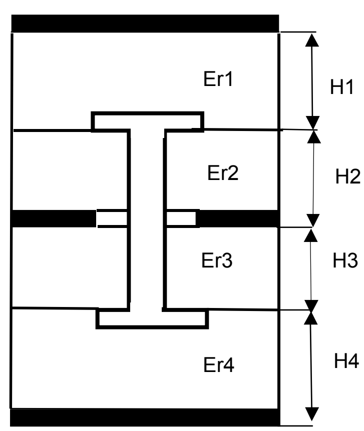


Figure 13. Via Configuration

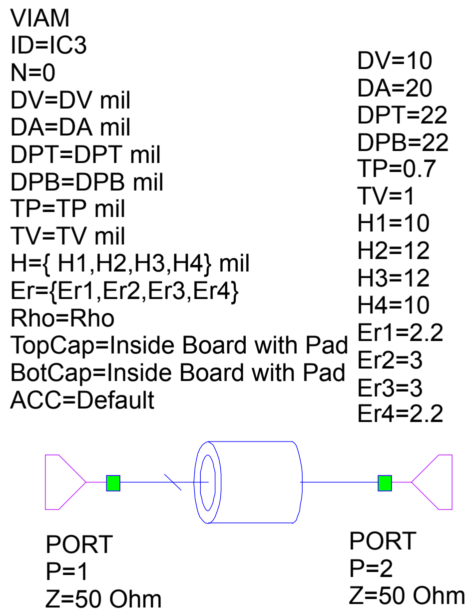


Figure 14. Schematic

6. Series connection of two vias can be implemented if the first VIAM BotCap parameter is "Inside Board with Cap" and the second VIAM TopCap parameter is "Inside Board no Cap". These settings avoid pad doubling.
7. Sometimes it may be beneficial if a via is connected to a coplanar waveguide line running across the ground plane. To provide this connection set either TopCap or BotCap to "Inside Antipad no Pad". This option sets the via cap as it appears in "Basic stackup" and assumes that capture cap impact at this end is low. To account for pad inside antipad (see the "Topology" section) set TopCap/BotCap either to "Inside Antipad with Pad" or to "Inside Board & Antipad with Pad".
8. In general, you can create a stack of vias using the two previous recommendations. If the impact of pads inside antipads is neglected, you should select "Inside Antipad no Pad" for stacking; to account for multiple pads inside antipads stack up several VIAMs with N=1 and apply (in turns) Top/BotCap = "Inside Antipad with Pad" and "Inside Antipad no Pad".
9. This model uses a disk cache, so if a project contains multiple instances of identical VIAM, only one instance simulates and saves results to the cache, and all identical (having the same parameter set) VIAM simply fetch these results from cache. Note that the cache keeps saved data, and any project on the same computer that contains VIAM with the same set of parameters reuses the cached data.

References

[1] Qizheng Gu, Y. Eric Yang, and M.Ali Tassoudji, "Modeling and Analysis of Vias in Multilayered Integrated Circuits," *IEEE Trans. on Microwave Theory and Tech.*, vol. 41, February 1993, pp. 206-214

[2] Qizheng Gu, M.Ali Tassoudji et al, "Coupled Noise Analysis for Adjacent Vias in Multilayered Digital Circuits," *IEEE Trans. on Circ. and Syst.*, vol. 41, December 1994, pp. 796-804

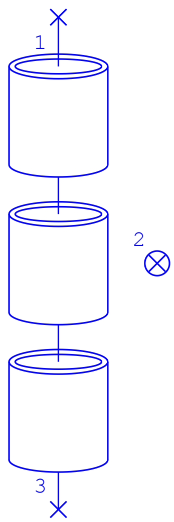
[3] B.Tomasic and A. Hessel, "Linear Array of Coaxially Fed Monopole Elements in a Parallel Plate Waveguide," *IEEE Trans. on Antennas and Prop.*, vol. 36, April 1988, pp. 449-462

[4] M. Goldfarb and R. Pucel, "Modeling Via Hole in Microstrip," *IEEE Microwave and Guided Wave Lett.*, vol. 1, June 1991, pp. 135-137

[5] David Meeker home page: http://femm.foster-miller.net/~dmeeker/ Follow this link for information: http://femm.foster-miller.net/Archives/readme.htm FEMM Manual: http://femm.foster-miller.net/Archives/doc/manual.pdf

Multiport Single Via in Multilayer Board: VIAM2

Symbol



Summary

VIAM2 models an isolated via in a multilayer printed circuit board. Similar to VIAM, VIAM2 assumes that all dielectric layers crossed by the via are separated by infinite perfect conducting grounded planes. VIAM2 differs from VIAM in its ability to attach multiple VIAMC elements representing per-layer coupling between two VIAM2 elements. VIAM2 is a dynamic model; its schematic symbol changes (adds/removes terminals) when the number of board layers changes. See VIAM for additional details.

Topology

See VIAM.

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	V1
N	Number of dielectric layers		1
DV	External diameter of via hole	Length	W[a]
DA	Diameter of antipad	Length	L[a]
DPT	Diameter of top capture pad	Length	L[a]
DPB	Diameter of bottom capture pad	Length	L[a]
TP	Thickness of capture pad	Length	T[a]
H	Heights of dielectric layers (vector)	Length	{0.1 mm}
Er	Relative dielectric constants of dielectric layers (vector)		{1}
Rho	Via metal bulk resistance relative to gold		1

Name	Description	Unit Type	Default
TopCap	Switch "Board Top"/"Inside Board with Pad/Inside Board No Pad/Inside Antipad No Pad"	Length	"Board Top"
BotCap	Switch "Board Top"/"Inside Board with Pad/Inside Board No Pad/Inside Antipad No Pad"	Length	"Board Top"
*Acc	Switch "Default/High"		"Default"

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See "Default Values" for details.

** indicates a secondary parameter*

Parameter Details

N. This parameter defines the number of dielectric layers in a multilayered board separated by ground planes (see the "Basic stackup" figure in the "Topology" section of VIAM). It is important to note that *only layers from the "Basic stackup" are counted toward N*. The actual number of layers VIAM2 uses depends on the options selected with the TopCap and BotCap parameters (see the following details for these parameters.) Note that (contrary to VIAM) VIAM2 does not allow a zero value of N (see "Parameter Restrictions and Recommendations".)

DPT, DPB. The diameters of the capture pads attached to the via top end (Port #1) and to the via bottom end (Port # N+2). Certain options of the TopCap and BotCap parameters imply that the respective pad is missing.

H, Er. These vector parameters define the height and dielectric constant of every dielectric layer in the board setup. Note that these vectors always include heights and dielectric constants of layers defined by the TopCap and BotCap parameters. TopCap adds values to the vector head while BotCap adds values to the vector tail. You must provide correct values for each layer the model uses.

TopCap, BotCap. Each of these switching parameters has four user-selectable options. These options allow the *addition* of via caps of various configurations to each via end of the basic multilayer configuration presented in the "Basic stackup" figure in the "Topology" section.

TopCap defines the configuration of the via cap at Port #1 and BotCap defines the configuration of the via cap at Port #N+2.

All available configuration/options are presented in the "Topology" section (see VIAM) in figures with corresponding captions. The "Board Top" option requires the addition of *one more value* to vectors H and Er in addition to the layer values needed for "Basic stackup"; the "Inside Board with Pad " and "Inside Board no Pad" options require the addition of *two more values* to vectors H and Er; the "Inside Antipad no Pad" option does not add any values to H and Er because it actually leaves the via end "as is" in "Basic stackup."

Note that TopCap and BotCap can be set independently to different options. The size of vectors H and Er are *defined by both selected options*.

Acc. The default value of this parameter excludes the contribution of higher modes into the via model. Setting Acc=High adds the evaluation of series representing the contribution of higher modes and slightly increases simulation time. In most cases Acc=Default provides sufficient accuracy but for very long vias crossing many dielectric layers Acc=High might be beneficial. Note that the value of Acc should match the value of Acc set for attached VIAMC elements.

Parameter Restrictions and Recommendations

1. The number of layers N in the Basic Stackup must be 1<=N<=30. Note that N=0 is forbidden for VIAM2 but allowed for VIAM.
2. This model assumes that only dominant mode TMo propagates in each layer over all frequency sweep. The heights of all layers are checked for normal operation below the cutoff frequency of the higher mode propagating in respective parallel-plate waveguide. If the evaluation frequency exceeds the cutoff threshold the model generates an error and reports the highest sweep frequency allowed for the specified separation between ground planes.

Implementation Details

The via is modeled as a set of segments contained within ground-separated dielectric layers. Each segment is modeled as cylindrical antenna/monopole radiating between perfectly conducting parallel plates. Modeling techniques used are based on publications [1]-[4] [6–81]. Via cap capacitance is treated by means of a quasi-static FEM technique based on [5] [6–81]. Port reference planes are located at the centers of capture pads. Port #1 is associated with the top end of the via and Port #N+2 is associated with the bottom end of the via. Port #2..#N+1 are coupling ports. The only element that should be connected to any of these ports is VIAMC (see the example in the "Recommendations for Use" section).

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

Recommendations for Use

1. If no coupling, VIAMC elements are connected to Ports #2..#N+1, then VIAM2 behaves similar to the VIAM element.
2. To provide the most accurate evaluation of coupling, all VIAM2 elements connected with VIAMC elements must have identical parameters DV, DA, and Acc.
3. Restriction N>=1 (at least one dielectric layer bounded by two ground planes) is mandatory because each VIAMC element evaluates coupling due to TMo mode propagating in the respective layer. VIAM2 assumes that this coupling is predominant and neglects coupling between via caps (see VIAM, VIAMD).
4. Building a network of coupled vias requires one VIAMC element per layer, per each pair of VIAM2 elements included in a coupled set of vias. For efficiency, a good practice may be to create separate subcircuits for each layer and connect VIAM2 elements to subcircuits using named connectors (NCONN) and named ports (PORT_NAME). See the following example.
5. This model uses a disk cache, so if a project contains multiple instances of identical VIAM2 only one instance simulates and saves results to the cache, and all identical (having the same parameter set) VIAM2 elements simply fetch these results from cache. Note that the cache keeps saved data, and any project on the same computer that contains VIAM2 with the same set of parameters reuses the cached data.

The following example demonstrates how coupling between vias in a PCB may affect coupling between signal traces. In this example, three edge-coupled traces on top of a PCB board connect to three edge-coupled striplines inside the board. The board is comprised of four FR-4 layers. Ground planes separate dielectric layers and the bottom of the board is also metallized. Microstrips and striplines are connected with three blind vias. Note that captions reside **under** the figures.

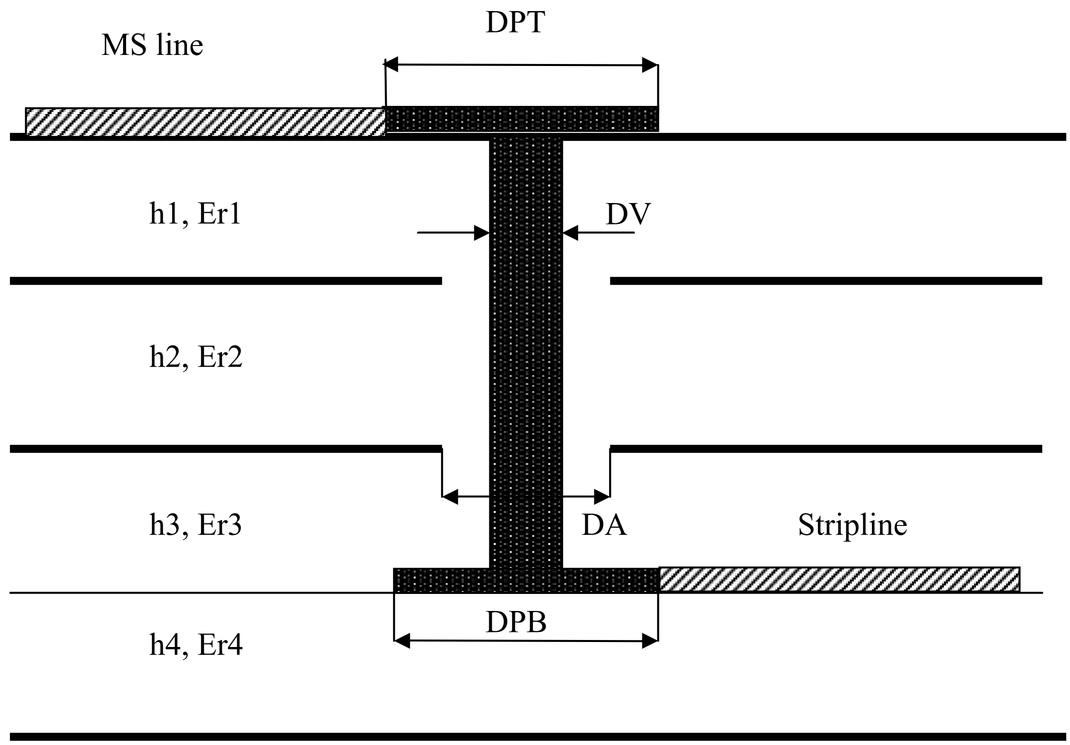


Figure 1. Via Cross View

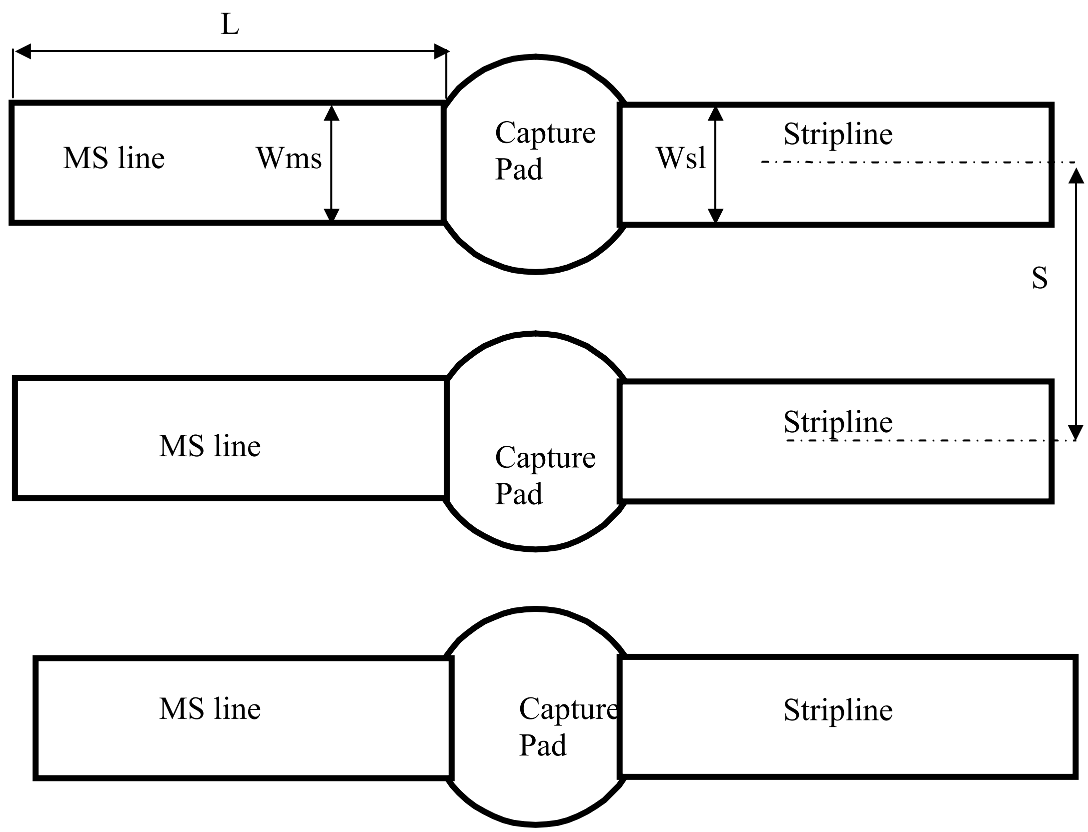


Figure 2. Via Top View

The following are material properties and dimension denotations (dimensions in mils). t1, t2 etc. are loss tangents.

Board stack	Via	Lines
Er1=4.5	DV=10	Wms=18
Er2=4.5	DA=20	Wsl=8
Er3=4.5	DPT=40	L=500
Er4=4.5	DPB=40	T=1
h1=10		S=80
h2=20		
h3=10		
h4=10		
t1=0.02		
t2=0.02		
t3=0.02		
t4=0.02		

Figure 3. Dimensions in mils

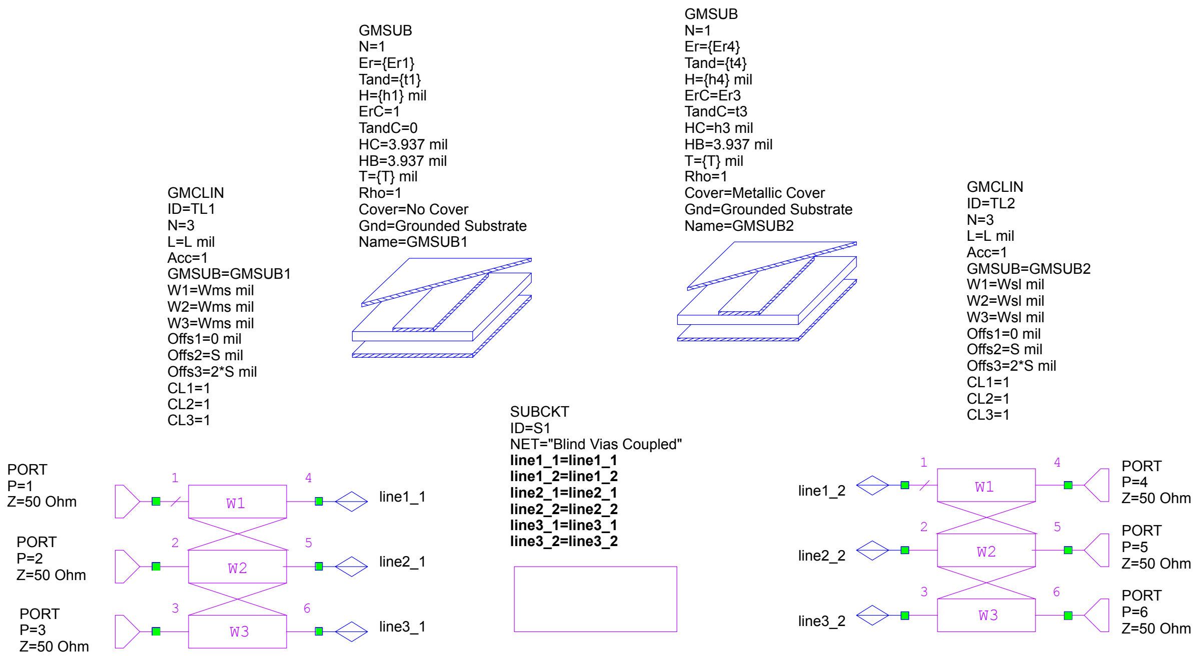


Figure 4. Schematic: Three coupled lines

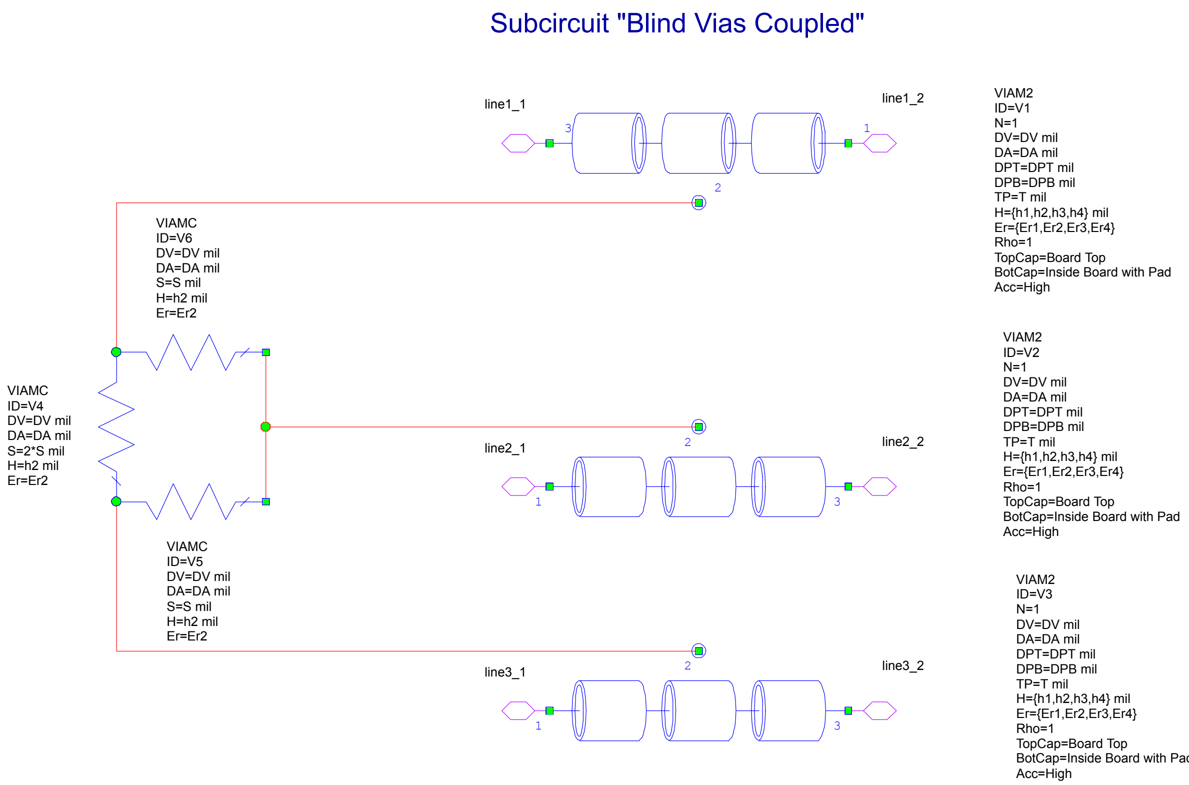


Figure 5. Subcircuit with coupled vias

Simulation results demonstrate losses for cases where vias are not accounted for, when vias are accounted for but coupling between vias is neglected, and when coupling between vias is factored in.

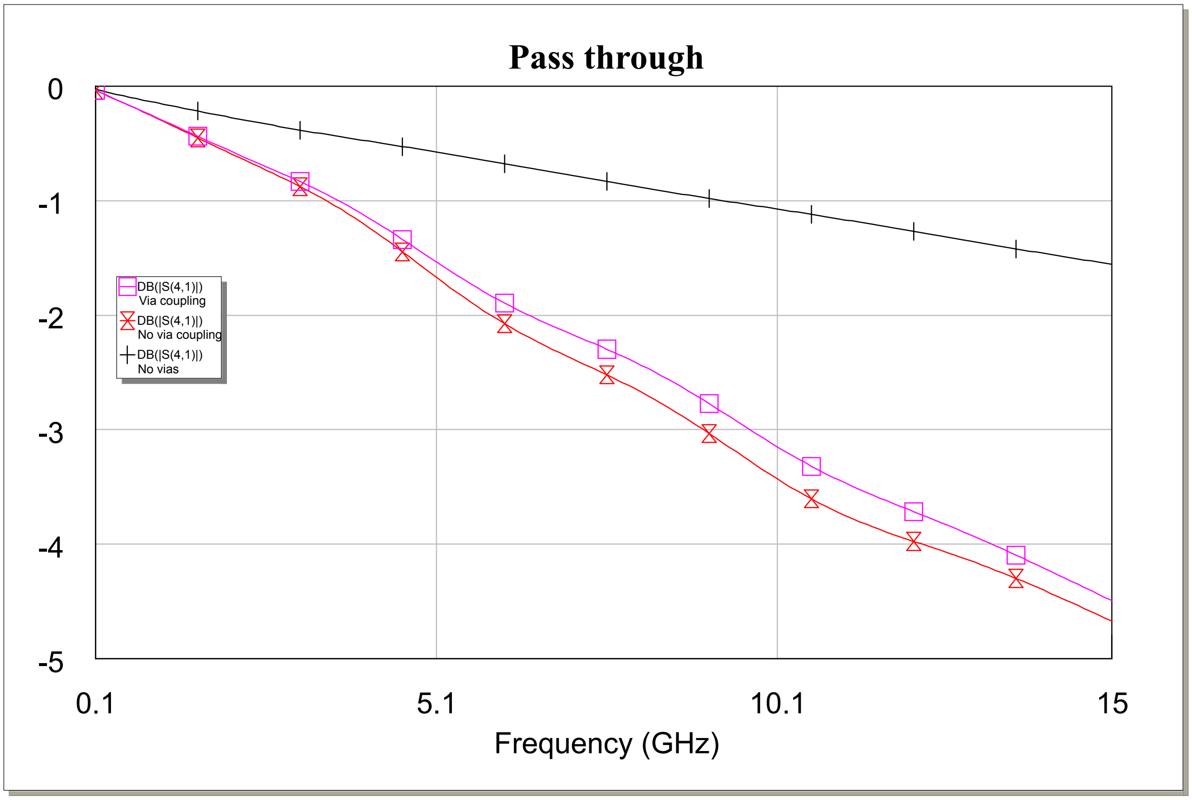


Figure 6. Simulation results: S41 in dB - Pass through loss

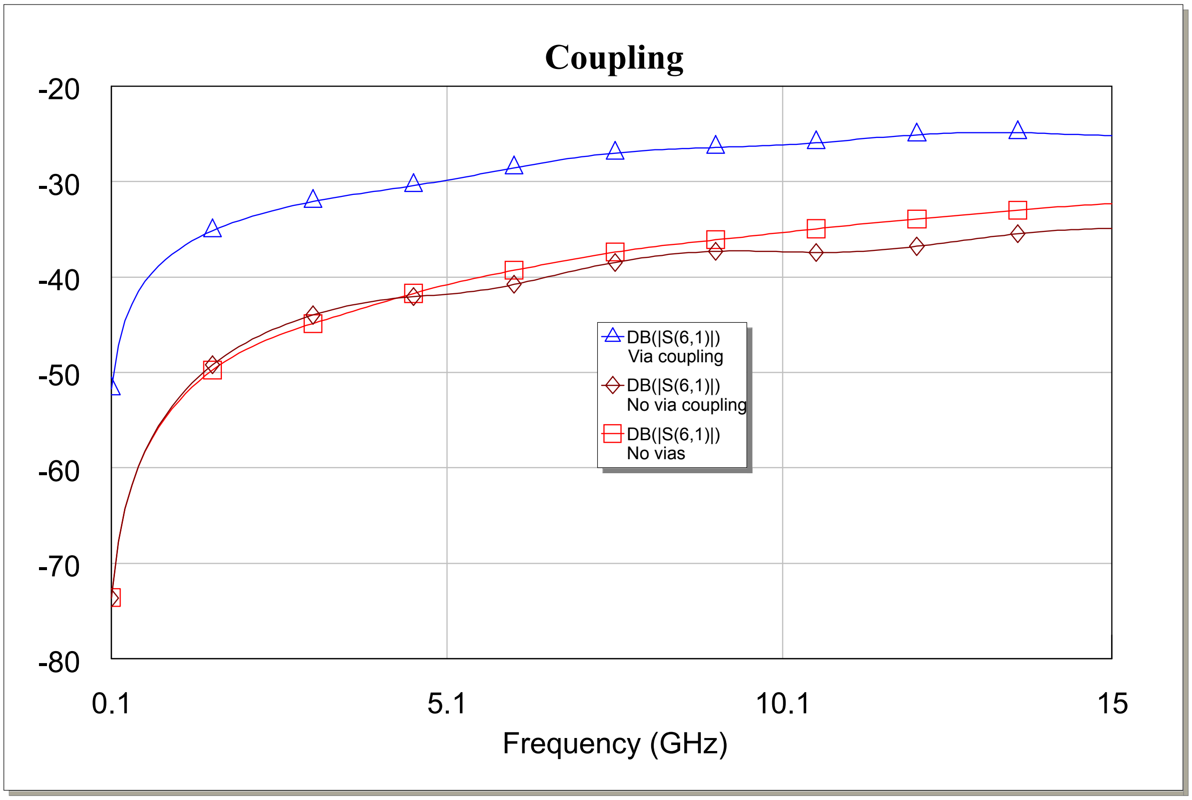


Figure 7. Simulation results: S61 in dB - Coupling loss

References

[1] Qizheng Gu, Y. Eric Yang, and M. Ali Tassoudji, "Modeling and Analysis of Vias in Multilayered Integrated Circuits," *IEEE Trans. on Microwave Theory and Tech.*, vol. 41, February 1993, pp. 206-214

[2] Qizheng Gu, M.Ali Tassoudji et. al., "Coupled Noise Analysis for Adjacent Vias in Multilayered Digital Circuits," *IEEE Trans. on Circ. and Syst.*, vol. 41, December 1994, pp. 796-804

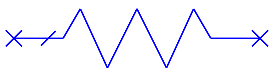
[3] B. Tomasic and A. Hessel, "Linear Array of Coaxially Fed Monopole Elements in a Parallel Plate Waveguide," *IEEE Trans. on Antennas and Prop.*, vol. 36, April 1988, pp. 449-462

[4] M. Goldfarb and R. Pucel, "Modeling Via Hole in Microstrip," *IEEE Microwave and Guided Wave Lett.*, vol. 1, June 1991, pp. 135-137

[5] David Meeker home page: http://femm.foster-miller.net/~dmeeker/ Follow this link for information: http://femm.foster-miller.net/Archives/readme.htm FEMM Manual: http://femm.foster-miller.net/Archives/doc/manual.pdf

Lump Coupling between Vias in Multilayer Board: VIAMC

Symbol



Summary

VIAMC represents a lump coupling between two similar vias in a multilayer printed circuit board. Coupling is evaluated as electromagnetic interference between sections of two vias confined within a user-selected dielectric layer from the board stackup. You can select any layer sandwiched between two ground planes because coupling evaluation is based on interaction of the single TMo mode propagating along a parallel-plate waveguide made by two adjacent ground planes. VIAMC can be used only in conjunction with the VIAM2 element and it must attach to dedicated coupling ports of VIAM2. See VIAM2 for additional details.

Topology

See VIAM2.

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	V1
DV	External diameter of via hole	Length	W[a]
DA	Diameter of antipad	Length	L[a]
H	Height of dielectric layer	Length	0.1 mm
Er	Relative dielectric constant of dielectric layer		1
*Acc	Switch "Default/High"		"Default"

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See "Default Values" for details.

** indicates a secondary parameter*

Parameter Details

H, Er. These parameters define the height and dielectric constant of the dielectric layer selected from the board stackup.

Acc. The default value of this parameter excludes the contribution of higher modes into the via model. Setting Acc=High adds the evaluation of series representing the contribution of higher modes and slightly increases simulation time. In most cases, Acc=Default provides sufficient accuracy but for very long vias crossing many dielectric layers Acc=High might be beneficial. Note that the value of Acc should match the value of Acc set for attached VIAM2 elements.

Parameter Restrictions and Recommendations

1. This model assumes that only dominant mode TMo propagates in the selected layer over all frequency sweep. The height and dielectric constant of the layer are checked for cutoff frequency of the higher mode propagating in the respective parallel-plate waveguide. Usually, this cutoff frequency has an order of hundred(s) gigahertz. Specifics of VIAMC implementation demand the cutoff frequency to be over 10 GHz. If for any reason (most possibly user error) cutoff frequency is below 10 GHz, VIAMC issues an error message and stops simulation.

Implementation Details

Implementation of this element constitutes a lumped approximation of electromagnetic coupling between two identical vias comprised of via segments confined to a single dielectric layer. This coupling is evaluated in absence of other vias (similar to the VIAMD element).

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Recommendations for Use

1. VIAMC must connect only to side (coupling) ports of VIAM2 elements.
2. To provide accurate evaluation of coupling all elements, VIAM2 elements connected with VIAMC elements must have identical DV, DA, and Acc parameters and values.
3. Building a network of coupled vias requires one VIAMC element per layer, per each pair of VIAM2 elements included in a coupled set of vias. For efficiency, a good practice may be to create separate subcircuits for each layer and connect VIAM2 elements to subcircuits using named connectors (NCONN) and named ports (PORT_NAME). For more information, see the example in VIAM2.
4. This model uses a disk cache, so if a project contains multiple instances of identical VIAMC only one instance simulates and saves results to the cache, and all identical (having the same parameter set) VIAMC elements simply fetch these results from the cache. Note that the cache keeps saved data, and any project on the same computer that contains VIAMC with the same set of parameters reuses the cached data.

References

[1] Qizheng Gu, Y. Eric Yang, and M. Ali Tassoudji, "Modeling and Analysis of Vias in Multilayered Integrated Circuits," *IEEE Trans. on Microwave Theory and Tech.*, vol. 41, February 1993, pp. 206-214

[2] Qizheng Gu, M.Ali Tassoudji et. al., "Coupled Noise Analysis for Adjacent Vias in Multilayered Digital Circuits," *IEEE Trans. on Circ. and Syst.*, vol. 41, December 1994, pp. 796-804

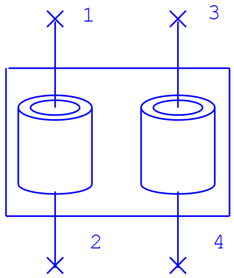
[3] B. Tomasic and A. Hessel, "Linear Array of Coaxially Fed Monopole Elements in a Parallel Plate Waveguide," *IEEE Trans. on Antennas and Prop.*, vol. 36, April 1988, pp. 449-462

[4] M. Goldfarb and R. Pucel, "Modeling Via Hole in Microstrip," *IEEE Microwave and Guided Wave Lett.*, vol. 1, June 1991, pp. 135-137

[5] David Meeker home page: http://femm.foster-miller.net/~dmeeker/ Follow this link for information: http://femm.foster-miller.net/Archives/readme.htm FEMM Manual: http://femm.foster-miller.net/Archives/doc/manual.pdf

Differential Vias in Multilayer Board: VIAMD

Symbol



Summary

VIAMD models a pair of identical coupled vias in a multilayer printed circuit board. This model assumes that all dielectric layers crossed by vias are separated by infinite perfect conducting grounded planes. Each ground plane has two antipads and vias pass through the center of each antipad. Capture pads are located at the via ends. The environment of each capture pad is user-controlled via model parameters. These parameters implement all typical via configurations: Thru, buried, and blind. You can take a pad out from one or both via ends and implement a series connection of several vias.

It is imperative for vias to cross at least one dielectric layer with ground planes with an antipad at the top and bottom.

Multilayer via analysis is based on the theory of long cylindrical antenna excited by a frill of magnetic current radiating in a parallel plate waveguide and on the theory of monopole radiating into a parallel plate waveguide. This model also uses quasi-static FEM analysis for accurate evaluation of capacitances of complex conducting configurations in a multilayered dielectric.

Topology

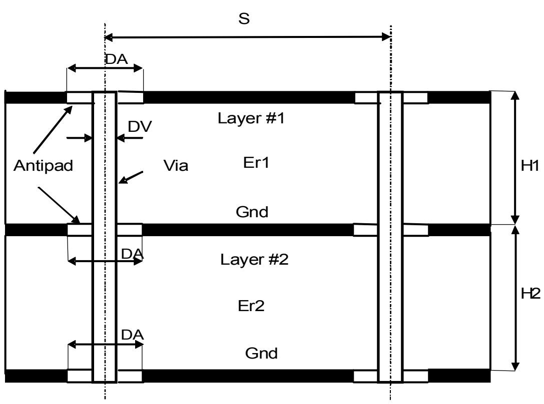


Figure 1. Basic stackup

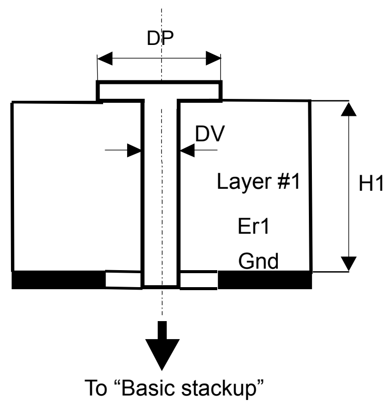


Figure 2. Option "Board Top"

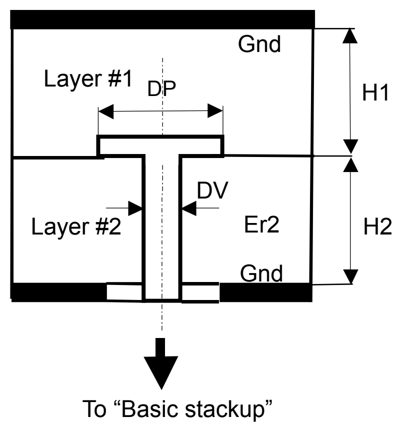


Figure 3. Option "Inside Board with Pad"

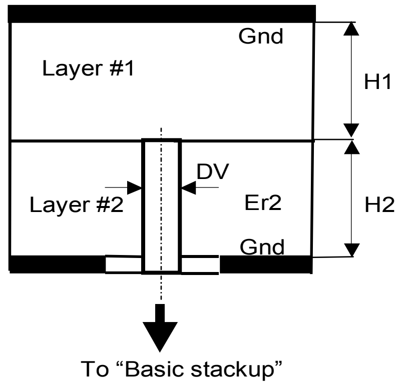


Figure 4. Option "Inside Board No Pad"

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	PKG1
N	Number of dielectric layers		1
DV	External diameter of via hole	Length	W[a]
DA	Diameter of antipad	Length	L[a]
DPT	Diameter of top capture pad	Length	L[a]
DPB	Diameter of bottom capture pad	Length	L[a]
S	Distance between via axes	Length	L[a]
TP	Thickness of capture pad	Length	T[a]
H	Heights of dielectric layers (vector)	Length	{0.1 mm}
Er	Relative dielectric constants of dielectric layers (vector)		{1}
Rho	Via metal bulk resistance relative to gold		1
TopCap	Switch "Board Top"/"Inside Board with Pad/Inside Board No Pad/Inside Antipad No Pad"	Length	"Board Top"
BotCap	Switch "Board Top"/"Inside Board with Pad/Inside Board No Pad/Inside Antipad No Pad"	Length	"Board Top"
*Acc	Switch "Default/High"		"Default"

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See "Default Values" for details.

** indicates a secondary parameter*

Parameter Details

N. This parameter defines the number of dielectric layers in a multilayered board separated by ground planes (see the "Basic stackup" figure in the "Topology" section). **NOTE:** Only layers from the "Basic stackup" are counted toward N. The actual number of layers VIAM uses depends on the options selected with the TopCap and BotCap parameters (see the following details for these parameters.) Note that VIAM **does not** allow a zero value of N (see "Parameter Restrictions and Recommendations".)

DPT, DPB. The diameters of the capture pads attached to the via top ends (Ports #1 and #3) and to the via bottom ends (Ports #2 and #4). Certain options of the TopCap and BotCap parameters imply that the respective pad is missing.

H, Er. These vector parameters define the height and dielectric constant of every dielectric layer in the board setup. Note that these vectors always include heights and dielectric constants of layers defined by the TopCap and BotCap parameters. TopCap adds values to the vector head while BotCaps adds values to the vector tail. You must provide correct values for each layer the model uses.

TopCap, BotCap. Each of these switching parameters has four user-selectable options. These options allow the *addition of via caps* of various configurations to each via end of the basic multilayer configuration presented in the "Basic stackup" figure in the "Topology" section.

TopCap defines the configuration of the via caps at Ports #1 and #3 and BotCap defines the configuration of the via caps at Ports #2 and #4.

All available configuration/options are presented in the "Topology" section in figures with corresponding captions. The "Board Top" option requires the addition of *one more value* to vectors H and Er in addition to the layer values needed for "Basic stackup"; the "Inside Board with Pad " and "Inside Board no Pad" options require the addition of *two more values* to vectors H and Er; the "Inside Antipad no Pad" option does not add any values to H and Er because it actually leaves the via end "as is" in "Basic stackup."

Note that TopCap and BotCap can be set independently to different options. The size of vectors H and Er are *defined by both selected options*.

Acc. The default value of the Acc parameter excludes the contribution of higher modes into the via model. Setting Acc=High adds the evaluation of series representing the contribution of high modes and slightly increases simulation time. In most cases Acc=Default provides sufficient accuracy, but for very long vias crossing many dielectric layers and for tightly coupled vias (small S), Acc=High might be beneficial.

Parameter Restrictions and Recommendations

1. The number of layers N in Basic stackup must be 1<=N<=30. Note that N=0 is not allowed for VIAMD.
2. This model assumes that only dominant mode TMo propagates in each layer over all frequency sweep. The heights of all layers are checked for normal operation below the cutoff frequency of the higher mode propagating in respective parallel-plate waveguide. If the frequency exceeds the cutoff threshold the model generates an error and reports the highest sweep frequency allowed for the specified separation between ground planes.

Implementation Details

The via is modeled as a set of segments contained within ground-separated dielectric layers. Each segment is modeled as cylindrical antenna/monopole radiating between perfectly conducting parallel plates. Modeling techniques used are based on publications [1]-[4] [6–88]. Via cap capacitance is treated by means of a quasi-static FEM technique based on [5] [6–88]. Port reference planes are located at the centers of capture pads. Ports #1 and #3 are associated with the top end of the via and ports #2 and #4 are associated with the bottom end of the via.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Recommendations for Use

See examples of various via implementations in the VIAM model documentation (note that examples for N=0 are not valid for VIAMD).

This model uses a disk cache, so if a project contains multiple instances of identical VIAMD only one instance simulates and saves results to the cache, and all identical (having the same parameter set) VIAMD simply fetch these results from cache. Note that the cache keeps saved data and any project on the same computer that contains VIAMD with the same set of parameters reuses the cached data.

References

[1] Qizheng Gu, Y. Eric Yang, and M.Ali Tassoudji, "Modeling and Analysis of Vias in Multilayered Integrated Circuits," *IEEE Trans. on Microwave Theory and Tech.*, vol. 41, February 1993, pp. 206-214

[2] Qizheng Gu, M.Ali Tassoudji et al, "Coupled Noise Analysis for Adjacent Vias in Multilayered Digital Circuits," *IEEE Trans. on Circ. and Syst.*, vol. 41, December 1994, pp. 796-804

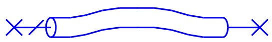
[3] B.Tomasic and A. Hessel, "Linear Array of Coaxially Fed Monopole Elements in a Parallel Plate Waveguide," *IEEE Trans. on Antennas and Prop.*, vol. 36, April 1988, pp. 449-462

[4] M. Goldfarb and R. Pucel, "Modeling Via Hole in Microstrip," *IEEE Microwave and Guided Wave Lett.*, vol. 1, June 1991, pp. 135-137

[5] David Meeker home page: http://femm.foster-miller.net/~dmeeker/ Follow this link for information: http://femm.foster-miller.net/Archives/readme.htm FEMM Manual: http://femm.foster-miller.net/Archives/doc/manual.pdf

Round Wire Inductor (Closed Form): WIRE

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	IC1
L	Wire length	Length	L[a]
Dia	Wire diameter	Length	W[a]
Rho	Metal resistivity relative to gold		Rho[a]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory.

Implementation Details

This component models a length of conductor having a round section. The model is implemented as a lumped element series inductor and resistor whose values are derived from the dimensions of the wire.

Layout

This element uses a special layout cell for a short circuit. The layout cell allows the elements connected on either side of the element to look through this element. For example, when a MLIN is hooked to a TFCM (capacitor) model and the project has bridge code configured, the line draws the proper interconnect between the line and the cap. If this element is placed between the line and the cap, the layout still draws the same.

You typically do not assign artwork cells to these items.

Recommendations for Use

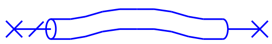
Discontinuity models function most accurately when attached to lines that match their corresponding edges. Directly connecting discontinuity models to one another reduces their accuracy.

References

[1] Wadell, Brian C. Transmission Line Design Handbook, Artech House 1991.

Round Bond Wire Over Substrate: WIRE1

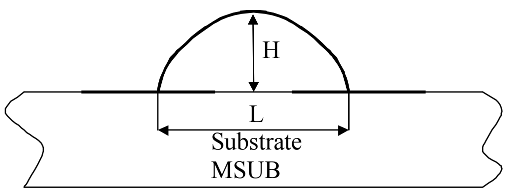
Symbol



Summary

WIRE1 models a round bond wire connecting two conducting pads or microstrip lines. The model takes into account substrate proximity and the wire's arc shape

Structure



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	TL1
L	Wire length (distance between pads)	Length	L[a]
D	Wire diameter	Length	W[a]
H	Wire midpoint elevation over the substrate	Length	W[a]
Rho	Metal resistivity relative to gold		1
MSUB	Substrate definition	Text	MSUB1[b]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See the Microwave Office Layout Guide for details.

[b]Modify only if schematic contains multiple substrates. See the "Using Elements With Model Blocks" for details.

Parameter Details

H. The wire has an arc shape and H is the height of its summit point.

L. L is the shortest straight distance between points where wire is affixed to the pads

Implementation Details

This model is based on a quasi-static electromagnetic analysis described in [1]. It accounts for losses in metal and in substrate dielectric. Dispersion is partly included.

To decrease the calculation time for schematics that contain several of these elements, the electromagnetic analyses are cached; (saved and reused in later analyses). Thus, the first analysis of a circuit containing multiple bond wires takes longer than subsequent analyses.

NOTE: The quasi-static electromagnetic models use lengthy numerical algorithms which may lead to a noticeable increase in simulation time for schematics that employ many such models.

If wire elevation is very small in comparison with the substrate thickness, simulation time may also increase noticeably.

Layout

This element uses a special layout cell for a short circuit. The layout cell allows the elements connected on either side of the element to look through this element. For example, when a MLIN is hooked to a TFCM (capacitor) model and the project has bridge code configured, the line draws the proper interconnect between the line and the cap. If this element is placed between the line and the cap, the layout still draws the same.

You typically do not assign artwork cells to these items.

Recommendations for Use

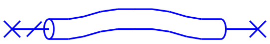
For a wide range of substrate and wire parameters, the closed-form model WIRE2 may provide accuracy approaching that of WIRE1. However, the analysis of WIRE2 is based on an assumption that H is large relative to the substrate thickness, so WIRE2 issues a warning if the wire is placed too close to the substrate. WIRE1 may be useful for validation of designs using WIRE2.

References

[1] M.B. Bazdar, A.R. Djordjevic, R.F. Harrington, and T.K. Sarkar, "Evaluation of quasi-static matrix parameters for multiconductor transmission lines using Galerkin's method," IEEE Trans. Microwave Theory Tech., vol. MTT-42, July 1994, pp. 1223-1228.

Round Bond Wire Over Substrate: WIRE2

Symbol



Summary

WIRE2 models a round bond wire connecting two conducting pads or microstrip lines. Model accounts for substrate proximity and is valid for wire reasonably elevated over the substrate.

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	IC1
L	Wire length (distance between pads)	Length	L[a]
D	Wire diameter	Length	W
H	Wire midpoint elevation over the substrate	Length	W
Rho	Metal resistivity relative to Gold		1
MSUB	Substrate definition	Text	MSUB1[b]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See the Microwave Office Layout Guide for details.

[b]Modify only if schematic contains multiple substrates. See "Using Elements With Model Blocks" for details.

Parameter Restrictions and Recommendations

The wire elevation H should exceed 0.3 times the substrate's thickness. The model issues a warning if this limitation is violated.

Layout

This element uses a special layout cell for a short circuit. The layout cell allows the elements connected on either side of the element to look through this element. For example, when a MLIN is hooked to a TFCM (capacitor) model and the project has bridge code configured, the line draws the proper interconnect between the line and the cap. If this element is placed between the line and the cap, the layout still draws the same.

You typically do not assign artwork cells to these items.

Recommendations for Use

According to reference [1], for best accuracy the wire elevation H should be large. However, for a wide range of substrate and wire parameters WIRE2 provides good accuracy in comparison to the more complicated and more time consuming model WIRE1. If you are concerned about the accuracy when small values of H are used, you should use WIRE1 for validation.

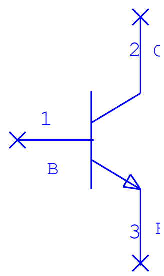
References

[1] Kuester, E.F, Chang, D.C., "Propagating Modes Along a Thin Wire Located Above a Grounded Dielectric Slab", IEEE Trans. on Microwave Theory and Tech., Vol MTT-25 No.12, December 1977, pp. 1065-1069

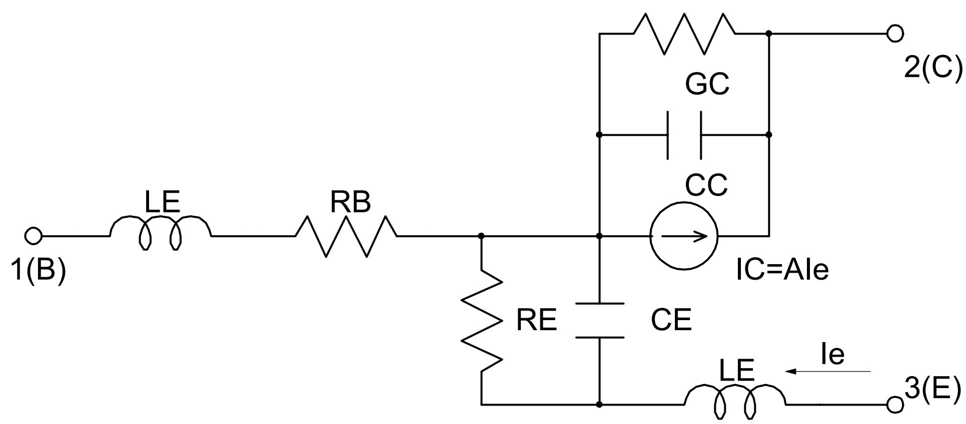
Linear Devices

Bipolar Transistor (Closed Form): BIP

Symbol



Equivalent Circuit



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	B1
A	Magnitude of DC current gain (alpha)		0.8
T	Current gain time delay	Time	0 ns
F	-3dB frequency for current gain	Frequency	0 GHz
CC	Collector capacitance	Capacitance	0 pF
GC	Collector conductance	Conductance	0 S
RB	Base resistance	Resistance	1 ohm
LB	Base inductance	Inductance	0 nH
CE	Emitter capacitance	Capacitance	0 pF
RE	Emitter resistance	Resistance	1 ohm
LE	Emitter inductance	Inductance	0 nH

Implementation Details

Implements a bipolar junction transistor with DC gain specified by A.

The frequency-dependent gain A(f) is given by:

$$A(f) = \begin{cases} A\dfrac{e^{-j2\pi \mathrm{ft}}}{1 + j\left(\frac{f}{F}\right)} & F > 0 \\ \mathrm{Ae}^{-j2\pi \mathrm{ft}} & F = 0 \end{cases}$$

NOTES:

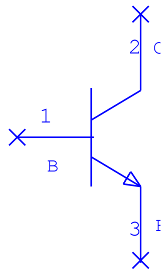
1. A must be in the range $0 < A < 1$
2. F must be greater than or equal to 0
3. Assumed to be noiseless.

Layout

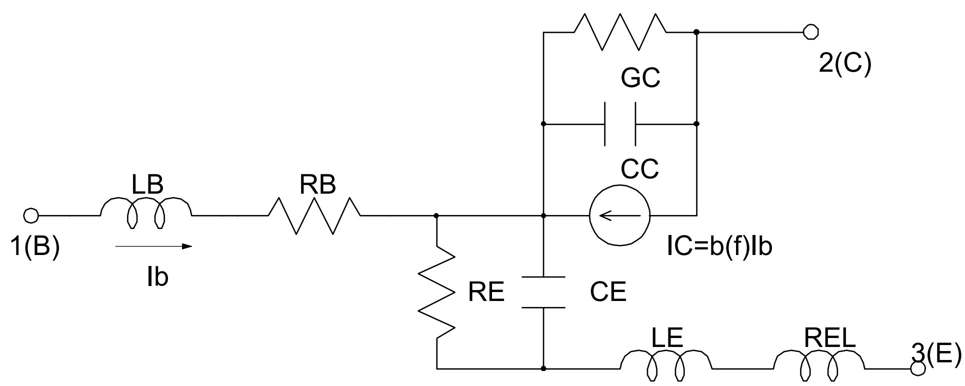
This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Bipolar Transistor Beta Controlled (Closed Form): BIPB

Symbol



Equivalent Circuit



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	B1
B	Magnitude of DC current gain (beta)		100
A	Current gain phase offset	Angle	0 Deg
T	Current gain time delay	Time	0 ns
CC	Collector capacitance	Capacitance	0 pF
GC	Collector conductance	Conductance	0 S
RB	Base resistance	Resistance	1 ohm
LB	Base inductance	Inductance	0 nH
CE	Emitter capacitance	Capacitance	0 pF
RE	Emitter resistance	Resistance	1 ohm
LE	Emitter inductance	Inductance	0 nH
REL	Emitter load resistance	Resistance	1 ohm

Implementation Details

Implements a bipolar junction transistor with DC gain specified by B.

The frequency-dependent current gain β(f) is given by

$$\beta(f) = \mathrm{B}e^{-j(2\pi \mathrm{fT} - A)}$$

NOTES:

1. B must be greater than zero.

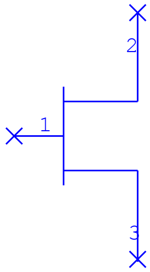
2. Assumed to be noiseless.

Layout

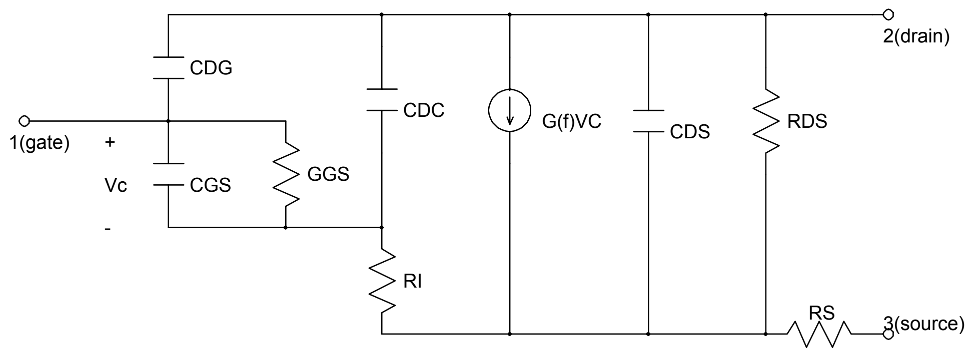
This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Field Effect Transistor (Closed Form): FET

Symbol



Equivalent Circuit



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	F1
G	Magnitude of transconductance at DC (beta)	Conductance	0.1 S
T	Time delay of the transconductance	Time	0 ns
F	Roll off frequency of the transconductance	Frequency	0 GHz
CGS	Gate to source capacitance	Capacitance	0 pF
GGS	Gate to source conductance	Conductance	1 S
RI	Channel resistance	Resistance	1 ohm
CDG	Drain to gate capacitance	Capacitance	0 pF
CDC	Dipole layer capacitance	Capacitance	0 pF
CDS	Drain to source capacitance	Capacitance	0 pF
RDS	Drain to source resistance	Resistance	100 ohm
RS	Source resistance	Resistance	0 ohm

Implementation Details

Implements a FET transistor with DC transconductance specified by G.

The frequency-dependent transconductance G(f) is given by

$$G(f) = \begin{cases} G\dfrac{e^{-j2\pi\mathrm{ft}}}{1 + j\left(\frac{f}{F}\right)} & F > 0 \\ \mathrm{Ge}^{-j2\pi\mathrm{ft}} & F = 0 \end{cases}$$

Layout

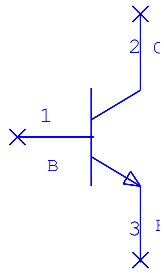
This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Restrictions

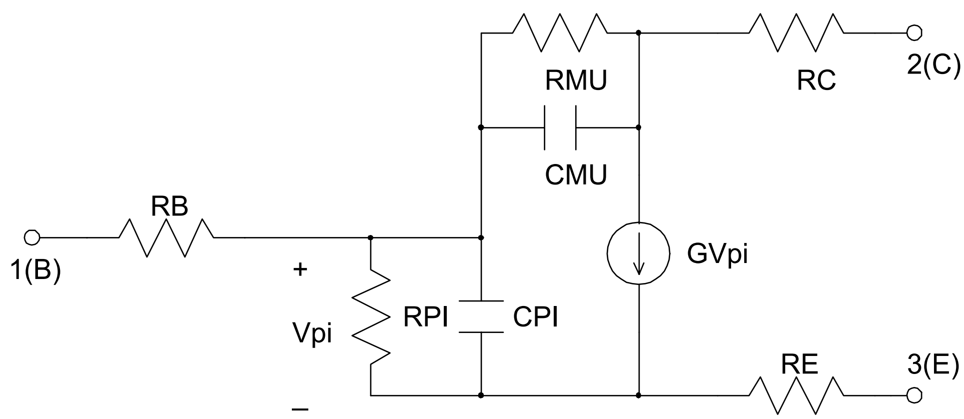
1. Assumed to be noiseless.

Hybrid-Pi Bipolar Transistor (Closed Form): HYBPI

Symbol



Topology



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	B1
G	Magnitude of transconductance	Conductance	0.1 S
T	Transit time	Time	0 ns
CPI	Base to emitter capacitance (PI)	Capacitance	0 pF
RPI	Base to emitter resistance (PI)	Resistance	10 ohm
CMU	Base to collector capacitance (MU)	Capacitance	0 pF
RMU	Base to collector resistance (MU)	Resistance	1000 ohm
RB	Base resistance	Resistance	1 ohm
RC	Collector resistance	Resistance	1 ohm
RE	Emitter resistance	Resistance	1 ohm

Layout

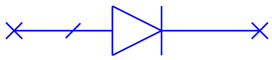
This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

Restrictions

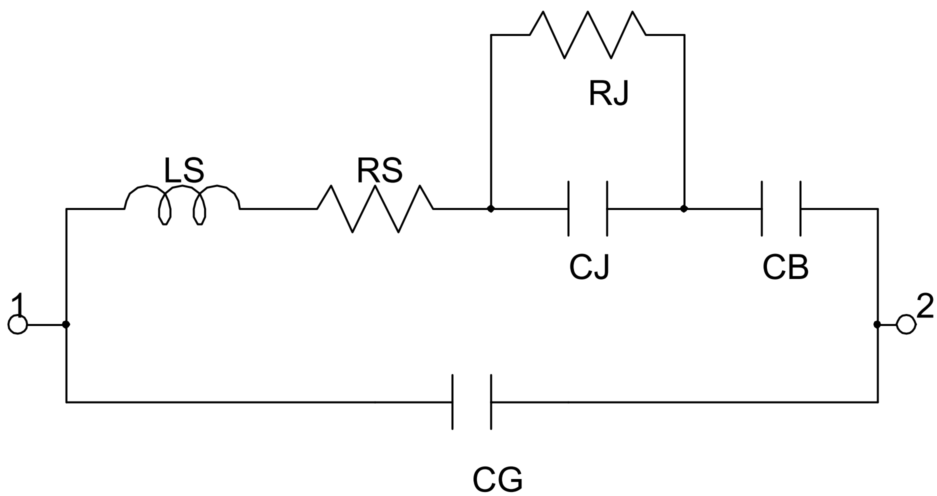
1. Assumed to be noiseless.
2. RPI must be greater than zero.
3. RMU must be greater than zero.

Pin Diode Chip Model (Closed Form): PIN

Symbol



Topology



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	D1
CJ	Junction capacitance	Capacitance	0 pF
RJ	Junction resistance	Resistance	1000 ohm
RS	Diode series resistance	Resistance	1 ohm
LS	Bond wire inductance	Inductance	0 nH
CB	By-pass capacitance	Capacitance	0 pF
CG	External capacitance	Capacitance	0 pF

Implementation Details

Implements a chip diode.

Layout

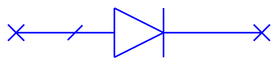
This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Restrictions

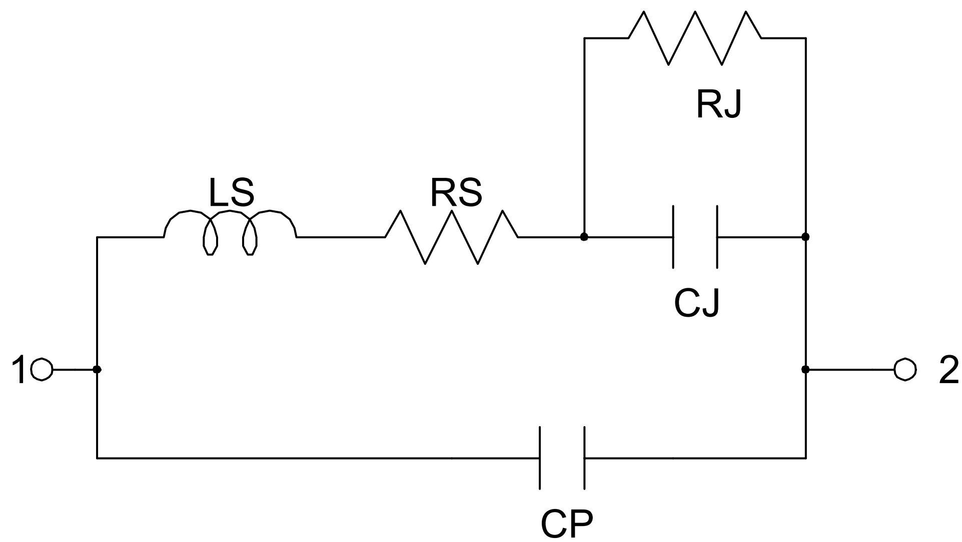
1. Assumed to be noiseless.

Pin Diode Packaged Model (Closed Form): PIN2

Symbol



Topology



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	D1
CJ	Junction capacitance	Capacitance	0 pF
RJ	Junction resistance	Resistance	1000 ohm
RS	Diode series resistance	Resistance	1 ohm
LS	Bond wire inductance	Inductance	0 nH
CP	Package capacitance	Capacitance	0 pF

Implementation Details

Implements a packaged PIN diode.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

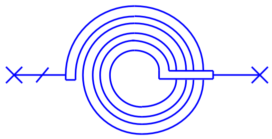
Restrictions

1. Assumed to be noiseless.

Lumped Element

Flat Circular Spiral Wire Inductor: AIRIND

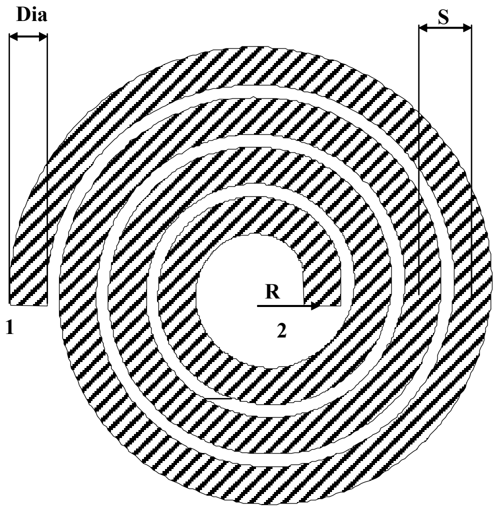
Symbol

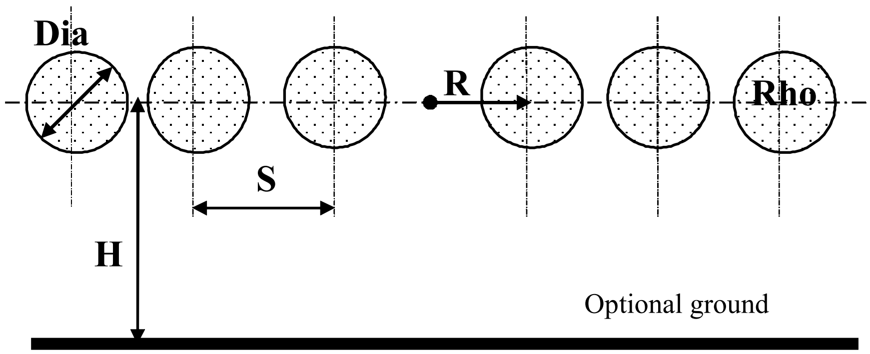


Summary

This circuit component models a flat round spiral inductor made of the wire of a cylindrical cross-section. This model allows you to either place the inductor in a free space or to place it above a perfect electrical ground. The modeling approach is based on an evaluation of self and mutual inductances, capacitances and resistances between all spiral turns.

Topology





Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	MSP1
NT	Number of turns (>=1)		3

Name	Description	Unit Type	Default
Dia	Wire diameter	Length	25 um
S	Spacing between turns (centers of wires)	Length	100 um
R	Inside radius of innermost turn (measured from center of spiral to the wire centerline)	Length	15 um
H	Inductor elevation above optional ground	Length	15 um
Rho	Wire metal bulk resistivity relative to gold		1
Gnd	Switch "No ground/Ground"		No ground
*Nseg	Number of linear segment per turn		6

** indicates a secondary parameter*

Parameter Details

NT. The number of spiral turns forming the inductor. NT is equal to the number of times the wire intersects the horizontal center line of the inductor (except for the innermost crossing at port 2 - see "Topology") plus the fraction of the turn from the last intersection to port 1. For example, "Topology" presents an inductor with NT=3.5.

R. The radius of the innermost turn (measured from the spiral center to the wire centerline).

H. The inductor elevation above optional ground, measured from the wire centerline. If Gnd is set to "No ground" the model ignores this parameter.

Gnd. Toggles between "No ground" and "Ground." "Ground" makes the model use parameter H.

Nseg. The number of linear wire segments a model uses to approximate a single spiral turn (see "Implementation Details"). Nseg must be greater than or equal to 6.

Parameter Restrictions and Recommendations

1. NT should greater than or equal to 1.
2. Models check that Nseg $\geq$6.
3. The inductor size (diameter D_0 measured along the inductor horizontal centerline) may be evaluated as D_0 = 2R+2S(NT-1)+S+Dia

Implementation Details

Each spiral turn is broken into Nseg linear wire segments and the model evaluates self and mutual RLC parameters of all segments (including the image wire segments if Gnd is set to "Ground").This model controls the length of each segment and issues an error message if the wire diameter Dia exceeds the length of any segment . This model accounts for a skin effect and partly for radiation loss.

Note that port 1 is assigned to external end point of a spiral and Port 2 is assigned to internal end point of a spiral.

Recommendations for Use

AIRIND predicts the frequency behavior of a wire inductor below and in the vicinity of the first self-resonance frequency.

User discretion is advised if model issues a warning about segment length being too small. AIRIND assumes that each segment is represented by infinitely thin filament which means Lseg >>. Thus, very short (relative to wire diameter) segments may cause additional errors. Model issues warning message if Lseg<6*Dia. This message contains segment number. All segments with **numbers less than violating** are also in violation. Usually those are segments that belong to a few internal turns. Thus user gets an approximate estimation of percentage of segments with reduced accuracy.

Table of conversion AWG (American Wire Gauge) to millimeters (for those who prefer wire gauge for denoting wire diameter).

AWG	Wire diameter (mm)	AWG	Wire diameter (mm)	AWG	Wire diameter (mm)
12	2.053	25	0.4547	38	0.1007
13	1.828	26	0.4049	39	0.08969
14	1.628	27	0.3606	40	0.07987
15	1.450	28	0.3211	41	0.07112
16	1.291	29	0.2859	42	0.06334
17	1.150	30	0.2546	43	0.05641
18	1.024	31	0.2268	44	0.05023
19	0.9116	32	0.2019	45	0.04473
20	0.8118	33	0.1798	46	0.03983
21	0.7229	34	0.1601	47	0.03547
22	0.6438	35	0.1426	48	0.03160
23	0.5733	36	0.1270	49	0.02813
24	0.5106	37	0.1131	50	0.02505

Layout

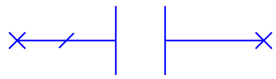
The layout for this cell has hard-coded model layers. When you first use this layout cell, a layer named "AirInductor" is added to your drawing layer and model layer list (if they are not already there). Using the model layer mapping, you can assign these layers to draw on any drawing layer. The 3D layout displays the shapes as round wires.

This model has a layout-only parameter that controls how the layout cell draws. You access the parameters by selecting the item in the layout, right-clicking and choosing **Shape Properties** to display the Cell Options dialog box, then clicking the **Parameters** tab.

Name	Value	Units	Description
NSeg	32	Integer	Number of linear segments approximating one spiral turn.

Capacitor (Closed Form): CAP

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Name	Text	C1
C	Capacitance	Capacitance	0 pF

Implementation Details

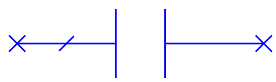
Implements an ideal capacitor.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

Capacitor with Frequency-dependent Capacitance, Parasitic and Loss: CAPQP

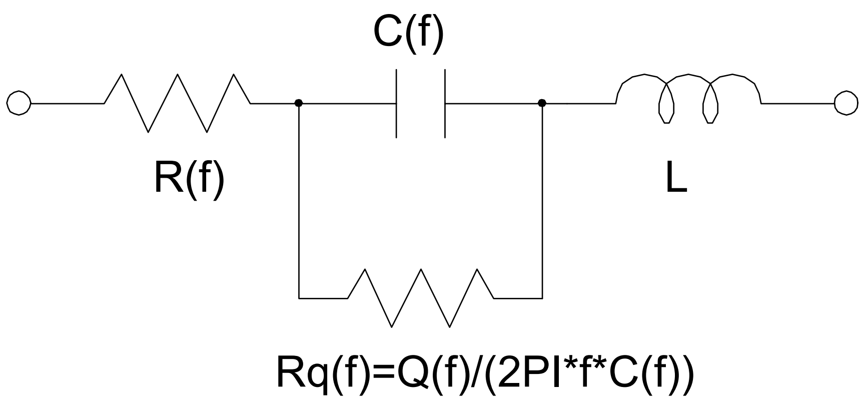
Symbol



Summary

CAPQP models a capacitor with user-specified frequency dependence of capacitance and loss. Frequency dependencies are specified as lookup tables (vectors) in model parameters. This model uses interpolation to obtain parameter values at each project evaluation frequency.

Equivalent Circuit



Parameters

Name	Description	Unit Type	Default
ID	Name	Text	CAP#
F	Vector of frequencies at which C, Q, and R are specified.	Frequency	{1}
C	Capacitance (vector)	Capacitance	Li[a]
Q	Quality factor (vector)	Scalar	{1000}
L	Inductance	Parasitic inductance	C[a]
R	Series resistance (vector)	Resistance	{0}

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See the Microwave Office Layout Guide for details.

Parameter Details

F. Vector of frequencies at which L, Q, and R parameters are specified. Frequencies must be sequential and specified in ascending order.

C. Vector of series capacitance C(f) (see the "Equivalent Circuit" section) specified in capacitance project units. You must specify each vector entry at the corresponding frequency entry from frequency vector F.

Q. Vector of quality factor; specifies series resistance Rq (f) (see the "Equivalent Circuit" section). You must specify each vector entry at the corresponding frequency entry from frequency vector F.

L. Parasitic inductance (see the "Equivalent Circuit" section).

R. Vector of shunt resistance R(f) (see the "Equivalent Circuit" section) specified in resistance project units. You must specify each vector entry at the corresponding frequency entry from frequency vector F.

Parameter Restrictions and Recommendations

1. The size of vector parameters C, Q, and R must be equal to the size of frequency vector F.
2. If the project evaluation frequency is out of range of frequencies in F, then C, Q, and R parameters are extrapolated as constant values equal to the first/last entries of corresponding vectors. No warning is issued.
3. You can specify the vector in three ways: First, by entering it as a right side value of model parameter, for example R={100,102,110,113,120}; second, by specifying the vector elsewhere in the equation; and third, by specifying the vector in a column or row of a text file. The third way provides a convenient and flexible method of specifying the C, Q, and R parameters at a single location. For example, you can create a *capqp.txt* file containing space separated columns of C, Q, and R. The first column must represent frequency in project units (note that changing the project default frequency units demands manual scaling of frequencies in this file). Import or link this file to your project and name it, for example, CAPQP_1. Now you can specify, for example, parameter R as R = Col(datafile("CAPQP_1"),4) so that the values of vector R are copied to the model from column 4 of *capqp.txt* imported under the name capqp_1. If you prefer to deploy your data row-wise use R = Row(datafile("CAPQP_1"),2).
4. If your project uses text file input to feed data to this model be aware what frequency, resistance, inductance or conductance units this file implies. Your project default units may differ from those in your data file. If this happens, you must scale input values, multiplying the call of function Col or Row by a scaling coefficient. For example, if your project uses capacitance in picofarads and the data file contains data in Farads you may get capacitance data from column 2 of data file CAPQP_1 such as: C= 1e+12*Col(datafile("CAPQP_1"),2).

Implementation Details

This model is implemented as a series connection of lossy capacitor and frequency-dependent resistor. Parasitic inductance is in series with a lossy capacitor.

Model implementation is based on linear interpolation of C, Q, and R parameters at each project evaluation frequency. Interpolation uses user-supplied lookup tables via parameters. If the project evaluation frequency is out of range of frequencies in F, then C, Q, and R parameters are extrapolated as constant values equal to the first/last entries of corresponding vectors.

Layout

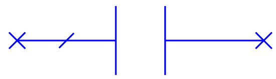
This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Recommendations for Use

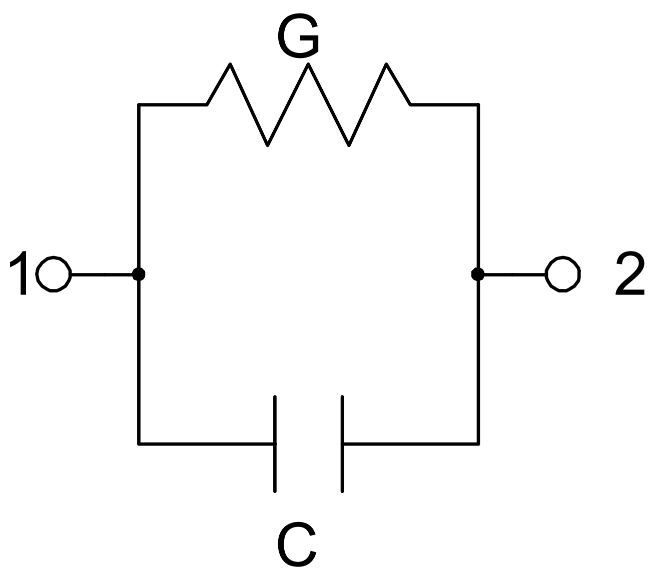
Time transient analysis is sensitive to the number of frequency points. Also, time domain measurements may be inaccurate due to non-causal behavior of the model at specified parameter values.

Capacitor with Q (Closed Form): CAPQ

Symbol



Equivalent Circuit



Parameters

Name	Description	Unit Type	Default
ID	Name	Text	C1
C	Capacitance	Capacitance	1 pF
Q	Q at FQ		0
FQ	Freq at which Q is determined	Frequency	0 GHz
ALPH	Scaling exponent for Q		1
DCMod	DC modeling	Vector text (pull-down)	Lossless

Parameter Details

DCMod. Specifies the DC behavior of the model. **Lossless** indicates that the model is modeled as an ideal capacitor at DC. **Lossy** indicates that loss is taken into account at DC.

Implementation Details

Implements a capacitor with frequency-dependent Q.

$$Q(f) = Q\left(\frac{f}{\mathrm{FQ}}\right)^{\mathrm{ALPH}}$$

The admittance of the capacitor is given by:

$$Y = G + \mathrm{jB} = 2\pi \mathrm{fC}\left(\frac{1}{Q(f)} + j\right)$$

$$\left(Q_C = \frac{B}{G} = \frac{2\pi \mathrm{fC}}{G} \quad \text{or} \quad G = \frac{2\pi \mathrm{fC}}{Q_C}\right)$$

In these equations, f is the simulation frequency.

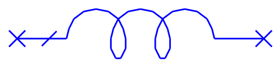
NOTE: FQ must be greater than or equal to zero.

Layout

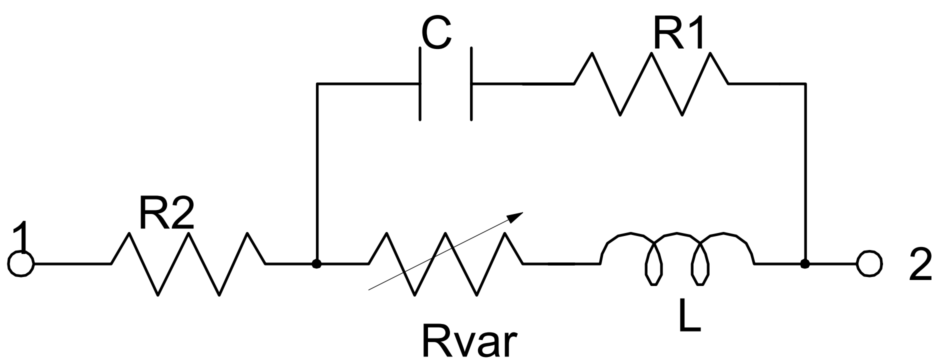
This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

Coilcraft Inductor (Closed Form): CCIND

Symbol



Topology



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	L1
R1	Resistance in series with C	Resistance	1 ohm
R2	Resistance in series with inductance	Resistance	1 ohm
C	Resonating capacitance	Capacitance	1 pF
L	Inductance	Inductance	1 nH
K	Frequency-dependent resistance coefficient		1e-6

Implementation Details

Modeled as an equivalent circuit (above) where

$$R_{\mathrm{var}} = K \cdot \sqrt{f}$$

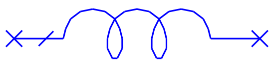
and f is the simulation frequency in Hz. See the Coilcraft website for a detailed model description.

Layout

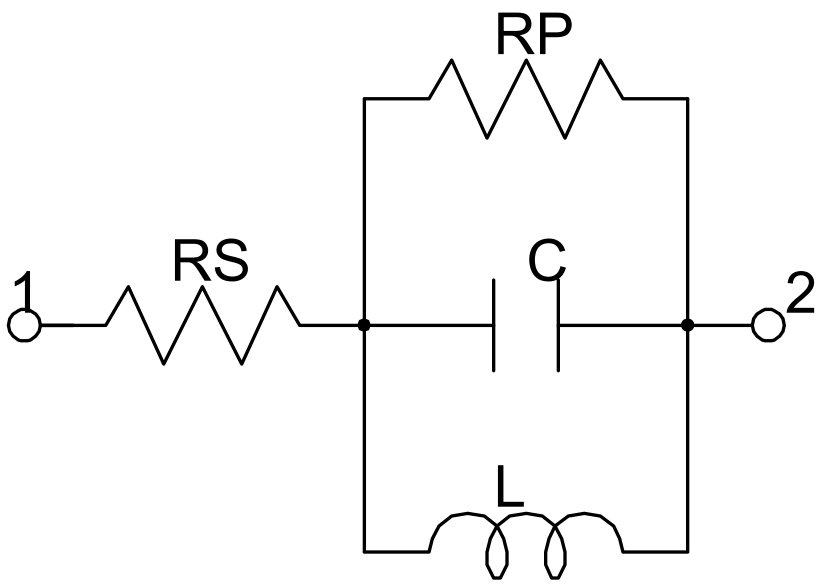
This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Coilcraft Power Chip Inductor (Closed Form): CCIND2

Symbol



Topology



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	L1
RS	Resistance in series with L and C	Resistance	1 ohm
RP	Resistance in parallel with L and C	Resistance	1 ohm
C	Resonating capacitance	Capacitance	1 pF
L	Inductance	Inductance	1 nH

Implementation Details

This element is modeled as an equivalent circuit (shown above).

See the Coilcraft website for a detailed model description.

Layout

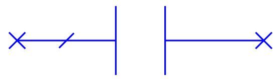
This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Restrictions

CCIND2 was based on measured data taken in the frequency range from 100 kHz to the self-resonant frequency of the inductor. Simulation results outside of this frequency range should be used with caution. Please contact Coilcraft for additional information.

Chip Capacitor - 3D EM Cell (Closed Form): CHCAP_EM

Symbol



Summary

This element should only be used in 3D EM documents. The intent of this model is to create a 3D parameterized cell for a chip capacitor for 3D EM analysis. See “Using 3D EM Elements ” for details on using 3D pCells with Analyst.

Parameters

Name	Description	Unit Type	Default
ID	Name	Text	IC1
L	Plate length	Length	40 um
W	Plate width	Length	40 um
D	Thickness of filling dielectric	Length	2 um
T	Plate thickness	Length	2 um
Rho	Metal resistivity relative to gold		1
Er	Relative dielectric constant of filling dielectric		1
Tand	Loss tangent of filling dielectric		0
*DIE_NAME	Filling dielectric name for 3D EM cell		""
*Orientation	Horizontal plates/Vertical plates		"Horizontal plates"

Layout

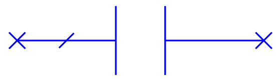
This element does not have a layout that can be used in schematic layout. When you first use this layout cell, a layer named "CHCAP_EM" is added to your drawing layer and model layer list (if they are not already there). Using the model layer mapping, you can assign these layers to draw on any drawing layer.

3D EM Layout

This element has a layout cell specifically for Analyst 3D EM layouts. See “Using 3D EM Elements ” for details on using 3D pCells with Analyst. The parameters of the model fully define a parallel plate capacitor.

Multiplate Chip Capacitor - 3D EM Cell: CHCAP2_EM

Symbol

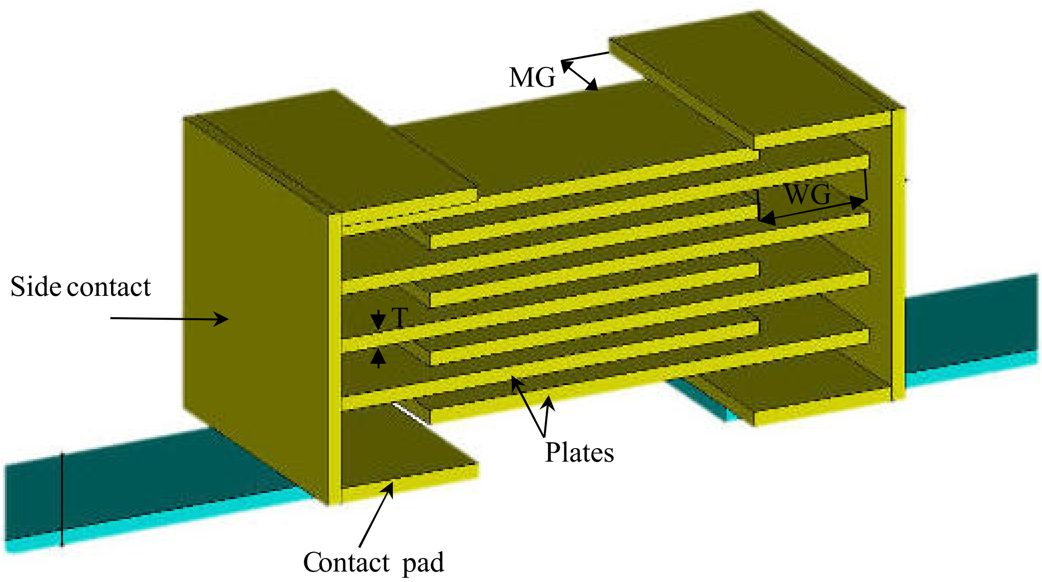


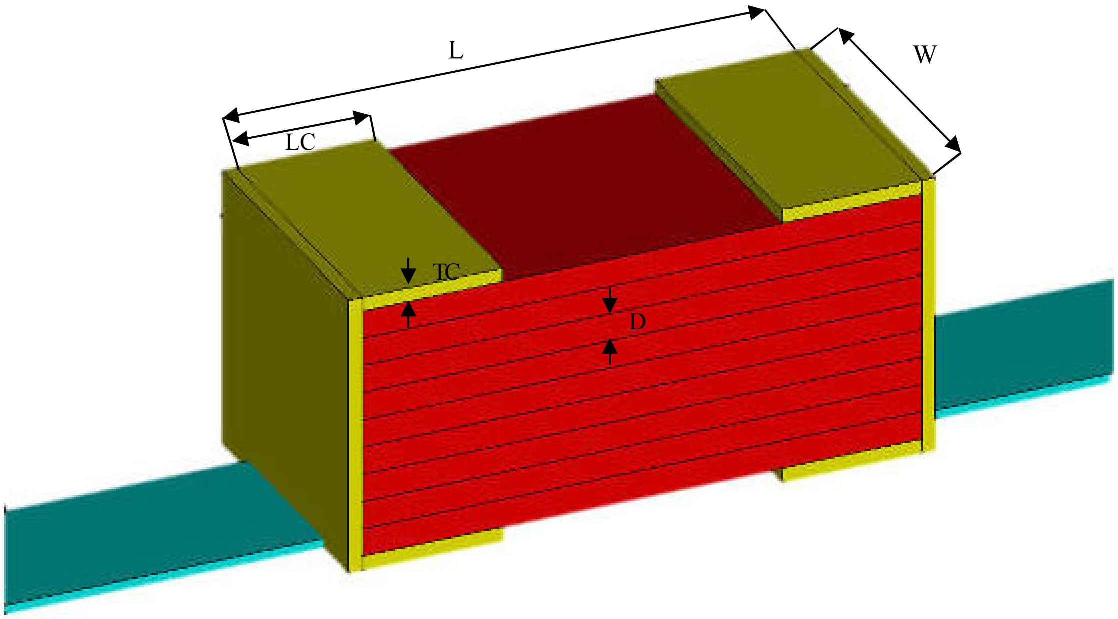
Summary

CHCAP2_EM is a dynamic model of a multiplate ceramic chip capacitor. You can specify the number of interdigitally stacked metal plates, arrange plates and select dielectric properties, and set size and properties of contact pads.

This element is for use in 3D EM documents only. It is intended to create a 3D parameterized cell for a multiplate chip capacitor for 3D EM analysis. See "Using 3D EM Elements " for details on using 3D pCells with Analyst.

Topology





Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	IC1
N	Number of plates		5
L	Length of plates (less contact thickness)	Length	2 mm
W	Width of plates	Length	2 mm
WG	Interdigital gap between plate tip and side contact	Length	0.1 mm
MG	Plate side recess into capacitor	Length	0.08 mm
D	Thickness of dielectric layers	Length	0.2 mm
T	Plate thickness	Length	0.02 mm
Rho	Plate metal bulk resistivity relative to gold		1.0
Er	Relative dielectric constant of filling dielectric		10
Tand	Loss tangent of filling dielectric		1e-4
LC	Length of contact pad	Length	0.3 mm
TC	Thickness of contact pad	Length	0.1 mm
Rho *DIE_NAME	Contact pad metal bulk resistivity relative to gold Filling dielectric name for 3D EM cell		1.0 ""

** indicates a secondary parameter*

Parameter Details

N. Specifies the total number of interdigitally stacked plates.

L. Specifies the length of the capacitor (thickness of side contacts excluded), (see the "Topology" section).

WG. Specifies the interdigital gap between the plate end and the side contact (see the "Topology" section).

MG. Specifies the recess of the plate side into the dielectric (see the "Topology" section).

Parameter Restrictions and Recommendations

1. The number of dielectric layers inside the capacitor is N+1. A metal plate sits on top of each layer (except the top layer). Half of the plates connect to the left contact pad; the rest of the plates connects to the right contact pad.
2. The total length of contact pads 2*LC cannot be greater than 0.95*L.
3. A 3D cell sets the lower cap on T and TC values equal to 0.1 micron.
4. The DIE_NAME (hidden) parameter may be used to assign different dielectric materials to multiple instances of CHCAP2_EM.

Layout

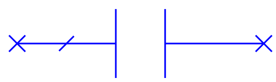
The layout for this cell has hard-coded model layers. When you first use this layout cell, a layer named "CHCAP2_EM" is added to your drawing layer and model layer list (if they are not already there). Using the model layer mapping, you can assign these layers to draw on any drawing layer.

3D EM Layout

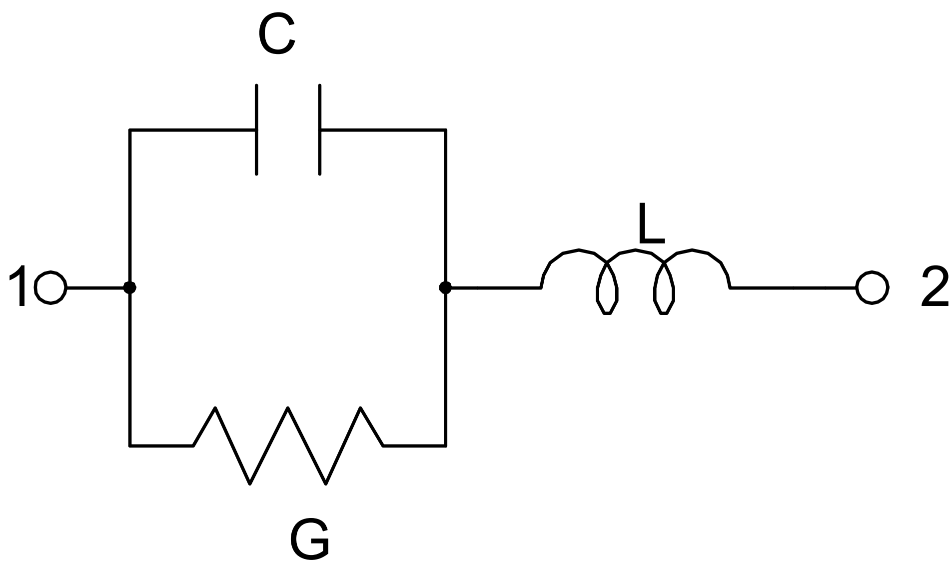
This element has a layout cell specifically for Analyst 3D EM layouts. See "Using 3D EM Elements " for details on using 3D pCells with Analyst.

Chip Capacitor (Closed Form): CHIPCAP

Symbol



Equivalent Circuit



Parameters

Name	Description	Unit Type	Default
ID	Name	Text	C1
C	Capacitance	Capacitance	1 pF
Q	Q		0
FQ	Frequency at which Q is evaluated	Frequency	0 GHz
FR	Series-resonant frequency	Frequency	0 GHz
ALPH	Exponent for scaling Q		1
DCMod	DC modeling	Vector text (pull-down)	Lossless

Parameter Details

DCMod. Specifies the DC behavior of the model. **Lossless** indicates that the model is an ideal capacitor at DC. **Lossy** indicates that loss is taken into account at DC.

Parameter Notes

Entering zero (or leaving the default value) for FQ or Q makes the loss resistance infinite.

Entering zero for FR or C makes the series inductance zero.

Implementation Details

Implements a chip capacitor with frequency-dependent Q.

$$Q(f) = Q\left(\frac{f}{\mathrm{FQ}}\right)^{\mathrm{ALPH}}$$

$$L = \frac{1}{(2\pi \cdot \mathrm{FR})^2 \cdot C}$$

The impedance of the capacitor is given by:

$$Z = \frac{1}{2\pi\mathrm{fC}\left(\frac{1}{Q(f)} + j\right)} + j2\pi f L$$

Layout

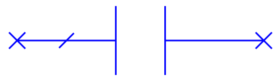
This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Restrictions

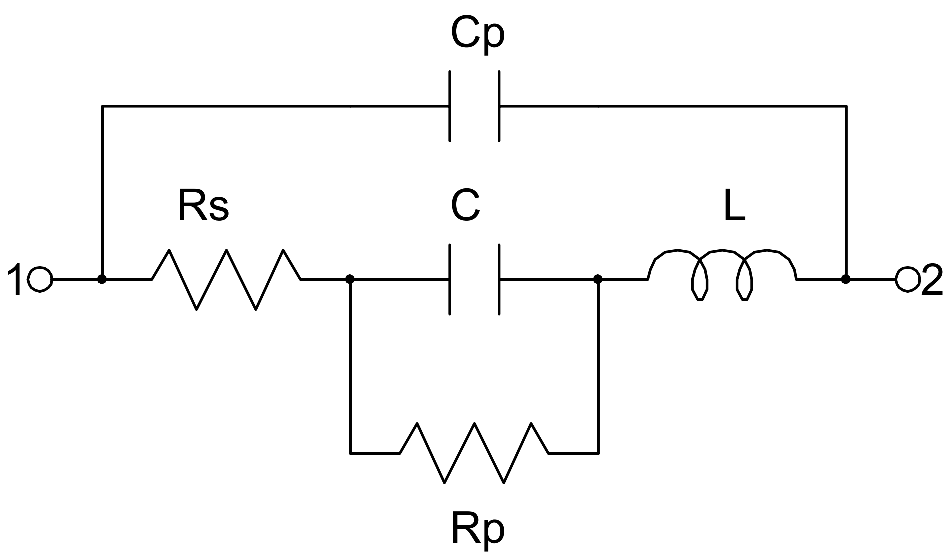
1. C, Q, FR, and FQ must be greater than or equal to zero.

Advanced Chip Capacitor Model (Closed Form): CHIPCAP2

Symbol



Equivalent Circuit



Parameters

Name	Description	Unit Type	Default
ID	Name	Text	C1
C	Capacitance	Capacitance	1.0 pF
Q	Q		0.0
TAND	Dielectric loss tangent		0.0
FQ	Frequency at which Q is evaluated	Frequency	0.0 GHz
FRS	Series-resonant frequency	Frequency	0.0 GHz
FRQ	Parallel-resonant frequency	Frequency	0.0 GHz
ALPH	Exponent for scaling Q		1.0

Parameter Details

FQ and **Q**. Entering zero (or leaving the default value) makes the loss resistance infinite.

FRS or **C**. Entering zero makes the series inductance zero.

FRP. Entering zero makes Cp zero.

Implementation Details

CHIPCAP2 is an advanced chip capacitor model, which includes several important phenomena:

- Finite, frequency-dependent Q
- Series and parallel resonance
- Dielectric losses.

These phenomena are evident in real, chip capacitors, and thus should be included in a meaningful model.

Unlike CHIPCAP, the dominant effect in determining the Q is series resistance; this is usually the case with chip capacitors. Thus, the Q of CHIPCAP2 scales more correctly with frequency than CHIPCAP. For this reason, using CHIPCAP2 is recommended in new designs.

The Q is scaled as

$$Q(f) = Q\left(\frac{\mathrm{FQ}}{f}\right)^{\mathrm{ALPH}}$$

where

$$Q = \frac{1}{2\pi FQR_sC}$$

When *ALPH*=1 (the default),

$$Q(f) = \frac{1}{2\pi fR_sC}$$

The inductance is

$$L = \frac{1}{(2\pi \cdot \mathrm{FRS})^2 \cdot C}$$

and the capacitance that accounts for parallel resonance, C_p,, is

$$C_p = \frac{1}{\omega_p^2 L_{\mathrm{eq}}}$$

where ω_p=2π·FRP. L_{eq} is the equivalent inductance,

$$\omega_p L_{\mathrm{eq}} = \omega_p L - \frac{1}{\omega_p C}$$

From these expressions, C_p can be expressed as

$$C_p = \frac{C}{\frac{\omega_p^2}{\omega_s^2} - 1}$$

Layout

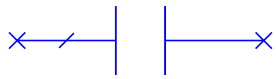
This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Restrictions

1. *C, Q, FRS, FRP,* and *FQ* must be greater than or equal to zero.
2. *FRP* must be greater than *FRS*.

Simple Chip Capacitor (Closed Form): CHIPCAPS

Symbol



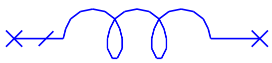
Summary

Parameters

Name	Description	Unit Type	Default
ID	Name	Text	IC1
L	Plate length		100 um
W	Plate width		40 um
D	Thickness of filling dielectric		2 um
T	Plate thickness		2 um
Rho	Metal resistivity relative to gold		1
ErF	Relative dielectric constant of filling dielectric		1
TandF	Loss tangent of filling dielectric		0
MSUB	Substrate definition		
*DIE_NAME	Filling dielectric name for 3D EM cell		""

Toroidal Coil Inductor (Closed Form): CIND

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	L1
N	Number of turns		1
AL	Inductance index	Inductance	1 nH

Implementation Details

Inductance calculated as

$$L = N^2 \cdot \mathrm{AL}$$

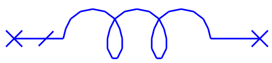
$$N,\ \mathrm{AL} > 0$$

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Air Wound Coil: COIL

Symbol



Summary

COIL models an air core wire coil using closed form approximation for frequency dependent resistance and inductance of cylindrical wire.

Parameters

Name	Description	Unit Type	Default
ID	Name	Text	IND1
N	Number of turns	Scalar	4
R	Coil inner radius	Length	5 mm
L	Coil length	Length	5 mm
Rho	Bulk resistance of wire metal normalized to gold	Scalar	1
DIA_AWG	Wire diameter specification mode	Text	"Diameter"
DIA or AWG	Diameter/AWG of coil wire (name depends on the value of parameter DIA_AWG)	Text	40 mkm/"26" (the default value depends on the value of parameter DIA_AWG)

Parameter Details

L. Represents a longitudinal length of a coil.

DIA_AWG. This is a switch that allows you to specify wire diameter according to user preferences: In project length units (for example, microns, or millimeters) or in American Wire Gauge (AWG).

Diameter/AWG. The name and value of the last parameter of COIL changes dynamically depending on the value of parameter DIA_AWG. If DIA_AWG=Diameter then the name of the last parameter becomes "Diameter" and you can enter any value in project length units; if DIA_AWG=AWG then the name of the last parameter becomes "AWG" and you can select the AWG number from a drop-down list.

Parameter Restrictions and Recommendations

1. The value of parameter L should be greater than or equal to N*Diameter where Diameter is the wire diameter. This model corrects L if L is less than N*Diameter.
2. Minimal value of inner radius R is 0.25 mm. If user specifies R<0.25 mm then COIL internally sets R=0.25 mm.

Implementation Details

This model is implemented as a series connection of lumped inductor and lumped frequency-dependent resistor.

Layout

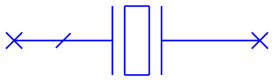
This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Recommendations for Use

COIL is more accurate if L >0.7*R and if L does not exceed by much the product N*Diameter, (if coil is relatively tightly wound).

Crystal Resonator (Closed Form): CRYSTAL

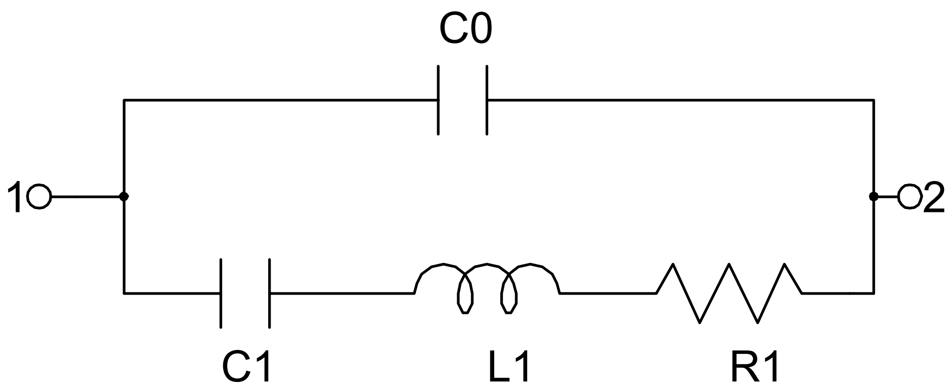
Symbol



Summary

CRYSTAL is a linear model of a crystal using a series RLC resonator in parallel with a static capacitance (see the equivalent circuit). This model calculates the motional inductance, from the series resonant frequency and the motional capacitance.

Equivalent Circuit



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	CR1
Fs	Series Resonant Frequency	GHz	.025
R1	Motional Resistance	ohm	50
C1	Motional Capacitance	pF	2e-4
C0	Static Capacitance	pF	2
L1	Computed Motional Inductance	nH	2.026e8

Parameter Details

Fs, R1, C1, and C0. These values are typically given in the data sheet.

L1. This is a calculated, uneditable parameter.

Implementation Details

This element is implemented as an aggregate model of an SLRC element in shunt with a capacitor. The motional inductance, L1 is calculated from the series resonant frequency Fs, and the motional capacitance, C1.

$$L1 = \frac{1}{(2\pi F_s)^2 \cdot C1}$$

References

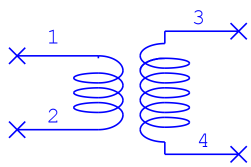
[1] J.R. Vig, "Quartz Crystal Resonators and Oscillators: A Tutorial." US Army Communications Command Attn: AMSEL-RD-C2-PT, Jan. 2001, pp. 3-20-3-23.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

Pair of Coupled Circular Wire Solenoids: CSOLIND

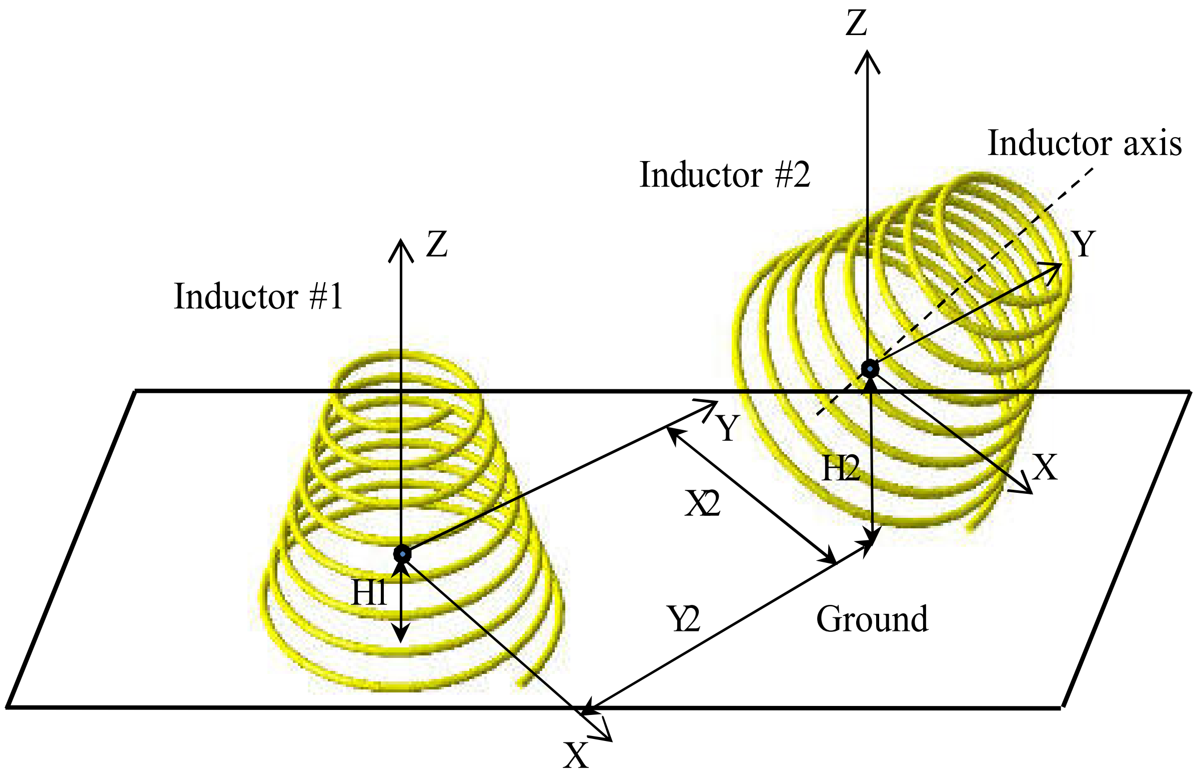
Symbol



Summary

This circuit component models a pair of electromagnetically coupled circular solenoids (optionally tapered) made of the wire of a cylindrical cross-section and situated above infinite perfectly conducting ground. The modeling approach is based on evaluation of self and mutual inductances, capacitances, and resistances between all spiral turns.

Topology



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	IND1
NT1	Number of turns of solenoid #1 (>=1)		5
NT2	Number of turns of solenoid #2(>=1)		5

Name	Description	Unit Type	Default
Dia	Wire diameter	Length	25 um
DS1	Start diameter of solenoid #1 (measured between wire centerlines)	Length	500 um
DS2	Start diameter of solenoid #2 (measured between wire centerlines)	Length	500 um
DE1	End diameter of solenoid #1(measured between wire centerlines)	Length	250 um
DE2	End diameter of solenoid #2 (measured between wire centerlines)	Length	250 um
PS1	Pitch of solenoid #1 (measured between wire centerlines along solenoid axis)	Length	80 um
PS2	Pitch of solenoid #2 (measured between wire centerlines along solenoid axis)	Length	80 um
H1	Elevation of solenoid #1 above ground	Length	300 um
H2	Elevation of solenoid #2 above ground	Length	300 um
X2	X-position of solenoid #2 relative to solenoid #1	Length	1500 um
Y2	Y-position of solenoid #2 relative to solenoid #1	Length	0 um
RotX1	Solenoid #1 angle of rotation about local X-axis	Degrees	0
RotX2	Solenoid #2 angle of rotation about local X-axis	Degrees	0
RotY1	Solenoid #1 angle of rotation about local Y-axis	Degrees	0
RotY2	Solenoid #2 angle of rotation about local Y-axis	Degrees	0
RotZ1	Solenoid #1 angle of rotation about local Z-axis	Degrees	0
RotZ2	Solenoid #2 angle of rotation about local Z-axis	Degrees	0
*Nts1	Solenoid #1 number of linear segments per turn		8
*Nts2	Solenoid #2 number of linear segments per turn		8
Rho	Wire metal bulk resistivity relative to gold		1
*Er	Relative dielectric constant of encasing material		1
*Tand	Loss tangent of encasing material		0

** indicates a secondary parameter*

Parameter Details

NT1, NT2 The numbers of spiral turns forming solenoids.

Dia. The diameter of solenoid wire.

DS1, DS2, DE1, DE2. Respectively, the start and end diameters of a tapered solenoid (solenoid spiral starts at the bottom). This model checks if DE<=DS and issues an error message if this condition does not hold.

PS1, PS2. The axial distance **(measured along the solenoid axis)** between wire centerlines of two adjacent spiral turns (pitch). For cylindrical and tapered solenoids, this model checks if the distance between adjacent turns measured along the cone/cylinder generatrix is greater than Dia.

H1, H2. The elevation of the solenoid start point (see "Topology") above ground.

RotX1, RotX2, RotY1, RotY2, RotZ1, RotZ2. The tilt of solenoid axes is controlled by these angles. RotX1, RotX2 rotate the solenoid axis about local axis X; RotY1, RotY2 rotate the solenoid axis about axis Y; RotZ1, RotZ2 rotate the solenoid about local axis Z (see "Topology" for a definition of local axes). Rotation angles are positive when rotations appear counter-clockwise and the axis about which rotations occur points toward the observer. This model tracks the position of all conducting parts and issues an error if a combination of specified parameters makes a solenoid(s) touch ground or another solenoid. Changes made to the solenoid angular position reflect on its image on the 3D Layout view.

Nts1, Nts2. The number of linear wire segments a model uses to approximate a single spiral turn (see "Implementation Details"). Nseg must be greater than or equal to 6.

Er, Tand. Parameters of material used for optional encasing. The defaults are Er=1, Tand=0.

Parameter Restrictions and Recommendations

1. NT should be greater or equal to 1.
2. Model checks that Nts≥6.
3. Model checks if DE≤DS.
4. This model checks PS for each inductor. Note that PS is an "axial pitch" (the distance between turn wire centerlines measured along the inductor axis). For conical solenoids this model checks "external pitch" (the distance between turn wire centerlines measured along the cone generatrix). This external pitch must be greater than Dia. When DE=DS, axial and external pitches are equal in value.

Implementation Details

Each spiral turn is broken into Nts linear wire segments and the model evaluates self and mutual RLC parameters of all segments including the image wire segments (due to ground pesence).This model controls the length of each segment and issues an error message if the length of any segment exceeds the wire diameter Dia or warning if segment length is too small (see Recomenadtions for Use). This model accounts for a skin effect and partly for radiation loss.

Recommendations for Use

Note that Port 1 is assigned to the start point of inductor #1 and Port 2 is assigned to the end point of inductor #1 (start point at the bottom and end point at the top). The same is true for ports 3 and 4 attached to inductor #2.

This model predicts the frequency behavior of the following solenoids and in the vicinity of the first self-resonance frequency.

User discretion is advised if this model issues a warning about segment length being too small. CSOLIND assumes that each segment is represented by infinitely thin filament, which means Lseg >>Dia. Thus, very short (relative to wire diameter) segments may cause additional errors. This model issues a warning message if Lseg<6*Dia. This message contains the segment number. All segments with **numbers greater than that in violation** are also in violation. If DE<DS, segments in violation often belong to a few top turns. You get an approximate estimation of the percentage of segments with reduced accuracy.

Table of conversion AWG (American Wire Gauge) to millimeters (for those who prefer wire gauge for denoting wire diameter).

AWG	Wire diameter (mm)	AWG	Wire diameter (mm)	AWG	Wire diameter (mm)
12	2.053	25	0.4547	38	0.1007
13	1.828	26	0.4049	39	0.08969
14	1.628	27	0.3606	40	0.07987
15	1.450	28	0.3211	41	0.07112
16	1.291	29	0.2859	42	0.06334
17	1.150	30	0.2546	43	0.05641
18	1.024	31	0.2268	44	0.05023
19	0.9116	32	0.2019	45	0.04473
20	0.8118	33	0.1798	46	0.03983
21	0.7229	34	0.1601	47	0.03547
22	0.6438	35	0.1426	48	0.03160
23	0.5733	36	0.1270	49	0.02813
24	0.5106	37	0.1131	50	0.02505

Layout

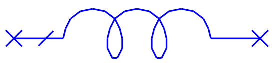
The layout for this cell has hard-coded model layers. When you first use this layout cell, a layer named "Solenoid" is added to your drawing layer and model layer list (if they are not already there). Using the model layer mapping, you can assign these layers to draw on any drawing layer. The 2D layout displays a circle the size of the DS parameter plus the Dia size. The 3D layout displays the shapes as a solenoid.

This model has a layout only parameter that controls how the layout cell draws. You access the parameters by selecting the item in the layout, right-clicking and choosing **Shape Properties** to display the Cell Options dialog box, then clicking the **Parameters** tab.

Name	Value	Units	Description
NSeg	32	Integer	Number of linear segments approximating one spiral turn.

Inductor (Closed Form): IND

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Name	Text	L1
L	Inductance	Inductance	0 nH

Implementation Details

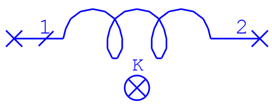
Implements an ideal inductor.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

Inductor with Optional Coupling (Closed Form): INDK

Symbol



Summary

INDK implements an ideal inductor with an option to include coupling to other INDK elements. The coupling is modeled by other elements (INDM or K) connected between the "K" ports of two INDK elements.

Parameters

Name	Description	Unit Type	Default
ID	Name	Text	L1
L	Inductance	Inductance	1 nH

Implementation Details

The INDK element implements an ideal inductor with the option to be mutually coupled to other INDK elements. The coupling between two INDK elements can be represented by using one of two elements: the INDM element, which represents the mutual inductance between the two INDK elements, or the K element, which represents the coupling coefficient. If nothing is connected to the "K" port of the INDK element, the inductor is considered ideal.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Recommendations for Use

Figure 1 and Figure 2 show how these elements are typically used. Figure 3 shows how to model mutual coupling between more than two inductors.

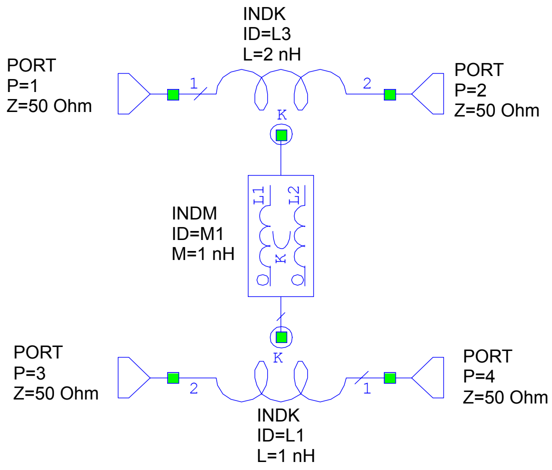


Figure 1. Typical use of the INDK and INDM elements to model mutual coupling between two inductors. The INDM element is used to model the mutual coupling between the two INDK elements by specifying an actual mutual inductance.

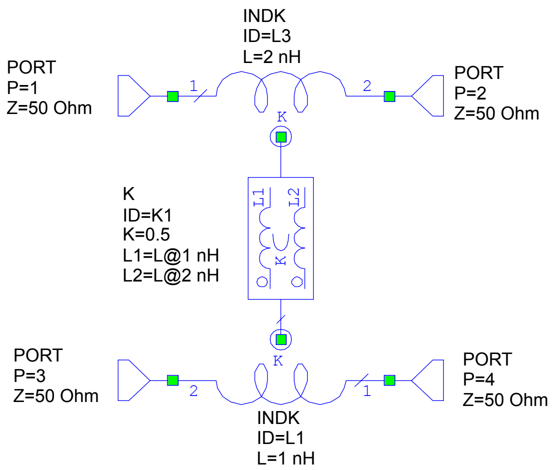


Figure 2. Typical use of the INDK and K elements to model mutual coupling between two inductors. The K element is used to model the mutual coupling between the two INDK elements by specifying the coupling coefficient, K, and the inductances of the two coupled inductors.

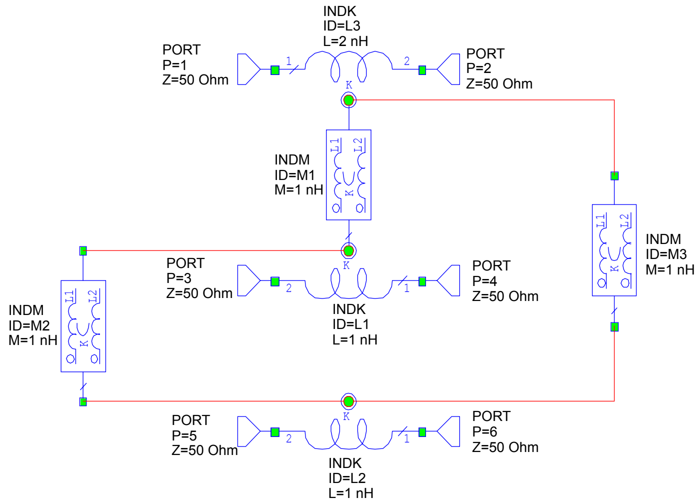
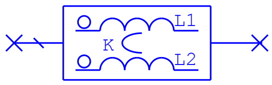


Figure 3. Typical use of the INDK and INDM elements to model mutual coupling between more than two inductors. K elements could also be used in place of the INDM elements.

Mutual Inductance (Closed Form): INDM

Symbol



Summary

INDM is used to represent the mutual coupling between inductors (INDK elements).

Parameters

Name	Description	Unit Type	Default
ID	Name	Text	L1
M	Mutual inductance	Inductance	1 nH

Implementation Details

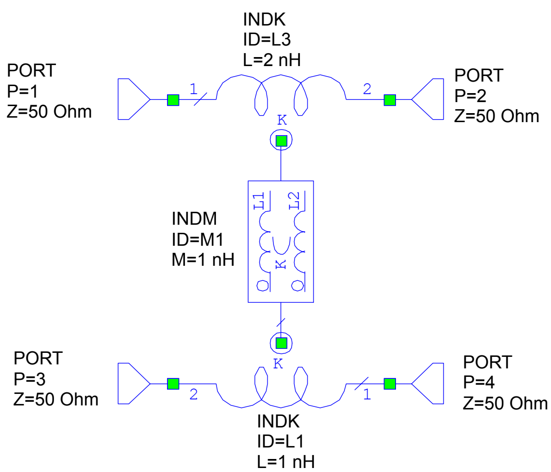
The INDM element represents the mutual coupling between inductors. This element is meant to be used between two INDK elements and should be connected to the "K" ports of those elements.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Recommendations for Use

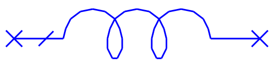
The following figure shows a typical use of INDM.



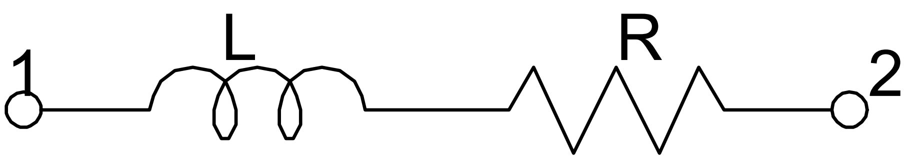
Use of the INDK and INDM elements to model mutual coupling between two inductors. The INDM element is used to model the mutual coupling between the two INDK elements by specifying an actual mutual inductance.

Inductor With Q (Closed Form): INDQ

Symbol



Topology



Parameters

Name	Description	Unit Type	Default
ID	Name	Text	L1
L	Inductance	Inductance	1 nH
Q	Q		0
FQ	Frequency at which Q is evaluated	Frequency	0 GHz
ALPH	Scaling factor for Q		1
DCMod	DC modeling	Vector text (pull-down)	Lossless

Parameter Details

DCMod. Specifies the DC behavior of the model. **Lossless** indicates that the model is modeled as an ideal inductor at DC. **Lossy** indicates that loss is taken into account at DC.

Implementation Details

Implements an inductor with frequency-dependent Q.

$$Q(f) = Q\left(\frac{f}{\mathrm{FQ}}\right)^{\mathrm{ALPH}}$$

The impedance of the inductor is given by:

$$Z = R + \mathrm{jX} = 2\pi \mathrm{fL}\left(\frac{1}{Q(f)} + j\right)$$

$$\left(Q_L = \frac{X}{R} = \frac{2\pi \mathrm{fL}}{R} \quad \text{or} \quad R = \frac{2\pi \mathrm{fL}}{Q_L}\right)$$

Layout

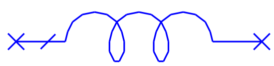
This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

Restrictions

FQ must be greater than or equal to zero.

Inductor with Frequency-dependent Inductance, Parasitic and Loss: INDQP

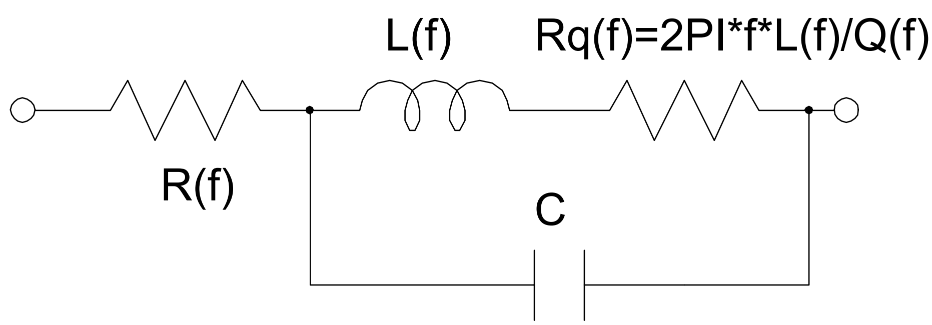
Symbol



Summary

INDQP models an inductor with user-specified frequency-dependence of inductance and loss. Frequency dependencies are specified as lookup tables (vectors) in model parameters. This model uses interpolation to obtain parameter values at each project evaluation frequency.

Equivalent Circuit



Parameters

Name	Description	Unit Type	Default
ID	Name	Text	IND#
F	Vector of frequencies at which L, Q, and R are specified	Frequency	{1}
L	Inductance (vector)	Inductance	Li[a]
Q	Quality factor (vector)	Scalar	{1000}
C	Capacitance	Parasitic capacitance	C[a]
R	Series resistance (vector)	Resistance	{0}

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See the Microwave Office Layout Guide for details.

Parameter Details

F. Vector of frequencies at which L, Q, and R parameters are specified. Frequencies must be sequential and specified in ascending order.

L. Vector of series inductance L(f) (see the "Equivalent Circuit" section) specified in inductance project units. You must specify each vector entry at the corresponding frequency entry from frequency vector F.

Q. Vector of quality factor; specifies series resistance Rq (f) (see the "Equivalent Circuit" section). You must specify each vector entry at the corresponding frequency entry from frequency vector F.

C. Parasitic capacitance (see the "Equivalent Circuit" section).

R. Vector of series resistance R(f) (see the "Equivalent Circuit" section) specified in resistance project units. You must specify each vector entry at the corresponding frequency entry from frequency vector F.

Parameter Restrictions and Recommendations

1. The size of vector parameters L, Q, and R must be equal to the size of frequency vector F.
2. If the project evaluation frequency is out of range of frequencies in F, then L, Q, and R parameters are extrapolated as constant values equal to the first/last entries of corresponding vectors. No warning is issued.
3. You can specify the vector in three ways: First, by entering it as a right side value of the model parameter, for example R={100,102,110,113,120}; second, by specifying the vector elsewhere in the equation; and third, by specifying the vector in a column or row of a text file. The third way provides a convenient and flexible method of specifying L, Q, and R parameters at a single location. For example, you can create the file *indqp.txt* containing space separated columns of L, Q, and R. The first column must represent frequency in project units (note that changing project default frequency units demands manual scaling of frequencies in this file). Import or link this file to your project and name it, for example, INDQP_1. Now you can specify, for example, parameter R as R = Col(datafile("INDQP_1"),4) so that values of vector R are copied to the model from column 4 of file *indqp.txt* imported under the name indqp_1. If you prefer to deploy your data row-wise use R = Row(datafile("INDQP_1"),2).
4. If your project uses a text file input to feed data to this model be aware what frequency, resistance, inductance or conductance units this file implies. Your project default units may differ from those in your data file. If this happens, you must scale input values, multiplying the call of function Col or Row by a scaling coefficient. For example, if your project uses inductance in nanohenries and the data file contains data in microhenries you may get inductance data from column 2 of the data file INDQP_1 such as: L= 1e+3*Col(datafile("INDQP_1"),2).

Implementation Details

This model is implemented as a series connection of lossy inductor and frequency-dependent resistor. A parasitic capacitor shunts a lossy inductor.

Model implementation is based on linear interpolation of L, Q, and R parameters at each project evaluation frequency. Interpolation uses user-supplied lookup tables via parameters. If the project evaluation frequency is out of range of frequencies in F, then L, Q, and R parameters are extrapolated as constant values equal to the first/last entries of corresponding vectors.

Layout

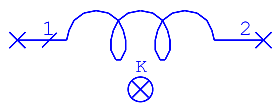
This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Recommendations for Use

Time transient analysis is sensitive to the number of frequency points. Also, time domain measurements may be inaccurate due to non-causal behavior of the model at specified parameter values.

Inductor with Series Resistive Loss and Optional Coupling (Closed Form): INDRK

Symbol



Summary

INDRK implements an ideal inductor in series with a resistor with an option to include coupling to other INDRK and/or INDK elements. The coupling is modeled by other elements (INDM, INDK, or INDRM) connected between the "K" ports of two INDRK (or INDK) elements.

Parameters

Name	Description	Unit Type	Default
ID	Name	Text	L1
R	Resistor	Resistance	0 ohm
L	Inductance	Inductance	1 nH

Implementation Details

The INDRK element implements an ideal inductor in series with the resistor and with the option to be mutually coupled to other INDRK/INDK elements. The coupling between two INDRK/INDK elements can be represented by using one of three elements: the INDM element, which represents the mutual inductance between the two INDRK/INDK elements, the INDRM element, which represents the mutual inductance with resistive loss between the two INDRK/INDK elements, or the K element, which represents the coupling coefficient. If nothing is connected to the "K" port of the INDK element, the inductor L is considered ideal (in series with the resistor R.)

Layout

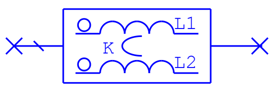
This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Recommendations for Use

See "Recommendations for Use" for the INDK model.

Mutual Inductance with Resistive Loss (Closed Form): INDRM

Symbol



Summary

INDRM is used to represent the mutual coupling between inductors if resistive coupling loss is present (INDK elements).

Parameters

Name	Description	Unit Type	Default
ID	Name	Text	L1
R	Mutual resistance	Resistance	0 ohm
M	Mutual inductance	Inductance	0 nH

Implementation Details

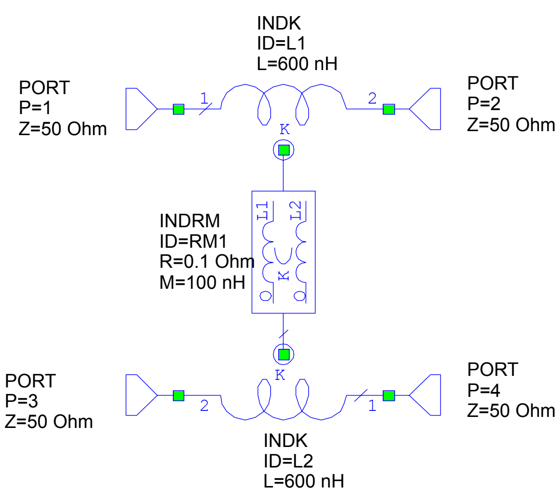
The INDRM element represents the mutual coupling between inductors. Parameter R represents the optional lossy coupling that exists, for example, in coupled Tx lines (R is in series with M). This element is meant to be used between two INDK elements and should be connected to the "K" ports of those elements.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Recommendations for Use

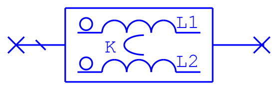
The following figure shows the typical use of INDK and INDM elements to model mutual coupling between two inductors.



The INDRM element is used to model the mutual coupling between the two INDK elements by specifying an actual mutual inductance and resistance.

Inductor Coupling Coefficient (Closed Form): K

Symbol



Summary

K is used to represent the mutual coupling between inductors (INDK elements).

Parameters

Name	Description	Unit Type	Default
ID	Name	Text	K2
K	Coupling Coefficienct		0
*L1	Inductor 1 Value		L@1 nH
*L2	Inductor 2 Value		L@2 nH

** indicates a secondary parameter*

Implementation Details

K represents the mutual coupling between inductors. This element is for use between two INDK elements and should be connected to the "K" ports of those elements. The mutual inductance is calculated as:

$$M = K \cdot \sqrt{L1 \cdot L2}$$

where M is the mutual inductance, K is the coupling coefficient, L1 is the inductance connected at port 1, and L2 is the inductance connected at port 2.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

Recommendations for Use

Figure 1 shows how this element is typically used.

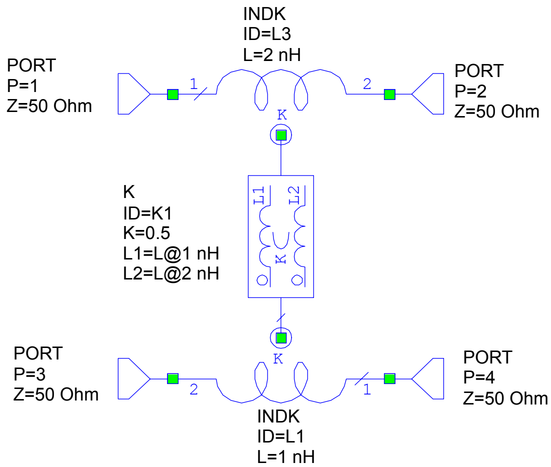
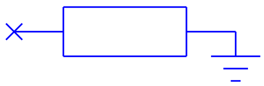


Figure 1. Typical use of the INDK and K elements to model mutual coupling between two inductors. The K element is used to model the mutual coupling between the two INDK elements by specifying the coupling coefficient, K, and the inductances of the two coupled inductors.

Grounded Resistor (Closed Form): LOAD

Symbol



Parameters

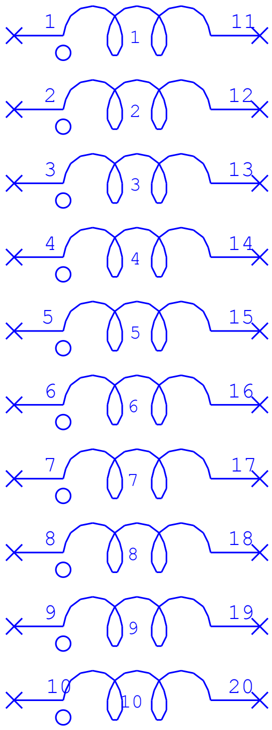
Name	Description	Unit Type	Default
ID	Element ID	Text	Z1
Z	Load impedance	Resistance	50 ohm

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Mutually Coupled Coils 10 Inductors: MUC10

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	M1
L1	Self-inductance of coil 1	Inductance	1 nH
R1	Resistance of coil 1	Inductance	0 ohm
L2	Self-inductance of coil 2	Inductance	1 nH
R2	Resistance of coil 2	Resistance	0 ohm
L3	Self-inductance of coil 3	Inductance	1 nH
R3	Resistance of coil 3	Resistance	0 ohm
L4	Self-inductance of coil 4	Inductance	1 nH
R4	Resistance of coil 4	Resistance	0 ohm
L5	Self-inductance of coil 5	Inductance	1 nH
R5	Resistance of coil 5	Resistance	0 ohm
L6	Self-inductance of coil 6	Inductance	1 nH
R6	Resistance of coil 6	Resistance	0 ohm
L7	Self-inductance of coil 7	Inductance	1 nH
R7	Resistance of coil 7	Resistance	0 ohm

Name	Description	Unit Type	Default
L8	Self-inductance of coil 8	Inductance	1 nH
R8	Resistance of coil 8	Resistance	0 ohm
L9	Self-inductance of coil 9	Inductance	1 nH
R9	Resistance of coil 9	Resistance	0 ohm
L10	Self-inductance of coil 10	Inductance	1 nH
R10	Resistance of coil 10	Resistance	0 ohm
K1_2	Coupling coefficient between coils 1 and 2		0
K1_3	Coupling coefficient between coils 1 and 3		0
K1_4	Coupling coefficient between coils 1 and 4		0
K1_5	Coupling coefficient between coils 1 and 5		0
K1_6	Coupling coefficient between coils 1 and 6		0
K1_7	Coupling coefficient between coils 1 and 7		0
K1_8	Coupling coefficient between coils 1 and 8		0
K1_9	Coupling coefficient between coils 1 and 9		0
K1_10	Coupling coefficient between coils 1 and 10		0
K2_3	Coupling coefficient between coils 2 and 3		0
K2_4	Coupling coefficient between coils 2 and 4		0
K2_5	Coupling coefficient between coils 2 and 5		0
K2_6	Coupling coefficient between coils 2 and 6		0
K2_7	Coupling coefficient between coils 2 and 7		0
K2_8	Coupling coefficient between coils 2 and 8		0
K2_9	Coupling coefficient between coils 2 and 9		0
K2_10	Coupling coefficient between coils 2 and 10		0
K3_4	Coupling coefficient between coils 3 and 4		0
K3_5	Coupling coefficient between coils 3 and 5		0
K3_6	Coupling coefficient between coils 3 and 6		0
K3_7	Coupling coefficient between coils 3 and 7		0
K3_8	Coupling coefficient between coils 3 and 8		0
K3_9	Coupling coefficient between coils 3 and 9		0
K3_10	Coupling coefficient between coils 3 and 10		0
K4_5	Coupling coefficient between coils 4 and 5		0
K4_6	Coupling coefficient between coils 4 and 6		0
K4_7	Coupling coefficient between coils 4 and 7		0
K4_8	Coupling coefficient between coils 4 and 8		0
K4_9	Coupling coefficient between coils 4 and 9		0
K4_10	Coupling coefficient between coils 4 and 10		0
K5_6	Coupling coefficient between coils 5 and 6		0

Name	Description	Unit Type	Default
K5_7	Coupling coefficient between coils 5 and 7		0
K5_8	Coupling coefficient between coils 5 and 8		0
K5_9	Coupling coefficient between coils 5 and 9		0
K5_10	Coupling coefficient between coils 5 and 10		0
K6_7	Coupling coefficient between coils 6 and 7		0
K6_8	Coupling coefficient between coils 6 and 8		0
K6_9	Coupling coefficient between coils 6 and 9		0
K6_10	Coupling coefficient between coils 6 and 9		0
K7_8	Coupling coefficient between coils 7 and 8		0
K7_9	Coupling coefficient between coils 7 and 9		0
K7_10	Coupling coefficient between coils 7 and 10		0
K8_9	Coupling coefficient between coils 8 and 9		0
K8_10	Coupling coefficient between coils 8 and 10		0
K9_10	Coupling coefficient between coils 9 and 10		0

Implementation Details

Pin numbers for each coil are assigned by the convention i = coupled port and N+i= output port for each coil, N = number of coils. So for coil 1, 1 is the coupled port and 11 is the output port for 10 coils. Vci is the voltage across coil i, i=1, ...,N:

$$V_{ci} = (R_i + j\omega L_i) \cdot I_i + \sum_{j=1}^{N} j\omega M_{ij} I_j$$

$$M_{\mathrm{ij}} = K_{\mathrm{ij}} \cdot \sqrt{L_i \cdot L_j}$$

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Restrictions

$$L_i > 0$$

$$R_i > 0$$

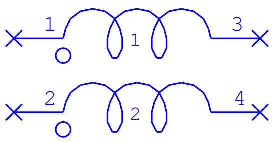
$$-1 < K_{\mathrm{ij}} < 1$$

where

$$-1 < K_{\mathrm{ij}} < 1$$

Mutually Coupled Coils 2 Inductors (Closed Form): MUC2

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	M1
L1	Self-inductance of coil 1	Inductance	1 nH
R1	Resistance of coil 1	Resistance	0 ohm
L2	Self-inductance of coil 2	Inductance	1 nH
R2	Resistance of coil 2	Resistance	0 ohm
K1_2	Coupling coefficient between coils 1 and 2		0

Implementation Details

V_{ci} is the voltage across coil I, I=1, ...,N:

$$V_{ci} = (R_i + j\omega L_i) \cdot I_i + \sum_{j=1}^{N} j\omega M_{ij} I_j$$

$$M_{ij} = K_{ij} \cdot \sqrt{L_i \cdot L_j}$$

$$L_i > 0$$

$$R_i > 0$$

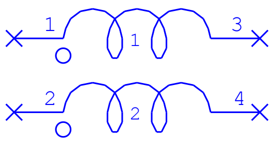
$$-1 < K_{ij} < 1$$

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

(Obsolete) Lossless Mutually Coupled Coils, 2 Inductors (Closed Form): MUC2_M

Symbol



Summary

This element is OBSOLETE and is replaced by the Mutually Coupled Coils 2 Inductors (Closed Form) (MUC2) element.

Parameters

Name	Description	Unit Type	Default
L1	Self-inductance of coil 1	Inductance	1 nH
L2	Self-inductance of coil 2	Inductance	1 nH
M	Mutual inductance	Inductance	0 nH

Implementation Details

$$V_{ci} = (R_i + j\omega L_i) \cdot I_i + \sum_{j=1}^{N} j\omega M_{ij} I_j$$

$$M_{ij} = K_{ij} \cdot \sqrt{L_i \cdot L_j}$$

$$L_i > 0$$

$$R_i > 0$$

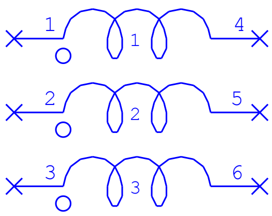
$$-1 < K_{ij} < 1$$

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

Mutually Coupled Coils 3 Inductors (Closed Form): MUC3

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	M1
L1	Self-inductance of coil 1	Inductance	1 nH
R1	Resistance of coil 1	Resistance	0 ohm
L2	Self-inductance of coil 2	Inductance	1 nH
R2	Resistance of coil 2	Resistance	0 ohm
L3	Self-inductance of coil 3	Inductance	1 nH
R3	Resistance of coil 3	Resistance	0 ohm
K1_2	Coupling coefficient between coils 1 and 2		0
K1_3	Coupling coefficient between coils 1 and 3		0
K2_3	Coupling coefficient between coils 2 and 3		0

Implementation Details

V_{ci} is the voltage across coil I, I=1, ...,N:

$$V_{ci} = (R_i + j\omega L_i) \cdot I_i + \sum_{j=1}^{N} j\omega M_{ij} I_j$$

$$M_{ij} = K_{ij} \cdot \sqrt{L_i \cdot L_j}$$

$$L_i > 0$$

$$R_i > 0$$

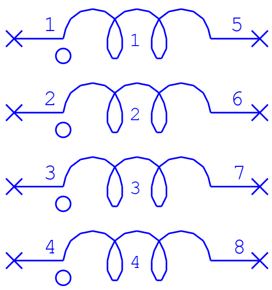
$$-1 < K_{ij} < 1$$

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

Mutually Coupled Coils 4 Inductors (Closed Form): MUC4

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	M1
L1	Self-inductance of coil 1	Inductance	1 nH
R1	Resistance of coil 1	Resistance	0 ohm
L2	Self-inductance of coil 2	Inductance	1 nH
R2	Resistance of coil 2	Resistance	0 ohm
L3	Self-inductance of coil 3	Inductance	1 nH
R3	Resistance of coil 3	Resistance	0 ohm
L4	Self-inductance of coil 4	Inductance	1 nH
R4	Resistance of coil 4	Resistance	0 ohm
K1_2	Coupling coefficient between coils 1 and 2		0
K1_3	Coupling coefficient between coils 1 and 3		0
K1_4	Coupling coefficient between coils 1 and 4		0
K2_3	Coupling coefficient between coils 2 and 3		0
K2_4	Coupling coefficient between coils 2 and 4		0
K3_4	Coupling coefficient between coils 3 and 4		0

Implementation Details

Pin numbers for each coil are assigned by the convention i = coupled port and N+i= output port for each coil, N = number of coils. So for coil 1, 1 is the coupled port and 5 is the output port for 4 coils. V_{ci} is the voltage across coil I, I=1, ...,N:

$$V_{ci} = (R_i + j\omega L_i) \cdot I_i + \sum_{j=1}^{N} j\omega M_{ij} I_j$$

$$M_{ij} = K_{ij} \cdot \sqrt{L_i \cdot L_j}$$

$$L_i > 0$$

$$R_i > 0$$

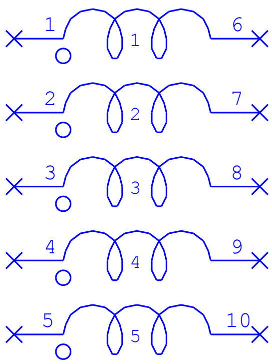
$$-1 < K_{\mathrm{ij}} < 1$$

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Mutually Coupled Coils 5 Inductors (Closed Form): MUC5

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	M1
L1	Self-inductance of coil 1	Inductance	1 nH
R1	Resistance of coil 1	Resistance	0 ohm
L2	Self-inductance of coil 2	Inductance	1 nH
R2	Resistance of coil 2	Resistance	0 ohm
L3	Self-inductance of coil 3	Inductance	1 nH
R3	Resistance of coil 3	Resistance	0 ohm
L4	Self-inductance of coil 4	Inductance	1 nH
R4	Resistance of coil 4	Resistance	0 ohm
L5	Self-inductance of coil 5	Inductance	1 nH
R5	Resistance of coil 5	Resistance	0 ohm
K1_2	Coupling coefficient between coils 1 and 2		0
K1_3	Coupling coefficient between coils 1 and 3		0
K1_4	Coupling coefficient between coils 1 and 4		0
K1_5	Coupling coefficient between coils 1 and 5		0
K2_3	Coupling coefficient between coils 2 and 3		0
K2_4	Coupling coefficient between coils 2 and 4		0
K2_5	Coupling coefficient between coils 2 and 5		0
K3_4	Coupling coefficient between coils 3 and 4		0
K3_5	Coupling coefficient between coils 3 and 5		0
K4_5	Coupling coefficient between coils 4 and 5		0

Implementation Details

Pin numbers for each coil are assigned by the convention i = coupled port and N+i= output port for each coil, N = number of coils. So for coil 1, 1 is the coupled port and 6 is the output port for 5 coils. Vci is the voltage across coil I, I=1, ...,N:

$$V_{ci} = (R_i + j\omega L_i) \cdot I_i + \sum_{j=1}^{N} j\omega M_{ij} I_j$$

$$M_{\mathrm{ij}} = K_{\mathrm{ij}} \cdot \sqrt{L_i \cdot L_j}$$

$$L_i > 0$$

$$R_i > 0$$

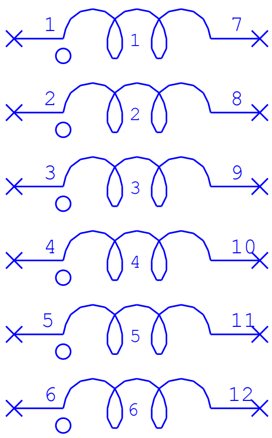
$$-1 < K_{\mathrm{ij}} < 1$$

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

Mutually Coupled Coils 6 Inductors (Closed Form): MUC6

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	M1
L1	Self-inductance of coil 1	Inductance	1 nH
R1	Resistance of coil 1	Resistance	0 ohm
L2	Self-inductance of coil 2	Inductance	1 nH
R2	Resistance of coil 2	Resistance	0 ohm
L3	Self-inductance of coil 3	Inductance	1 nH
R3	Resistance of coil 3	Resistance	0 ohm
L4	Self-inductance of coil 4	Inductance	1 nH
R4	Resistance of coil 4	Resistance	0 ohm
L5	Self-inductance of coil 5	Inductance	1 nH
R5	Resistance of coil 5	Resistance	0 ohm
L6	Self-inductance of coil 6	Inductance	1 nH
R6	Resistance of coil 6	Resistance	0 ohm
K1_2	Coupling coefficient between coils 1 and 2		0
K1_3	Coupling coefficient between coils 1 and 3		0
K1_4	Coupling coefficient between coils 1 and 4		0
K1_5	Coupling coefficient between coils 1 and 5		0
K1_6	Coupling coefficient between coils 1 and 6		0
K2_3	Coupling coefficient between coils 2 and 3		0
K2_4	Coupling coefficient between coils 2 and 4		0
K2_5	Coupling coefficient between coils 2 and 5		0
K2_6	Coupling coefficient between coils 2 and 6		0

Name	Description	Unit Type	Default
K3_4	Coupling coefficient between coils 3 and 4		0
K3_5	Coupling coefficient between coils 3 and 5		0
K3_6	Coupling coefficient between coils 3 and 6		0
K4_5	Coupling coefficient between coils 4 and 5		0
K4_6	Coupling coefficient between coils 4 and 6		0
K5_6	Coupling coefficient between coils 5 and 6		0

Implementation Details

Pin numbers for each coil are assigned by the convention i = coupled port and N+i= output port for each coil, N = number of coils. So for coil 1, 1 is the coupled port and 7 is the output port for 6 coils. Vci is the voltage across coil I, I=1, ...,N:

$$V_{ci} = (R_i + j\omega L_i) \cdot I_i + \sum_{j=1}^{N} j\omega M_{ij} I_j$$

$$M_{\text{ij}} = K_{\text{ij}} \cdot \sqrt{L_i \cdot L_j}$$

$$L_i > 0$$

$$R_i > 0$$

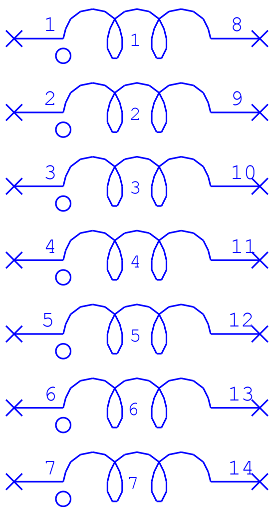
$$-1 < K_{\text{ij}} < 1$$

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

Mutually Coupled Coils 7 Inductors (Closed Form): MUC7

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	M1
L1	Self-inductance of coil 1	Inductance	1 nH
R1	Resistance of coil 1	Resistance	0 ohm
L2	Self-inductance of coil 2	Inductance	1 nH
R2	Resistance of coil 2	Resistance	0 ohm
L3	Self-inductance of coil 3	Inductance	1 nH
R3	Resistance of coil 3	Resistance	0 ohm
L4	Self-inductance of coil 4	Inductance	1 nH
R4	Resistance of coil 4	Resistance	0 ohm
L5	Self-inductance of coil 5	Inductance	1 nH
R5	Resistance of coil 5	Resistance	0 ohm
L6	Self-inductance of coil 6	Inductance	1 nH
R6	Resistance of coil 6	Resistance	0 ohm
L7	Self-inductance of coil 7	Inductance	1 nH
R7	Resistance of coil 7	Resistance	0 ohm
K1_2	Coupling coefficient between coils 1 and 2		0
K1_3	Coupling coefficient between coils 1 and 3		0
K1_4	Coupling coefficient between coils 1 and 4		0
K1_5	Coupling coefficient between coils 1 and 5		0
K1_6	Coupling coefficient between coils 1 and 6		0

Name	Description	Unit Type	Default
K1_7	Coupling coefficient between coils 1 and 7		0
K2_3	Coupling coefficient between coils 2 and 3		0
K2_4	Coupling coefficient between coils 2 and 4		0
K2_5	Coupling coefficient between coils 2 and 5		0
K2_6	Coupling coefficient between coils 2 and 6		0
K2_7	Coupling coefficient between coils 2 and 7		0
K3_4	Coupling coefficient between coils 3 and 4		0
K3_5	Coupling coefficient between coils 3 and 5		0
K3_6	Coupling coefficient between coils 3 and 6		0
K3_7	Coupling coefficient between coils 3 and 7		0
K4_5	Coupling coefficient between coils 4 and 5		0
K4_6	Coupling coefficient between coils 4 and 6		0
K4_7	Coupling coefficient between coils 4 and 7		0
K5_6	Coupling coefficient between coils 5 and 6		0
K5_7	Coupling coefficient between coils 5 and 7		0
K6_7	Coupling coefficient between coils 6 and 7		0

Implementation Details

Pin numbers for each coil are assigned by the convention i = coupled port and N+i= output port for each coil, N = number of coils. So for coil 1, 1 is the coupled port and 8 is the output port for 7 coils. Vci is the voltage across coil I, I=1, ...,N:

$$V_{ci} = (R_i + j\omega L_i) \cdot I_i + \sum_{j=1}^{N} j\omega M_{ij} I_j$$

$$M_{\text{ij}} = K_{\text{ij}} \cdot \sqrt{L_i \cdot L_j}$$

$$L_i > 0$$

$$R_i > 0$$

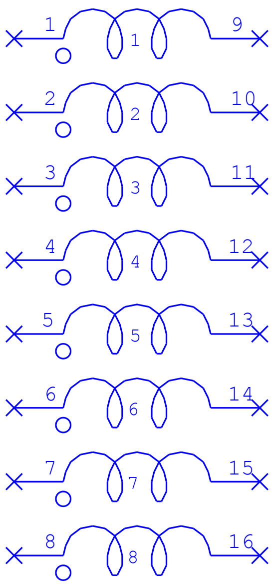
$$-1 < K_{\text{ij}} < 1$$

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

Mutually Coupled Coils 8 Inductors (Closed Form): MUC8

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	M1
L1	Self-inductance of coil 1	Inductance	1 nH
R1	Resistance of coil 1	Resistance	0 ohm
L2	Self-inductance of coil 2	Inductance	1 nH
R2	Resistance of coil 2	Resistance	0 ohm
L3	Self-inductance of coil 3	Inductance	1 nH
R3	Resistance of coil 3	Resistance	0 ohm
L4	Self-inductance of coil 4	Inductance	1 nH
R4	Resistance of coil 4	Resistance	0 ohm
L5	Self-inductance of coil 5	Inductance	1 nH
R5	Resistance of coil 5	Resistance	0 ohm
L6	Self-inductance of coil 6	Inductance	1 nH
R6	Resistance of coil 6	Resistance	0 ohm
L7	Self-inductance of coil 7	Inductance	1 nH
R7	Resistance of coil 7	Resistance	0 ohm
L8	Self-inductance of coil 8	Inductance	1 nH
R8	Resistance of coil 8	Resistance	0 ohm
K1_2	Coupling coefficient between coils 1 and 2		0

Name	Description	Unit Type	Default
K1_3	Coupling coefficient between coils 1 and 3		0
K1_4	Coupling coefficient between coils 1 and 4		0
K1_5	Coupling coefficient between coils 1 and 5		0
K1_6	Coupling coefficient between coils 1 and 6		0
K1_7	Coupling coefficient between coils 1 and 7		0
K1_8	Coupling coefficient between coils 1 and 8		0
K2_3	Coupling coefficient between coils 2 and 3		0
K2_4	Coupling coefficient between coils 2 and 4		0
K2_5	Coupling coefficient between coils 2 and 5		0
K2_6	Coupling coefficient between coils 2 and 6		0
K2_7	Coupling coefficient between coils 2 and 7		0
K2_8	Coupling coefficient between coils 2 and 8		0
K3_4	Coupling coefficient between coils 3 and 4		0
K3_5	Coupling coefficient between coils 3 and 5		0
K3_6	Coupling coefficient between coils 3 and 6		0
K3_7	Coupling coefficient between coils 3 and 7		0
K3_8	Coupling coefficient between coils 3 and 8		0
K4_5	Coupling coefficient between coils 4 and 5		0
K4_6	Coupling coefficient between coils 4 and 6		0
K4_7	Coupling coefficient between coils 4 and 7		0
K4_8	Coupling coefficient between coils 4 and 8		0
K5_6	Coupling coefficient between coils 5 and 6		0
K5_7	Coupling coefficient between coils 5 and 7		0
K5_8	Coupling coefficient between coils 5 and 8		0
K6_7	Coupling coefficient between coils 6 and 7		0
K6_8	Coupling coefficient between coils 6 and 8		0
K7_8	Coupling coefficient between coils 7 and 8		0

Implementation Details

Pin numbers for each coil are assigned by the convention i = coupled port and N+i= output port for each coil, N = number of coils. So for coil 1, 1 is the coupled port and 9 is the output port for 8 coils. Vci is the voltage across coil I, I=1, ...,N:

$$V_{ci} = (R_i + j\omega L_i) \cdot I_i + \sum_{j=1}^{N} j\omega M_{ij} I_j$$

$$M_{\mathrm{ij}} = K_{\mathrm{ij}} \cdot \sqrt{L_i \cdot L_j}$$

$$L_i > 0$$

$$R_i > 0$$

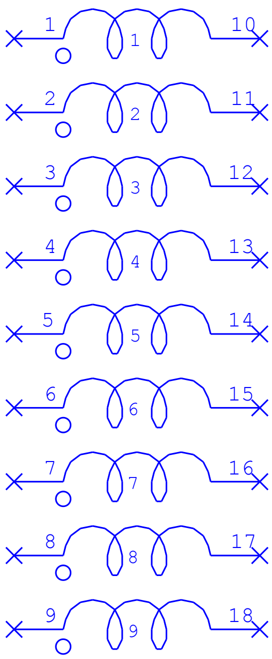
$$-1 < K_{ij} < 1$$

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Mutually Coupled Coils 9 Inductors (Closed Form): MUC9

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	M1
L1	Self-inductance of coil 1	Inductance	1 nH
R1	Resistance of coil 1	Resistance	0 ohm
L2	Self-inductance of coil 2	Inductance	1 nH
R2	Resistance of coil 2	Resistance	0 ohm
L3	Self-inductance of coil 3	Inductance	1 nH
R3	Resistance of coil 3	Resistance	0 ohm
L4	Self-inductance of coil 4	Inductance	1 nH
R4	Resistance of coil 4	Resistance	0 ohm
L5	Self-inductance of coil 5	Inductance	1 nH
R5	Resistance of coil 5	Resistance	0 ohm
L6	Self-inductance of coil 6	Inductance	1 nH
R6	Resistance of coil 6	Resistance	0 ohm
L7	Self-inductance of coil 7	Inductance	1 nH
R7	Resistance of coil 7	Resistance	0 ohm
L8	Self-inductance of coil 8	Inductance	1 nH
R8	Resistance of coil 8	Resistance	0 ohm

Name	Description	Unit Type	Default
L9	Self-inductance of coil 9	Inductance	1 nH
R9	Resistance of coil 9	Resistance	0 ohm
K1_2	Coupling coefficient between coils 1 and 2		0
K1_3	Coupling coefficient between coils 1 and 3		0
K1_4	Coupling coefficient between coils 1 and 4		0
K1_5	Coupling coefficient between coils 1 and 5		0
K1_6	Coupling coefficient between coils 1 and 6		0
K1_7	Coupling coefficient between coils 1 and 7		0
K1_8	Coupling coefficient between coils 1 and 8		0
K1_9	Coupling coefficient between coils 1 and 9		0
K2_3	Coupling coefficient between coils 2 and 3		0
K2_4	Coupling coefficient between coils 2 and 4		0
K2_5	Coupling coefficient between coils 2 and 5		0
K2_6	Coupling coefficient between coils 2 and 6		0
K2_7	Coupling coefficient between coils 2 and 7		0
K2_8	Coupling coefficient between coils 2 and 8		0
K2_9	Coupling coefficient between coils 2 and 9		0
K3_4	Coupling coefficient between coils 3 and 4		0
K3_5	Coupling coefficient between coils 3 and 5		0
K3_6	Coupling coefficient between coils 3 and 6		0
K3_7	Coupling coefficient between coils 3 and 7		0
K3_8	Coupling coefficient between coils 3 and 8		0
K3_9	Coupling coefficient between coils 3 and 9		0
K4_5	Coupling coefficient between coils 4 and 5		0
K4_6	Coupling coefficient between coils 4 and 6		0
K4_7	Coupling coefficient between coils 4 and 7		0
K4_8	Coupling coefficient between coils 4 and 8		0
K4_9	Coupling coefficient between coils 4 and 9		0
K5_6	Coupling coefficient between coils 5 and 6		0
K5_7	Coupling coefficient between coils 5 and 7		0
K5_8	Coupling coefficient between coils 5 and 8		0
K5_9	Coupling coefficient between coils 5 and 9		0
K6_7	Coupling coefficient between coils 6 and 7		0
K6_8	Coupling coefficient between coils 6 and 8		0
K6_9	Coupling coefficient between coils 6 and 9		0
K7_8	Coupling coefficient between coils 7 and 8		0
K7_9	Coupling coefficient between coils 7 and 9		0

Name	Description	Unit Type	Default
K8_9	Coupling coefficient between coils 8 and 9		0

Implementation Details

Pin numbers for each coil are assigned by the convention i = coupled port and N+i= output port for each coil, N = number of coils. So for coil 1, 1 is the coupled port and 10 is the output port for 9 coils. Vci is the voltage across coil I, I=1, ...,N:

$$V_{ci} = (R_i + j\omega L_i) \cdot I_i + \sum_{j=1}^{N} j\omega M_{ij} I_j$$

$$M_{\mathrm{ij}} = K_{\mathrm{ij}} \cdot \sqrt{L_i \cdot L_j}$$

$$L_i > 0$$

$$R_i > 0$$

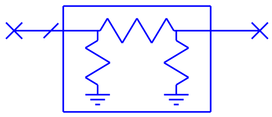
$$-1 < K_{\mathrm{ij}} < 1$$

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Resistive Pi-Pad (Closed Form): PIPAD

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	P1
Z1	Input impedance	Resistance	50 ohm
Z2	Output impedance	Resistance	50 ohm
DB	Attenuation in DB	DB	0 dB

Implementation Details

Implements an ideal attenuator. Z1 and Z2 specify the input and output resistances. If port 2 is terminated in Z2 then port 1 will be matched to Z1. Likewise if port 1 is terminated in Z1 then port 2 will be matched to Z2. Both resistances must be greater than zero.

DB specifies the attenuation in dB. The absolute value of DB is always used, and must be greater than DBmin defined below.

$$\mathrm{DB}_{\mathrm{min}} = 10 \cdot \log 10(A_{\mathrm{min}})$$

where, for

$$Z1 > Z2$$

$$A_{\mathrm{min}} = 2\frac{Z1}{Z2}\left(1 + \sqrt{1 - \frac{Z2}{Z1}}\right) - 1$$

$$Z2 > Z1$$

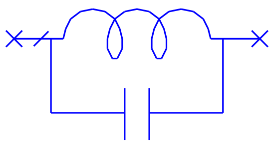
$$A_{\mathrm{min}} = 2\frac{Z2}{Z1}\left(1 + \sqrt{1 - \frac{Z1}{Z2}}\right) - 1$$

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Parallel LC (Closed Form): PLC

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Name	Text	LC1
L	Inductance	Inductance	1 nH
C	Capacitance	Capacitance	1 pF

Implementation Details

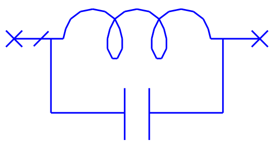
Parallel inductance and capacitance for use in high Q circuits.

Layout

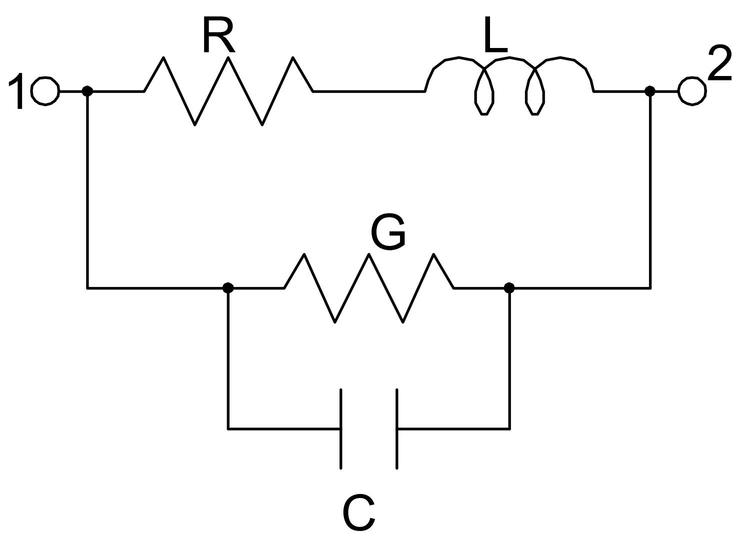
This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Parallel LC with Q (Closed Form): PLCQ

Symbol



Topology



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	LC1
C	Capacitance	Capacitance	1 pF
QC	Quality factor for capacitor		0
FQC	Reference frequency for QC	Frequency	0 GHz
ALPHC	Scaling factor for Qc		1
L	Inductance	Inductance	1 nH
QL	Quality factor for inductor		0
FQL	Reference frequency for QL	Frequency	0 GHz
ALPHL	Scaling factor for QL		1
DCMod	DC modeling	Vector text (pull-down)	Lossless

Parameter Details

DCMod. Specifies the DC behavior of the model. **Lossless** indicates that the model is modeled as an ideal capacitor in parallel with an ideal inductor at DC. **Lossy** indicates that loss is taken into account at DC.

Implementation Details

$$\mathrm{QL}(f) = Q\left(\frac{f}{\mathrm{FQL}}\right)^{\mathrm{ALPHL}}$$

$$\mathrm{QC}(f) = Q\left(\frac{f}{\mathrm{FQC}}\right)^{\mathrm{ALPHC}}$$

The admittance of the resonant circuit is given by:

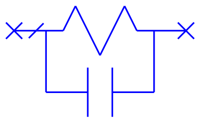
$$Y = 2\pi f C\left(\frac{1}{\mathrm{QC}(f)} + j\right) + \frac{1}{2\pi f L\left(\frac{1}{\mathrm{QL}(f)} + j\right)}$$

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Parallel RC (Closed Form): PRC

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Name	Text	RC1
R	Resistance	Resistance	1000 ohm
C	Capacitance	Capacitance	1 pF

Implementation Details

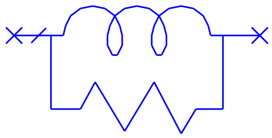
Parallel ideal resistor and capacitor.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Parallel RL (Closed Form): PRL

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Name	Text	RL1
R	Resistance	Resistance	1000 ohm
L	Inductance	Inductance	1 nH

Implementation Details

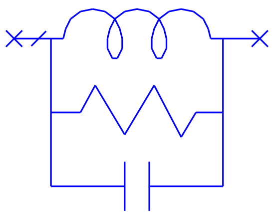
Parallel inductance and resistance for use in high Q circuits.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Parallel RLC (Closed Form): PRLC

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Name	Text	RLC1
R	Resistance	Resistance	1000 ohm
L	Inductance	Inductance	1 nH
C	Capacitance	Capacitance	1 pF

Implementation Details

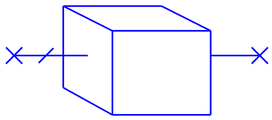
Parallel resistance, inductance and capacitance for use in high Q circuits.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

QFN MMIC Package: QFN_EM

Symbol

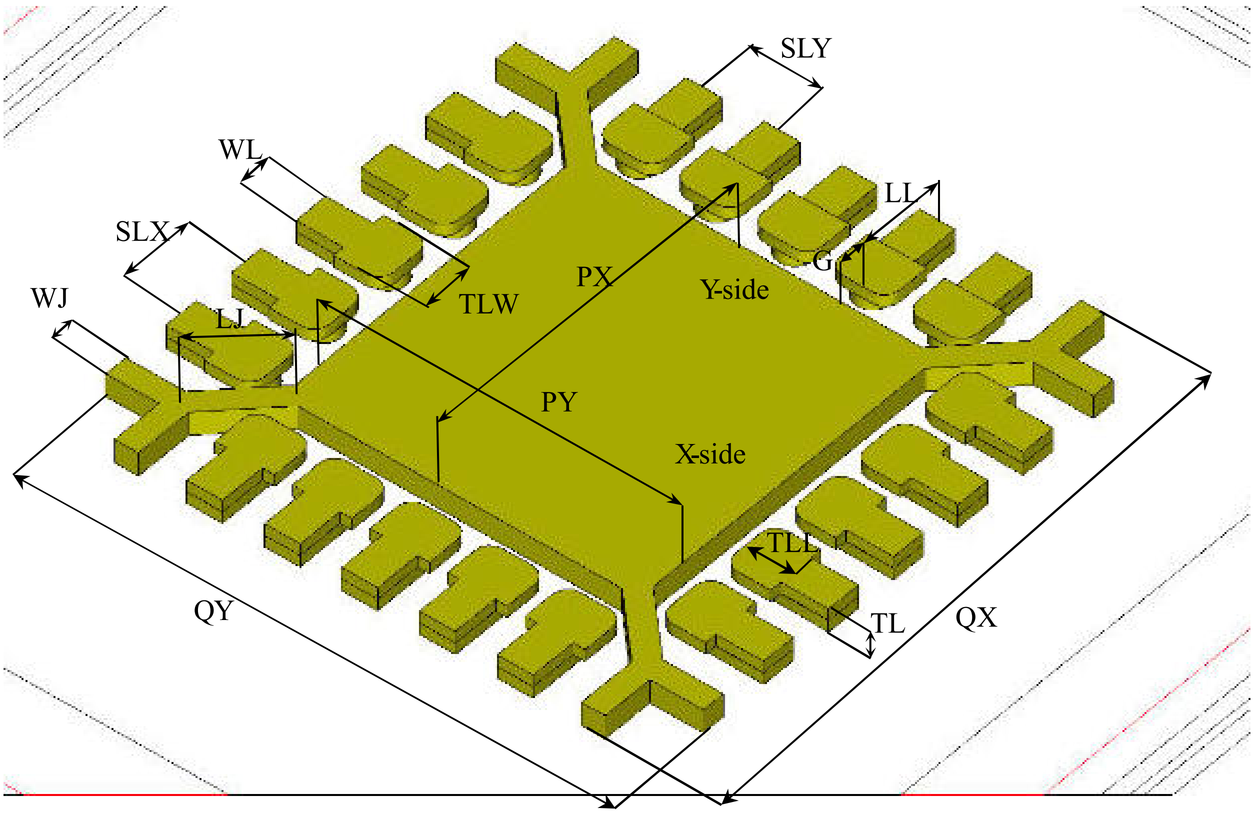


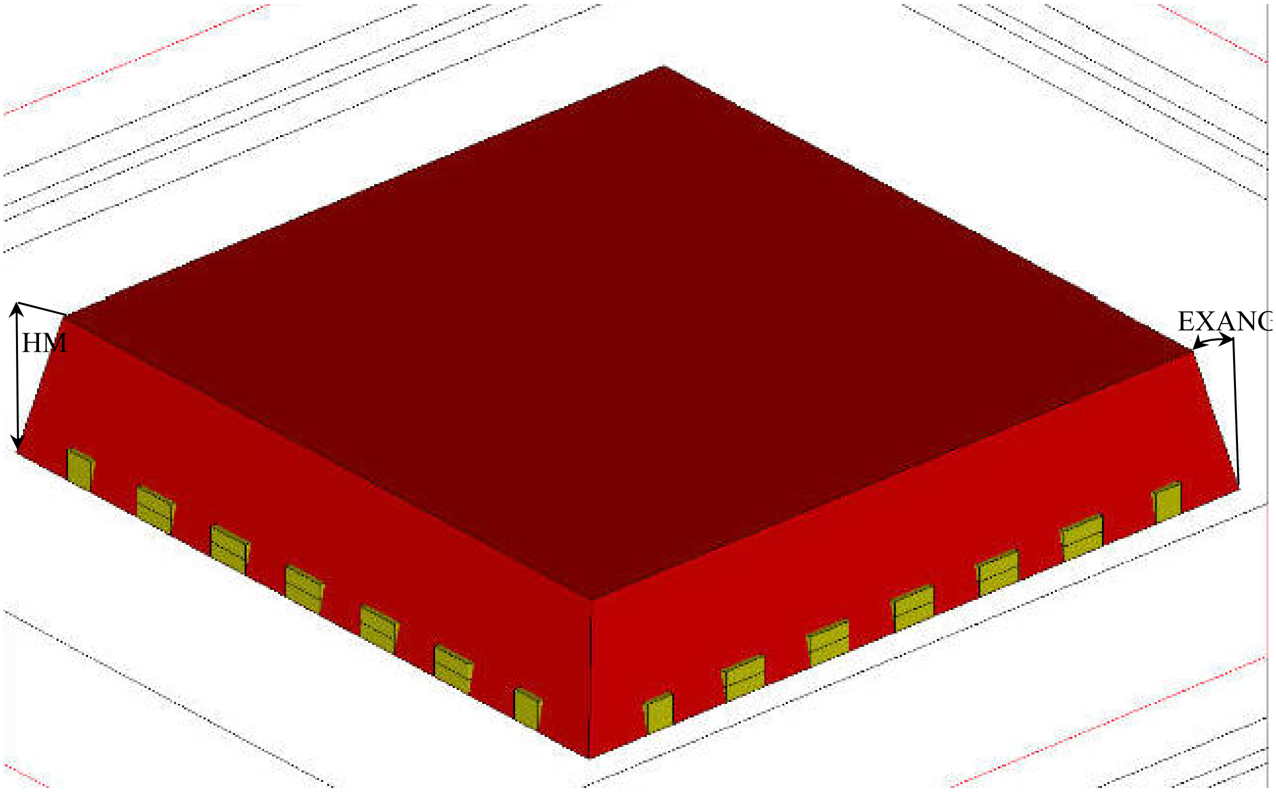
Summary

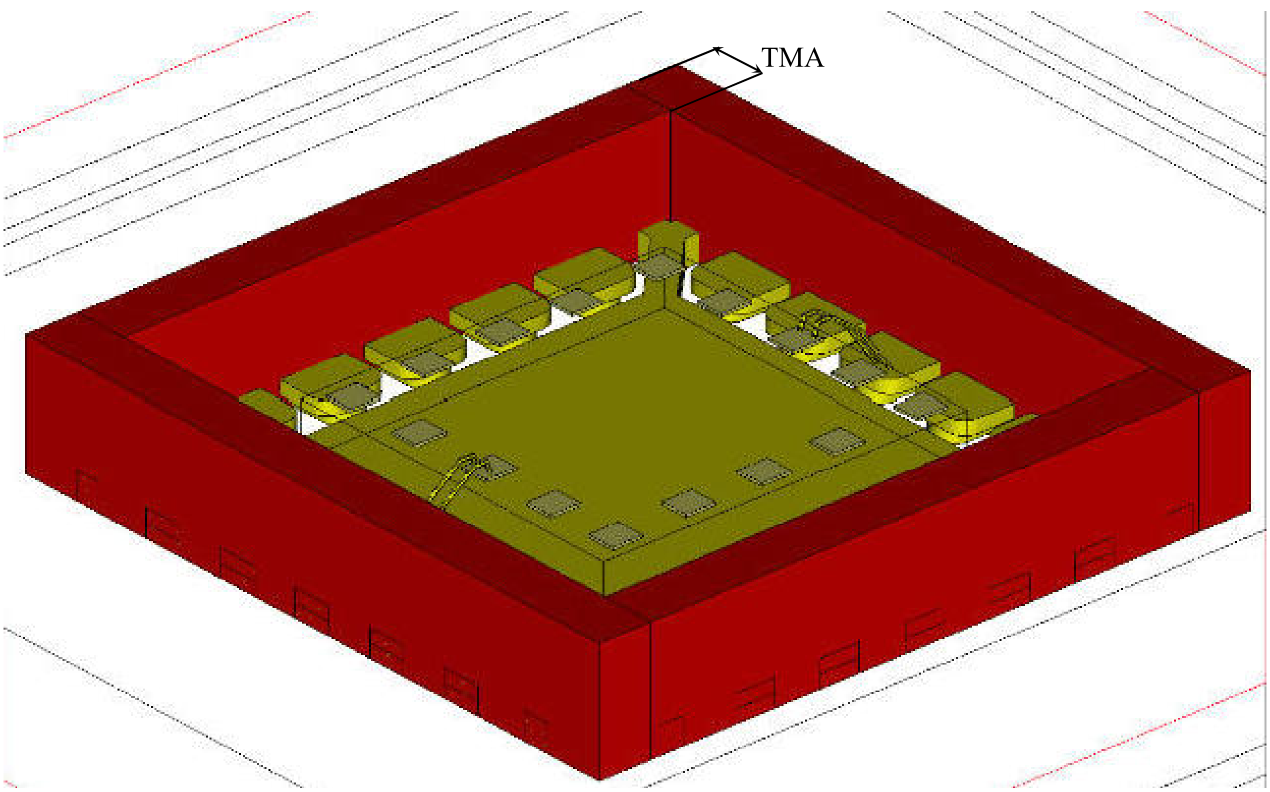
QFN_EM is a QFN MMIC package model. You can specify the number of leads, lead pitch, overmolding, the method of singulation, materials and other package parameters and properties.

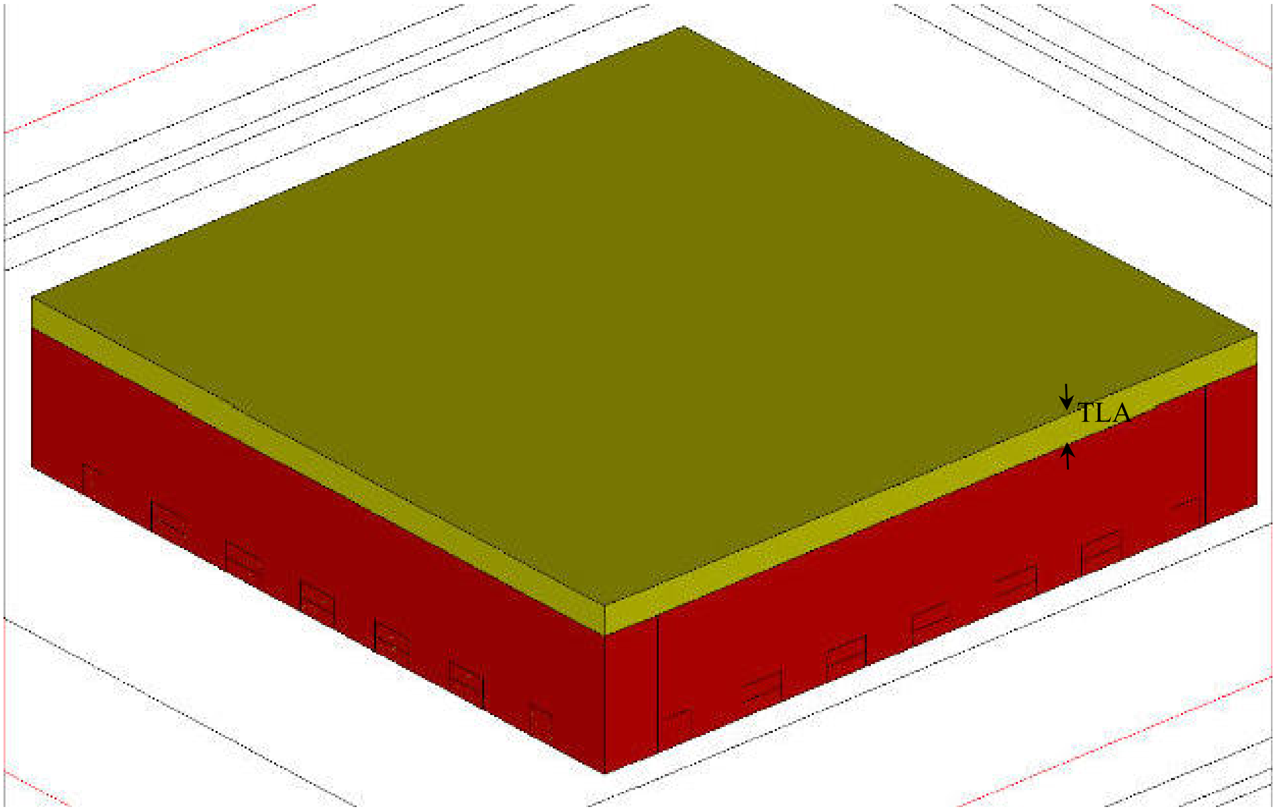
This element is for use in 3D EM documents only. It is intended to create a 3D parameterized cell for Gull-Wing lead(s) for 3D EM analysis. See "Using 3D EM Elements " for details on using 3D pCells with Analyst.

Topology




HM
EXANG


TMA



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	IC
QX	Package X size (for reference only)	Length	5 mm
QY	Package Y size (for reference only)	Length	5 mm
NLX	Number of X side leads		5
NLY	Number of Y side leads		5
SLX	Lead pitch on X side	Length	0.65 mm
SLY	Lead pitch on Y side	Length	0.65 mm
WL	Lead width	Length	0.3 mm
LL	Lead Length	Length	0.8 mm
TL	Lead and die pad metal thickness	Length	0.2 mm
TLW	Width of lead T-extension	Length	0.5 mm
TLL	length of lead T-extension	Length	0.4 mm
G	Distance between lead and die pad	Length	0.2 mm
PX	Die pad X size	Length	3 mm
PY	Die pad Y size	Length	3 mm

Name	Description	Unit Type	Default
HM	Overmold (package) height	Length	0.9 mm
TMA	Thickness of package walls (air cavity with lead option)	Length	0.9 mm
TLA	Thickness of package lid (air cavity with lead option)	Length	0.9 mm
RhoL	Leadframe metal bulk resistivity relative to gold		0.7
RhoLD	Package lead metal bulk resistivity relative to gold		1.1
ErP	Relative dielectric constant of overmold dielectric		4
TandP	Loss tangent of overmold dielectric		0.001
ErLD	Relative dielectric constant of lid dielectric		4.6
TandLD	Loss tangent of lid dielectric		0.0037
LJ	Length of corner jumper	Length	0.6 mm
WJ	Width of corner jumper	Length	0.3 mm
EXANG	Angle of punch press excision	Angle	0
METL_NAME	Leadframe metal name for 3D EM cell	Text	Copper
METL_NAME	Package lid metal name for 3D EM cell	Text	Aluminum
OMOLD_NAME	Overmold dielectric name for 3D EM cell	Text	OMold
LIDD_NAME	Lid dielectric name for 3D EM cell	Text	Glass
TestPack	Switch Full package/Lidframe only	Text	Lidframe only
Singul	Singulation method switch Saw/Punch	Text	Saw
AirCav	Switch No air cavity/Air cavity with lid/Air cavity w/o lid	Text	No air cavity
LidMtr	Switch Dielectric lid/Metal lid	Text	Dielectric lid

Parameter Details

QX, QY. Parameters QX and QY specify overall package dimensions and are only for reference. Pcell calculates the actual dimensions from other parameters:

$Q_x = P_x + 2LL + 2G$

$Q_y = P_y + 2LL + 2G$

Layout

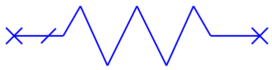
The layout for this cell has hard-coded model layers. When you first use this layout cell, a layer named "QFNPACK" is added to your drawing layer and model layer list (if they are not already there). Using the model layer mapping, you can assign these layers to draw on any drawing layer.

3D EM Layout

This element has a layout cell specifically for Analyst 3D EM layouts. See "Using 3D EM Elements " for details on using 3D pCells with Analyst.

Resistor (Closed Form): RES

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Name	Text	R1
R	Resistance	Resistance	1 ohm

Implementation Details

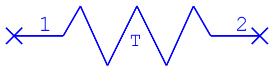
Implements an ideal resistor.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Resistance With Noise Temperature: REST

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	IN1
R	Resistance	Resistance	1 ohm
T	Temperature for noise calculation	Temperature	24 DegC

Implementation Details

The noise current density is given by:

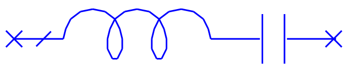
$$\frac{\mathrm{in}^2}{\Delta f} = 4 \cdot \mathrm{KT} \cdot \frac{1}{R}$$

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Series LC (Closed Form): SLC

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Name	Text	LC1
L	Inductance	Inductance	1 nH
C	Capacitance	Capacitance	1 pF

Implementation Details

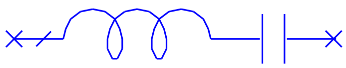
Series inductance and capacitance for use in high Q circuits.

Layout

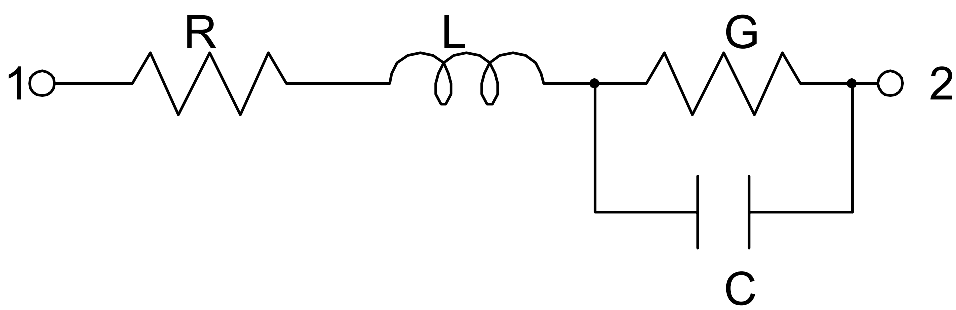
This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

Series LC with Q (Closed Form): SLCQ

Symbol



Topology



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	LC1
C	Capacitance	Capacitance	1 pF
QC	Quality factor for capacitor		0
FQC	Reference frequency for QC	Frequency	0 GHz
ALPHC	Scaling factor for Qc		1
L	Inductance	Inductance	1 nH
QL	Quality factor for inductor		0
FQL	Reference frequency for QL	Frequency	0 GHz
ALPHL	Scaling factor for QL		1
DCMod	DC modeling	Vector text (pull-down)	Lossless

Parameter Details

DCMod. Specifies the DC behavior of the model. **Lossless** indicates that the model is modeled as an ideal capacitor in series with an ideal inductor at DC. **Lossy** indicates that loss is taken into account at DC.

Implementation Details

$$\mathrm{QL}(f) = Q\left(\frac{f}{\mathrm{FQL}}\right)^{\mathrm{ALPHL}}$$

$$\mathrm{QC}(f) = Q\left(\frac{f}{\mathrm{FQC}}\right)^{\mathrm{ALPHC}}$$

The impedance of the resonant circuit is given by:

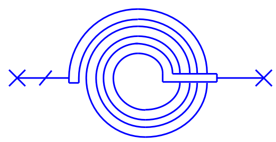
$$Z = 2\pi f L\left(\frac{1}{\mathrm{QL}(f)} + j\right) + \frac{1}{2\pi f C\left(\frac{1}{\mathrm{QC}(f)} + j\right)}$$

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Circular Wire Solenoid: SOLIND

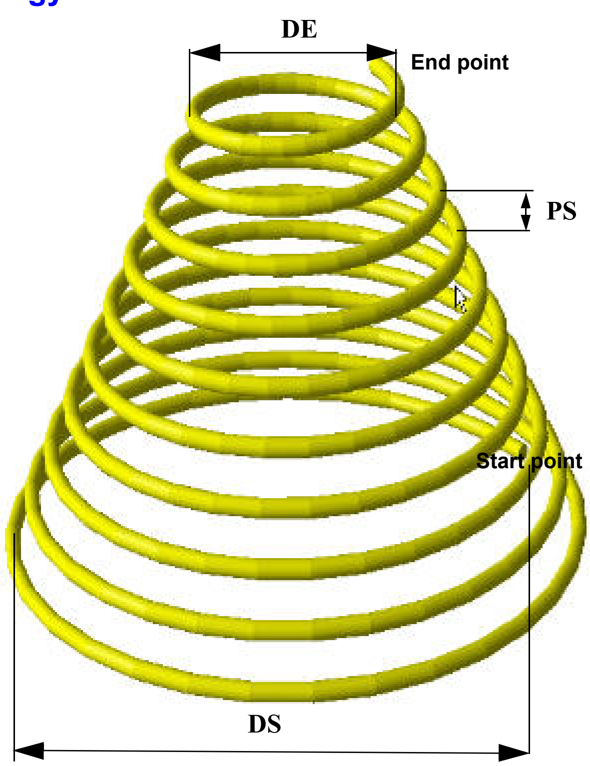
Symbol

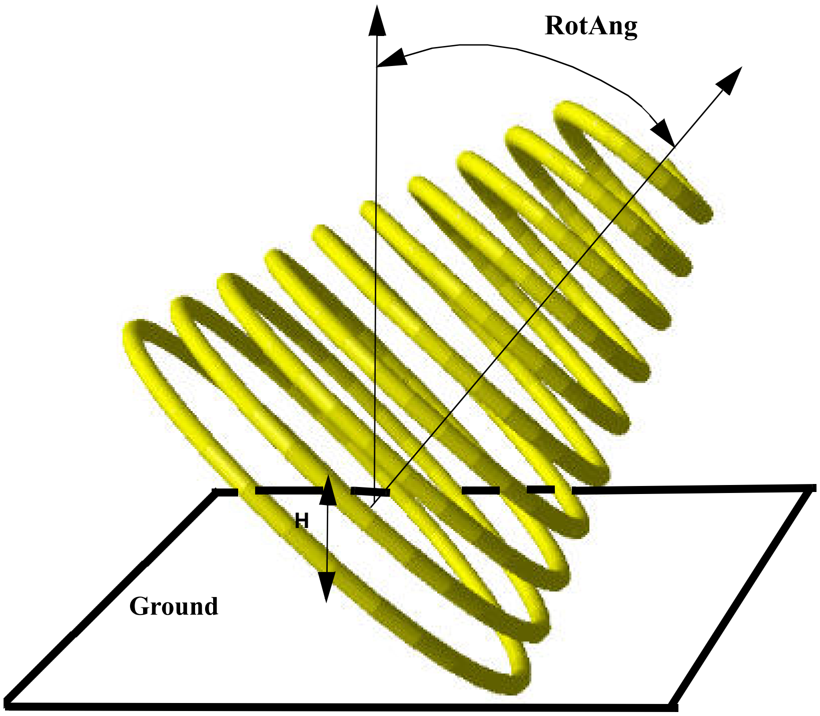


Summary

This circuit component models a circular solenoid (optionally tapered) made of the wire of a cylindrical cross-section situated either in free space (air) or above infinite perfectly conducting ground. The modeling approach is based on evaluation of self and mutual inductances, capacitances and resistances between all spiral turns.

Topology





Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	IND1
NT	Number of turns (>=1)		10
Dia	Wire diameter	Length	25 um
DS	Start diameter of solenoid (measured between wire centerlines)	Length	500 um
DE	End diameter of solenoid (measured between wire centerlines)	Length	500 um
PS	Pitch of solenoid (measured between wire centerlines along solenoid axis)	Length	100 um
H	Elevation of solenoid above optional ground	Length	100 um
RotAng	Rotation angle (meridional tilt) of solenoid axis relative to normal to ground	Degrees	0
Rho	Wire metal bulk resistivity relative to gold		1
Gnd	Switch "No ground/Ground"		No ground
*Nseg	Number of linear segment per turn		6

** indicates a secondary parameter*

Parameter Details

NT. The number of spiral turns forming the solenoid.

Dia. The diameter of solenoid wire.

DS, DE. Respectively, the start and end diameters of a tapered solenoid. Model checks if DE<=DS and issues error message if this condition does not hold.

PS. The axial **(measured along solenoid axis)** distance between wire centerlines of two adjacent spiral turns (pitch).

H. Elevation of start point of solenoid (see Topology) above ground. If Gnd="No ground" this parameter is not used

RotAng. Tilt of solenoid axis (and elevation H) make sense only if Gnd is set to "Ground". If Gnd="No ground" RotAng is not used. Model keeps track of position of all conducting parts and issues an error if combination of specified H, RotAng, DS and Dia makes solenoid to touch ground. Changes made to solenoid angular position reflect on respective image on 3D layout view.

Nseg. The number of linear wire segments a model uses to approximate a single spiral turn (see "Implementation Details"). Nseg must be greater than or equal to 6.

Parameter Restrictions and Recommendations

1. NT should be greater or equal to 1.
2. Model checks that Nseg≥6.
3. Model checks if DE≤DS.

Implementation Details

Each spiral turn is broken into Nseg linear wire segments and the model evaluates self and mutual RLC parameters of all segments (including the image wire segments if Gnd is set to "Ground").This model controls the length of each segment and issues an error message if the length of any segment exceeds the wire diameter Dia. This model accounts for a skin effect and partly for radiation loss.

Recommendations for Use

Note that port 1 is assigned to the start point of a spiral and Port 2 is assigned to end point of a spiral (see positions of start and end points on Topology).

This model predicts the frequency behavior of the solenoid below and in the vicinity of the first self-resonance frequency.

User discretion is advised if model issues a warning about segment length being too small. SOLIND assumes that each segment is represented by infinitely thin filament which means Lseg >>Dia. Thus, very short (relative to wire diameter) segments may cause additional errors. Model issues warning message if Lseg<6*Dia. This message contains segment number. All segments with **numbers greater than violating** are also in violation. In case DE<DS segments in violation often belong to a few top turns. Thus user gets an approximate estimation of percentage of segments with reduced accuracy.

Table of conversion AWG (American Wire Gauge) to millimeters (for those who prefer wire gauge for denoting wire diameter).

AWG	Wire diameter (mm)	AWG	Wire diameter (mm)	AWG	Wire diameter (mm)
12	2.053	25	0.4547	38	0.1007
13	1.828	26	0.4049	39	0.08969

AWG	Wire diameter (mm)	AWG	Wire diameter (mm)	AWG	Wire diameter (mm)
14	1.628	27	0.3606	40	0.07987
15	1.450	28	0.3211	41	0.07112
16	1.291	29	0.2859	42	0.06334
17	1.150	30	0.2546	43	0.05641
18	1.024	31	0.2268	44	0.05023
19	0.9116	32	0.2019	45	0.04473
20	0.8118	33	0.1798	46	0.03983
21	0.7229	34	0.1601	47	0.03547
22	0.6438	35	0.1426	48	0.03160
23	0.5733	36	0.1270	49	0.02813
24	0.5106	37	0.1131	50	0.02505

Layout

The layout for this cell has hard-coded model layers. When you first use this layout cell, a layer named "Solenoid" is added to your drawing layer and model layer list (if they are not already there). Using the model layer mapping, you can assign these layers to draw on any drawing layer. The 2D layout displays a circle the size of the DS parameter plus the Dia size. The 3D layout displays the shapes as a solenoid.

This model has a layout only parameter that controls how the layout cell draws. You access the parameters by selecting the item in the layout, right-clicking and choosing **Shape Properties** to display the Cell Options dialog box, then clicking the **Parameters** tab.

Name	Value	Units	Description
NSeg	32	Integer	Number of linear segments approximating one spiral turn.

Series RC (Closed Form): SRC

Symbol

Parameters

Name	Description	Unit Type	Default
ID	Name	Text	RC1
R	Resistance	Resistance	1 ohm
C	Capacitance	Capacitance	1 pF

Implementation Details

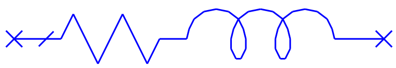
Series ideal resistor and capacitor. This element is preferable to the use of separate elements, as it eliminates a node.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Series RL (Closed Form): SRL

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Name	Text	RL1
R	Resistance	Resistance	1 ohm
L	Inductance	Inductance	1 nH

Implementation Details

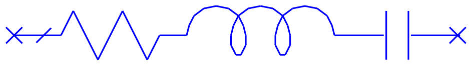
Series inductance and resistance for use in high Q circuits.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

Series RLC (Closed Form): SRLC

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Name	Text	RLC1
R	Resistance	Resistance	1 ohm
L	Inductance	Inductance	1 nH
C	Capacitance	Capacitance	1 pF

Implementation Details

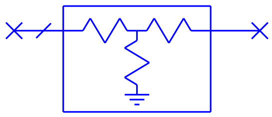
Series inductance, resistance and capacitance for use in high Q circuits.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Resistive T-Pad (Closed Form): TEEPAD

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	P1
Z1	Input impedance	Resistance	50 ohm
Z2	Output impedance	Resistance	50 ohm
DB	Attenuation in DB	DB	0 dB

Implementation Details

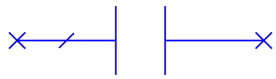
Implements an ideal attenuator in the same manner as PIPAD.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Thin Film Capacitor (Closed Form): TFC

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Element Name	Text	V1
W	Conductor width	Length	20 um
L	Conductor length	Length	20 um
T	Dielectric thickness	Length	1 um
ER	Relative dielectric constant		1
RHO	Metal bulk resistivity normalized to gold (NOT USED IN THIS MODEL)		0
TAND	Dielectric loss tangent		0

Implementation Details

This component models a thin film capacitor. This model accounts for fringing capacitance. Current implementation of TFC ignores parameter RHO.

Layout

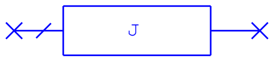
The TFC layout object obtains its drawing layer information from the LPF file that is loaded for the project. The drawing layers are defined within a block that begins with $CAP_DEFINE_BEGIN and $CAP_DEFINE_END, respectively (see "The Layout Process File (LPF)" for more information on working with LPF files). If this block is not defined, the capacitor is drawn on the Default layer (if the Default layer is not defined, it is drawn on the first layer that is defined in Layer Setup).

References

[1] I. Hurt, "A Computer-Aided Design for Hybrid Circuits," IEEE Transactions on Components, Hybrids, and Manufacturing Technology, vol.CHMT-3, Feb.1988, pp. 294-304.

Idealized Admittance Inverter: YINV

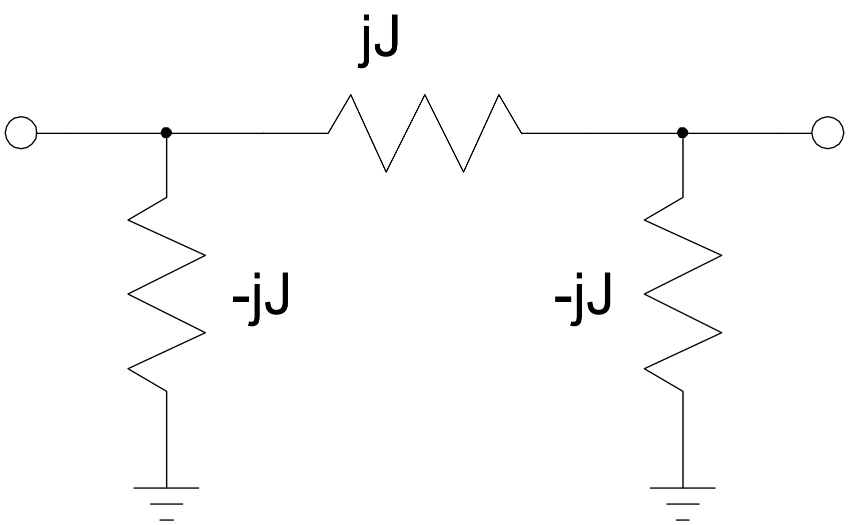
Symbol



Summary

YINV represents an idealized admittance J-inverter intended for use in filter synthesis.

Equivalent Circuit



Parameters

Name	Description	Unit Type	Default
ID	Name	Text	YI1
J	Inverter parameter	Conductance	0.02 S

Parameter Details

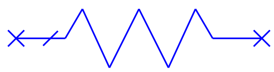
J. Parameter J has a dimension of conductance and represents an imaginary part of pure reactive frequency-independent admittances making a PI-circuit (see "Equivalent Circuit" where j is an imaginary unit).

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Complex Frequency Dependent Resistor (Closed Form): ZFREQ

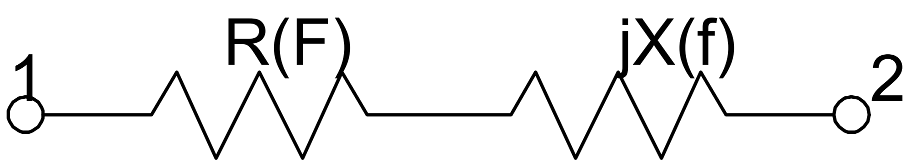
Symbol



Summary

ZFREQ provides computation of circuit parameters for a general complex resistor whose behavior in the frequency domain is governed by a user-supplied table of real part R and imaginary part X.

Equivalent Circuit



Parameters

Name	Description	Unit Type	Default
ID	Name	Text	Z1
F	Vector of Frequencies at which R, X are specified	Frequency	1 GHz
R	Vector of R	Resistance	50
X	Vector of X	Resistance	0

Parameter Details

F. Vector of frequencies at which R and X parameters are specified. Frequencies must be sequential and specified in ascending order.

R. Vector of active (real) resistance (see the "Equivalent Circuit" section) specified in project resistance units per meter. You must specify each vector entry at the corresponding frequency entry from frequency vector F.

X. Vector of reactive (imaginary) resistance (see the "Equivalent Circuit" section) specified in project resistance units per meter. Each vector entry must be specified at the corresponding frequency entry from frequency vector F.

Parameter Restrictions and Recommendations

1. The sizes of vector parameters R and X must be equal to the size of frequency vector F.
2. To provide sufficient accuracy of interpolation (see the "Implementation Details" section) parameters F, R, and X must contain a reasonable number of data points (typically >=10).
3. You can specify a vector in three ways:

- Entered as a right-side value of a model parameter, for example R={100,102,110,113,120}

- Specified elsewhere in an equation
- Specified in a column or row of a text file. This provides a convenient and flexible method of specification of R and X parameters in a single location. You can create, for example, a *Load.txt* file containing space separated columns of R and X. The first column must represent frequency in project units (note that changing the project default frequency units demands manual scaling of frequencies in this file). Import or link this file to your project and name it, for example, ZFREQ1. Now you can specify parameter F as F = Col(datafile("ZFREQ1"),1). The values of vector F are copied to the model from column 1 of the *Load.txt* file, which was imported under the name ZFREQ1. If you prefer to arrange your data by row, use F = Row(datafile("ZFREQ1"),1).

Implementation Details

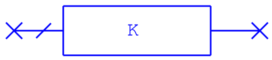
Model implementation is based on linear interpolation of R and X parameters at each project evaluation frequency. Interpolation uses user-supplied via parameters look-out tables. If project operational frequency is out of range of frequencies in F, then R and X parameters are extrapolated as constant values equal to the first/last entries of corresponding vectors.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Idealized Impedance Inverter: ZINV

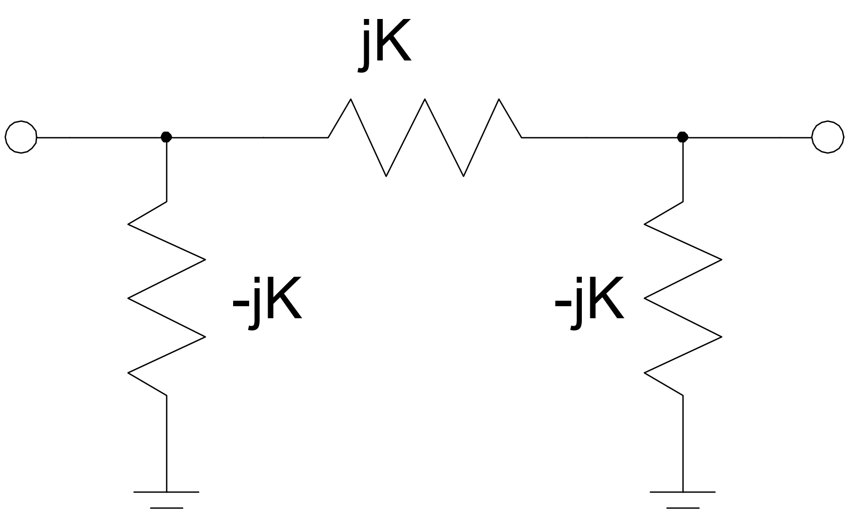
Symbol



Summary

ZINV represents an idealized impedance K-inverter intended for use in filter synthesis.

Equivalent Circuit



Parameters

Name	Description	Unit Type	Default
ID	Name	Text	ZI1
K	Inverter parameter.	Resistance	50 ohm

Parameter Details

K. Parameter K has a dimension of impedance and represents an imaginary part of pure reactive frequency-independent impedances making a PI-circuit (see "Equivalent Circuit" where j is an imaginary unit).

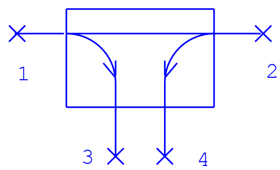
Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Measurement Devices

Dual Power Sampler (Closed Form): DPWRSMP

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Name	Text	U1
R	Port resistance		50

Implementation Details

Simulates a frequency-independent, lossless, directional coupler. The 4-port S parameters are given below.

$$S = \begin{bmatrix} 0 & 1 & 0 & 0 \\ 1 & 0 & 0 & 0 \\ 1 & 0 & 0 & 0 \\ 0 & 1 & 0 & 0 \end{bmatrix}$$

Inserting this element into a circuit and connecting resistors to port 3 and port 4 provides a way to sample voltage or power. The resistors should have the same value as R.

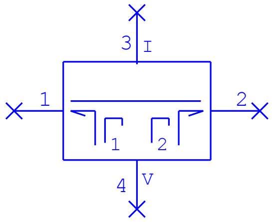
Layout

This element uses a special layout cell for a short circuit. The layout cell allows the elements connected on either side of the element to look through this element. For example, when a MLIN is hooked to a TFCM (capacitor) model and the project has bridge code configured, the line draws the proper interconnect between the line and the cap. If this element is placed between the line and the cap, the layout still draws the same.

You typically do not assign artwork cells to these items.

(Obsolete) Gamma Probe (Closed Form): GPROBE

Symbol



Summary

This element is OBSOLETE and is replaced by the Gamma Probe (Closed Form) (GPROBE2).

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	XF1
Rsense	Current sense resistance		0.0001

Parameter Details

Rsense. This value must be nonzero, however it can be made small enough such that the probe circuit may be considered lossless.

Implementation Details

The gamma probe (GPROBE) element is an insertable element that is placed on the input and output pins of active devices. When placed, it is invisible to the circuit so that the through-signal path is not disturbed. This probe calculates the two reflection coefficients for the node into which it is inserted (Gamma 1 looks to the left and Gamma 2 looks to the right). The reflection coefficients themselves can be plotted, but the main thing to view is the STAB_GP measurement, which is the product of the two reflection coefficients. If this product is greater than 1 at any frequency or load (or source pull point), then the device is unstable.

The stability measurements STAB_GP and STAB_GPN and the internal gamma measurements GAM1_GP and GAM2_GP require that the excitation ports (connected to nodes 1 and 2) are specified as part of the measurement. You must also specify the ports connected to the voltage and current sampling terminals of the GPROBE element for these measurements (the measurement requires that ports be placed on the V and I nodes of the GPROBE element).

You should use GPROBE in circuits without feedback across the active devices. If this feedback is present, GPROBEM may provide better accuracy. See "Performing Internal Stability Analysis" for more information on the operation of GPROBE. Implementation details of GPROBEM (GPROBE is analogous) can be found in the document listed in "References" (see Fig.3); this document is available for download from the NI AWR Knowledge Base (search for GPROBEM).

Layout

This element uses a special layout cell for a short circuit. The layout cell allows the elements connected on either side of the element to look through this element. For example, when a MLIN is hooked to a TFCM (capacitor) model and the project has bridge code configured, the line draws the proper interconnect between the line and the cap. If this element is placed between the line and the cap, the layout still draws the same.

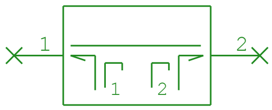
You typically do not assign artwork cells to these items.

References

[1] C. Campbell and S. Brown "Modified S-Probe Circuit Element for Stability Analysis" (white paper)

Gamma Probe (Closed Form): GPROBE2

Symbol



Summary

GPROBE2 establishes a reference plane where it is inserted into the signal path, and offers a means of determining the reflection coefficients required for internal stability measurements. It otherwise acts like a short circuit, so it does not affect the behavior or layout of the circuit in any way.

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	GP1

Implementation Details

The gamma probe (GPROBE2) element is an insertable element that should be connected in series with the input and output pins of active devices in multi-stage circuits. When connected, it is invisible to the circuit so that the through-signal path is not disturbed. This probe calculates the two reflection coefficients at the reference plane where it is inserted ("looking out" of each of its own nodes). The reflection coefficients themselves can be plotted, but the primary measurement to view is the STAB_GP2, which is a function of the product of the two reflection coefficients. If this measurement is greater than 1 at any frequency or load (or source pull point), then the circuit may be unstable. See "Performing Internal Stability Analysis" for more information on the operation and use of GPROBE2, and internal stability analysis in general.

NOTE: The GPROBE and GPROBEM elements are now obsolete and should be replaced with the GPROBE2 element. The new approach is much more efficient as no new ports are needed in the schematic.

Layout

This element uses a special layout cell for a short circuit. The layout cell allows the elements connected on either side of the element to look through this element. For example, when a MLIN is hooked to a TFCM (capacitor) model and the project has bridge code configured, the line draws the proper interconnect between the line and the cap. If this element is placed between the line and the cap, the layout still draws the same.

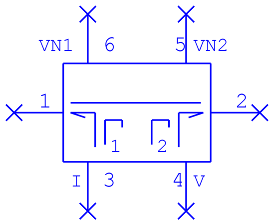
You typically do not assign artwork cells to these items.

References

[1] C. Campbell and S. Brown "Modified S-Probe Circuit Element for Stability Analysis" (white paper)

(Obsolete) Modified Gamma Probe (Closed Form): GPROBEM

Symbol



Summary

This element is OBSOLETE and is replaced by the Gamma Probe (Closed Form) (GPROBE2). The GPROBEM element is similar to the GPROBE element. However, the GPROBE element may give incorrect results in certain cases. This modified element attempts to correct these errors.

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	GP2
Rsense	Current sense resistance	Resistance	0.0001
Rlarge	Open circuit resistance	Resistance	1e+009

Parameter Details

Rsense. This value must be nonzero, however it can be made small enough such that the probe circuit may be considered lossless.

Rlarge. This value can be made arbitrarily large such that ports attached to nodes VN1, VN2 do not load the probe circuit.

Implementation Details

GPROBEM is used to measure the internal reflection coefficients at the reference plane. The modified gamma probe GPROBEM element is an insertable element that is placed on the input and output pins of active devices. When placed, GPROBEM is invisible to the circuit so that the through-signal path is not disturbed. This probe calculates the two reflection coefficients Gamma 1 and Gamma 2 for the node into which it is inserted (Gamma 1 looks to the left and Gamma 2 looks to the right). The reflection coefficients themselves can be plotted, but the main thing to look at is the STAB_GPM measurement, which is the product of the two reflection coefficients. If this product is greater than 1 at any frequency or load (or source pull point) then the device is unstable.

The stability measurements STAB_GPM and STABN_GPM require that the ports connected to the voltage and current sampling terminals of the GPROBEM element are specified for these measurements (the measurement requires that ports be placed on the V and I nodes of the GPROBEM). The same is true for VN1 and VN2 nodes (voltage sampling terminals respectively on the side of node 1 and the side of node 2).

GPROBEM provides better accuracy than GPROBE in circuits with feedback across the active devices that are being tested. See "Performing Internal Stability Analysis" for more information on the operation of GPROBE (the operation of GPROBEM is similar). Implementation details can be found in the document listed in "References" (see Fig.3); this document is available for download from the NI AWR Knowledge Base (search for GPROBEM).

Layout

This element uses a special layout cell for a short circuit. The layout cell allows the elements connected on either side of the element to look through this element. For example, when a MLIN is hooked to a TFCM (capacitor) model and the project has bridge code configured, the line draws the proper interconnect between the line and the cap. If this element is placed between the line and the cap, the layout still draws the same.

You typically do not assign artwork cells to these items.

References

[1] C. Campbell and S. Brown "Modified S-Probe Circuit Element for Stability Analysis" (white paper)

Current Meter: I_METER

Symbol

Parameters

Name	Description	Unit Type	Default
ID	Current meter name	Text	AMP1

Implementation Details

When this current meter is inserted into a circuit path it measures the total current through the meter which is a sum of the DC, and forward- and reverse-traveling current. You can use I_METER with Itime and Iharm measurements (see “Time Domain Current: Itime ” and “Frequency Domain Current: Iharm ”), as well as other Nonlinear Current measurements.

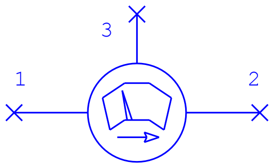
Layout

This element uses a special layout cell for a short circuit. The layout cell allows the elements connected on either side of the element to look through this element. For example, when a MLIN is hooked to a TFCM (capacitor) model and the project has bridge code configured, the line draws the proper interconnect between the line and the cap. If this element is placed between the line and the cap, the layout still draws the same.

You typically do not assign artwork cells to these items.

Current Meter: I_METER2

Symbol



Summary

I_METER2 measures the current through pins 1 and 2 and sets the voltage of the 3rd pin equal to the measured current. I_METER2 is equivalent to an ideal current-controlled voltage source with zero input and output resistances and unity gain. This element is useful in transient co-simulation that is modeled in VSS using the NL_S_ENV system block. The output signals of the NL_S_ENV block are the port voltages of the selected ports in the circuit schematic. To make a circuit schematic current available as a signal in the system simulation, it can be measured using I_METER2, and the 3rd pin should be connected to an output port. The port impedance does not load the circuit being measured.

Parameters

Name	Description	Unit Type	Default
ID	Current meter name	Text	IM1

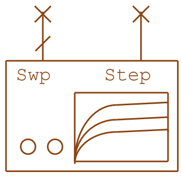
Layout

This element uses a special layout cell for a short circuit. The layout cell allows the elements connected on either side of the element to look through this element. For example, when a MLIN is hooked to a TFCM (capacitor) model and the project has bridge code configured, the line draws the proper interconnect between the line and the cap. If this element is placed between the line and the cap, the layout still draws the same.

You typically do not assign artwork cells to these items.

DC Current/Voltage Curve Tracer: IVCURVE

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	IV1
VSWEEP_start	Swept voltage start value	Voltage	0 V
VSWEEP_stop	Swept voltage stop value	Voltage	4 V
VSWEEP_step	Swept voltage step value	Voltage	1 V
VSTEP_start	Stepped voltage start value	Voltage	0 V
VSTEP_stop	Stepped voltage stop value	Voltage	4 V
VSTEP_step	Stepped voltage step value	Voltage	1 V

Implementation Details

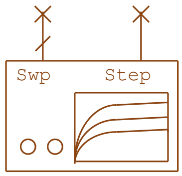
Produces a swept voltage for I-V curve trace measurements.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

DC Current Driven Curve Tracer: IVCURVEI

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	IV1
VSWEEP_start	Swept voltage start value	Voltage	0 V
VSWEEP_stop	Swept voltage stop value	Voltage	4 V
VSWEEP_step	Swept voltage step value	Voltage	1 V
ISTEP_start	Stepped current start value	Current	0 mA
ISTEP_stop	Stepped current stop value	Current	10 mA
ISTEP_step	Stepped current step value	Current	2 mA

Implementation Details

Produces a swept voltage while stepping the current. This element is useful for I-V characterization of current-controlled devices such as bipolar transistors.

AWR simulators add some series resistance to this element during simulation. This may affect results if injecting current into a large resistance. You can change the value of the resistance by choosing **Options > Default Circuit Options** to display the Circuit Options dialog box, then clicking the **AWR Sim** tab. In the Convergence Aids section, edit the **Series source resistance** value.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

Measurement Probe: M_PROBE

Symbol

Summary

M_PROBE identifies an element terminal for voltage and current measurements of all kinds. You must place it on an element terminal-- it does not measure voltage in the middle of a wire. The advantage of using M_PROBE is that after simulation, you can drag it around a schematic to display the same measurement at different element terminals, without editing a measurement or requiring a new simulation. To move the M_PROBE without dragging, right-click it and choose Dynamic Probe, then click any node or terminal to move it there, or double-click on a SUBCKT block to open the lower-level circuit and click to place the probe there. Press the **Esc** key to exit dynamic mode.

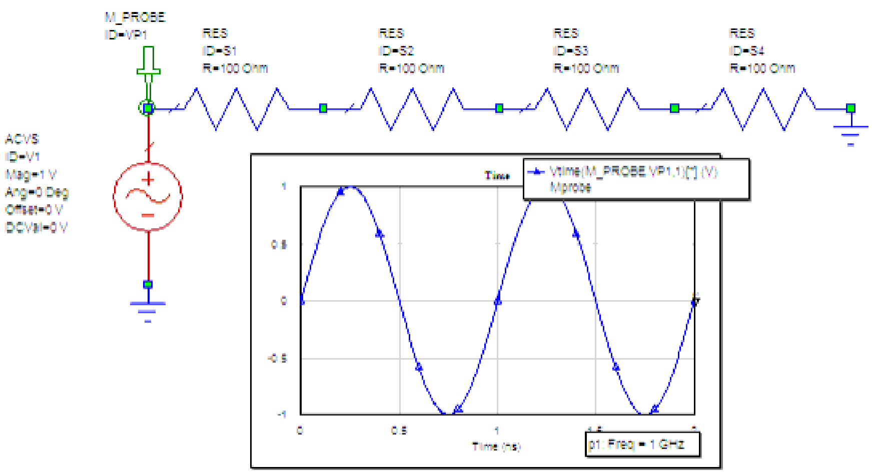
Some measurements such as VTimeD can use two M_PROBEs. To make both of the probes dynamic at the same time, select one probe, **Shift**-click the next one, right-click and choose Dynamic Probe. The first probe moves to the position of the mouse click and the second probe moves to the same relative location. If the second probe is not placed where you want it, **Shift**-click at the desired location.

Parameters

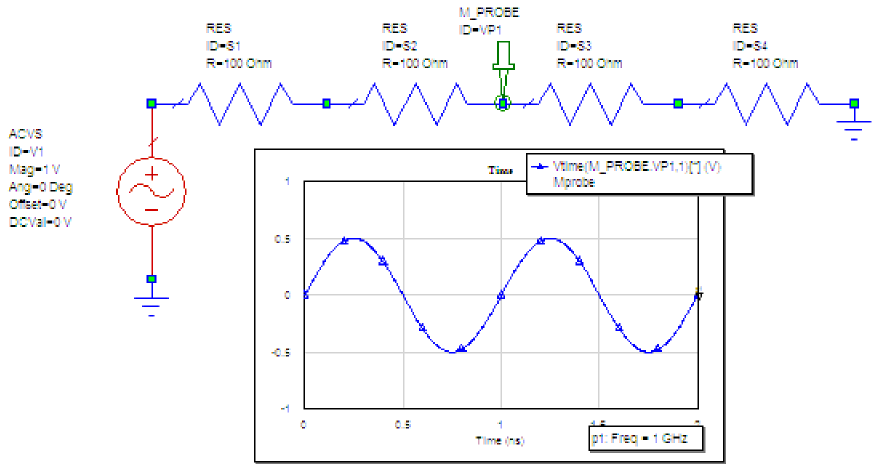
Name	Description	Unit Type	Default
ID	Measurement Probe name	Text	VP1

Implementation Details

In the following schematic, the M_PROBE is at the input voltage source.



Click and drag the M_PROBE to move it to a new location to update graphs.



In current measurements, current flowing into the selected element terminal is positive, unless the element is a voltage or current source.

If you set transient simulation options to save results only at meters, the M_PROBE is NOT included in this list since you can change its location without requiring a new simulation.

Layout

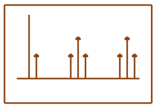
This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Recommendations for Use

In circuit schematics, do not use "pin-to-pin" connections at element terminals where current is measured. In the previous example, you should space the elements apart and connect them by wires for current measurements. The M_PROBE current measurement are ambiguous if the desired element terminal is at the same location as one or more others.

Analyze Nonlinear Noise: NLNOISE

Symbol



Summary

NLNOISE instructs the simulator to compute noise in a nonlinear circuit following large-signal analysis. See "Noise Analysis" for noise analysis details.

Parameters

Name	Description	Unit Type	Default
ID	Noise control name		NS1
PortTo	Output port		2
PortFrom	Input port		1
NFstart	Noise freq. start	Frequency	0.1 GHz
NFend	Noise freq. end	Frequency	0.2 GHz
NFsteps	Number of points in noise freq. sweep		5
SwpType	Sweep type		LINEAR
*LSTone	Tone index of large signal excitation		{1}
*SSTone	Tone index of noise signal excitation		2
*OverSamp	Sampling factor		4
*SparseTol	Drop tolerance for matrix sparcification		0.001

** indicates a secondary parameter*

Parameter Details

PortTo and **PortFrom.** Noise figure is computed with respect to these ports. The noise spectral density measurement computes noise at PortTo.

NFstart, **NFend**, and **NFsteps.** Specify the sideband frequencies, relative to the large signal harmonics, at which to measure noise. See the following figure.

SwpType. Determines the spacing between the frequency steps. Allowable sweep types are linear (LINEAR) and logarithmic (LOG).

LSTone. Lists the large signal tone(s) in the harmonic balance simulation whose sideband noise is to be taken into account. The value of this parameter is either "**{1}**" or "**{1,2}**". If the harmonic balance simulation includes low magnitude tone(s) like the RF input to a mixer, then a separate test bench schematic should be used for nonlinear noise analysis, in which the low- power harmonic balance tone(s) are removed. For more details, see "Noise Analysis".

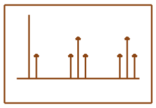
SSTone. Assigns a tone number to the frequency list specified by NFstart, NFend, and NFsteps. This should be the largest tone number in LSTone plus 1.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Analyze Nonlinear Stability Properties: NLSTABILITY

Symbol



Summary

NLSTABILITY instructs the simulator to analyze stability in a nonlinear circuit following large-signal analysis. See "Nonlinear Internal Stability Analysis" for nonlinear stability analysis details.

Parameters

Name	Description	Unit Type	Default
ID	Stability control name		ST1
Fstart	Small-signal frequency start	Frequency	0.1 GHz
Fend	Small-signal frequency end	Frequency	0.2 GHz
Fsteps	Number of points in small-signal frequency sweep		5
SwpType	Sweep type		LINEAR

Parameter Details

Fstart and **Fend.** Define the small-signal frequency range.

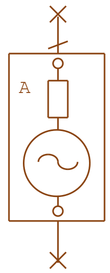
SwpType. Allowable sweep types are linear (LINEAR) and logarithmic (LOG).

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Oscillator Analysis Probe: OSCAPROBE

Symbol



Summary

OSCAPROBE initiates a large-signal oscillator simulation. See "Nonlinear Oscillator Analysis" for the recommended placement of OSCAPROBE and for details of oscillator analysis.

Equivalent Circuit

OSCAPROBE is an ideal source in series with an ideal impedance element. The impedance presents an open circuit at all frequencies other than the fundamental frequency of oscillation.

Parameters

Name	Description	Unit Type	Default
ID	Element ID		X1
*F0	Initial guess at oscillation frequency (only used by APLAC)[a]		1 GHz
Fstart	Frequency range start	Frequency	1 GHz
Fend	Frequency range end	Frequency	2 GHz
Fsteps	Max numb. of freq. points in search		200
*VpMag	Guess at mag. of probe voltage (only used by APLAC)[a]		
*VpPh	Guess at ph. of probe voltage (not used)[a]		
*VpMax	Maximum mag of probe voltage in search	Voltage	8
Vsteps	Maximum number of voltage steps in search	Voltage	40
*Iter	Max number of iterations for oscillator analysis		200
*Damp	Convergence parameter for oscillator analysis		1
*Tone	Tone index for oscillation frequency (not implemented)		1
*Ftol	Frequency relative tolerance (not used)[a]		1e-3
*Pert	Perturbation factor		1e-6
*InitHarm	Number of harmonics in initial search		
*NLMethod	Solution method: Samanski=0, NNES=1		1
*IterExtrapSweep	Sweep iteration to start extrapolation at (large number to turn off)		5

Name	Description	Unit Type	Default
*OptMethod	Optimization method used to find the oscillation conditions (only used by APLAC). Try changing this parameter if oscillation conditions are not found.		MinMax
*TranKick	Transient analysis kick		Yes

[a]Elements designated as "not used" are manipulated by the simulator during oscillator analysis. Their values have no effect on simulation.

* indicates a secondary Parameter

Parameter Details

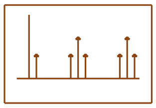
See “Nonlinear Oscillator Analysis” for the theory behind oscillator analysis and for a detailed discussion of the oscillator probe and its parameters.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

Analyze Oscillator Noise: OSCNOISE

Symbol



Summary

OSCNOISE instructs the simulator to compute oscillator phase noise following large-signal oscillator analysis.

Parameters

Name	Description	Unit Type	Default
ID	Oscillator		NS1
OFstart	Noise freq. start (offset from large signal)	Frequency	1e-6 GHz
OFend	Noise freq. end (offset from large signal)	Frequency	1e-3 GHz
OFsteps	Number of points in sweep		5
SwpType	Sweep type		LOG
*OverSamp	Sampling factor		2
*SparseTol	Drop tolerance for matrix sparsification		0.001
Harm	Harmonics for noise computation		{1,2}
*ConversionNoise	Compute conversion noise		No

** indicates a secondary parameter*

Parameter Details

OFstart and **OFend.** Define the noise frequency range. For oscillators, noise is specified as an offset from the large-signal frequency.

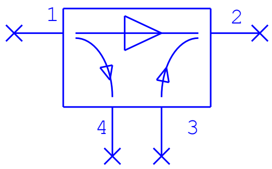
SwpType. Allowable sweep types are linear (LINEAR) and logarithmic (LOG).

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

Oscillator Open-Loop Test Element (Closed Form): OSCTEST

Symbol

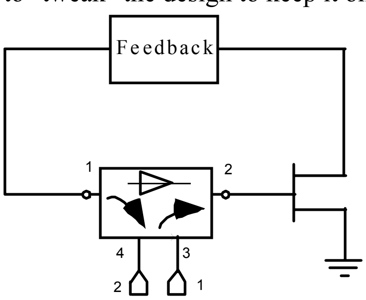


Summary

OSCTEST is used to determine loop gain in oscillator design and to break the feedback loop of an oscillator in the forward direction, at the fundamental frequency. When this is done, a source at port 3 is used to replace the feedback signal and the feedback itself is measured at port 4. This allows you to determine the open-loop gain of the circuit under large-signal conditions.

The following figure illustrates the use of OSCTEST. A large-signal excitation is applied to port 1, and the large signal S-parameter S21, at the fundamental frequency is monitored. The cutoff frequency FC must be set to any value between the fundamental frequency and the second harmonic. When the excitation is weak, |S21| must be greater than unity, or the circuit will not oscillate. As the excitation level is increased, |S21| decreases as the circuit enters saturation, and at some frequency and some excitation level, |S21| = 1 and its phase is zero. This point corresponds to the oscillation frequency, and the output power under these conditions is the oscillator's output power.

Before OSCTEST can be used under large-signal conditions, an initial small-signal design of the oscillator should be made. The circuit should be adjusted so that |S21| > 1 and its phase is zero, under small-signal conditions, at the desired frequency of oscillation. As the excitation is increased, the frequency of zero phase shift changes, and it may be necessary to "tweak" the design to keep it on the right frequency.



Parameters

Name	Description	Units Type	Default
ID	Element ID	Text	O1
*FC	f1 < fc < 2*f1	Frequency	1.0 GHz
*Z0	Port 1 and 2 impedance	Resistance	50 ohm

** indicates a secondary parameter*

Equations

At frequencies below FC, OSCTEST has the following S-parameters:

$$S_{12} = S_{41} = S_{23} = 1.0 \angle 0$$

At frequencies above FC, the S-parameters are

$$S_{12} = S_{21} = S_{41} = 1.0 \angle 0$$

All other S-parameters are zero.

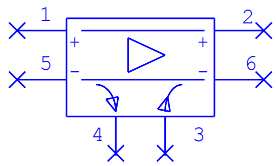
Layout

This element uses a special layout cell for a short circuit. The layout cell allows the elements connected on either side of the element to look through this element. For example, when a MLIN is hooked to a TFCM (capacitor) model and the project has bridge code configured, the line draws the proper interconnect between the line and the cap. If this element is placed between the line and the cap, the layout still draws the same.

You typically do not assign artwork cells to these items.

Oscillator Open-Loop Element Differential Excitation: OSCTEST2

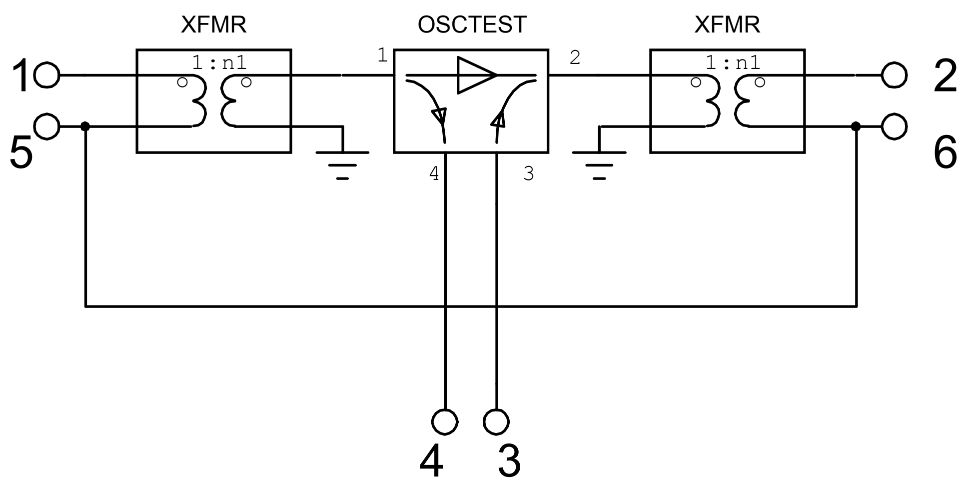
Symbol



Summary

OSCTEST2 is a type of directional coupler useful for loop gain computation in oscillators. It presents a short circuit everywhere except at a specified frequency (or range of frequencies) near the intended frequency of oscillation. OSCTEST2 is a more general form of OSCTEST, to which it degenerates when terminals 5 and 6 are grounded. It is more general than OSCTEST in the sense that it may be used to compute open-loop gain in differential oscillators.

Topology



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	OD1
FC	Loop frequency	Ghz	_FREQ*1e-9
*Z0	Port impedance	ohm	50

* indicates a secondary Parameter

Parameter Details

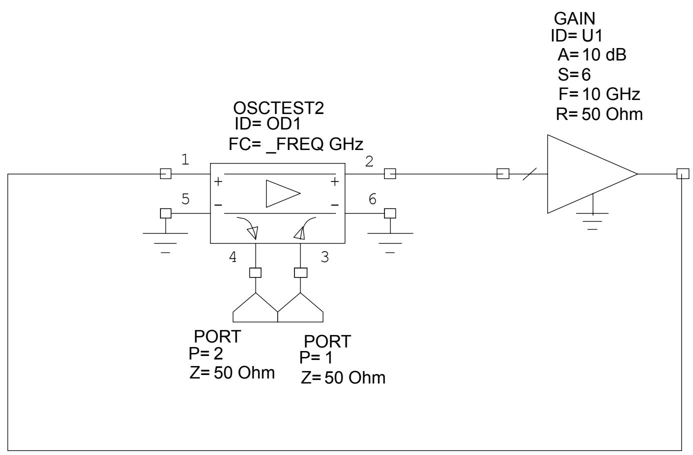
The easiest way to specify FC is to utilize the built-in variable _FREQ (for linear simulations) or _FREQH1 (for large-signal simulations). Note that _FREQ is a number (or array of numbers) equal to the project frequency in Hz, so if FC is specified in user units (GHz by default) the FC entry must be scaled appropriately (_FREQ*1e-9). If **Schematic Dependent Parameters Use Base Units** is selected on the Project Options dialog box **Schematic/Diagrams** tab, no scaling is necessary since parameter FC will automatically be calculated in base (MKS) units.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

Recommendations for Use

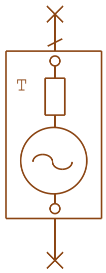
To compute loop gain, break the loop so that the large arrow points in the direction of forward gain, as in the following figure.



The S21 parameter of the circuit should equal the open-loop S21 of the GAIN stage.

(Obsolete) Oscillator Tuning Probe: OSCTPROBE

Symbol



Summary

This element is OBSOLETE and is replaced by the Oscillator Analysis Probe (OSCAPROBE) to simulate oscillators, then optimize any number of parameters to achieve the desired oscillation frequency (OSC_FREQ measurement). The details for OSCTPROBE are maintained here for backward compatibility.

OSCTPROBE initiates the tuning mode of oscillator analysis, whereby a circuit parameter is varied such that the circuit oscillates at a specified frequency.

Equivalent Circuit

OSCTPROBE is an ideal source in series with an ideal impedance element. The impedance presents an open circuit at all frequencies other than the fundamental frequency of oscillation.

Parameters

Name	Description	Unit Type	Default
ID	Element ID		X1
Freq	Target frequency of oscillation	Frequency	1 GHz
Par	Tuning parameter(<type>.<id>.<param>)		<type>.<id>.<param>
*Par_low	Lower parameter constraint (optional, mks)	[a]	0
*Par_high	Upper parameter constraint (optional, mks)	[a]	0
*VpMag	Guess at mag. of probe voltage (not used)[b]		
*VpPh	Guess at ph. of probe voltage (not used)[b]		
*VpMax	Maximum probe voltage in search (optional)	Voltage	3 V
Vsteps	Maximum number of voltage points in search	Voltage	15
*Iter	Max number of iterations for oscillator analysis		200
*Damp	Convergence parameter for oscillator analysis		1
*Ftol	Frequency relative tolerance (not used)[b]		1e-3

[a] Parameter values are entered in MKS units.

[b] Elements designated as "not used" are manipulated by the simulator in the course of oscillator analysis. Their values have no effect on simulation.

** indicates a secondary parameter*

Parameter Details

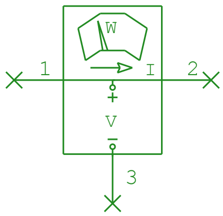
See “Nonlinear Oscillator Analysis” for the theory behind oscillator analysis.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

General Power Meter (3-port): P_METER3

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	P1

Implementation Details

When this current meter is inserted into a circuit path it can simultaneously measure DC, and forward- and reverse-traveling current. You can use P_METER3 with Ptime and Pharm measurements, as well as other Nonlinear Power measurements.

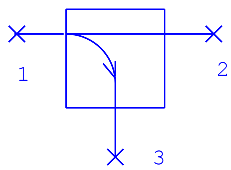
Layout

This element uses a special layout cell for a short circuit. The layout cell allows the elements connected on either side of the element to look through this element. For example, when a MLIN is hooked to a TFCM (capacitor) model and the project has bridge code configured, the line draws the proper interconnect between the line and the cap. If this element is placed between the line and the cap, the layout still draws the same.

You typically do not assign artwork cells to these items.

Power Sampler (Closed Form): PWRSMP

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Name	Text	U1
R	Port resistance		50

Implementation Details

Simulates a frequency-independent, lossless, directional coupler. The 3-port S parameters are given below.

$$S = \begin{bmatrix} 0 & 1 & 0 \\ 1 & 0 & 0 \\ 1 & 0 & 0 \end{bmatrix}$$

Inserting this element into a circuit and connecting a resistor to port 3 provides a way to sample voltage or power. The resistor should have the same value as R.

Layout

This element uses a special layout cell for a short circuit. The layout cell allows the elements connected on either side of the element to look through this element. For example, when a MLIN is hooked to a TFCM (capacitor) model and the project has bridge code configured, the line draws the proper interconnect between the line and the cap. If this element is placed between the line and the cap, the layout still draws the same.

You typically do not assign artwork cells to these items.

Stability Probe: STAB_PROBE

Symbol

Summary

STAB_PROBE identifies a stability probe that can be used to gather data for STAN stability analysis.

Parameters

Name	Description	Unit Type	Default
ID	Stability Probe Name	Text	SP1
STAN	Store STAN data		Yes
TYPE	Probe type		Current
DIFFERENTIAL	Differential Probe		No

Implementation Details

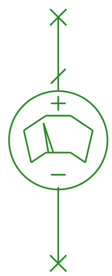
The probe TYPE defines what type of excitation the probe delivers to the circuit in the small signal analysis done at the large signal operating point. A **Current** TYPE is an open circuit in HB analysis and current source in small signal analysis. A **Voltage** TYPE is a short circuit in HB analysis and voltage source in small signal analysis. The probe can be made differential so you can add a series voltage probe.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Voltage Meter: V_METER

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Volt meter name	Text	VM1

Implementation Details

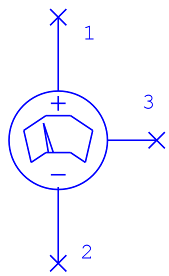
When connected between two nodes, this voltage meter measures the differential voltage between them. You can use V_METER with Vtime and Vharm measurements, as well as other Nonlinear Voltage measurements.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

Voltage Meter: V_METER2

Symbol



Summary

V_METER2 measures the voltage between pins 1 and 2 and sets the voltage of the 3rd pin equal to the measured voltage. V_METER2 is equivalent to an ideal voltage-controlled voltage source with infinite input resistance and zero output resistance and unity gain. This element is useful in transient co-simulation that is modeled in VSS using the NL_S_ENV system block. The output signals of the NL_S_ENV block are the port voltages of the selected ports in the circuit schematic. To make a circuit schematic voltage available as a signal in the system simulation, it can be measured using V_METER2, and the 3rd pin should be connected to an output port. The port impedance does not load the circuit being measured.

Parameters

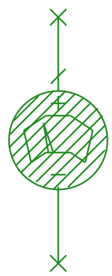
Name	Description	Unit Type	Default
ID	Volt meter name	Text	VM1

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

Noise Voltage Meter: V_NSMTR

Symbol



Summary

V_NSMTR is used to define the node (when the other end is connected to ground) or the node pair at which to calculate the noise voltage or perform other nonlinear noise measurements.

Parameters

Name	Description	Unit Type	Default
ID	Volt meter name	Text	VNS1
InpSrc	AC Input source ID	Text	V1

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Recommendations for Use

Nonlinear noise measurements are calculated at the ports or at the nodes (node pairs) denoted by V_NSMTR. To display a nonlinear noise measurement at the port of the circuit, you do not need to use V_NSMTR. V_NSMTR is primarily used for linear AC noise analysis, or to enable the display of nonlinear noise voltage measurements at the nodes other than the ports of the circuit.

The InpSrc parameter applies to AC noise analysis. It identifies the AC input source at which the equivalent input noise is measured (for example, the NOISEI measurement).

Voltage Probe: V_PROBE

Symbol

Summary

V_PROBE measures voltage at a node.

Equivalent Circuit

V_PROBE is a grounded voltmeter (V_METER)

Parameters

Name	Description	Unit Type	Default
ID	Element ID		VP1

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Microstrip

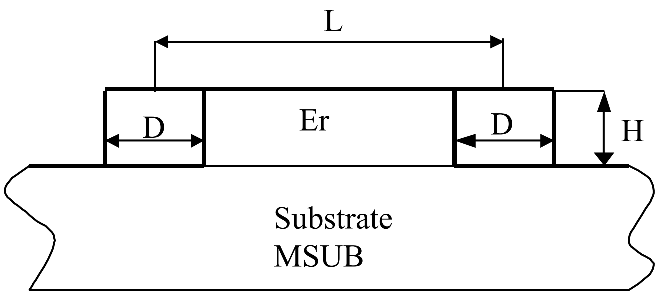
Airbridge with Optional Insulating Layer (Closed Form): ABRIDGE

Symbol

Summary

ABRIDGE models a flat bond wire or a conducting strip (bridge) connecting two conducting pads or microstrip lines. Wire connects to pads by means of two cylindrical vias. Model accounts for insulating dielectric layer under the bridge and for underlying substrate.

Topology



Parameters

Name	Description	Unit Type	Default
ID	Name	Text	TL1
W	Bridge conductor width	Length	W[a]
L	Bridge conductor length	Length	L[a]
D	Via diameter	Length	W[a]
H	Via/Insulating layer height	Length	W[a]
Er	Dielectric constant of insulating layer		1
MSUB	Substrate definition	Text	MSUB1[b]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See Microwave Office Layout Guide for details.

[b]Modify only if schematic contains multiple substrates. See the “Using Elements With Model Blocks” for details.

Parameter Restrictions and Recommendations

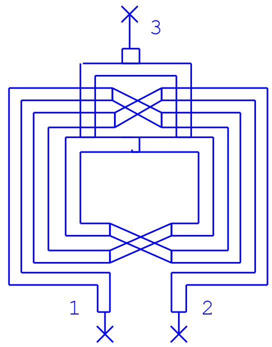
ABRIDGE uses closed form models MLIN (microstrip line) and VIA, so all limitations of these models apply to ABRIDGE.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

Microstrip Rectangular Differential Inductor (EM Quasi-Static): DIFRIND

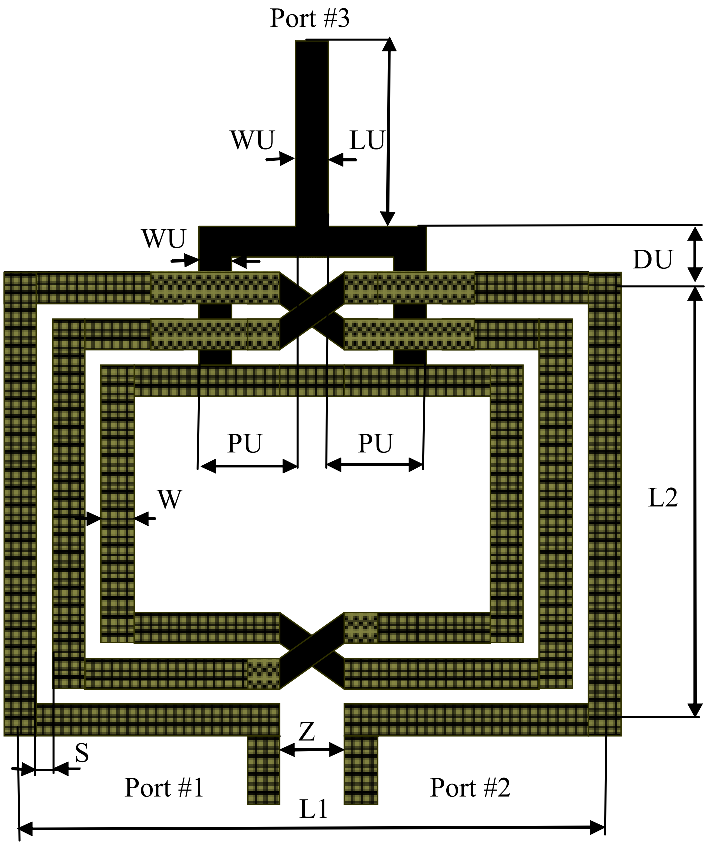
Symbol



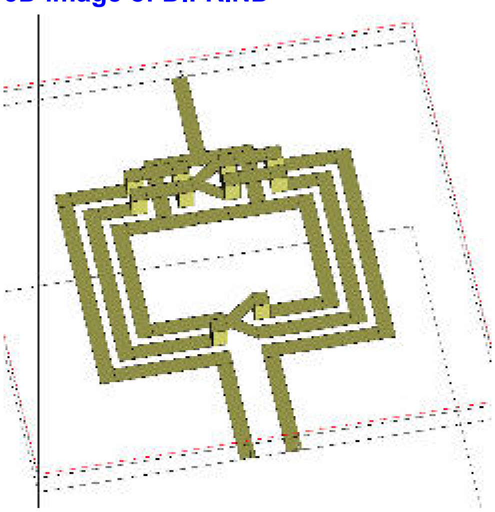
Summary

DIFRIND models a rectangular differential microstrip inductor/transformer with a center tap. This model is based on an evaluation of self and mutual inductances, capacitances and resistances between all parallel segments; this evaluation is based on an accurate quasi-static model of an arbitrary number of edge-coupled microstrip lines. The center tap is implemented as an underpass bridge that is common for GaAs designs. A differential inductor may replace two inductors joined by a common AC ground in differential amplifier or oscillator circuits.

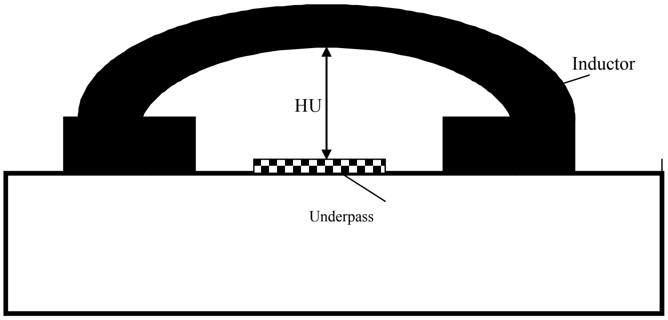
Topology (2D image of DIFRIND)



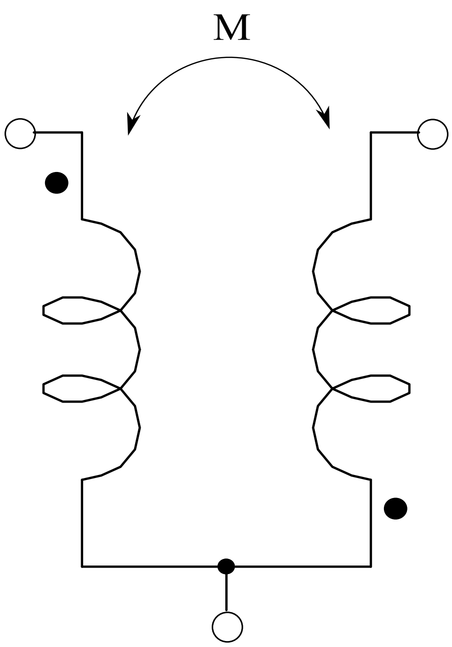
3D image of DIFRIND



Cross-section of underpass strip and overpassing section of inductor winding



Equivalent Circuit



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	IND
N	Number of turns(>=3, odd value only)		3
L1	Size of inductor horizontal side (w/ cross-over gap)	Length	80 um
L2	Size of inductor vertical side (w/o cross-over gap)	Length	155 um
W	Width of inductor windings	Length	10 um
S	Spacing between inductor windings	Length	5 um
Z	Gap for cross-over between windings	Length	20 um
TU	Thickness of underpass metal	Length	1 um
RhoU	Underpass metal bulk resistivity normalized to gold		1
WU	Width of underpass	Length	10 um

Name	Description	Unit Type	Default
HU	Height of dielectric above underpass	Length	1 um
LU	Length of underpass extension beyond inductor outline	Length	0 um
PU	Length of underpass horizontal shoulder	Length	30 um
DU	Length of underpass vertical shoulder that extends beyond inductor outline	Length	15 um
ErU	Relative dielectric constant of dielectric above underpass		1
TandU	Loss tangent of dielectric above underpass		0
*Acc	Accuracy parameter		1
MSUB	Substrate definition	Text	MSUB1[a]

[a]Modify only if schematic contains multiple substrates. See “Using Elements With Model Blocks” for details.

** indicates a secondary parameter*

Parameter Details

N. The number of turns/windings forming the differential inductor. Actually, the differential inductor comprises two connected sub-inductors; each winding comprises two halves, each belonging to a different sub-inductor. Sub-inductors connect at port #3. The number of windings N is limited to odd values only and must be greater or equal to 3. This model runs a layout feasibility check before performing calculations.

Acc. The accuracy parameter. The default value for Acc is 1. If Acc is less than 1 or greater than 10 it is set automatically to 1.

Parameter Restrictions and Recommendations

1. N >=3
2. N takes only odd values. Even values of N result in a layout that differs from the layout assumed for the current implementation of DIFRIND: This happens because even N mandates the location of port 3 at the same side of the inductor where ports 1 and 2 are located (odd N places port 3 at the opposite side). The proximity of all ports makes GaAs (and similar technologies) design relatively complicated; it also may result in undesirable stray coupling between inputs.
3. This model checks the underpass horizontal shoulder width PA to verify it is small enough to contact the top of the innermost winding (see "Topology") and if PA is large enough to prevent the underpass vertical shoulder to span the cross-over gap Z (see "Topology").
4. This model checks the sizes of internal openings because you may specify N, W, and S so that all windings cannot be squeezed into dimensions L1 and/or L2.
5. The Accuracy parameter Acc is limited to $1 \le Acc \le 10$. A larger value of Acc increases the density of mesh used in computations. The accuracy of model parameters may gain slightly from increasing Acc at the expense of a noticeable increase in computation time. Generally, a good trade-off between accuracy and computation time is to set Acc to 1.

Implementation Details

To decrease the calculation time for schematics that contain several DIFRIND inductors, cache is implemented for this model. This means that during the first evaluation of a schematic the most time-consuming intermediate parameters (characteristics of coupled lines) for each inductor instance are stored in the disk cache. Each inductor model checks this cache looking for its duplicate. Duplicate inductors copy the appropriate parameters from the disk cache, saving substantially on their recalculation.

Note that the model caches only frequency-independent characteristics of coupled lines, but recalculates the large equivalent circuit network (derived from coupled line characteristics) at each swept frequency. Thus, if the number of swept frequency points is large (say, 300) the total time spent on equivalent circuit evaluation may substantially exceed the time for evaluation of coupled line characteristics. In this particular case, time saving due to caching may be relatively moderate.

NOTE: The implementation of EM Quasi-Static models relies heavily on the involved numerical algorithms. This may lead to a noticeable increase in simulation time for schematics that employ many such models.

Layout

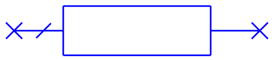
This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

References

[1] M.B. Bazdar, A.R. Djordjevic, R.F. Harrington, and T.K. Sarkar, "Evaluation of quasi-static matrix parameters for multiconductor transmission lines using Galerkin's method," IEEE Trans. Microwave Theory Tech., vol. MTT-42, July 1994, pp. 1223-1228

RFIC Microstrip Line (FEM Quasi-Static): FM1LIN

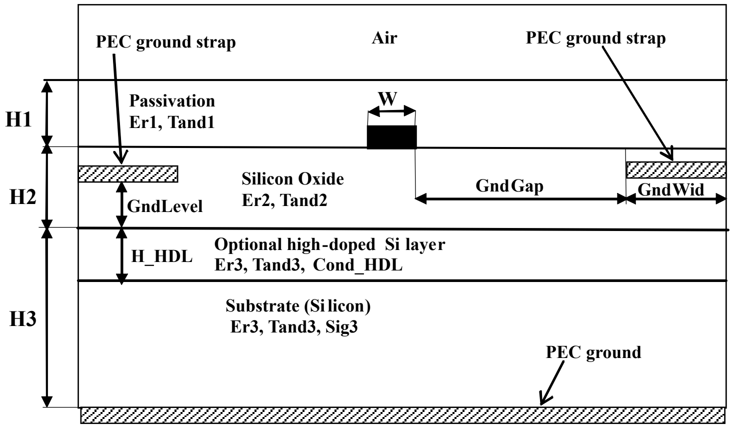
Symbol



Summary

FM1LIN models a section of microstrip line situated either atop or within insulating layers located on top of a conducting substrate. This stack is typical for many RFIC manufacturing technologies. A backing ground plane is always present. Optional perfectly conducting straps may be placed at both sides of line at arbitrary level to model lateral ground. FM1LIN can account for presence of conducting (high-doped) layer between insulating layers and conducting substrate.

Topology



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	TL1
W	Width of conductor	Length	W[a]
L	Conductor length	Length	L[a]
MRFSUB	Substrate definition	Text	MRFSUB1[b]
*IsGndStrap	Ground straps presence (Switch No/Yes)		No
*GndGap	Distance between ground strap and inductance edge	Length	10
*GndWid	Width of ground strap	Length	10
*GndLevel	Height of ground strap above substrate	Length	10
*IsHDLl	High-doped layer presence atop of substrate (Switch No/Yes)		No
*H_HDL	Bulk conductance of high-doped layer	Siemens/m	700
*Cond_HDL	Height of high-doped layer	Length	2 um
*SaveToFile	Switch "Save to txt file"=Yes/No		"No"

Name	Description	Unit Type	Default
*FileName	Name of text file with computed model parameters	String	Same as model name

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See Microwave Office Layout Guide for details.

[b]Modify only if schematic contains multiple substrates. See the "Using Elements With Model Blocks" for details.

** indicates a secondary parameter*

Parameter Details

MRFSUB. See the MRFSUB documentation. Note that notations H1, H2, H3, Er1, Er2, Er3, Tand1, Tand2, Tand3, Sig3 on the cross-sectional view above are MRFSUB respective parameters.

SaveToFile. The parameter is hidden by default and set to No. You can toggle this parameter to Yes and No. When set to Yes, the model creates a text file (the default is "model_name.txt") at the current project location. This text file contains characteristic impedances, propagation constants, effective dielectric constants, and values of RLGC line parameters at each project frequency. Each row contains respective values computed at the frequency specified in the first column (frequency in GHz, R in ohms/m, L in H/m, G in S/m, C in F/m).

FM1LIN creates a text file that contains complex characteristic impedance Re(Zo), Im(Zo), complex effective dielectric constant Re(Eeff), Im (Eff) (see the traditional effective dielectric constant in column 12), and Loss (dB/m) in columns 2-6. Columns 7-10 contain R, L, G, and C. Column 11 contains the propagation constant Beta in Rad/m. Note that column 12 contains the traditional effective dielectric constant Er_Eff that does not account for loss. The total number of columns in the file is 12.

The created text file might be linked or imported to a project as a data file, and the frequency behavior of any of the above parameters may be viewed using the proper data measurement. Note that the first column (frequency) is always in GHz so these measurements might be incompatible with other measurements placed on the same graph; thus, it may be best to place the data measurements on a separate dedicated graph.

FileName. By default this parameter is hidden and is set to FM1LIN.TXT. You can change the file name for each model instance to an arbitrary name with a length not exceeding 64 symbols.

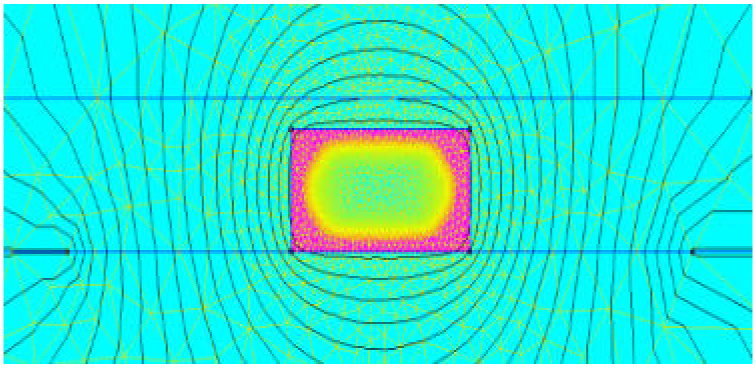
Parameter Restrictions and Recommendations

1. There are no limits on T except possible aggravations due to an overly thin T (see "Implementation Details"). Note that the conductor must stay within the passivation layer so the conductor thickness T must be less than 0.95*H1.
2. To exclude passivation, set MRFSUB parameters Er1=1, Tand1=0. Assign a value to H1 that well exceeds T to avoid an overly thin air layer between the top of the conductor and the top of the mock "passivation" layer (see "Implementation Details").

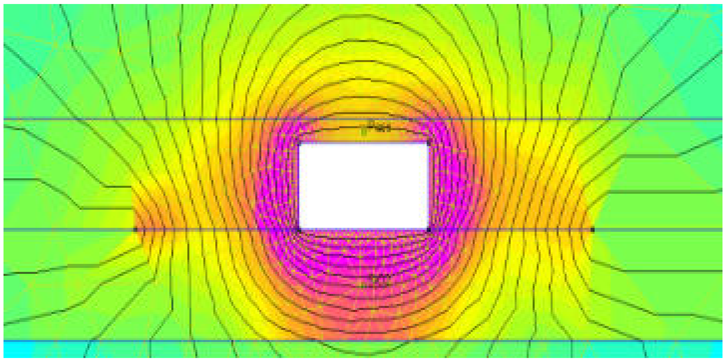
Implementation Details

The 2D Finite Element Method (FEM) in conjunction with the quasi-static problem formulation provides a very stable solution for RFIC's most common dimensions and frequency range. The FEM engine is partially based on the FEMM solver ([1] [10–11]); it comprises a mesher ([2] [10–11]) and two independent solvers: an electrical solver and a magnetic solver. The mesher generates a mesh that covers the entire cross-section including a cross-section of conductors. Usually, this method consumes a reasonable amount of computing resources; however, in certain situations memory consumption and run time may escalate because specified dimensions force the mesher to generate overly dense mesh. Commonly, this happens due to an error in the parameter specification that results in creation of extremely narrow layers.

The following figure demonstrates the typical distribution of an electric current across conductors cross-section and contours of magnetic potential. Note that this model provides mesh fine enough (and large enough) to reveal small details of skin effect and current crowding.



This figure demonstrates the distribution of an electric field and equipotential lines. Note how the closely located ground straps affect field distribution.



Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See “Cell Options Dialog Box: Layout Tab ” for Cell Options dialog box **Layout** tab details.

See “The Layout Process File (LPF)” for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

Recommendations for Use

Computation time substantially grows with mesh size. Mesh inside conductors may heavily contribute into total mesh size because size of a mesh cell inside conductor is governed by value of skin depth at the highest simulation frequency. Mesh does not vary with frequency so the same fine mesh works at lower frequencies. Frequency-independent mesh results in improved stability and consistency but with some sacrifice in solution time.

NOTE: The implementation of FEM Quasi-Static models relies on temporary intermediate text files. This model creates these files temporarily in the project directory and deletes them subsequently. There may be several dozen files, and they may use up to 100 MB of disk space, so make sure your hard drive has sufficient free space.

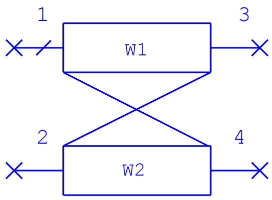
References

[1] David Meeker home page: http://femm.foster-miller.net/~dmeeker/ Follow this link for information: http://femm.foster-miller.net/Archives/readme.htm FEMM Manual: http://femm.foster-miller.net/Archives/doc/manual.pdf

[2] Jonathan Richard Shewchuk. Triangle. A Two-Dimensional Quality Mesh Generator and Delaunay Triangulator. Follow this link for information and download: http://www-2.cs.cmu.edu/~quake/triangle.html

2 RFIC Coupled Microstrip Lines (FEM Quasi-Static): FM2CLIN

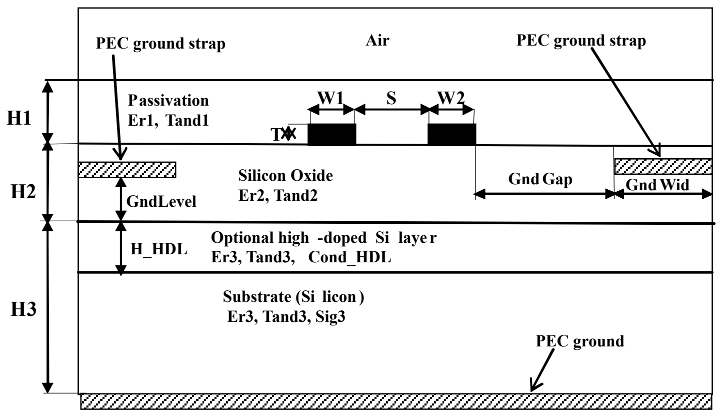
Symbol



Summary

FM2CLIN models a section of two edge-coupled microstrip lines situated either atop or within insulating layers located on top of a conducting substrate. This stack is typical for many RFIC manufacturing technologies. A backing ground plane is always present. Optional perfectly conducting straps may be placed at both sides of the line at an arbitrary level to model lateral ground. FM2CLIN can account for the presence of a conducting (high-doped) layer between insulating layers and the conducting substrate.

Topology



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	TL1
W1	Width of conductor #1	Length	W[a]
W2	Width of conductor #2	Length	W[a]
S	Spacing between conductors	Length	W[a]
L	Conductor length	Length	L[a]
MRFSUB	Substrate definition	Text	MRFSUB1[b]
*IsGndStrap	Ground straps presence (Switch No/Yes)		No
*GndGap	Distance between ground strap and inductance edge	Length	10
*GndWid	Width of ground strap	Length	10
*GndLevel	Height of ground strap above substrate	Length	10

Name	Description	Unit Type	Default
*IsHDLl	High-doped layer presence atop of substrate (Switch No/Yes)		No
*H_HDL	Bulk conductance of high-doped layer	Siemens/m	700
*Cond_HDL	Height of high-doped layer	Length	2 um
*SaveToFile	Switch "Save to txt file"=Yes/No		"No"
*FileName	Name of text file with computed model parameters	String	Same as model name

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See Microwave Office Layout Guide for details.

[b]Modify only if schematic contains multiple substrates. See "Using Elements With Model Blocks" for details.

** indicates a secondary parameter*

Parameter Details

MRFSUB. See the MRFSUB documentation. Note that notations H1, H2, H3, Er1, Er2, Er3, Tand1, Tand2, Tand3, Sig3 on the cross-sectional view above are MRFSUB respective parameters.

SaveToFile. The parameter is hidden by default and set to No. You can toggle this parameter to Yes and No. When set to Yes, the model creates a text file (the default is "model_name.txt") at the current project location. This text file contains characteristic impedances, propagation constants, effective dielectric constants, and values of RLGC line parameters at each project frequency. Each row contains respective values computed at the frequency specified in the first column (frequency in GHz, R in ohms/m, L in H/m, G in S/m, C in F/m).

This model implies that in the general case ($W_1 \neq W_2$), two modes, namely, C and P exist in a modeled structure (see [2]). Note that in a symmetrical structure ($W_1=W_2$) C-mode corresponds to even mode and P-mode corresponds to odd mode.

FM2CLIN outputs complex characteristic impedances Re(Zc1), Im(Zc1), Re(Zp1), Im(Zp1), Re(Zc2), Im(Zc2), Re(Zp2), Im(Zp2), complex effective dielectric constants Re(EeffC), Im(EeffC), Re(EeffP), Im(EeffP) (see traditional effective dielectric constants in columns 38, 39), and losses for C and P modes LossC (dB/m), LossP(dB/m) to columns 2-15. Entries of R, L, G, and C matrices are distributed among columns 16-31 in respective order. Columns 32-37 contain Re(Rc), Im(Rc), Re(Rp), Im(Rp), BetaC, BetaP where Rc is the ratio of C - mode voltage in the second line to C - mode voltage in the first line; Rp is the same ratio in P-mode case (see details in [3], section 4.3.1); BetaC and BetaP are propagation constants of C- and P-modes in Rad/m. Note that columns 38 and 39 contain traditional effective dielectric constants ErC_Eff and ErP_Eff (they do not account for losses). The total number of columns in the text file is 39.

The text file that is created might be linked or imported to a project as a data file and frequency behavior of any previously mentioned parameter may be viewed using the proper data measurement. Note that the first column (frequency) is always in GHz so these measurements might be incompatible with other Microwave Office measurements placed on the same graph; thus, you may prefer to place the aforementioned data measurements on a separate dedicated graph.

FileName. By default this parameter is hidden and is set to FM2CLIN.TXT. You can change the file name for each model instance to an arbitrary name with a length not exceeding 64 symbols.

Parameter Restrictions and Recommendations

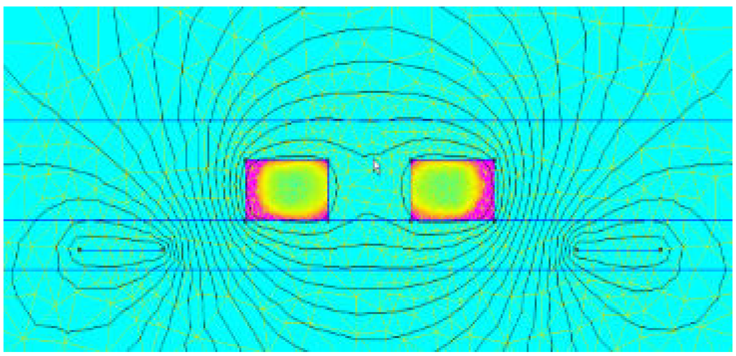
1. There are no limits on T except possible aggravations due to an overly thin T (see "Implementation Details"). Note that the conductor must stay within the passivation layer so the conductor thickness T must be less than 0.95*H1.

2. To exclude passivation, set MRFSUB parameters Er1=1, Tand1=0. Assign a value to H1 that well exceeds T to avoid an overly thin air layer between the top of the conductor and the top of the mock "passivation" layer (see "Implementation Details").

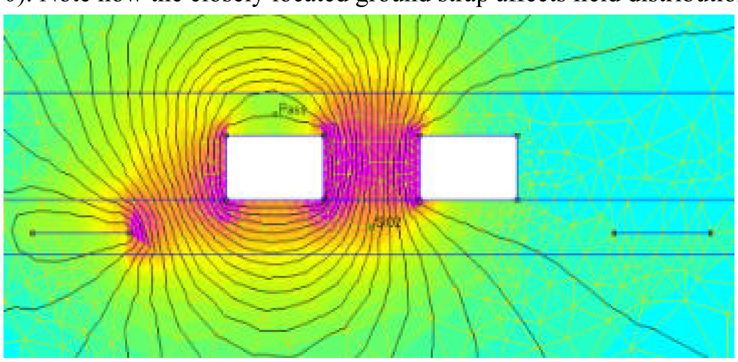
Implementation Details

The 2D Finite Element Method (FEM) in conjunction with the quasi-static problem formulation provides a very stable solution for RFIC's most common dimensions and frequency range. The FEM engine is partially based on the FEMM solver ([1] [10–15]); it comprises a mesher ([2] [10–15]) and two independent solvers: an electrical solver and a magnetic solver. The mesher generates a mesh that covers the entire cross-section including a cross-section of conductors. Usually, this method consumes a reasonable amount of computing resources; however, in certain situations memory consumption and run time may escalate because specified dimensions force the mesher to generate overly dense mesh. Commonly, this happens due to an error in the parameter specification that results in creation of extremely narrow layers.

The following figure demonstrates the typical distribution of an electric current across a conductor cross-section and contours of magnetic potential (conductors are at 1A of impressed current each). Note that this model provides mesh fine enough (and large enough) to reveal small details of skin effect and current crowding.



This figure demonstrates the distribution of an electric field and equipotential lines (conductors are at potentials 1 and 0). Note how the closely located ground strap affects field distribution.



Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.

4. Click **OK** to use the new line type in the layout.

See “Cell Options Dialog Box: Layout Tab ” for Cell Options dialog box **Layout** tab details.

See “The Layout Process File (LPF)” for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

Recommendations for Use

Computation time substantially grows with mesh size. Mesh inside conductors may heavily contribute into total mesh size because size of a mesh cell inside conductor is governed by value of skin depth at the highest simulation frequency. Mesh does not vary with frequency so the same fine mesh works at lower frequencies. Frequency-independent mesh results in improved stability and consistency but with some sacrifice in solution time.

NOTE: The implementation of FEM Quasi-Static models relies on temporary intermediate text files. This model creates these files temporarily in the project directory and subsequently deletes them. There may be several dozen files and they may use up to 100 MB of disk space, so ensure that your hard drive has sufficient free space.

References

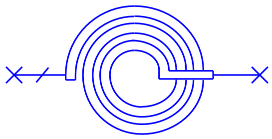
[1] David Meeker home page: http://femm.foster-miller.net/~dmeeker/ Follow this link for information: http://femm.foster-miller.net/Archives/readme.htm FEMM Manual: http://femm.foster-miller.net/Archives/doc/manual.pdf

[2] Jonathan Richard Shewchuk. Triangle. A Two-Dimensional Quality Mesh Generator and Delaunay Triangulator. Follow this link for information and download: http://www-2.cs.cmu.edu/~quake/triangle.html

[3]R. Mongia, I. Bahl, and P. Bhartia, RF and Microwave Coupled-Line Circuits, Artech House, Norwood, MA, 1999.

(Obsolete) Circular Spiral RFIC Microstrip Inductor (FEM Quasi-Static): FMCIND

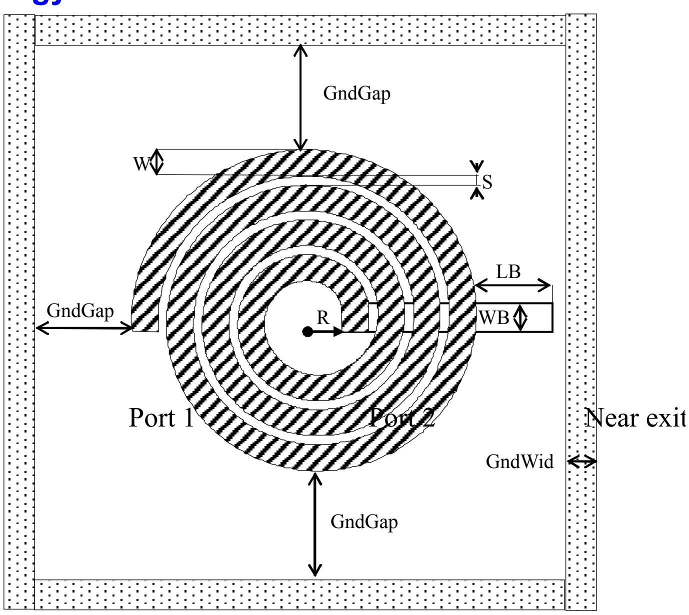
Symbol



Summary

This element is OBSOLETE and is replaced by the Circular Spiral RFIC Inductor (FEM Quasi-Static) (FMCIND2) element. This circuit component models an RFIC microstrip circular spiral inductor with a strip underpass. The inductor is located atop an insulating dielectric and covered with a passivation layer. This model is based on an evaluation of self and mutual inductances, capacitances, admittances, and resistances between all spiral turns. The calculation of these circuit parameters is based on an accurate FEM quasi-static model of coupled microstrip circular rings arranged on/into the dielectric (silicon oxide) layer above the conducting substrate. This model accounts for the passivation layer, optional ground straps surrounding the inductor, and for the presence of the optional conducting (high-doped) layer between insulated layers and the substrate.

Topology



Cross-sectional Views

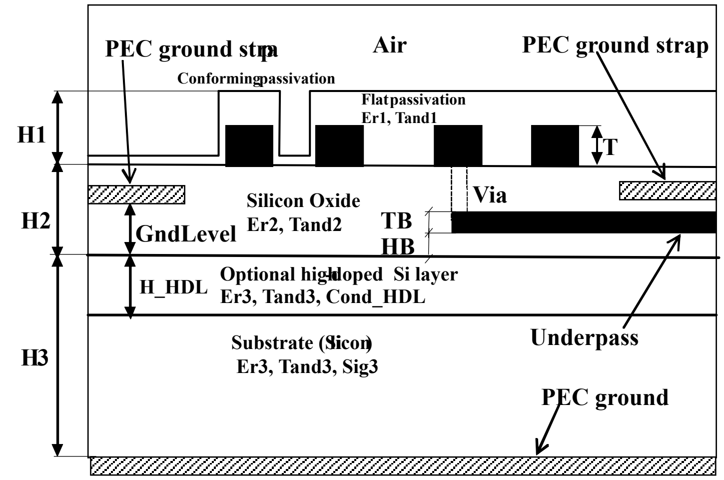


Figure 1. Suspended underpass (CMOS RFIC)

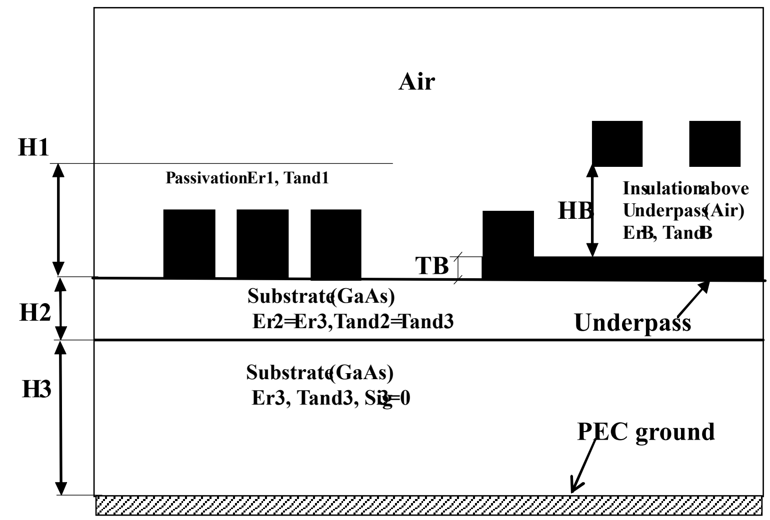


Figure 2. On-substrate underpass (GaAs RFIC)

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	MI1
NT	Number of turns		3
W	Conductor width	Length	10 um
S	Conductor spacing	Length	2 um
RI	Internal radius of a spiral	Length	20 um
UExit	Underpass exits toward near/far side of spiral (Switch "Near exit/Far exit")		Far exit

Name	Description	Unit Type	Default
UPass	Underpass floats in insulating oxide/sits immediately on substrate surface (Switch "Suspended/On-substrate underpass)		Suspended
WB	Width of underpass conductor	Length	10 um
HB	Height of underpass conductor above substrate/Height of inductor above underpass (Switch "Suspended/On-substrate underpass)	Length	2 um
LB	Length of underpass conductor extension beyond inductor	Length	0 um
TB	Thickness of underpass conductor	Length	1 um
ErB	Relative dielectric constant of dielectric above underpass (used only if UPass="On-substrate")		1 um
TandB	Loss tangent of dielectric above underpass (used only if UPass="On-substrate")		1 um
RhoB	Underpass metal bulk resistivity normalized to gold		1
MRFSUB	Substrate definition	Text	MRFSUB1[a]
*IsGndStrap	Ground straps present (Switch No/Yes)		No
*GndGap	Distance between ground strap and inductance edge	Length	10
*GndWid	Width of ground strap	Length	10
*GndLevel	Height of ground strap above substrate	Length	10
*IsHDL	High-doped layer present atop of substrate (Switch No/Yes)		No
*H_HDL	Bulk conductance of high-doped layer	Siemens/m	700
*Cond_HDL	Height of high-doped layer	Length	2 um
*PassWrap	Flat/Conforming passivation (Switch Flat passivation/Conforming passivation)		Flat passivation
*HP	Thickness of conforming passivation layer between turns	Length	2 um

[a]Modify only if schematic contains multiple substrates. See the "Using Elements With Model Blocks" for details.

** indicates a secondary parameter*

Parameter Details

NT. Number of turns NT must provide an integer number of half-turns, that is 1, 1.5,2, 2.5 etc. Model accepts any value for NT in the range 1..50 but if NT does not provide integer number of half-turns model changes NT to closest suitable value and issues a warning.

UExit. This switch allows direction of the underpass either to the nearest turn (Near exit, see Topology) or to span the internal opening before "diving" under the conducting turns (Far exit).

UPass. This switch allows to select inductor design. Setting "Suspended underpass" (see Cross-sectional Views, Figure 1) implies that all inductor turns are located atop of insulating dielectric (oxide) and underpass is "floating" in oxide above substrate surface. Setting "On-substrate underpass" (see Cross-sectional Views, Figure 2) implies that substrate is non-conductive, all inductor turns sit immediately on the substrate, portion of turn metal is elevated above substrate and makes a kind of arc above the underpass trace.

HB. Depends on UPass setting.

UPass = "Suspended substrate": Height of the underpass is the distance between the substrate top (top of high-doped layer if present) and the bottom of the underpass conductor.

UPass = "On-substrate underpass":HB is essentially elevation of portion of inductor turns above underpass.

ErB, TandB. Usage depends on UPass setting.

UPass = "Suspended substrate": model just ignores ErB and TandB.

UPass = "On-substrate underpass": ErB and TandB represent characteristics of material filling spacing between arched inductor turns and underpass strip (often air).

LB. The length of the extension of underpass conductor beyond the external edge of inductor body.

MRFSUB. See the MRFSUB documentation. Note that notations H1, H2, H3, Er1, Er2, Er3, Tand1, Tand2, Tand3, Sig3 on the cross-sectional view above are MRFSUB respective parameters.

H1, H2, Er1, Er2, Tand1, Tand2, Tand3, Sig3. Settings of these parameters depend on setting of switch UPass.

UPass = "Suspended substrate": said parameters must be set as shown on Cross-sectional Views, Figure 1.

UPass = "Suspended substrate": said parameters must be set as shown on Cross-sectional Views, Figure 2. If substrate height is Hsub than it is recommended to set H2=0.25*Hsub, H3=0.75*Hsub (user must make sure that H2+H3=Hsub). Make sure also that Er2=Er3=ErSub,Tand2=Tand3=TandSub where ErSub, TandSub are substrate characteristics.

IsGndStrap. IsGndStrap=Yes installs additional (to backing ground plane) lateral ground implemented as ground straps around inductor (see topology). IsGndStrap=No makes model use only backing ground plane. Latter value of this switch makes model to ignore parameters GndGap, GndWid, GndLevel. IsGndStrap must be left at default value "No" if parameter UPass="On-substrate underpass".

IsHDL. IsHDL=Yes converts upper portion of substrate into high-doped layer. This layer has height H_HDL, dielectric constant Er3, loss tangent Tand3, and conductance Cond_HDL. Is_HDL=No makes model to ignore parameters H_HDL and Cond_HDL. IsHDL must be left at default value "No" if parameter UPass="On-substrate underpass".

PassWrap. Default value "Flat passivation" provides flat passivation layer covering inductor turns. Value "Conforming passivation" makes thin passivation layer to conform cross-sectional contour as shown at Cross-sectional Views, Figure 1.

HP. This parameter is ignored if PassWrap="Flat passivation", else HP provides thickness of passivation layer that covers the gap between turns. Model implies that passivation layer has the same thickness at sides and bottom of the gap (Cross-sectional Views, Figure 1).

Parameter Restrictions and Recommendations

1. $1 \leq NT \leq 50$
2. There are no limits on T except possible aggravations due to overly thin T (see "Implementation Details"). Note that the conductor must stay confined within passivation layer so the conductor thickness T must be less than 0.95*H1
3. Selection PassWrap=Yes makes the passivation layer to conform the conductor contour (See Cross-sectional Views, Figure 1). Note that PassWrap=Yes makes passivation thickness vary: on the conductor top it equals H1-T, at the valley between the conductors (both sides and bottom) the passivation thickness is Hp. Models limits the width of this valley (gap between passivation dielectric walls) at 0.1 micron.

4. To exclude passivation, set MRFSUB parameters Er1=1 and Tand1=0. Assign a reasonable value to H1 that well exceeds T to avoid an overly thin air layer between the top of the conductor and the top of the mock "passivation" layer (see "Implementation Details").
5. To model a suspended MEMS (micro-electromechanical systems) inductor, set Er2=1.
6. Setting UPass="On-substrate" implies that inductor is located on non-conducting lossy substrate, switch IsGndStrap set to "No ground straps", switch IsHDL set to "No HDL", sum of H2 and H3 is equal to substrate height, material characteristics of layer H2 are identical to those of layer H3. Passivation switch PassWrap can be set to any desirable option.

Implementation Details

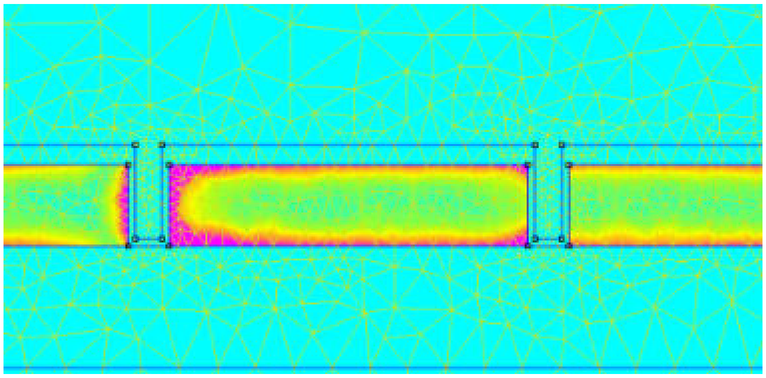
Spiral is treated as a set of concentric rings connected in series with each other and with capacitively coupled strip underpass. Problem is solved as axisymmetric 2D layout of interacting lossy thick metal traces sitting atop of insulation layer. Underpass is treated as a lossy thick transmission line running inside insulation layer (alternatively, underpass may be placed atop of substrate). Entire stackup has conducting substrate underneath of insulating layer.

Lateral grounds are optional and are implemented as PEC infinitely thin rings (ring shape is mandatory due to axial symmetry of layout); user can place them arbitrary within insulation layer).

Passivation layer may be made either flat or conformal.

The 2D Finite Element Method (FEM) in conjunction with quasi-static problem formulation provides a very stable solution for RFICs most common dimensions and frequency range. An FEM engine is partially based on a FEMM solver ([1] [10–21]); it comprises mesher ([2] [10–21]) and two independent solvers: An electrical solver and a magnetic solver. The mesher generates mesh that covers the entire cross-section including the cross-section of conductors. Usually, this method consumes a reasonable amount of computing resources; however, in certain situations memory consumption and run time may escalate because some specified dimensions force the mesher to generate an overly dense mesh. Commonly, this happens due to an error in parameter specifications that results in creation of extremely narrow layers.

The following figure demonstrates the typical distribution of current across a conductor cross-section (conforming passivation). Note that model provides mesh fine enough (and large enough) to reveal small details of skin effect and current crowding.



Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.

2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See "Cell Options Dialog Box: Layout Tab " for Cell Options dialog box **Layout** tab details.

See "The Layout Process File (LPF)" for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

Recommendations for Use

Computation time substantially grows with mesh size. Mesh inside conductors may heavily contribute into total mesh size because size of a mesh cell inside conductor is governed by value of skin depth at the highest simulation frequency. Mesh does not vary with frequency so the same fine mesh works at lower frequencies. Frequency-independent mesh results in improved stability and consistency but with some sacrifice in solution time.

NOTE: The implementation of FEM Quasi-Static models relies on temporary intermediate text files. This model creates these files temporarily in the project directory and subsequently deletes them. There may be several dozen files and they may take up to 100 MB of disk space, so make sure that your project hard drive has sufficient free space.

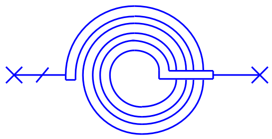
References

[1] David Meeker home page: http://femm.foster-miller.net/~dmeeker/ Follow this link for information: http://femm.foster-miller.net/Archives/readme.htm FEMM Manual: http://femm.foster-miller.net/Archives/doc/manual.pdf

[2] Jonathan Richard Shewchuk. Triangle. A Two-Dimensional Quality Mesh Generator and Delaunay Triangulator. Follow this link for information and download: http://www-2.cs.cmu.edu/~quake/triangle.html

Circular Spiral RFIC Inductor (FEM Quasi-Static): FMCIND2

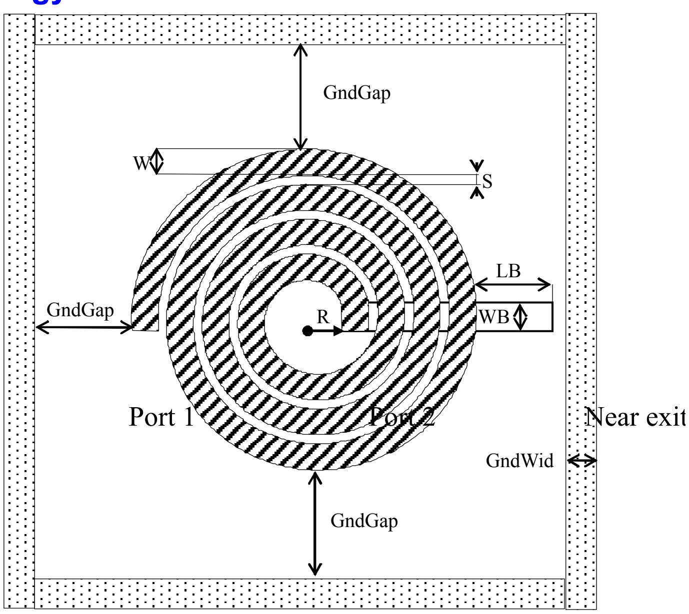
Symbol



Summary

This circuit component models an RFIC microstrip circular spiral inductor with a strip underpass. The inductor is located atop an insulating dielectric and covered with a passivation layer. This model is based on an evaluation of self and mutual inductances, capacitances, admittances, and resistances between all spiral turns. The calculation of these circuit parameters is based on an accurate FEM quasi-static model of coupled microstrip circular rings arranged on/into the dielectric (silicon oxide) layer above the conducting substrate. This model accounts for the passivation layer, optional ground straps surrounding the inductor, and for the presence of the optional conducting (high-doped) layer between insulated layers and the substrate. Simulation speed-up tools include disk cache and AFS (Advanced Frequency Sweep).

Topology



Cross-sectional Views

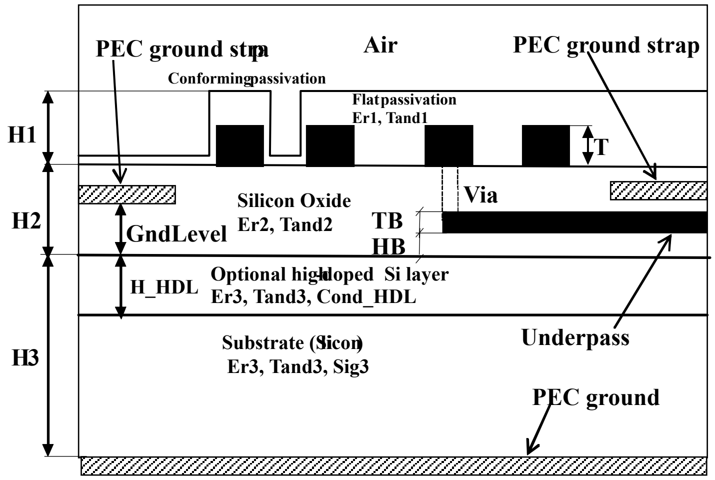


Figure 1. Suspended Underpass (CMOS RFIC)

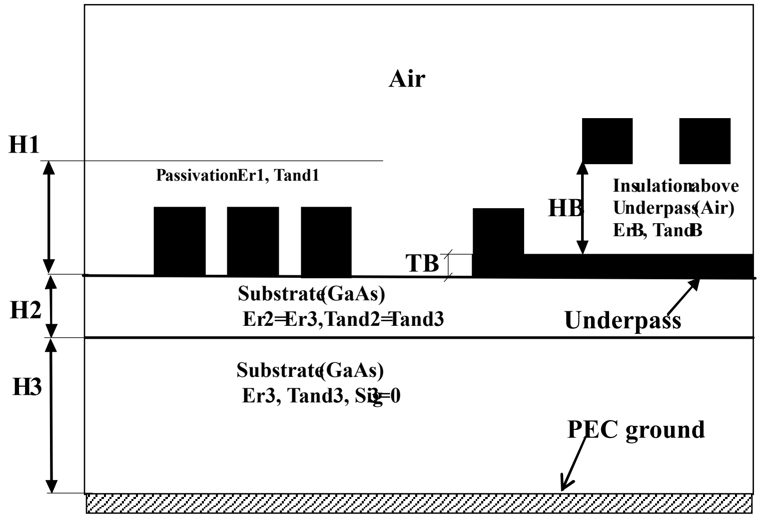


Figure 2. On-substrate Underpass (GaAs RFIC)

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	MI1
NT	Number of turns		3
W	Conductor width	Length	10 um
S	Conductor spacing	Length	2 um
RI	Internal radius of a spiral	Length	20 um
UExit	Underpass exits toward near/far side of spiral (Switch "Near exit/Far exit")		Far exit

Name	Description	Unit Type	Default
UPass	Underpass floats in insulating oxide/sits immediately on substrate surface (Switch "Suspended/On-substrate underpass)		Suspended
WB	Width of underpass conductor	Length	10 um
HB	Height of underpass conductor above substrate/Height of inductor above underpass (Switch "Suspended/On-substrate underpass)	Length	2 um
LB	Length of underpass conductor extension beyond inductor	Length	0 um
TB	Thickness of underpass conductor	Length	1 um
ErB	Relative dielectric constant of dielectric above underpass (used only if UPass="On-substrate")		1 um
TandB	Loss tangent of dielectric above underpass (used only if UPass="On-substrate")		1 um
RhoB	Underpass metal bulk resistivity normalized to gold		1
MRFSUB	Substrate definition	Text	MRFSUB1[a]
*IsGndStrap	Ground straps present (Switch No/Yes)		No
*GndGap	Distance between ground strap and inductance edge	Length	10
*GndWid	Width of ground strap	Length	10
*GndLevel	Height of ground strap above substrate	Length	10
*IsHDL	High-doped layer present atop of substrate (Switch No/Yes)		No
*Cond_HDL	Bulk conductance of high-doped layer	Siemens/m	700
*H_HDL	Height of high-doped layer	Length	2 um
*PassWrap	Flat/Conforming passivation (Switch Flat passivation/Conforming passivation)		Flat passivation
*HP	Thickness of conforming passivation layer between turns	Length	2 um
UseAFS	Use/Do not use AFS for model speed up (Switch UseAFS/NoAFS)		UseAFS

[a]Modify only if the schematic contains multiple substrates. See the "Using Elements With Model Blocks" for details.

** indicates a secondary parameter*

Parameter Details

NT. Number of turns; must represent an integer number of half-turns that is equal to 1, 1.5, 2, 2.5 and so on. This model accepts any value for NT in the range 1..50 but if NT does comply with the above limitation the model changes NT to the closest suitable value and issues a warning.

UExit. This switch allows the direction of the underpass either to the nearest turn (Near the exit, see the "Topology" section) or to span the internal opening before "diving" under the conducting turns (Far exit). Certain combinations of UExit and NT values are prohibited (see the "Parameter Restrictions and Recommendations" section).

UPass. This switch allows you to select the inductor design. Setting "Suspended underpass" (see "Cross-sectional Views", Figure 1) implies that all inductor turns are located atop insulating dielectric (oxide) and that the underpass is "floating" in oxide above the substrate surface. Setting "On-substrate underpass" (see "Cross-sectional Views", Figure 2) implies that the substrate is non-conductive, all inductor turns sit immediately on the substrate, and the portion of turn metal is elevated above the substrate and makes an arc above the underpass trace.

HB. Usage depends on the UPass setting:

UPass=Suspended substrate - Height of the underpass is the distance between the substrate top (top of high-doped layer if present) and the bottom of the underpass conductor.

UPass=On-substrate underpass - HB is essentially the elevation of the portion of inductor turns above the underpass.

ErB, TandB. Usage depends on UPass setting:

UPass=Suspended substrate - the model ignores ErB and TandB.

UPass=On-substrate underpass - ErB and TandB represent the characteristics of material filling spacing between arched inductor turns and the underpass strip (often air).

LB. The length of the extension of underpass conductor beyond the external edge of the inductor body.

MRFSUB. See MRFSUB. Note that notations H1, H2, H3, Er1, Er2, Er3, Tand1, Tand2, Tand3, and Sig3 in "Cross-sectional Views" are MRFSUB respective parameters.

H1, H2, Er1, Er2, Tand1, Tand2, Tand3, Sig3. Settings depend on the setting of switch UPass:

UPass=Suspended substrate - parameters must be set as shown in "Cross-sectional Views" Figure 1.

UPass=Suspended substrate - parameters must be set as shown in "Cross-sectional Views" Figure 2. If the substrate height is Hsub then you should set H2=0.25*Hsub and H3=0.75*Hsub (you must make sure that H2+H3=Hsub). Ensure also that Er2=Er3=ErSub,Tand2=Tand3=TandSub where ErSub and TandSub are substrate characteristics.

IsGndStrap. "Yes" installs additional (to backing ground plane) lateral ground implemented as ground straps around the inductor (see the "Topology" section). "No" makes the model use only a backing ground plane. The latter value of this switch makes the model ignore parameters GndGap, GndWid, and GndLevel. This parameter must be left at the default value "No" if parameter UPass=On-substrate underpass.

IsHDL. "Yes" converts the upper portion of the substrate into a high-doped layer. This layer has height H_HDL, dielectric constant Er3, loss tangent Tand3, and conductance Cond_HDL. "No" makes this model ignore parameters H_HDL and Cond_HDL. IsHDL must be left at the default value "No" if parameter UPass=On-substrate underpass.

PassWrap. The default value "Flat passivation" provides a flat passivation layer covering inductor turns. "Conforming passivation" makes a thin passivation layer to conform the cross-sectional contour as shown in "Cross-sectional Views" Figure 1.

HP. This parameter is ignored if PassWrap=Flat passivation, otherwise HP provides the thickness of passivation layer that covers the gap between turns. This model implies that the passivation layer has the same thickness at the sides and bottom of the gap (see "Cross-sectional Views" Figure 1).

UseAFS. This model can optionally use Advanced Frequency Sweep (UseAFS is the default setting). If UseAFS=NoAFS then this model simulates at each frequency point from the specified frequency sweep. If UseAFS=UseAFS then this model does not simulate at each frequency point; instead, it simulates at several automatically selected frequency points and uses the results to obtain a very accurate frequency-dependent approximation valid through the entire frequency sweep. See the specifics regarding AFS interaction with disk cache in the "Implementation Details" section.

Layout cell NSeg. Note that the corresponding layout cell of this model has parameter NSeg (to edit NSeg, select the corresponding layout cell, right-click and choose **Shape Properties** to display the Cell Options dialog box, then click the **Parameters** tab). NSeg defines the order of regular circumscribed polygon used to draw the spiral shape. The default value is 64. Larger values (up to 360) make the contour smoother at the cost of slower drawing.

Layout cell BrGap (since v7.51). You should select the FMCIND2 layout cell on the **Layout** tab of the Element Options dialog box. The layout cell of this model has parameter BrGap (to edit BrGap select the corresponding layout shell, right-click and choose **Shape Properties** to display the Cell Options dialog box, then click the **Parameters** tab). This parameter is valid only for certain line types, like Plated Metal Line, when the inductor body is comprised of two metals (top and bottom) connected by a via along all windings, and the bridge is built on the bottom metal. In this case, the bridge conductor may be short-circuited by windings at the bottom metal level. To avoid this, this layout cell provides a gap in the bottom metal to allow the bridge to exit from the inductor center untouched. BrGap is a user-defined extension to the width of this gap (the default is 5 microns). You should use the layout 3D view to check the actual gap width. In rare cases when parameter W is small and BrGap is relatively large, approximately half of the internal winding may be stripped of the bottom metal; in this case decreasing BrGap may help to restore the bottom metal back to internal winding.

Parameter Restrictions and Recommendations

1. 1<=NT<=50
2. There are no limits on T except possible aggravations due to overly thin T (see the "Implementation Details" section). Note that the conductor must stay confined within the passivation layer so the conductor thickness T must be less than 0.95*H1.
3. Selection PassWrap=Yes makes the passivation layer conform to the conductor contour (see "Cross-sectional Views" Figure 1). Note that PassWrap=Yes makes passivation thickness vary: on the conductor top it equals H1-T, at the valley between the conductors (both sides and bottom) the passivation thickness is Hp. Models limit the width of this valley (gap between passivation dielectric walls) at 0.1 micron.
4. To exclude passivation, set MRFSUB parameters Er1=1 and Tand1=0. Assign a reasonable value to H1 that well exceeds T to avoid an overly thin air layer between the top of the conductor and the top of the mock "passivation" layer (see the "Implementation Details" section).
5. To model a suspended MEMS (micro-electromechanical systems) inductor, set Er2=1.
6. Setting UPass=On-substrate implies that the inductor is located on a non-conducting lossy substrate, IsGndStrap is set to "No ground straps", IsHDL is set to "No HDL", the sum of H2 and H3 is equal to the substrate height, and the material characteristics of layer H2 are identical to those of layer H3. Passivation switch PassWrap can be set to any desirable option.
7. FMCIND2 does not allow you to use frequency-dependent model material parameter Cond_HDL and substrate material parameters Er1, Er2, Er3, Tand1, Tand2, Tand3, and Sig3.
8. Certain combinations of NT and UExit are not allowed. This model does not allow the bridge to cross port 1, and changes the bridge exit to avoid this crossing (a warning is issued and the layout displays the actual bridge position). If NT=n (n is an integer) then Near Exit is prohibited; if you set UExit=Near Exit, the model resolves this conflict by pointing the bridge "south" between Near Exit and Far Exit (Far Exit may add excessive bridge length). If NT=n+1/2, then Far Exit is prohibited; if you set UExit=Far Exit then this model overrides and sets UExit=Near Exit.

Implementation Details

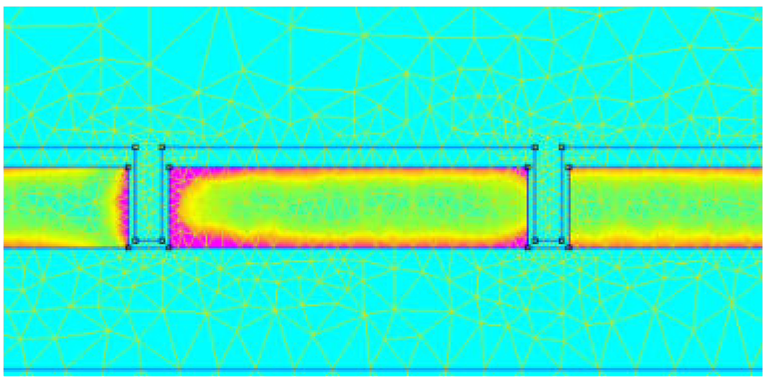
The spiral is treated as a set of concentric rings connected in series with each other and with a capacitively coupled strip underpass. The problem is solved as an axisymmetric 2D layout of interacting lossy thick metal traces sitting atop an insulation layer. The underpass is treated as a lossy thick transmission line running inside an insulation layer (alternatively, the underpass may be placed atop a substrate). The entire stackup has a conducting substrate underneath an insulating layer.

Lateral grounds are optional and are implemented as PEC infinitely thin rings (the ring shape is mandatory due to the axial symmetry of layout); you can place them arbitrarily within the insulation layer).

The Passivation layer can be either flat or conformal.

The 2D Finite Element Method (FEM) in conjunction with the quasi-static problem formulation provides a very stable solution for RFICs most common dimensions and frequency range. An FEM engine is partially based on a FEMM solver ([1] [10–28]); it comprises the mesher ([1] [10–28]) and two independent solvers: an electrical solver and a magnetic solver. The mesher generates mesh that covers the entire cross-section including the cross-section of conductors. Usually, this method consumes a reasonable amount of computing resources; however, in certain situations memory consumption and run time may escalate because some specified dimensions force the mesher to generate an overly dense mesh. This commonly occurs due to an error in parameter specifications that results in creation of extremely narrow layers.

The following figure demonstrates the typical distribution of current across a conductor cross-section (conforming passivation). Note that this model provides mesh fine enough (and large enough) to reveal small details of skin effect and current crowding.



Disk Cache and AFS

Depending on the setting of UseAFS, the model caches either model Y-matrices at each sweep frequency (UseAFS=NoAFS) or frequency-dependent approximation (UseAFS=UseAFS).

If UseAFS=NoAFS then all frequencies in a sweep are part of cache search criteria. This means that all subsequent runs of identical models at the same frequency sweep use disk cache for speed-up. However, if even one frequency differs from those in a cache (or one frequency is added to the sweep) this model runs full-blown simulation.

When UseAFS=UseAFS this model includes first and last (extreme) sweep frequencies along with model parameters in the cache search criteria. This means that once approximation results are written to the cache, all subsequent sweeps with arbitrary frequency step and identical extreme frequencies are served with AFS approximation obtained from the cache without the need to run AFS again. However, any sweep with even one extreme frequency distinct from those in the cache demands a new run of the AFS engine.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.

4. Click **OK** to use the new line type in the layout.

See "Cell Options Dialog Box: Layout Tab " for Cell Options dialog box **Layout** tab details.

See "The Layout Process File (LPF)" for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

Recommendations for Use

Computation time substantially grows with mesh size. The mesh inside conductors may heavily contribute to total mesh size because the size of a mesh cell inside a conductor is governed by the value of skin depth at the highest simulation frequency. Mesh does not vary with frequency, so the same fine mesh works at lower frequencies. Frequency-independent mesh results in improved stability and consistency but with some sacrifice in solution time.

NOTE: The implementation of FEM Quasi-Static models relies on temporary intermediate text files. This model creates these files temporarily in the project directory and subsequently deletes them. There may be several dozen files and they may take up to 100 MB of disk space, so make sure that your project hard drive has sufficient free space.

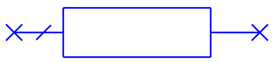
References

[1] David Meeker home page: http://femm.foster-miller.net/~dmeeker/; For more information go to: http://femm.foster-miller.net/Archives/readme.htm; For the FEMM manual go to: http://femm.foster-miller.net/Archives/doc/manual.pdf

[2] Jonathan Richard Shewchuk. Triangle. A Two-Dimensional Quality Mesh Generator and Delaunay Triangulator. For information and download go to: http://www-2.cs.cmu.edu/~quake/triangle.html

Microstrip Line with Extended Range of Design Parameters (FEM Quasi-Static): FMLINX

Symbol

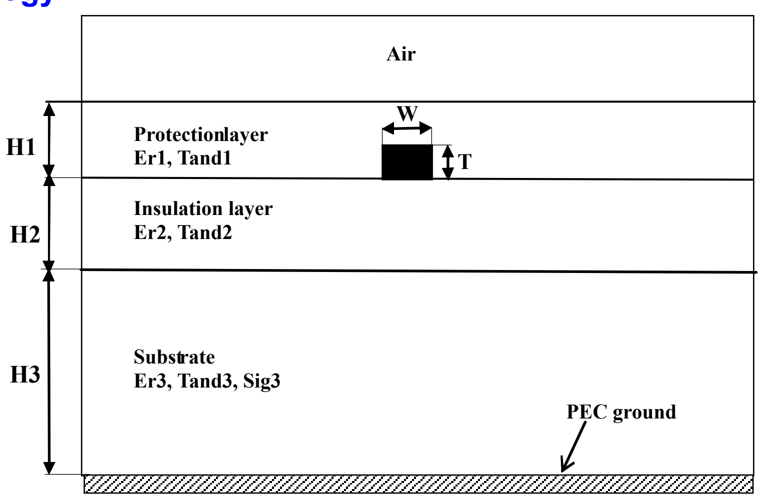


Summary

FMLINX models a section of microstrip line situated either atop or within insulating layers located on top of an optionally conducting substrate. The substrate is backed with a ground plane. FMLINX stays operational for very wide as well as very narrow conductors, provides accurate results for conductors with thickness exceeding width, and works with conducting substrates. This model does not account for non-TEM dispersion effects.

FMLINX is constructed as an X-model (table-based interpolation) using the results of a EM 2-D quasi-static cross-sectional analysis based on the Finite Elements Method (FEM). For a detailed discussion of the X-models see “ EM-based Models (X-models) ”. The Quasi-static FEM technique provides an accurate solution in the presence of a conducting/lossy dielectric stack at a price of a longer processor time spent on 2D meshing and solution of large systems of linear equations. FMLINX gains outstanding computational speed increases due to the table-based interpolation.

Topology



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	TL1
W	Width of conductor	Length	W[a]
L	Conductor length	Length	L[a]
MRFSUB	Substrate definition	Text	MRFSUB1[b]
*AutoFill	AutoFill DataBase if not equal to 0		0

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory.

[b]Modify only if the schematic contains multiple substrates.

** indicates a secondary parameter*

Parameter Details

W. Conductor width is an independent parameter. You can set it to an arbitrary value that complies with the restrictions listed in the "Parameter Restrictions and Recommendations" section.

MRFSUB. Substrate parameters are listed in the description of MRFSUB. All MRFSUB parameters are fixed parameters for FMLINX. Changes made to these parameters may cause a new table autofill if MWO does not find a matching table for the specified set of MRFSUB parameters.

Autofill. The AutoFill parameter is a secondary (hidden by default) input which allows you to specify that the entire interpolation table should be filled automatically at the current values. To initiate this filling process, set this parameter to 1. During normal operation, you should set this parameter to zero. To access the hidden parameter, double-click the schematic element.

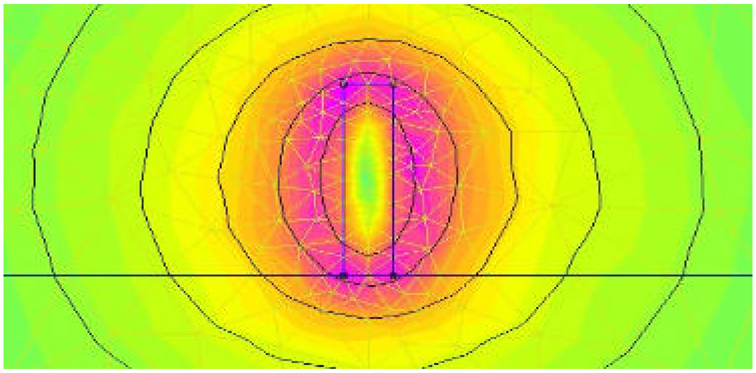
Parameter Restrictions and Recommendations

1. FMLINX implies that ratio W/(H2+H3) lies within a predefined range of $0.001 \le W/(H2 + H3) \le 5$. Outside of this range, this model extrapolates output parameters and issues a warning. FMLINX considers that the absolute minimal value of conductor width Wmin=0.02 micron, so at W/(H2+H3) equal to the smallest value 0.001, the total value of H2+H3 must be less than or equal to 20 microns. So, if H2+H3<20 micron FMLINX displays an error message.
2. Changes made to fixed parameters T, Rho, H1, H2, H3, Er1, Er2, Er3, Tand1, Tand2, Tand3, and Sig3 may initiate an autofill of a new table (the *FMLINX*.EMX* file in the *EM_Models* folder) if FMLINX does not find a matching table. The autofill procedure may take a while due to large conductor widths that might be involved in the process, particularly if H2+H3 is large (for example, at H2+H3=640 micron maximum W is 5*640=3200 micron). NI AWR recommends keeping tabs on created tables. A good practice is to give them meaningful names that help to identify the design for which the particular table was created.
3. FMLINX sets a cap to conductor thickness at 0.1 micron due to possible meshing problems.
4. To model classic microstrip line on the substrate with parameters H_sub, Er_sub, and Tand_sub, you must set Er1=1, H1>T, Tand1=0, Er2=Er3=Er_sub, Tand2=Tand3=Tand_sub, and Sig3=0, and provide H2+H3=Hsub.
5. Try not to set H1 very close to T (if your design allows) because a very thin gap between the conductor top and boundary H1/Air may cause very fine mesh.
6. The layer with parameters H1, Er1, and Tand1 is indented to serve as a protection layer.
7. For conductive substrate, some insulation between the conductor and substrate must be provided via the layer with parameters H2, Er2, and Tand2.

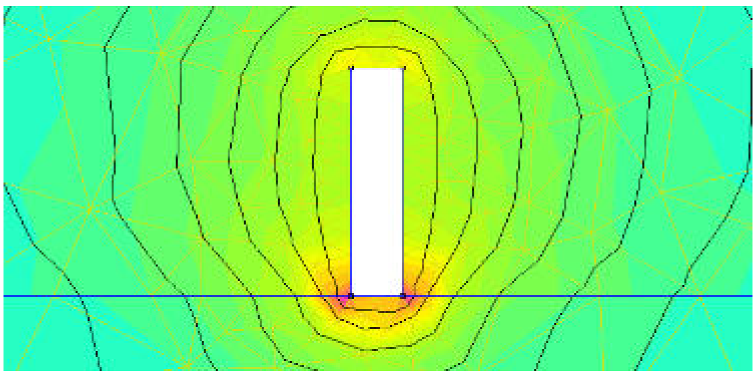
Implementation Details

The 2D Finite Element Method (FEM) in conjunction with the quasi-static problem formulation provides a very stable solution for RFIC's most common dimensions and frequency range. The FEM engine is partially based on the FEMM solver ([1] [10–31]); it comprises a mesher ([2] [10–32]) and two independent solvers: an electrical solver and a magnetic solver. The mesher generates a mesh that covers the entire cross-section including a cross-section of conductors. Usually, this method consumes a reasonable amount of computing resources; however, in certain situations memory consumption and run time may escalate because specified dimensions force the mesher to generate overly dense mesh. Commonly, this happens due to an error in the parameter specification that results in creation of extremely narrow layers.

The following figure demonstrates the distribution of magnetic field in the vicinity of the conductor with W<<T and contours of magnetic potential.



This figure demonstrates the distribution of an electric field in the vicinity of the conductor with W<<T and equipotential contours.



This X-model was developed under research performed at NI AWR Corporation. The full set of details of the implementation are considered proprietary in nature.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See "Cell Options Dialog Box: Layout Tab " for Cell Options dialog box **Layout** tab details.

See "The Layout Process File (LPF)" for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

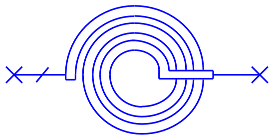
References

[1] David Meeker home page: http://femm.foster-miller.net/~dmeeker/ Follow this link for information: http://femm.foster-miller.net/Archives/readme.htm FEMM Manual: http://femm.foster-miller.net/Archives/doc/manual.pdf

[2] Jonathan Richard Shewchuk. Triangle. A Two-Dimensional Quality Mesh Generator and Delaunay Triangulator. Follow this link for information and download: http://www-2.cs.cmu.edu/~quake/triangle.html

(Obsolete) Octagonal Spiral RFIC Microstrip Inductor (FEM Quasi-Static): FMOCIND

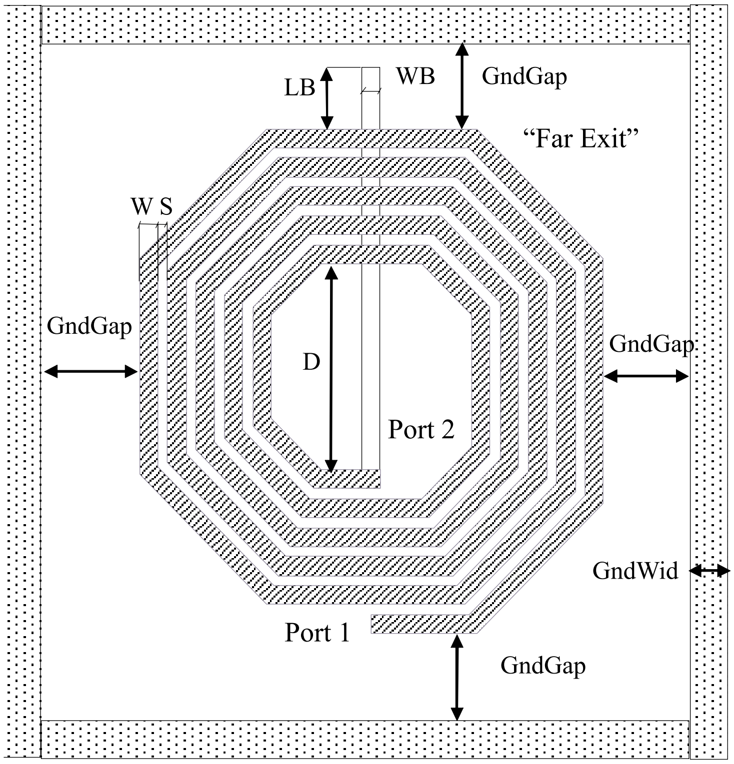
Symbol



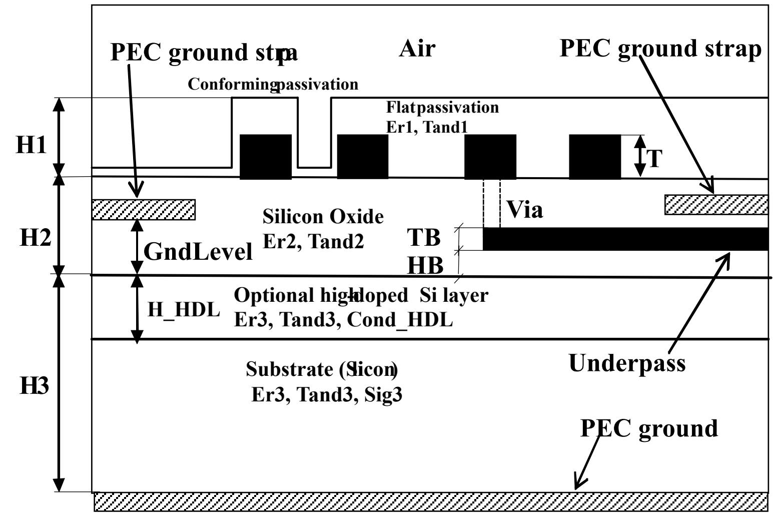
Summary

This element is OBSOLETE and is replaced by the Octagonal Spiral RFIC Inductor (FEM Quasi-Static) (FMOCIND2) element. This circuit component models an RFIC microstrip octagonal spiral inductor with a strip underpass. The inductor is located atop an insulating dielectric and covered with a passivation layer. This model is based on an evaluation of self and mutual inductances, capacitances, admittances, and resistances between all spiral turns. The calculation of these circuit parameters is based on an accurate FEM quasi-static model of coupled microstrip lines arranged on/into the dielectric (silicon oxide) layer above the conducting substrate. This model accounts for the passivation layer, optional ground straps surrounding the inductor, and for the presence of the optional conducting (high-doped) layer between insulated layers and the substrate.

Topology



Cross-sectional View



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	MI1
NT	Number of turns		3
W	Conductor width	Length	10 um
S	Conductor spacing	Length	2 um
D	Internal diameter measured between internal edge of contact pad of Port 2 and edge of opposite spiral segment	Length	20 um
UExit	Underpass exits toward near/far side of spiral (Switch "Near exit/Far exit")		Far exit
WB	Width of underpass conductor	Length	10 um
HB	Height of underpass conductor above substrate/Height of inductor above underpass (Switch "Suspended/On-substrate underpass)	Length	2 um
LB	Length of underpass conductor extension beyond inductor	Length	0 um
TB	Thickness of underpass conductor	Length	1 um
RhoB	Underpass metal bulk resistivity normalized to gold		1
MRFSUB	Substrate definition	Text	MRFSUB1[a]
*IsGndStrap	Ground straps present (Switch No/Yes)		No
*GndGap	Distance between ground strap and inductance edge	Length	10
*GndWid	Width of ground strap	Length	10
*GndLevel	Height of ground strap above substrate	Length	10
*IsHDL	High-doped layer present atop of substrate (Switch No/Yes)		No
*H_HDL	Bulk conductance of high-doped layer	Siemens/m	700
*Cond_HDL	Height of high-doped layer	Length	2 um

Name	Description	Unit Type	Default
*PassWrap	Flat/Conforming passivation (Switch Flat passivation/Conforming passivation)		Flat passivation
*HP	Thickness of conforming passivation layer between turns	Length	2 um

[a]Modify only if schematic contains multiple substrates. See the "Using Elements With Model Blocks" for details.

** indicates a secondary parameter*

Parameter Details

NT. Number of turns NT must provide an integer number of quarter-turns, that is 1, 1.25, 1.5 etc. Model accepts any value for NT in the range 1..50 but if NT does not provide integer number of quarter-turns model changes NT to the closest suitable value.

UExit. This switch allows direction of the underpass either to the nearest turn (Near exit, see Topology) or to span the internal opening before "diving" under the conducting turns (Far exit).

HB. Height of the underpass is the distance between the substrate top (top of high-doped layer if present) and the bottom of the underpass conductor.

LB. The length of the extension of underpass conductor beyond the external edge of inductor body.

MRFSUB. See the MRFSUB documentation. Note that notations H1, H2, H3, Er1, Er2, Er3, Tand1, Tand2, Tand3, Sig3 on the cross-sectional view above are MRFSUB respective parameters.

sGndStrap. IsGndStrap=Yes installs additional (to backing ground plane) lateral ground implemented as ground straps around inductor (see topology). IsGndStrap=No makes model use only backing ground plane. Latter value of this switch makes model to ignore parameters GndGap, GndWid, GndLevel.

IsHDL. IsHDL=Yes converts upper portion of substrate into high-doped layer. This layer has height H_HDL, dielectric constant Er3, loss tangent Tand3, and conductance Cond_HDL. Is_HDL=No makes model to ignore parameters H_HDL and Cond_HDL.

PassWrap. Default value "Flat passivation" provides flat passivation layer covering inductor turns. Value "Conforming passivation" makes thin passivation layer to conform cross-sectional contour as shown at Cross-sectional View.

HP. This parameter is ignored if PassWrap="Flat passivation", else HP provides thickness of passivation layer that covers the gap between turns. Model implies that passivation layer has the same thickens at sides and bottom of the gap (Cross-sectional View).

Parameter Restrictions and Recommendations

1. $1 \leq NT \leq 50$
2. There are no limits on T except possible aggravations due to overly thin T (see "Implementation Details"). Note that the conductor must stay confined within passivation layer so the conductor thickness T must be less than 0.95*H1
3. Selection PassWrap=Yes makes the passivation layer to conform the conductor contour (See Cross-sectional View). Note that PassWrap=Yes makes passivation thickness vary: on the conductor top it equals H1-T, at the valley between the conductors (both sides and bottom) the passivation thickness is Hp. Models limits the width of this valley (gap between dielectric walls) at 0.1 micron.

4. To exclude passivation, set MRFSUB parameters Er1=1 and Tand1=0. Assign a reasonable value to H1 that well exceeds T to avoid an overly thin air layer between the top of the conductor and the top of the mock "passivation" layer (see "Implementation Details").
5. To model a suspended MEMS (micro-electromechanical systems) inductor, set Er2=1.

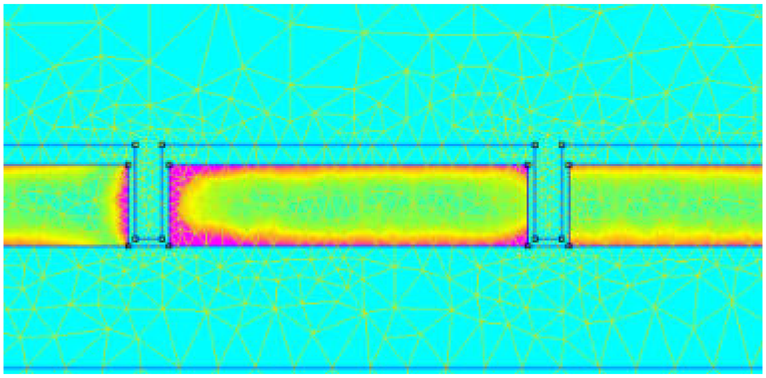
Implementation Details

Lateral grounds are optional and are implemented as PEC infinitely thin strips surrounding inductor (see Topology); user can place them arbitrary within insulation layer).

Passivation layer may be made either flat or conformal.

The 2D Finite Element Method (FEM) in conjunction with quasi-static problem formulation provides a very stable solution for RFICs most common dimensions and frequency range. An FEM engine is partially based on a FEMM solver ([1] [10–37]); it comprises mesher ([2] [10–37]) and two independent solvers: An electrical solver and a magnetic solver. The mesher generates mesh that covers the entire cross-section including the cross-section of conductors. Usually, this method consumes a reasonable amount of computing resources; however, in certain situations memory consumption and run time may escalate because some specified dimensions force the mesher to generate an overly dense mesh. Commonly, this happens due to an error in parameter specifications that results in creation of extremely narrow layers.

The following figure demonstrates the typical distribution of current across a conductor cross-section (conforming passivation). Note that model provides mesh fine enough (and large enough) to reveal small details of skin effect and current crowding.



Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See "Cell Options Dialog Box: Layout Tab " for Cell Options dialog box **Layout** tab details.

See "The Layout Process File (LPF)" for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

Recommendations for Use

Computation time substantially grows with mesh size. Mesh inside conductors may heavily contribute into total mesh size because size of a mesh cell inside conductor is governed by value of skin depth at the highest simulation frequency. Mesh does not vary with frequency so the same fine mesh works at lower frequencies. Frequency-independent mesh results in improved stability and consistency but with some sacrifice in solution time.

NOTE: The implementation of FEM Quasi-Static models relies on temporary intermediate text files. This model creates these files temporarily in the project directory and subsequently deletes them. There may be several dozen files and they may take up to 100 MB of disk space, so make sure that your project hard drive has sufficient free space.

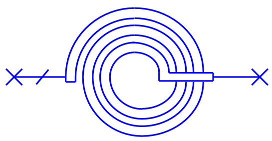
References

[1] David Meeker home page: http://femm.foster-miller.net/~dmeeker/ Follow this link for information: http://femm.foster-miller.net/Archives/readme.htm FEMM Manual: http://femm.foster-miller.net/Archives/doc/manual.pdf

[2] Jonathan Richard Shewchuk. Triangle. A Two-Dimensional Quality Mesh Generator and Delaunay Triangulator. Follow this link for information and download: http://www-2.cs.cmu.edu/~quake/triangle.html

Octagonal Spiral RFIC Inductor (FEM Quasi-Static): FMOCIND2

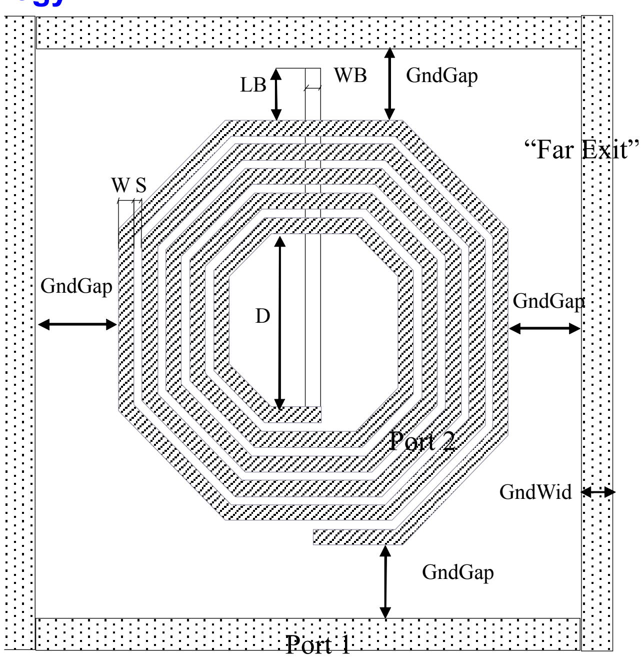
Symbol



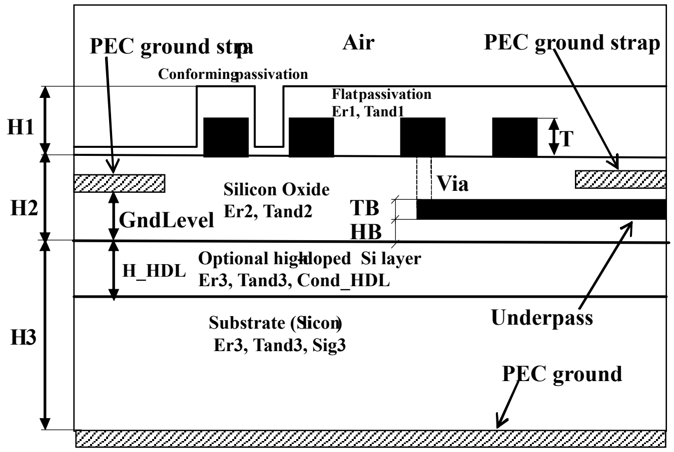
Summary

This circuit component models an RFIC microstrip octagonal spiral inductor with a strip underpass. The inductor is located atop an insulating dielectric and covered with a passivation layer. This model is based on an evaluation of self and mutual inductances, capacitances, admittances, and resistances between all spiral turns. The calculation of these circuit parameters is based on an accurate FEM quasi-static model of coupled microstrip lines arranged on/into the dielectric (silicon oxide) layer above the conducting substrate. This model accounts for the passivation layer, optional ground straps surrounding the inductor, and for the presence of the optional conducting (high-doped) layer between insulated layers and the substrate. Simulation speed up tools include disk cache and AFS (Advanced Frequency Sweep).

Topology



Cross-sectional View



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	MI1
NT	Number of turns		3
W	Conductor width	Length	10 um
S	Conductor spacing	Length	2 um
D	Internal diameter measured between internal edge of contact pad of Port 2 and edge of opposite spiral segment	Length	20 um
UExit	Underpass exits toward near/far side of spiral ("Near exit/Far exit")		Far exit
WB	Width of underpass conductor	Length	10 um
HB	Height of underpass conductor above substrate	Length	2 um
LB	Length of underpass conductor extension beyond inductor	Length	0 um
TB	Thickness of underpass conductor	Length	1 um
RhoB	Underpass metal bulk resistivity normalized to gold		1
MRFSUB	Substrate definition	Text	MRFSUB1[a]
*IsGndStrap	Ground straps present (No/Yes)		No
*GndGap	Distance between ground strap and inductance edge	Length	10
*GndWid	Width of ground strap	Length	10
*GndLevel	Height of ground strap above substrate	Length	10
*IsHDL	High-doped layer present atop of substrate (No/Yes)		No
*Cond_HDL	Bulk conductance of high-doped layer	Siemens/m	700
*H_HDL	Height of high-doped layer	Length	2 um
*PassWrap	Flat/Conforming passivation (Flat passivation/Conforming passivation)		Flat passivation
*HP	Thickness of conforming passivation layer between turns	Length	2 um

Name	Description	Unit Type	Default
UseAFS	Use/Do not use AFS for model speed up (UseAFS/NoAFS)		UseAFS

[a]Modify only if the schematic contains multiple substrates. See "Using Elements With Model Blocks" for details.

** indicates a secondary parameter*

Parameter Details

NT. Number of turns NT must provide an integer number of quarter-turns, that is 1, 1.25, 1.5 etc. This model accepts any value for NT in the range 1..50 but if NT does not provide an integer number of quarter-turns the model changes NT to the closest suitable value.

UExit. Allows the direction of the underpass either to the nearest turn (Near exit, see the "Topology" section) or to span the internal opening before "diving" under the conducting turns (Far exit).

HB. Height of the underpass is the distance between the substrate top (top of high-doped layer if present) and the bottom of the underpass conductor.

LB. The length of the extension of underpass conductor beyond the external edge of inductor body.

MRFSUB. See MRFSUB. Note that notations H1, H2, H3, Er1, Er2, Er3, Tand1, Tand2, Tand3, and Sig3 on the cross-sectional view above are MRFSUB respective parameters.

IsGndStrap. IsGndStrap=Yes installs additional (to backing ground plane) lateral ground implemented as ground straps around the inductor (see the "Topology" section). IsGndStrap=No makes this model use only a backing ground plane. This value makes the model ignore the GndGap, GndWid, and GndLevel parameters.

IsHDL. IsHDL=Yes converts the upper portion of substrate into a high-doped layer. This layer has height H_HDL, dielectric constant Er3, loss tangent Tand3, and conductance Cond_HDL. Is_HDL=No makes the model ignore parameters H_HDL and Cond_HDL.

PassWrap. The default value "Flat passivation" provides a flat passivation layer covering inductor turns. Setting this to "Conforming passivation" makes a thin passivation layer to conform to the cross-sectional contour as shown in the Cross-sectional View.

HP. This parameter is ignored if PassWrap="Flat passivation", otherwise HP provides the thickness of the passivation layer that covers the gap between turns. This model implies that the passivation layer has the same thickness at the sides and bottom of the gap (see the Cross-sectional View).

UseAFS. This model can optionally use Advanced Frequency Sweep (UseAFS is the default setting). If UseAFS="NoAFS" than this model simulates at each frequency point from the specified frequency sweep. If UseAFS="UseAFS" then this model does not simulate at each frequency point; instead, it simulates at several automatically selected frequency points and uses results to obtain very accurate frequency-dependent approximation valid through entire frequency sweep. See the specifics regarding AFS interaction with disk cache in the "Implementation Details" section.

Parameter Restrictions and Recommendations

1. $1 \le NT \le 50$
2. There are no limits on T except possible aggravations due to overly thin T (see "Implementation Details"). Note that the conductor must stay confined within the passivation layer so the conductor thickness T must be less than 0.95*H1.

3. Selection PassWrap=Yes makes the passivation layer to conform the conductor contour (See Cross-sectional View). Note that PassWrap=Yes makes passivation thickness vary: on the conductor top it equals H1-T, at the valley between the conductors (both sides and bottom) the passivation thickness is Hp. Models limits the width of this valley (gap between dielectric walls) at 0.1 micron.
4. To exclude passivation, set MRFSUB parameters Er1=1 and Tand1=0. Assign a reasonable value to H1 that well exceeds T to avoid an overly thin air layer between the top of the conductor and the top of the mock "passivation" layer (see the "Implementation Details" section).
5. To model a suspended MEMS (micro-electromechanical systems) inductor, set Er2=1.
6. FMOCIND2 does not allow to use frequency-dependent model material parameter Cond_HDL and substrate material parameters Er1, Er2, Er3, Tand1, Tand2, Tand3, Sig3.

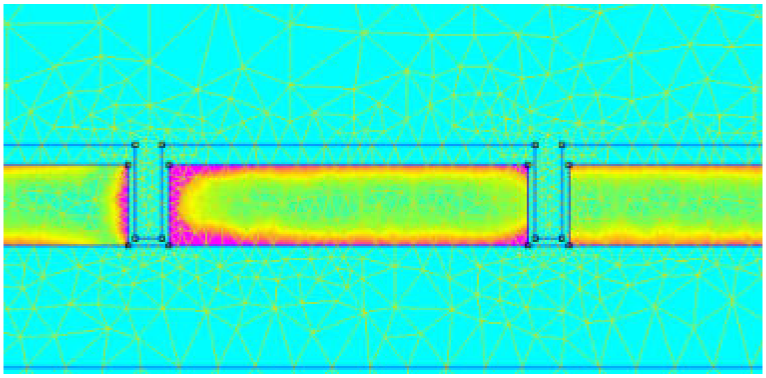
Implementation Details

Lateral grounds are optional and are implemented as PEC infinitely thin strips surrounding inductor (see Topology); user can place them arbitrary within insulation layer).

Passivation layer may be made either flat or conformal.

The 2D Finite Element Method (FEM) in conjunction with quasi-static problem formulation provides a very stable solution for RFICs most common dimensions and frequency range. An FEM engine is partially based on a FEMM solver ([[1] [10–42]); it comprises mesher ([2] [10–42]) and two independent solvers: An electrical solver and a magnetic solver. The mesher generates mesh that covers the entire cross-section including the cross-section of conductors. Usually, this method consumes a reasonable amount of computing resources; however, in certain situations memory consumption and run time may escalate because some specified dimensions force the mesher to generate an overly dense mesh. Commonly, this happens due to an error in parameter specifications that results in creation of extremely narrow layers.

The following figure demonstrates the typical distribution of current across a conductor cross-section (conforming passivation). Note that model provides mesh fine enough (and large enough) to reveal small details of skin effect and current crowding.



Disk cache and AFS.

Depending on the UseAFS setting, this model caches either model Y-matrices at each sweep frequency (UseAFS="NoAFS") or frequency-dependent approximation (UseAFS="UseAFS").

If UseAFS="NoAFS" then all frequencies in a sweep are the part of cache search criteria. This means that all subsequent runs of identical models at the same frequency sweep will use disk cache for speed up. However, if even one frequency differs from those in a cache (or one frequency added to sweep) model will run full blown simulation.

There is a certain specific in the way how model treats cache in case UseAFS="UseAFS". Model includes first and last (extreme) sweep frequencies along with model parameters into cache search criteria. This means that once approximation results had been written to cache all subsequent sweeps with arbitrary frequency step and identical extreme frequencies will be served with AFS approximation obtained from cache without need to spend time on running AFS again. However, any sweep with even one extreme frequency distinct from those in cache will demand new run of AFS engine.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See “Cell Options Dialog Box: Layout Tab ” for Cell Options dialog box **Layout** tab details.

See “The Layout Process File (LPF)” for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

Recommendations for Use

Computation time substantially grows with mesh size. Mesh inside conductors may heavily contribute into total mesh size because size of a mesh cell inside conductor is governed by value of skin depth at the highest simulation frequency. Mesh does not vary with frequency so the same fine mesh works at lower frequencies. Frequency-independent mesh results in improved stability and consistency but with some sacrifice in solution time.

NOTE: The implementation of FEM Quasi-Static models relies on temporary intermediate text files. This model creates these files temporarily in the project directory and subsequently deletes them. There may be several dozen files and they may take up to 100 MB of disk space, so make sure that your project hard drive has sufficient free space.

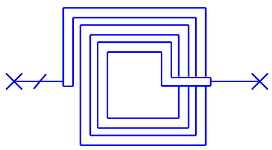
References

[1] David Meeker home page: http://femm.foster-miller.net/~dmeeker/ Follow this link for information: http://femm.foster-miller.net/Archives/readme.htm FEMM Manual: http://femm.foster-miller.net/Archives/doc/manual.pdf

[2] Jonathan Richard Shewchuk. Triangle. A Two-Dimensional Quality Mesh Generator and Delaunay Triangulator. Follow this link for information and download: http://www-2.cs.cmu.edu/~quake/triangle.html

(Obsolete) Rectangular Spiral RFIC Microstrip Inductor (FEM Quasi-Static): FMRIND

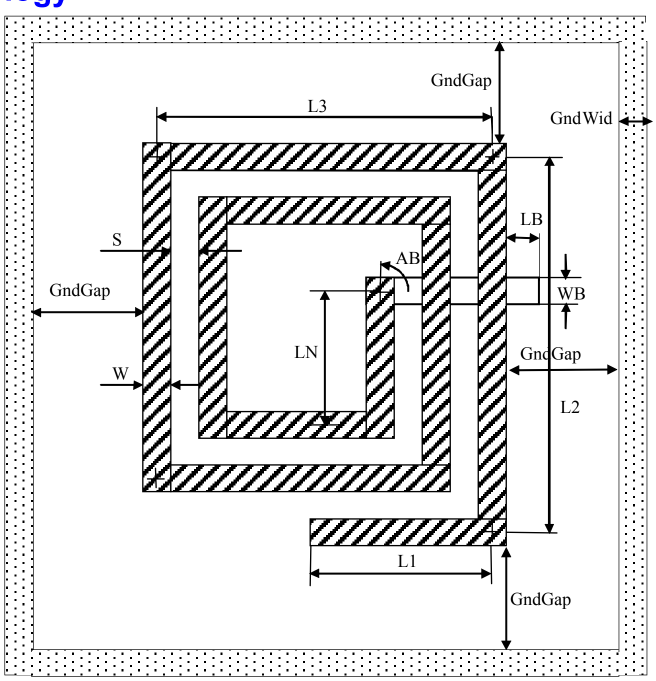
Symbol



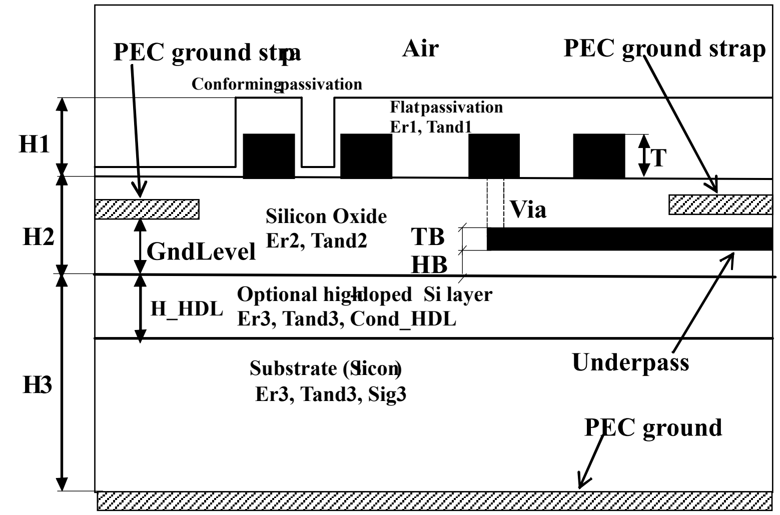
Summary

This element is OBSOLETE and is replaced by the Rectangular Spiral RFIC Inductor (FEM Quasi-Static) (FMRIND2) element. This circuit component models an RFIC microstrip rectangular inductor with a strip underpass. The inductor is located atop the insulating dielectric and covered with a passivation layer. This model is based on an evaluation of self and mutual inductances, capacitances, admittances, and resistances between all segments. Calculation of these circuit parameters is based on an accurate 2D FEM quasi-static model of coupled microstrip lines arranged on/into the dielectric (silicon oxide) layer above the conducting substrate. This model accounts for the passivation layer, optional ground straps surrounding the inductor, and for the presence of a conducting (high-doped) layer between insulated layers and the substrate.

Topology



Cross-sectional View



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	MI1
NS	Number of linear segments (>=4)		15
L1	Length of first segment	Length	80 um
L2	Length of second segment	Length	155 um
L3	Length of third segment	Length	165 um
LN	Length of last segment	Length	35 um
AB	Angle of underpass departure		0 deg
W	Conductor width	Length	10 um
S	Conductor spacing	Length	5 um
WB	Width of underpass conductor	Length	10 um
HB	Height of underpass conductor above substrate	Length	2 um
LB	Length of underpass conductor extension beyond inductor	Length	0 um
TB	Thickness of underpass conductor	Length	1 um
RhoB	Underpass metal bulk resistivity normalized to gold		1
MRFSUB	Substrate definition	Text	MRFSUB1[a]
*IsGndStrap	Ground straps presence (Switch No/Yes)		No
*GndGap	Distance between ground strap and inductance edge	Length	10
*GndWid	Width of ground strap	Length	10
*GndLevel	Height of ground strap above substrate	Length	10
*IsHDLl	High-doped layer presence atop of substrate (Switch No/Yes)		No
*H_HDL	Bulk conductance of high-doped layer	Siemens/m	700

Name	Description	Unit Type	Default
*Cond_HDL	Height of high-doped layer	Length	2 um
*PassWrap	Flat/Conforming passivation (Switch Flat passivation/Conforming passivation)		Flat passivation
*HP	Thickness of conforming passivation layer between turns	Length	2 um

[a]Modify only if schematic contains multiple substrates. See the "Using Elements With Model Blocks" for details.

** indicates a secondary parameter*

Parameter Details

NS. The number of linear conductor segments forming an inductor. NS should be greater than 4 and less than NSMAX. The value of NSMAX can be evaluated from the condition LNMAX >0, where

LNMAX = L2-(NS-2)(W+S)/2 for even NS

LNMAX = L3-(NS-3)(W+S)/2 for odd NS

The layout feasibility check is run before performing calculations.

LN. The length of the last segment LN should not exceed LNMAX (see above). If the LN value is too large the model automatically sets LN to LNMAX and issues a warning. LN also should not be less than (W+WB)/2. If the LN value is too small the model automatically sets LN to (W+WB)/2 and issues a warning.

AB. The angle AB (degrees) defines the direction of underpass departure from the end of the last segment. Only 0, 90, 180 and 270 are allowed for AB. The Zero angle has an underpass that is parallel to L1 and goes to the opposite direction. The angle is measured counterclockwise. Any intermediate value of AB is set to the closest acceptable value.

The underpass is not allowed *to overlap the last segment*. If this happens, the model changes AB so that the underpass departs in the opposite direction.

Important: The layout cell overrides the setting for AB and sets the underpass exit to the opposite direction if overlapping occurs.

HB. The height of the underpass is the distance between the upper substrate boundary (the top of the high-doped layer if present) and the bottom of the underpass conductor.

LB. This is the length of the part of underpass conductor that sticks out from the external edge of conducting turns.

MRFSUB. See the MRFSUB documentation. Note that notations H1, H2, H3, Er1, Er2, Er3, Tand1, Tand2, Tand3, Sig3 on the cross-sectional view above are MRFSUB respective parameters.

Out90deg (Layout cell parameter only): Note that the layout cell of this model has an Out90deg parameter (to edit values of this parameter select the corresponding layout cell, right-click on it and choose **Shape Properties >Parameters**). Setting this parameter to a nonzero value means that the orientation of a face at port 2 provides connection to an external circuit via a right (90deg) bend. Correspondingly, setting this parameter to zero means that the orientation of a face at port 2 provides an"in line" (no bend) connection to an external circuit. The default value is zero. Setting it to a nonzero value (for example, to 1) doesn't affect the electrical properties of the model; no bend component is added automatically. You can attach any bend model to port 2 if needed.

HP. This parameter is ignored if PassWrap="Flat passivation", else HP provides thickness of passivation layer that covers the gap between turns. Model implies that passivation layer has the same thickens at sides and bottom of the gap (Cross-sectional View).

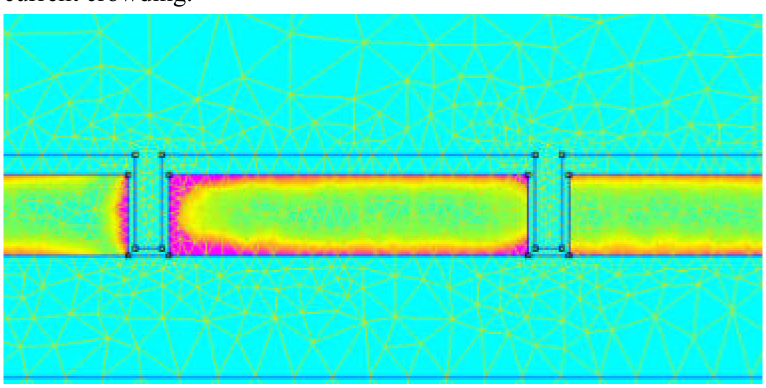
Parameter Restrictions and Recommendations

1. NS should be greater than 4 and less than NSMAX. The value of NSMAX can be evaluated from condition LNMAX >0 (see "Parameter Details").
2. There are no limits on T except possible aggravations due to an overly thin T (see "Implementation Details"). Note that the conductor must stay within the passivation layer so the conductor thickness T must be less than 0.95*H1.
3. Selection PassWrap=Yes makes the passivation layer skirt the conductor contour. Note that when PassWrap=Yes the passivation thickness varies: on the conductor top it equals H1-T; at the valley between conductors (both sides and bottom) passivation thickness is 1/3*(H1-T).
4. To exclude passivation, set MRFSUB parameters Er1=1 and Tand1=0. Set H1 large enough (H1>>T) to avoid an overly thin air layer between the top of the conductor and the top of the mock "passivation" layer (see "Implementation Details").
5. To model a suspended MEMS (micro-electromechanical systems) inductor, set Er2=1.

Implementation Details

2D Finite Element Method (FEM) in conjunction with quasi-static problem formulation provides a very stable solution for RFICs most common dimensions and frequency range. The FEM engine is partially based on the FEMM solver ([1] [10–47]); it comprises a mesher ([2] [10–47]) and two independent solvers: an electrical solver and a magnetic solver. The mesher generates a mesh that covers the entire cross-section including the cross-section of conductors. Usually, this method consumes a reasonable amount of computing resources; however, in certain situations memory consumption and run time may escalate because some specified dimensions force the mesher to generate overly dense mesh. Commonly, this happens due to an error in parameter specifications that results in creation of extremely narrow layers.

The following figure demonstrates the typical distribution of current across conductors cross-section (conforming passivation). Note that the model provides mesh fine enough (and large enough) to reveal small details of skin effect and current crowding.



Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See "Cell Options Dialog Box: Layout Tab " for Cell Options dialog box **Layout** tab details.

See "The Layout Process File (LPF)" for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

Recommendations for Use

Computation time substantially grows with mesh size. Mesh inside conductors may heavily contribute into total mesh size because size of a mesh cell inside conductor is governed by value of skin depth at the highest simulation frequency. Mesh does not vary with frequency so the same fine mesh works at lower frequencies. Frequency-independent mesh results in improved stability and consistency but with some sacrifice in solution time.

NOTE: The implementation of FEM Quasi-Static models relies on temporary intermediate text files. This model creates these files temporarily in the project directory and subsequently deletes them. There may be several dozen files and they may use up to 100 MB of disk space, so ensure that your hard drive has sufficient free space.

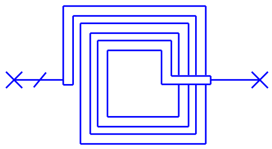
References

David Meeker home page: http://femm.foster-miller.net/~dmeeker/ For more information: http://femm.foster-miller.net/Archives/readme.htm For the FEMM Manual: http://femm.foster-miller.net/Archives/doc/manual.pdf

Jonathan Richard Shewchuk. Triangle. A Two-Dimensional Quality Mesh Generator and Delaunay Triangulator. For information and download: http://www-2.cs.cmu.edu/~quake/triangle.html

Rectangular Spiral RFIC Inductor (FEM Quasi-Static): FMRIND2

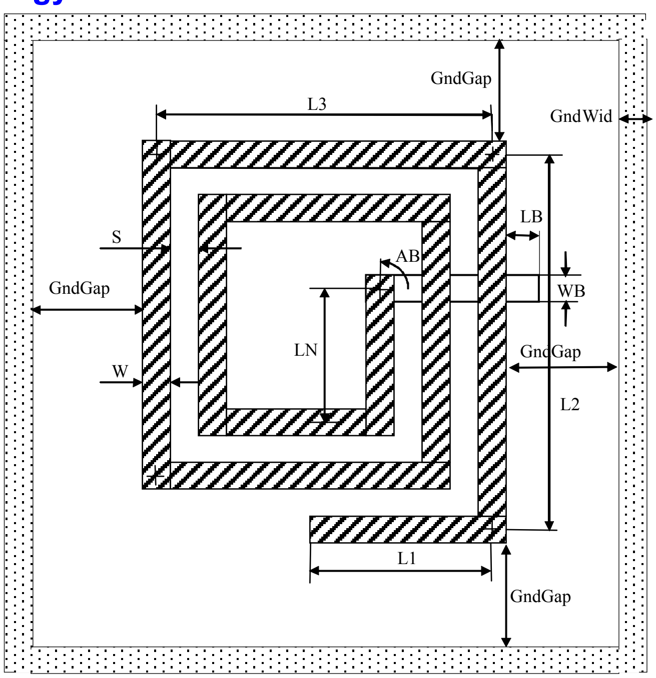
Symbol



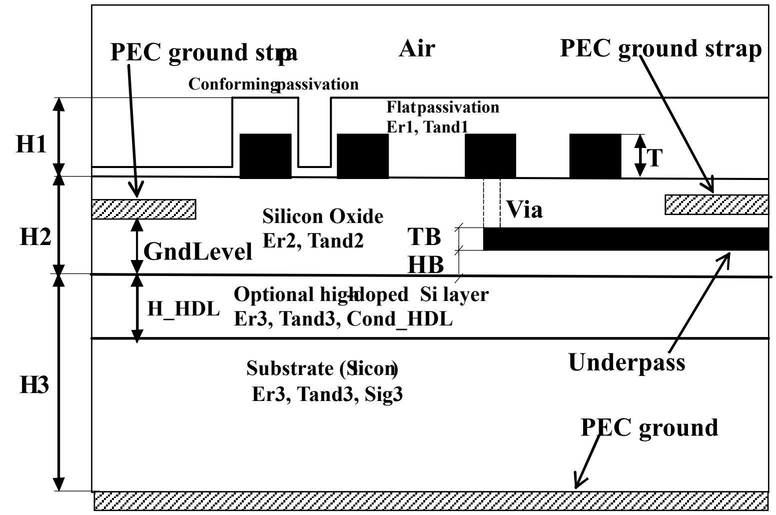
Summary

This circuit component models an RFIC microstrip rectangular inductor with a strip underpass. The inductor is located atop the insulating dielectric and covered with a passivation layer. This model is based on an evaluation of self and mutual inductances, capacitances, admittances, and resistances between all segments. Calculation of these circuit parameters is based on an accurate 2D FEM quasi-static model of coupled microstrip lines arranged on/into the dielectric (silicon oxide) layer above the conducting substrate. This model accounts for the passivation layer, optional ground straps surrounding the inductor, and for the presence of a conducting (high-doped) layer between insulated layers and the substrate. Simulation speed-up tools include disk cache and AFS (Advanced Frequency Sweep).

Topology



Cross-sectional View



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	MI1
NS	Number of linear segments (>=4)		15
L1	Length of first segment	Length	80 um
L2	Length of second segment	Length	155 um
L3	Length of third segment	Length	165 um
LN	Length of last segment	Length	35 um
AB	Angle of underpass departure	Angle	0 deg
W	Conductor width	Length	10 um
S	Conductor spacing	Length	5 um
WB	Width of underpass conductor	Length	10 um
HB	Height of underpass conductor above substrate	Length	2 um
LB	Length of underpass conductor extension beyond inductor	Length	0 um
TB	Thickness of underpass conductor	Length	1 um
RhoB	Underpass metal bulk resistivity normalized to gold		1
MRFSUB	Substrate definition	Text	MRFSUB1[a]
*IsGndStrap	Ground straps presence (No/Yes)		No
*GndGap	Distance between ground strap and inductance edge	Length	10
*GndWid	Width of ground strap	Length	10
*GndLevel	Height of ground strap above substrate	Length	10
*IsHDL1	High-doped layer presence atop of substrate (No/Yes)		No
*Cond_HDL	Bulk conductance of high-doped layer	Siemens/m	700

Name	Description	Unit Type	Default
*H_HDL	Height of high-doped layer	Length	2 um
*PassWrap	Flat/Conforming passivation (Flat passivation/Conforming passivation)		Flat passivation
*HP	Thickness of conforming passivation layer between turns	Length	2 um
UseAFS	Use/Do not use AFS for model speed up (UseAFS/NoAFS)		UseAFS

[a]Modify only if the schematic contains multiple substrates. See "Using Elements With Model Blocks" for details.

** indicates a secondary parameter*

Parameter Details

NS. The number of linear conductor segments forming an inductor. NS should be greater than 4 and less than NSMAX. The value of NSMAX can be evaluated from the condition LNMAX >0, where

LNMAX = L2-(NS-2)(W+S)/2 for even NS

LNMAX = L3-(NS-3)(W+S)/2 for odd NS

The layout feasibility check is run before performing calculations.

LN. The length of the last segment LN should not exceed LNMAX (see LNMAX). If the LN value is too large the model automatically sets LN to LNMAX and issues a warning. LN also should not be less than (W+WB)/2. If the LN value is too small the model automatically sets LN to (W+WB)/2 and issues a warning.

AB. The angle AB (degrees) defines the direction of underpass departure from the end of the last segment. Only 0, 90, 180 and 270 are allowed for AB. The Zero angle has an underpass that is parallel to L1 and goes in the opposite direction. The angle is measured counterclockwise. Any intermediate value of AB is set to the closest acceptable value.

The underpass is not allowed to overlap the last segment. If this occurs, the model changes AB so that the underpass departs in the opposite direction. **NOTE:**: The layout cell overrides the setting for AB and sets the underpass exit to the opposite direction if overlapping occurs.

HB. The height of the underpass is the distance between the upper substrate boundary (the top of the high-doped layer if present) and the bottom of the underpass conductor.

LB. The length of the part of underpass conductor that sticks out from the external edge of conducting turns.

MRFSUB. See MRFSUB. Note that notations H1, H2, H3, Er1, Er2, Er3, Tand1, Tand2, Tand3, and Sig3 in "Cross-sectional View" are MRFSUB respective parameters.

Out90deg (Layout cell parameter only): Note that the layout cell of this model has an Out90deg parameter (to edit these parameter values select the corresponding layout cell, right-click and choose **Shape Properties** to display the Cell Options dialog box, then click the **Parameters** tab). Setting this parameter to a nonzero value means that the orientation of a face at port 2 provides a connection to an external circuit via a right (90deg) bend. Correspondingly, setting this parameter to zero means that the orientation of a face at port 2 provides an "in line" (no bend) connection to an external circuit. The default value is zero. Setting it to a nonzero value (for example, to 1) does not affect the electrical properties of the model; no bend component is added automatically. You can attach any bend model to port 2 if needed.

HP. This parameter is ignored if the PassWrap parameter is "Flat passivation", otherwise HP provides the thickness of the passivation layer that covers the gap between turns. This model implies that the passivation layer has the same thickness at the sides and bottom of the gap (See the "Cross-sectional View" section).

UseAFS. This model can optionally use Advanced Frequency Sweep ("UseAFS" is the default). If UseAFS is set to "NoAFS" then this model simulates at each frequency point from the specified frequency sweep. If UseAFS is set to "UseAFS", then this model does not simulate at each frequency point; instead, it simulates at several automatically selected frequency points and uses results to obtain a very accurate frequency-dependent approximation valid through the entire frequency sweep. For information about AFS interaction with a disk cache, see the "Implementation Details" section.

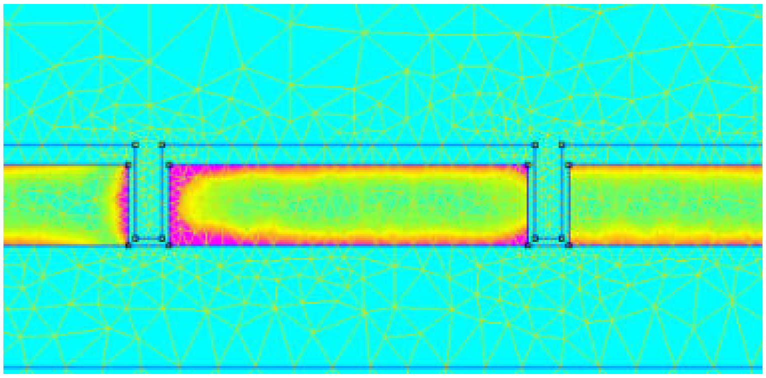
Parameter Restrictions and Recommendations

1. NS should be greater than 4 and less than NSMAX. The value of NSMAX can be evaluated from the condition LNMAX >0 (see the "Parameter Details" section).
2. There are no limits on T except possible aggravations due to an overly thin T (see the "Implementation Details" section). Note that the conductor must stay within the passivation layer so the conductor thickness T must be less than 0.95*H1.
3. When PassWrap is set to Yes, the passivation layer skirts the conductor contour and the passivation thickness varies: on the conductor top it equals H1-T; at the valley between conductors (both sides and bottom) the passivation thickness is 1/3*(H1-T).
4. To exclude passivation, set MRFSUB parameters Er1 to 1 and Tand1 to 0. Set H1 large enough (H1>>T) to avoid an overly thin air layer between the top of the conductor and the top of the mock "passivation" layer (see the "Implementation Details" section).
5. To model a suspended MEMS (micro-electromechanical systems) inductor, set Er2 to 1.
6. FMRIND2 does not allow the use of the Cond_HDL frequency-dependent model material parameter and substrate material parameters Er1, Er2, Er3, Tand1, Tand2, Tand3, and Sig3.

Implementation Details

2D Finite Element Method (FEM) in conjunction with quasi-static problem formulation provides a very stable solution for RFICs most common dimensions and frequency range. The FEM engine is partially based on the FEMM solver ([1] [10–53]); it comprises a mesher ([2] [10–53]) and two independent solvers: an electrical solver and a magnetic solver. The mesher generates a mesh that covers the entire cross-section including the cross-section of conductors. Usually, this method consumes a reasonable amount of computing resources; however, in certain situations memory consumption and run time may escalate because some specified dimensions force the mesher to generate overly dense mesh. Commonly, this happens due to an error in parameter specifications that results in creation of extremely narrow layers.

The following figure demonstrates the typical distribution of current across conductors cross-section (conforming passivation). Note that the model provides mesh fine enough (and large enough) to reveal small details of skin effect and current crowding.



Disk Cache and AFS.

Depending on the UseAFS setting, FMRIND2 caches either model Y-matrices at each sweep frequency (UseAFS is "NoAFS") or frequency-dependent approximation (UseAFS is "UseAFS").

If UseAFS is "NoAFS" then all frequencies in a sweep are part of the cache search criteria. This means that all subsequent runs of identical models at the same frequency sweep use disk cache for increased speed. However, if even one frequency differs from those in a cache (or one frequency is added to sweep) this model runs a full-blown simulation.

When UseAFS is "UseAFS", this model includes the first and last (extreme) sweep frequencies along with the model parameters in the cache search criteria. This means that once approximation results are written to the cache, all subsequent sweeps with arbitrary frequency step and identical extreme frequencies are served with AFS approximation obtained from the cache without needing to run AFS again. However, any sweep with even one extreme frequency distinct from those in the cache initiates a new run of the AFS engine.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See “Cell Options Dialog Box: Layout Tab ” for Cell Options dialog box **Layout** tab details.

See “The Layout Process File (LPF)” for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

Recommendations for Use

Computation time substantially grows with mesh size. Mesh inside conductors may heavily contribute to the total mesh size because the size of a mesh cell inside a conductor is governed by the value of the skin depth at the highest simulation frequency. Mesh does not vary with frequency, so the same fine mesh works at lower frequencies. Frequency-independent mesh results in improved stability and consistency, but with some sacrifice in solution time.

NOTE: The implementation of FEM Quasi-Static models relies on temporary intermediate text files. This model creates these files temporarily in the project directory and subsequently deletes them. There may be several dozen files and they may use up to 100 MB of disk space, so ensure that your hard drive has sufficient free space.

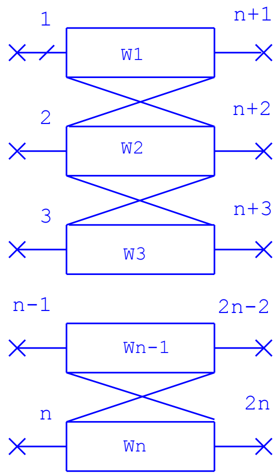
References

[1] David Meeker home page: http://femm.foster-miller.net/~dmeeker/ Follow this link for information: http://femm.foster-miller.net/Archives/readme.htm FEMM Manual: http://femm.foster-miller.net/Archives/doc/manual.pdf

[2] Jonathan Richard Shewchuk. Triangle. A Two-Dimensional Quality Mesh Generator and Delaunay Triangulator. Follow this link for information and download: http://www-2.cs.cmu.edu/~quake/triangle.html

General Multilayer Conducting Substrate Coupled Lines (FEM Quasi-Static): GFMCLIN

Symbol



Summary

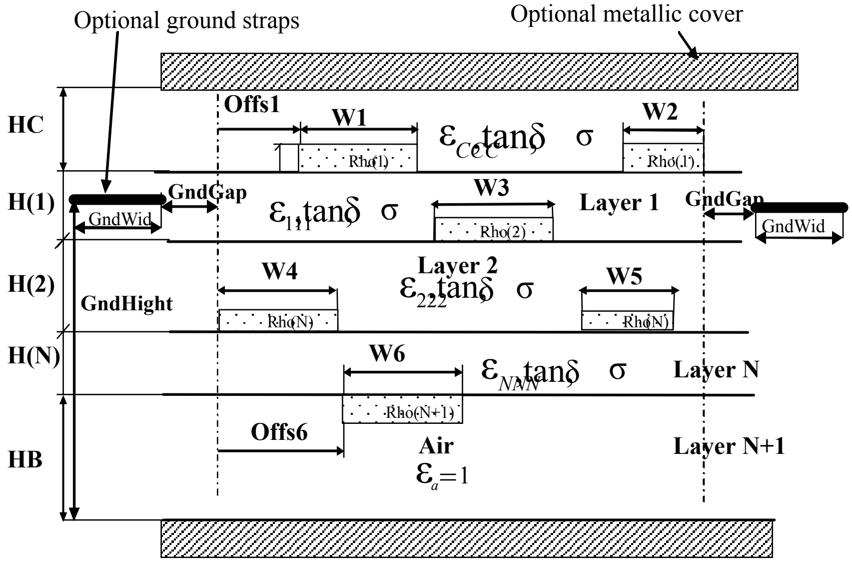
GFMCLIN models a section of several (1 to 50) edge- and/or offset-broadside- coupled microstrip lines arranged within unshielded or shielded (optional upper cover) stratified inhomogeneous substrate. Substrate layers have arbitrary heights and are made of various materials; optionally the substrate may be suspended. A backing ground plane is always present. GFMCLIN can account for the presence of optional metallic side walls in conjunction with a metallic cover, and for the trapezoidal shape of conductors cross-section. You can apply a solder mask over the top layer of conductors. GFMCLIN can also model a surface finish made of metal with magnetic properties, and account for the presence of magnetic substrate layers. Model has option of modeling frequency dependence of substrate permeability. Simulation speed-up tools include disk cache and AFS (Advanced Frequency Sweep).

GFMCLIN implements the Finite Element Method (FEM) quasi-static modeling technique. GFMCLIN is a dynamic or scalable model; it accepts a number of lines as input so the model and its schematic symbol expands/shrinks as the number of lines increases/decreases.

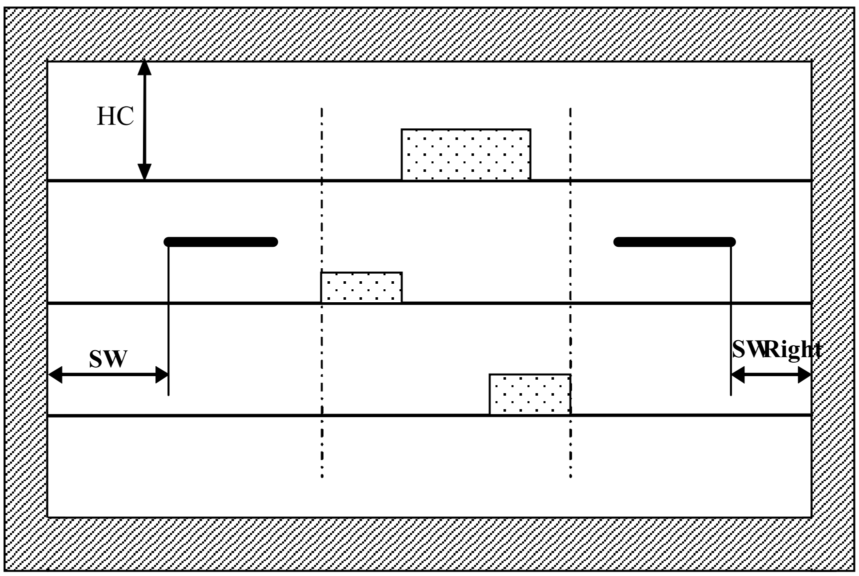
GFMCLIN differs from GMCLIN as follows:

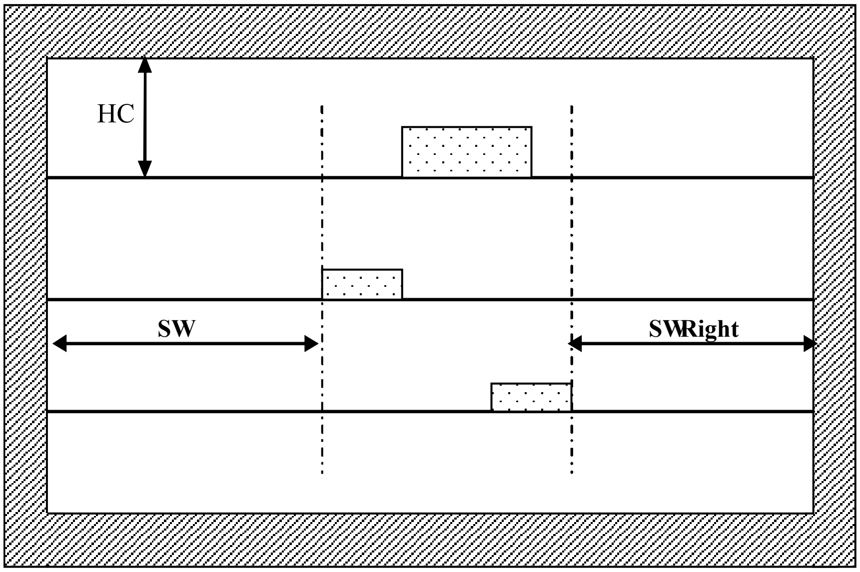
- Allows you to specify conductance for dielectric layers.
- Allows you to specify bulk resistivity for each metal layer (GMCLIN assigns identical Rho to all metal layers).
- Allows you to place side walls of metallic enclosure at unequal distances from a conductor structure.
- Accounts for actual distribution of current density inside a conductor.
- Solves for line parameters at each frequency from frequency sweep (GMCLIN interpolates frequency behavior between higher and lower frequency points).
- Allows placement of additional lateral (coplanar) grounds to model an RFIC environment.
- Allows modeling of magnetic substrates with frequency dependent permeability of substrate layers as well as magnetic surface finish.

Topology

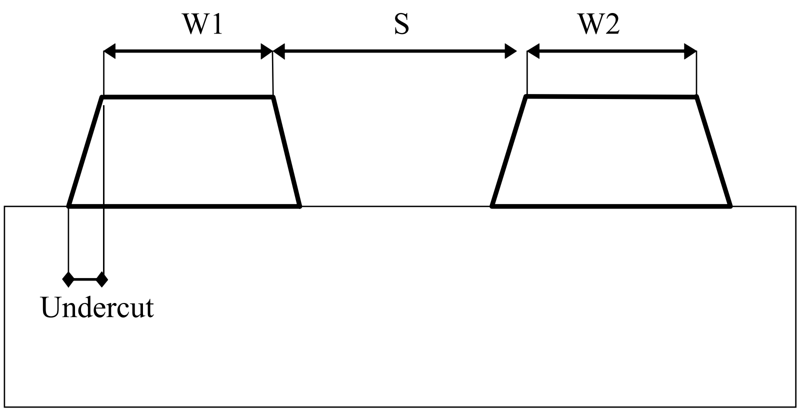


Definitions of SW and SWRight depend on the presence/absence of ground straps.

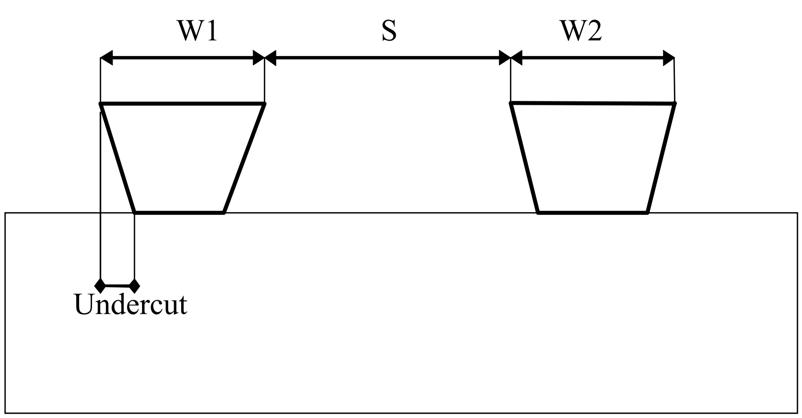




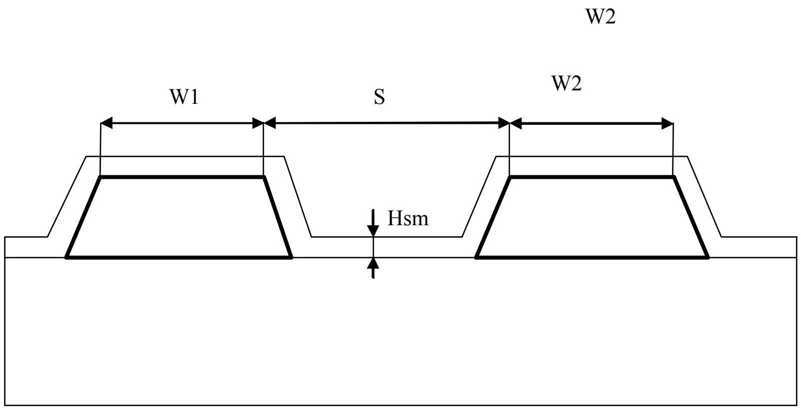
Definition of positive etch undercut (spacing S is defined as the difference between offsets).



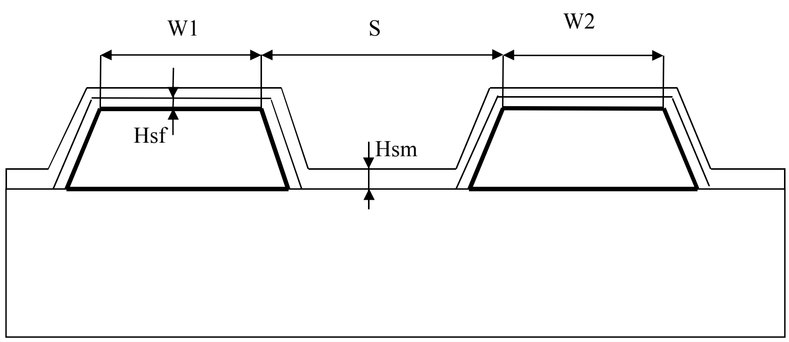
Definition of negative etch undercut (spacing S is defined as the difference between offsets).



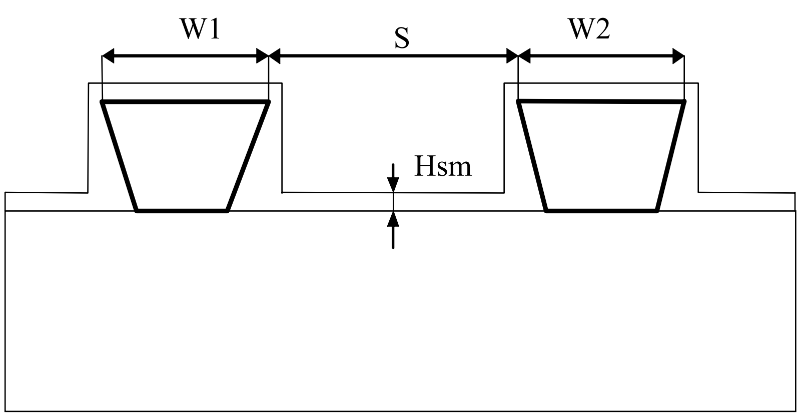
Configuration of optional conformal solder mask. Case undercut≥0.



Configuration of optional conformal solder mask over the surface finish metal. Case undercut≥0.



Configuration of optional conformal solder mask. Case undercut<0.



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	TL1
N	Number of conductors		2
L	Conductor length	Length	L[a]

Name	Description	Unit Type	Default
GMSUB	Substrate Definition	Text	GMSUB1[b]
*IsGndStrap	Switch "IsGndStrap"=No Ground Straps/Ground Straps		"No Ground straps"
*GndGap	Distance between ground strap and adjacent conductor	Length	10 micron
*GndWid	Width of ground strap	Length	10 micron
*GndHeight	Height of ground strap above backing ground	Length	1 micron
*SaveToFile	Switch "Save to txt file"=Yes/No		"No"
*FileName	Name of text file with computed model parameters	String	Same as model name
UseAFS	Switch "UseAFS"=UseAFS/NoAFS		"UseAFS"
*UseUnd	Switch "UseUnd"=No undercut/Use undercut		"No undercut"
*IsSldMask	Switch "IsSldMask"=No conformal solder mask/Conformal solder mask		"No conformal solder mask"
*Hsmt	Thickness of conformal solder mask layer	Length	10 micron
*Ersm	Relative dielectric constant of conformal solder mask layer		1
*TDsm	Loss tangent of solder mask layer		0
*IsSurFin	Switch "IsSldMask"=No surface finish/Yes surface finish mask		"No surface finish"
*Hsf	Thickness of surface finish metal	Length	10 micron
*Rhosf	Resistivity of surface finish metal normalized to gold		1
*MusfP	Real part of relative permeability of surface finish metal		1
*MusfPP	Imaginary part of relative permeability of surface finish metal		0
*IsMagSub	Switch "IsMagSub"=Not magnetic substrate/Magnetic substrate		"Not magnetic substrate"
*FreqProf	Frequency sweep of substrate relative Mu		1 GHz
*SubMurF	Frequency profile of substrate relative Mu (real part)		1
*SubMuiF	Frequency profile of substrate relative Mu (imaginary part)		1
*SubCntrl	Permissions to apply Mu frequency profile to substrate layers		0
*IsSubF	Switch "IsSubF" = Substrate Mu constant vs frequency/Substrate Mu varies vs frequency		"Substrate Mu constant vs frequency"
Wi, i=1..nn-number of lines	Width of conductor No i	Length	W[a]
Offsi, i=1..nn-number of lines	Offset of conductor No i	Length	W[a]
CLi, i=1..nn-number of lines	Number of layer containing conductor No i		1

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See "Intelligent Cells (iCells)" for details.

[b]Modify only if the schematic contains multiple substrates. See "Using Elements With Model Blocks" for details.

** indicates a secondary parameter*

Parameter Details

See GMnCLIN and GMCLIN for a detailed description of the majority of the GFMCLIN parameters. Parameters specific to GFMCLIN (or used differently) are described as follows:

GMSUB: RhoV. GFMCLIN uses the Rho parameter differently than the GMnCLIN and GMCLIN models do. Instead, GFMCLIN gets information about conductor resistivity from the vector parameter RhoV. This parameter contains bulk resistivity (relative to gold) for the conductor metal specified for each dielectric layer. GFMCLIN waits for the RhoV vector to contain an entry for each dielectric layer, even for layers that do not carry conductors. Thus, the length of the RhoV vector must be equal to N if the substrate is grounded. If the substrate is suspended, RhoV must have N+1 entries.

GMSUB: Sigma. This vector parameter specifies the conductances of materials used for dielectric layers. Sigma must supply conductance for each dielectric layer (except the suspended substrate air layer under the substrate, so the total number of entries for Sigma must always be N).

GMSUB: SigmaC. Specifies the conductance of the dielectric that makes the layer under the metallic cover (if present).

GMSUB: SW and SWRight. GFMCLIN uses these parameters only if the GMSUB Cover parameter is "Metallic Box"; all other values of Cover make SW and SWRight irrelevant. See the last two figures under "Topology" for more information on how GFMCLIN interprets SW and SWRight.

If IsGndStrap is "No Ground Straps," GFMCLIN considers SW as the distance from the left edge of the leftmost conductor to the left sidewall of the metallic box/enclosure. SWRight is the distance from the right edge of the rightmost conductor to the right sidewall.

If IsGndStrap is "Ground Straps," GFMCLIN interprets SW as the distance from the left edge of the left ground strap to the left sidewall of the metallic box/enclosure. SWRight is the distance from the right edge of the right ground strap to the right sidewall.

Note that the SW and SWRight parameters may have different values (GMnCLIN considers the distances to the left and right side walls identical and equal to SW).

GMSUB: Undercut parameter. GFMCLIN optionally uses this vector parameter to apply its values as etch undercuts to conductors on each layer. Etch undercut specifies non-rectangular (trapezoidal) distortion of conductor cross-section due to manufacturing errors. Undercut may take positive, zero, and negative values (see the figures in "Topology"). All conductors that belong to the same layer have an identical undercut equal to the corresponding entry of vector Undercut. Note that conductor width is always defined as the top side of a cross-sectional trapezoid. Spacing between adjacent conductors is measured between the closest top vertices of respective trapezoids. See the restriction on the Undercut value in "Parameter Restrictions and Recommendations". Note that the vector parameter Undercut must contain exactly N entries if the GMSUB Gnd parameter is "Grounded Substrate", and exactly N+1 entries if Gnd is "Suspended Substrate". This model does not check length and does not use the contents of the GMSUB Undercut parameter if the UseUnd parameter is "No undercut".

GMSUB: MuP, MuPP. These vector parameters specify the real (MuP) and imaginary (MuPP) parts of permeabilities of materials used for dielectric layers. If control parameter IsMagSub = "Non-magnetic substrate" (default) than parameters MuP and MuPP are not used. If control parameter IsMagSub = "Magnetic substrate" then MuP and MuPP must supply permeabilities for each dielectric layer (except the suspended substrate air layer under the substrate, so the number of entries for MuP and MuPP must always be N).

OffsX. The relative horizontal offset of the conductor #X left edge from the left edge of the left-most conductor. All offsets must be positive. Offset of the left-most conductor (or conductors) may be set to zero. GFMCLIN looks for the smallest offset and subtracts it from other offsets. The left edge of the left-most conductors marks a reference plane to determine the distance to lateral ground straps and optional metallic sidewalls.

CL1. Specifies the number of the layer that carries the conductor on top of it. If the conductor protrudes upward into layer #m (it occupies the lower part of layer #m but sits on layer #m+1), you must set CL1 to m+1. If the conductor has a negative thickness and is recessed into layer #m (it occupies the upper part of layer #m), you must set CL1 to m.

IsGndStrap. This switch either places ("Ground Straps") or does not place ("No Ground Straps") identical perfectly conducting straps (lateral grounds) at both sides of the conductor structure. The default is "No Ground Straps".

GndGap. The distance between the left edge of the left-most (or right edge of the right-most) conductor and lateral ground strap (if present). If IsGndStrap is set to "No Ground Strap" the value of this parameter is irrelevant.

GndWid. The width of the lateral ground straps (if present). If IsGndStrap is set to "No Ground Strap" the value of this parameter is irrelevant.

GndHeight. The height of the lateral ground straps (if present) above the backing ground. If IsGndStrap is set to "No Ground Strap" the value of this parameter is irrelevant.

SaveToFile. This parameter is hidden by default and set to No. You can toggle the parameter to Yes. If you select Yes, GFMCLIN creates a text file named *default model_name.txt* at the current project location. This text file contains a table of values of RLGC line parameters at each project frequency. Each row contains RLGC values computed at the frequency specified in the first column (frequency in GHz, R in ohms/m, L in H/m, G in S/m, C in F/m).

The structure of this text file is essentially the same if N>2 (see the description of the text file structure for N=1 in GM1LIN). The model outputs RLGC matrices to columns that immediately follow the frequency column. Entries of each matrix are placed column-wise; the first column is first: R11, R21, R31.. Rn1; and then the second column: R12, R22, R32,..Rn2 and so on. The total number of columns in the file is 4*n*n+1, where n is the number of lines.

GFMCLIN at N=2 implies the existence of two modes, namely, C and P (see [2] [10–67]), and places additional columns in the output text file. Note that if a system of coupled lines is fully symmetrical (as it might be for edge-coupled microstrip lines; whereas broadside coupled microstrip lines are always asymmetrical), C mode corresponds to even mode and P mode corresponds to odd mode.

GFMCLIN at N=2 outputs complex characteristic impedances Re(Zc1), Im(Zc1), Re(Zp1), Im(Zp1), Re(Zc2), Im(Zc2), Re(Zp2), Im(Zp2); complex effective dielectric constants Re(EeffC), Im(EeffC), Re(EeffP), Im(EeffP) (find traditional effective dielectric constants in columns 38, 39), and losses for C and P modes LossC (dB/m), LossP(dB/m) to columns 2-15. Entries of R, L, G, and C matrices are distributed among columns 16-31 in corresponding order (see above). Columns 32-37 contain Re(Rc), Im(Rc), Re(Rp), Im(Rp), BetaC, and BetaP; where Rc is ratio of C mode voltage in the second line to C mode voltage in the first line; Rp is the same ratio in P mode (see details in [2] [10–67], section 4.3.1) and BetaC and BetaP are propagation constants of C and P modes in Rad/m. Note that columns 38 and 39 contain traditional effective dielectric constants ErC_Eff and ErP_Eff (they do not account for losses). The total number of columns in the output text file is 39.

You can link or import the created text file to a project as a data file and view the frequency behavior of any of the above parameters using the proper data measurement. Note that the first column (frequency) is always in GHz, so these measurements might be incompatible with other MWO measurements placed on the same graph; you may prefer to place these data measurements on a separate dedicated graph.

FileName. By default this parameter is hidden and is set to *gfmclin.txt*. You can change the file name for each model instance to an arbitrary name with a length not exceeding 64 symbols.

UseAFS. This model can optionally use Advanced Frequency Sweep (UseAFS is the default setting of this switch). If UseAFS is "NoAFS" the model simulates at each frequency point from the specified frequency sweep. If UseAFS is "UseAFS" the model does not simulate at each frequency point. Instead, it simulates at several automatically selected frequency points and uses results to obtain very accurate frequency-dependent approximation for each entry of RLGC matrix valid through the entire frequency sweep. See "Implementation Details" for the specifics regarding Advance Frequency Sweep (AFS) interaction with disk cache. See also restriction on usage of UseAFS parameter with magnetics in "Parameter Restrictions and Recommendations".

UseUnd. This parameter specifies if the model uses the GMSUB vector parameter Undercut (previously described). The model does not check length and does not use the contents of the GMSUB Undercut parameter if the UseUnd parameter is "No undercut".

IsSldMask, Hsm, Ersm, TDsm. Conductors on top of a dielectric stack can be covered by a conformal dielectric layer that does not belong to the regular dielectric stack. This layer is called “solder mask,” a common term used in the PCB industry. Any material used as a conformal passivation layer fits as well. The IsSldMask parameter informs the model if the Hsm, Ersm, TDsm parameters are relevant or not. Note that a solder mask can be applied only to conductors that sit on top of a dielectric stack, and only if their thickness is positive (they are not recessed but stick out of the dielectric). If the model cannot find conductors eligible for the solder mask application it does not apply the solder mask. See restriction on Hsm value in "Parameter Restrictions and Recommendations".

IsSurFin, Hsf, Rhosf, MusfP, MusfPP. Conductors on top of a dielectric stack may have a protective coating called surface finish. Popular PCB surface finish ENIG (Electroless Nickel Immersion Gold) contains nickel that exhibits noticeable magnetic properties. GFMCLIN allows you to add a single protective metal layer to top conductors (for ENIG, the presence of a very thin top protective gold layer is neglected) if IsSurfFin = "Surface finish metal," and specify its parameters: thickness (Hsf), relative resistivity (Rhosf), and relative complex magnetic permeability MusfP+J*MusfPP (here P and PP stand for one and two primes - standard denotations for real and imaginary part of permeability). Note that surface finish can be applied only to conductors that sit on top of a dielectric stack, and only if their thickness is positive (they are not recessed but stick out upward from dielectric). If the model cannot find top conductors eligible for surface finish application it does not apply the surface finish at all. See also restriction on usage of UseAFS parameter with magnetics in "Parameter Restrictions and Recommendations".

IsMagSub. Informs the model if GMSUB parameters MuP and MuPP are relevant. If IsMagSub = "Magnetic substrate," then the model checks vectors MuP and Mupp (real and imaginary parts of relative permeabilities) for errors and applies their values to material properties of substrate dielectric layers. If IsMagSub = "Non-magnetic substrate" then the model applies default free space permeability values to all substrate dielectric layers. See also restriction on usage of UseAFS parameter with magnetics in "Parameter Restrictions and Recommendations".

FreqProf, SubMurF, SubMuiF, SubCntrl. These parameters provide frequency dependent permeability for selected substrate layers. Vector parameters FreqProf, SubMurF, and SubMuiF combined define Mu (permeability) frequency profile: FreqProf is a frequency sweep (valid only for substrate permeabilities), SubMurF and SubMuiF are values of real and imaginary parts of substrate permeabilities specified at FreqProf frequencies. Note that only sole Mu frequency profile is allowed, meaning that all layers to which this profile applies are assumed made of the same material. Entries of vector SubCntrl should be either 1 or 0: SubCntrl[k] =1 grants permission to apply Mu frequency profile to permeability of layer #k; SubCntrl[k] =0 prohibits to apply Mu frequency profile to permeability of layer #k.

Note that sizes of vectors SubMurF and SubMuiF must be equal to size of vector FreqProf; size of vector SubCntrl must be equal to number of substrate layers.

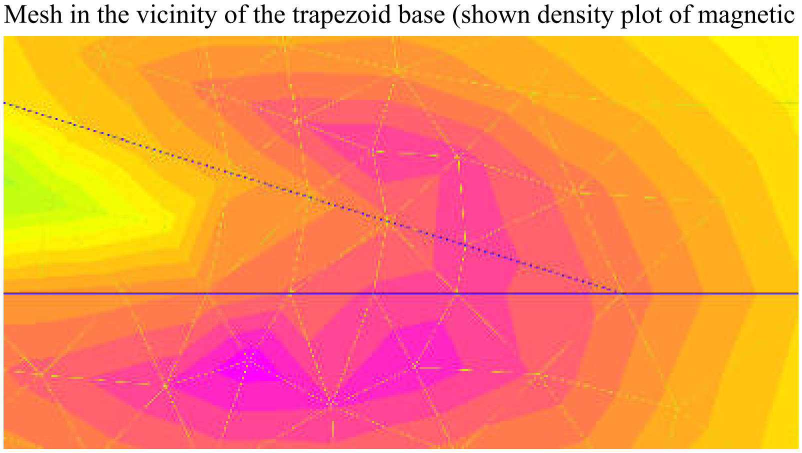
IsSubF. Informs the model if permeability frequency profile is relevant. If IsSubF="Substrate Mu constant vs frequency" then permeabilities of all substrate layers are assumed constant vs frequency. If IsSubF="Substrate Mu varies vs frequency" than model attempts to apply permeability frequency profile specified by parameters FreqProf, SubMurF, and SubMuiF to selected substrate layers; layer selection is provided by entries of parameter SubCntrl.

Parameter Restrictions and Recommendations

1. Total number of layers cannot exceed 30.

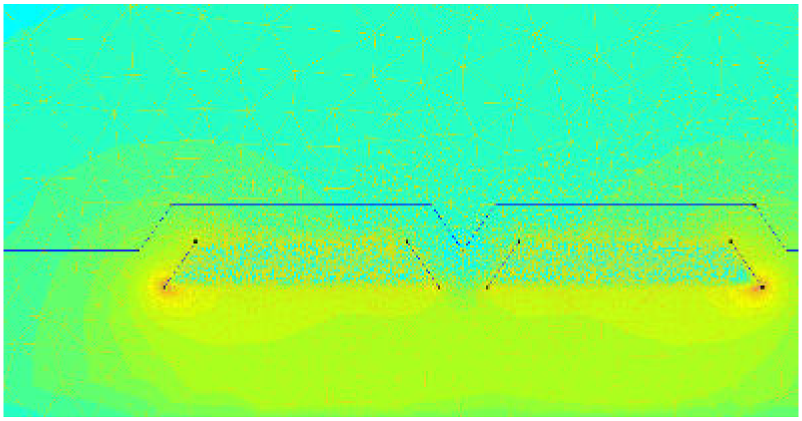
2. Number of conductors N cannot exceed 50.
3. See detailed information on parameter restrictions and recommendations in GMnCLIN under "Conductor Numbering and Disposition". The GMFCLIN, GMnCLIN, and GMCLIN models are very particular about correct conductor numbering.
4. Do not use very small (for example, below 0.1) values of RhoV (bulk resistivity of conductors in layers, vector parameter of GMSUB) and Rhosf (bulk resistivity of surface finish metal) because GFMCLIN cannot model perfect conductors and may have issues with metals of very low bulk resistivity. To provide high accuracy of modeling current distribution inside the conductors, maximum mesh cell size is proportional to the metal skin depth at the highest frequency of the frequency sweep. Skin depth is proportional to the square root of bulk resistivity so mesh may contain millions of cells for low resistivity metals. Extremely big mesh may result in a long simulation time and lead to other computational issues.
5. Do not use infinitely thin conductors and conductors with a thickness less than 0.1 micron.
6. Do not use excessively big (for example, above 20) values of real/imaginary parts of surface finish metal permeability MusfP and MusfPP, because GFMCLIN may have difficulties with very big mesh. To provide high accuracy of modeling current distribution inside the conductors, maximum mesh cell size is proportional to the metal skin depth at the highest frequency of the frequency sweep. If magnetic surface finish is present, skin depth is inversely proportional to the square root of surface finish metal permeability so large values of permeability may contribute to the creation of multimillion cell mesh. Extremely big mesh may result in a long simulation time and lead to other computational issues.
7. If UseAFS is "UseAFS" and GFMCLIN cannot obtain approximation data from a disk cache, AFS approximation is activated. AFS needs to perform a simulation at seven frequency points minimum, so a frequency sweep must have at least seven frequency points. If the number of frequency points is less than seven the model stops and issues a corresponding error message. If UseAFS is "UseAFS" and GFMCLIN successfully gets data from a disk cache, it uses cached approximation data at each specified frequency point even if the number of frequency points is less than seven. If UseAFS is "NoAFS" the model performs calculations at each frequency point.
8. Note that option UseAFS="UseAFS" is incompatible with use of magnetic properties (IsSurFin="Surface finish metal" and/or IsMagSub="Magnetic substrate"); model always simulates magnetics at each point of frequency sweep.
9. The absolute value of the undercut for each layer is capped at W/2 (where W is conductor width) if the undercut is negative. A positive undercut is limited by the maximum etch factor EF (defined as *EF=Undercut(i)/T* where *T* is conductor thickness, *Undercut(i)* is the *i*-th entry of vector parameter Undercut, and *i* is the layer number) at EF≤2.95. This value is defined by the abilities of the FEM mesher to cope with fine mesh that develops at small acute angle adjoining the trapezoid base.

   Mesh in the vicinity of the trapezoid base (shown density plot of magnetic field) where etch factor = 2.7.



10. Positive undercut also limits permissible values of minimal spacings between adjacent conductors on the same layer because conductor offsets (and accordingly, spacings) are measured to (between) **top cross-sectional vertices** (see the figures in "Topology"). GFMCLIN issues an error message if the distance between adjacent **base vertices** of neighboring conductors gets less that 0.1 micron. The same occurs for the minimum allowable values of the GndGap parameters and the GMSUB SW and SWRight parameters.

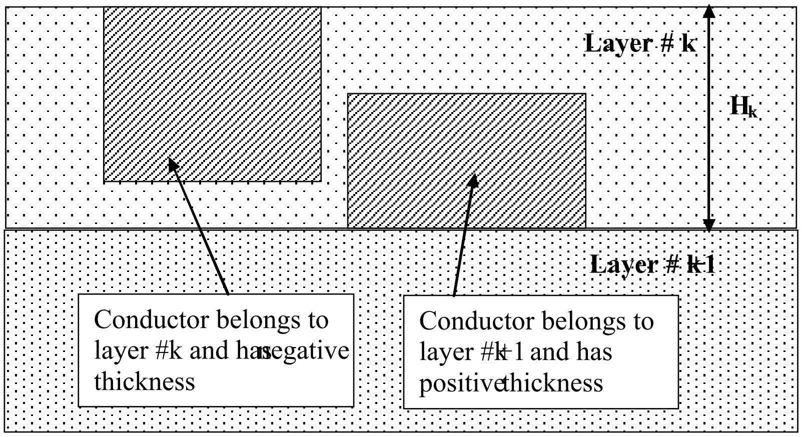
11. The application of a "solder mask" onto an existing top-layer layout may limit the minimal spacing allowed between adjacent conductors. This limitation comes from drawing limitations that imply an "inverse trapezoid" shape of the solder mask layer between conductors and disallows the "V-notch" shape when the top horizontal part of the solder mask layer between conductors vanishes (see the following figure). If said limitation is met, GFMCLIN issues an error message such as: "Spacing #1 in layer #1 must be greater than 0.1 micron (also check undercut and optional solder mask)." Note that if no solder mask parameters effect specified spacing, the error message does not mention the solder mask. Typically, this occurs with a thick solder mask layer and relatively small spacing specified and may be interpreted as a recommendation to substitute a flat layer for a conformal layer (to set IsSldMask to “No solder mask” and merely add a flat solder mask layer to dielectric stack).

    Limiting case (a V-notch in the solder mask layer with a horizontal stretch missing) for spacing between adjacent conductors if a solder mask is present.

12. If the thickness of the conformal mask layer Hsm exceeds five thicknesses of the top layer conductors (if any), the model issues a warning: "Conformal solder mask requested. Solder mask thickness Hsm is greater than 5T (five conductor thicknesses) so conformal fit is not feasible. Additional flat top layer may provide same results as conformal layer." This is an explicit recommendation to substitute a flat layer for a conformal layer, that is, to set IsSldMask to "No solder mask" and merely add a flat solder mask layer to the dielectric stack.

13. GMFCLIN allows "interdigital" positioning of conductors that belong to adjacent layers so that they all take up space within a common dielectric layer (see the following figure). GMCLIN does not allow this arrangement because it checks the total height of conductors and compares it to the height of the dielectric layer Hk into which they protrude, not paying attention to their lateral offset.



GFMCLIN allows this layout while GMnCLIN and GMCLIN consider it a user error.

14. Permeability frequency profile parameters FreqProf, SubMurF, and SubMuiF may be defined as data file(s) as well as scripting equations. Use equation functions col() or row () to read profile from data file. To use scripting equations you need to place scripting function calculating (and returning) vectors SubMurF and SubMuiF into the project scripting module named Equations. These functions can be referenced in schematic equations (see "Using Scripted Equation Functions").

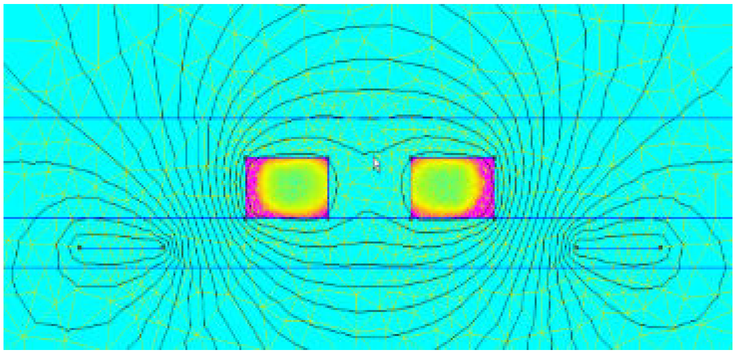
Implementation Details

GFMCLIN implementation is based on the FEM technique described in [1] [10–67] and applied to quasi-static approximation of Maxwell equations. It accounts for losses in metal and in substrate dielectric. Dispersion is included partly as frequency-dependence contribution that comes from losses (substrate polarization and eddy current loss, conductor resistive loss). Non-TEM dispersion is not included.

The 2D Finite Element Method (FEM) in conjunction with the quasi-static problem formulation provides a very stable solution for RFIC's most common dimensions and frequency range. The FEM engine is partially based on the FEMM solver ([3] [10–67]); it comprises a mesher ([4] [10–67]) and two independent solvers: an electrical solver and a magnetic solver. The mesher generates a mesh that covers the entire cross-section including a cross-section of conductors. Usually, this method consumes a reasonable amount of computing resources; however, in certain situations memory consumption and run time may increase because specified dimensions force the mesher to generate overly dense mesh. Commonly, this happens due to an error in the parameter specification that results in creation of extremely narrow layers.

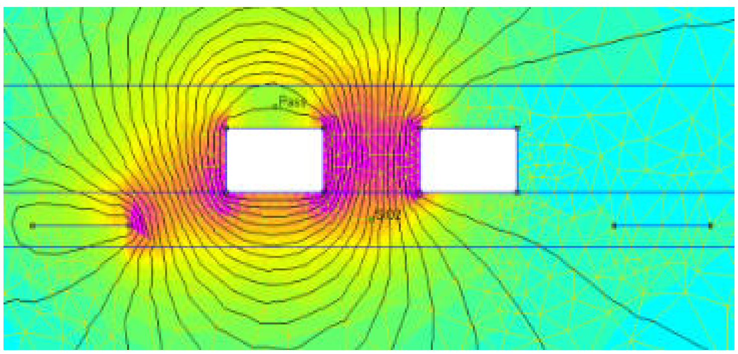
Another cause may also result in fine mesh. Generally, the densest mesh is concentrated in the vicinity (and within) conductors. This is intentional because a model accurately defines fields inside conductors and sets the smallest mesh cell size inside the conductor to a value proportional to either the skin depth of the conductor metal at the highest evaluation frequency or to the minimal cross-sectional dimension, whichever is less. If you specify, for example, a gold conductor with cross-sectional dimensions of 10x1 microns, at the highest frequency 20 GHz, you can presume the mesh will be of reasonable size. Setting the conductor width to 2000 microns or the highest frequency to 200 GHz may substantially hinder the modeling process, however.

The following figure demonstrates the typical distribution of an electric current across a conductor cross-section and contours of magnetic potential (conductors are at 1A of impressed current each). Note that this model provides mesh fine enough (and large enough) to reveal the small details of skin effect and current crowding.



The following figure demonstrates the distribution of an electric field and equipotential lines (conductors are at potentials 1 and 0). Note how the closely located ground strap affects field distribution.

Eventually, the model extracts RLGC per-unit-length matrices from computed electric and magnetic fields and uses them to obtain external circuit parameters.



Disk Cache and AFS

Depending on the setting of the UseAFS switch, the model caches either RLGC matrices at each sweep frequency (UseAFS is "NoAFS") or frequency-dependent approximations for each RLGC matrix entry (UseAFS is "UseAFS").

If UseAFS is "NoAFS" then all frequencies in a sweep are part of the cache search criteria and all subsequent runs of identical models at the same frequency sweep use disk cache for speed-up. However, if even one frequency differs from those in a cache (or one frequency added to a sweep) the model runs a full simulation.

When UseAFS is "UseAFS" the model includes first and last (extreme) sweep frequencies along with model parameters into cache search criteria. Once approximation results are written to cache, all subsequent sweeps with arbitrary frequency step and the same extreme frequencies are handled with AFS approximation obtained from the cache without needing to run AFS again. Any sweep with even one extreme frequency distinct from those in the cache results in running the AFS engine.

Layout

The project LPF must contain some structures with predefined names for each dielectric layer populated with conductors (note that every conductor has the number of the assigned dielectric layer specified in the respective parameter CLn, where n is the conductor number). Structure names must be based on the template ML_LINE_X, where X is equal to the number of the dielectric layer. All conductors that belong to layer X are comprised of material layers described in ML_LINE_X.

For example, if GMCLIN defines seven conductors, substrate GMSUB defines eight dielectric layers, and conductors are distributed over dielectric layers in accordance with the following table:

Conductor #	CL value
1	1
2	2
3	3
4	3
5	3
6	6
7	6

From the table you can see that conductor #1 sits on dielectric layer #1, conductor #2 sits on dielectric layer #2, conductors #3, #4, and #5 are on layer #3, and conductors #6 and #7 sit on layer #6. This arrangement indicates that only layers #1, 2, 3, and 6 are populated with conductors and you need to specify structures ML_LINE_1, ML_LINE_2, ML_LINE_3, and ML_LINE_6.

If a structure with the corresponding name is not found, the name of the missing structure is drawn on the error layer.

Note that the number of dielectric layers (and X in ML_LINE_X) defined in GMSUB cannot exceed 30.

Note also that the layout cell assigns to each conductor #n the linetype index equal to the value of the corresponding parameter CLn, that is equal to the number of the dielectric layer assigned to conductor #n.

Recommendations for Use

See the GMnCLIN "Recommendation for Use" section for details and usage examples.

Computation time substantially grows with mesh size. Mesh inside conductors may heavily contribute to total mesh size because the size of a mesh cell inside a conductor is governed by the value of skin depth at the highest simulation frequency. Mesh does not vary with frequency, so the same fine mesh works at lower frequencies. Frequency-independent mesh results in improved stability and consistency, but with some sacrifice in solution time.

Note that AFS may dramatically reduce simulation time for sweeps that contain hundreds or even thousands of frequency points (AFS is incompatible with use of magnetic properties).

NOTE: The implementation of FEM Quasi-Static models relies on temporary intermediate text files. This model creates these files temporarily in the project directory and subsequently deletes them. There may be several dozen files and they may use up to 100 MB of disk space, so ensure that your hard drive has sufficient free space.

Caution regarding units of data in saved text files: If a project that reads saved RLGC text files uses frequency, resistance, inductance, or conductance units different from GHz, Ohm, Henry, or Siemens, you may need to scale input values manually.

How to use frequency dependent substrate permeabilities (Mu frequency profile)

To use frequency dependent substrate permeabilities, first of all it is necessary to set general magnetic substrate permission: IsMagSub="Magnetic substrate". Then you need to set permission to apply Mu frequency profile IsSubF="Substrate Mu varies vs frequency". Fill entries of substrate vector parameters MuP and MuPP (Mu constant vs frequency) including entries for layers intended to use with frequency dependent permeabilities (just put any values for respective entries, for example MuP=1 and MuPP=0). For permitted (see details on SubCntrl role below) layers, profile values will overwrite values specified by MuP and MuPP anyway. Fill vector parameters that define Mu frequency profile: FreqProf, SubMurF, SubMuiF, SubCntrl. You should fill out all SubCntrl entries (number of SubCntrl entries must be equal to the number of all substrate layers). Non-zero value of SubCntrl entry #k is a permission to overwrite permeability of substrate layer #k by Mu frequency profile. Zero value of SubCntrl entry means that permeability of respective layer will keep value specified by substrate parameters MuP and MuPP. Keep in mind that only sole Mu frequency profile is allowed, meaning that all layers to which this profile applies are assumed made of the same material.

Relation between frequency sweep and Mu frequency profile

If entire frequency sweep is within FreqProf range than model silently interpolates real and imaginary values of Mu using SubMurF and SubMuiF as lookup tables. If first and/or last sweep frequencies are out of FreqProf range than GFMCLIN issues respective warning and extrapolates beyond FreqProf range as constant value equal either to first and/or last FreqProf entries.

References

[1] Jianming Jin,"The Finite Element Method on Electromagnetics," 2nd edition, Wiley, NY, 2002.

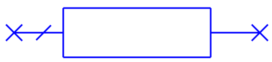
[2] R. Mongia, I. Bahl, and P. Bhartia, RF and Microwave Coupled-Line Circuits, Artech House, Norwood, MA, 1999.

[3] David Meeker home page: http://femm.foster-miller.net/~dmeeker/ Follow this link for information: http://femm.foster-miller.net/Archives/readme.htm FEMM Manual: http://femm.foster-miller.net/Archives/doc/manual.pdf

[4] Jonathan Richard Shewchuk. Triangle. A Two-Dimensional Quality Mesh Generator and Delaunay Triangulator. Follow this link for information and download: http://www-2.cs.cmu.edu/~quake/triangle.html

Single Line on General Multilayer Substrate (EM Quasi-Static): GM1LIN

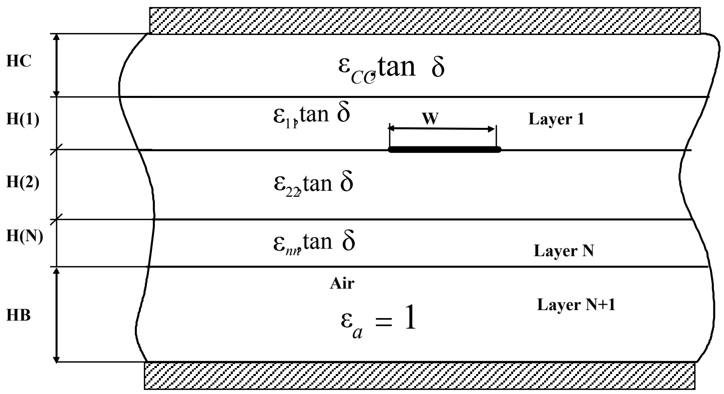
Symbol



Summary

GM1LIN models a section of single microstrip line arranged within an unshielded or shielded (top cover is optional) stratified inhomogeneous substrate. Substrate layers have arbitrary heights and are made of various materials; optionally the substrate may be suspended. Backing ground plane is always present. GM1LIN can account for presence of optional metallic side walls in conjunction with metallic cover (metallic enclosure). This model can evaluate the metal surface roughness effect using surface profile height "Rgh" specified via substrate GMSUB.

Topology



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	TL1
W	Width of conductor	Length	W[a]
CL1	Conductor #1 layer number		1
L	Conductor length	Length	L[a]
Acc	Accuracy parameter		1
GMSUB	Substrate Definition	Text	GMSUB1[b]
*SaveToFile	Save/Do not Save Data to Text File		"No"
*FileName	Name of text file with computed model parameters	String	Same as model name

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See the Microwave Office Layout Guide for details.

[b]Modify only if schematic contains multiple substrates. See “Using Elements With Model Blocks” for details.

** indicates a secondary parameter*

Parameter Details

GMSUB. Multilayer substrate parameters are listed in GMSUB model description. Substrate is comprised of layered substrate itself, optional metallic cover, backing ground plane and optional air layers separating substrate from backing ground plane (suspended substrate) and from cover (covered substrate). Optional metallic side walls are available in conjunction with metallic cover. Optional top and bottom air layers are specified by means of corresponding switches included as GMSUB parameters (see GMSUB description, parameter switches Cover and Gnd). GMSUB implies that thickness of conductors on each layer is specified via vector parameter T. All conductors on the same layer have similar thickness. This thickness may vary between layers. Each dielectric layer must have corresponding entry in vector T even if this specific layer does not carry conductors. Model checks if the length of vector T is equal to the number of dielectric layers that might carry conductors. Surface roughness is specified by the "Rgh" parameter (RMS height of surface profile). If Rgh=0, then evaluation of the roughness effect is skipped.

CL1. Parameter CL1 specifies the number of layer that carries the conductor atop of it. It means that if conductor protrudes upward into layer #m (that is, it takes some bottom space from layer #m) but sits onto the layer #m+1, parameter CL1 must be set to value m+1. If the conductor has negative thickness and is recessed into the layer #m (that is, takes some top space from layer #m), parameter CL1 must be set to value m.

Acc. The parameter Acc is the accuracy parameter. The default value for Acc is 1. If Acc is less than 1 or greater than 10 it is set automatically to 2.

SaveToFile. Parameter is hidden by default and set to No. User can toggle this parameter to Yes and No. If Yes is selected model creates a text file (named by default model_name.txt) at the current project location. This text file contains table of values of RLGC line parameters at each project frequency. Each row contains RLGC values computed at the frequency specified in the first column (frequency in GHz, R in ohms/m, L in H/m, G in S/m, C in F/m). Model GM1LIN creates file gm1lin.txt that contains complex characteristic impedance Re(Zo), Im(Zo), complex effective dielectric constant Re(Eeff), Im (Eff) (please find traditional effective dielectric constant in column 12), and Loss (dB/m) in columns 2-6. Columns 7-10 contain R, L, G, and C. Column 11 contains propagation constant Beta in Rad/m. Note that column 12 contains traditional effective dielectric constant Er_Eff that does not account for loss. Total number of columns in file gm1lin.txt is 12.

Created text file might be linked or imported to project as a data file and frequency behavior of any above mentioned parameter may be viewed using the proper data measurement. Note that first column (frequency) is always in GHz so these measurements might be incompatible with other Microwave Office measurements placed on the same graph; thus, user might prefer to place aforementioned data measurements on separate dedicated graph.

FileName. By default this parameter is hidden and is set to GM1LIN.TXT. User can change file name for each model instance to arbitrary name with length of file name not exceeding 64 symbols.

Parameter Restrictions and Recommendations

1. Total number of layers cannot exceed 30.
2. Accuracy parameter A is limited to $1<Acc<10$. Larger value of Acc increases density of mesh used in computations. Accuracy of model parameters may gain slightly from increasing Acc at the expense of noticeable growth of computation time. As a rule of thumb, good trade-off between accuracy and computation time is to set Acc to 1.
3. Number of dielectric layers must include only those layers that may carry conductors on their top surface. Layers are enumerated from top to bottom. Note that if GMSUB switch parameter Cover is set to "Metallic Cover" or "Metallic B ox" the topmost layer adjacent to cover is not included in count; likewise, if substrate switch parameter Gnd is set to "Suspended Substrate" the bottom air layer is not included in count. Actually, layer number 0 may be displayed in error messages, e.g. in case when substrate is covered and thickness of conductor atop the layer #1 exceeds 95% of thickness of layer located above layer #1, message refers to this "undercover" layer as layer #0.

4. This model does not impose restrictions on the conductor thickness (thickness may be zero, positive, or negative). Negative thickness means that the conductor is recessed into the substrate. User supplies conductor thickness for each layer that may carry conductors (see description of vector parameter T in GMSUB documentation). See also below paragraph 4 regarding thickness of conductor in case of suspended substrate. Proceed with caution setting fictitious thicknesses for layers that do not actually carry conductors: excessive thickness may cause error message if it exceeds 95% of thickness of adjacent dielectric layer. To keep on the safe side you may set these thicknesses to zero.
5. If substrate is suspended, (N+1)th entry of thicknesses vector T (see substrate parameters) refers to conductor that is expected to be located at the bottom surface of layer N; model automatically assigns negative thickness to this conductor. It means that such a conductor extends downwards from the bottom surface of the substrate.
6. If GMSUB switch Cover is set to "Metallic Box" it means that conductor and substrate are confined within metallic enclosure. Top cover is placed at distance HC (sets in GMSUB) above dielectric layer #1, bottom ground plane is always present (position defined by GMSUB parameters Gnd and HB, see GMSUB documentation), and left and right side walls are offset by value of GMSUB parameter SW from respectively left and right edges of line conductor.

Implementation Details

Model implementation is based on EM Quasi-Static technique described in [1] [10–71]. It accounts for losses in metal and in substrate dielectric. Dispersion is partly included. Evaluation of the surface roughness impact is based on the improved Hammerstad formula [2] [10–71],[3] [10–71].

Layout

The project LPF must contain a structure with a predefined name for the dielectric layer that carries a conductor. See “Structure Type Definitions” for more information on adding structures to LPF files. The structure name must be based on the template ML_LINE_X, where X is equal to the number of the dielectric layer specified with GMSUB (for example, ML_LINE_2 or ML_LINE_23). The conductor is comprised of material layers described in the structure ML_LINE_X.

If a structure with the corresponding name is not found, the name of the missing structure is drawn on the error layer.

Note that the number of dielectric layers (and X in ML_LINE_X) defined in GMSUB cannot exceed 30.

Note also that the layout cell assigns to the conductor the linetype index equal to the value of the CL1 parameter, that is equal to the number of the dielectric layer assigned to the conductor.

Recommendations for Use

Primary RLGC parameters as well as characteristic line parameters might be of interest for circuit designer; however, Microwave Office single line models do not cater this information to user explicitly. GM1LIN models accommodate unique opportunity to obtain exhaustive information about values of line parameters at any frequency from operational frequency range. Due to the fact that GM1LIN may be configured to represent any EM Quasi-Static model of a single line (except MM1LIN that allows conductive substrate) it also may be used as means of extracting additional information not available directly from such models as MEMLIN, SEMLIN, S1LIN etc. See documentation on substrate GMSUB for examples of implementation of commonly used single layer configurations (microstrip, stripline, suspended, inverted, etc.)

NOTE: The implementation of EM Quasi-Static models relies heavily on the involved numerical algorithms. This may lead to a noticeable increase in simulation time for schematics that employ many such models.

If thickness of any layer is too small in comparison with the thickness of another layer, simulation time may also noticeably grow.

Caution regarding units of data in saved text file: If a project that reads saved text file uses frequency, resistance, inductance or conductance units different from GHz, ohm, henry or siemens you may need to scale input values manually.

References

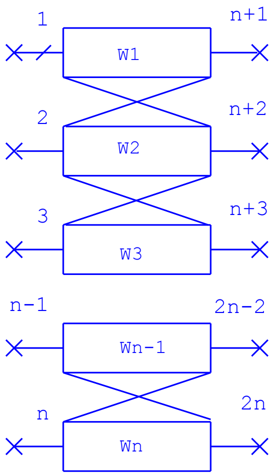
[1] M.B. Bazdar, A.R. Djordjevic, R.F. Harrington, and T.K. Sarkar, "Evaluation of quasi-static matrix parameters for multiconductor transmission lines using Galerkin's method," IEEE Trans. Microwave Theory Tech., vol. MTT-42, July 1994, pp. 1223-1228.

[2] E.O. Hammerstad (edited by F. Bekkadal), Microstrip handbook,Trondheim: Norwegian Institute of Technology, 1985

[3] S.P. Morgan, Jr., "Effects of surface roughness on eddy current losses at microwave frequencies," J. Appl. Phys., vol. 20, 1949, pp. 352-362

Multiple Coupled Lines on General Multilayer Substrate (Dynamic Model) (EM Quasi-Static): GMCLIN

Symbol

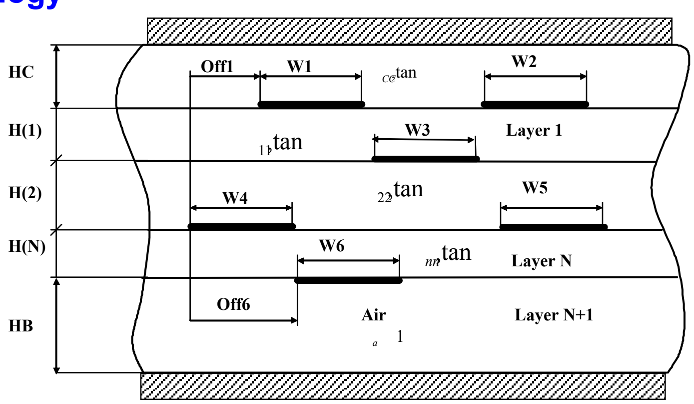


Summary

GMCLIN models a section of several (2 to 50) edge and/or offset broadside coupled microstrip lines arranged within unshielded or shielded (upper cover is optional) stratified inhomogeneous substrate. Substrate layers have arbitrary heights and are made of various materials; optionally the substrate may be suspended. A backing ground plane is always present. GMCLIN can account for the presence of optional metallic side walls in conjunction with a metallic cover.

GMCLIN implements the same modeling techniques as the GM2CLIN through GM10CLIN models. In addition, GMCLIN is a dynamic or scalable model; it accepts a number of lines as input so the model and its schematic symbol expand/shrink as the number of lines increase/decrease. This model can evaluate the metal surface roughness effect using the surface profile height "Rgh" specified via substrate GMSUB.

Topology



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	TL1
N	Number of conductors		2
L	Conductor length	Length	L[a]
Acc	Accuracy parameter		1
GMSUB	Substrate Definition	Text	GMSUB1[b]
*SaveToFile	Switch "Save to txt file"=Yes/No		"No"
*FileName	Name of text file with computed model parameters	String	Same as model name
Wi, i=2..nn-number of lines	Width of conductor #n	Length	W[a]
Offsi, i=2..nn-number of lines	Offset of conductor #n	Length	W[a]
CLi, i=2..nn-number of lines	Number of layer containing conductor #n		1

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See "Intelligent Cells (iCells)" for details.

[b]Modify only if schematic contains multiple substrates. See "Using Elements With Model Blocks" for details.

** indicates a secondary parameter*

Parameter Details

See GMnCLIN for a detailed description of GMCLIN parameters. GMCLIN differs from GMnCLIN in parameter order only: For GMnCLIN models, the L, Acc, GMSUB, Save to File, and FileName parameters follow the widths (W*i*), offsets (Offs*i*), and layers that conductors belong to (CL*i*) parameters; for GMCLIN models, the L, Acc, GMSUB, Save to File, and FileName parameters precede the W*i*, Offs*i*, and CL*i* parameters.

Parameter Restrictions and Recommendations

1. Total number of layers cannot exceed 30.
2. Number of conductors N cannot exceed 50.
3. Surface roughness restriction:

   GMnCLIN implies that Rho is identical across the conductor layers; as is surface roughness, Rgh. Rgh should not exceed half of the conductor thickness. If this occurs GMnCLIN issues a warning.
4. To learn about restrictions and recommendations common for GMCLIN and GMnCLIN, see GMnCLIN.

Implementation Details

- Model implementation is based on the EM Quasi-Static technique described in [1] [10–75]. It accounts for losses in metal and in substrate dielectric. Dispersion is partly included.
- To apply Method of Moments for analysis, a quasi-static model creates 1D mesh covering contours of all conductors. The mesh is made of linear segments (pulses) of varying length. The length of a pulse is relatively big at the conductor center; it decreases toward the conductor edges to reveal the charge distribution across the conductor. If the conductor

width is too large it may cause the pulse size to approach zero for pulses close to the edge. In these rare cases the model may display a "Length of pulse #nnn equal to zero" error message.

- Evaluation of the surface roughness impact is based on the improved Hammerstad formula [3] [10–75],[4] [10–75].

Layout

The project LPF must contain some structures with predefined names for each dielectric layer populated with conductors (note that every conductor has the number of the assigned dielectric layer specified in the respective parameter CLn, where n is the conductor number). See "Structure Type Definitions" for more information on adding structures to LPF files. Structure names must be based on the template ML_LINE_X, where X is equal to the number of the dielectric layer. All conductors that belong to layer X are comprised of material layers described in ML_LINE_X.

For example, if GMCLIN defines 7 conductors, substrate GMSUB defines 8 dielectric layers, and conductors are distributed over dielectric layers in accordance with the following table:

Conductor #	CL value
1	1
2	2
3	3
4	3
5	3
6	6
7	6

Per the table you see that conductor #1 sits on dielectric layer #1, conductor #2 sits on dielectric layer #2, conductors #3, #4, and #5 are on layer #3, and conductors #6 and #7 sit on layer #6. This arrangement indicates that only layers #1, 2, 3, and 6 are populated with conductors, and you need to specify structures ML_LINE_1, ML_LINE_2, ML_LINE_3, and ML_LINE_6.

If a structure with the corresponding name is not found, the name of the missing structure is drawn on the error layer.

Note that the number of dielectric layers (and X in ML_LINE_X) defined in GMSUB cannot exceed 30.

Note also that the layout cell assigns to each conductor #n the linetype index equal to the value of the corresponding parameter CLn, that is equal to the number of the dielectric layer assigned to conductor #n.

Recommendations for Use

See this section in GMnCLIN for details. The section also includes examples of usage.

NOTE: The implementation of EM Quasi-Static models relies heavily on the involved numerical algorithms. This may lead to a noticeable increase in simulation time for schematics that employ many such models.

If a layer thickness is too small compared to the thickness of another layer, simulation time may also noticeably increase.

Caution regarding units of data in saved text files: If a project that reads saved text files uses frequency, resistance, inductance or conductance units different from GHz, ohm, henry or siemens, you may need to scale input values manually.

References

[1] M.B. Bazdar, A.R. Djordjevic, R.F. Harrington, and T.K. Sarkar, "Evaluation of quasi-static matrix parameters for multiconductor transmission lines using Galerkin's method," IEEE Trans. Microwave Theory Tech., vol. MTT-42, July 1994, pp. 1223-1228.

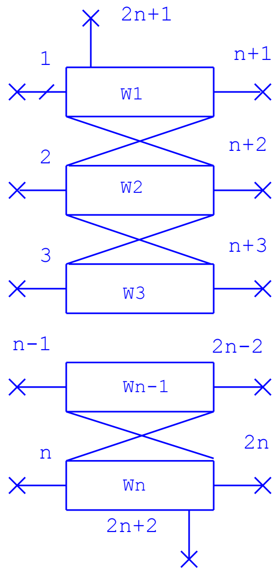
[2] R. Mongia, I. Bahl, and P. Bhartia, RF and Microwave Coupled-Line Circuits, Artech House, Norwood, MA, 1999.

[3] E.O. Hammerstad (edited by F. Bekkadal), Microstrip handbook, Trondheim: Norwegian Institute of Technology, 1985

[4] S.P. Morgan, Jr., "Effects of surface roughness on eddy current losses at microwave frequencies," J. Appl. Phys., vol. 20, 1949, pp. 352-362

Asymmetric Tapped Multiple Coupled Lines on General Multilayer Substrate (Dynamic Model) (EM Quasi-Static): GMCLINAT

Symbol



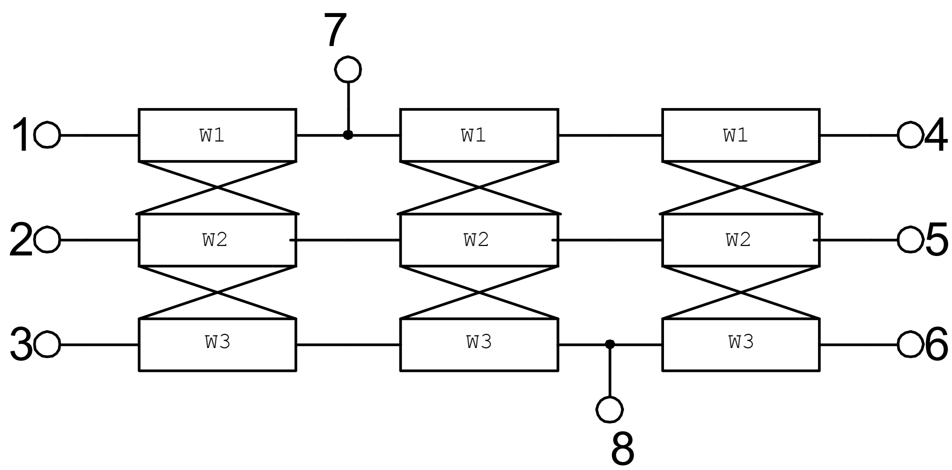
Summary

GMCLINAT is intended for use in tapped interdigital filters. GMCLINAT models three cascaded sections of several (2 to 50) edge and/or offset broadside coupled microstrip lines with two tap ports attached between sections (see GMCLIN).

This model is based on GMCLIN, uses the same substrate GMSUB, and has the same parameters (except section lengths) as GMCLIN.

GMCLINAT has an advantage over cascading three GMCLIN models because it runs three times faster.

Equivalent Circuit



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	TL1
N	Number of conductors		2
L1	Conductor length of 1st section	Length	L[a]
L2	Conductor length of 2nd section	Length	L[a]
L3	Conductor length of 3rd section	Length	L[a]
Acc	Accuracy parameter		1
GMSUB	Substrate Definition	Text	GMSUB1[b]
*SaveToFile	Switch "Save to txt file"=Yes/No		"No"
*FileName	Name of text file with computed model parameters	String	Same as model name
Wi, i=2..nn-number of lines	Width of conductor #n	Length	W[a]
Offsi, i=2..nn-number of lines	Offset of conductor #n	Length	W[a]
CLi, i=2..nn-number of lines	Number of layer containing conductor #n		1

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See "Intelligent Cells (iCells)" for details.

[b]Modify only if schematic contains multiple substrates. See "Using Elements With Model Blocks" for details.

** indicates a secondary parameter*

Parameter Details

See GMCLIN for a detailed description of GMCLINAT parameters. GMCLINAT differs from GMCLIN only in length parameters.

L1. Length of the front section between input ports (Ports 1..3 in "Equivalent Circuit") and the first tap port (Port 7 in "Equivalent Circuit").

L2. Length of the middle section between the first and second tap ports (Ports 7 and 8 in "Equivalent Circuit").

L3. Length of the tail section between second tap port and output ports (Ports 4..6 in "Equivalent Circuit").

Parameter Restrictions and Recommendations

1. The total number of layers cannot exceed 30.
2. The number of conductors N cannot exceed 50.
3. To learn about restrictions and recommendations common for GMCLIN, GMnCLIN, and GMCLINAT, see GMnCLIN.

Implementation Details

Model implementation is based on the EM Quasi-Static technique described in [1] [10–78]. It accounts for losses in metal and in substrate dielectric. Dispersion is partly included.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

Recommendations for Use

You can set any section length (L1, L2 or L3) equal to zero. For example, setting L2=0 converts an asymmetric tapped line into a symmetric tapped line (see GMCLINST).

See this section in GMnCLIN for details. The section also includes examples of usage.

NOTE: The implementation of EM Quasi-Static models relies heavily on the involved numerical algorithms. This may lead to a noticeable increase in simulation time for schematics that employ many such models.

If a layer thickness is too small compared to the thickness of another layer, simulation time may also noticeably increase.

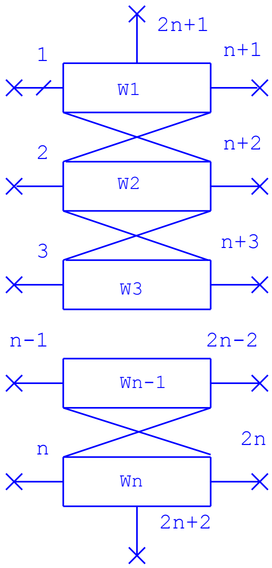
References

[1] M.B. Bazdar, A.R. Djordjevic, R.F. Harrington, and T.K. Sarkar, "Evaluation of quasi-static matrix parameters for multiconductor transmission lines using Galerkin's method," IEEE Trans. Microwave Theory Tech., vol. MTT-42, July 1994, pp. 1223-1228.

[2]R. Mongia, I. Bahl, and P. Bhartia, RF and Microwave Coupled-Line Circuits, Artech House, Norwood, MA, 1999.

Symmetric Tapped Multiple Coupled Lines on General Multilayer Substrate (Dynamic Model) (EM Quasi-Static): GMCLINST

Symbol



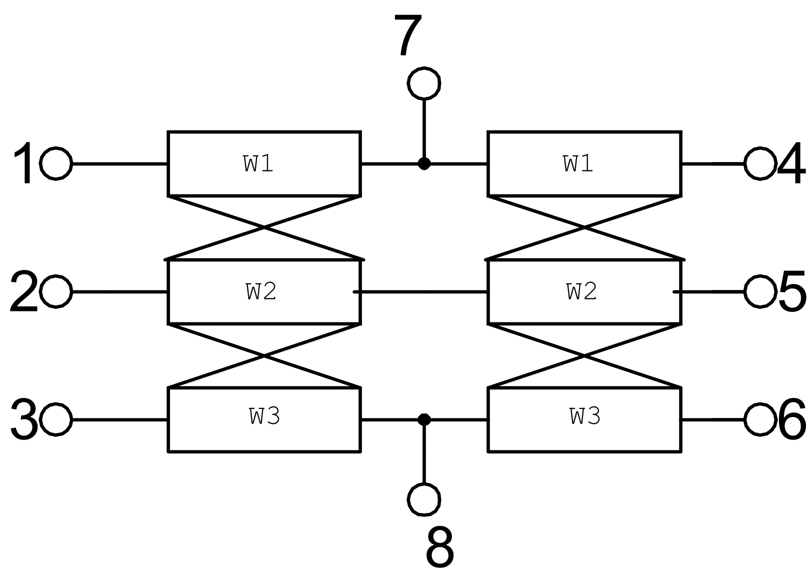
Summary

GMCLINST is intended for use in tapped interdigital filters. GMCLINST models two cascaded sections of several (2 to 50) edge and/or offset broadside coupled microstrip lines with two tap ports attached between sections (see GMCLIN for a detailed summary).

This model is based on the GMCLIN model; it uses the same substrate GMSUB and has the same parameters (except section lengths).

GMCLINAT has an advantage over cascading two GMCLIN models because it runs twice as fast.

Equivalent Circuit



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	TL1
N	Number of conductors		2
L1	Conductor length of 1st section	Length	L[a]
L2	Conductor length of 2nd section	Length	L[a]
Acc	Accuracy parameter		1
GMSUB	Substrate Definition	Text	GMSUB1[b]
*SaveToFile	Switch "Save to txt file"=Yes/No		"No"
*FileName	Name of text file with computed model parameters	String	Same as model name
Wi, i=2..nn-number of lines	Width of conductor #n	Length	W[a]
Offsi, i=2..nn-number of lines	Offset of conductor #n	Length	W[a]
CLi, i=2..nn-number of lines	Number of layer containing conductor #n		1

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See the Microwave Office Layout Guide for details

[b]Modify only if schematic contains multiple substrates. See "Using Elements With Model Blocks" for details.

** indicates a secondary parameter*

Parameter Details

See GMCLIN for a detailed description of GMCLINST parameters. GMCLINST differs from GMCLIN only in its length parameters.

L1. Length of front section between the input ports (Ports 1..3 in "Equivalent Circuit") and tap port.

L2. Length of tail section between the second tap port and output ports (Ports 4..6 in "Equivalent Circuit").

Parameter Restrictions and Recommendations

1. The total number of layers cannot exceed 30.
2. The number of conductors N cannot exceed 50.
3. For more information about restrictions and recommendations common for GMCLIN, GMnCLIN, and GMCLINST, see GMnCLIN .

Implementation Details

Model implementation is based on the EM Quasi-Static technique described in [1] [10–81]. It accounts for losses in metal and in substrate dielectric. Dispersion is partly included.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Recommendations for Use

See this section in GMnCLIN for details. The section also includes examples of usage.

NOTE: The implementation of EM Quasi-Static models relies heavily on the involved numerical algorithms. This may lead to a noticeable increase in simulation time for schematics that employ many such models.

If a layer thickness is too small compared to the thickness of another layer, simulation time may also noticeably increase.

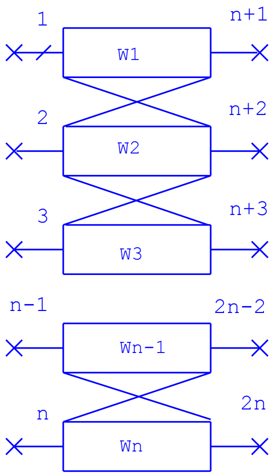
References

[1] M.B. Bazdar, A.R. Djordjevic, R.F. Harrington, and T.K. Sarkar, "Evaluation of quasi-static matrix parameters for multiconductor transmission lines using Galerkin's method," IEEE Trans. Microwave Theory Tech., vol. MTT-42, July 1994, pp. 1223-1228.

[2]R. Mongia, I. Bahl, and P. Bhartia, RF and Microwave Coupled-Line Circuits, Artech House, Norwood, MA, 1999.

Multiple Coupled Lines on General Multilayer Substrate (n=2 to 10) (EM Quasi-Static): GMnCLIN

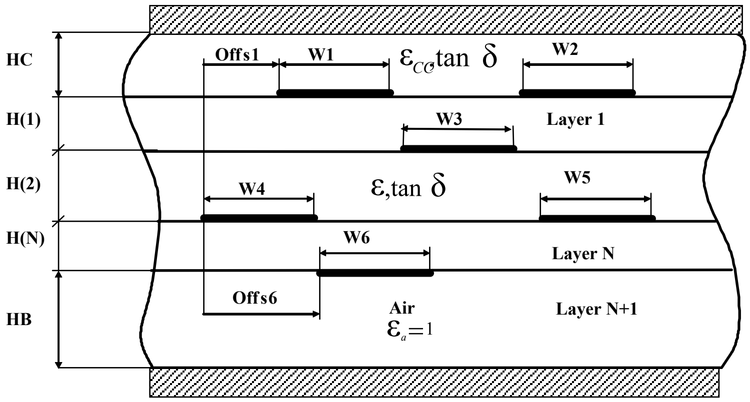
Symbol

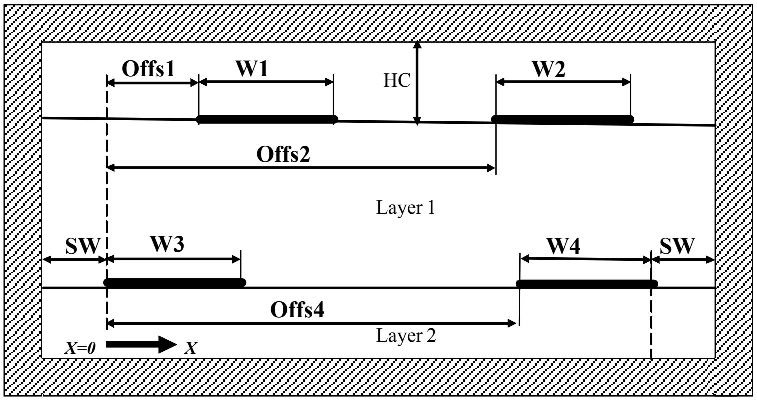


Summary

GMnCLIN models a section of several (2 to 10) edge- and/or offset broadside-coupled microstrip lines arranged within unshielded or shielded (optional upper cover) stratified inhomogeneous substrate. Substrate layers have arbitrary heights and are made of various materials; optionally the substrate may be suspended. A backing ground plane is always present. GMnCLIN models can account for the presence of optional metallic side walls in conjunction with a metallic cover (metallic enclosure). This model can evaluate the metal surface roughness effect using surface profile height "Rgh" specified via substrate GMSUB.

Topology





Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	TL1
Wi, i=2..nn-number of lines	Width of conductor #n	Length	W[a]
Offi, i=2..nn-number of lines	Offset of conductor #n	Length	W[a]
CLi, i=2..nn-number of lines	Number of layer containing conductor #n		1
L	Conductor length	Length	L[a]
Acc	Accuracy parameter		1
GMSUB	Substrate Definition	Text	GMSUB1[b]
*SaveToFile	Switch "Save to txt file"=Yes/No		"No"
*FileName	Name of text file with computed model parameters	String	Same as model name

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See Microwave Office Layout Guide for details.

[b]Modify only if the schematic contains multiple substrates. See “Using Elements With Model Blocks” for details.

** indicates a secondary parameter*

Parameter Details

GMSUB. Multilayer substrate parameters are listed in the GMSUB model description. The substrate is comprised of the layered substrate itself, an optional metallic cover, a backing ground plane and optional air layers separating the substrate from the backing ground plane (suspended substrate) and from the cover (covered substrate). Optional metallic side walls are available in conjunction with the metallic cover. Optional top and bottom air layers are specified by means of corresponding switches included as GMSUB parameters (see the GMSUB model Cover and Gnd parameters). GMSUB implies that the thickness of conductors on each layer is specified via vector parameter T. All conductors on the same layer have a similar thickness. This thickness may vary between layers. Each dielectric layer must have a corresponding entry in vector T even if this specific layer does not carry conductors. This model checks if the length of vector T is equal to the number of dielectric layers that might carry conductors. Surface roughness is specified by the "Rgh" parameter (RMS height of surface profile). If Rgh=0, then evaluation of the roughness effect is skipped.

OffsX. The relative horizontal offset of the conductor #X left edge from the left edge of the left-most conductor. All offsets must be positive. Offset of the left-most conductor (or conductors) may be set to zero. The model looks for the smallest offset and subtracts it from other offsets. The left edge of the left-most conductors marks a reference plane to determine the distance to optional metallic side walls.

See the "Topology" section and the example explained in the "Parameter Restrictions and Recommendations" section.

CL1. Specifies the number of the layer that carries the conductor on top of it. If the conductor protrudes upward into layer #m (that is, it occupies some lower part of layer #m but sits on layer #m+1), CL1 must be set to value m+1. If the conductor has a negative thickness and is recessed into layer #m (that is, it occupies the upper part of layer #m), CL1 must be set to m.

Acc. The accuracy parameter. The default value for Acc is 1. If it is less than 1 or greater than 10 it is set automatically to 2.

SaveToFile. This parameter is hidden by default and set to No. You can toggle the parameter to Yes or No. If Yes is selected, the model creates a text file (named *default model_name.txt*) at the current project location. This text file contains a table of values of RLGC line parameters at each project frequency. Each row contains RLGC values computed at the frequency specified in the first column (frequency in GHz, R in ohms/m, L in H/m, G in S/m, C in F/m).

The structure of this text file is essentially the same for GMnCLIN models, n>2 (see the description of the text file structure in GM1LIN). All models output RLGC matrices to columns that immediately follow the frequency column. Entries of each matrix are placed column-wise; the first column is first: R11, R21, R31.. Rn1; and then the second column: R12, R22, R32,..Rn2. The total number of columns in the file is 4*n*n+1, where n is the number of lines.

The GM2CLIN model implies the existence of two modes, namely, C and P (see [2]) and places additional columns to the *gm2clin.txt* file. Note that if a system of coupled lines is fully symmetrical (as might be the case of edge-coupled microstrip lines; whereas, broadside coupled microstrip lines are always asymmetrical), C-mode corresponds to even mode and P-mode corresponds to odd mode.

GM2CLIN outputs complex characteristic impedances Re(Zc1), Im(Zc1), Re(Zp1), Im(Zp1), Re(Zc2), Im(Zc2), Re(Zp2), Im(Zp2); complex effective dielectric constants Re(EeffC), Im(EeffC), Re(EeffP), Im(EeffP) (traditional effective dielectric constants are in columns 38, 39), and losses for C and P modes LossC (dB/m), LossP(dB/m) to columns 2-15. Entries of R, L, G, and C matrices are distributed among columns 16-31 in corresponding order (see above). Columns 32-37 contain Re(Rc), Im(Rc), Re(Rp), Im(Rp), BetaC, BetaP; where Rc is the ratio of C-mode voltage in the second line to C-mode voltage in the first line; Rp is the same ratio in P-mode (see details in [2], section 4.3.1) and BetaC and BetaP are propagation constants of C- and P-modes in Rad/m. Note that columns 38 and 39 contain traditional effective dielectric constants ErC_Eff, ErP_Eff (they do not account for losses). The total number of columns in the *gm2clin.txt* file is 39.

The created text file might be linked or imported to a project as a data file and frequency behavior of any above mentioned parameter may be viewed using the proper data measurement. Note that the first column (frequency) is always in GHz so these measurements might be incompatible with other MWO measurements placed on the same graph; you may prefer to use these data measurements on a separate dedicated graph.

FileName. By default this parameter is hidden and is set to model *name.txt*. For example, the GM1LIN model saves parameters to a *gm1lin.txt* file and GM7CLIN saves its parameters to a *gm7clin.txt* file. You can change the file name for each model instance to an arbitrary name with a length not exceeding 64 symbols.

Parameter Restrictions and Recommendations

Arrangement and numbering of layers:

1. Total number of layers cannot exceed 30.
2. Conductors are assumed located on top of dielectric layers.
3. The number of dielectric layers must include only those layers that may carry conductors on their top surface. Layers are enumerated from top to bottom. Note that if the Cover parameter is set to "Metallic Cover" or "Metallic Box" the topmost layer adjacent to the cover is excluded from the count; likewise, if the substrate Gnd parameter is set to "Suspended Substrate," the bottom air layer is not included in the count. For example, for a simple suspended (covered or uncovered) substrate you must set N=1. Nevertheless, conductors that extend downwards into the bottom air layer belong to layer N+1 and the corresponding CLn parameter must be set to N+1 (in the case of a simple suspended substrate, to 2). Layer number 0 might be displayed in error messages (when a substrate is covered and the thickness of conductors on top of layer #1 exceeds 95% of the thickness of the layer located above layer #1), this message refers to this "undercover" layer as layer #0.

Conductor numbering and disposition:

- Conductors are numbered from left to right in each layer first; after completion, numbering in the upper layer continues in the next layer (see the "Topology" section). This model implies that all conductors in Layer 1 are numbered from left to right, for example, from 1 to 2; a single conductor in Layer 2 is number 3; conductors in layer 3 are numbered 4 and 5, and so on. The CLn and Offn parameters must be arranged in corresponding order. The suggested numbering scheme is mandatory; wrong numbering causes errors.
- The numbering of conductors displayed in error messages may vary. Note that the conductor number in messages such as "Offset of conductor 3 must be greater than zero" is always the conductor order number according to the numbering above. Certain error messages, however, that mention layer number refer to the relative conductor number on the corresponding layer. For example, the message "Thickness of strip #1 on layer #6 exceeds 95% of thickness of layer #5" implies that the leftmost strip (#1 on layer 6) on the dielectric layer 6 is almost as thick as the upper adjacent layer 5.

Thickness of conductors:

- This model does not impose restrictions on the conductor thickness (thickness may be zero, positive, or negative). A negative thickness means that the conductor is recessed into the substrate. You supply the conductor thickness for each layer that may carry conductors (see the description of the T parameter in GMSUB). It is implied that all conductors located on top of the same dielectric layer have the same thickness. Use caution when setting fictitious thicknesses for layers that do not actually carry conductors. Excessive thickness may cause an error message if it exceeds 95% of the thickness of the adjacent dielectric layer. As a precaution you may set these thicknesses to zero.

If the substrate is suspended, the (N+1)th entry of thicknesses vector T (see the substrate parameters) refers to conductors that are expected to be arranged over the bottom surface of layer N. These conductors actually belong to layer N+1. This model automatically assigns a negative thickness to these conductors, so they extend downward from the bottom surface of the substrate and are recessed into air layer N+1 (see conductor 6 in the "Topology" section).

Conductor offsets and side walls:

- The Offs1 and Offs2 parameters have a reference plane located at the left edge of the left-most conductor (see the "Topology" section where the second figure shows four conductors confined in a metallic enclosure). In this setup, conductor #3 is the left-most one, thus you should set Offs3 to zero. The position of the reference plane can be described as X=0, defining X as a horizontal axis looking to the right and originating at the left edge of the left-most conductor #3. All offsets must be referenced to position X=0.
- If the GMSUB Cover parameter is set to "Metallic Box", conductors and substrate are confined within a metallic enclosure. The top cover is placed at the distance HC (a GMSUB parameter) above dielectric layer #1, and the bottom ground plane is always present (as defined by the GMSUB Gnd and HB parameters).

- The left side wall, in terms of coordinate X, is located at X=-SW. The right side wall is offset by the distance SW from the right edge of the rightmost conductor #4 and is located at X=Offs4+W4+SW. This disposition provides symmetrical placement of conductor setup between the side walls of the enclosure.

Other restrictions:

1. The Accuracy parameter (Acc) is limited to 1≤Acc≤10. A larger value of Acc increases the density of mesh used in computations. The accuracy of model parameters may gain slightly from increasing Acc at the expense of a noticeable increase in computation time. Generally, a good trade-off between accuracy and computation time is to set Acc to 1.
2. The Offn parameter has an upper limit. If any pair of conductors' Offn/W ratio (W is the largest width in the pair) exceeds 200, GMnCLIN issues a warning.

Surface roughness restriction:

GMnCLIN implies that Rho is identical across the conductor layers; as is surface roughness, Rgh. Rgh should not exceed half of the conductor thickness. If this occurs GMnCLIN issues a warning.

Implementation Details

- GMnCLIN implementation is based on the EM Quasi-Static technique described in [1] [10–90]. It computes matrices of per-unit-length RLGC parameters and uses them to evaluate circuit parameters of coupled lines.
- This model accounts for losses in metal and in substrate dielectric. Dispersion is partly included (in part of frequency-dependence of output parameters due to the presence of frequency-dependent losses).
- Evaluation of the surface roughness impact is based on the improved Hammerstad formula [3] [10–90],[4] [10–90].

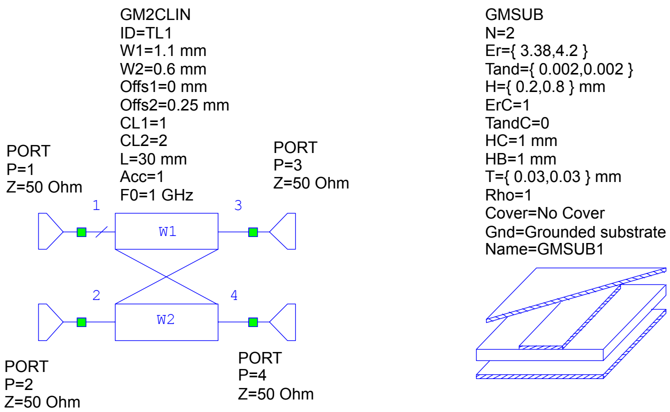
Recommendations for Use

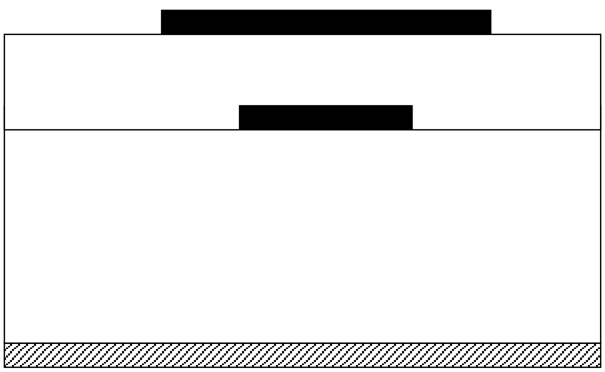
Primary RLGC parameters as well as characteristic line parameters might be of interest to a circuit designer; however, MWO coupled line models do not provide this information explicitly. GMnCLIN models accommodate a unique opportunity to obtain exhaustive information about the values of line parameters at any frequency from the operational frequency range. Because GMnCLIN can be configured to represent any EM Quasi-Static model of coupled lines (except MM2LIN which allows conductive substrate) you may also use it as a means of extracting additional information not available directly from such models as MnCLIN, SnCLIN, SBCPL, MEM2LIN, SEM2LIN etc. See GMSUB for examples of implementation of commonly used single layer configurations.

For GM2CLIN models only: C- and P-modes propagating in two coupled lines might swap their places at certain frequency points. This swap displays as an abrupt change in the curves representing the measurements based on a text file generated by the GM2CLIN model. This problem is handled in the model by checking the abrupt "steps" in frequency dependence of Re (Rc) and swapping modes back. Unfortunately, if two adjacent frequency points are way off, the values of Re(Rc) may differ more than 40% (the threshold that is hardcoded in the model) and unwanted swap may occur resulting in dips on smooth curves (usually occurring at relatively low frequencies). To remove these dips, simply make the frequency mesh finer in their vicinity.

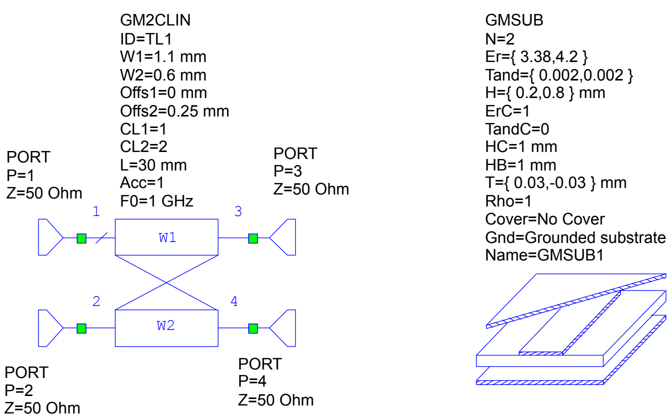
The following is an example of two configurations of a microstrip broadside coupler:

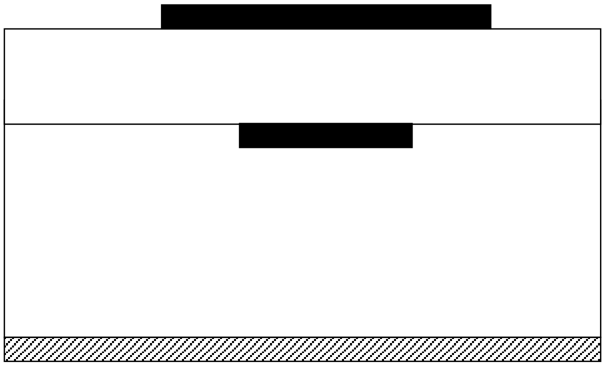
Configuration #1. Metal traces are printed and etched separately on the top sides of sheets made of top and bottom dielectric materials; the bottom metal extends upward.





Configuration #2. Metal traces are printed and etched on both sides of a sheet of top dielectric; this dielectric was installed on top of the bottom dielectric; the bottom metal extends downward.

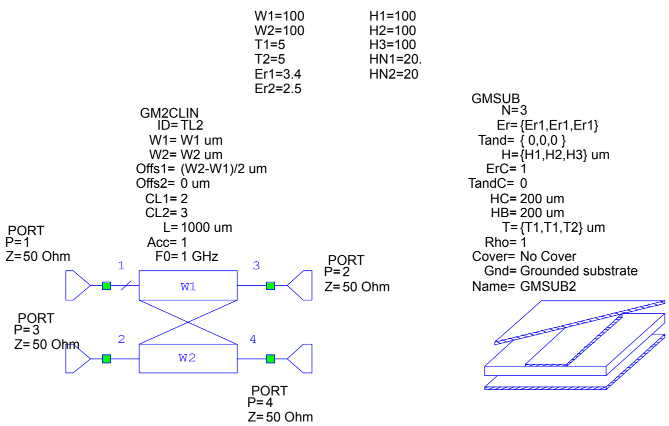


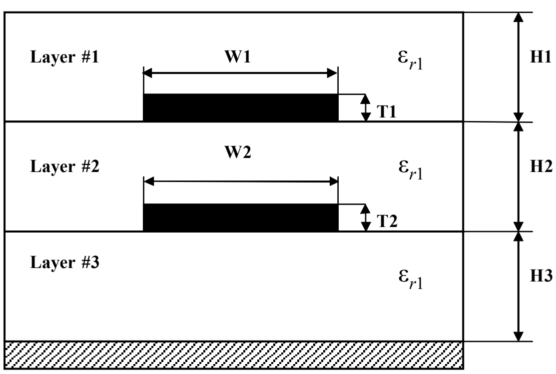


The second configuration differs from the first only by the sign of the second entry of vector T (a GMSUB parameter).

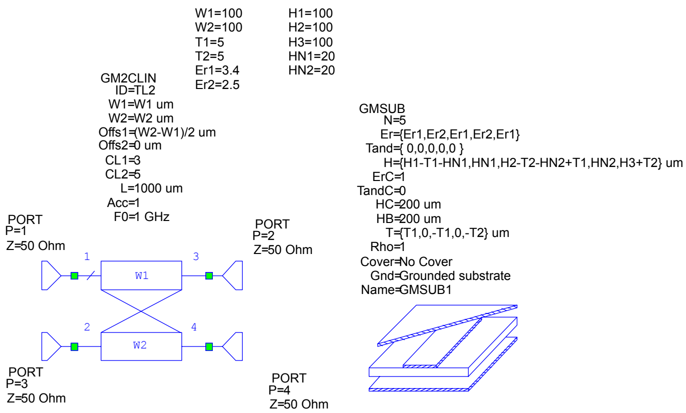
The following is a more sophisticated implementation of a multilayer dielectric stack:

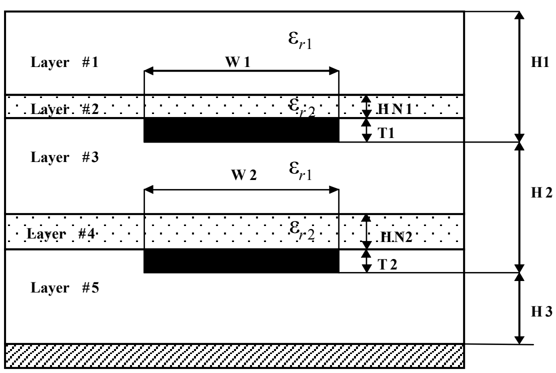
Configuration #1. This configuration looks like the configuration in the previous example. Both conductors "float" into the homogeneous dielectric; the dielectric is broken into three layers to provide a specified arrangement of conductors.





Configuration #2. This configuration is an update of configuration #1 of this example. It appeared necessary to place additional dielectric layers made of different material (for example, glue or resin) on top of each conductor. The distances H1, H2, and H3 must remain the same. The problem that arises if you try to place these additional layers is that you cannot merely arrange a layer on top of a metal conductor, because a metal cannot occupy vertically more than 95% of a layer thickness into which it protrudes. GMnCLIN issues an error message because the modeling algorithm allows only a single horizontal edge of a conductor to coincide or to be in the immediate vicinity of the boundary between dielectrics. To work around this problem you need to rework the layout and create one which applies negative thickness to both conductors. The following figures show how you might implement this workaround. Note how the components of vectors H and T are specified in terms of layer and conductors thickness.





NOTE: The implementation of EM Quasi-Static models relies heavily on the involved numerical algorithms. This may lead to a noticeable increase in simulation time for schematics that employ many such models.

If the thickness of any layer is too small when compared to the thickness of another layer, simulation time may also noticeably increase. You should avoid large ratios for SW/Hstack and HC/Hstack (Hstack is the total height of a dielectric stack plus optional HC and HB) with GFMnCLIN and GMCLIN models due to a possible reduction in accuracy. You should not set these ratios above 30-40.

Caution regarding units of data in saved text file: If a project that reads saved text files uses frequency, resistance, inductance or conductance units different from GHz, ohm, henry or siemens you may need to scale input values manually.

Layout

The project LPF must contain some structures with predefined names for each dielectric layer populated with conductors (note that every conductor has the number of the assigned dielectric layer specified in the respective CLn parameter, where n is the conductor number). Structure names must be based on the template ML_LINE_X, where X is equal to the number of the dielectric layer. All conductors that belong to layer X are comprised of material layers described in ML_LINE_X.

For example, if GMCLIN defines seven conductors, substrate GMSUB defines eight dielectric layers, and conductors are distributed over dielectric layers in accordance with the following table:

Conductor #	CL value
1	1
2	2
3	3
4	3
5	3
6	6
7	6

From the table you see that conductor #1 sits on dielectric layer #1, conductor #2 sits on dielectric layer #2, conductors #3, #4, and #5 are on layer #3, and conductors #6 and #7 sit on layer #6. This arrangement indicates that only layers #1, 2, 3, and 6 are populated with conductors and you need to specify structures ML_LINE_1, ML_LINE_2, ML_LINE_3, and ML_LINE_6.

If the structure with the corresponding name is not found, the name of the missing structure is drawn on the error layer.

Note that the number of dielectric layers (and X in ML_LINE_X) defined in GMSUB cannot exceed 30.

Note also that the layout cell assigns to each conductor #n the linetype index equal to the value of the corresponding parameter CLn, that is equal to the number of the dielectric layer assigned to conductor #n.

References

[1] M.B. Bazdar, A.R. Djordjevic, R.F. Harrington, and T.K. Sarkar, "Evaluation of quasi-static matrix parameters for multiconductor transmission lines using Galerkin's method," IEEE Trans. Microwave Theory Tech., vol. MTT-42, July 1994, pp. 1223-1228.

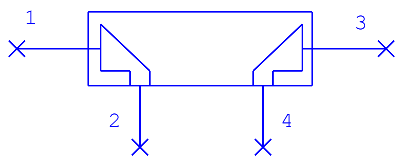
[2] R. Mongia, I. Bahl, and P. Bhartia, RF and Microwave Coupled-Line Circuits, Artech House, Norwood, MA, 1999.

[3] E.O. Hammerstad (edited by F. Bekkadal), Microstrip handbook,Trondheim: Norwegian Institute of Technology, 1985

[4] S.P. Morgan, Jr., "Effects of surface roughness on eddy current losses at microwave frequencies," J. Appl. Phys., vol. 20, 1949, pp. 352-362

Lange Coupler Manifold (Static Solution): LMAN

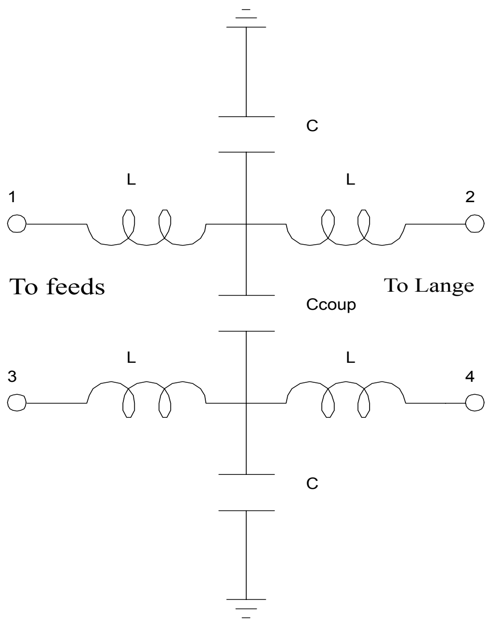
Symbol



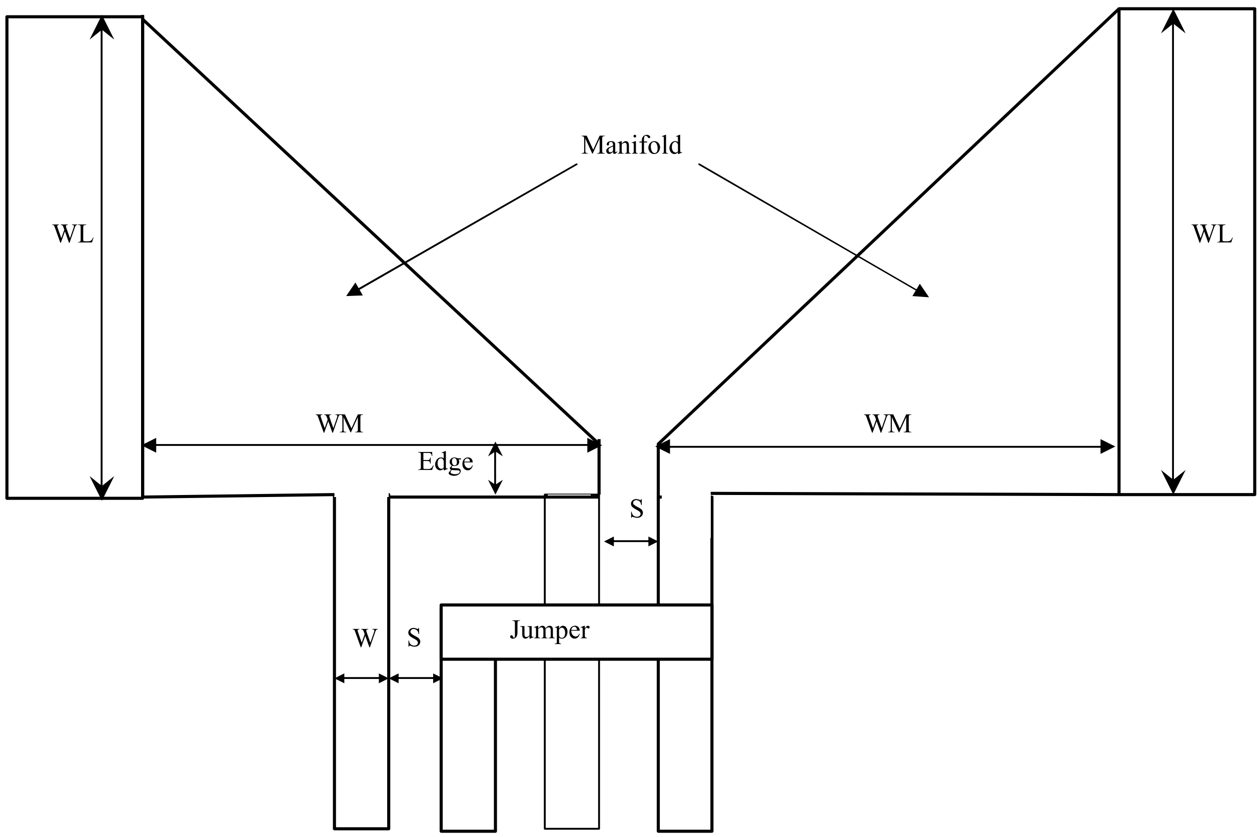
Summary

LMAN models transitions (metal manifolds) between microstrip feeding lines and the body of a Lange coupler. LMAN is intended for use in conjunction with the MLANGE2 (Microstrip Lange Coupler, EM Quasi-Static) element. The layout of the two models match.

Equivalent Circuit



Topology



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	XV1
N	Number of Lange coupler fingers	Scalar	4
W	Lange coupler finger width	Length	W[a]
S	Spacing between Lange coupler fingers	Length	W[a]
WL	Width of feeding microstrip line	Length	W[a]
WM	Width of manifold side attached to Lange coupler	Length	W[a]
MSUB	Substrate Definition	Text	MSUB[b]
Edge	Height of manifold edge **(layout parameter only)**	Length	0

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See the Microwave Office Layout Guide for details.

[b]Modify only if the schematic contains multiple substrates. See "Using Elements With Model Blocks" for details.

Parameter Details

N, W, S. These parameters are used to calculate the size of the manifold side attached to the Lange coupler. They are also used in the layout cell to calculate the positions of faces that provide connection to Lange fingers (see the "Topology" section).

Edge. The electrical model does not use this parameter. This parameter is set in Layout View only. To set the Edge, right-click the LMAN shape in the Layout View, choose **Shape Properties** to display the Cell Options dialog box, then click the **Parameters** tab. Set the Edge to the desired value.

Parameter Restrictions and Recommendations

1. Parameter N must be even and within range $4 \le N \le 16$.
2. Parameter WL must be equal to the width of the microstrip line attached to the manifold.
3. Parameter WM defines the side of the manifold attached to the Lange coupler. WM must be big enough to provide a connection to all fingers attached to the manifold (see the "Topology" section). The minimal value of WM is equal to WMmin = 2[N/4](W+S)+W where [] is the integer. If WM<WMmin, then the model sets WM=WMmin and issues a warning.

Implementation Details

Capacitances C, Ccoup, and inductance L (see the "Equivalent Circuit" section) are evaluated using Method of Moments. Implementation is based on a static Green's function technique in conjunction with approximation of Green's function in the spectral domain. Polarization and conductive losses are not included.

To decrease the calculation time for schematics that contain several manifolds, cache is implemented for this model. During the first evaluation of schematic capacitances for each manifold, instances are stored in memory cache and saved to the hard drive. Each manifold model in every new schematic/project checks this cache looking for its duplicate. Duplicate manifolds copy the manifold capacitances from memory cache, saving substantial recalculation time.

LMAN assumes that both parts of the manifold have a triangular shape. Setting the Edge parameter to a nonzero value modifies the manifold shape in layout, but has no effect on calculated electrical parameters. The layout cell sets the maximum limit on the value of Edge at WL/2.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See "Cell Options Dialog Box: Layout Tab " for Cell Options dialog box **Layout** tab details.

See "The Layout Process File (LPF)" for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

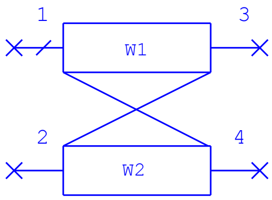
Recommendations for Use

LMAN is intended for use in conjunction with MLANGE2. LMAN provides a layout matching the layout of MLANGE2.

The implementation of this element relies on the involved numerical algorithms. This may lead to an increase in simulation time for schematics that employ many differing instances of LMAN.

2 Coupled Microstrip Lines (EM Quasi-Static): M2CLIN

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Name	Text	TL1
W1	Conductor 1 width - Nodes 1&&3	Length	W[a]
W2	Conductor 1 width - Nodes 2&&4	Length	W[a]
S	Gap width 1&&2	Length	W[a]
L	Conductor length	Length	L[a]
Acc	Accuracy parameter		1
MSUB	Substrate definition	Text	MSUB1[b]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See the Microwave Office Layout Guide for details.

[b]Modify only if schematic contains multiple substrates. See "Using Elements With Model Blocks" for details.

Implementation Details

This circuit component models a section of two edge coupled microstrip lines.

The parameters W1, W2 (strip widths), S (gap between strips), and L (line length) are dimensions entered in the default length units.

The parameter Acc is the accuracy parameter ($1 \leq Acc \leq 10$). The default value for Acc is 1. If Acc is less than 1 or greater than 10 it sets automatically to 2. Larger value of Acc increase density of mesh used in computations. Accuracy of model parameters may gain slightly from increasing Acc at the expense of noticeable growth of computation time. As a rule of thumb, good trade-off between accuracy and computation time is to set Acc to 1.

The parameter MSUB specifies the microstrip substrate element, which defines additional cross sectional parameters of the transmission line. If blank, a default is used.

This component doesn't impose restrictions on the conductor thickness (thickness may be zero, positive, or negative). Negative thickness means that the conductor is recessed into the substrate.

The component accounts for losses in metal and in substrate dielectric. Dispersion is not included.

NOTE: The implementation of EM Quasi-Static models heavily relies on the involved numerical algorithms. This may lead to the noticeable increase of computation time for some schematics that employ many models of this kind.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See "Cell Options Dialog Box: Layout Tab " for Cell Options dialog box **Layout** tab details.

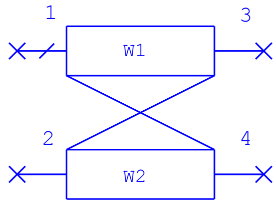
See "The Layout Process File (LPF)" for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

References

[1] B. Bazdar, A.R. Djordjevic, R.F. Harrington, and T.K. Sarkar, "Evaluation of quasi-static matrix parameters for multiconductor transmission lines using Galerkin's method," IEEE Trans. Microwave Theory Tech., vol. MTT-42, July 1994, pp. 1223-1228.

Asymmetric Edge Coupled Microstrip Lines (Closed Form): MACLIN

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Name	Text	TL1
W1	Conductor 1 width - nodes 1&&3	Length	W[a]
W2	Conductor 2 width = nodes 2&&4	Length	W[a]
S	Gap width conductors 1&&2	Length	40 um
L	Physical length	Length	L[a]
MSUB	Substrate definition	Text	MSUB1[b]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See the Microwave Office Layout Guide for details.

[b]Modify only if schematic contains multiple substrates. See "Using Elements With Model Blocks" for details.

Implementation Details

This circuit component models a section of two edge coupled asymmetrical microstrip lines.

The parameters W1, W2 (strip widths), S (gap between strips), and L (line length) are dimensions entered in the default length units. The parameter MSUB specifies the microstrip substrate element, which defines additional cross sectional parameters of the transmission line. If blank, a default is used.

NOTE:

This model has two implementations: basic implementation and modified implementation. Basic implementation provides frequency-dependent behavior that matches reference [1] [10–98]; unfortunately, formulation [1] [10–98] is not very good in low frequency and DC region. Modified implementation improves low frequency and DC behavior but demonstrates frequency behavior somehow different from those of basic implementation at frequencies below 10 GHz. For example, model may underestimate total loss if dielectric loss substantially exceeds conductor loss. MACLIN is not intended for modeling substrates with very large loss tangents (say, above 0.02): Loss of this magnitude should be modeled by EM Quasi-Static models.

How to switch between implementations

Right-click on schematic that contain MCLIN model(s), select Options from pull-down menu, than select Modeling Options tab, go to pull-down list "Model compatibility version".

To set all MCLIN models in this schematic to basic implementation select Version 5.53 from pull-down list.

To set all MCLIN models in this schematic to modified implementation select Version 6.0 or Version 6.5 from pull-down list.

Note that switch also affects MCLIN, MCFIL, SCLIN.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See “Cell Options Dialog Box: Layout Tab ” for Cell Options dialog box **Layout** tab details.

See “The Layout Process File (LPF)” for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

Recommendations for Use

Modified implementation of MACLIN better works in time-domain than basic implementation; however, in critical time-domain designs EM Quasi-Static models like GM2CLIN are preferable.

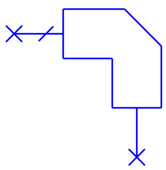
References

[1] R.Mongia, I.Bahl, and P.Bhartia, RF and Microwave Coupled-Line Circuits. Boston: Artech House, 1999, ch.4.

[2] P.K. Ikalainen and G.L. Matthaei, "Wideband, Forward-Coupling Microstrip Hybrids With High Directivity," IEEE Trans. Microwave Theory Tech., vol. MTT-35, Aug. 1987, pp. 719-725

(Obsolete) Microstrip Bend Mitered Arbitrary Angle (Closed Form): MBEND

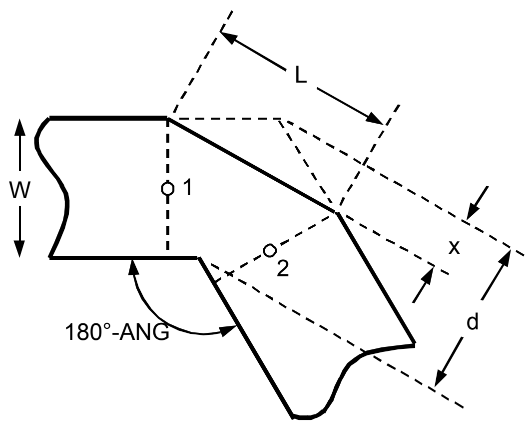
Symbol



Summary

This element is OBSOLETE and is replaced by the Microstrip Bend Mitered 90deg (EM Based) (MBEND90X) element if ANG is set to "90 Deg", otherwise it is replaced by the Microstrip Bend Optimally Mitered Arbitrary Angle (Closed Form) (MBENDA) element.

Topology



Parameters

MBEND$ is a Microstrip iCell and has no parameters. See "Intelligent Cells (iCells)" for more information.

Name	Description	Unit Type	Default
ID	Name	Text	TL1
W	Conductor width	Length	W[a]
ANG	Angle of bend	Angle	90 Deg
M	Miter size		0
MSUB	Substrate definition	Text	MSUB#[b]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See Microwave Office Layout Guide for details.

[b]Modify only if schematic contains multiple substrates. See the "Using Elements With Model Blocks" for details.

Implementation Details

MSUB indicates the name of the MSUB element to use for substrate characteristics. If blank, a default is used. The reference planes for the nodes are indicated by the dashed lines. The width of the line at both nodes is given by W. The amount of bend is given by ANG. The bend is mitered according to M, defined as M = x/d with the dimensions related by

$$d - x = \sqrt{2} \cdot W \cdot (1 - M)$$

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See "Cell Options Dialog Box: Layout Tab " for Cell Options dialog box **Layout** tab details.

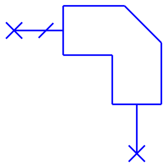
See "The Layout Process File (LPF)" for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

Reference

[1] B. Wadell, Transmission Line Design Handbook, p. 294,1991.

(Obsolete) Microstrip Bend Mitered 50% Mitered 90deg Bend (Closed Form): MBEND2

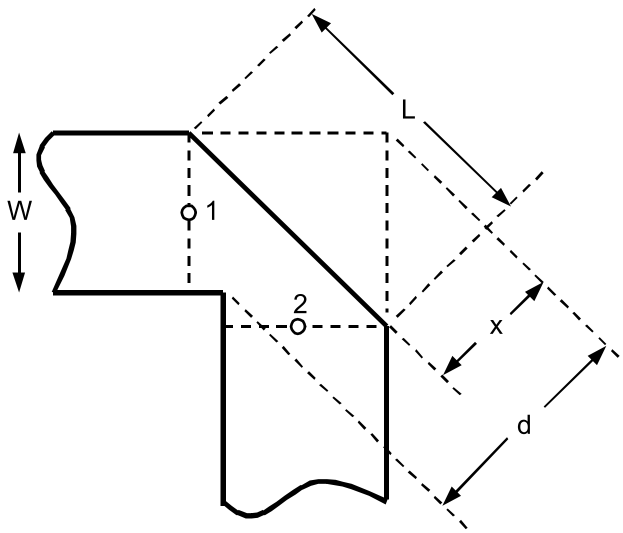
Symbol



Summary

This element is OBSOLETE and is replaced by the Microstrip Bend Mitered 90deg (EM Based) (MBEND90X) or Microstrip Bend Optimally Mitered Arbitrary Angle (Closed Form) (MBENDA) element.

Topology



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	TL1
W	Conductor width	Length	W[a]
MSUB	Substrate definition	Text	MSUB#[b]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See Microwave Office Layout Guide for details.

[b]Modify only if schematic contains multiple substrates. See the "Using Elements With Model Blocks" for details.

Implementation Details

MSUB indicates the name of the MSUB element to use for substrate characteristics. If blank, a default is used. The reference planes for the nodes are indicated by the dashed lines. The width of the line at both nodes is given by W. The right-angle bend is 50% mitered, that is, chamfered so that line of cut has 45 degrees skew.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See “Cell Options Dialog Box: Layout Tab ” for Cell Options dialog box **Layout** tab details.

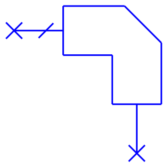
See “The Layout Process File (LPF)” for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

Reference

[1] M.Kirschning, R.H.Jansen, and N.H.L.Koster, "Measurement and computer-aided modeling of microstrip discontinuities by an improved resonator method," IEEE Microwave Theory Tech. Symp. Digest, 1983, pp. 495-497.

(Obsolete) Microstrip Bend Optimally Mitered 90deg (Closed Form): MBEND3

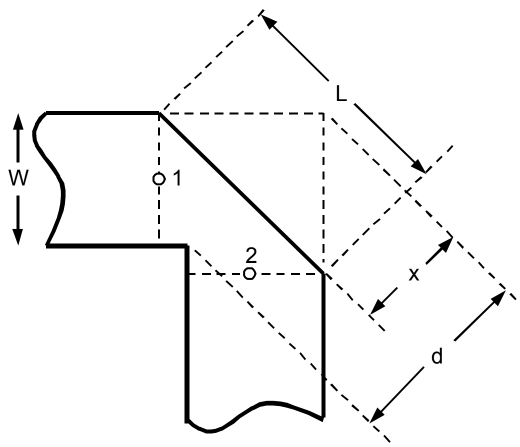
Symbol



Summary

This element is OBSOLETE and is replaced by the Microstrip Bend Mitered 90deg (EM Based) (MBEND90X) or Microstrip Bend Optimally Mitered Arbitrary Angle (Closed Form) (MBENDA) element.

Topology



Parameters

MBEND3$ is a Microstrip iCell and has no parameters. See "Intelligent Cells (iCells)" for more information.

Name	Description	Unit Type	Default
ID	Name	Text	TL1
W	Conductor width	Length	W[a]
MSUB	Substrate definition	Text	MSUB#[b]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See Microwave Office Layout Guide for details.

[b]Modify only if schematic contains multiple substrates. See the "Using Elements With Model Blocks" for details.

Implementation Details

MSUB indicates the name of the MSUB element to use for substrate characteristics. If blank, a default is used. The reference planes for the nodes are indicated by the dashed lines. The width of the line at both nodes is given by W. The right-angle bend is optimally mitered, with the dimensions related by

$$\frac{L}{W} = \sqrt{2} \cdot \left(1.04 + 1.3\varepsilon^{-1.35\frac{W}{b}}\right)$$

$$d - x = \sqrt{2} \cdot w - \frac{L}{2}$$

with the following restrictions

$$\frac{W}{b} \geq 25$$

$$\varepsilon_r \leq 25$$

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See “Cell Options Dialog Box: Layout Tab ” for Cell Options dialog box **Layout** tab details.

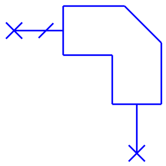
See “The Layout Process File (LPF)” for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

Reference

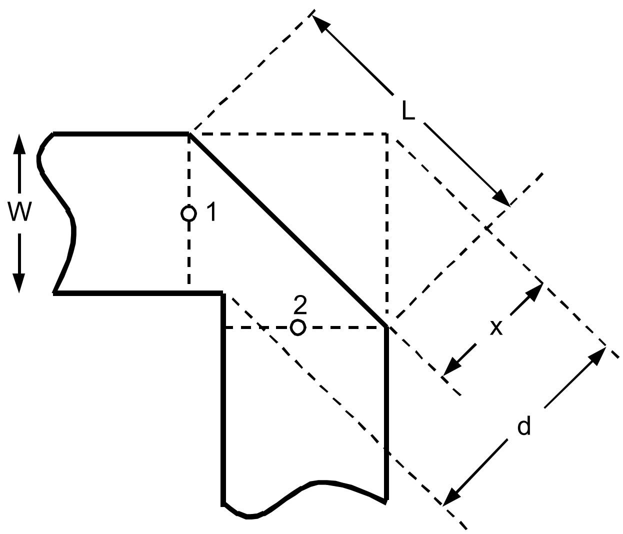
[1] B. Wadell, Transmission Line Design Handbook, p. 291,1991.

Microstrip Bend Mitered 90deg (EM Based): MBEND90X

Symbol



Topology



Parameter

Name	Description	Unit Type	Default
ID	Element ID	Text	MS1
W	Conductor width	Length	W[a]
M	Miter fraction (0-0.9)		0
MSUB	Substrate definition	Text	MSUB[b]
*AutoFill	AutoFill dataBase if not equal to 0		0

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See Microwave Office Layout Guide for details.

[b]Modify only if schematic contains multiple substrates. See the "Using Elements With Model Blocks" for details.

** indicates a secondary parameter*

Parameter Details

W. The intelligent version of this model automatically assumes this width is equal to the attached models at that node.

Implementation Details

This model implements an X-model of the Microstrip 90x Mitered Bend. For a more detailed discussion of the X-models, see *"Schematics and System Diagrams"*. This model does not include the effects of dielectric, conductor or radiation losses.

The bend is mitered according to M, defined as M = x/d with the dimensions related by

$$d - x = \sqrt{2} \cdot w \cdot (1 - M)$$

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See "Cell Options Dialog Box: Layout Tab " for Cell Options dialog box **Layout** tab details.

See "The Layout Process File (LPF)" for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

Restrictions

$0.25 \le W/H \le 8$ Recommended	ErNom $\le$ 16.0; Recommended
$0 \le$ Frequency $\le F_{max}$[a] Recommended	ErNom $\ge$ 1 ; Required
$0 \le M \le 0.99$ Required	(100*|ErNom-ErNom|)/ErNom $\le$ 10; Recommended
$M \le 0.90$ Recommended	(100*|ErNom-ErNom|)/ErNom $\le$ 20; Required

[a]Fmax(3)= The frequency limits of this model are dynamic with respect to the dimensions of the discontinuity.

F_{max} dynamic frequency limit is displayed to the user via warning messages for the relative size, and dielectric in use. Importantly, this recommended frequency limit will change as a function of the largest width in the discontinuity for a given substrate definition. The frequency limit warns the user that at least one of the transmission lines constructing the discontinuity is approaching the frequency limit where multiple modes can propagate. See "Upper Frequency Limitations" for more information.

In this table, H and ErNom are MSUB parameters.

Recommendations for Use

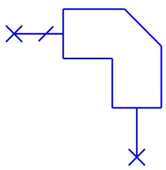
Discontinuity models function most accurately when attached to lines that match their corresponding edges. Directly connecting discontinuity models to one another reduces their accuracy.

References

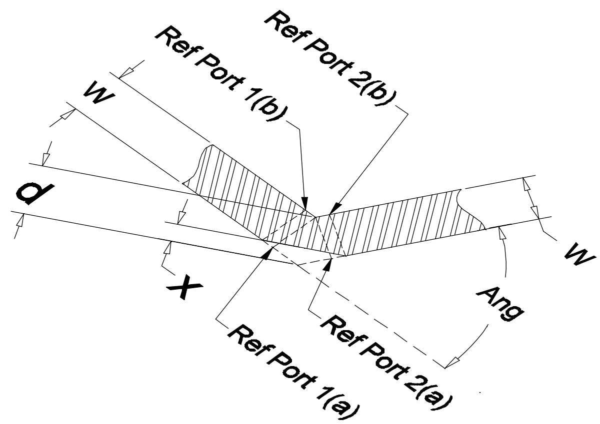
This model was developed under research performed at NI AWR Corporation. and the details of the implementation are considered proprietary in nature.

Microstrip Bend Optimally Mitered Arbitrary Angle (Closed Form): MBENDA

Symbol



Topology



Parameters

MBENDA$ is a Microstrip iCell and has no parameters. See "Intelligent Cells (iCells)" for more information.

Miter (M) is computed as a function of Angle and W/H.

Name	Description	Unit Type	Default
ID	Element ID	Text	TL1
W	Conductor width	Length	W[a]
ANG	Angle of bend	Angle	90 Deg
MSUB	Substrate definition	Text	MSUB#[b]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See the Microwave Office Layout Guide for details.

[b] Modify only if schematic contains multiple substrates. See "Using Elements With Model Blocks" for details.

Restrictions

0 ≤ ANG ≤ 120 Recommended

T/W ≤ 0.1 Recommended

W/H ≥ 0.25 Required

$\omega_r \leq 16.0$ Recommended

$1 \le \omega_r$ Required

Implementation Details

This circuit component models an optimum Miter of an arbitrary angled bend in the Microstrip signal conductor. The model assumes a Quasi TEM mode of propagation matched to a Microstrip line of width (W) and incorporates the effects of dielectric and conductive losses. The basis of the model is a curve fit of experimentally determined optimum miters at specific angles within the specified range. At this time, the optimal miter percentage must be interpolated from the following table. The parameter W (Strip Width) is a dimension entered in the default length units. The parameter MSUB specifies the Microstrip Substrate element, which defines additional cross sectional parameters of the transmission line. M = 100 (x/d)

W/H	0 Deg	30 Deg	60 Deg	90 Deg	120 Deg
0.5	0%	12%	45%	75%	98%
1.0	0%	19%	41%	63%	92%
2.0	0%	7%	31%	56%	79%

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See “Cell Options Dialog Box: Layout Tab ” for Cell Options dialog box **Layout** tab details.

See “The Layout Process File (LPF)” for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

Recommendations for Use

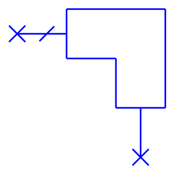
Discontinuity models function most accurately when attached to lines that match their corresponding edges. Directly connecting discontinuity models to one another reduces their accuracy.

References

Wadell, Brian C. "Transmission Line Design Handbook" Artech House 1991, pp. 294-296

(Obsolete) Microstrip Bend Unmitered 90deg (Closed Form): MBENDR

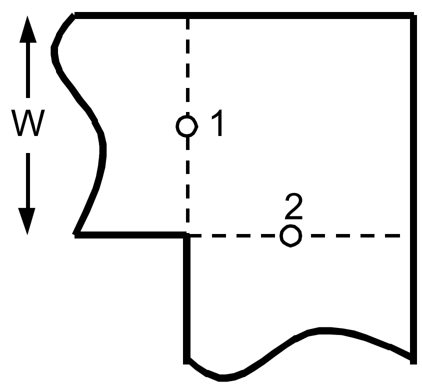
Symbol



Summary

This element is OBSOLETE and is replaced by the Microstrip Bend Mitered 90deg (EM Based) (MBEND90X) or Microstrip Bend Optimally Mitered Arbitrary Angle (Closed Form) (MBENDA) element.

Topology



Parameters

MBENDR$ is a Microstrip iCell and has no parameters. See "Intelligent Cells (iCells)" for more information.

Name	Description	Unit Type	Default
ID	Element ID	Text	TL1
W	Conductor width	Length	W[a]
MSUB	Substrate definition	Text	MSUB#[b]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See Microwave Office Layout Guide for details.

[b]Modify only if schematic contains multiple substrates. See the "Using Elements With Model Blocks" for details.

Implementation Details

MSUB indicates the name of the MSUB element to use for substrate characteristics. If blank, a default is used. The reference planes for the nodes are indicated by the dashed lines. The width of the line at both nodes is given by W. The right-angle bend is not mitered.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See "Cell Options Dialog Box: Layout Tab " for Cell Options dialog box **Layout** tab details.

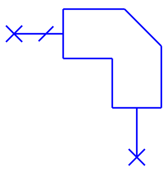
See "The Layout Process File (LPF)" for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

Reference

[1] B. Wadell, Transmission Line Design Handbook, p. 289,1991.

Microstrip Bend, Unmitered 90deg, Unequal Widths (EM Based): MBENDRWX

Symbol



Summary

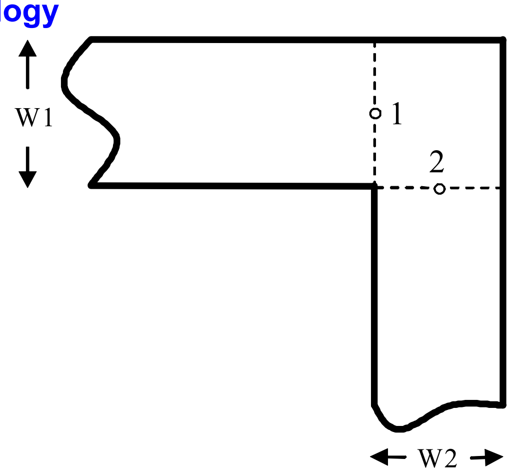
MBENDRWX models an umitered right bend of microstrip. Lines connected to MBENDRWX may have different widths.

This model is constructed as an X-model (table-based interpolation) using the results of EM analysis based on Method of Moments. For a more detailed discussion of the X-models see “ EM-based Models (X-models) ” .

This model does not include the effects of dielectric/conductor/radiation losses; it also implies that the conductor thickness is zero.

MBENDRWX$ is the corresponding intelligent cell (iCell). An iCell model is identical to its non-iCell equivalent with the following exception: Certain dimension-related parameters are not explicitly user specified; rather, they are automatically and dynamically determined by the dimensions of the attached elements. See “Intelligent Cells (iCells)” for a detailed discussion of how to use iCells, their advantages, and their limitations.

Topology



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	MS1
W1	Conductor width @ Port 1	Length	W[a]
W2	Conductor width @ Port 2	Length	W[a]
MSUB	Substrate definition	Text	See[b,c]

Name	Description	Unit Type	Default
*AutoFill	AutoFill dataBase if not equal to 0		0

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory.

[b]If only one MSUB is present in the schematic, this substrate is automatically used. If multiple MSUB substrate definitions are present, you must specify which to use.

[c]Modify only if the schematic contains multiple substrates.

* indicates a secondary Parameter

Parameter Details

W1, W2. Conductors widths are independent parameters.

MSUB. Substrate parameters are listed in the description of the MSUB model.

Autofill. The AutoFill parameter is a hidden input which allows you to specify that the entire interpolation table should be filled automatically at the current values. To instigate this filling process, this parameter should be set equal to 1. During normal operation, this parameter should be set equal to zero. You can access the hidden parameter by double-clicking on the schematic element.

Parameter Restrictions and Recommendations

1. MBENDRWX implies that the ratio Wmax/H lies within a **predefined** range of $0.25 \le W_{max}/H \le 8$ and the ratio Wmin/H lies within a **predefined** range of $0.2 \le W_{min}/W_{max} \le 1$ (H is substrate thickness, Wmax = max(W1,W2), Wmin =min(W1,W2)). Also, model implies that $W_{min}/H \ge 0.005$. Outside of these ranges, MBENDRWX extrapolates output parameters and issues a warning.
2. Bend is modeled as a two-port discontinuity. Reference planes corresponding to the ports are located as shown in Topology.
3. Nominal dielectric constant ErNom is fixed parameter. Microwave Office provides pre-generated (pre-filled) tables for a several typical values of MBENDRWX fixed parameter. Changes to any fixed parameter may start the automatic filling process if Autofill is set to 1. Changes to Tand and Rho do not affect results and do not instigate the tables filling because MBENDRWX always implies that Tand=0 and Rho=1.
4. You can change any fixed parameter to create corresponding tables.
5. The dielectric constant Er of the substrate MSUB is a statistical parameter. It means that models account for relative deviation of Er from ErNom within 20%; larger deviation demands a new fill of model tables.
6. The frequency limits of this model are dynamic with respect to the dimensions of the bend as discontinuity, to the line width. This dynamic frequency limit is displayed via warning messages for the relative size, and dielectric in use. Note that this recommended frequency limit changes as a function of the largest width in the discontinuity for a given substrate definition. The frequency limit warns you that at least one of the transmission lines constructing the discontinuity is approaching the frequency limit where multiple modes can propagate. See “Upper Frequency Limitations” for more information.

Implementation Details

This model was developed under research performed at NI AWR Corporation. The details of the implementation are considered proprietary in nature.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See "Cell Options Dialog Box: Layout Tab " for Cell Options dialog box **Layout** tab details.

See "The Layout Process File (LPF)" for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

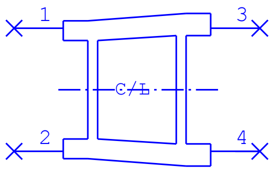
Recommendations for Use

To create new tables for a substrate dielectric constant different from those supplied with Microwave Office, you must set ErNom = Er = needed-value-of-dielectric-constant, set AutoFill = 1 and simulate. You should allow several hours (the actual time depends on your computer capabilities) for generating tables.

In exchange for speed increase, small errors resulting from the interpolation should be expected and the range of the input parameters is restricted.

Microstrip Branch-Line Coupler (Aggregate): MBLCOUP

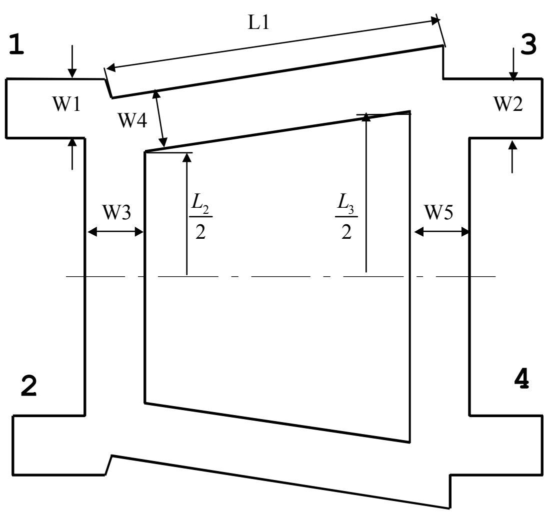
Symbol



Summary

MBLCOUP models microstrip branch-line coupler (90° hybrid). Model is a general one, i.e. it allows arbitrary lengths and widths of feeding lines as well as of series and shunt arms.

Topology



Parameters

Name	Description	Unit Type	Default
ID	Name	Text	TL1
W1	Width of feeding line at ports 1 and 2	Length	W[a]
W2	Width of feeding line at port 3 and 4	Length	W[a]
W3	Width of the line connecting ports 1 and 2	Length	W[a]
W4	Width of the line connecting ports 1,3 and 2,4	Length	W[a]
W5	Width of the Iine connecting ports 3 and 4	Length	W[a]
L1	Length of the line connecting ports 1,3 and 2,4	Length	L[a]
L2	Length of the Line connecting ports 1 and 2	Length	L[a]
L3	Length of the line connecting ports 3 and 4	Length	L[a]

Name	Description	Unit Type	Default
MSUB	Substrate definition	Text	MSUB1[b]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See the Microwave Office Layout Guide for details.

[b] Modify only if schematic contains multiple substrates. See "Using Elements With Model Blocks" for details.

Parameter Details

L1. Series arm connecting ports 1,3 and series arm connecting ports 2,4 have the same length L1.

L2. Full length of shunt arm connecting ports 1 and 2.

L3. Full length of shunt arm connecting ports 3 and 4.

Parameter Restrictions and Recommendations

1. MBLCOUP uses MLIN and MSTEP (microstrip line and microstrip step of width) models, so all limitations of these models are applicable to MBLCOUP.
2. For equal split 3 dB coupler (equal power split and 90 ^{0} phase shift between ports 3 and 4):

   $W_1=W_2=W_3=W_5$;

   $L_1=L_2=L_3=\lambda_{eff}/4$; (quarter of effective wavelength)

   W4 provides characteristic impedance of series branches equal to 1/√2 times characteristic impedance of feeding lines.
3. Branch-line coupler with $L_2=L_3$ and $L_2 \neq L_3$ may provide more feasible design (see Reference [2] [10–116]).

Implementation Details

MBLCOUP implies that coupler is mirror symmetrical relative to horizontal axis (see Topology).

Recommendations for Use

Feeding lines at ports may have different characteristic impedances. This is provided by proper choice of width parameters W1, W2 (see Topology).

To ensure specified power split, matching, isolation and phase shift between branched lines parameters may be derived in accordance with the recommendations in References [1], [2]. However, user may set parameters to values of his choice or use them for circuit optimization.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

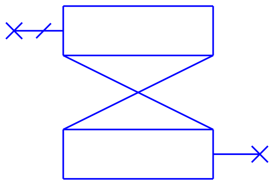
References

[1] Mongia, R., Bahl,I.J. and Bhartia P., RF and microwave coupled-line couplers, Artech House Inc., Norwood, MA, 1999, pp. 243-247

[2] Toker, C., Saglam, M., Ozme, M. and Gunalp, N.,"Branch-Line Couplers Using Unequal Line Lengths," IEEE Trans. Microwave Theory Tech., vol. MTT-49, April 2001, pp. 718-721

Coupled Microstrip Line Section for Filters (Closed Form): MCFIL

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Device ID	Text	TL1
W	Conductor widths	Length	W[a]
S	Gap width	Length	40 um
L	Conductor length	Length	L[a]
MSUB	Substrate definition	Text	MSUB1[b]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See the Microwave Office Layout Guide for details.

[b]Modify only if schematic contains multiple substrates. See the "Using Elements With Model Blocks" for details.

Implementation Details

MSUB indicates the name of the MSUB element to use for substrate characteristics. If blank, a default is used. This is a non-floating line. This is a coupled line model with the end effect included for the open ended line. One side of each coupled line is the ground plane.

W is the width of the line (the widths of the two lines must be the same), S is the width of the gap between the lines, and L is the physical length (in length units, not the electrical length in degrees).

This model is most accurate in the following ranges (where H is the substrate thickness):

$0.1 < W/H < 10$

$0.1 < S/H < 10$

$1 < \varepsilon_r < 18$

Outside these ranges, the accuracy degrades gracefully, and the results may still be useful. Therefore, no error message is displayed if these limits are exceeded.

NOTE:

This model has two implementations: basic and modified. Basic implementation provides frequency-dependent behavior that matches reference [1]; unfortunately, formulation [1] is not very good at low frequency and DC region. Modified implementation improves low frequency and DC behavior but demonstrates frequency behavior somehow different from those of basic implementation at frequencies below 10 GHz. For example, the model may underestimate total loss if dielectric loss substantially exceeds conductor loss. MCFIL is not intended for modeling substrates with exorbitant loss tangents (for example, above 0.02): Loss of this magnitude should be modeled by EM quasi-static models.

To switch between implementation, right-click the schematic that contains the MCFIL model(s), choose **Options**, then on the **AWR Sim** tab, specify the **Model compatibility version**.

To set all MCFIL models in this schematic to basic implementation, select **Version 5.5**.

To set all MCFIL models in this schematic to modified implementation, select **Version 6.0** or **Version 6.5**.

Note that this selection also affects MCLIN, MACLIN, and SCLIN models.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See "Cell Options Dialog Box: Layout Tab " for Cell Options dialog box **Layout** tab details.

See "The Layout Process File (LPF)" for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

Recommendations for Use

Modified implementation of MCFIL is specifically recommended for use in time-domain designs.

Reference

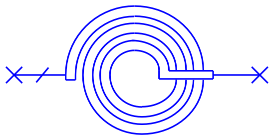
[1] M. Kirschning and R. H. Jansen, "Accurate Wide-Range Design Equations for the Frequency-Dependent Characteristic of Parallel Coupled Microstrip Lines," IEEE Trans. Microwave Theory Tech., Vol. MTT-32, p. 83, 1984.

[2] D. Kajfez, Z. Paunovic, and S. Pavlin, "Simplified Design of Lange Coupler," IEEE Trans. Microwave Theory Tech., Vol. MTT-26, Oct., 1978, p. 806.

[3] N. Alexopoulos and S. Maas, "Characteristics of Microstrip Directional Couplers on Anisotropic Substrates," IEEE Trans. Microwave Theory Tech., Vol. MTT-30, No. 8, pp. 1267-70 (Aug., 1982).

Circular Microstrip Inductor Without Bridge: MCINDN

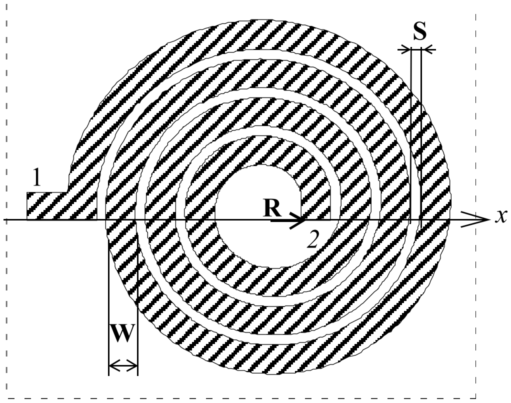
Symbol



Summary

This circuit component models a microstrip round spiral inductor without a bridge. You are free to implement any bridge model to attach port 2 to an external circuit. This model is based on an evaluation of self and mutual inductances, capacitances and resistances between all spiral turns.

Topology



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	MSP1
NT	Number of turns (>=1)		3.5
W	Conductor width	Length	10 um
S	Conductor spacing	Length	5 um
R	Inside radius of innermost turn	Length	15 um
MSUB	Substrate definition	Text	MSUB[a]

[a]Modify only if the schematic contains multiple substrates. See "Using Elements With Model Blocks" for details.

Parameter Details

NT. This is the number of spiral turns forming an inductor. NT is equal to the number of times the conductor intersects the axis X (except for the innermost crossing at port 2 - see "Topology") plus the fraction of turn from the last intersection to port 1. For example, "Topology" displays an inductor with NT=3.5.

R. This is a minimal radius of the inner side of the innermost turn.

Layout cell NSeg: Note that the corresponding layout cell of this model has parameter NSeg (to edit NSeg, select the corresponding layout cell, right- click it and choose **Shape Properties > Parameters**). NSeg defines the order of regular circumscribed polygon that is used to draw the spiral shape. The default value is 64. Larger values of Nseg make the contour smoother at the cost of slower drawing.

Parameter Restrictions and Recommendations

1. NT should not be less than 1
2. Model accuracy may decline if ratio W/S exceeds 5
3. The inductor size (diameter D_0 measured along axis X) may be evaluated as:

$$D_0 = 2R + 2(W+S)\mathrm{NT} - 0.5(W+S)$$

Implementation Details

To decrease the calculation time for schematics that contain several microstrip inductors, cache is implemented for this model. During the first evaluation of the schematic the most time consuming intermediate parameters for each inductor instance are stored in the memory cache. Each inductor model checks this cache looking for its duplicate. Duplicate inductors copy the appropriate parameters from the memory cache, saving substantially on their recalculation.

NOTE: The implementation of EM Quasi-Static models relies heavily on the involved numerical algorithms. This may lead to a noticeable increase in simulation time for schematics that employ many such models.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See "Cell Options Dialog Box: Layout Tab " for Cell Options dialog box **Layout** tab details.

See "The Layout Process File (LPF)" for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

Recommendations for Use

For new projects, you should use layout cell MCINDN2. To do so, select the MCINDN2 layout cell on the **Layout** tab of the Element Options dialog box.

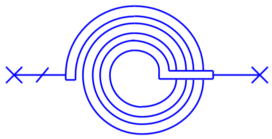
References

[1] M.B. Bazdar, A.R. Djordjevic, R.F. Harrington, and T.K. Sarkar, "Evaluation of quasi-static matrix parameters for multiconductor transmission lines using Galerkin's method," IEEE Trans. Microwave Theory Tech., vol. MTT-42, July 1994, pp. 1223-1228

[2] A.Niknejad, R.Meyer, "Analysis, design, and optimization of spiral inductors and transformers for Si RF IC's" IEEE Journal of Solid Circuits, vol.33, No 10, 1998, pp. 1470-1481.

Circular Microstrip Inductor With Strip Bridge (EM Quasi-Static): MCINDS

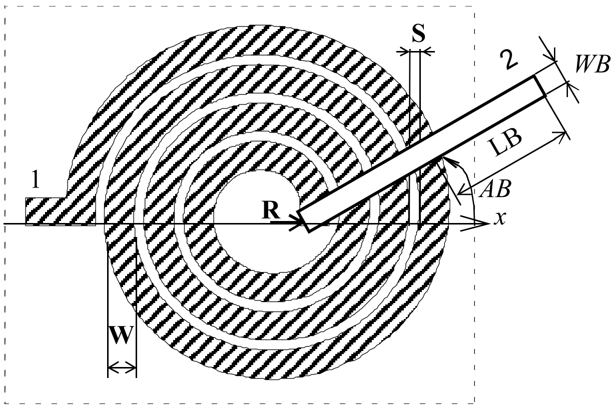
Symbol



Summary

This circuit component models a microstrip round spiral inductor with a strip bridge. This model is based on evaluation of self and mutual inductances, capacitances and resistances between all spiral turns. The bridge conductor crosses the inductor on its top and is capacitively coupled to all intersected turns.

Topology



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	MSP1
NT	Number of turns (>=1)		3.5
W	Conductor width	Length	10 um
S	Conductor spacing	Length	5 um
R	Inside radius of innermost turn	Length	15 um
AB	Angle of bridge departure	Angle	0 deg
WB	Width of bridge strip conductor	Length	10 um
HB	Height of bridge dielectric	Length	2 um
LB	Length of bridge extension beyond inductor	Length	0
EPSB	Relative dielectric constant of bridge dielectric		1
TDB	Loss tangent of bridge dielectric		0
TB	Thickness of bridge strip	Length	1 um
RhoB	Bridge metal bulk resistivity normalized to gold		1

Name	Description	Unit Type	Default
MSUB	Substrate definition	Text	MSUB1[a]

[a]Modify only if the schematic contains multiple substrates. See "Using Elements With Model Blocks" for details.

Parameter Details

NT. The number of spiral turns forming the inductor. NT is equal to the number of times the conductor intersects the axis X (except for the innermost crossing at port 2 - see the "Topology" section) plus the fraction of turn from the last intersection to port 1. For example, the "Topology" section displays an inductor with NT=3.5.

R. A minimal radius of the inner side of the innermost turn.

AB. Angle AB (degrees) defines the direction of bridge departure from the end of the last segment. Only 0, 90, 180 and 270 values are allowed. A zero angle has a bridge that is parallel to axis X and departs to the axis X positive direction; a 180-degree bridge departs in the opposite direction. Angle AB is measured counterclockwise. Any intermediate value of AB is set to the closest acceptable value.

LB. The bridge extension length from the outer conductor edge.

HB, EPSB, and TDB. These parameters characterize the dielectric layer that insulates the bridge from the inductor.

Layout cell NSeg. Note that the corresponding layout cell of this model has the NSeg parameter (to edit NSeg, select the corresponding layout cell, right- click and choose **Shape Properties** to display the Cell Options dialog box, then click the **Parameters** tab). NSeg defines the order of regular circumscribed polygon that is used to draw the spiral shape. The default value is 64. Larger values of Nseg (up to 360 allowed) make the contour smoother at the cost of slower drawing.

Layout cell BrGap (since v7.51). You should select the MCINDS2 layout cell on the **Layout** tab of the Element Options dialog box. The layout cell of this model has parameter BrGap (to edit BrGap select the corresponding layout cell, right-click and choose **Shape Properties** to display the Cell Options dialog box, then click the **Parameters** tab.) This parameter is valid only for certain line types, like Plated Metal Line, when the inductor body is comprised of two metals (top and bottom) connected by a via along all windings, and the bridge is built on the bottom metal. In this case, the bridge conductor may be short-circuited by windings at the bottom metal level. To avoid this, the layout cell provides a gap in the bottom metal to allow the bridge to exit from the inductor center untouched. BrGap is a user-defined extension to the width of this gap (the default is 5 microns). You should use the layout 3D view to check the actual gap width. In rare cases when parameter W is small and BrGap is relatively large, approximately half of the internal winding may be stripped of the bottom metal; in this case decreasing BrGap may help to restore the bottom metal back to internal winding.

Parameter Restrictions and Recommendations

1. NT should not be less than 1
2. This model allows an arbitrary value of NT>1, but *you should use the integer number of turns plus the integer number of quarter turns* because the layout cell uses NT rounded to a quarter of a turn and the model checks to determine if the bridge crosses port 1 in the layout (see restriction number 5).
3. Model accuracy may decline if ratio W/S exceeds 5.
4. The inductor size (diameter D_0 measured along axis X) may be evaluated as:

$$D_0 = 2R + 2(W + S)\mathrm{NT} - 0.5(W + S)$$

5. Certain combinations of NT and AB are not allowed after v7.51. This model does not allow the bridge to cross port 1 and changes AB to avoid this crossing (a warning displays and the layout displays the actual bridge position *if (and*

only if) you select layout cell MCINDS2. To this end, this model rounds NT to a quarter of a turn (just to evaluate the position of port 1 in the layout, the actual NT stays intact) and makes its choice: If NT=n (n is integer) then AB=0 is prohibited; if 0 is specified the model sets AB=270. If NT=n+1/2, then AB=180 is prohibited and the model sets AB=0. If NT=n+1/4, then AB=90 is off limits and the model sets AB=0. If NT=n+3/4, then AB=270 is prohibited and the model sets AB=0.

Implementation Details

To decrease the calculation time for schematics that contain several microstrip inductors, a cache is implemented for this model. During the first evaluation of the schematic the most time consuming intermediate parameters for each inductor instance are stored in the memory cache. Each inductor model checks this cache looking for its duplicate. Duplicate inductors copy the appropriate parameters from memory cache, saving substantially on their recalculation.

NOTE: The implementation of EM Quasi-Static models relies heavily on the involved numerical algorithms. This may lead to a noticeable increase in simulation time for schematics that employ many such models.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See “Cell Options Dialog Box: Layout Tab ” for Cell Options dialog box **Layout** tab details.

See “The Layout Process File (LPF)” for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

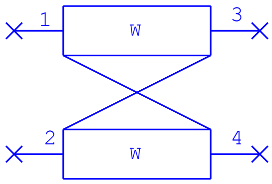
References

[1] M.B. Bazdar, A.R. Djordjevic, R.F. Harrington, and T.K. Sarkar, "Evaluation of quasi-static matrix parameters for multiconductor transmission lines using Galerkin's method," IEEE Trans. Microwave Theory Tech., vol. MTT-42, July 1994, pp. 1223-1228

[2] A.Niknejad, R.Meyer, "Analysis, design, and optimization of spiral inductors and transformers for Si RF IC's" IEEE Journal of Solid Circuits, vol.33, No 10, 1998, pp. 1470-1481.

Symmetric Edge Coupled Microstrip Lines (Closed Form): MCLIN

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Name	Text	TL1
W	Conductor width	Length	W[a]
S	Gap width	Length	40 um
L	Conductor length	Length	L[a]
MSUB	Substrate definition	Text	MSUB1[b]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See the Microwave Office Layout Guide for details.

[b]Modify only if schematic contains multiple substrates. See "Using Elements With Model Blocks" for details

Implementation Details

MSUB indicates the name of the MSUB element to use for substrate characteristics. If blank, a default is used. This is a non-floating line. One side of each coupled line is the ground plane.

W is the width of the line (the widths of the two lines must be the same), S is the width of the gap between the lines, and L is the physical length (i.e., in length units, not the electrical length in degrees).

This model is most accurate in the following ranges (where H is the substrate thickness):

$0.1 < W/H < 10$

$0.1 < S/H < 10$

$1 < \varepsilon_r < 18$

Outside these ranges, the accuracy degrades gracefully, and the results may still be useful. Therefore, no error message is given if these limits are exceeded.

Important

This model has two implementations: basic implementation and modified implementation. Basic implementation provides frequency-dependent behavior that matches reference [1] [10–126]; unfortunately, formulation [1] [10–126] is not very good in low frequency region and at DC. Modified implementation improves low frequency and DC behavior but demonstrates frequency behavior somehow different from those of basic implementation at frequencies well below 10 GHz: It can underestimate loss. Usually, difference is small but some loss-critical design like coupled line bandpass filters in 1-2 GHz range may be too optimistic in passband loss prediction. Also, modified implementation may underestimate total loss if dielectric loss substantially exceeds conductor loss.

To switch between implementation, right-click the schematic that contains the MCLIN model(s), choose **Options**, then on the **AWR Sim** tab, specify the **Model compatibility version**.

To set all MCLIN models in this schematic to basic implementation, select **Version 5.5**.

To set all MCLIN models in this schematic to modified implementation, select **Version 6.0** or **Version 6.5**.

Note that this selection also affects MACLIN, MCFIL, and SCLIN models.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See “Cell Options Dialog Box: Layout Tab ” for Cell Options dialog box **Layout** tab details.

See “The Layout Process File (LPF)” for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

Recommendations for Use

Basic implementation is recommended for frequency-domain applications.

Modified implementation is recommended for time-domain applications.

MCLIN (both implementations) is not intended for modeling substrates with very large loss tangents (say, above 0.02): Loss of this magnitude should be modeled by EM Quasi-Static models (M2CLIN, GM2CLIN).

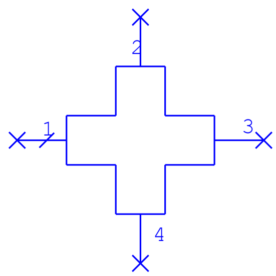
In critical frequency-domain and time-domain applications EM Quasi-Static models like M2CLIN or GM2CLIN are preferable.

Reference

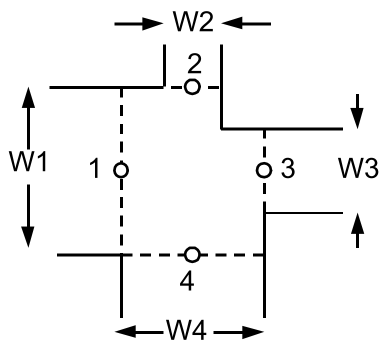
[1] M. Kirschning and R. H. Jansen, "Accurate Wide-Range Design Equations for the Frequency-Dependent Characteristic of Parallel Coupled Microstrip Lines," IEEE Trans. Microwave Theory Tech., Vol. MTT-32, p. 83, 1984.

Microstrip Cross - Junction (Closed Form): MCROSS

Symbol



Topology



Parameters

MCROSS$ is a Microstrip iCell and has no parameters. See “Intelligent Cells (iCells)” for more information.

Name	Description	Unit Type	Default
ID	Name	Text	TL1
W1	Conductor width @ Node 1	Length	W[a]
W2	Conductor width @ Node 2	Length	W[a]
W3	Conductor width @ Node 3	Length	W[a]
W4	Conductor width @ Node 4	Length	W[a]
MSUB	Substrate definition	Text	MSUB1[b]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See the Microwave Office Layout Guide for details.

[b]Modify only if the schematic contains multiple substrates. See “Using Elements With Model Blocks” for details.

Implementation Details

A junction of four microstrip lines. The reference planes for the junction are the ends of the respective lines; that is, the reference plane at any node is the side of the widest of the lines connected to that node. MSUB indicates the name of the MSUB element to use for substrate characteristics. If blank, a default is used.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See "Cell Options Dialog Box: Layout Tab " for Cell Options dialog box **Layout** tab details.

See "The Layout Process File (LPF)" for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

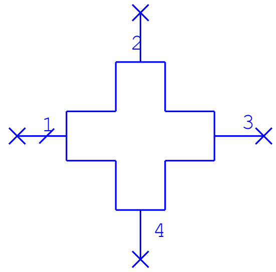
Recommendations for Use

This model does not provide rotation symmetry of results; it is fully symmetrical relative to the axis connecting terminals 2 and 4. A higher degree of accuracy may be reached using the MCROSS2 or MCROSSX models.

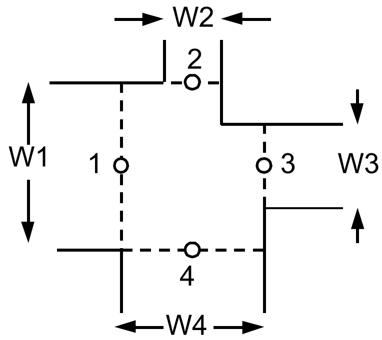
Discontinuity models function most accurately when attached to lines that match their corresponding edges. Directly connecting discontinuity models to one another reduces their accuracy.

Microstrip Cross - Junction (Closed Form): MCROSS2

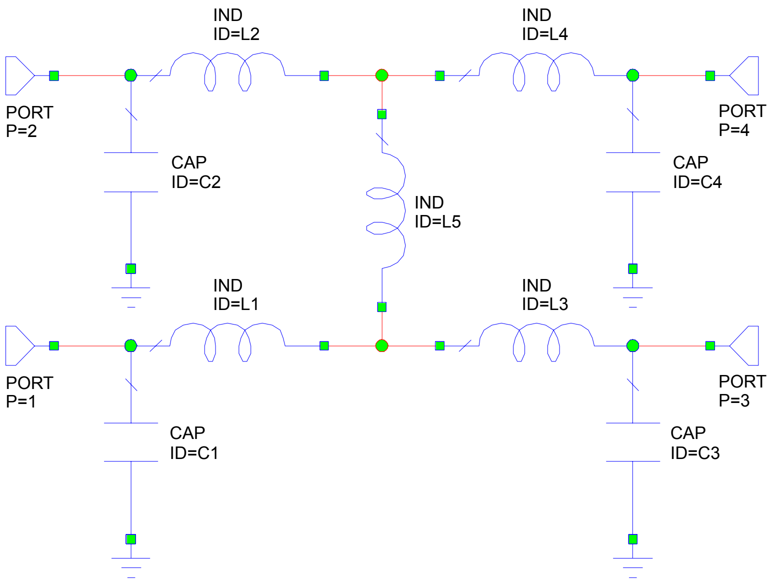
Symbol



Topology



Equivalent Circuit



Parameters

MCROSS2$ is a Microstrip iCell and has no parameters. See "Intelligent Cells (iCells)" for more information.

Name	Description	Unit Type	Default
ID	Name	Text	TL1
W1	Conductor width @ Node 1	Length	W[a]
W2	Conductor width @ Node 2	Length	W[a]
W3	Conductor width @ Node 3	Length	W[a]
W4	Conductor width @ Node 4	Length	W[a]
MSUB	Substrate definition	Text	MSUB1[b]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See the Microwave Office Layout Guide for details.

[b]Modify only if the schematic contains multiple substrates. See “Using Elements With Model Blocks” for details.

Parameter Restrictions and Recommendations

Where Wx is W1, W2, W3, or W4:

- Error Limits
  - Wx > 0
  - T > 0
  - H > 0
  - Er <= 50
- Warning Limits
  - 0.02 <= Wx/H <= 8.0

Implementation Details

A junction of four microstrip lines. The reference planes for the junction are the ends of the respective lines; that is, the reference plane at any node is the side of the widest of the lines connected to that node. MSUB indicates the name of the MSUB element to use for substrate characteristics. If blank, a default is used.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See “Cell Options Dialog Box: Layout Tab ” for Cell Options dialog box **Layout** tab details.

See “The Layout Process File (LPF)” for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

Recommendations for Use

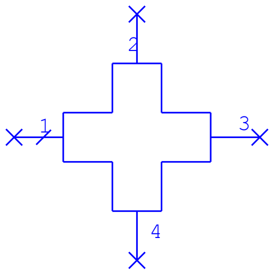
Discontinuity models function most accurately when attached to lines that match their corresponding edges. Directly connecting discontinuity models to one another reduces their accuracy.

References

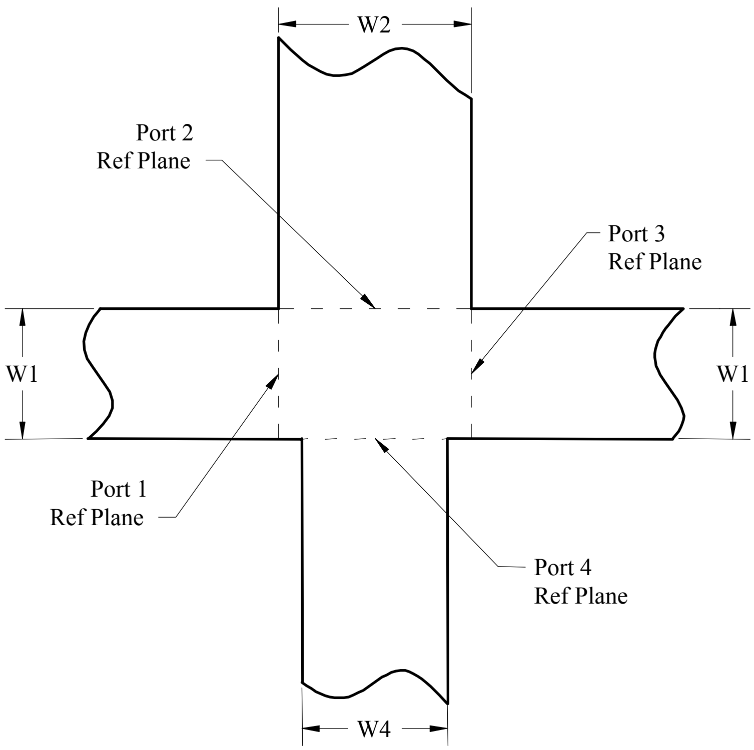
[1] K. C. Gupta, R. Garg, I. J. Bahl, and P. Bhartia, Microstrip Lines and Slotlines, 2nd ed., Artech House, Mass., 1996.

Microstrip Cross Junction (Closed Form): MCROSSX

Symbol



Topology



Parameters

MCROSSX$ is a Microstrip iCell and has no parameters. See "Intelligent Cells (iCells)" for more information.

Name	Description	Unit Type	Default
ID	Element ID	Text	MX1
W1	Conductor Width @ Node1&3	Length	W[a]
W2	Conductor Width @ Node 2	Length	W[a]
*W3	Conductor Width @ Node 3 (layout only)	Length	
W4	Conductor Width @ Node 4	Length	W1[a]

Name	Description	Unit Type	Default
MSUB	Substrate definition	Text	MSUB1[b]
*AutoFill	AutoFill database if not equal to 0	Integer	0

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See Microwave Office Layout Guide for details.

[b]Modify only if the schematic contains multiple substrates. See the "Using Elements With Model Blocks" for details.

** indicates a secondary parameter*

The intelligent version of this model automatically assumes these widths are equal to the attached models at that node.

Implementation Details

This model implements an X-model of the microstrip Cross-junction. For a more detailed discussion of the X-models see " EM-based Models (X-models) ". This model does not include the effects of dielectric, conductor or radiation losses, nor does it account for the effect of conductor thickness. Further, this model assumes that the widths at Port 1 and 3 are equal to W1.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See "Cell Options Dialog Box: Layout Tab " for Cell Options dialog box **Layout** tab details.

See "The Layout Process File (LPF)" for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

Restrictions

Independent Parameters:	W1, W2, W4 and Frequency
Scalable Parameters:	MSUB H- Substrate Height
Fixed Parameters:	MSUB Er Nominal
Statistical Parameters:	Er

$0.25 \leq W_{widest}/H \leq 4$ Required	$\varepsilon_{r\ nominal} \leq 16.0$ Recommended
$0.25 \leq W_{others}/W_{widest} \leq 1$	$1 \leq \varepsilon_{r\ nominal}$ Required
$0 \leq Frequency \leq F_{max}$ Recommended	$(100*\|\varepsilon_r-\varepsilon_{r\ nominal}\|)/\varepsilon_{r\ nominal} \leq 10\%$ Recommended
	$(100*\|\varepsilon_r-\varepsilon_{r\ nominal}\|)/\varepsilon_{r\ nominal} \leq 20\%$ Required

W_{widest} = The largest dimension of W1, W2, and W4

W_{others} = W1,W2, and W4 excluding W_{widest}

FW_{max}=The frequency limits of this model are dynamic with respect to the dimensions of the discontinuity. This dynamic frequency limit is displayed to the user via warning messages for the relative size, and dielectric in use. Importantly, this recommended frequency limit will change as a function of the largest width in the discontinuity for a given substrate definition. The frequency limit warns the user that at least one of the transmission lines constructing the discontinuity is approaching the frequency where multiple modes can propagate. See “Upper Frequency Limitations” for more information.

Recommendations for Use

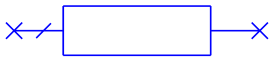
Discontinuity models function most accurately when attached to lines that match their corresponding edges. Directly connecting discontinuity models to one another reduces their accuracy.

References

This model was developed under research performed at NI AWR Corporation and the details of the implementation are considered proprietary in nature.

Meander Microstrip Line with Radiused Corners: MCTRACE

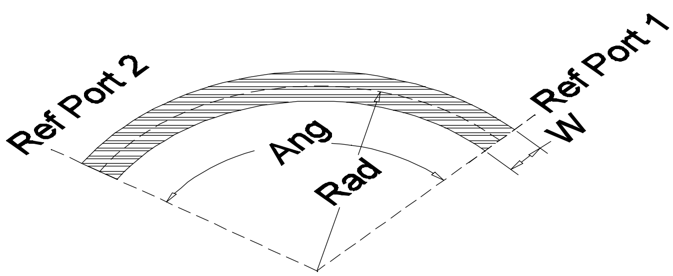
Symbol

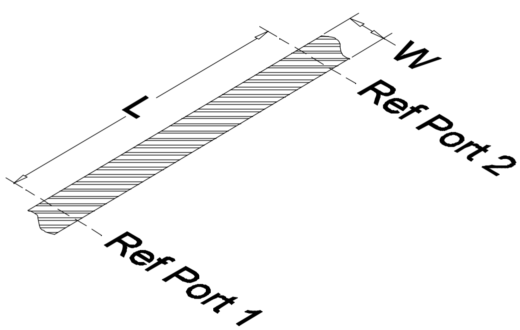


Summary

The MCTRACE element is a microstrip line that can be routed by graphically editing the element in the layout view. The model of the element is constructed by cascading curved microstrip bends (implemented either as MCURVE or MCURVE2) and microstrip lines (MLIN). You can specify what implementation of curved bend is used.

Topology





Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	TL1
W	Conductor width	Length	W[a]
L	Calculated center line length	Length	L[a]
R	Radius of curvature of centerline	Length	Rad[a]
MSUB	Substrate definition	Text	MSUB#[b]
*DB	Bend position array	Length	Changed in Layout

Name	Description	Unit Type	Default
*RB	Bend angle array (degrees)	Angle	Changed in Layout
BImp	Switch between curved bend implementations: "Closed Form"/"EM Based"		"Closed Form"
NHM	Number of higher modes. Effective if BImp="EM Based"		1

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory.

[b]If only one MSUB is present in the schematic, this substrate is automatically used. If multiple MSUB substrate definitions are present, the user must specify.

** indicates a secondary parameter*

Parameter Details

NHM. The number of higher modes accounted for in the equivalent model of curved waveguide with magnetic walls (see MCURVE2).

R. ("Topology" denotes it as Rad) A radius of the conductor centerline (the conductor is located symmetrically between the arcs of the internal radius R-W/2 and the external radius R+W/2).

Parameter Restrictions and Recommendations

1. The internal radius of the curved bend (R-W/2) should not be less than or equal to zero, so the model disallows setting the radius as R <=W/2.
2. If BImp="EM Based" then for the NHM parameter, MCTRACE accepts values in the range of 0≤NHM≤10. Generally, the default value of NHM=1 provides reasonable accuracy and a short computation. To improve accuracy, you may increase NHM to 3 at the expense of a longer computation. NHM >3 is not recommended due to the increased computational time without a gain in accuracy.
3. If BImp="EM Based" then for certain combinations of conductor width W and evaluation frequency, model may issue a warning: "EM Based curved bend model selected. Model operates above higher-mode cutoff frequency. Simulation results may be wrong in this region." To eliminate this message, you should reduce the upper limit of the frequency sweep.
4. For large values of R, MTRACE provides results consistent with a stretch of straight microstrip line of equal length; however, very large values of R are not recommended. This model limits the ratio R/W to 500.
5. If L < 2*DB or DB < 2*R, then the internal radius of the curved bend shrinks to be able to fit between the two straight segments of the MCTRACE.

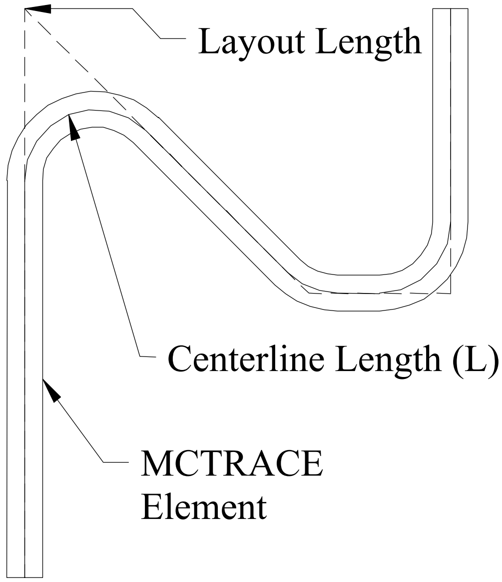
Implementation Details

If BImp="Closed Form" curved bend is modeled as a section of straight microstrip line with corrected length (similar to MCURVE).

If BImp="EM Based" curved bend is modeled as a section of curved rectangular waveguide with magnetic walls (similar to MCURVE2).

In general, graphical editing of MCTRACE is done using mouse commands and is exactly the same as MTRACE2. Double-clicking the element activates the blue grab diamonds used for manipulating the bends and lines.

One significant difference is noticeable when altering the MCTRACE element compared to MTRACE2. For MCTRACE, the model parameter, L, is altered when graphically altering the trace, such as adding or changing bends. In practice, the layout length (defined as follows) remains constant during these operations, while the center line length (parameter L) changes due to the shortening effect caused by the introducing or modifying curves.



Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See “Cell Options Dialog Box: Layout Tab ” for Cell Options dialog box **Layout** tab details.

See “The Layout Process File (LPF)” for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

Recommendations for Use

NOTE: If BImp="EM Based" NHM >3 may result in a longer computation and is generally not recommended.

References

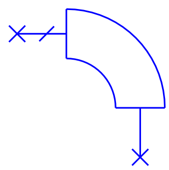
[1] Weisshaar, A., Tripathi, V.K. "Perturbation Analysis and Modeling of Curved Microstrip Bends," IEEE Trans. on MTT, Vol. 38, No. 10, October 1990, pp.1449-1454.

[2] Levin, L. et al, Electromagnetic waves and curved structures, London: Peter Peregrinus Ltd, 1977.

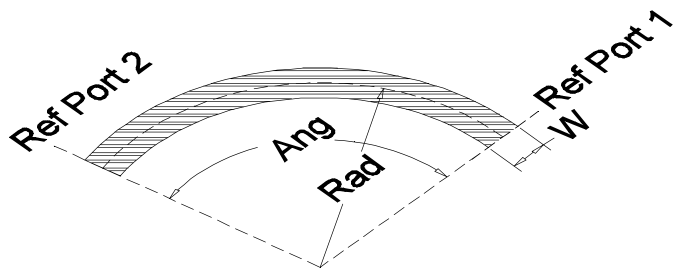
[3] Numerical Techniques for Microwave and Millimeter-Wave Passive Structures, Ed. by T. Itoh, New York: John Wiley & Sons, 1989, Chapter 7.

Microstrip Radiused Corner (Closed Form): MCURVE

Symbol



Topology



Parameters

MCURVE$ is a Microstrip iCell and has no parameters. See "Intelligent Cells (iCells)" for more information.

Name	Description	Unit Type	Default
ID	Element ID	Text	TL1
W	Conductor width	Length	40 um[a]
ANG	Angle of bend	Angle	90 Deg[a]
R	Radius of curvature to centerline	Length	50 um
MSUB	Substrate definition	Text	MSUB[b]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory.

[b]If only one MSUB is present in the schematic, this substrate is automatically used. If multiple MSUB substrate definitions are present, the user must specify.

Implementation Details

MSUB indicates the name of the MSUB element to use for substrate characteristics. If blank, a default is used. The reference planes for the nodes are indicated by the dashed lines. The width of the line at both nodes is given by W. The amount of bend is given by ANG.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.

3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See “Cell Options Dialog Box: Layout Tab ” for Cell Options dialog box **Layout** tab details.

See “The Layout Process File (LPF)” for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

Recommendations for Use

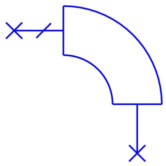
Discontinuity models function most accurately when attached to lines that match their corresponding edges. Directly connecting discontinuity models to one another reduces their accuracy.

Reference

[1] B. Wadell, Transmission Line Design Handbook, p. 297,1991.

Curved Microstrip Bend (EM Mode-Matching): MCURVE2

Symbol

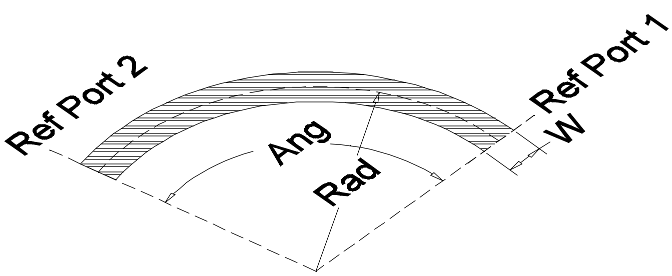


Summary

MCURVE2 models a stretch of microstrip line curved along a circular arc. Modeling is based on a mode matching technique that provides better accuracy than the MCURVE model, although at the expense of a longer computation.

MCURVE2$ is the corresponding intelligent cell (iCell). An iCell model is identical to its non-iCell equivalent with the following exception: Certain dimension-related parameters are not explicitly user-specified; rather, they are automatically and dynamically determined by the dimensions of the attached elements. See "Intelligent Cells (iCells)" for a detailed discussion of how to use iCells, and their advantages and limitations.

Topology



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	TL1
W	Conductor width	Length	W[a]
ANG	Angle of bend	Angle	90 deg
R	Radius of curvature to centerline	Length	L[a]
NHM	Number of higher modes		1
MSUB	Substrate definition	Text	MSUB[b]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory.
[b]Modify only if the schematic contains multiple substrates.

Parameter Details

NHM. The number of higher modes accounted for in the equivalent model of curved waveguide with magnetic walls.

R. ("Topology" denotes it as Rad) A radius of the conductor centerline (the conductor is located symmetrically between the arcs of the internal radius R-W/2 and the external radius R+W/2).

Parameter Restrictions and Recommendations

1. To avoid a decrease in accuracy, radius R should not be less than conductor width W, so MCURVE2 issues a warning if R <=W.
2. For the NHM parameter, MCURVE2 accepts values in the range of $0 \leq NHM \leq 10$. Generally, the default value of NHM=1 provides reasonable accuracy and a short computation. To improve accuracy, you may increase NHM to 3 at the expense of a longer computation. NHM >3 is not recommended due to the increased computational time without a gain in accuracy.
3. For certain combinations of conductor width W and evaluation frequency model may issue a warning: "MCURVE2 operates above the higher-mode cutoff frequency. Simulation results may be wrong in this region." To eliminate this message, you should reduce the upper limit of the evaluation frequency range.
4. MCURVE2 may lose accuracy if curvature of bend is too big, that is, ratio R/W is too small. Best accuracy is achieved if $R/W \geq 2$.
5. For large values of R, MCURVE2 provides results consistent with a stretch of straight microstrip line of equal length; however, very large values of R are not recommended. This model limits the ratio R/W to 500.

Implementation Details

This model was implemented after publications [1] [10–141] and [2] [10–142]. Two fundamental approximations are used:

- Propagation in the microstrip line is modeled as propagation in a rectangular waveguide with lateral magnetic walls (see the advantages and limitations of this method in [3] [10–142].)
- The perturbation technique is applied to obtain modes of curved rectangular waveguide (see [2] [10–142]).

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See “Cell Options Dialog Box: Layout Tab ” for Cell Options dialog box **Layout** tab details.

See “The Layout Process File (LPF)” for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

Recommendations for Use

NOTE: NHM >3 results in a long computation and generally is not recommended

References

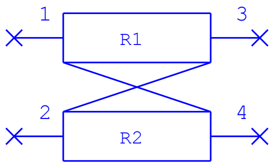
[1] Weisshaar, A., Tripathi, V.K. "Perturbation Analysis and Modeling of Curved Microstrip Bends," IEEE Trans. on MTT, Vol. 38, No. 10, October 1990, pp.1449-1454.

[2] Levin, L. et al, Electromagnetic waves and curved structures, London: Peter Peregrinus Ltd, 1977.

[3] Numerical Techniques for Microwave and Millimeter-Wave Passive Structures, Ed. by T. Itoh, New York: John Wiley & Sons, 1989, Chapter 7.

Multitrace Curved Coupled Microstrip Bend (Average-potential Method): MCURVEN

Symbol

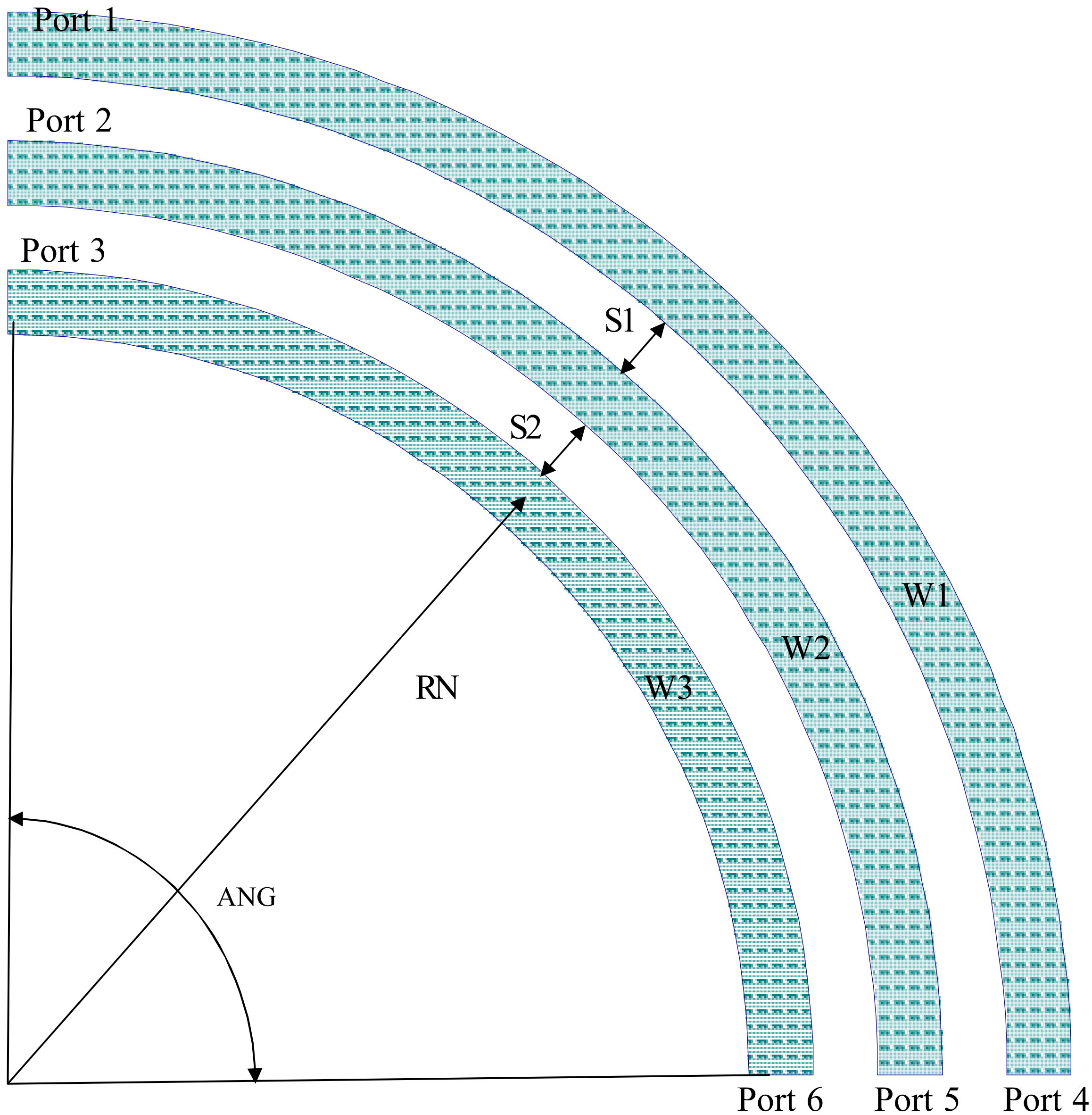


Summary

MCURVEN models a set of short coupled microstrip lines curved along circular arcs. Each microstrip arc is approximated by a set of thin straight segments and an RLGC network is built using the Average-potential technique in conjunction with the approximation of static Green's functions. The RLGC network represents coupling between all curved microstrip lines and accounts for resistive and dielectric losses.

MCURVEN is a dynamic or scalable model; it accepts a number of lines as input so the model and its schematic symbol expands/shrinks as the number of lines increases/decreases.

Topology



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	TL1
N	Number of conductors	Scalar	1
ANG	Angle of bend	Angle	90 deg
RN	Minimal trace radius	Length	30 microns
Nseg	Number of straight segments per conductor	Scalar	6
MSUB	Substrate definition	Text	MSUB[a]

Name	Description	Unit Type	Default
Wi, i=1..N	Width of conductor # i	Length	10 micron
Si, i=1..N-1	Spacing between edges of conductors # i and #i+1	Length	20 micron

[a]Modify only if the schematic contains multiple substrates.

Parameter Details

RN. Radius of the internal conductor (minimal length) centerline. Conductors are numbered from external conductor down to internal.

Si. Spacing between adjacent edges of the conductor #i and conductor #i+1. Note that spacing (like conductors) are numbered from external conductor down to internal.

Parameter Restrictions and Recommendations

1. Conductors should not overlap; ensure that Si> 0 is always true.
2. MCURVEN may lose accuracy if the bend is too tight or the conductor width is too big. This model targets bends, not buses, and is recommended for modeling narrow and relatively short curved traces, for example 2<R/W<10, so it is not as accurate when R/W>10.

Implementation Details

- Each curve is represented by a set of Nseg straight filament segments connected in series. All segments are coupled through capacitive and inductive networks. The average-potential technique developed for this model extends formulations from [1] [10–145] to the case of a grounded dielectric substrate.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See “Cell Options Dialog Box: Layout Tab ” for Cell Options dialog box **Layout** tab details.

See “The Layout Process File (LPF)” for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

Recommendations for Use

MCURVEN targets bends, not buses, so modeling of narrow (W/R<0.5) relatively short (R/W<10) traces is recommended.

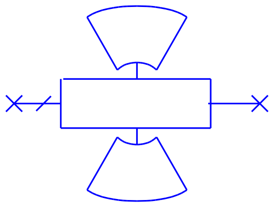
References

[1] Grover, F.W., Inductance Calculations, Mineola, New York: Dover Publishing, 2004

[2] Bahl, I.J., Lumped Elements for RF and Microwave Circuits, Norwood, MA: Artech House, 2003

(Obsolete) Microstrip Radial Stub Double Shunt (Closed Form): MDRSTUB

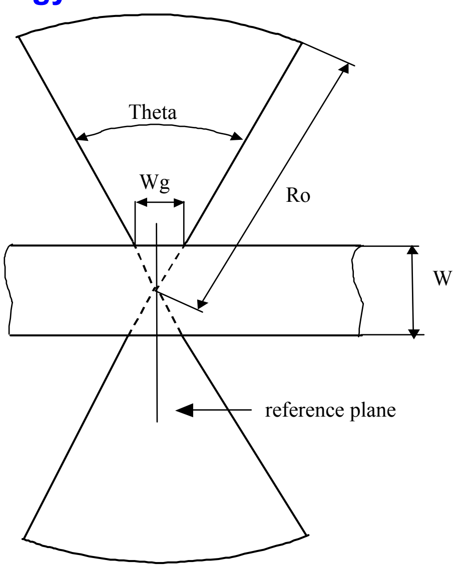
Symbol



Summary

This element is OBSOLETE and is replaced by the Radial Stub, Double Shunt (Closed Form) (MDRSTUB2) element.

Topology



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	ST1
Ro	Outer radius of the stub	Length	L[a]
W	Conductor width	Length	W[a]
Wg	Width of crossing of stub and microstrip line	Length	W[a]
Theta	Angle of the stub	Angle	90 Deg
MSUB	Substrate definition	Text	MSUB#[b]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See Microwave Office Layout Guide for details.

[b]Modify only if schematic contains multiple substrates. See the "Using Elements With Model Blocks" for details.

Implementation Details

This circuit component models a double-shunt "butterfly" radial stub on a single layer substrate (series stubs should be modeled with MRSTUB). Both stubs are of the same size.The model is a two port through element. The parameters W (width of the input microstrip), Wg (width of crossing of stub and microstrip line), and Ro (outer radius) are dimensions entered in the default length units. Theta is a radial stub angle entered in degrees. The parameter MSUB specifies the substrate element, which defines additional cross sectional parameters of the transmission line and substrate characteristics. If blank, a default is used. The model is valid up to its second open-circuit resonance and for $9 \leq \text{Theta} \leq 180$.

The component accounts for losses in metal and in substrate dielectric. Dispersion is not included.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See "Cell Options Dialog Box: Layout Tab " for Cell Options dialog box **Layout** tab details.

See "The Layout Process File (LPF)" for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

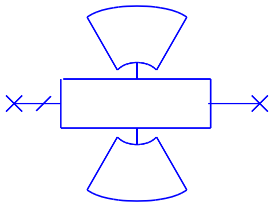
References

[1] S.L. March, "Analyzing lossy radial-line stubs," IEEE Trans. Microwave Theory Tech., vol. MTT-33, March 1985, pp. 269-271

[2] F.Giannini, R.Ruggieri, and J.Vrba, "Shunt-connected microstrip radial stubs", IEEE Trans. Microwave Theory Tech., vol. MTT-34, March 1986, pp. 363-366

Radial Stub, Double Shunt (Closed Form): MDRSTUB2

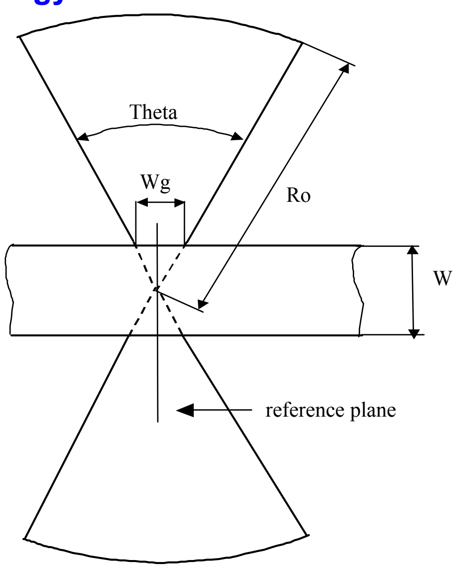
Symbol



Summary

MDRSTUB2 models a double-shunt "butterfly" microstrip radial stub that shunts microstrip line. Both stubs are the same size. The model is a two-port through element.

Topology



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	ST1
Ro	Outer radius of the stub	Length	L[a]
Wg	Width of crossing of stub and microstrip line	Length	W[a]
W	Conductor width	Length	W[a]
Theta	Angle of the stub	Angle	90 Deg
MSUB	Substrate definition	Text	MSUB1[b]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See the Microwave Office Layout Guide for details.

[b]Modify only if schematic contains multiple substrates. See "Using Elements With Model Blocks" for details.

Parameter Restrictions and Recommendations

1. Theta must be within the range $9^o \leq$ Theta $\leq 160^o$.
2. Model is feasible if

$$\left(R_O - \frac{Wg}{2sin(Theta/2)}\right) \Big/ R_O > 0.001$$

.

Implementation Details

Model implementation is based on cascading multiple segments of lossy microstrip transmission lines described in [1]. It accounts for losses in metal and in substrate dielectrics. Radiation loss is not included.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See “Cell Options Dialog Box: Layout Tab ” for Cell Options dialog box **Layout** tab details.

See “The Layout Process File (LPF)” for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

Recommendations for Use

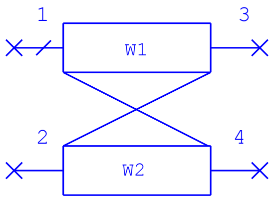
MDRSTUB2 replaces the MDRSTUB element, which is obsolete.

References

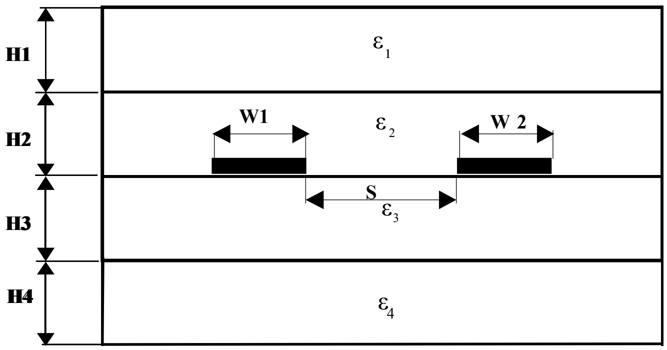
[1] B.C.Wadell, Transmission Line Design Book, Artech House, Boston-London, 1991, pp.306-307

2 Embedded Edge Coupled Microstrip Lines (EM Quasi-Static): MEM2LIN

Symbol



Topology



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	TL1
W1	Conductor 1 width - Nodes 1&&3	Length	W[a]
W2	Conductor 2 width - Nodes 2&4	Length	W[a]
S	Gap width conductors 1&2	Length	40 um
L	Conductor length	Length	L[a]
Acc	Accuracy parameter		1
MSUB4	Substrate definition	Text	MSUB4#[b]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See Microwave Office Layout Guide for details.

[b]If only one MSUB4 is present in the schematic, this substrate is automatically used. If multiple MSUB4 substrate definitions are present, the user must specify.

Implementation Details

This circuit component models a section of two coupled microstrip lines embedded into a four layered substrate.

The parameters W1, W2 (strips widths), S(spacing between lines) and L (line length) are dimensions entered in the default length units.

The parameter Acc is the accuracy parameter (1≤AccParam≤10). The default value for Acc is 1. If Acc is less than 1 or greater than 10 it is sets automatically to 2. Larger value of Acc increase density of mesh used in computations. Accuracy of model parameters may gain slightly from increasing Acc at the expense of noticeable growth of computation time. As a rule of thumb, good trade-off between accuracy and computation time is to set Acc to 1.

The parameter MSUB4 specifies the four-layer microstrip substrate element, which defines additional cross sectional parameters of the transmission line. If blank, a default is used. Conductors are implied to reside at the boundary between layer 2 and layer 3.

This component doesn't impose restrictions on the conductor thickness (thickness may be zero, positive, or negative). Negative thickness means that the conductor is recessed into the layer 3.

MEM2LIN may be used to implement various kinds of coupled striplines: buried, coated, inverted, and suspended simply by selecting relevant dielectric constants and thicknesses.

Covered microstrip line may be implemented with SEM2LIN (embedded coupled striplines).

The component accounts for losses in metal and in substrate dielectric. Dispersion is not included.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See “Cell Options Dialog Box: Layout Tab ” for Cell Options dialog box **Layout** tab details.

See “The Layout Process File (LPF)” for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

Recommendation

When implementing suspended, coated etc. lines don't make, if possible, thickness of any layer too small in comparison with other layers: the calculation time may noticeably grow.

Warning

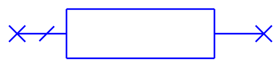
The implementation of EM Quasi-Static models heavily relies on the involved numerical algorithms. This may lead to the noticeable increase of computation time for some schematics that employ many models of this kind.

References

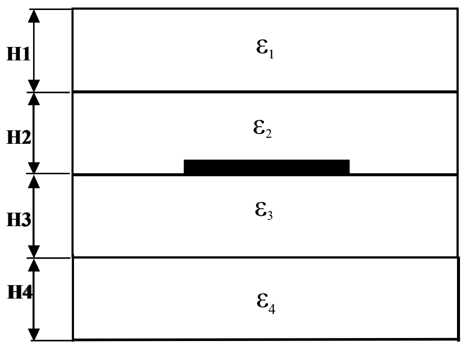
[1] M.B. Bazdar, A.R. Djordjevic, R.F. Harrington, and T.K. Sarkar, "Evaluation of quasi-static matrix parameters for multiconductor transmission lines using Galerkin's method," IEEE Trans. Microwave Theory Tech., vol. MTT-42, July 1994, pp. 1223-1228.

Embedded Microstrip Line (EM Quasi-Static): MEMLIN

Symbol



Topology



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	TL1
W	Conductor width	Length	W[a]
L	Conductor length	Length	L[a]
Acc	Accuracy parameter		1
MSUB4	Substrate definition	Text	MSUB4#[b]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See Microwave Office Layout Guide for details.

[b]Modify only if schematic contains multiple substrates. See "Using Elements With Model Blocks" for details.

Implementation Details

This circuit component models a section of single microstrip line embedded into four layered substrate.

The parameters W (strip widths) and L (line length) are dimensions entered in the default length units.

The parameter Acc is the accuracy parameter (1<=Acc<=10). The default value for Acc is 1. If Acc is less than 1 or greater than 10 it is sets automatically to 2. Larger value of Acc increase density of mesh used in computations. Accuracy of model parameters may gain slightly from increasing Acc at the expense of noticeable growth of computation time. As a rule of thumb, good trade-off between accuracy and computation time is to set Acc to 1.

The parameter MSUB4 specifies the four-layer microstrip substrate element, which defines additional cross sectional parameters of the transmission line. If blank, a default is used. Conductor is implied to reside on the boundary between layer 2 and layer 3.

This component doesn't impose restrictions on the conductor thickness (thickness may be zero, positive, or negative). Negative thickness means that the conductor is recessed into the layer 3.

MEMLIN may be used to implement various kinds of microstrip lines: buried, coated, inverted, and suspended simply by selecting relevant dielectric constants and thicknesses.

Covered microstrip line may be implemented with SEMLIN (embedded stripline).

The component accounts for losses in metal and in substrate dielectric. Dispersion is not included.

NOTES: It is recommended that when implementing suspended, coated etc. lines, do not make, if possible, the thickness of any layer too small in comparison with other layers or the calculation time may noticeably grow.

The implementation of EM Quasi-Static models heavily relies on the involved numerical algorithms. This may lead to the noticeable increase of computation time for some schematics that employ many models of this kind.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See "Cell Options Dialog Box: Layout Tab " for Cell Options dialog box **Layout** tab details.

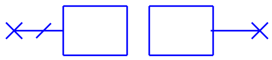
See "The Layout Process File (LPF)" for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

References

[1] M.B. Bazdar, A.R. Djordjevic, R.F. Harrington, and T.K. Sarkar, "Evaluation of quasi-static matrix parameters for multiconductor transmission lines using Galerkin's method," IEEE Trans. Microwave Theory Tech., vol. MTT-42, July 1994, pp. 1223-1228.

(Obsolete) Microstrip Gap in Conductor (Closed Form): MGAP

Symbol



Summary

This element is OBSOLETE and is replaced by the Microstrip Gap (EM Quasi-Static) (MGAP2) or Microstrip Gap (EM Based) (MGAPX) element.

Parameters

MGAP$ is a Microstrip iCell and has no parameters. See "Intelligent Cells (iCells)" for more information.

Name	Description	Unit Type	Default
ID	Name	Text	TL#[a]
W	Conductor width	Length	W[b]
S	Gap width	Length	W[b]
MSUB	Substrate definition	Text	MSUB[c]

[a]# is automatically incremented as additional TL Element ID's are added to the schematic.

[b]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory.

[c]If only one MSUB is present in the schematic, this substrate is automatically used. If multiple MSUB substrate definitions are present, you must specify which to use.

Implementation Details

W (width of microstrip conductor), S (length of the gap) are entered in the default length units.

MSUB indicates the name of the MSUB element to use for substrate characteristics. If blank, a default is used.

Model is represented by a symmetric π-circuit comprising two frequency-independent capacitances; it provides accuracy about 7% accurate in the ranges

$0.1 < S/W < 1.$

$2.5 < \varepsilon_r < 10$

where ε_r is the substrate relative permittivity.

NOTE: This model is considered obsolete due to low accuracy; MGAP2 is recommended for use instead MGAP.

Layout

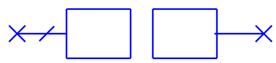
This element does not have a visual layout cell. The layout cell creates a gap between the two microstrip or stripline elements that connect to each side. The size of the gap is determined by the S parameter. Since there is no layer to draw, linetypes do not apply. You can select the layout cell to snap the microstrip or stripline elements to it.

References

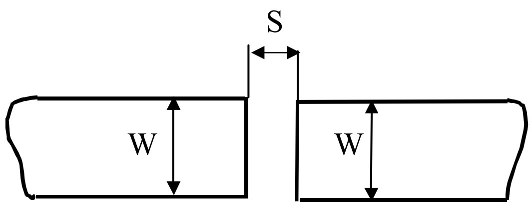
[1] R. Garg, I.Bahl, "Microstrip Discontinuities," International Journal of Electronics, vol. 45, No. 1, 1978, pp. 81-87.

Microstrip Gap (EM Quasi-Static): MGAP2

Symbol



Topology



Parameters

MGAP2$ is a Microstrip iCell and has no parameters. See “Intelligent Cells (iCells)” for more information.

Name	Description	Unit Type	Default
ID	Name	Text	TL#[a]
W	Conductor Width	Length	W[b]
S	Gap Width	Length	W[b]
MSUB	Substrate definition	Text	MSUB1[c]

[a]# is automatically incremented as additional TL Element ID's are added to the schematic.

[b]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory.

[c]If only one MSUB is present in the schematic, this substrate is automatically used. If multiple MSUB substrate definitions are present, you must specify which to use.

Implementation Details

W (width of microstrip conductor), S (length of the gap) are entered in the default length units.

MSUB indicates the name of the MSUB element to use for substrate characteristics. If blank, a default is used.

This model is valid within the following ranges (where H is the substrate thickness):

$0.5 < W/H < 2.5$

$0.1 < S/H < 1$

$1 < \varepsilon_r < 216$

Model is represented by a symmetric π-circuit comprising two frequency-independent capacitances. Model is EM-based because capacitance values are interpolated over the data collected with electromagnetic simulation at a single frequency point; model is quasi-static because this frequency is low enough to allow capacitance approximation and dispersion is not accounted for.

Layout

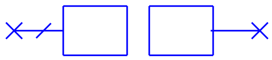
This element does not have a visual layout cell. The layout cell creates a gap between the two microstrip or stripline elements that connect to each side. The size of the gap is determined by the S parameter. Since there is no layer to draw, linetypes do not apply. You can select the layout cell to snap the microstrip or stripline elements to it.

Recommendations for Use

Discontinuity models function most accurately when attached to lines that match their corresponding edges. Directly connecting discontinuity models to one another reduces their accuracy.

Microstrip Gap (EM Based): MGAPX

Symbol

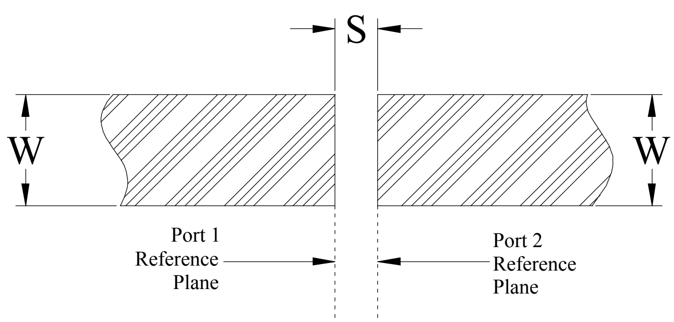


Summary

The MGAPX element models the electrical characteristics of a gap in the signal conductor for a microstrip transmission line system. The model is based upon multiple full wave electromagnetic simulations performed on the present substrate configuration. As the results of these EM simulations are stored, these lengthy simulations only have to be performed once.

MGAPX$ is the corresponding intelligent cell (iCell). An iCell model is identical to its non-iCell equivalent with the following exception: Certain dimension-related parameters are not explicitly specified by the user; rather, they are automatically and dynamically determined by the dimensions of the attached elements. See "Intelligent Cells (iCells)" for a detailed discussion of how to use iCells, their advantages, and their limitations.

Topology



Parameters

Name	Description	Unit Type	Default
ID	Name	Text	TL1
W	Conductor width	Length	W[a]
S	Gap width	Length	W[a]
MSUB	Substrate definition	Text	MSUB#[b]
*AutoFill	AutoFill database if not equal to 0		0

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See the Microwave Office Layout Guide for details.

[b]Modify only if schematic contains multiple substrates. See "Using Elements With Model Blocks" for details.

** indicates a secondary parameter*

Parameter Restrictions and Recommendations

1. 0.05 ≤ S/H ≤ 1.0 Recommended
2. 0.1 ≤ W/H ≤ 4.0 Recommend

Implementation Details

MGAPX is implemented as an EM-based model, or "X-model". An X-model is based on stored full-wave EM simulations and is the most accurate of models available. See " EM-based Models (X-models) " for a detailed discussion of X-models, their advantages, and their limitations.

Layout

This element does not have a visual layout cell. The layout cell creates a gap between the two microstrip or stripline elements that connect to each side. The size of the gap is determined by the S parameter. Since there is no layer to draw, linetypes do not apply. You can select the layout cell to snap the microstrip or stripline elements to it.

Recommendations for Use

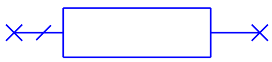
Discontinuity models function most accurately when attached to lines that match their corresponding edges. Directly connecting discontinuity models to one another reduces their accuracy.

References

This model was developed under research performed at NI AWR Corporation. The full set of details of the implementation are considered proprietary in nature.

Inverted Microstrip Line (EM Quasi-Static): MI1LIN

Symbol



Summary

MI1LIN models a section of single microstrip line with a conductor strip placed on the lower surface of a suspended substrate (suspended substrate is a single layer substrate elevated over the infinite grounded plane).

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	TL1
W	Conductor width	Length	W[a]
L	Conductor length	Length	L[a]
Acc	Accuracy parameter		1
MPSUB	Substrate definition	Text	MPSUB1[b]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See the Microwave Office Layout Guide for details.

[b]Modify only if schematic contains multiple substrates. See the "Using Elements With Model Blocks" for details.

Parameter Details

MPSUB. Suspended substrate parameters are listed in the MPSUB model description.

Acc. The accuracy parameter. The default value for Acc is 1. If Acc is less than 1 or greater than 10 it is set automatically to 2.

Parameter Restrictions and Recommendations

1. Accuracy parameter Acc is limited to $1 \le Acc \le 10$. A larger value of Acc increases the density of mesh used in computations. Accuracy of model parameters may gain slightly from increasing Acc at the expense of noticeable growth of computation time. As a rule of thumb, a good trade-off between accuracy and computation time is to set Acc to 1.
2. This component does not impose restrictions on the conductor thickness (thickness may be zero, positive, or negative). Negative thickness means that the conductor is recessed into the substrate.

Implementation Details

Model implementation is based on EM Quasi-Static technique described in [1] [10–160]. It accounts for losses in metal and in substrate dielectric. Dispersion is partly included

Recommendations for Use

NOTE: The implementation of EM Quasi-Static models relies heavily on the involved numerical algorithms. This may lead to a noticeable increase in simulation time for schematics that employ many such models.

If thickness of any layer is too small in comparison with the thickness of another layer, simulation time may also noticeably grow.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See “Cell Options Dialog Box: Layout Tab ” for Cell Options dialog box **Layout** tab details.

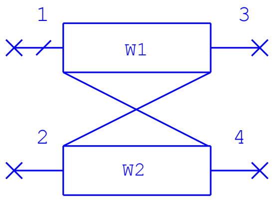
See “The Layout Process File (LPF)” for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

References

[1] M.B. Bazdar, A.R. Djordjevic, R.F. Harrington, and T.K. Sarkar, "Evaluation of quasi-static matrix parameters for multiconductor transmission lines using Galerkin's method," IEEE Trans. Microwave Theory Tech., vol. MTT-42, July 1994, pp. 1223-1228.

2 Inverted Edge Coupled Microstrip Lines (EM Quasi-Static): MI2CLIN

Symbol



Summary

MI2LIN models sections of two-edge coupled asymmetric microstrip lines with conductor strips placed on the lower surface of a suspended substrate (the suspended substrate is a single-layer substrate elevated over the infinite grounded plane).

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	TL1
W1	Conductor 1 Width - Nodes 1&13	Length	W[a]
W2	Conductor 2 Width - Nodes 2&14	Length	W[a]
W3	Conductor 3 Width - Nodes 3&15	Length	W[a]
W4	Conductor 4 Width - Nodes 4&116	Length	W[a]
W5	Conductor 5 Width - Nodes 5&17	Length	W[a]
W6	Conductor 6 Width - Nodes 6&18	Length	W[a]
W7	Conductor 7 Width - Nodes 7&19	Length	W[a]
W8	Conductor 8 Width - Nodes 8&20	Length	W[a]
W9	Conductor 9 Width - Nodes 9&21	Length	W[a]
W10	Conductor 10 Width - Nodes 10&22	Length	W[a]
W11	Conductor 11 Width - Nodes 11&23	Length	W[a]
W12	Conductor 12 Width - Nodes 12&24	Length	W[a]
S1	Gap Width Conductors 1&2	Length	W[a]
S2	Gap Width Conductors 2&3	Length	W[a]
S3	Gap Width Conductors 3&4	Length	W[a]
S4	Gap Width Conductors 4&5	Length	W[a]
S5	Gap Width Conductors 5&6	Length	W[a]
S6	Gap Width Conductors 6&7	Length	W[a]
S7	Gap Width Conductors 7&8	Length	W[a]
S8	Gap Width Conductors 8&9	Length	W[a]
S9	Gap Width Conductors 9&10	Length	W[a]
S10	Gap Width Conductors 10&11	Length	W[a]

Name	Description	Unit Type	Default
S11	Gap Width Conductors 11&12	Length	W[a]
L	Conductor length	Length	L[a]
Acc	Accuracy parameter		1
MPSUB	Substrate definition	Text	MPSUB1[b]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See the Microwave Office Layout Guide for details.

[b]Modify only if schematic contains multiple substrates. See "Using Elements With Model Blocks" for details.

Parameter Details

MPSUB. See MPSUB for a list of suspended substrate parameters.

Acc. The accuracy parameter. The default value is 1. If Acc is less than 1 or greater than 10 it is set automatically to 2.

Parameter Restrictions and Recommendations

1. The Accuracy parameter A is limited to 1≤Acc≤10. A larger value of Acc increases the density of mesh used in computations. The accuracy of model parameters may gain slightly from increasing Acc at the expense of a noticeably increased computation time. Generally, a good trade-off between accuracy and computation time is to set Acc to 1.
2. This component does not impose restrictions on the conductor thickness (thickness may be zero, positive, or negative). A negative thickness indicates that the conductor is recessed into the substrate.

Implementation Details

Model implementation is based on the EM Quasi-Static technique described in [1] [10–163]. It accounts for losses in metal and in substrate dielectric. Dispersion is partly included

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See "Cell Options Dialog Box: Layout Tab " for Cell Options dialog box **Layout** tab details.

See "The Layout Process File (LPF)" for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

Recommendations for Use

NOTE: The implementation of EM Quasi-Static models relies heavily on the involved numerical algorithms. This may lead to a noticeable increase in simulation time for schematics that employ many such models.

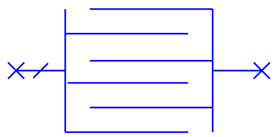
If the thickness of any layer is too small compared to another layer, simulation time may also noticeably grow.

References

M.B. Bazdar, A.R. Djordjevic, R.F. Harrington, and T.K. Sarkar, "Evaluation of quasi-static matrix parameters for multiconductor transmission lines using Galerkin's method," IEEE Trans. Microwave Theory Tech., vol. MTT-42, July 1994, pp. 1223-1228.

Microstrip Interdigital Capacitor (Aggregate): MICAP

Symbol

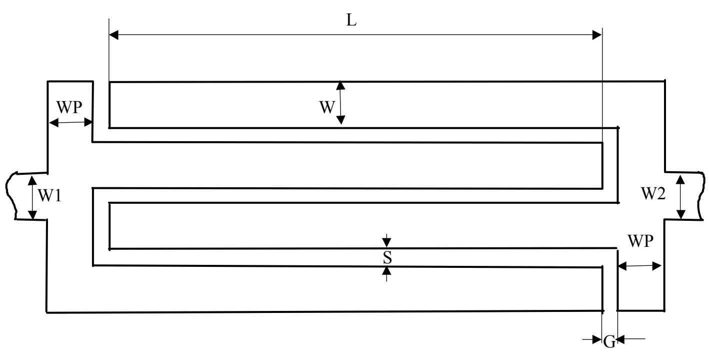


Summary

MICAP models a microstrip interdigital capacitor that has an emphasized transverse metallic strip connecting fingers at both input ports. MICAP takes advantage of the EM Quasi-Static coupled lines approach to considering interaction between all fingers.

MICAP$ is the corresponding intelligent cell (iCell). An iCell model is identical to its non-iCell equivalent with the following exception: Certain dimension-related parameters are not explicitly user-specified; rather, they are automatically and dynamically determined by the dimensions of the attached elements. See "Intelligent Cells (iCells)" for a detailed discussion of how to use iCells, their advantages, and their limitations.

Topology



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	MI1
W	Finger width	Length	W[a]
S	Spacing between adjacent fingers	Length	S[a]
G	End gap width	Length	S[a]
L	Length of the overlap region of the fingers	Length	L[a]
N	Number of fingers		4
WP	Width of the fingers transverse interconnect	Length	W[a]
W1	Width of the feeding line at port 1	Length	W[a]
W2	Width of the feeding line at port 2	Length	W[a]

Name	Description	Unit Type	Default
MSUB	Substrate definition	Text	MSUB1[a]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See the Microwave Office Layout Guide for details.

Parameter Details

W1. This parameter is secondary for the MICAP$ iCell model.

W2. This parameter is secondary for the MICAP$ iCell model.

Parameter Restrictions and Recommendations

1. The number of fingers N must be 4 ≤N≤ 16. You should use the MICAP1 model instead for all capacitors with N=2 and N=3.
2. Conductor thickness is set via substrate parameters. MICAP does not impose restrictions on thickness except for the requirement to be non-negative.

Implementation Details

The EM Quasi-Static technique allows you to model a microstrip interdigital capacitor with a wide range of conductor thicknesses.

MICAP accounts for the effect of phase shift along the microstrip line that connects fingers. It also includes the effect of the presence of width steps at ports.

To apply Method of Moments for analysis, a quasi-static model creates 1D mesh covering contours of all conductors. The mesh is made of linear segments (pulses) of varying length. The length of a pulse is relatively big at the conductor center; it decreases toward the conductor edges to reveal the charge distribution across the conductor. If the conductor width is prohibitively large it may cause the pulse size to approach zero for pulses close to the edge. In these rare cases the model may display a “Length of pulse #nnn equal to zero” error message.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See “Cell Options Dialog Box: Layout Tab ” for Cell Options dialog box **Layout** tab details.

See “The Layout Process File (LPF)” for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

Recommendations for Use

This model should be used for microstrip interdigital capacitors that have a prominent metallic strip of width WP connecting fingers at both ports. If this is not so, and one or both widths of feeding lines exceed 50% of capacitor width, you should use the MICAP1 model. MICAP1 is geared toward interdigital capacitors that are incorporated into microstrip line having a width equal to that of the capacitor.

MICAP accounts for losses in metal and in substrate dielectric. Dispersion is partly included.

To decrease the calculation time for schematics that contain several microstrip interdigital capacitors, cache is implemented for this model. This means that during the first evaluation of a schematic the most time-consuming intermediate parameters for each capacitor instance are stored in memory cache. Each interdigital capacitor model checks this cache looking for its duplicate. Duplicate capacitors copy the appropriate parameters from the memory cache, saving substantially on their recalculation.

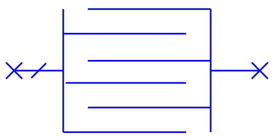
NOTE: The implementation of EM Quasi-Static models relies heavily on the involved numerical algorithms. This may lead to a noticeable increase in simulation time for schematics that employ many such models.

References

[1] B. Bazdar, A.R. Djordjevic, R.F. Harrington, and T.K. Sarkar, "Evaluation of quasi-static matrix parameters for multiconductor transmission lines using Galerkin's method," IEEE Trans. Microwave Theory Tech., vol. MTT-42, July 1994, pp. 1223-1228.

Microstrip Interdigital Capacitor, (No Steps at Ports) (EM Quasi-Static): MICAP1

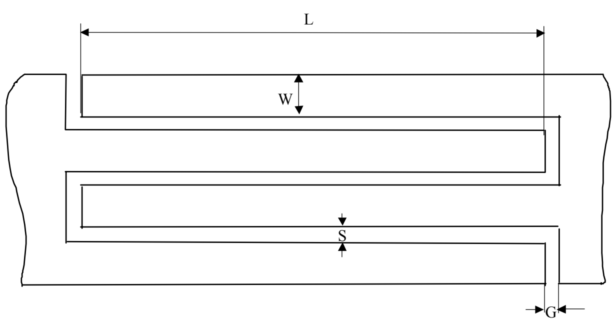
Symbol



Summary

MICAP1 models a microstrip interdigital capacitor that connects to microstrip lines without width steps. Model takes advantage of EM Quasi-Static coupled lines approach to consider interaction between all fingers.

Topology



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	M1
W	Finger width	Length	W[a]
S	Spacing between adjacent fingers	Length	S[a]
G	End gap width	Length	S[a]
L	Length of the overlap region of the fingers	Length	L[a]
N	Number of fingers		4
W1	Width of the feeding line at port 1 (Read-Only parameter)	Length	W[a]
W2	Width of the feeding line at port 2 (Read-Only parameter)	Length	W[a]
MSUB	Substrate definition	Text	MSUB1[b]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See Microwave Office Layout Guide for details.

[b]Modify only if schematic contains multiple substrates. See the “Using Elements With Model Blocks” for details.

Parameter Details

W1, W2. Parameters W1, W2 are read-only (output) parameters, represent the capacitor total width at ports 1 and 2 and are evaluated in the model as N(W + S)-S.

Parameter Restrictions and Recommendations

1. The number of fingers N must be $2 \leq N \leq 16$.
2. Conductor thickness is set via substrate parameters.Model doesn't impose restrictions on thickness except requirement to be non-negative.

Implementation Details

EM Quasi Static technique allows user to model microstrip interdigital capacitor with wide range of conductor thicknesses.

This model accounts for the effect of phase shift along the microstrip line that connects fingers. It also includes the effect of the presence of width step at one port.

To apply Method of Moments for analysis, a quasi-static model creates 1D mesh covering contours of all conductors. The mesh is made of linear segments (pulses) of varying length. The length of a pulse is relatively big at the conductor center; it decreases toward the conductor edges to reveal the charge distribution across the conductor. If the conductor width is prohibitively large it may cause the pulse size to approach zero for pulses close to the edge. In these rare cases the model may display a “Length of pulse #nnn equal to zero” error message.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See “Cell Options Dialog Box: Layout Tab ” for Cell Options dialog box **Layout** tab details.

See “The Layout Process File (LPF)” for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

Recommendations for Use

The model should be used for microstrip interdigital capacitors that connect to feeding microstrip line without width step. If this is not the case and the emphasized metallic strip that connects end of fingers at input ports is present at both sides (or at one side) it is recommended that model MICAP (or MICAP2) is used instead MICAP1. MICAP is geared toward interdigital capacitor that connects to microstrip lines having widths that are small comparing to capacitor width (less than 50% of capacitor width); MICAP2 has width step only at one port.

The model accounts for losses in metal and in substrate dielectric. Dispersion is partly included.

To decrease the calculation time for schematics that contain several microstrip interdigital capacitors cache is implemented for this model. It means that during the first evaluation of schematic the most time consuming intermediate parameters for each capacitor instance are being stored in memory cache. Each interdigital capacitor model checks this cache looking for its duplicate. Duplicate capacitors copy the appropriate parameters from memory cache saving substantially on their recalculation.

Layout recommendation: The MICAP1 layout cell does not draw any lines between the tips of fingers because it implies that the fingers protrude directly from the ends of lines attached to MICAP ports. You can evaluate the widths of these lines either using a simple formula (see "Parameter Details") or using the values of the MICAP1 output parameters W1 and W2.

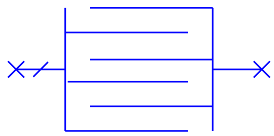
NOTE: The implementation of EM Quasi-Static models relies heavily on the involved numerical algorithms. This may lead to a noticeable increase in simulation time for schematics that employ many such models.

References

[1] B. Bazdar, A.R. Djordjevic, R.F. Harrington, and T.K. Sarkar, "Evaluation of quasi-static matrix parameters for multiconductor transmission lines using Galerkin's method," IEEE Trans. Microwave Theory Tech., vol. MTT-42, July 1994, pp. 1223-1228.

Microstrip Interdigital Capacitor, (Step at Port 1, No Step at Port 2) (EM Quasi-Static): MICAP2

Symbol

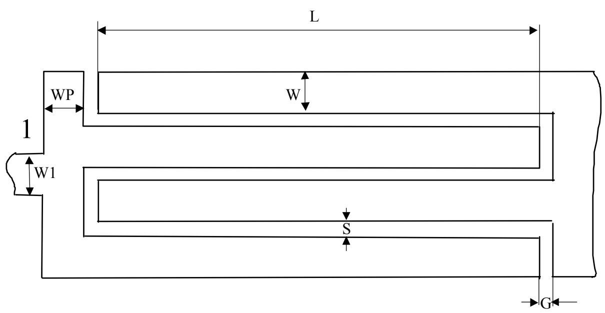


Summary

MICAP2 models a microstrip interdigital capacitor that connects to microstrip lines with a width step at port 1 and without a width step at port 2. The capacitor should have an emphasized metallic strip connecting the ends of fingers at port 1. MICAP2 takes advantage of the EM Quasi-Static coupled lines approach to considering interaction between all fingers.

MICAP2$ is the corresponding intelligent cell (iCell). An iCell model is identical to its non-iCell equivalent with the following exception: Certain dimension-related parameters are not explicitly user-specified; rather, they are automatically and dynamically determined by the dimensions of the attached elements. See "Intelligent Cells (iCells)" for a detailed discussion of how to use iCells, their advantages, and their limitations.

Topology



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	MI1
W	Finger width	Length	W[a]
S	Spacing between adjacent fingers	Length	S[a]
G	End gap width	Length	S[a]
L	Length of the overlap region of the fingers	Length	L[a]
N	Number of fingers		4
WP	Width of the fingers transverse interconnect	Length	W[a]
W1	Conductor Width @ Node 1	Length	W[a]
W2	Width of the feeding line at port 2 (Read-Only parameter)	Length	W[a]

Name	Description	Unit Type	Default
MSUB	Substrate definition	Text	MSUB1[b]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See Microwave Office Layout Guide for details.

[b]Modify only if the schematic contains multiple substrates. See the "Using Elements With Model Blocks" for details.

Parameter Details

W1. This parameter is secondary for the MICAP2$ iCell model.

W2. A read-only (output) parameter that represents the capacitor total width at ports 2, and is evaluated in the model as N(W + S)-S.

Parameter Restrictions and Recommendations

1. The number of fingers N must be 4 ≤N≤ 16.
2. Conductor thickness is set via substrate parameters. MICAP2 does not impose restrictions on thickness except for the requirement to be non-negative.

Implementation Details

The EM Quasi-Static technique allows you to model a microstrip interdigital capacitor with a wide range of conductor thicknesses.

This model accounts for the effect of phase shift along the microstrip line that connects fingers. It also includes the effect of the presence of width step at one port.

To apply Method of Moments for analysis, a quasi-static model creates 1D mesh covering contours of all conductors. The mesh is made of linear segments (pulses) of varying length. The length of a pulse is relatively big at the conductor center; it decreases toward the conductor edges to reveal the charge distribution across the conductor. If the conductor width is prohibitively large it may cause the pulse size to approach zero for pulses close to the edge. In these rare cases the model may display a "Length of pulse #nnn equal to zero" error message.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See "Cell Options Dialog Box: Layout Tab " for Cell Options dialog box **Layout** tab details.

See "The Layout Process File (LPF)" for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

Recommendations for Use

This model should be used for microstrip interdigital capacitors that have a prominent metallic strip of width WP connecting fingers at port 1. Capacitor and microstrip line have the same width at port 2. This model is convenient for modeling capacitors grounded at port 2. If W1exceeds 50% of the capacitor width you should use the MICAP1 model. MICAP1 is geared toward interdigital capacitors that are incorporated into microstrip line having a width equal to that of the capacitor. You should also use MICAP1 for all capacitors with N=2 and N=3.

This model accounts for losses in metal and in substrate dielectric. Dispersion is partly included.

To decrease the calculation time for schematics that contain several microstrip interdigital capacitors, cache is implemented for this model. This means that during the first evaluation of a schematic, the most time-consuming intermediate parameters for each capacitor instance are stored in a memory cache. Each interdigital capacitor model checks this cache looking for its duplicate. Duplicate capacitors copy the appropriate parameters from the memory cache, saving substantially on their recalculation.

Layout recommendation: The MICAP2 layout cell does not draw any lines between the tips of fingers at port 2 because it implies that the fingers protrude directly from the ends of the attached line. You can evaluate the width of this line either using a simple formula (see the "Parameter Details" section) or using the value of output parameter W2.

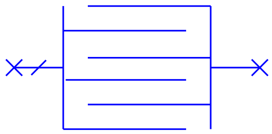
NOTE: The implementation of EM Quasi-Static models relies heavily on the involved numerical algorithms. This may lead to a noticeable increase in simulation time for schematics that employ many such models.

References

[1] B. Bazdar, A.R. Djordjevic, R.F. Harrington, and T.K. Sarkar, "Evaluation of quasi-static matrix parameters for multiconductor transmission lines using Galerkin's method," IEEE Trans. Microwave Theory Tech., vol. MTT-42, July 1994, pp. 1223-1228.

Microstrip Interdigital Capacitor: MICAP3

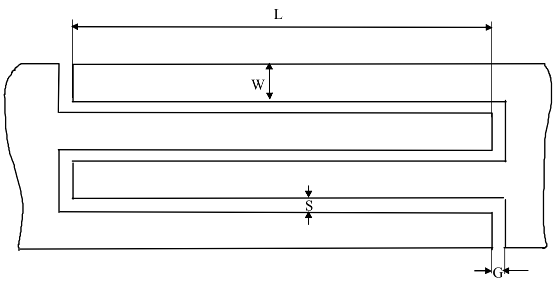
Symbol



Summary

MICAP3 models a microstrip interdigital capacitor that connects to microstrip lines without width steps at ports and uses a closed-form approximations.

Topology



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	MI1
W	Finger width	Length	W[a]
S	Spacing between adjacent fingers	Length	S[a]
L	Length of the overlap region of the fingers	Length	L[a]
N	Number of fingers		4
W1	Width of the feeding line at port 1 (Read-Only parameter)	Length	W[a]
W2	Width of the feeding line at port 2 (Read-Only parameter)	Length	W[a]
MSUB	Substrate definition	Text	MSUB1[b]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See the Microwave Office Layout Guide for details.

[b]Modify only if schematic contains multiple substrates. See “Using Elements With Model Blocks” for details.

Parameter Details

W1, W2. Parameters W1, W2 are read-only (output) parameters, represent the capacitor total width at ports 1 and 2 and are evaluated in the model as N(W + S)-S.

Parameter Restrictions and Recommendations

1. The number of fingers N must be N≥ 2. Model and layout cell do not restrict the upper limit of N, but user discretion is advised. Generally, target capacitance is not expected to exceed 1 pF.
2. Model does not account for gap G between finger tips and traverse lines (see Topology) but implies that G has the same order of magnitude as spacing between adjacent fingers S. Layout cell of MICAP3 has additional parameter G which allows user to modify G on layout only.

Implementation Details

Model is represented by a Pi-circuit with symmetrical shunt capacitors and series RC circuit.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See “Cell Options Dialog Box: Layout Tab ” for Cell Options dialog box **Layout** tab details.

See “The Layout Process File (LPF)” for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

Recommendations for Use

The model should be used for microstrip interdigital capacitors that connect to feeding microstrip line without width step. If this is not the case and the emphasized metallic strip that connects fingers at input ports is present at both sides (or at one side) it is recommended that model MICAP (or MICAP2) is used instead MICAP3. MICAP is geared toward interdigital capacitor that connects to microstrip lines having widths that are small comparing to capacitor width (less than 50% of capacitor width); MICAP2 has width step only at one port.

This model sacrifices high accuracy for high speed of computations and is convenient for automatic and manual optimization. It also does not set explicitly the upper limit of fingers number (however, large number of fingers may cause the noticeable phase shift of field distribution along capacitor width and invalidate some assumptions made for this model). For more reliable results usage of time consuming but more accurate model MICAP1 is recommended.

The model accounts for losses in metal and in substrate dielectric. Dispersion is partly included.

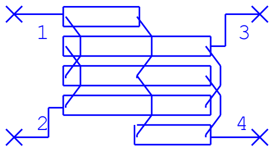
Layout recommendation: The layout cell of MICAP3 does not draw any lines between the tips of fingers because it implies that the fingers protrude directly from the ends of lines attached to MICAP3 ports. You can evaluate the widths of these lines either using a simple formula (see "Parameter Details") or using the values of MICAP3 output parameters W1 and W2.

References

[1] Mongia, R., Bahl, I. and Bhartia P., RF and Microwave Coupled-Line Circuits, Artech House Inc., Norwood, MA, 1999, pp.370-372

Microstrip Lange Coupler (Closed Form): MLANG

Symbol



Summary

MLANG models the multiline interdigital Lange coupler based on N edge-coupled microstrip lines (closed form technique). MLANG features an improved layout cell and can be used in conjunction with LMAN (Lange Coupler Manifold).

Parameters

Name	Description	Unit Type	Default
ID	Name	Text	TL1
N	Number of fingers		4
W	Finger width	Length	W[a]
S	Width of gap between fingers	Length	W[a]
L	Finger length	Length	L[a]
MSUB	Substrate definition	Text	MSUB[b]
Gap	Gap between open end of shortened finger and manifold **(layout parameter only)**	Length	0

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory.

[b]If only one MSUB is present in the schematic, this substrate is automatically used. If multiple MSUB substrate definitions are present, you must specify which to use.

Parameter Details

N. Number of fingers

W. Width of the fingers (the widths of all the fingers must be the same).

S. Width of gaps between the fingers (must be identical).

Parameter Restrictions and Recommendations

1. This model is most accurate in the following ranges (where H is the substrate thickness and ε is the substrate dielectric constant):

   $0.1<W/H<10$

   $0.1<S/H<10$

   $1<\varepsilon_r<18$

   Outside these ranges, the accuracy degrades gracefully, and the results may still be useful, so no error message is displayed if these limits are exceeded. Do not use this model for two-strip couplers; use MCLIN or M2CLIN instead.

2. The Gap parameter defines the distance between the open end of a shortened finger and the edge of the manifold (see the "Topology" section of MLANGE2). This parameter has no effect on the electrical characteristics of MLANG. You can set the Gap value in the Layout View only. To set the Gap, right-click the MLANG shape, choose **Shape Properties** to display the Cell Options dialog box, then click the **Parameters** tab. Set the Gap to the desired value.

Implementation Details

MLANG uses closed form representation from [2] and formulations for odd/even impedances of coupled microstrip lines from [1].

This is a lossless element, dispersion is included.

Note that the MLANG symbol has terminal numbering that differs from those of the obsolete element MLANGE, namely terminal numbers 3 and 4 are swapped. This numbering matches terminal numbering of the MLANGE2 element. If terminal 1 is input, then 2 is coupled, 3 is isolated, and 4 is direct.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See “Cell Options Dialog Box: Layout Tab ” for Cell Options dialog box **Layout** tab details.

See “The Layout Process File (LPF)” for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

Recommendations for Use

MLANG can be used in conjunction with the LMAN element, which provides a layout intended to match the MLANG layout.

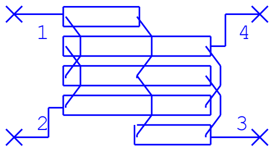
References

[1] M. Kirschning, R. Jansen, "Accurate Wide-Range Design Equations for the Frequency-Dependent Characteristics of Parallel Coupled Microstrip Lines", IEEE Trans. Microwave Theory Tech., vol. MTT-32, 1984, p. 83-90.

[2] D. Kajfez, Z. Paunovic, S. Pavlin, "Simplified Design of Lange Coupler", IEEE Trans. Microwave Theory Tech., vol. MTT-26, 1978, pp. 806-808.

(Obsolete) Microstrip Lange Coupler (Closed Form): MLANGE

Symbol



Summary

This element is OBSOLETE and is replaced by the Microstrip Lange Coupler (Closed Form) (MLANG) element.

Parameters

Name	Description	Unit Type	Default
ID	Name	Text	TL1
N	Number of strips		4
W	Conductor widths	Length	W[a]
S	Gap widths	Length	W[a]
L	Conductor Lengths	Length	L[a]
MSUB	Substrate definition	Text	MSUB[b]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory.

[b]If only one MSUB is present in the schematic, this substrate is automatically used. If multiple MSUB substrate definitions are present, you must specify which to use.

Implementation Details

This component models the multiline interdigital Lange coupler based on N edge-coupled microstrip lines.

N is the number of strips, W is the width of the strips (the widths of all the strips must be the same), S is the width of the gaps between the strips (which also must be identical), and L is the physical length of lines. W, S, and L are entered in the default length units.

MSUB indicates the name of the MSUB element to use for substrate characteristics. If blank, a default is used.

This model is most accurate in the following ranges (where H is the substrate thickness):

$0.1<W/H<10$

$0.1<S/H<10$

$1<\varepsilon_r<18$

Outside these ranges, the accuracy degrades gracefully, and the results may still be useful. Therefore, no error message is given if these limits are exceeded. Do not use this model for two-strip couplers; use MCLIN or M2CLIN instead.

This is a lossless element, dispersion is included.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See "Cell Options Dialog Box: Layout Tab " for Cell Options dialog box **Layout** tab details.

See "The Layout Process File (LPF)" for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

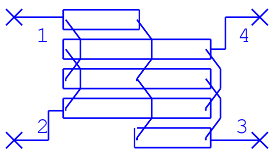
References

[1] M.Kirschning, R.Jansen, "Accurate Wide-Range Design Equations for the Frequency-Dependent Characteristics of Parallel Coupled Microstrip Lines,", IEEE Trans. Microwave Theory Tech., vol. MTT-32, 1984, p. 83-90.

[2] D. Kajfez, Z.Paunovic, S.Pavlin, "Simplified Design of Lange Coupler,", IEEE Trans. Microwave Theory Tech., vol. MTT-26, 1978, pp. 806-808.

(Obsolete) Microstrip Lange Coupler (EM Quasi-Static): MLANGE1

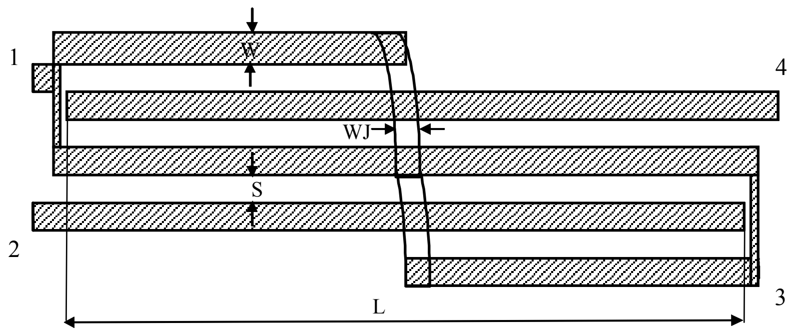
Symbol



Summary

This element is OBSOLETE and is replaced by the Microstrip Lange Coupler (EM Quasi-Static) (MLANGE2) element. MLANGE1 models the multiline interdigital Lange coupler based on multiple edge-coupled microstrip lines. This model accurately accounts for coupling of lines with arbitrary thickness.

Topology



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	TL
N	Number of lines		4
W	Line width	Length	W[a]
S	Spacing between lines	Length	W[a]
L	Length of coupled lines	Length	L[a]
WJ	Width of jumper	Length	W[a]
Acc	Accuracy parameter		1
MSUB	Substrate definition	Text	MSUB1[b]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See the Microwave Office Layout Guide for details.

[b]Modify only if schematic contains multiple substrates. See "Using Elements With Model Blocks" for details.

Parameter Details

Acc. If Acc is less than 1 or greater than 10 it is set automatically to 2.

WJ. The width of a jumper (flat bond ribbon, bond wire) connecting fingers. For bond wire having diameter D, set WJ=2D.

Parameter Restrictions and Recommendations

1. Number of finger N should be even and within the range 4≤N≤16. .
2. The accuracy parameter Acc is limited to 1≤Acc≤10. A larger value increases the density of mesh used in computations. The accuracy of model parameters may gain slightly from increasing Acc at the expense of the noticeable growth of computation time. In general, a good trade-off between accuracy and computation time is to set Acc to 1
3. This component does not impose restrictions on the conductor thickness.

Implementation Details

Model implementation is based on EM Quasi-Static technique described in [1]. It accounts for losses in metal and in substrate dielectrics. Dispersion is partly included.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See "Cell Options Dialog Box: Layout Tab " for Cell Options dialog box **Layout** tab details.

See "The Layout Process File (LPF)" for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

Recommendations for Use

To decrease the calculation time for schematics that contain several MLANGE1 elements, cache is implemented for this model. During the first evaluation of a schematic, the most time-consuming intermediate parameters for each MLANGE1 instance are stored in memory cache. Each MLANGE1 model checks this cache to look for its duplicate. Duplicate models copy the appropriate parameters from the memory cache, saving substantially on their recalculation.

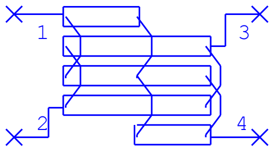
NOTE: The implementation of EM Quasi-Static models relies heavily on the involved numerical algorithms. This may lead to a noticeable increase in simulation time for schematics that employ many such models.

References

[1] M.B. Bazdar, A.R. Djordjevic, R.F. Harrington, and T.K. Sarkar, "Evaluation of quasi-static matrix parameters for multiconductor transmission lines using Galerkin's method," IEEE Trans. Microwave Theory Tech., vol. MTT-42, July 1994, pp. 1223-1228.

Microstrip Lange Coupler (EM Quasi-Static): MLANGE2

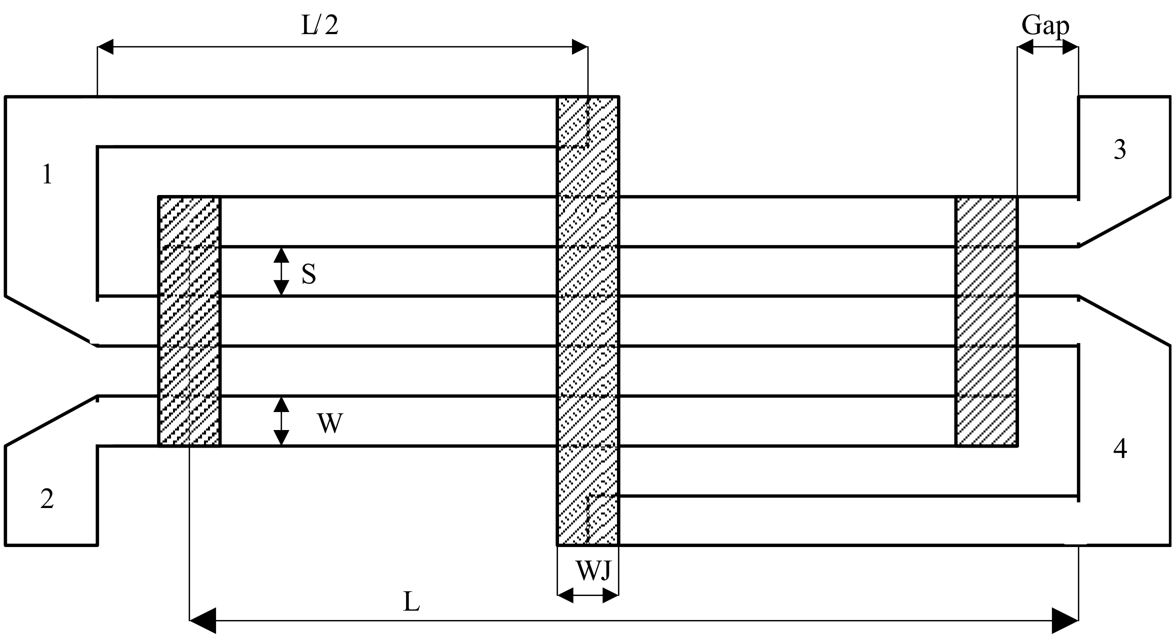
Symbol



Summary

MLANGE2 models the multiline interdigital Lange coupler based on multiple edge-coupled microstrip lines. This model accurately accounts for coupling of lines with arbitrary thickness. MLANGE2 features an improved layout cell and you can use it in conjunction with the LMAN (Lange Coupler Manifold) element.

Topology



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	TL
N	Number of lines		4
W	Line width	Length	W[a]
S	Spacing between lines	Length	W[a]
L	Length of coupled lines	Length	L[a]
WJ	Width of jumper	Length	W[a]

Name	Description	Unit Type	Default
TJ	Thickness of jumper	Length	W[a]
RhoJ	Bulk resistivity of jumper metal normalized to gold	Scalar	1
Acc	Accuracy parameter		1
MSUB	Substrate definition	Text	MSUB1[b]
Gap	Gap between open end of shortened finger and manifold **(layout parameter only)**	Length	0

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See the Microwave Office Layout Guide for details.

[b]Modify only if a schematic contains multiple substrates. See “Using Elements With Model Blocks” for details.

Parameter Details

Acc. If Acc is less than 1 or greater than 10 it is set automatically to 2.

WJ, TJ. The width and thickness of a jumper (MLANGE2 uses the model of a flat metal ribbon) connecting fingers. To use for the jumpers bond wire of diameter D, set WJ and TJ to provide equality of cross-sectional perimeters, that is, $\pi D = 2(WJ+TJ)$.

Parameter Restrictions and Recommendations

1. The number of finger N should be even and within the range $4 \le N \le 16$.
2. The accuracy parameter Acc is limited to $1 \le Acc \le 10$. A larger value increases the density of mesh used in computations. The accuracy of model parameters may gain slightly from increasing Acc at the expense of the noticeable growth of computation time. In general, a good trade-off between accuracy and computation time is to set Acc to 1
3. This component does not impose restrictions on conductor thickness.
4. The Gap parameter defines the distance between the open end of a shortened finger and the edge of the manifold (see the "Topology" section). This parameter has no effect on the electrical characteristics of MLANGE2. You can set the Gap value in the Layout View only. To set the Gap, right-click the MLANG2 shape, choose **Shape Properties** to display the Cell Options dialog box, then click the **Parameters** tab. Set the Gap to the desired value.
5. The RIBBON element used to model junctions may issue warnings regarding the values of parameters WJ, TJ, and/or RhoJ. RIBBON expects that $0.01 \le WJ/TJ \le 100$, $RhoJ \le 100$ and displays a warning if values are out of range.

Implementation Details

Model implementation is based on the EM Quasi-Static technique described in [1], which accounts for losses in metal and in substrate dielectrics. Dispersion is partly included.

Note that the MLANGE2 symbol has terminal numbering that differs from those of the obsolete element MLANGE1, namely terminal numbers 3 and 4 are swapped. If terminal 1 is input, then 2 is coupled, 3 is isolated, and 4 is direct.

To apply Method of Moments for analysis, a quasi-static model creates 1D mesh covering contours of all conductors. The mesh is made of linear segments (pulses) of varying length. The length of a pulse is relatively big at the conductor center; it decreases toward the conductor edges to reveal the charge distribution across the conductor. If the conductor width is prohibitively large it may cause the pulse size to approach zero for pulses close to the edge. In these rare cases the model may display a “Length of pulse #nnn equal to zero” error message.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See "Cell Options Dialog Box: Layout Tab " for Cell Options dialog box **Layout** tab details.

See "The Layout Process File (LPF)" for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

Recommendations for Use

NOTE: The implementation of EM Quasi-Static models relies on the involved numerical algorithms. This may lead to a noticeable increase in simulation time for schematics that employ many such models.

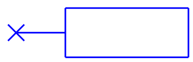
You can use MLANGE2 in conjunction with the LMAN element. LMAN provides a layout intended to match the layout of MLANGE2.

References

[1] M. B. Bazdar, A. R. Djordjevic, R. F. Harrington, and T. K. Sarkar, "Evaluation of quasi-static matrix parameters for multi-conductor transmission lines using Galerkin's method", IEEE Trans. Microwave Theory Tech., vol. MTT-42, July 1994, pp. 1223-1228.

Open Microstrip Line With End Effect (Closed Form): MLEF

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	TL1
W	Conductor width	Length	W[a]
L	Conductor length	Length	L[a]
MSUB	Substrate definition	Text	MSUB#[b]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See the Microwave Office Layout Guide for details.

[b]Modify only if schematic contains multiple substrates. See "Using Elements With Model Blocks" for details.

Implementation Details

Parameters are the same as for MLIN.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See "Cell Options Dialog Box: Layout Tab " for Cell Options dialog box **Layout** tab details.

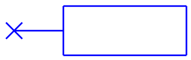
See "The Layout Process File (LPF)" for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

Reference

[1] M. Kirshning, R. H. Jansen, and N. H. L. Koster, Elec. Ltrs., Vol. 17, p. 123, 1981.

Open Microstrip Line with End Effect (EM Base): MLEFX

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	MO1
W	Conductor width	Length	W[a]
L	Conductor length	Length	L[a]
MSUB	Substrate definition	Text	MSUB#[b]
*AutoFill	AutoFill database if not equal to 0		0

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See the Microwave Office Layout Guide for details.

[b]Modify only if schematic contains multiple substrates. See "Using Elements With Model Blocks" for details.

** indicates a secondary parameter*

Implementation Details

This model implements an X-model of the microstrip open ended line. For a more detailed discussion of the X-models see " EM-based Models (X-models) " . This model does not include the effects of dielectric, conductor or radiation losses.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See "Cell Options Dialog Box: Layout Tab " for Cell Options dialog box **Layout** tab details.

See "The Layout Process File (LPF)" for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

Restrictions

$0.05 \le W/H \le 8$ Recommended	$\varepsilon_{rnominal} \le 16$ Recommended
$0 \le \text{Frequency} \le F_{max}$[a] Recommended	$1 \le \varepsilon_{rnominal}$ Required
$(100*\lvert\varepsilon_r - \varepsilon_{rnominal}\rvert) \le 20\%$ Required	$(100*\lvert\varepsilon_r - \varepsilon_{rnominal}\rvert)/\varepsilon_{rnominal} \le 10\%$ Recommended

[a]The frequency limits of this model are dynamic with respect to the dimensions of the discontinuity. This dynamic frequency limit is displayed to the user via warning messages for the relative size, and dielectric in use. Importantly, this recommended frequency limit will change as a function of the

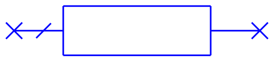
largest width in the discontinuity for a given substrate definition. The frequency limit warns the user that at least one of the transmission lines constructing the discontinuity is approaching the frequency limit where multiple modes can propagate. See the section marked "Upper Frequency Limitations" in the General Discussion of the X-models.

References

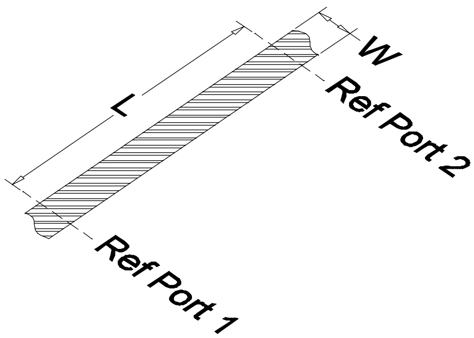
This model was developed under research performed at NI AWR Corporation. The details of the implementation are considered proprietary in nature.

Microstrip Line (Closed Form): MLIN

Symbol



Topology



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	TL1
W	Conductor width	Length	W[a]
L	Conductor length	Length	L[a]
MSUB	Substrate definition	Text	MSUB#[b]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See the Microwave Office Layout Guide for details.

[b]Modify only if schematic contains multiple substrates. See "Using Elements With Model Blocks" for details.

Parameter Details

MSUB. This parameter specifies the microstrip substrate element, which defines additional cross sectional parameters of the transmission line. You can also add this substrate to the Global Definitions window and then reference the material from any schematic.

Implementation Details

This circuit component models a length of Microstrip Transmission Line. The model assumes a Quasi-TEM mode of propagation and incorporates the effects of dielectric and conductive losses. The parameters W (Strip Width) and L (Strip Length) are lengths entered in the default length units.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.

3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See “Cell Options Dialog Box: Layout Tab ” for Cell Options dialog box **Layout** tab details.

See “The Layout Process File (LPF)” for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

Restrictions

$0.02 \leq W/H \leq 20$ recommended

$T/W \leq 0.7$ recommended

$T/H \leq 0.5$ recommended

$\varepsilon_r \leq 16$ recommended

$1 \leq \varepsilon_r$ required

Tand ≥ 0 required

$0 \leq$ Rho ≤ 1000 required

$0 \leq$ Rho ≤ 100 recommended

References

[1] E. Hammerstad and O. Jensen, IEEE MTT-S International Microwave Symposium Digest, p. 407, 1980.

[2] I. J. Bahl and D. K. Trivedi, "A Designer's Guide to Microstrip Line," Microwaves, p. 174, May, 1977.

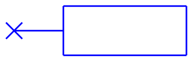
[3] S. March, "Microstrip Packaging: Watch the Last Step," Microwaves, p. 83, Dec., 1981.

[4] R. Pucel, D. J. Masse, and C. P. Hartwing, IEEE Trans. Microwave Theory Tech., Vol. MTT-16, p. 342, 1968.

[5] G. Wells and P. Pramanick, Int. J. Microwave and mmWave Computer-Aided Design, Vol. 5, p. 287, 1995.

(Obsolete) Open Microstrip Line Without End Effect (Closed Form): MLOC

Symbol



Summary

This element is OBSOLETE and is replaced by the Open Microstrip Line With End Effect (Closed Form) (MLEF) or Open Microstrip Line with End Effect (EM Base) (MLEFX) element.

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	TL1
W	Conductor width	Length	W[a]
L	Conductor length	Length	L[a]
MSUB	Substrate definition	Text	MSUB#[b]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See the Microwave Office Layout Guide for details.
[b]Modify only if schematic contains multiple substrates. See the "Using Elements With Model Blocks" for details.

Implementation Details

Parameters are the same as for MLIN.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See "Cell Options Dialog Box: Layout Tab " for Cell Options dialog box **Layout** tab details.

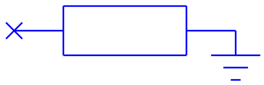
See "The Layout Process File (LPF)" for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

Reference

[1] M. Kirshning, R. H. Jansen, and N. H. L. Koster, Elec. Ltrs., Vol. 17, p. 123, 1981.

Shorted Microstrip Line without End Effect (Closed Form): MLSC

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	TL1
W	Conductor width	Length	W[a]
L	Conductor length	Length	L[a]
MSUB	Substrate definition	Text	MSUB#[b]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See the Microwave Office Layout Guide for details.

[b]Modify only if schematic contains multiple substrates. See "Using Elements With Model Blocks" for details.

Implementation Details

Parameters are the same as for MLIN.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

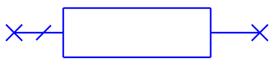
1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See "Cell Options Dialog Box: Layout Tab " for Cell Options dialog box **Layout** tab details.

See "The Layout Process File (LPF)" for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

2-Layer (Protective Layer/Substrate) Microstrip Line (EM Quasi-Static): MM1LIN

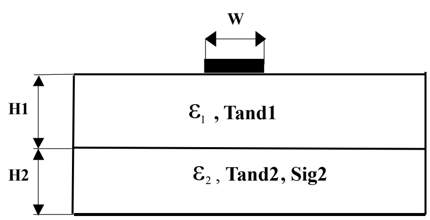
Symbol



Summary

MM1LIN models a section of single microstrip line having a conductor strip placed on the upper surface of a two-layered substrate. Lower layer of the substrate may have conductive loss.

Topology



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	TL1
W	Conductor width	Length	W[a]
L	Conductor length	Length	L[a]
Acc	Accuracy parameter		1
MSUB2	Substrate definition	Text	MSUB1[b]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See Microwave Office Layout Guide for details.

[b]Modify only if schematic contains multiple substrates. See the "Using Elements With Model Blocks" for details.

Parameter Details

Acc. The parameter Acc is the accuracy parameter. The default value for Acc is 1. If Acc is less than 1 or greater than 10 it is set automatically to 2.

Parameter Restrictions and Recommendations

1. Accuracy parameter Acc is limited to $1 \le Acc \le 10$. Larger value of Acc increases density of mesh used in computations. Accuracy of model parameters may gain slightly from increasing Acc at the expense of noticeable growth of computation time. As a rule of thumb, good trade-off between accuracy and computation time is to set Acc to 1.
2. This component does not impose restrictions on the conductor thickness (thickness may be zero, positive, or negative). Negative thickness means that the conductor is recessed into the substrate.
3. Substrate parameters Tand1, Tand2 (loss tangents for upper and lower layers) should not exceed 3.0; substrate parameter Sig2 (bulk conductance of the lower layer) shouldn't exceed 3 S/m.

Implementation Details

Model implementation is based on EM Quasi-Static technique described in [1] [10–192]. It accounts for losses in metal and in substrate dielectrics. Polarization loss (Tand1,Tand2) are allowed in both substrate layers. Conductive loss is allowed in lower layer only. Dispersion is partly included.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See “Cell Options Dialog Box: Layout Tab ” for Cell Options dialog box **Layout** tab details.

See “The Layout Process File (LPF)” for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

Recommendations for Use

This element may be used to model microstrip line over semiconductor substrate insulated with oxide or another semiconductor. For example, MM1LIN may model microstrip line on SiO2/Si substrate or on GaAs/Si substrate. However, model impose limitations on allowed level of polarization and conductive losses, that is, for instance, only high resistance silicon (HRS) substrates may be modeled.

NOTE: The implementation of EM Quasi-Static models relies heavily on the involved numerical algorithms. This may lead to a noticeable increase in simulation time for schematics that employ many such models.

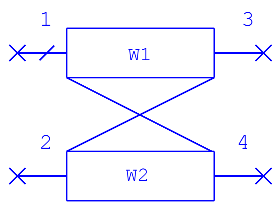
If thickness of any layer is too small in comparison with the thickness of another layer, simulation time may also noticeably grow.

References

[1] M.B. Bazdar, A.R. Djordjevic, R.F. Harrington, and T.K. Sarkar, "Evaluation of quasi-static matrix parameters for multiconductor transmission lines using Galerkin's method," IEEE Trans. Microwave Theory Tech., vol. MTT-42, July 1994, pp. 1223-1228.

2 Two-layer Edge (Protective Layer/Substrate) Coupled Microstrip Lines (EM Quasi-Static): MM2CLIN

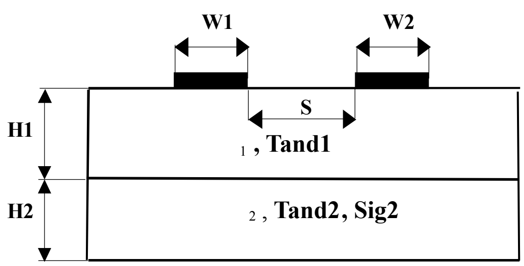
Symbol



Summary

MM2CLIN models section of two edge coupled microstrip lines placed on the upper surface of a two-layered substrate. Lower layer of the substrate may have conductive loss.

Topology



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	TL1
W1	Width of conductor 1	Length	W[a]
W2	Width of conductor 2	Length	W[a]
S	Spacing between conductors	Length	L[a]
L	Conductor length	Length	L[a]
Acc	Accuracy parameter		1
MSUB2	Substrate definition	Text	MSUB[b]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See the Microwave Office Layout Guide for details.

[b]Modify only if schematic contains multiple substrates. See "Using Elements With Model Blocks" for details

Parameter Details

Acc. The parameter Acc is the accuracy parameter. The default value for Acc is 1. If Acc is less than 1 or greater than 10 it is set automatically to 2.

Parameter Restrictions and Recommendations

1. Accuracy parameter A is limited to 1≤Acc≤10. Larger value of Acc increases density of mesh used in computations. Accuracy of model parameters may gain slightly from increasing Acc at the expense of noticeable growth of computation time. As a rule of thumb, good trade-off between accuracy and computation time is to set Acc to 1.
2. This component does not impose restrictions on the conductor thickness (thickness may be zero, positive, or negative). Negative thickness means that the conductor is recessed into the substrate.
3. Substrate parameters Tand1, Tand2 (loss tangents for upper and lower layers) should not exceed 3.0; substrate parameter Sig2 (bulk conductance of the lower layer) shouldn't exceed 3 S/m.

Implementation Details

Model implementation is based on EM Quasi-Static technique described in [1] [10–194]. It accounts for losses in metal and in substrate dielectrics. Polarization loss (Tand1,Tand2) are allowed in both substrate layers. Conductive loss is allowed in lower layer only. Dispersion is partly included.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See "Cell Options Dialog Box: Layout Tab " for Cell Options dialog box **Layout** tab details.

See "The Layout Process File (LPF)" for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

Recommendations for Use

This element may be used to model microstrip line over semiconductor substrate insulated with oxide or another semiconductor. For example, MM2LIN may model microstrip line on SiO2/Si substrate or on GaAs/Si substrate. However, this model imposes limitations on the allowed level of polarization and conductive losses, for instance, only high resistance silicon (HRS) substrates may be modeled.

NOTE: The implementation of EM Quasi-Static models relies heavily on the involved numerical algorithms. This may lead to a noticeable increase in simulation time for schematics that employ many such models.

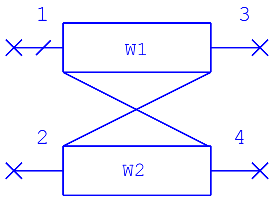
If thickness of any layer is too small in comparison with the thickness of another layer, simulation time may also noticeably grow.

References

M.B. Bazdar, A.R. Djordjevic, R.F. Harrington, and T.K. Sarkar, "Evaluation of quasi-static matrix parameters for multiconductor transmission lines using Galerkin's method," IEEE Trans. Microwave Theory Tech., vol. MTT-42, July 1994, pp. 1223-1228.

2 Two-Layer Broadside Coupled Microstrip Lines (EM Quasi-Static): MMBCPL

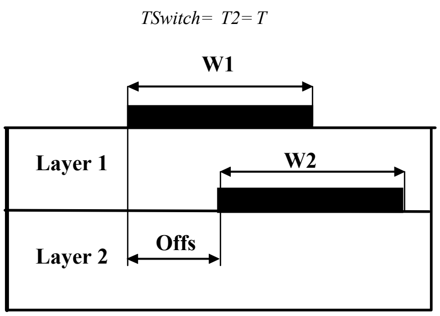
Symbol

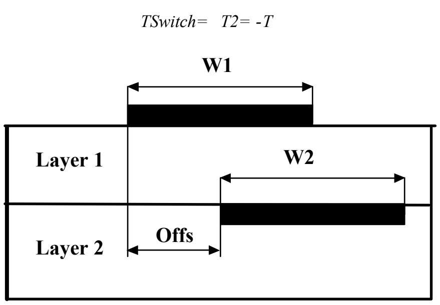


Summary

MMBCPL models a section of broadside coupled microstrip lines arranged at the two opposite surfaces of a top layer of two-layered substrate. The lower line may be displaced laterally relative to the left edge of the upper line.

Topology





Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	TL1
W1	Top conductor width	Length	W[a]
W2	Bottom conductor width	Length	W[a]
L	Conductor length	Length	L[a]
Offs	Horizontal offset	Length	L[a]
Acc	Accuracy parameter		1
TSwitch	Bottom Conductor Thickness T2= T or T2= -T		T2=T

Name	Description	Unit Type	Default
MSUB2	Substrate definition	Text	MSUB21[b]
SNAME1	Structure name from lpf file for top conductor	Text	TOP_BCLIN
SNAME2	Structure name from lpf file for bottom conductor	Text	BOT_BCLIN

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See the Microwave Office Layout Guide for details.

[b]Modify only if schematic contains multiple substrates. See "Using Elements With Model Blocks" for details.

Parameter Details

MSUB2. Two-layer substrate. Parameters are listed in the MSUB2 model description.

Acc. The accuracy parameter. The default value for Acc is 1. If Acc is less than 1 or greater than 10 it is set automatically to 2.

TSwitch. Defines how the bottom conductor protrudes: upward or downward. Direction is defined by a combination of the sign of parameter T and the value of TSwitch. TSwitch="T2=T" T>0 makes both conductors extend their thickness upward; TSwitch="T2=T" and T<0 makes them extend downward. Setting TSwitch="T2=-T" makes the bottom conductor extend the opposite direction relative to the top conductor (if T>0 it extends downward, if T<0 it extends upward.) The default is T>0 and TSwitch="T2=T". See the "Topology" section where two possible layouts are displayed.

Offs. Offs is a relative horizontal offset of the lower line left edge from the left edge of the upper line. Offs may be positive (the lower line is displaced to the right), zero (both lines are aligned) or negative (the lower line is displaced to the left).

Parameter Restrictions and Recommendations

1. Acc is limited to 1≤Acc≤10. A larger value of Acc increases the density of mesh used in computations. The accuracy of model parameters may gain slightly from increasing Acc at the expense of an increase in computation time. Generally, a good trade-off between accuracy and computation time is to set Acc to 1.
2. MMBCPL does not impose restrictions on conductor thickness (the thickness may be zero, positive, or negative). If thickness parameter T (see the MPSUB parameters) is positive, MMBCPL implies that both strips are leveled up over the corresponding layer boundaries. If T is negative, MMBCPL reassesses the upper strip into the lower layer and extends the lower strip conductor body downward.
3. Substrate parameters Tand1 and Tand2 (loss tangents for upper and lower layers) should not exceed 3.0; the substrate parameter Sig2 (bulk conductance of the lower layer) should not exceed 3 S/m.
4. SNAME1 and SNAME2 are for layout only and have no effect on the electrical performance.

Implementation Details

Model implementation is based on the EM Quasi-Static technique described in [1] [10–197]. It accounts for losses in metal and in substrate dielectric. Dispersion is partly included

Layout

When you specify SNAME1 and SNAME2, the structures with corresponding names are identified in a $STRUCT_TYPE_BEGIN section of the LPF file. If a structure with the corresponding name is not found, the name of the missing structure is drawn on the error layer. The structure named SNAME1 must contain a text line starting with the line type name used for the top conductor. A structure named SNAME2 must contain a text line starting with the line type name used for the bottom conductor. In a structure named SNAME1=TOP_TRACE and SNAME2= BOT_TRACE with line type names "Top Copper" and "Bot Copper", you need to add the following code to the LPF file (user defined line types may contain any number of layers; structures must contain only one text line):

```
$LINE_TYPE_BEGIN "Top Copper"
  ! -> Layer   offset   minWidth   flags
  "Cu_1"  0  5e-005  0
$LINE_TYPE_END
$LINE_TYPE_BEGIN "Bot Copper"
  ! -> Layer   offset   minWidth   flags
  "Cu_2"  0  5e-005  0
$LINE_TYPE_END
$STRUCT_TYPE_BEGIN "TOP_TRACE"
   "Top Copper" 0 0 0
$STRUCT_TYPE_END
$STRUCT_TYPE_BEGIN "BOT_TRACE"
   "Bot Copper" 0 0 0
$STRUCT_TYPE_END
```

Note that text inside structures contains the line type names in quotation marks followed by three blank separated zeros.

Recommendations for Use

MMBCPL may be used to model microstrip line over semiconductor substrate insulated with oxide or another semiconductor. For example, MMBCPL may model microstrip line on SiO2/Si substrate or on GaAs/Si substrate. However, this model imposes limitations on the allowed level of polarization and conductive losses (for instance, only high resistance silicon (HRS) substrates may be modeled).

NOTE: The implementation of EM Quasi-Static models relies heavily on the involved numerical algorithms. This may lead to a noticeable increase in simulation time for schematics that employ many such models.

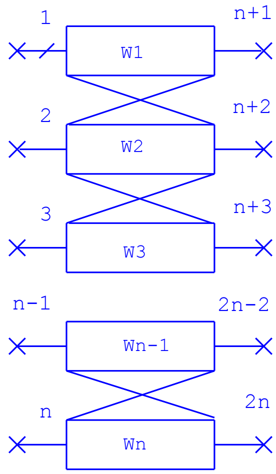
If the thickness of any layer is too small in comparison to the thickness of another layer, simulation time may also noticeably increase.

References

[1] M.B. Bazdar, A.R. Djordjevic, R.F. Harrington, and T.K. Sarkar, "Evaluation of quasi-static matrix parameters for multiconductor transmission lines using Galerkin's method," IEEE Trans. Microwave Theory Tech., vol. MTT-42, July 1994, pp. 1223-1228.

Multiple Edge Coupled Microstrip Lines (EM Quasi-Static): MNCLIN

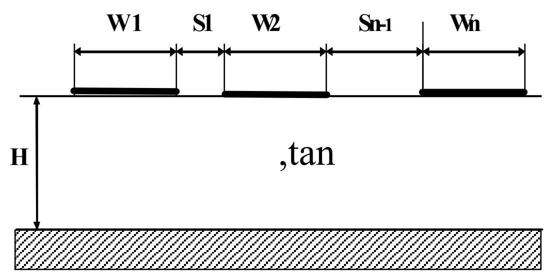
Symbol



Summary

MNCLIN models a section of several (1 to 50) edge coupled/single microstrip lines arranged on a single-layer substrate. A backing ground plane is always present. MNCLIN implements the same modeling techniques as the M2CLIN,...M16CLIN models. In addition, MNCLIN is a dynamic or scalable model; it accepts a number of lines as input parameters so the model and its schematic symbol expands/shrinks as the number of lines increases/decreases. This model uses disk cache to speed up simulation.

Topology



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	TL1
N	Number of conductors		2
L	Conductor length	Length	L[a]
Acc	Accuracy parameter		1
MSUB	Substrate Definition	Text	MSUB1[b]
*SaveToFile	"Save to txt file"=Yes/No		"No"

Name	Description	Unit Type	Default
*FileName	Name of text file with computed model parameters	String	Same as model name
Wi, i=1..nn-number of lines	Width of conductor No i	Length	W[a]
Si, i=1..n-1n-number of lines	Spacing between conductors No i and No i+1	Length	W[a]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See the Microwave Office Layout Guide for details.

[b]Modify only if the schematic contains multiple substrates. See "Using Elements With Model Blocks" for details.

** indicates a secondary parameter*

Parameter Details

See MxCLIN for a detailed description of MNCLIN parameters. MNCLIN differs from MXCLIN in parameter order only: For MXCLIN models, the L, Acc, and MSUB parameters follow the widths (Wi) and spacings(Si); for the MNCLIN model, the L, Acc, MSUB, Save to File, and FileName parameters precede the Wi and Si parameters.

SaveToFile. This parameter is hidden by default and set to "No". You can toggle the parameter to "Yes" or "No". If set to "Yes", the model creates a text file (named *MnCLIN.txt*) at the current project location. This text file contains a table of values of per-unit-length RLGC line parameters at each project frequency. Each row contains RLGC values computed at the frequency specified in the first column (frequency in GHz, R in ohms/m, L in H/m, G in S/m, C in F/m).

The structure of this text file is essentially the same for n>2. All models output RLGC matrices to columns that immediately follow the frequency column. Entries of each matrix are placed column-wise; the first column is first: R11, R21, R31.. Rn1; and then the second column: R12, R22, R32,..Rn2 etc. The total number of columns in the file is 4*n*n+1, where n is the number of lines.

If N=2, MNCLIN implies the existence of two modes, namely, C and P (see [2] [10–201]) and places additional columns in the *text* file. Note that if a system of coupled lines is fully symmetrical (as might be the case with edge-coupled microstrip lines) C-mode corresponds to even mode and P-mode corresponds to odd mode.

For N=2, MNCLIN outputs complex characteristic impedances Re(Zc1), Im(Zc1), Re(Zp1), Im(Zp1), Re(Zc2), Im(Zc2), Re(Zp2), and Im(Zp2); complex effective dielectric constants Re(EeffC), Im(EeffC), Re(EeffP), and Im(EeffP) (see traditional effective dielectric constants in columns 38, 39); and losses for C and P modes LossC (dB/m), LossP(dB/m) to columns 2-15. Entries of R, L, G, and C matrices are distributed among columns 16-31 in corresponding order (see the previous). Columns 32-37 contain Re(Rc), Im(Rc), Re(Rp), Im(Rp), BetaC, and BetaP; where Rc is the ratio of C - mode voltage in the second line to C - mode voltage in the first line; Rp is the same ratio in P-mode (see details in [2] [10–201], section 4.3.1) and BetaC and BetaP are propagation constants of C- and P-modes in Rad/m. Note that column 38 and 39 contain traditional effective dielectric constants ErC_Eff and ErP_Eff (they do not account for losses). The total number of columns in the file is 39 (at N=2).

If N=, MNCLIN creates a file that contains complex characteristic impedance Re(Zo) and Im(Zo), complex effective dielectric constant Re(Eeff) and Im (Eff) (see traditional effective dielectric constant in column 12), and Loss (dB/m) in columns 2-6. Columns 7-10 contain R, L, G, and C. Column 11 contains propagation constant Beta in Rad/m. Note that column 12 contains traditional effective dielectric constant Er_Eff that does not account for loss. Total number of columns in the file is 12 (at N=1).

The created text file might be linked or imported to a project as a data file and you can view the frequency behavior of any above mentioned parameter using the proper data measurement. Note that the first column (frequency) is always in

GHz so these measurements might be incompatible with other MWO measurements placed on the same graph; you may prefer to place these data measurements on a separate dedicated graph.

FileName. By default this parameter is hidden and is set to model *MnCLIN.TXT.* You can change the file name for each model instance to an arbitrary name with a length not exceeding 64 symbols.

Parameter Restrictions and Recommendations

1. The number of conductors N cannot exceed 50.
2. For more information about restrictions and recommendations common to MNCLIN and MXCLIN, see MxCLIN.

Implementation Details

1. Model implementation is based on the EM Quasi-Static technique described in [1] [10–201]. It computes matrices of per-unit-length RLGC parameters and uses them to evaluate circuit parameters of coupled lines. This model saves frequency-independent RLGC matrices to disk cache. Before calculation of RLGC matrix models, it checks to see if the disk cache contains data that has been saved earlier with the same set of input parameters. If a match is found, the model reads RLGC matrices from a disk cache and saves time on their calculation. If no match is found, the model calculates RLGC matrices and places a new record into the disk cache. All subsequent runs of any project containing this model with the identical set of input parameters use the disk cache for speed up.
2. This model accounts for losses in metal and in substrate dielectric. Dispersion is partly included (in part of frequency dependence of output parameters due to presence of frequency dependent losses).
3. To apply Method of Moments for analysis, a quasi-static model creates 1D mesh covering contours of all conductors. The mesh is made of linear segments (pulses) of varying length. The length of a pulse is relatively big at the conductor center; it decreases toward conductor edges to reveal the charge distribution across conductor. If the conductor width is too large it may cause the pulse size to approach zero for pulses close to edge. In these rare cases the model may display a “Length of pulse #nnn equal to zero” error message.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See “Cell Options Dialog Box: Layout Tab ” for Cell Options dialog box **Layout** tab details.

See “The Layout Process File (LPF)” for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

Recommendations for Use

See MxCLIN for details.

NOTE: The implementation of EM Quasi-Static models relies heavily on the involved numerical algorithms. This may lead to a noticeable increase in simulation time for schematics that employ many such models.

If a layer thickness is too small compared to the thickness of another layer, simulation time may also noticeably increase.

Caution regarding units of data in saved text files: If a project that reads saved text files uses frequency, resistance, inductance or conductance units different from GHz, ohm, henry or siemens, you may need to scale input values manually.

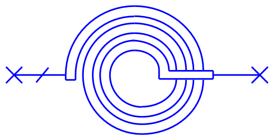
References

[1] M.B. Bazdar, A.R. Djordjevic, R.F. Harrington, and T.K. Sarkar, "Evaluation of quasi-static matrix parameters for multiconductor transmission lines using Galerkin's method," IEEE Trans. Microwave Theory Tech., vol. MTT-42, July 1994, pp. 1223-1228.

[2] R. Mongia, I. Bahl, and P. Bhartia, RF and Microwave Coupled-Line Circuits, Artech House, Norwood, MA, 1999.

Octagonal Spiral Inductor (Average Potential Method): MOCTIND

Symbol

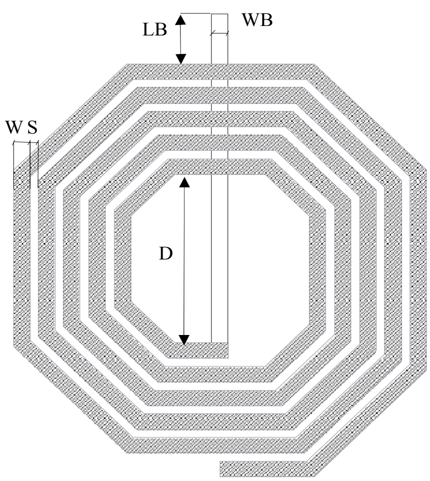


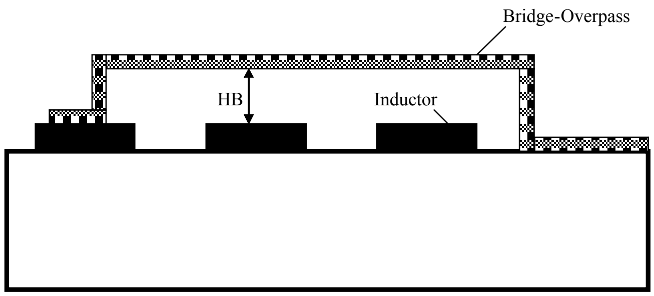
Summary

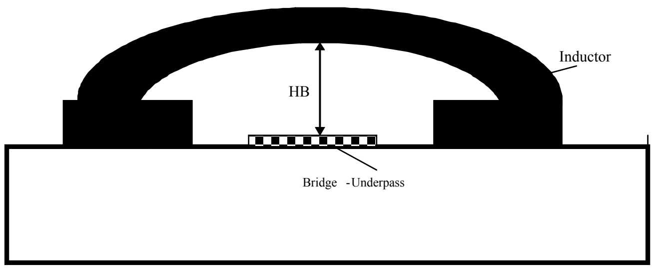
MOCTIND models a microstrip octagonal spiral inductor with a strip underpass/overpass. It is intended for modeling octagonal inductors on alumina and GaAs substrates. The inductor is located on the top of a ground-backed dielectric substrate. This model is based on evaluation of self and mutual inductances, capacitances, admittances, and resistances between all spiral turns. These equivalent circuit parameters are obtained using the average potential method in conjunction with the approximation of static Green's functions.

MOCTIND has a big speed advantage over the FEM based FMOCIND.

Topology







Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	MI1
NT	Number of turns		3
W	Conductor width	Length	10 um
S	Conductor spacing	Length	2 um
D	Internal diameter measured between internal edge of contact pad of Port 2 and edge of opposite spiral segment	Length	20 um
UExit	Underpass exits toward near/far side of spiral ("Near exit/Far exit")		Far exit
WB	Width of underpass conductor	Length	10 um
HB	Height of underpass conductor above substrate/Height of inductor above underpass ("Suspended/On-substrate underpass")	Length	2 um
LB	Length of underpass conductor extension beyond inductor	Length	0 um
TB	Thickness of underpass conductor	Length	1 um
ErB	Relative dielectric constant of dielectric under overpass bridge		1
TandB	Loss tangent of dielectric under overpass bridge		1
RhoB	Underpass metal bulk resistivity normalized to gold		1
*BrMode	Switch Overpass/Underpass	Text	Overpass
MSUB	Substrate definition	Text	MSUB1[a]

[a]Modify only if the schematic contains multiple substrates. See "Using Elements With Model Blocks" for details.

** indicates a secondary parameter*

Parameter Details

NT. Number of turns NT must provide an integer number of quarter-turns, that is 1, 1.25, 1.5 etc. This model accepts any value for NT in the range 1..50 but if NT does not provide an integer number of quarter-turns the model changes NT to the closest suitable value.

UExit. Allows the direction of the underpass either to the nearest turn ("Near exit", see the "Topology" section) or to span the internal opening before "diving" under the conducting turns ("Far exit").

HB. Height of the underpass is the distance between the substrate top (top of high-doped layer if present) and the bottom of the underpass conductor.

LB. The length of the extension of underpass conductor beyond the external edge of inductor body.

BrMode. This switch defines the position of the bridge connected to port 2 relative to the inductor metal (see the "Topology" section). The Overpass value makes the model imply that the bridge runs above the inductor windings and is separated from the inductor's metal by deposited dielectric or air. The Underpass value implies that the bridge runs on the top of the substrate while the inductor windings form arcs above the bridge; thus, the bridge actually runs under windings and makes the underpass (GaAs specific).

Parameter Restrictions and Recommendations

$1 \leq NT \leq 50$

Implementation Details

The Average-potential technique developed for this model is a generalization of formulations derived in [1] [10–204] for the case of grounded dielectric substrate.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See “Cell Options Dialog Box: Layout Tab ” for Cell Options dialog box **Layout** tab details.

See “The Layout Process File (LPF)” for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

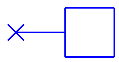
References

[1] Grover, F.W., Inductance Calculations, Mineola, New York: Dover Publishing, 2004

[2] Bahl, I.J., Lumped Elements for RF and Microwave Circuits, Norwood, MA: Artech House, 2003

Microstrip Open Circuit with End Effect (Closed Form): MOPEN

Symbol



Parameters

MOPEN$ is a Microstrip iCell and has no parameters. See "Intelligent Cells (iCells)" for more information.

Name	Description	Unit Type	Default
ID	Name	Text	TL1
W	Conductor width	Length	W[a]
MSUB	Substrate definition	Text	MSUB#[b]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See the Microwave Office Layout Guide for details.

[b]Modify only if schematic contains multiple substrates. See "Using Elements With Model Blocks" for details.

Implementation Details

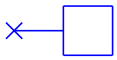
MSUB indicates the name of the MSUB element to use for substrate characteristics. If blank, a default is used.

Layout

This element does not have an assigned layout cell nor does it use line types to determine its layout. The element models an open end effect.

Microstrip Open End Effect (EM Base): MOPENX

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	MO1
W	Conductor width	Length	W[a]
MSUB	Substrate definition	Text	MSUB#[b]
*AutoFill	AutoFill DataBase if not equal to 0		0

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See the Microwave Office Layout Guide for details.

[b]Modify only if schematic contains multiple substrates. See "Using Elements With Model Blocks" for details.

** indicates a secondary parameter*

Implementation Details

This model implements an X-model of the microstrip open-ended line. For more X-model information see " EM-based Models (X-models) ". This model does not include the effects of dielectric, conductor or radiation losses.

Layout

This element does not have an assigned layout cell nor does it use line types to determine its layout. The element models an open end effect.

Restrictions

$0.05 \leq W/H \leq 8$ Recommended	$\varepsilon_r \leq 16.0$ Recommended
$0 \leq$ Frequency $\leq F_{max}$ Recommended	$1 \leq \varepsilon_{rnominal}$ Required
$(100*\lvert\varepsilon_r-\varepsilon_{rnominal}\rvert) \leq 20$ Required	$(100*\lvert\varepsilon_r-\varepsilon_{rnominal}\rvert)/\varepsilon_{rnominal} \leq 10$ Recommended

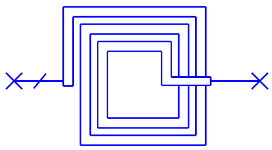
F_{max}= The frequency limits of this model are dynamic with respect to the dimensions of the discontinuity. This dynamic frequency limit is displayed to the user via warning messages for the relative size, and dielectric in use. Importantly, this recommended frequency limit will change as a function of the largest width in the discontinuity for a given substrate definition. The frequency limit warns the user that at least one of the transmission lines constructing the discontinuity is approaching the frequency limit where multiple modes can propagate. See the section marked "Upper Frequency Limitations" in the General Discussion of the X-models.

References

This model was developed under research performed at NI AWR Corporation. The details of the implementation are considered proprietary in nature.

Rectangular Microstrip Inductor without Airbridge/Underpass (EM Quasi-Static): MRINDNB2

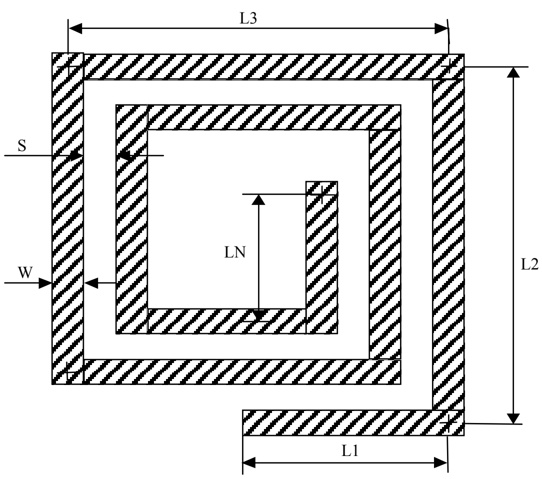
Symbol



Summary

This circuit component models a microstrip rectangular inductor without airbridge at port 2. You can implement any model of bridge to attach port 2 to an external circuit. This model is based on an evaluation of self and mutual inductances, capacitances, and resistances between all parallel segments, which in turn is based on an accurate quasi-static model of an arbitrary number of edge-coupled microstrip lines. The bridge conductor crosses the inductor on its top and is capacitively coupled to all crossed segments. MRINDNB2 is an updated version of MRINDNBR; the update provides better compliance with HSPICE requirements.

Topology



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	MI1
NS	Number of linear segments (>=4)		15
L1	Length of first segment	Length	80 um
L2	Length of second segment	Length	155 um
L3	Length of third segment	Length	165 um
LN	Length of last segment	Length	35 um
W	Conductor width	Length	10 um
S	Conductor spacing	Length	5 um

Name	Description	Unit Type	Default
*Acc	Accuracy parameter		1
MSUB	Substrate definition	Text	MSUB1[a]

[a]Modify only if schematic contains multiple substrates. See "Using Elements With Model Blocks" for details.

** indicates a secondary parameter*

Parameter Details

NS. The number of linear conductor segments forming the inductor. NS should be greater than 4 and less than NSMAX. The value of NSMAX can be evaluated from the condition LNMAX >0, where

LNMAX = L2-(NS-2)(W+S)/2 for even NS

LNMAX = L3-(NS-3)(W+S)/2 for odd NS

The layout feasibility check is run before performing calculations.

LN. The length of the last segment LN should not exceed LNMAX (see previous). If you define too large a value of LN, the model automatically sets LN to LNMAX and issues a warning. LN also should not be less than W/2. If you define too small a value of LN, the model automatically sets LN to W/2 and issues a warning.

Acc. The accuracy parameter. The default value for Acc is 1. If Acc is less than 1 or greater than 10 it is set automatically to 1.

In90deg, Out90deg (layout cell parameter only): Note that the corresponding layout cell of this model has In90deg and Out90deg parameters (to edit these parameter values select the corresponding layout cell, right-click and choose **Shape Properties**). On the **Parameters** tab of the dialog box that displays, setting these parameters to a nonzero value means that the location of faces at the junction either at port 1 (In90deg) or at port 2 (Out90deg) provides a connection to an external circuit via a right (90deg) bend. Correspondingly, setting these parameters to zero means that the location of a face at the corresponding junction provides an "in line" connection to an external circuit. Default values are zeros. Setting it to nonzero values (for example, to 1) doesn't affect the electrical properties of the model; no bend component is added automatically and you must attach the model of bend to the corresponding port at schematics.

Parameter Restrictions and Recommendations

1. NS should be greater than 4 and less than NSMAX. The value of NSMAX can be evaluated from the condition LNMAX >0 (see previous).
2. The Acc parameter is limited to 1≤ACC≤10. Larger values of Acc increase the density of the mesh used in computations. The accuracy of model parameters may improve slightly from increasing Acc, at the expense of a noticeable increase in computation time. Generally, a good trade-off between accuracy and computation time is to set Acc to 1.

Implementation Details

To decrease the calculation time for schematics that contain several MRINDNB2 inductors, cache is implemented for this model. This means that during the first evaluation of a schematic the most time-consuming intermediate parameters for each inductor instance are being stored in a disk cache. Each inductor model checks this cache looking for its duplicate. Duplicate inductors copy the appropriate parameters from the memory cache, saving substantially on their recalculation.

Note that the model caches only frequency-independent characteristics of coupled lines, but recalculates the large equivalent circuit network (derived from coupled line characteristics) at each swept frequency. Thus, if the number of swept frequency points is large (for example, 300) the total time spent on equivalent circuit evaluation may substantially exceed the time for evaluation of coupled line characteristics. In this case, time saving due to caching may be relatively moderate.

NOTE: The implementation of EM Quasi-Static models relies heavily on the involved numerical algorithms. This may lead to a noticeable increase in simulation time for schematics that employ many such models.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See “Cell Options Dialog Box: Layout Tab ” for Cell Options dialog box **Layout** tab details.

See “The Layout Process File (LPF)” for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

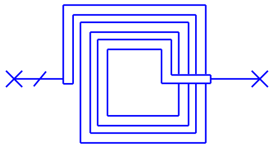
References

[1] M. B. Bazdar, A. R. Djordjevic, R. F. Harrington, and T. K. Sarkar, "Evaluation of quasi-static matrix parameters for multiconductor transmission lines using Galerkin's method", IEEE Trans. Microwave Theory Tech., vol. MTT-42, July 1994, pp. 1223-1228

[2] M. Kirschning, R. H. Jansen, N. H .L. Koster, "Measurement and computer-aided modeling of microstrip discontinuities by an improved resonator method," IEEE MTT-S International Microwave Symposium Digest, 1983, pp. 495-497.

Rectangular Microstrip Inductor without Bridge (EM Quasi-Static): MRINDNBR

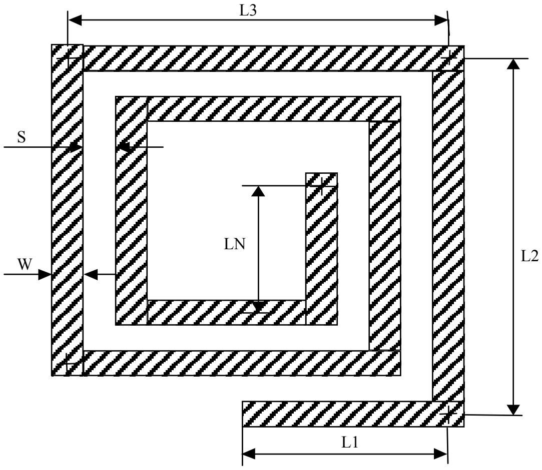
Symbol



Summary

This circuit component models a microstrip rectangular inductor without bridge at internal port 2. You can implement any model of bridge to attach port 2 to an external circuit. MRINDNBR is based on evaluation of self- and mutual-inductances, capacitances, and resistances between all parallel segments, which is based on an accurate quasi-static model of an arbitrary number of edge-coupled microstrip lines.

Topology



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	MI1
NS	Number of linear segments (>=4)		15
L1	Length of first segment	Length	80 um
L2	Length of second segment	Length	155 um
L3	Length of third segment	Length	165 um
LN	Length of last segment	Length	35 um
W	Conductor width	Length	10 um
S	Conductor spacing	Length	5 um
MSUB	Substrate definition	Text	MSUB1[a]

[a]Modify only if schematic contains multiple substrates. See "Using Elements With Model Blocks" for details.

Parameter Details

NS. The number of linear conductor segments forming the inductor. NS should be greater than 4 and less than NSMAX. The value of NSMAX can be evaluated from the condition LNMAX >0, where

LNMAX = L2-(NS-2)(W+S)/2 for even NS

LNMAX = L3-(NS-3)(W+S)/2 for odd NS

The layout feasibility check is run before performing calculations.

LN. The length of the last segment LN should not exceed LNMAX (see previous). If you define too large a value of LN, the model automatically sets LN to LNMAX and issues a warning. LN also should not be less than W/2. If you define too small a value of LN, the model automatically sets LN to W/2 and issues a warning.

In90deg, Out90deg. (Layout cell): Note that the corresponding layout cell of this model has parameters In90deg, Out90deg (to edit values of these parameters, select the corresponding layout cell, right-click and choose **Shape Properties**). On the **Parameters** tab of the dialog box that displays, setting these parameters to nonzero values means that the location of faces at the junction either at port 1 (In90deg) or at port 2 (Out90deg) provides connection to an external circuit via right (90deg) bend. Correspondingly, setting these parameters to zero means that the location of the face at the corresponding junction provides an "in line" connection to an external circuit. The default values are zeros. Setting to nonzero values (for example, to 1) does not affect electrical properties of the model. No bend component is added automatically and you must attach the model of bend to the corresponding port at schematics.

Parameter Restrictions and Recommendations

1. NS should be greater than 4 and less than NSMAX. The value of NSMAX can be evaluated from the condition LNMAX >0 (see previous).

Implementation Details

To decrease the calculation time for schematics that contain several MRINDNBR inductors, cache is implemented for this model. This means that during the first evaluation of a schematic the most time-consuming intermediate parameters for each inductor instance are being stored in memory cache. Each inductor model checks this cache looking for its duplicate. Duplicate inductors copy the appropriate parameters from memory cache, saving substantially on their recalculation.

Note that the model caches only frequency-independent characteristics of coupled lines, but recalculates the large equivalent circuit network (derived from coupled line characteristics) at each swept frequency. Thus, if the number of swept frequency points is large (say, 300) the total time spent on equivalent circuit evaluation may substantially exceed the time for evaluation of coupled line characteristics. In this particular case, time saving due to caching may be relatively moderate.

NOTE: The implementation of EM Quasi-Static models relies heavily on the involved numerical algorithms. This may lead to a noticeable increase in simulation time for schematics that employ many such models.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.

2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See "Cell Options Dialog Box: Layout Tab " for Cell Options dialog box **Layout** tab details.

See "The Layout Process File (LPF)" for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

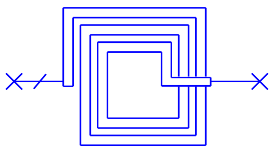
References

[1] M.B. Bazdar, A.R. Djordjevic, R.F. Harrington, and T.K. Sarkar, "Evaluation of quasi-static matrix parameters for multiconductor transmission lines using Galerkin's method," IEEE Trans. Microwave Theory Tech., vol. MTT-42, July 1994, pp. 1223-1228

[2] M. Kirschning, R.H. Jansen, N.H.L. Koster, "Measurement and computer-aided modeling of microstrip discontinuities by an improved resonator method," IEEE MTT-S International Microwave Symposium Digest, 1983, pp. 495-497.

(Obsolete) Rectangular Microstrip Inductor with Strip Bridge (EM Quasi-Static): MRINDSB2

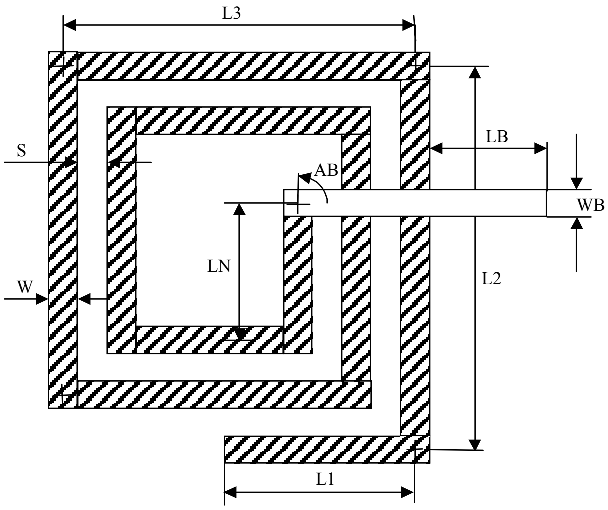
Symbol

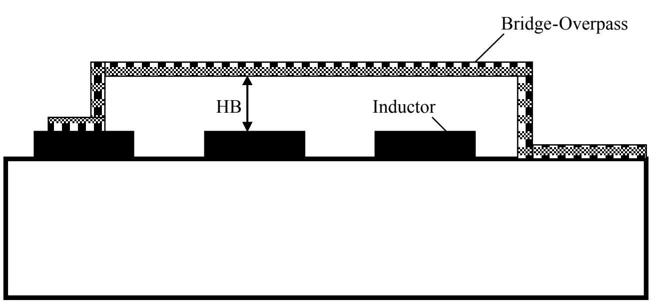


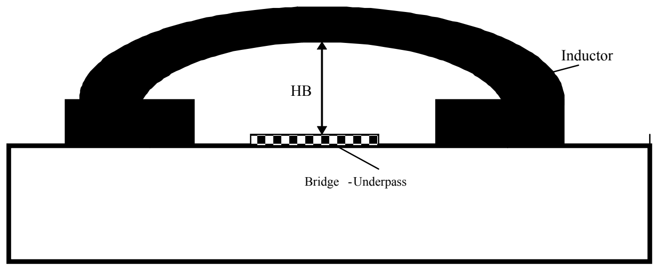
Summary

This element is OBSOLETE and is replaced by the Rectangular Microstrip Inductor with Strip Bridge (EM Quasi-Static) (MRINDSB3) element. MRINDSB2 models a microstrip rectangular inductor with strip bridge. This model is based on an evaluation of self and mutual inductances, capacitances and resistances between all parallel segments, which in turn is based on an accurate quasi-static model of an arbitrary number of edge-coupled microstrip lines. MRINDSB2 is an updated version of MRINDSBR. The update provides better compliance with HSPICE requirements. MRINDSB2 also features a switch parameter to allow an underpass bridge for GaAs designs.

Topology







Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	MI1
NS	Number of linear segments (>=4)		15
L1	Length of first segment	Length	80 um
L2	Length of second segment	Length	155 um
L3	Length of third segment	Length	165 um
LN	Length of last segment	Length	35 um
AB	Angle of bridge departure	Angle	0 deg
W	Conductor width	Length	10 um
S	Conductor spacing	Length	5 um
WB	Width of bridge strip conductor	Length	10 um
HB	Height of bridge dielectric	Length	2 um
LB	Length of bridge extension beyond inductor	Length	0 um
EPSB	Relative dielectric constant of bridge dielectric		1
TDB	Loss tangent of bridge dielectric		0
TB	Thickness of bridge strip	Length	1 um
RhoB	Bridge metal bulk resistivity normalized to gold		1
BrMode	Switch Overpass/Underpass	Text	Overpass
*Acc	Accuracy parameter		1
MSUB	Substrate definition	Text	MSUB1[a]

[a]Modify only if schematic contains multiple substrates. See “Using Elements With Model Blocks” for details.

* indicates a secondary Parameter

Parameter Details

NS. The number of linear conductor segments forming the inductor. NS should be greater than 4 and less than NSMAX. The value of NSMAX can be evaluated from the condition LNMAX >0, where

LNMAX = L2-(NS-2)(W+S)/2 for even NS

LNMAX = L3-(NS-3)(W+S)/2 for odd NS

The layout feasibility check is run before performing calculations.

LN. The length of the last segment LN should not exceed LNMAX (see above). If you define too large a value of LN, the model automatically sets LN to LNMAX and issues a warning. LN also should not be less than (W+WB)/2. If you define too small a value of LN the model automatically sets LN to (W+WB)/2 and issues a warning.

AB. Angle AB (degrees) defines the direction of the bridge departure from the end of the last segment. Only 0, 90, 180 and 270 are allowed. A zero angle has a bridge that is parallel to L1 and goes to the opposite direction. Angle is measured counterclockwise. Any intermediate value of AB is set to the closest acceptable value.

MRINDSB2 does not allow the bridge to overlap the last segment. If this occurs, the model changes AB so that the bridge departs in the opposite direction (this is made for correct evaluation of the capacitance coupling between bridge and inductor segments).

NOTE:: The layout cell does not reflect a change in bridge orientation in version 5.53 and earlier versions. From version 6.53 forward, a new version of layout cell overrides the setting for AB and sets the bridge exit to the opposite direction if overlapping occurs. By default, projects created in version 5.53 and earlier versions use an old version layout cell; while new projects created in 6.53 and later use a new layout cell. You can update the layout cell assigned to MRINDSB2 by double-clicking the model symbol in the schematic window, and selecting either the new layout cell (MRINDSBR2) or the old layout cell (MRINDSBR) on the **Layout** tab.

You should not allow the bridge to cross the first segment (the length of this segment is controlled by parameter L1) due to parasitic coupling between input and output and possible issues with layout. If MRINSB2 discovers this intersection it issues a warning and a recommendation to review inductor layout.

BrMode. This switch defines the position of the bridge connected to port 2 relative to the inductor metal (see "Topology"). The Overpass value makes the model imply that the bridge runs above the inductor windings and is separated from the inductor's metal by deposited dielectric or air. The Underpass value implies that the bridge runs atop of the substrate while the inductor windings form arcs above the bridge; thus, the bridge actually runs under windings and makes the underpass (GaAs specific).

Acc. The parameter Acc is the accuracy parameter. The default value for Acc is 1. If Acc is less than 1 or greater than 10 it is set automatically to 1.

Out90deg (layout cell parameter only): Note that the layout cell for this model has a Out90deg parameter (to edit values of this parameter select the corresponding layout cell, right-click it and choose **Shape Properties/Parameters**). Setting this parameter to a nonzero value means that the orientation of a face at port 2 provides a connection to an external circuit via right (90deg) bend. Correspondingly, setting this parameter to zero means that the orientation of a face at port 2 provides an"in line" (no bend) connection to an external circuit. The default value is zero. Setting this to a nonzero value (say, to 1) doesn't affect the electrical properties of the model; no bend component is added automatically. You can attach any bend model to the port 2 if needed.

Parameter Restrictions and Recommendations

1. NS should be greater than 4 and less than NSMAX. The value of NSMAX can be evaluated from the condition LNMAX >0 (see above).
2. Accuracy parameter Acc is limited to 1≤ACC≤10. Larger value of Acc increases density of mesh used in computations. Accuracy of model parameters may gain slightly from increasing Acc at the expense of noticeable growth of computation time. As a rule of thumb, good trade-off between accuracy and computation time is to set Acc to 1.

Implementation Details

To decrease the calculation time for schematics that contain several MRINDSB2 inductors, cache is implemented for this model. This means that during the first evaluation of a schematic the most time consuming intermediate parameters for each inductor instance are being stored in the disk cache. Each inductor model checks this cache looking for its duplicate. Duplicate inductors copy the appropriate parameters from the disk cache, saving substantially on their recalculation.

Note that the model caches only frequency-independent characteristics of coupled lines, but recalculates the large equivalent circuit network (derived from coupled line characteristics) at each swept frequency. Thus, if the number of swept frequency points is large (say, 300) the total time spent on equivalent circuit evaluation may substantially exceed the time for evaluation of coupled line characteristics. In this particular case, time saving due to caching may be relatively moderate.

NOTE: The implementation of EM Quasi-Static models relies heavily on the involved numerical algorithms. This may lead to a noticeable increase in simulation time for schematics that employ many such models.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See “Cell Options Dialog Box: Layout Tab ” for Cell Options dialog box **Layout** tab details.

See “The Layout Process File (LPF)” for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

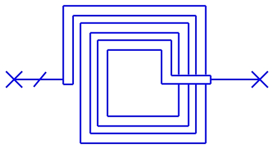
References

[1] M.B. Bazdar, A.R. Djordjevic, R.F. Harrington, and T.K. Sarkar, "Evaluation of quasi-static matrix parameters for multiconductor transmission lines using Galerkin's method," IEEE Trans. Microwave Theory Tech., vol. MTT-42, July 1994, pp. 1223-1228

[2] M. Kirschning, R.H. Jansen, N.H.L.Koster, "Measurement and computer-aided modeling of microstrip discontinuities by an improved resonator method," IEEE MTT-S International Microwave Symposium Digest, 1983, pp. 495-497.

Rectangular Microstrip Inductor with Strip Bridge (EM Quasi-Static): MRINDSB3

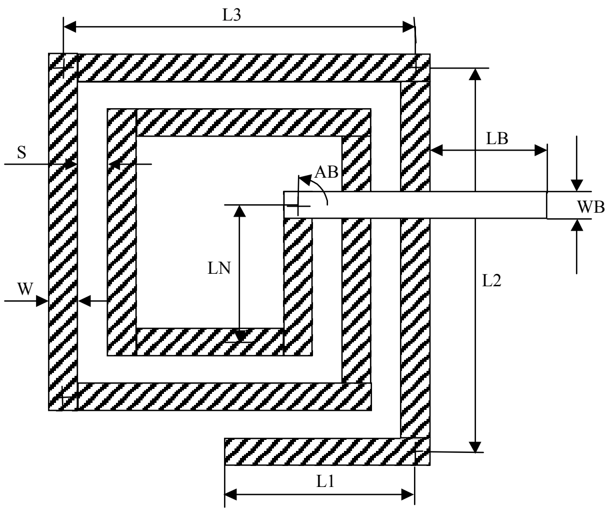
Symbol

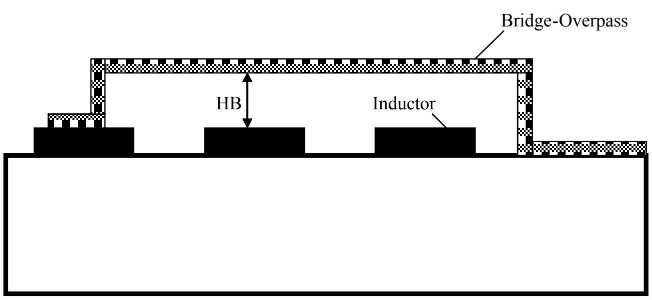


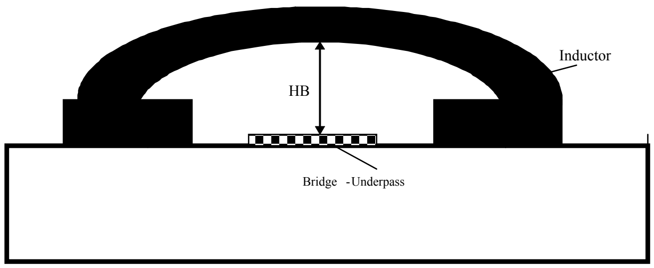
Summary

MRINDSB3 models a microstrip rectangular inductor with strip bridge. This model is based on an evaluation of self and mutual inductances, capacitances, and resistances between all parallel segments, which in turn is based on an accurate quasi-static model of an arbitrary number of edge-coupled microstrip lines. MRINDSB3 is an updated version of MRINDSBR. The update provides better compliance with HSPICE requirements. MRINDSB3 also features a switch parameter to allow an underpass bridge for GaAs designs.

Topology







Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	MI1
NS	Number of linear segments (>=4)		15
L1	Length of first segment	Length	80 um
L2	Length of second segment	Length	155 um
L3	Length of third segment	Length	165 um
LN	Length of last segment	Length	35 um
AB	Angle of bridge departure	Angle	0 deg
W	Conductor width	Length	10 um
S	Conductor spacing	Length	5 um
WB	Width of bridge strip conductor	Length	10 um
HB	Height of bridge dielectric	Length	2 um
LB	Length of bridge extension beyond inductor	Length	0 um
EPSB	Relative dielectric constant of bridge dielectric		1
TDB	Loss tangent of bridge dielectric		0
TB	Thickness of bridge strip	Length	1 um
RhoB	Bridge metal bulk resistivity normalized to gold		1
BrMode	Switch Overpass/Underpass	Text	Overpass
*Acc	Accuracy parameter		1
MSUB	Substrate definition	Text	MSUB1[a]

[a]Modify only if schematic contains multiple substrates. See "Using Elements With Model Blocks" for details.

** indicates a secondary parameter*

Parameter Details

NS. The number of linear conductor segments forming the inductor. NS should be greater than 4 and less than NSMAX. The value of NSMAX can be evaluated from the condition LNMAX >0, where

LNMAX = L2-(NS-2)(W+S)/2 for even NS

LNMAX = L3-(NS-3)(W+S)/2 for odd NS

The layout feasibility check is run before performing calculations.

LN. The length of the last segment LN should not exceed LNMAX (see previous). If you define too large a value of LN, the model automatically sets LN to LNMAX and issues a warning. LN also should not be less than (W+WB)/2. If you define too small a value of LN the model automatically sets LN to (W+WB)/2 and issues a warning.

AB. Angle AB (degrees) defines the direction of the bridge departure from the end of the last segment. Only 0, 90, 180 and 270 are allowed. A zero angle has a bridge that is parallel to L1 and goes to the opposite direction. Angle is measured counterclockwise. Any intermediate value of AB is set to the closest acceptable value.

MRINDSB3 does not allow the bridge to overlap the last segment. If this occurs, the model changes AB so that the bridge departs in the opposite direction (this is made for correct evaluation of the capacitance coupling between bridge and inductor segments).

MRINDSB3 does not allow the bridge to cross the first segment (the length of which is controlled by parameter L1) due to parasitic coupling between input and output and possible issues with layout. If MRINSB3 discovers this intersection, it displays an error and a message recommending review of the inductor layout.

BrMode. This switch defines the position of the bridge connected to port 2 relative to the inductor metal (see "Topology"). The Overpass value makes the model imply that the bridge runs above the inductor windings and is separated from the inductor metal by deposited dielectric or air. The Underpass value implies that the bridge runs atop the substrate while the inductor windings form arcs above the bridge; thus, the bridge actually runs under windings and makes the underpass (GaAs specific).

Acc. The accuracy parameter. The default value for Acc is 1. If Acc is less than 1 or greater than 10 it is set automatically to 1.

Out90deg. (layout cell parameter only): Note that the layout cell for this model has a Out90deg parameter (to edit values of this parameter select the corresponding layout cell, right-click and choose **Shape Properties**). On the **Parameters** tab of the dialog box that displays, setting this parameter to a nonzero value means that the orientation of a face at port 2 provides a connection to an external circuit via right (90deg) bend. Correspondingly, setting this parameter to zero means that the orientation of a face at port 2 provides an "in line" (no bend) connection to an external circuit. The default value is zero. Setting this to a nonzero value (for example, to 1) does not affect the electrical properties of the model; no bend component is added automatically. You can attach any bend model to the port 2 if needed.

Parameter Restrictions and Recommendations

1. NS should be greater than 4 and less than NSMAX. The value of NSMAX can be evaluated from the condition LNMAX >0 (see previous).
2. The Acc parameter is limited to $1 \leq ACC \leq 10$. Larger values of Acc increase the density of the mesh used in computations. The accuracy of model parameters may improve slightly from increasing Acc, at the expense of a noticeable increase in computation time. Generally, a good trade-off between accuracy and computation time is to set Acc to 1.

Implementation Details

To decrease the calculation time for schematics that contain several MRINDSB3 inductors, cache is implemented for this model. This means that during the first evaluation of a schematic the most time-consuming intermediate parameters for each inductor instance are being stored in a disk cache. Each inductor model checks this cache looking for its duplicate. Duplicate inductors copy the appropriate parameters from the disk cache, saving substantially on their recalculation.

Note that the model caches only frequency-independent characteristics of coupled lines, but recalculates the large equivalent circuit network (derived from coupled line characteristics) at each swept frequency. Thus, if the number of swept frequency points is large (for example, 300) the total time spent on equivalent circuit evaluation may substantially exceed the time for evaluation of coupled line characteristics. In this case, time saving due to caching may be relatively moderate.

NOTE: The implementation of EM Quasi-Static models relies heavily on the involved numerical algorithms. This may lead to a noticeable increase in simulation time for schematics that employ many such models.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See “Cell Options Dialog Box: Layout Tab ” for Cell Options dialog box **Layout** tab details.

See “The Layout Process File (LPF)” for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

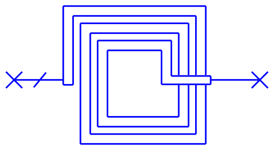
References

[1] M. B. Bazdar, A. R. Djordjevic, R. F. Harrington, and T. K. Sarkar, "Evaluation of quasi-static matrix parameters for multi-conductor transmission lines using Galerkin's method", IEEE Trans. Microwave Theory Tech., vol. MTT-42, July 1994, pp. 1223-1228

[2] M. Kirschning, R.H. Jansen, N.H.L. Koster, "Measurement and computer-aided modeling of microstrip discontinuities by an improved resonator method", IEEE MTT-S International Microwave Symposium Digest, 1983, pp. 495-497.

Rectangular Microstrip Inductor Strip Bridge (EM Quasi-Static): MRINDSBR

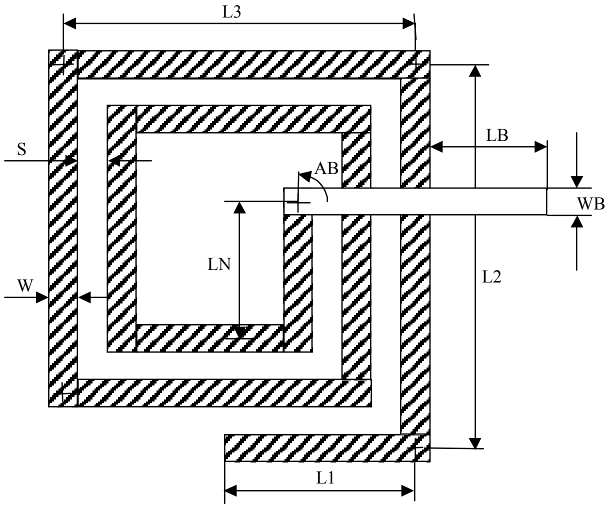
Symbol



Summary

This circuit component models a microstrip rectangular inductor with strip bridge. MRINDSBR is based on evaluation of self and mutual inductances, capacitances, and resistances between all parallel segments, which is based on accurate quasi-static model of arbitrary number of edge coupled microstrip lines.

Topology



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	MI1
NS	Number of linear segments (>=4)		15
L1	Length of first segment	Length	80 um
L2	Length of second segment	Length	155 um
L3	Length of third segment	Length	165 um
LN	Length of last segment	Length	35 um
AB	Angle of bridge departure	Angle	0 deg
W	Conductor width	Length	10 um
S	Conductor spacing	Length	5 um
WB	Width of bridge strip conductor	Length	10 um

Name	Description	Unit Type	Default
HB	Height of bridge dielectric	Length	2 um
LB	Length of bridge extension beyond inductor	Length	0 um
EPSB	Relative dielectric constant of bridge dielectric		1
TDB	Loss tangent of bridge dielectric		0
TB	Thickness of bridge strip	Length	1 um
RhoB	Bridge metal bulk resistivity normalized to gold		1
MSUB	Substrate definition	Text	MSUB1[a]

[a]Modify only if schematic contains multiple substrates. See "Using Elements With Model Blocks" for details.

Parameter Details

NS. The number of linear conductor segments forming the inductor. NS should be greater than 4 and less than NSMAX. The value of NSMAX can be evaluated from the condition LNMAX >0, where

LNMAX = L2-(NS-2)(W+S)/2 for even NS

LNMAX = L3-(NS-3)(W+S)/2 for odd NS

The layout feasibility check is run before performing calculations.

LN. The length of the last segment LN should not exceed LNMAX (see previous). If you define too large a value of LN, the model automatically sets LN to LNMAX and issues a warning. LN also should not be less than (W+WB)/2. If you define too small a value of LN, the model automatically sets LN to (W+WB)/2 and issues a warning.

AB. Angle AB (degrees) defines the direction of bridge departure from the end of the last segment. Only 0, 90, 180 and 270 are allowed for AB. A zero angle has a bridge that is parallel to L1 and goes to the opposite direction. Angle is measured counterclockwise. Any intermediate value of AB is set to the closest acceptable value.

Bridge is not allowed to overlap the last segment. If this occurs, the model changes AB so that the bridge departs in the opposite direction (this is made only for evaluation of capacitance coupling between bridge and inductor segments).

NOTE: The layout cell does not reflect a change in bridge orientation in version 5.53 and earlier versions. Starting with version 6.53, a new version of layout cell overrides the setting for AB and sets the bridge exit to the opposite direction if overlapping occurs. By default, projects created in version 5.53 and earlier versions use the old version of the layout cell. Projects created in 6.53 and later use the new layout cell by default. You can update the layout cell assigned to this model. To do so, double-click the model symbol in the Schematic window to open the Element Options dialog box for the model, click the **Layout** tab and select either MRINDSBR2 as a new layout cell or MRINDSBR as the old layout cell.

Out90deg (Layout cell parameter only): Note that the layout cell for this model has parameter a Out90deg parameter (to edit values of this parameter select the corresponding layout cell, right-click and choose **Shape Properties**). On the **Parameters** tab of the dialog box that displays, setting this parameter to nonzero means that the orientation of a face at port 2 provides a connection to an external circuit via right (90deg) bend. Correspondingly, setting this parameter to zero means that the orientation of a face at port 2 provides an "in line" (no bend) connection to external circuit. The default value is zero. Setting this to a nonzero value (for example, to 1) does not effect the electrical properties of the model; no bend component is added automatically. You can attach any bend model to the port 2 if needed.

Parameter Restrictions and Recommendations

1. NS should be greater than 4 and less than NSMAX. The value of NSMAX can be evaluated from the condition LNMAX >0 (see previous).

Implementation Details

To decrease the calculation time for schematics that contain several MRINDSBR inductors, cache is implemented for this model. This means that during the first evaluation of the schematic the most time-consuming intermediate parameters for each inductor instance are being stored in memory cache. Each inductor model checks this cache looking for its duplicate. Duplicate inductors copy the appropriate parameters from memory cache, saving substantially on their recalculation.

Note that the model caches only frequency-independent characteristics of coupled lines, but recalculates the large equivalent circuit network (derived from coupled line characteristics) at each swept frequency. Thus, if the number of swept frequency points is large (for example, 300) the total time spent on equivalent circuit evaluation may substantially exceed the time for evaluation of coupled line characteristics. In this case, the time saving due to caching may be relatively moderate.

Caution: The implementation of EM Quasi-Static models relies heavily on the involved numerical algorithms. This may lead to a noticeable increase in simulation time for schematics that employ many such models.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See "Cell Options Dialog Box: Layout Tab " for Cell Options dialog box **Layout** tab details.

See "The Layout Process File (LPF)" for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

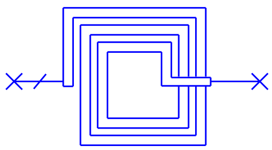
References

[1] M. B. Bazdar, A. R. Djordjevic, R. F. Harrington, and T. K. Sarkar, "Evaluation of quasi-static matrix parameters for multiconductor transmission lines using Galerkin's method", IEEE Trans. Microwave Theory Tech., vol. MTT-42, July 1994, pp. 1223-1228

[2] M. Kirschning, R.H. Jansen, N. H. L. Koster, "Measurement and computer-aided modeling of microstrip discontinuities by an improved resonator method", IEEE MTT-S International Microwave Symposium Digest, 1983, pp. 495-497.

Rectangular Microstrip Inductor with Wire Bridge (EM Quasi-Static): MRINDWBR

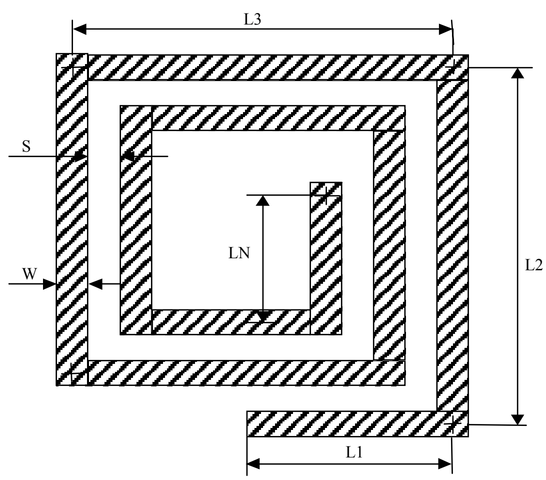
Symbol



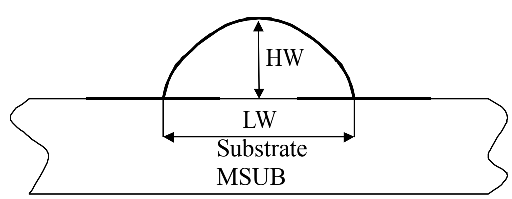
Summary

This circuit component models a microstrip rectangular inductor with wire bridge at an internal port. MRINDWBR is based on an evaluation of self- and mutual-inductances, capacitances, and resistances between all parallel segments, which is based on an accurate quasi-static model of an arbitrary number of edge-coupled microstrip lines.

Topology



Inductor



Wire bridge

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	MI1

Name	Description	Unit Type	Default
NS	Number of linear segments (>=4)		15
L1	Length of first segment	Length	80 um
L2	Length of second segment	Length	155 um
L3	Length of third segment	Length	165 um
LN	Length of last segment	Length	35 um
W	Conductor width	Length	10 um
S	Conductor spacing	Length	5 um
LW	Wire ridge length	Length	500 um
DiaW	Diameter of bridge wire	Length	20 um
HW	Wire bridge height	Length	500 um
RhoW	Bridge wire bulk resistivity normalized to gold		1
MSUB	Substrate definition	Text	MSUB1[a]

[a]Modify only if the schematic contains multiple substrates. See "Using Elements With Model Blocks" for details.

Parameter Details

NS. The number of linear conductor segments forming the inductor. NS should be greater than 4 and less than NSMAX. The value of NSMAX can be evaluated from the condition LNMAX >0, where

LNMAX = L2-(NS-2)(W+S)/2 for even NS

LNMAX = L3-(NS-3)(W+S)/2 for odd NS

The layout feasibility check is run before performing calculations.

LN. The length of the last segment LN should not exceed LNMAX (see previous). If you define too large a value of LN, the model automatically sets LN to LNMAX and issues a warning. LN also should not be less than W/2. If you define too small a value of LN, the model automatically sets LN to W/2 and issues a warning.

LW. The distance between bridge wire attachment points (see Wire bridge in the "Topology" section). LW should be large enough to allow the bridge to reach an attachment point beyond the inductor boundary.

HW. The maximal height of the wire bridge above the substrate.

In90deg, Out90deg. (Layout cell): Note that the corresponding layout cell of this model has In90deg and Out90deg parameters (to edit these parameters, select the corresponding layout cell, right-click and choose **Shape Properties** to display the Cell Options dialog box). On the **Parameters** tab, setting these parameters to nonzero values means that the location of faces at the junction either at port 1 (In90deg) or at port 2 (Out90deg) provides connection to an external circuit via a right (90deg) bend. Correspondingly, setting these parameters to zero means that the location of the face at the corresponding junction provides an "in line" connection to an external circuit. The default values are zeros. Setting these to nonzero values (for example, to 1) does not affect the electrical properties of the model. No bend component is added automatically and you must attach the model of bend to the corresponding port at schematics.

Parameter Restrictions and Recommendations

1. NS should be greater than 4 and less than NSMAX. The value of NSMAX can be evaluated from the condition LNMAX >0 (see previous).

2. You should enter a sufficient value of LW to provide a bridge long enough to reach an attachment point beyond the inductor boundary.

Implementation Details

To decrease the calculation time for schematics that contain several MRINDWBR inductors, cache is implemented for this model. During the first evaluation of a schematic, the most time-consuming intermediate parameters for each inductor instance are stored in memory cache. Each inductor model checks this cache looking for its duplicate. Duplicate inductors copy the appropriate parameters from memory cache, saving substantially on their recalculation.

Note that this model caches only frequency-independent characteristics of coupled lines, but recalculates the large equivalent circuit network (derived from coupled line characteristics) at each swept frequency. Thus, if the number of swept frequency points is large (for example, 300) the total time spent on equivalent circuit evaluation may substantially exceed the time for evaluation of coupled line characteristics. In this case, time saving due to caching may be relatively moderate.

This model does not account for coupling between bridge wire and inductor segments. However, the wire bridge is substrate-aware, so the HW parameter may affect MRINDWBR performance.

NOTE: The implementation of EM Quasi-Static models relies heavily on the involved numerical algorithms. This may lead to a noticeable increase in simulation time for schematics that employ many such models.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See “Cell Options Dialog Box: Layout Tab ” for Cell Options dialog box **Layout** tab details.

See “The Layout Process File (LPF)” for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

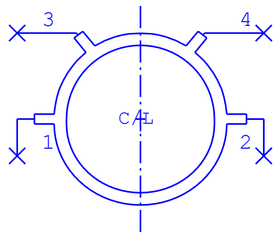
References

[1] M.B. Bazdar, A.R. Djordjevic, R.F. Harrington, and T.K. Sarkar, "Evaluation of quasi-static matrix parameters for multiconductor transmission lines using Galerkin's method," IEEE Trans. Microwave Theory Tech., vol. MTT-42, July 1994, pp. 1223-1228

[2] M. Kirschning, R.H. Jansen, N.H.L. Koster, "Measurement and computer-aided modeling of microstrip discontinuities by an improved resonator method," IEEE MTT-S International Microwave Symposium Digest, 1983, pp. 495-497.

(Obsolete) Microstrip Rat-Race Coupler (Aggregate): MRRCOUP

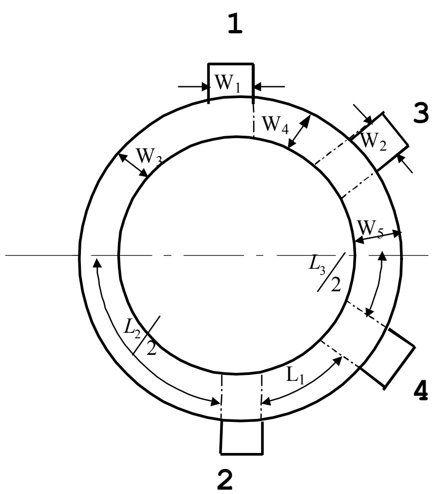
Symbol



Summary

This element is OBSOLETE and is replaced by the Microstrip Rat-Race Coupler (Aggregate) (MRRCOUP2) element. MRRCOUP models microstrip rat-race coupler (180° hybrid). Model is a general one, i.e. it allows arbitrary lengths and widths of feeding lines as well as of all sections of the ring lines.

Topology



Parameters

Name	Description	Unit Type	Default
ID	Name	Text	TL1
W1	Width of feeding line at ports 1 and 2	Length	W[a]
W2	Width of feeding line at port 3 and 4	Length	W[a]
W3	Width of the line connecting ports 1 and 2	Length	W[a]
W4	Width of the line connecting ports 1,3 and 2,4	Length	W[a]
W5	Width of the Iine connecting ports 3 and 4	Length	W[a]
L1	Length of the line connecting ports 1,3 and ports 2,4	Length	L[a]
L2	Length of the line connecting ports 1 and 2 (about 3/4 w/len)	Length	L[a]

Name	Description	Unit Type	Default
L3	Length of the line connecting ports 3 and 4	Length	L[a]
MSUB	Substrate definition	Text	MSUB1[b]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See the Microwave Office Layout Guide for details.

[b]Modify only if schematic contains multiple substrates. See "Using Elements With Model Blocks" for details.

Parameter Details

L1. Line connecting ports 1,3 and line connecting ports 2,4 have the same length L1.

L2. Full length of line connecting ports 1 and 2.

L3. Full length of line connecting ports 3 and 4.

Parameter Restrictions and Recommendations

1. MRRCOUP uses MCURVE and MTEE (microstrip arc line and microstrip T- junction) models, so all limitations of these models are applicable to MRRCOUP.
2. For equal split 3 dB coupler (equal power split and 180^{0} phase shift between ports 3 and 4):

   W_1=W_2;

   W_3=W_4=W_5

   L_1=L_3=$\lambda_{eff}/4$;(quarter of effective wavelength)

   L_2=$3\lambda_{eff}/4$

   W_3,W_4, and W_5 provide characteristic impedance of corresponding lines equal to $\sqrt{2}$ times characteristic impedance of feeding lines.

Implementation Details

MRRCOUP implies that coupler is mirror symmetrical relative to horizontal axis (see Topology).

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Recommendations for Use

Feeding lines at ports may have different characteristic impedances. This is provided by proper choice of width parameters W1, W2 (see Topology).

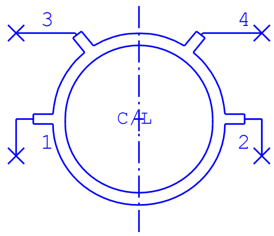
To ensure specified power split, matching, isolation and phase shift between branched lines parameters may be derived in accordance with the recommendations in References [1]. However, user may set parameters to values of his choice or use them for circuit optimization.

References

[1] Mongia, R., Bahl,I.J., and Bhartia P., RF and microwave coupled-line couplers, Artech House, Norwood, MA, 1999, pp. 260-264

Microstrip Rat-Race Coupler (Aggregate): MRRCOUP2

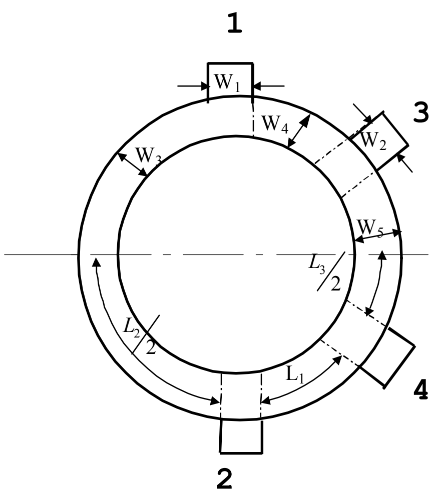
Symbol



Summary

MRRCOUP2 models a microstrip rat-race coupler (180° hybrid). MRRCOUP2 is a general model; it allows arbitrary lengths and widths of feeding lines and sections of the ring lines.

Topology



Parameters

Name	Description	Unit Type	Default
ID	Name	Text	TL1
W1	Width of feeding line at ports 1 and 2	Length	W[a]
W2	Width of the feeding line at port 3 and 4	Length	W[a]
W3	Width of the line connecting ports 1 and 2	Length	W[a]
W4	Width of the line connecting ports 1,3 and 2,4	Length	W[a]
W5	Width of the line connecting ports 3 and 4	Length	W[a]
L1	Length of the line connecting ports 1,3 and ports 2,4	Length	L[a]
L2	Length of the line connecting ports 1 and 2 (about 3/4 w/len)	Length	L[a]
L3	Length of the line connecting ports 3 and 4	Length	L[a]

Name	Description	Unit Type	Default
MSUB	Substrate definition	Text	MSUB1[b]

[a]User-modifiable default. Modify by editing the *default.lpf* file in the root installation directory. See the “The Layout Process File (LPF)” for details.
[b]Modify only if the schematic contains multiple substrates. See “Using Elements With Model Blocks” for details.

Parameter Details

L1. Line connecting ports 1,3 and line connecting ports 2,4 have the same length L1.

L2. Full length of line connecting ports 1 and 2.

L3. Full length of line connecting ports 3 and 4.

Parameter Restrictions and Recommendations

1. MRRCOUP2 uses MCURVE and MTEE (microstrip arc line and microstrip T- junction) models, so all limitations of these models apply to MRRCOUP2.
2. For equal split 3 dB coupler (equal power split and 180^o phase shift between ports 3 and 4):

   $W_1=W_2$;

   $W_3=W_4=W_5$

   $L_1=L_3=\lambda_{eff}/4$; (quarter of effective wavelength)

   $L_2=3\lambda_{eff}/4$

   W_3,W_4, and W_5 provide characteristic impedance of corresponding lines equal to √2 times the characteristic impedance of feeding lines.

Implementation Details

MRRCOUP2 implies that a coupler is mirror symmetrical relative to the horizontal axis (see the "Topology" section).

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

Recommendations for Use

Feeding lines at ports may have different characteristic impedances. This is provided by proper choice of width parameters W1, W2 (see the "Topology" section).

To ensure the specified power split, matching, isolation and phase shift between branched lines, parameters may be derived in accordance with the recommendations in Reference [1]. However, you may set parameters to values of your choice or use them for circuit optimization.

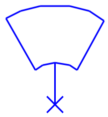
*NOTE:*The MRRCOUP element is considered obsolete due to errors in port numbering; MRRCOUP2 is recommended for use.

References

[1] Mongia, R., Bahl,I.J., and Bhartia P., RF and microwave coupled-line couplers, Artech House, Norwood, MA, 1999, pp. 260-264

(Obsolete) Microstrip Radial Stub Series (Closed Form): MRSTUB

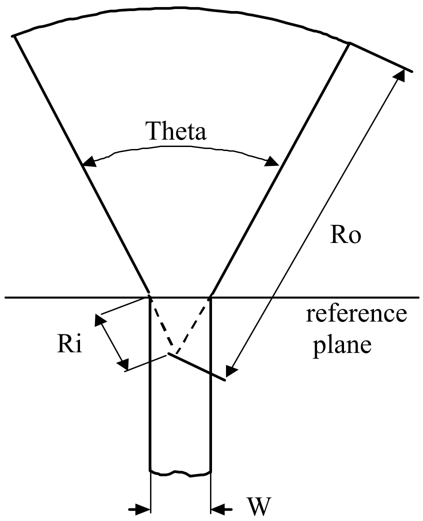
Symbol



Summary

This element is OBSOLETE and is replaced by the Microstrip Radial Stub Series (Closed Form) (MRSTUB2W) element.

Topology



Parameters

Name	Description	Unit Type	Default
ID	Name	Text	ST#[a]
Ro	Outer radius of the stub	Length	L[b]
Ri	Inner radius of the stub	Length	L[b]
Theta	Angle of the stub	Angle	90 Deg
MSUB	Substrate definition	Text	MSUB[c]

[a]# is automatically incremented as additional ST element ID's are added to the schematic.

[b]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory.

[c]If only one MSUB is present in the schematic, this substrate is automatically used. If multiple MSUB substrate definitions are present, the user must specify.

Implementation Details

This circuit component models a series radial stub on a single layer substrate. This stub is recommended for use as a termination (shunt stubs should be modeled with MSRSTUB or MDRSTUB). The parameters Ri (inner radius) and Ro (outer radius) are dimensions entered in the default length units.Theta is a radial stub angle entered in degrees. The parameter MSUB specifies the substrate element, which defines additional cross sectional parameters of the transmission line and substrate characteristics. If blank, a default is used. The stub is connected to a single node at the reference plane.

Inner radius Ri can be easily obtained from known microstrip width W:

$$R_i = \frac{W}{2\sin(\mathrm{Theta}/2)}$$

The model is valid up to its second open-circuit resonance and for $9 \leq \mathrm{Theta} \leq 180$.

The component accounts for losses in metal and in substrate dielectric. Dispersion is not included.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See “Cell Options Dialog Box: Layout Tab ” for Cell Options dialog box **Layout** tab details.

See “The Layout Process File (LPF)” for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

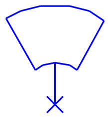
References

[1] S.L. March, "Analyzing lossy radial-line stubs," IEEE Trans. Microwave Theory Tech., vol. MTT-33, March 1985, pp. 269-271

[2] F.Giannini, R.Ruggieri, and J.Vrba, "Planar circuit analysis of microstrip radial stub", IEEE Trans. Microwave Theory Tech., vol. MTT-32, December 1984, pp. 1652-16551

(Obsolete) Microstrip Radial Stub Series (Closed Form): MRSTUB2

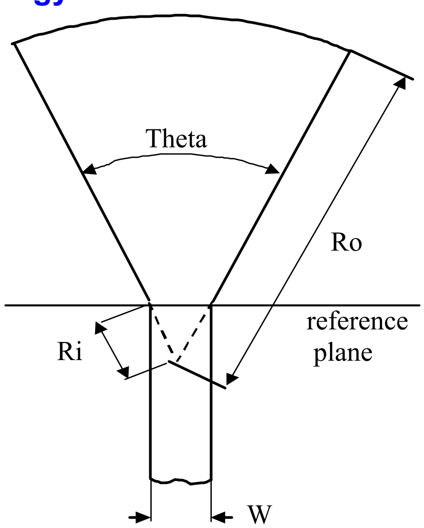
Symbol



Summary

This element is OBSOLETE and is replaced by the Microstrip Radial Stub Series (Closed Form) (MRSTUB2W) element. MRSTUB2 models a microstrip radial stub that terminates a microstrip line. The stub is connected to a single node at the reference plane.

Topology



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	ST1
Ri	Inner radius of the stub	Length	L[a]
Ro	Outer radius of the stub	Length	L[a]
Theta	Angle of the stub	Angle	90 Deg
MSUB	Substrate definition	Text	MSUB1[b]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See Microwave Office Layout Guide for details.

[b]Modify only if schematic contains multiple substrates. See the "Using Elements With Model Blocks" for details.

Parameter Details

Ri. The inner radius Ri can be easily obtained from the specified microstrip width W and stub angle Theta:

$$R_i = \frac{W}{2\sin(\mathrm{Theta}/2)}$$

Parameter Restrictions and Recommendations

1. Theta must be within the range $9 \leq \mathrm{Theta} \leq 160$.
2. This model is feasible if $(R_O - R_i)/R_O > 0.01$.

Implementation Details

Model implementation is based on cascading multiple segments of lossy microstrip transmission lines described in [1]. It accounts for losses in metal and in substrate dielectrics. Radiation loss is not included.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See "Cell Options Dialog Box: Layout Tab " for Cell Options dialog box **Layout** tab details.

See "The Layout Process File (LPF)" for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

Recommendations for Use

MRSTUB2 replaces the MRSTUB element, which is obsolete.

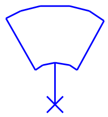
Discontinuity models function most accurately when attached to lines that match their corresponding edges. Directly connecting discontinuity models to one another reduces their accuracy.

References

[1] B.C.Wadell, Transmission Line Design Book, Artech House, Boston-London, 1991, pp.306-307

Microstrip Radial Stub Series (Closed Form): MRSTUB2W

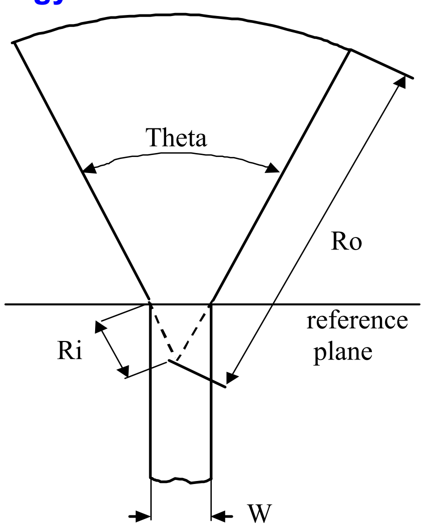
Symbol



Summary

MRSTUB2W models a microstrip radial stub that terminates a microstrip line. The stub is connected to a single node at the reference plane.

Topology



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	ST1
W	Width of connecting microstrip line	Length	L[a]
Ro	Outer radius of the stub	Length	L[a]
Theta	Angle of the stub	Angle	90 Deg
MSUB	Substrate definition	Text	MSUB1[b]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See the Microwave Office Layout Guide for details.

[b]Modify only if schematic contains multiple substrates. See "Using Elements With Model Blocks" for details.

Parameter Details

W. The conductor width of the connecting microstrip line. If needed, the inner radius Ri can be easily obtained from the specified microstrip width W and the stub angle Theta:

$$R_i = \frac{W}{2\sin(\text{Theta}/2)}$$

Parameter Restrictions and Recommendations

1. Theta must be within the range $9 \le \text{Theta} \le 160$.
2. This model is feasible if $(R_o\text{-}R_i)/R_o > 0.01$.

Implementation Details

Model implementation is based on cascading multiple segments of lossy microstrip transmission lines described in [1]. It accounts for losses in metal and in substrate dielectrics. Radiation loss is not included.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See "Cell Options Dialog Box: Layout Tab " for Cell Options dialog box **Layout** tab details.

See "The Layout Process File (LPF)" for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

Recommendations for Use

MRSTUB2W replaces the MRSTUB element, which is obsolete. MRSTUB2W is also recommended for use instead of MRSTUB2 because MRSTUB2W is easily accessible for measuring the W parameter, while MRSTUB2 relies on the less convenient internal radius Ri.

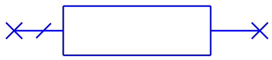
Discontinuity models function most accurately when attached to lines that match their corresponding edges. Directly connecting discontinuity models to one another reduces their accuracy.

References

[1] B.C.Wadell, Transmission Line Design Book, Artech House, Boston-London, 1991, pp.306-307

Suspended Substrate Microstrip Line (EM Quasi-Static): MS1LIN

Symbol



Summary

MS1LIN models a section of single microstrip line with a conductor strip placed on the upper surface of a suspended substrate (suspended substrate is a single layer substrate elevated over the infinite grounded plane).

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	TL1
W	Conductor width	Length	W[a]
L	Conductor length	Length	L[a]
Acc	Accuracy parameter		1
MPSUB	Substrate definition	Text	MPSUB1[b]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See the Microwave Office Layout Guide for details.

[b]Modify only if schematic contains multiple substrates. See "Using Elements With Model Blocks" for details.

Parameter Details

MPSUB. Suspended substrate parameters are listed in MPSUB model description.

Acc. The parameter Acc is the accuracy parameter. The default value for Acc is 1. If Acc is less than 1 or greater than 10 it is set automatically to 2.

Parameter Restrictions and Recommendations

1. Accuracy parameter A is limited to $1 \le Acc \le 10$. Larger value of Acc increases density of mesh used in computations. Accuracy of model parameters may gain slightly from increasing Acc at the expense of noticeable growth of computation time. As a rule of thumb, good trade-off between accuracy and computation time is to set Acc to 1.
2. This component does not impose restrictions on the conductor thickness (thickness may be zero, positive, or negative). Negative thickness means that the conductor is recessed into the substrate.

Implementation Details

Model implementation is based on EM Quasi-Static technique described in [1] [10–240]. It accounts for losses in metal and in substrate dielectric. Dispersion is partly included

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.

2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See “Cell Options Dialog Box: Layout Tab ” for Cell Options dialog box **Layout** tab details.

See “The Layout Process File (LPF)” for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

Recommendations for Use

NOTE: The implementation of EM Quasi-Static models relies heavily on the involved numerical algorithms. This may lead to a noticeable increase in simulation time for schematics that employ many such models.

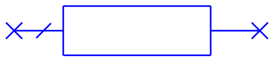
If thickness of any layer is too small in comparison with the thickness of another layer, simulation time may also noticeably grow.

References

[1] M.B. Bazdar, A.R. Djordjevic, R.F. Harrington, and T.K. Sarkar, "Evaluation of quasi-static matrix parameters for multiconductor transmission lines using Galerkin's method," IEEE Trans. Microwave Theory Tech., vol. MTT-42, July 1994, pp. 1223-1228.

Suspended Microstrip Line (EM Quasi-Static Based): MS1LINX

Symbol



Summary

MS1LINX models a section of single microstrip line with a conductor strip placed on the upper surface of a suspended substrate (suspended substrate is a single layer substrate elevated over the infinite grounded plane).

This model is constructed as an X-model (table-based interpolation) using the results of a EM 2-D quasi-static cross-sectional analysis based on Method of Moments. For a more detailed discussion of the X-models see " EM-based Models (X-models) ". EM 2-D quasi-static analysis is the same method as that used in the MS1LIN, however, MS1LINX gains large computational speed increases due to the table-based interpolation.

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	TL1
W	Conductor width	Length	W[a]
L	Line length	Length	L[a]
Acc	Accuracy parameter		1
MPSUB	Substrate definition	Text	See[b]
*AutoFill	Autofill database if not equal to 0		0

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory.

[b]If only one MPSUB is present in the schematic, this substrate is automatically used. If multiple MPSUB substrate definitions are present, you must specify which to use.

** indicates a secondary parameter*

Parameter Details

W. Conductor width is an independent parameter.

Acc. The input parameter Acc is an accuracy parameter influencing the EM 2-D quasi-static cross-sectional analysis, and can range from 1 to 10. This value is used to automatically fill the model tables if the necessary table is missing and parameter Autofill is set to 1. Higher values of Acc may improve accuracy but also may slow down the filling process. M1LINX does not use Acc in normal, table-based interpolation mode.

MPSUB. Suspended substrate parameters are listed in the MPSUB model description.

Autofill. The AutoFill parameter is a hidden input which allows you to specify that the entire interpolation table should be filled automatically at the current values. To instigate this filling process, set this parameter equal to 1. During normal operation, this parameter should be set equal to zero. You can access the hidden parameter by double-clicking on the schematic element.

Parameter Restrictions and Recommendations

1. MS1LINX implies that the ratio W/H1 (H1 is substrate thickness) lies within a *predefined* range $0.05 \le W/H1 \le 18.5$. Outside of this range, this model extrapolates output parameters and issues a warning.
2. Substrate thickness H1, substrate elevation above ground H2, nominal dielectric constant ErNom, dielectric tangent Tand, relative conductor bulk resistance Rho, and accuracy Acc are fixed parameters. Microwave Office provides pre-generated tables for several typical values of MS1LINX fixed parameters. Changes to any fixed parameter may start the automatic filling process (if Autofill is set to 1), the length of which depends on the Acc value. You can change any fixed parameter to create corresponding tables.
3. The dielectric constant Er of substrate MPSUB is a statistical parameter. It means that models account for the relative deviation of Er from ErNom within 20%; a larger deviation demands a new fill of the model tables.

Implementation Details

This model was developed under research performed at NI AWR Corporation. The full set of details of the implementation are considered proprietary in nature.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See "Cell Options Dialog Box: Layout Tab " for Cell Options dialog box **Layout** tab details.

See "The Layout Process File (LPF)" for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

Recommendations for Use

To create new tables for substrate dielectric constants different from those supplied with Microwave Office, you need to set ErNom = Er = needed-value-of-dielectric-constant, set AutoFill = 1 and simulate.

In exchange for a speed increase, small errors resulting from the interpolation should be expected and the range of the input parameters is restricted.

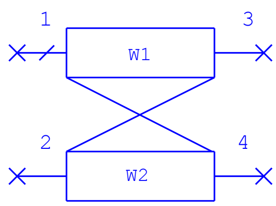
If you wish to implement values outside of the specified range of W/H1 (see *Parameter Restrictions and Recommendations*), you can use the MS1LIN element.

References

[1] M.B. Bazdar, A.R. Djordjevic, R.F. Harrington, and T.K. Sarkar, "Evaluation of quasi-static matrix parameters for multiconductor transmission lines using Galerkin's method," IEEE Trans. Microwave Theory Tech., vol. MTT-42, July 1994, pp. 1223-1228

2 Suspended Substrate Edge Coupled Microstrip Lines (EM Quasi-Static): MS2CLIN

Symbol



Summary

MS2CLIN models section of two edge coupled asymmetric microstrip lines on a suspended substrate, that is, on a single layer substrate elevated over the infinite grounded plane.

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	TL1
W1	Width of conductor 1	Length	W[a]
W2	Width of conductor 2	Length	W[a]
S	Spacing between conductors	Length	L[a]
L	Conductor length	Length	L[a]
Acc	Accuracy parameter		1
MPSUB	Substrate definition	Text	MPSUB1[b]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See the Microwave Office Layout Guide for details.

[b]Modify only if schematic contains multiple substrates. See "Using Elements With Model Blocks" for details.

Parameter Details

MPSUB. Suspended substrate parameters are listed in MPSUB model description.

Acc. The parameter Acc is the accuracy parameter. The default value for Acc is 1. If Acc is less than 1 or greater than 10 it is set automatically to 2.

Parameter Restrictions and Recommendations

1. Accuracy parameter A is limited to $1 \le Acc \le 10$. Larger value of Acc increases density of mesh used in computations. Accuracy of model parameters may gain slightly from increasing Acc at the expense of noticeable growth of computation time. As a rule of thumb, good trade-off between accuracy and computation time is to set Acc to 1.
2. This component does not impose restrictions on the conductor thickness (thickness may be zero, positive, or negative). Negative thickness means that the conductor is recessed into the substrate.

Implementation Details

Model implementation is based on EM Quasi-Static technique described in [1]. It accounts for losses in metal and in substrate dielectric. Dispersion is partly included

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See “Cell Options Dialog Box: Layout Tab ” for Cell Options dialog box **Layout** tab details.

See “The Layout Process File (LPF)” for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

Recommendations for Use

NOTE: The implementation of EM Quasi-Static models relies heavily on the involved numerical algorithms. This may lead to a noticeable increase in simulation time for schematics that employ many such models.

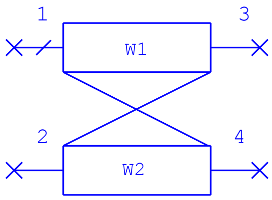
If thickness of any layer is too small in comparison with the thickness of another layer, simulation time may also noticeably grow.

References

[1] M.B. Bazdar, A.R. Djordjevic, R.F. Harrington, and T.K. Sarkar, "Evaluation of quasi-static matrix parameters for multiconductor transmission lines using Galerkin's method," IEEE Trans. Microwave Theory Tech., vol. MTT-42, July 1994, pp. 1223-1228.

2 Suspended Substrate Edge Coupled Microstrip Lines (EM Quasi-Static Based): MS2CLINX

Symbol



Summary

MS2CLINX models a section of two symmetric edge coupled microstrip lines with conductor strips placed on the upper surface of a suspended substrate (suspended substrate is a single layer substrate elevated above the infinite grounded plane).

This model is constructed as an X-model (table-based interpolation) using the results of a EM 2-D quasi-static cross-sectional analysis based on Method of Moments. For a more detailed discussion of the X-models see " EM-based Models (X-models) " . EM 2-D quasi-static analysis is the same method as that used in the MS2CLIN, however, MS2CLINX gains large computational speed increases due to the table-based interpolation.

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	TL1
W	Conductor width	Length	W[a]
S	Spacing between conductors	Length	W[a]
L	Line length	Length	L[a]
Acc	Accuracy parameter		1
MPSUB	Substrate definition	Text	See[b]
*AutoFill	Autofill database if not equal to 0		0

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory.

[b]If only one MPSUB is present in the schematic, this substrate is automatically used. If multiple MPSUB substrate definitions are present, you must specify which to use.

** indicates a secondary parameter*

Parameter Details

W. Conductor width is an independent parameter.

S. Spacing between conductors is an independent parameter.

Acc. The input parameter Acc is an accuracy parameter influencing the EM 2-D quasi-static cross-sectional analysis, and can range from 1 to 10. This value is used to automatically fill the model tables if the necessary table is missing and parameter Autofill is set to 1. Higher values of Acc may improve accuracy but also may slow the filling process. MS2CLINX does not use Acc in normal, table-based interpolation mode.

MPSUB. Suspended substrate parameters are listed in the MPSUB model description.

Autofill. The AutoFill parameter is a hidden input which allows you to specify that the entire interpolation table should be filled automatically at the current values. To instigate this filling process, this parameter should be set equal to 1. During normal operation, this parameter should be set equal to zero. You can access the hidden parameter by double-clicking on the schematic element.

Parameter Restrictions and Recommendations

1. MS2CLINX implies that the ratio W/H1 lies within the predefined range of $0.05 \leq W/H1 \leq 18$ and the ratio S/H1 lies within the predefined range of $0.025 \leq S/H1 \leq 70$ (H1 is substrate thickness) Outside of these ranges, the model extrapolates output parameters and issues a warning.
2. Substrate thickness H1, substrate elevation above ground H2, nominal dielectric constant ErNom, dielectric tangent Tand, relative conductor bulk resistance Rho, and accuracy Acc are fixed parameters. Microwave Office provides pre-generated tables for several typical values of MS2CLINX fixed parameters. Changes to any fixed parameter may start the automatic filling process (if Autofill is set to 1), the length of which depends on the Acc value. You can change any fixed parameter to create corresponding tables.
3. The dielectric constant Er of the substrate MPSUB is a statistical parameter. It means that models account for relative deviation of Er from ErNom within 20%; larger deviation demands a new fill of model tables.

Implementation Details

This model was developed under research performed at NI AWR Corporation. The full set of details of the implementation are considered proprietary in nature.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See “Cell Options Dialog Box: Layout Tab ” for Cell Options dialog box **Layout** tab details.

See “The Layout Process File (LPF)” for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

Recommendations for Use

To create new tables for the substrate dielectric constant different from those supplied with Microwave Office, you must set ErNom = Er = needed-value-of-dielectric-constant, set AutoFill = 1 and simulate.

In exchange for speed increase, small errors resulting from the interpolation should be expected and the range of the input parameters is restricted.

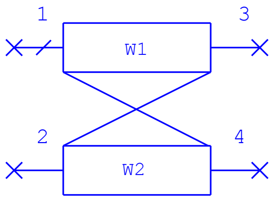
If you want to implement values outside of the specified ranges of W/H1, S/H1 (see *Parameter Restrictions and Recommendations*), you can use the MS2CLIN element.

References

[1] M.B. Bazdar, A.R. Djordjevic, R.F. Harrington, and T.K. Sarkar, "Evaluation of quasi-static matrix parameters for multiconductor transmission lines using Galerkin's method," IEEE Trans. Microwave Theory Tech., vol. MTT-42, July 1994, pp. 1223-1228

2 Suspended Substrate Broadside Coupled Microstrip Lines (EM Quasi-Static): MSBCPL

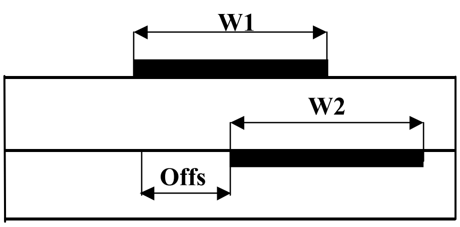
Symbol



Summary

MSBCPL models a section of broadside coupled microstrip lines located at the two opposite surfaces of a suspended substrate. The lower line may be displaced laterally relative to the left edge of the upper line.

Topology



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	TL1
W1	Upper conductor width	Length	W[a]
W2	Lower conductor width	Length	W[a]
L	Conductor length	Length	L[a]
Offs	Horizontal offset of lower line	Length	L[a]
Acc	Accuracy parameter		1
MPSUB	Substrate definition	Text	MPSUB1[b]
SNAME1	Structure name from lpf file for top conductor	Text	TOP_BCLIN
SNAME2	Structure name from lpf file for bottom conductor	Text	BOT_BCLIN

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See the Microwave Office Layout Guide for details.

[b]Modify only if the schematic contains multiple substrates. See "Using Elements With Model Blocks" for details.

Parameter Details

Acc. The accuracy parameter. The default value for Acc is 1. If Acc is less than 1 or greater than 10 it automatically defaults to 2.

Offs. Offs is a relative horizontal offset of lower line left edge from the left edge of the upper line. Offs may be positive (lower line is displaced to the right), zero (both lines are aligned) or negative (lower line is displaced to the left)

Parameter Restrictions and Recommendations

1. Acc is limited to 1≤Acc≤10. A larger value of Acc increases the density of mesh used in computations. The accuracy of model parameters may gain slightly from increasing Acc at the expense of a noticeable increase in computation time. Generally, a good trade-off between accuracy and computation time is to set Acc to 1.
2. This model does not impose restrictions on the conductor thickness (thickness may be zero, positive, or negative). If thickness parameter T (see MPSUB parameters) is positive, MSBCPL implies that the upper strip is leveled up over the substrate and the lower strip is leveled down toward the ground plane. If T is negative, both strips are recessed into the substrate.
3. SNAME1 and SNAME2 are for layout only and have no effect on the electrical performance.

Implementation Details

Model implementation is based on the EM Quasi-Static technique described in [1] [10–250]. It accounts for losses in metal and in substrate dielectric. Dispersion is partly included.

Layout

When you specify SNAME1 and SNAME2, the structures with corresponding names are identified in a $STRUCT_TYPE_BEGIN section of the LPF file. If a structure with the corresponding name is not found, the name of the missing structure is drawn on the error layer. A structure named SNAME1 must contain a text line starting with the line type name used for the top conductor. A structure named SNAME2 must contain a text line starting with the line type name used for the bottom conductor. In a structure named SNAME1=TOP_TRACE and SNAME2= BOT_TRACE with line type names "Top Copper" and "Bot Copper", you need to add the following code to the LPF file (user-defined line types may contain any number of layers; structures must contain only one text line):

```
$LINE_TYPE_BEGIN "Top Copper"
  ! -> Layer    offset    minWidth    flags
  "Cu_1"  0  5e-005  0
$LINE_TYPE_END
$LINE_TYPE_BEGIN "Bot Copper"
  ! -> Layer    offset    minWidth    flags
  "Cu_2"  0  5e-005  0
$LINE_TYPE_END
$STRUCT_TYPE_BEGIN "TOP_TRACE"
   "Top Copper" 0 0 0
$STRUCT_TYPE_END
$STRUCT_TYPE_BEGIN "BOT_TRACE"
   "Bot Copper" 0 0 0
$STRUCT_TYPE_END
```

Note that text inside structures contains line type names in quotation marks followed by three blank separated zeros.

Recommendations for Use

NOTE: The implementation of EM Quasi-Static models relies heavily on the involved numerical algorithms. This may lead to a noticeable increase in simulation time for schematics that employ many such models.

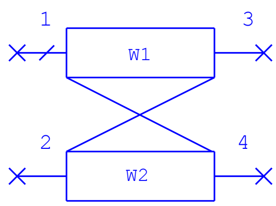
If the thickness of any layer is too small in comparison to the thickness of another layer, simulation time may also noticeably increase.

References

[1] M.B. Bazdar, A.R. Djordjevic, R.F. Harrington, and T.K. Sarkar, "Evaluation of quasi-static matrix parameters for multiconductor transmission lines using Galerkin's method," IEEE Trans. Microwave Theory Tech., vol. MTT-42, July 1994, pp. 1223-1228.

2 Suspended Substrate Broadside Coupled Microstrip Lines (EM Quasi-Static Based): MSBCPLX

Symbol

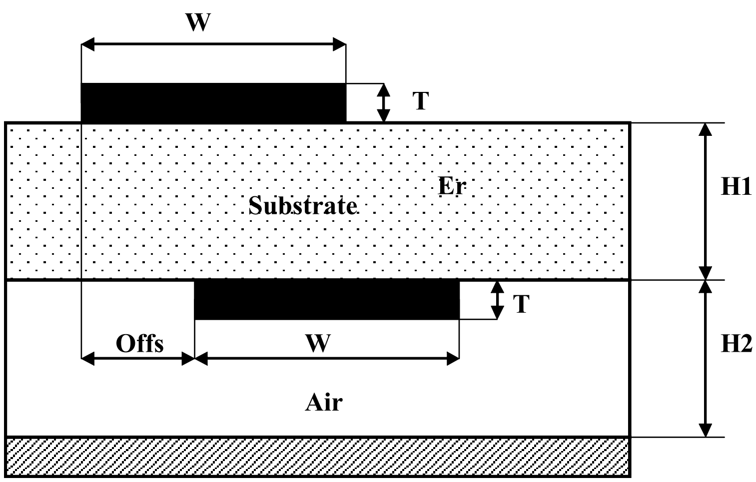


Summary

MSBCPLX models a section of two equal width broadside coupled microstrip lines with conductor strips placed on the top and bottom surfaces of a suspended substrate (suspended substrate is a single layer substrate elevated above the infinite grounded plane). Lateral offset is allowed for the bottom line.

MSBCPLX is constructed as an X-model (table-based interpolation) using the results of an EM 2-D quasi-static cross-sectional analysis based on Method of Moments. For a detailed discussion of X-models see “ EM-based Models (X-models) ”. EM 2-D quasi-static analysis is the same method as that used in the MSBCPL model, however, MSBCPLX gains large computational speed increases due to the table-based interpolation.

Topology



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	TL1
W	Conductors width	Length	W[a]
Offset	Offset of bottom conductors	Length	W[a]
L	Line length	Length	L[a]
Acc	Accuracy parameter		1

Name	Description	Unit Type	Default
MPSUB	Substrate definition	Text	See[b]
*AutoFill	Autofill database if not equal to 0		0
SNAME1	Structure name from lpf file for top conductor	Text	TOP_BCLIN
SNAME2	Structure name from lpf file for bottom conductor	Text	BOT_BCLIN

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory.

[b]If only one MPSUB is present in the schematic, this substrate is automatically used. If multiple MPSUB substrate definitions are present, you must specify which to use.

** indicates a secondary parameter*

Parameter Details

W. Conductor width is an independent parameter.

Offs. The offset of the bottom conductor is an independent parameter.

Acc. The input parameter Acc is an accuracy parameter influencing the EM 2-D quasi-static cross-sectional analysis, and can range from 1 to 10. This value is used to automatically fill the model tables if the necessary table is missing and if Autofill is set to 1. Higher values of Acc may improve accuracy but also may slow the filling process. MSBCPLX does not use Acc in normal, table-based interpolation mode.

MPSUB. Suspended substrate parameters are listed in the MPSUB model description.

Autofill. A hidden input which allows you to specify that the entire interpolation table should be filled automatically at the current values. To initiate this filling process, this parameter should be set to 1. During normal operation, this parameter should be set to zero. You can access the hidden parameter by double-clicking the schematic element.

Parameter Restrictions and Recommendations

1. MSBCPLX implies that the ratio W/H1 lies within a predefined range of $0.05 \le W/H1 \le 18$ and the ratio Offs/H1 lies within a predefined range of $0.025 \le S/H1 \le 70$ (H1 is substrate thickness). Outside of these ranges, the model extrapolates output parameters and issues a warning.
2. Substrate thickness H1, substrate elevation above ground H2, nominal dielectric constant ErNom, dielectric tangent Tand, relative conductor bulk resistance Rho, and accuracy Acc are fixed parameters. MWO provides pre-generated tables for several typical values of MSBCPLX fixed parameters. Changes to any fixed parameter may start the automatic filling process (if Autofill is set to 1), the length of which depends on the Acc value. You can change any fixed parameter to create the corresponding tables.
3. The dielectric constant Er of the substrate SPSUB is a statistical parameter. This means that models account for the relative deviation of Er from ErNom within 20%; a larger deviation demands a new fill of the model tables.
4. SNAME1 and SNAME2 are for layout only and have no effect on the electrical performance.

Implementation Details

This model was developed under research performed at NI AWR Corp. The full details of the implementation are considered proprietary in nature.

Layout

When you specify SNAME1 and SNAME2, the structures with corresponding names are identified in a $STRUCT_TYPE_BEGIN section of the LPF file. If a structure with the corresponding name is not found, the name of the missing structure is drawn on the error layer. The structure named SNAME1 must contain a text line starting with the line type name used for the top conductor. A structure named SNAME2 must contain a text line starting with the line type name used for the bottom conductor. In a structure named SNAME1=TOP_TRACE and SNAME2= BOT_TRACE with line type names "Top Copper" and "Bot Copper", you need to add the following code to the LPF file (user defined line types may contain any number of layers; structures must contain only one text line):

```
$LINE_TYPE_BEGIN "Top Copper"
  ! -> Layer   offset   minWidth   flags
  "Cu_1"  0  5e-005  0
$LINE_TYPE_END
$LINE_TYPE_BEGIN "Bot Copper"
  ! -> Layer   offset   minWidth   flags
  "Cu_2"  0  5e-005  0
$LINE_TYPE_END
$STRUCT_TYPE_BEGIN "TOP_TRACE"
   "Top Copper" 0 0 0
$STRUCT_TYPE_END
$STRUCT_TYPE_BEGIN "BOT_TRACE"
   "Bot Copper" 0 0 0
$STRUCT_TYPE_END
```

Note that text inside structures contains line type names in quotation marks followed by three blank separated zeros.

Recommendations for Use

To create new tables for the substrate dielectric constant different from those supplied with MWO, you must set ErNom = Er = needed-value-of-dielectric-constant, set AutoFill = 1 and simulate.

In exchange for a speed increase, you should expect small errors resulting from the interpolation, and the range of the input parameters is restricted.

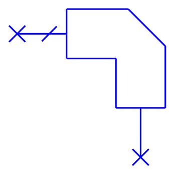
If you want to implement values outside of the specified ranges of W/H1, Offs/H1 (see the "Parameter Restrictions and Recommendations" section), you can use the MSBCPL element.

References

[1] M.B. Bazdar, A.R. Djordjevic, R.F. Harrington, and T.K. Sarkar, "Evaluation of quasi-static matrix parameters for multiconductor transmission lines using Galerkin's method," IEEE Trans. Microwave Theory Tech., vol. MTT-42, July 1994, pp. 1223-1228

Suspended Substrate Microstrip Bend, Mitered 90deg (EM Based): MSBND90X

Symbol



Summary

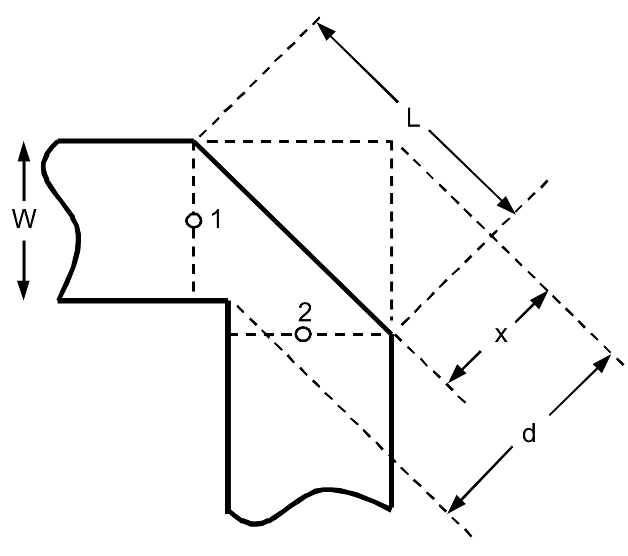
MSBND90X models a mitered bend of microstrip line placed on the top surface of a suspended substrate (suspended substrate is a single layer substrate elevated above the infinite grounded plane).

This model is constructed as an X-model (table-based interpolation) using the results of EM analysis based on Method of Moments. For a more detailed discussion of the X-models see " EM-based Models (X-models) " .

This model does not include the effects of dielectric/conductor/radiation losses; it also implies that conductor thickness is zero.

MSBND90X$ is the corresponding intelligent cell (iCell). An iCell model is identical to its non-iCell equivalent with the following exception: Certain dimension-related parameters are not explicitly user specified; rather, they are automatically and dynamically determined by the dimensions of the attached elements. See "Intelligent Cells (iCells)" for a detailed discussion of how to use iCells, their advantages, and their limitations.

Topology



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	MS1
W	Conductor width	Length	W[a]

Name	Description	Unit Type	Default
M	Miter fraction (0-0.9)		0
MPSUB	Substrate definition	Text	See[b],[c]
*AutoFill	Autofill database if not equal to 0		0

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory.

[b]If only one MPSUB is present in the schematic, this substrate is automatically used. If multiple MPSUB substrate definitions are present, you must specify which to use.

[c]Modify only if the schematic contains multiple substrates.

* indicates a secondary Parameter

Parameter Details

W. Conductor width is an independent parameter.

M. Parameter M (miter) is an independent parameter. M is defined as (see Topology)

$$M = \frac{x}{d}$$

. Other relations that hold for a right mitered bend are:

$$d = W\sqrt{2}$$

;

$$L = 2x$$

$$d - x = \sqrt{2} \cdot W \cdot (1 - M)$$

MPSUB. Suspended substrate parameters are listed in the model description.

Autofill. The AutoFill parameter is a hidden input which allows you to specify that the entire interpolation table should be filled automatically at the current values. To instigate this filling process, this parameter should be set equal to 1. During normal operation, this parameter should be set equal to zero. You can access the hidden parameter by double-clicking on the schematic element.

Parameter Restrictions and Recommendations

1. MSBND90X implies that the ratio W/H1 lies within a **predefined** range of $0.25 \le W/H1 \le 16$ and M lies within a **predefined** range of $0 \le M \le 0.9125$ (H1 is substrate thickness). Outside of these ranges, this model extrapolates output parameters and issues a warning.
2. Bend is modeled as a two-port discontinuity. Reference planes corresponding to ports are located as shown in Topology .
3. Substrate thickness H1, substrate elevation above ground H2, nominal dielectric constant ErNom, dielectric tangent Tand, and relative conductor bulk resistance Rho are fixed parameters. Microwave Office provides pre-generated (pre-filled) tables for several typical values of MSBND90X fixed parameters. Changes to any fixed parameter may start the automatic filling process if Autofill is set to 1. However, changes to Tand and Rho do not affect results and do not instigate the tables filling.
4. You can change any fixed parameter to create the corresponding tables.

5. The dielectric constant Er of the substrate MPSUB is a statistical parameter. It means that models accounts for the relative deviation of Er from ErNom within 20%; larger deviation demands a new fill of model tables.
6. The frequency limits of this model are dynamic with respect to the dimensions of the bend as discontinuity, to the line width. This dynamic frequency limit is displayed via warning messages for the relative size, and dielectric in use. Note that this recommended frequency limit changes as a function of the largest width in the discontinuity for a given substrate definition. The frequency limit warns you that at least one of the transmission lines constructing the discontinuity is approaching the frequency limit where multiple modes can propagate. See “Upper Frequency Limitations” for more information.

Implementation Details

This model was developed under research performed at NI AWR Corporation. The full set of details of the implementation are considered proprietary in nature.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See “Cell Options Dialog Box: Layout Tab ” for Cell Options dialog box **Layout** tab details.

See “The Layout Process File (LPF)” for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

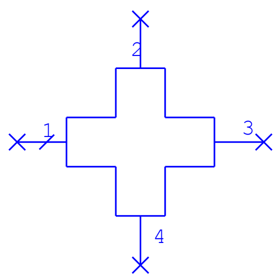
Recommendations for Use

To create new tables for the substrate dielectric constant different from those supplied with Microwave Office, you must set ErNom = Er = needed-value-of-dielectric-constant, set AutoFill = 1 and simulate. Please allow several hours (the actual time is dependent on your computer capabilities) for generating tables.

In exchange for speed increase, small errors resulting from the interpolation should be expected and the range of the input parameters is restricted.

Suspended Substrate Microstrip Cross - Junction Bend (EM Based): MSCROSSX

Symbol



Summary

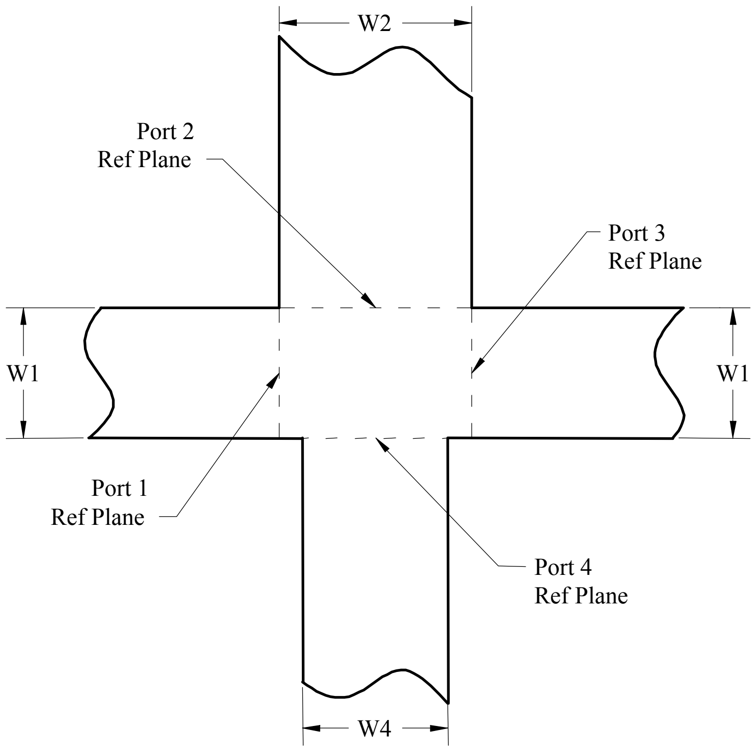
MSCROSSX models the cross junction of microstrip lines placed on the top surface of a suspended substrate (suspended substrate is a single layer substrate elevated above an infinite grounded plane and separated from it with an air layer).

This model is constructed as an X-model (table-based interpolation) using the results of EM analysis based on Method of Moments. For a more detailed discussion of the X-models see the " EM-based Models (X-models) " .

This model does not include effects of dielectric/conductor/radiation losses; it also implies that conductor thickness is zero.

MSCROSSX$ is the corresponding intelligent cell (iCell). An iCell model is identical to its non-iCell equivalent with the following exception: Certain dimension-related parameters are not explicitly user-specified; rather, they are automatically and dynamically determined by the dimensions of the attached elements. See "Intelligent Cells (iCells)" for a detailed discussion of how to use iCells, their advantages, and their limitations.

Topology



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	MX1
W1	Conductor Width @ Port 1	Length	W[a]
W2	Conductor Width @ Port 2	Length	W[a]
*W3	Conductor Width @ Port 3 (model always sets W3=W1; layout allows any W3)	Length	W[a]
W4	Conductor Width @ Port 4	Length	W[a]
MPSUB	Substrate definition	Text	MPSUB[bc]
*AutoFill	AutoFill dataBase if not equal to 0		0

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory.

[b]If only one MPSUB is present in the schematic, this substrate is automatically used. If multiple MPSUB substrate definitions are present, you must specify which to use.

[c]Modify only if the schematic contains multiple substrates.

Parameter Details

W1, W2, W4. Conductors widths are independent parameters.

W3. MSCROSSX does not use this parameter, it always sets W3=W1. However, layout cells read this parameter so it affects the layout.

MPSUB. Suspended substrate parameters are listed in the MPSUB model documentation.

Autofill. The AutoFill parameter is a hidden input which allows you to specify that the entire interpolation table should be filled automatically at the current values. To instigate this filling process, this parameter should be set equal to 1. During normal operation, this parameter should be set equal to zero. You can access the hidden parameter by double-clicking on the schematic element.

Parameter Restrictions and Recommendations

1. MSCROSSX implies that the ratio Wwidest/H1 lies within a predefined range of $0.25 \le W_{widest}/H1 \le 16$ where Wwidest is the maximum value of W1, W2, and W4 (H1 is substrate thickness); it also implies that the ratio Wothers/Wwidest lies within a predefined range of $0.2 \le W_{others}/W_{widest} \le 1$ where Wothers are all values of W1, W2, W4 excluding Wwidest. (Note that W3=W1). Outside of these ranges, this model extrapolates output parameters and issues a warning.
2. MSCROSSX is modeled as four-port discontinuity. Reference planes corresponding to ports are located as shown in the *Topology* section.
3. Substrate thickness H1, substrate elevation above ground H2, nominal dielectric constant ErNom, dielectric tangent Tand, and relative conductor bulk resistance Rho are fixed parameters. Microwave Office provides pre-generated (pre-filled) tables for several typical values of MSCROSSX fixed parameters. Changes to any fixed parameter may start the automatic filling process if Autofill is set to 1. However, changes to Tand and Rho do not affect results and do not instigate the tables filling.
4. You can change any fixed parameter to create corresponding tables.
5. The dielectric constant Er of substrate MPSUB is a statistical parameter. It means that models accounts for relative deviation of Er from ErNom within 20%; larger deviation demands a new fill of model tables.
6. The frequency limits of this model are dynamic with respect to the dimensions of the bend as discontinuity, to the line width. This dynamic frequency limit is displayed via warning messages for the relative size, and dielectric in use. Note that this recommended frequency limit changes as a function of the largest width in the discontinuity for a given substrate definition. The frequency limit warns you that at least one of the transmission lines constructing the discontinuity is approaching the frequency limit where multiple modes can propagate. See “Upper Frequency Limitations” for more information.

Implementation Details

This model was developed under research performed at NI AWR Corporation. The full set of details of the implementation are considered proprietary in nature.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See “Cell Options Dialog Box: Layout Tab ” for Cell Options dialog box **Layout** tab details.

See “The Layout Process File (LPF)” for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

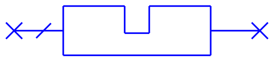
Recommendations for Use

To create new tables for the substrate dielectric constant different from those supplied with Microwave Office, you must set ErNom = Er = needed-value-of-dielectric-constant, set AutoFill = 1 and simulate. Please allow several hours (the actual time depends on your computer capabilities) for generating tables.

In exchange for a speed increase, small errors resulting from the interpolation should be expected and the range of the input parameters is restricted.

Microstrip Transverse Slit (Closed Form): MSLIT

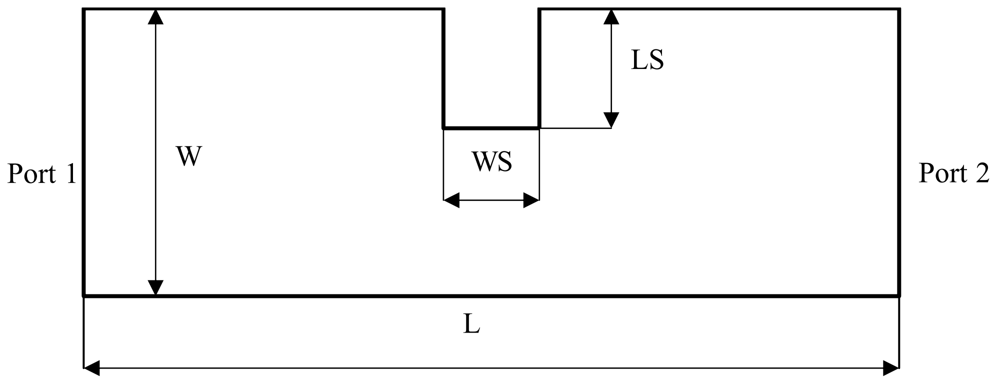
Symbol



Summary

MSLIT models a segment of microstrip line with a transverse slit cut in the middle of the segment. This model accounts for additional resistive loss caused by the slit.

Topology



Parameters

Name	Description	Unit Type	Default
ID	Name	Text	TL1
W	Conductor width	Length	W[a]
L	Conductor length	Length	L[b]
LS	Slit length	Length	L[b]
WS	Slit width	Length	W[a]
MSUB	Substrate definition	Text	MSUB#[c]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See the Microwave Office Layout Guide for details.

[b]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See the Microwave Office Layout Guide for details.

[c]Modify only if the schematic contains multiple substrates. See the "Using Elements With Model Blocks" for details.

Parameter Details

L. The slit is cut at equal distance L/2 from conductor ends.

Parameter Restrictions and Recommendations

1. Slit length LS cannot exceed 0.9W

2. Slit width WS cannot exceed 0.9L

Implementation Details

MSLIT accounts for slit excess inductance [1]. It also accounts for frequency-dependent excess resistance introduced by the slit (a modified technique based on [2]).

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See "Cell Options Dialog Box: Layout Tab " for Cell Options dialog box **Layout** tab details.

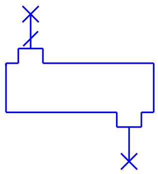
See "The Layout Process File (LPF)" for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

Recommendations for Use

The selection of the L parameter requires attention. The recommended value of L is in the range of 10..20WS.

Microstrip Pair of Open Stubs: MSOP

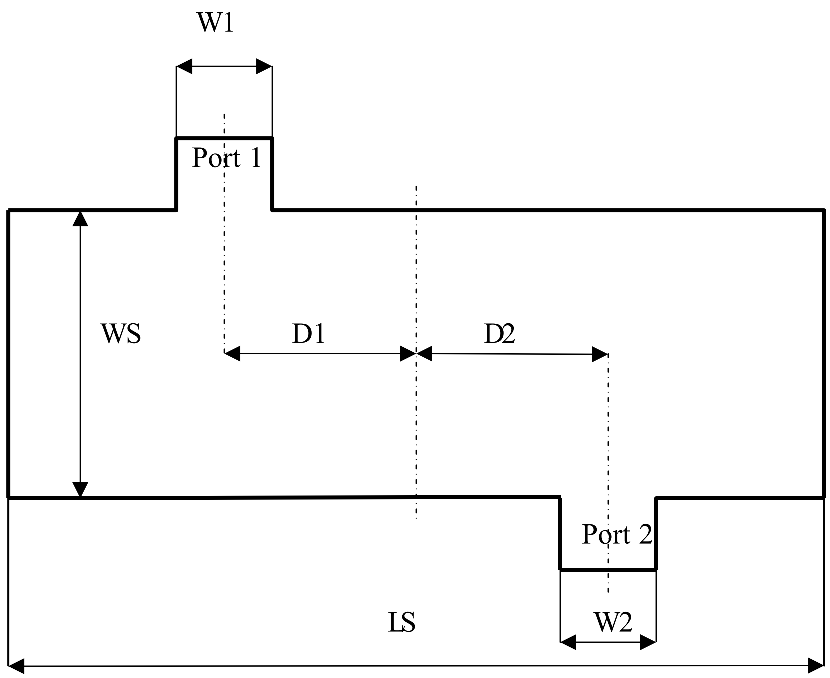
Symbol



Summary

MSOP models two microstrip open stubs attached back-to-back with two ports, allowing connection of two microstrip lines.

Topology



Parameters

Name	Description	Unit Type	Default
ID	Name	Text	TL1
WS	Stub width	Length	W[a]
LS	Stub total length	Length	L[b]
D1	Displacement of port #1 from stub center	Length	L[b]
D2	Displacement of port #2 from stub center	Length	L[b]

Name	Description	Unit Type	Default
W1	Width of feeding line at port #1	Length	W[a]
W2	Width of feeding line at port #2	Length	W[a]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See the Microwave Office Layout Guide for details.

[b]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See the Microwave Office Layout Guide for details.

Parameter Details

L. This is a total length of two equal length open stubs attached back-to-back.

Parameter Restrictions and Recommendations

1. Port displacement D1 and D2 can be positive and negative. For port 1, positive/negative displacement means that port 1 is on the left/right side of the stub center; for port 2, positive/negative displacement means that port 2 is on the right/left side of the stub center.

Implementation Details

MSOP is implemented as a network of microstrip segments connected via microstrip tees.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See “Cell Options Dialog Box: Layout Tab ” for Cell Options dialog box **Layout** tab details.

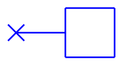
See “The Layout Process File (LPF)” for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

Recommendations for Use

This model accounts for metal and dielectric losses and can be used as a component of filters and phase shifters.

Microstrip Open Ended Effect (EM Based): MSOPENX

Symbol



Summary

MOPENX models an effect of the open end of a microstrip line placed on the top surface of a suspended substrate (suspended substrate is a single layer substrate elevated above the infinite grounded plane). This model should be used in conjunction with any suitable microstrip suspended line model.

This model is constructed as an X-model (table-based interpolation) using the results of EM analysis based on Method of Moments. For a more detailed discussion of the X-models see “ EM-based Models (X-models) ”.

This model does not include the effects of dielectric/conductor/radiation losses; it also implies that conductor thickness is zero.

MOPENX$ is the corresponding intelligent cell (iCell). An iCell model is identical to its non-iCell equivalent with the following exception: Certain dimension-related parameters are not explicitly user-specified; rather, they are automatically and dynamically determined by the dimensions of the attached elements. See “Intelligent Cells (iCells)” for a detailed discussion of how to use iCells, their advantages, and their limitations.

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	MO#[a]
W	Conductor width	Length	W[b]
MPSUB	Substrate definition	Text	See [c] [d]
*AutoFill	AutoFill dataBase if not equal to 0		0

[a]# is automatically incremented as additional MO element ID's are added to the schematic.

[b]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory.

[c]If only one MPSUB is present in the schematic, this substrate is automatically used. If multiple MPSUB substrate definitions are present, you must specify which to use.

[d] Modify only if the schematic contains multiple substrates.

** indicates a secondary parameter*

Parameter Details

W. Conductor width is an independent parameter.

MPSUB. Suspended substrate parameters are listed in the MPSUB model documentation.

Autofill. The AutoFill parameter is a hidden input which allows you to specify that the entire interpolation table should be filled automatically at the current values. To instigate this filling process, this parameter should be set equal to 1. During normal operation, this parameter should be set equal to zero. You can access the hidden parameter by double-clicking on the schematic element.

Parameter Restrictions and Recommendations

1. MSOPENX implies that the ratio W/H1 lies within a predefined range of $0.05 \leq W/H1 \leq 16$, where H1 is substrate thickness. Outside of these ranges, this model extrapolates output parameters and issues a warning.
2. An open end is modeled as a one-port discontinuity. The reference plane corresponding to the port is located at the edge of the open end of the line attached to this port; MSOPENX adds a small stretch of suspended microstrip line.
3. Substrate thickness H1, substrate elevation above ground H2, nominal dielectric constant ErNom, dielectric tangent Tand, and relative conductor bulk resistance Rho are fixed parameters. Microwave Office provides pre-generated (pre-filled) tables for several typical values of MSOPENX fixed parameters. Changes to any fixed parameter may start the automatic filling process if Autofill is set to 1. However, changes to Tand and Rho do not affect results and do not instigate the tables filling.
4. You can change any fixed parameter to create corresponding tables.
5. The dielectric constant Er of substrate MPSUB is a statistical parameter. This means that models account for a relative deviation of Er from ErNom within 20%; larger deviation demands a new fill of model tables.
6. The frequency limits of this model are dynamic with respect to the dimensions of the bend as discontinuity, actually, to the line width. This dynamic frequency limit is displayed via warning messages for the relative size, and dielectric in use. Note that this recommended frequency limit changes as a function of the largest width in the discontinuity for a given substrate definition. The frequency limit warns you that at least one of the transmission lines constructing the discontinuity is approaching the frequency limit where multiple modes can propagate. See "Upper Frequency Limitations" for more information.

Implementation Details

This model was developed under research performed at NI AWR Corporation. The full set of details of the implementation are considered proprietary in nature.

Layout

This element does not have an assigned layout cell nor does it use line types to determine its layout. The element models an open end effect.

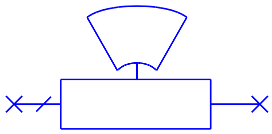
Recommendations for Use

To create new tables for the substrate dielectric constant different from those supplied with Microwave Office, you must set ErNom = Er = needed-value-of-dielectric-constant, set AutoFill = 1 and simulate. Please allow several hours (the actual time depends on your computer capabilities) for generating tables.

In exchange for a speed increase, small errors resulting from the interpolation should be expected and the range of the input parameters is restricted.

(Obsolete) Microstrip Radial Stub Shunt (Closed Form): MSRSTUB

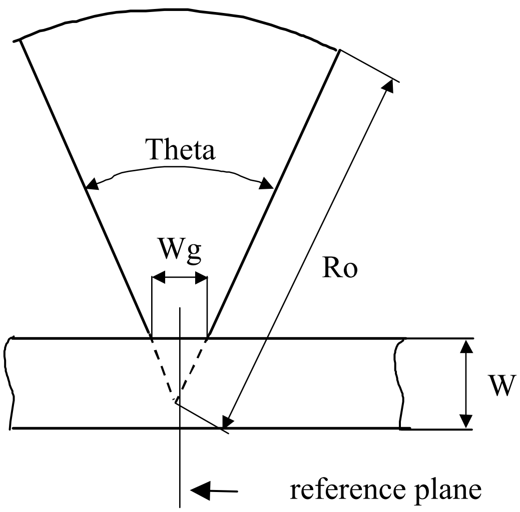
Symbol



Summary

This element is OBSOLETE and is replaced by the Microstrip Radial Stub Series (Closed Form) (MSRSTUB2) element.

Topology



Parameters

Name	Description	Unit Type	Default
ID	Name	Text	ST[a]
Ro	Outer radius of the stub	Length	L[b]
W	Conductor width	Length	W[b]
Wg	Width of crossing of stub and microstrip line	Length	W[b]
Theta	Angle of the stub	Angle	90 Deg
MSUB	Substrate definition	Text	MSUB[c]

[a]# is automatically incremented as additional ST element ID's are added to the schematic.

[b]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory.

[c]If only one MSUB is present in the schematic, this substrate is automatically used. If multiple MSUB substrate definitions are present, the user must specify.

Implementation Details

This circuit component models a single shunt radial stub on a single layer substrate (series stubs should be modeled with MRSTUB). The model is a two port through element. The parameters W (width of the input microstrip), Wg (width of crossing of stub and microstrip line), and Ro (outer radius) are dimensions entered in the default length units.Theta is a radial stub angle entered in degrees. The parameter MSUB specifies the substrate element, which defines additional cross sectional parameters of the transmission line and substrate characteristics. If blank, a default is used. The model is valid up to its second open-circuit resonance and for $9 \leq$ Theta ≤ 180.

The component accounts for losses in metal and in substrate dielectric. Dispersion is not included.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See “Cell Options Dialog Box: Layout Tab ” for Cell Options dialog box **Layout** tab details.

See “The Layout Process File (LPF)” for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

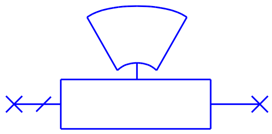
References

[1] S.L. March, "Analyzing lossy radial-line stubs," IEEE Trans. Microwave Theory Tech., vol. MTT-33, March 1985, pp. 269-271

[2]F.Giannini, R.Ruggieri, and J.Vrba, "Shunt-connected microstrip radial stubs", IEEE Trans. Microwave Theory Tech., vol. MTT-34, March 1986, pp. 363-366

Microstrip Radial Stub Series (Closed Form): MSRSTUB2

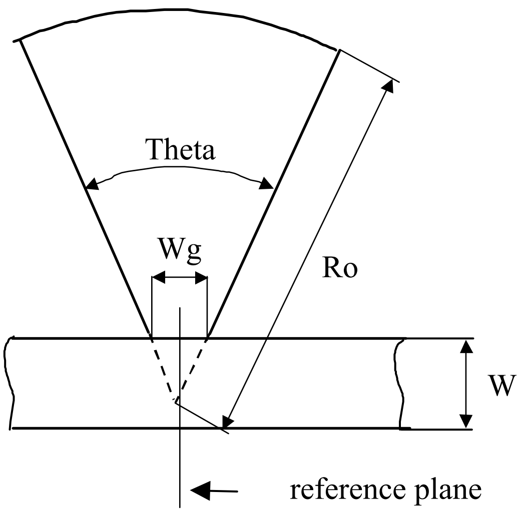
Symbol



Summary

MSRSTUB2 models a single microstrip radial stub that shunts microstrip line. This model is a two-port through element.

Topology



Parameters

Name	Description	Unit Type	Default
ID	Name	Text	ST[a]
Ro	Outer radius of the stub	Length	L[b]
W	Conductor width	Length	W[b]
Wg	Width of crossing of stub and microstrip line	Length	W[b]
Theta	Angle of the stub	Angle	90 Deg
MSUB	Substrate definition	Text	MSUB[c]

[a]# is automatically incremented as additional ST element ID's are added to the schematic.

[b]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory.

[c]If only one MSUB is present in the schematic, this substrate is automatically used. If multiple MSUB substrate definitions are present, you must specify which to use.

Parameter Restrictions and Recommendations

1. Theta must be within the range $9 \leq \text{Theta} \leq 160$.

2. This model is feasible if

$$\left(R_O - \frac{Wg}{2sin(Theta/2)}\right) / R_O > 0.001$$

.

Implementation Details

Model implementation is based on cascading multiple segments of lossy microstrip transmission lines described in [1] [10–270]. It accounts for losses in metal and in substrate dielectrics. Radiation loss is not included.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See "Cell Options Dialog Box: Layout Tab " for Cell Options dialog box **Layout** tab details.

See "The Layout Process File (LPF)" for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

Recommendations for Use

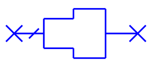
MSRSTUB2 replaces the MSRSTUB element, which is obsolete.

References

[1] B.C.Wadell, Transmission Line Design Book, Artech House, Boston-London, 1991, pp.306-307

Suspended Substrate Microstrip Step In Width with Offset (EM Based): MSSTEPX

Symbol



Summary

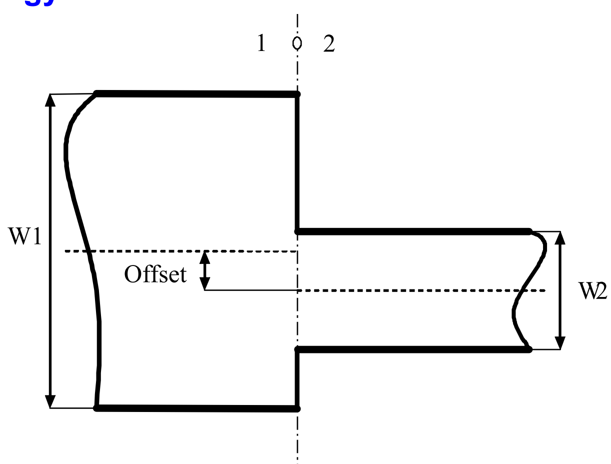
MSSTEPX models a step in width at the asymmetric junction of two microstrip lines placed on the top surface of a suspended substrate (suspended substrate is a single layer substrate elevated above the infinite grounded plane). This model allows a relative offset of lines.

This model is constructed as an X-model (table-based interpolation) using the results of EM analysis based on Method of Moments. For a more detailed discussion of the X-models see “ EM-based Models (X-models) ”.

This model does not include effects of dielectric/conductor/radiation losses; it also implies that conductor thickness is zero.

MSSTEPX$ is the corresponding intelligent cell (iCell). An iCell model is identical to its non-iCell equivalent with the following exception: Certain dimension-related parameters are not explicitly user-specified; rather, they are automatically and dynamically determined by the dimensions of the attached elements. See “Intelligent Cells (iCells)” for a detailed discussion of how to use iCells, their advantages, and their limitations.

Topology



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	MS1
W1	Conductor width @ node 1	Length	W[a]
W2	Conductor width @ node 2	Length	W[a]
Offset	Center-line offset dimension	Length	W[a]
MPSUB	Substrate definition	Text	MPSUB[bc]

Name	Description	Unit Type	Default
*AutoFill	Autofill database if not equal to 0		0

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory.

[b]If only one MPSUB is present in the schematic, this substrate is automatically used. If multiple MPSUB substrate definitions are present, you must specify which to use.

[c]Modify only if the schematic contains multiple substrates.

** indicates a secondary parameter*

Parameter Details

W1, W2. Conductors widths are independent parameters.

Offset. Offset is an independent parameter.

MPSUB. Suspended substrate parameters are listed in the MPSUB model documentation.

Autofill. The AutoFill parameter is a hidden input which allows you to specify that the entire interpolation table should be filled automatically at the current values. To instigate this filling process, this parameter should be set equal to 1. During normal operation, this parameter should be set equal to zero. You can access the hidden parameter by double-clicking on the schematic element.

Parameter Restrictions and Recommendations

1. MSTEPX implies that the ratio Wmax/H1 lies within a predefined range of $0.125 \le W_{max}/H1 \le 16$, the ratio Wmin/Wmax lies within a predefined range of $0.05 \le W_{min}/W_{max} \le 1$, and Offset/Wmax lies within a predefined range of $0 \le Offset/W_{max} \le 0.5$, where H1 is substrate thickness, Wmax is maximum value of W1, W2; and Wmin is the minimum value of W1, W2. Outside of these ranges, this model extrapolates output parameters and issues a warning.
2. MSSTEPX is modeled as a two-port discontinuity. Reference planes corresponding to ports are located as shown in the *Topology* section.
3. Substrate thickness H1, substrate elevation above ground H2, nominal dielectric constant ErNom, dielectric tangent Tand, and relative conductor bulk resistance Rho are fixed parameters. Microwave Office provides pre-generated (pre-filled) tables for a several typical values of MSSTEPX fixed parameters. Changes to any fixed parameter may start the automatic filling process if Autofill is set to 1. However, changes to Tand and Rho do not affect results and do not instigate the tables filling.
4. You can change any fixed parameter to create corresponding tables.
5. The dielectric constant Er of substrate MPSUB is a statistical parameter. It means that models account for relative deviation of Er from ErNom within 20%; larger deviation demands a new fill of model tables.
6. The frequency limits of this model are dynamic with respect to the dimensions of the bend as discontinuity, to the line width. This dynamic frequency limit is displayed via warning messages for the relative size, and dielectric in use. Note that the recommended frequency limit changes as a function of the largest width in the discontinuity for a given substrate definition. The frequency limit warns you that at least one of the transmission lines constructing the discontinuity is approaching the frequency limit where multiple modes can propagate. See “Upper Frequency Limitations” for more information.

Implementation Details

This model was developed under research performed at NI AWR Corporation. The full set of details of the implementation are considered proprietary in nature.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See "Cell Options Dialog Box: Layout Tab " for Cell Options dialog box **Layout** tab details.

See "The Layout Process File (LPF)" for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

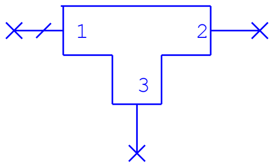
Recommendations for Use

To create new tables for the substrate dielectric constant different from those supplied with Microwave Office, you must set ErNom = Er = needed-value-of-dielectric-constant, set AutoFill = 1 and simulate. Please allow several hours (the actual time depends on your computer capabilities) for generating tables.

In exchange for speed increase, small errors resulting from the interpolation should be expected and the range of the input parameters is restricted.

Suspended Substrate Microstrip Tee - Junction (EM Based): MSTEEX

Symbol



Summary

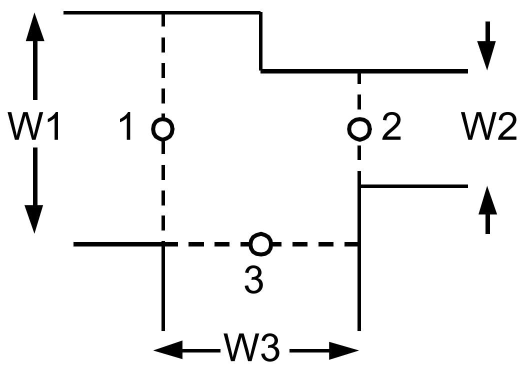
MSTEEX models a T-junction of three microstrip lines placed on the top surface of a suspended substrate (suspended substrate is a single layer substrate elevated above the infinite grounded plane).

This model is constructed as an X-model (table-based interpolation) using the results of EM analysis based on Method of Moments. For a more detailed discussion of the X-models see " EM-based Models (X-models) ".

This model does not include effects of dielectric/conductor/radiation losses; it also implies that conductor thickness is zero.

MSTEEX$ is the corresponding intelligent cell (iCell). An iCell model is identical to its non-iCell equivalent with the following exception: Certain dimension-related parameters are not explicitly user-specified; rather, they are automatically and dynamically determined by the dimensions of the attached elements. See "Intelligent Cells (iCells)" for a detailed discussion of how to use iCells, their advantages, and their limitations.

Topology



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	MT1
W1	Conductor width @ node 1	Length	W[a]
W2	Conductor width @ node 2	Length	W[a]
W3	Conductor width @ node 3	Length	W[a]
MPSUB	Substrate definition	Text	See[bc]

Name	Description	Unit Type	Default
*AutoFill	Autofill database if not equal to 0		0

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory.

[b]If only one MPSUB is present in the schematic, this substrate is automatically used. If multiple MPSUB substrate definitions are present, you must specify which to use.

[c]Modify only if the schematic contains multiple substrates.

** indicates a secondary parameter*

Parameter Details

W1, W2, W3. Conductors widths are independent parameters.

MPSUB. Suspended substrate parameters are listed in the MPSUB model documentation.

Autofill. A hidden input which allows you to specify that the entire interpolation table should be filled automatically at the current values. To instigate this filling process, this parameter should be set equal to 1. During normal operation, this parameter should be set equal to zero. You can access the hidden parameter by double-clicking on the schematic element.

Parameter Restrictions and Recommendations

1. MSTEEX implies that the ratio Wwidest/H1 lies within a predefined range of $0.25 \le W_{widest}/H1 \le 16$ where Wwidest is the maximum value of W1, W2, and W3 (H1 is substrate thickness); it also implies that the ratio Wothers/Wwidest lies within a predefined range of $0.2 \le W_{others}/W_{widest}/H1 \le 1$ where Wothers are all values of W1, W2, W3 excluding Wwidest. Outside of these ranges, this model extrapolates output parameters and issues a warning.
2. Tee is modeled as three-port discontinuity. Reference planes corresponding to ports are located as shown in the *Topology* section.
3. Substrate thickness H1, substrate elevation above ground H2, nominal dielectric constant ErNom, dielectric tangent Tand, and relative conductor bulk resistance Rho are fixed parameters. Microwave Office provides pre-generated (pre-filled) tables for several typical values of MSTEEX fixed parameters. Changes to any fixed parameter may start the automatic filling process if Autofill is set to 1. However, changes to Tand and Rho do not affect results and do not instigate the tables filling.
4. You can change any fixed parameter to create corresponding tables.
5. The dielectric constant Er of substrate MPSUB is a statistical parameter. It means that models account for relative deviation of Er from ErNom within 20%; larger deviation demands a new fill of model tables.
6. The frequency limits of this model are dynamic with respect to the dimensions of the bend as discontinuity, to the line width. This dynamic frequency limit is displayed via warning messages for the relative size and dielectric in use. Note that this recommended frequency limit changes as a function of the largest width in the discontinuity for a given substrate definition. The frequency limit warns you that at least one of the transmission lines constructing the discontinuity is approaching the frequency limit where multiple modes can propagate. See "Upper Frequency Limitations" for more information.

Implementation Details

This model was developed under research performed at NI AWR Corporation. The full set of details of the implementation are considered proprietary in nature.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See "Cell Options Dialog Box: Layout Tab " for Cell Options dialog box **Layout** tab details.

See "The Layout Process File (LPF)" for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

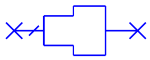
Recommendations for Use

To create new tables for the substrate dielectric constant different from those supplied with Microwave Office, you must set ErNom = Er = needed-value-of-dielectric-constant, set AutoFill = 1 and simulate. Please allow several hours (the actual time depends on your computer capabilities) for generating tables.

In exchange for a speed increase, small errors resulting from the interpolation should be expected and the range of the input parameters is restricted.

Microstrip Step in Width (Closed Form): MSTEP

Symbol



Parameters

MSTEP$ is a Microstrip iCell with no parameters. See “Intelligent Cells (iCells)” for more information.

Name	Description	Unit Type	Default
ID	Element ID	Text	TL1
W1	Conductor width @ node 1	Length	W[a]
W2	Conductor width @ node 2	Length	W[a]
MSUB	Substrate definition	Text	MSUB#[b]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See the Microwave Office Layout Guide for details.

[b]Modify only if schematic contains multiple substrates. See “Using Elements With Model Blocks” for details.

Implementation Details

MSUB indicates the name of the MSUB element to use for substrate characteristics. If blank, a default is used. The step is the reference plane. W1 and W2 are the widths of the connecting microstrips.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See “Cell Options Dialog Box: Layout Tab ” for Cell Options dialog box **Layout** tab details.

See “The Layout Process File (LPF)” for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

Recommendations for Use

Discontinuity models function most accurately when attached to lines that match their corresponding edges. Directly connecting discontinuity models to one another reduces their accuracy.

Reference

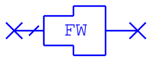
[1] G. Kompa, "Design of Stepped Microstrip Components," The Radio and Electronic Engineer, Vol. 48, p. 53, 1978.

[2] B. C. Wadell, Transmission-Line Design Handbook, Artech House, Norwood, MA, 1991.

[3] R. K. Hoffman, Handbook of Microwave Integrated Circuits, Artech House, Norwood, MA, 1987.

(Obsolete) Microstrip Step in Width (Mode Matching): MSTEP2

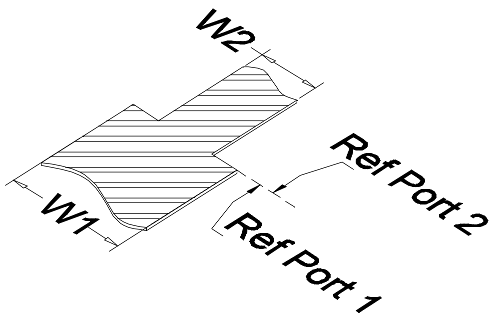
Symbol



Summary

This element is OBSOLETE and is replaced by the Microstrip Step in Width (Closed Form) (MSTEP) or Microstrip Step In Width Offset (EM Based) (MSTEPX) element.

Topology



Parameters

MSTEP2$ is a Microstrip iCell and has no parameters. See "Intelligent Cells (iCells)" for more information.

Name	Description	Unit Type	Default
ID	Element ID	Text	SL1
W1	Width of line @ port 1	Length	W[a]
W2	Width of line @ port 2	Length	W[a]
Modes	# of higher order modes		4
MSUB	Microstrip substrate definition	Text	MSUB#[b]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See Microwave Office Layout Guide for details.

[b]Modify only if schematic contains multiple substrates. See the "Using Elements With Model Blocks" for details.

Implementation Details

The model assumes a Quasi-TEM mode of incident and reflected propagating waves and also calculates the stored energy in higher order evanescent waves. The model is based on the Full Wave Solution of an equivalent parallel plate wave guide impedance step and incorporates the frequency-dependent nature of the parasitic inductance and capacitance and further incorporates the effects of dispersion. The model does not incorporate the resistive effects due to radiation or dielectric and conductive losses. The parameters W1 and W2 (Strip Widths) are lengths entered in the default length units. The parameter MSUB specifies the Microstrip Substrate element, which defines additional cross sectional parameters of the transmission line. The Modes parameter (Number of Higher Order Modes) indicates the number of modes to sum to determine the amount of stored energy in the junction. As the number of modes increase, the accuracy and calculation time increases. Further, as the aspect ratio (W1/W2) deviates away from unity (1) the number of modes required for a given accuracy increases.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See "Cell Options Dialog Box: Layout Tab " for Cell Options dialog box **Layout** tab details.

See "The Layout Process File (LPF)" for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

Restrictions

$0.05 \le W_{1\ or\ 2}/H \le 20$ Recommended	$\varepsilon_r \le 16.0$ Recommended
$T/W_{min} \le 0.5$ Recommended	$1 \le \varepsilon_r$ Required
$T/H \le 0.5$ Recommended	$0.05 \le W_1/W_2 \le 20$ Recommended
$1 \le$ Modes ≤ 20 Recommended	

References

[1] Menzel, W. and Wolff, I. "A Method of Calculating the Frequency Dependent Properties of Microstrip Discontinuities" IEEE Trans. On MTT Vol.25 pp.,107-112 February 1977

[2] Chu, T.S., Itoh, T. and Shih, Y.C. "Comparative Study of Mode-Matching Formulations for Microstrip Discontinuity Problems" IEEE Trans. on MTT Vol-33 No.10 Oct. 1985

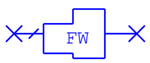
[3] Chu, T.S. and Itoh, T. "Analysis of Microstrip Step Discontinuity by the Modified Residue Calculus Technique" IEEE Trans. on MTT Vol-33 No.10 Oct. 1985

[4] Gupta, K.C. "Microstrip Lines and Slotlines" Artech 1996 pp.190

[5] Itoh, Tatsuo, "Numerical Techniques for Microwave and Millimeter-Wave Passive Structures" John Wiley & Sons Inc. 1989 p. 466-472

(Obsolete) Offset Microstrip Step Width (Mode Matching): MSTEPO

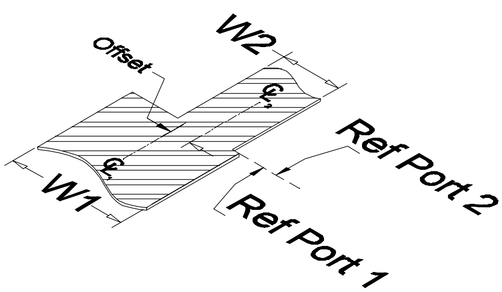
Symbol



Summary

This element is OBSOLETE and is replaced by the Microstrip Step In Width Offset (EM Based) (MSTEPX) element.

Topology



Parameters

MSTEPO$ is a Microstrip iCell and has no parameters. See "Intelligent Cells (iCells)" for more information.

Name	Description	Unit Type	Default
ID	Element ID	Text	SL1
W1	Conductor width @ node 1	Length	W[a]
W2	Conductor width @ node 2	Length	W[a]
Offset	Centerline offset dimension	Length	0 um
Modes	# of Higher order modes		8
MSUB	Microstrip substrate definition	Text	MSUB#[b]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See Microwave Office Layout Guide for details.

[b]Modify only if schematic contains multiple substrates. See the "Using Elements With Model Blocks" for details.

Implementation Details

The model assumes a Quasi-TEM mode of incident and reflected propagating waves and also calculates the stored energy in higher order evanescent waves. The model is based on the Full Wave Solution of an equivalent parallel plate wave guide impedance step and incorporates the effects of dispersion and the frequency-dependent nature of the parasitic inductance and capacitance. The model does not incorporate the resistive effects due to radiation or dielectric and conductive losses. The parameters W1, W2 (Strip Widths) and Offset are lengths entered in the default length units. The parameter MSUB specifies the Microstrip Substrate element, which defines additional cross sectional parameters of the transmission line. The Modes parameter (Number of Higher Order Odd and Even order Modes) indicates the number of modes to sum to determine the amount of stored energy in the junction. The amount of energy stored in the higher modes is increased due to the ability of the asymmetric step to excite odd ordered modes. As the number of modes increases, the accuracy and calculation time increases. Further, as the aspect ratio (W1/W2) deviates away from unity (1) the number of modes required for a given accuracy increases.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See "Cell Options Dialog Box: Layout Tab " for Cell Options dialog box **Layout** tab details.

See "The Layout Process File (LPF)" for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

Restrictions

$0.05 \leq W_{1\ or\ 2}/H \leq 20$ Recommended	$\varepsilon_r \leq 16.0$ Recommended
$T/W_{min} \leq 0.5$ Recommended	Offset $\leq$ |W1 - W2|/2
$T/H \leq 0.5$ Recommended	$0.05 \leq W1/W2 \leq 20$ Recommended
$1 \leq$ Modes ≤ 30 Recommended	

References

[1] Menzel, W. and Wolff, I. "A Method of Calculating the Frequency Dependent Properties of Microstrip Discontinuities" IEEE Trans. On MTT Vol.25 pp.,107-112 February 1977

[2] Chu, T.S., Itoh, T. and Shih, Y.C. "Comparative Study of Mode-Matching Formulations for Microstrip Discontinuity Problems" IEEE Trans. on MTT Vol-33 No.10 Oct. 1985

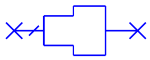
[3] Chu, T.S. and Itoh, T. "Analysis of Microstrip Step Discontinuity by the Modified Residue Calculus Technique" IEEE Trans. on MTT Vol-33 No.10 Oct. 1985

[4] Gupta, K.C. "Microstrip Lines and Slotlines" Artech 1996 pp.190

[5] Itoh, Tatsuo, "Numerical Techniques for Microwave and Millimeter-Wave Passive Structures" John Wiley & Sons Inc. 1989 p. 466-472

Microstrip Step In Width Offset (EM Based): MSTEPX

Symbol



Summary

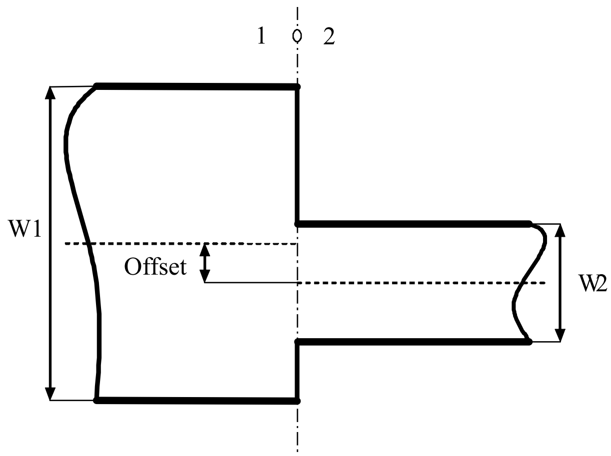
MSTEPX models a step in width at the asymmetric junction of two microstrip lines. Model allows relative offset of lines.

This model is constructed as an X-model (Table Based Interpolation) using the results of a EM analysis based on Method of Moments. For a more detailed discussion of the X-models see the " EM-based Models (X-models) ".

This model does not include effects of dielectric/conductor/radiation losses; it also implies that conductor thickness is zero.

MSTEPX$ is the corresponding intelligent cell (iCell). An iCell model is identical to its non-iCell equivalent with the following exception: Certain dimension-related parameters are not explicitly specified by the user; rather, they are automatically and dynamically determined by the dimensions of the attached elements. See "Intelligent Cells (iCells)" for a detailed discussion of how to use iCells, their advantages, and their limitations.

Topology



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	MS1
W1	Conductor width @ node 1	Length	W[a]
W2	Conductor width @ node 2	Length	W[a]
Offset	Centerline offset dimension	Length	W[a]
MSUB	Substrate definition	Text	See[bc]
*AutoFill	AutoFill database if not equal to 0		0

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory.

[b]If only one MSUB is present in the schematic, this substrate is automatically used. If multiple MSUB substrate definitions are present, you must specify which to use.
[c]Modify only if the schematic contains multiple substrates.

** indicates a secondary parameter*

Parameter Restrictions and Recommendations

$0.05 \leq W1\&W2 \leq 8.0$ Recommended	$Offset/W_{widest} \leq 0.5$ Recommended
$0.05 \leq W1/W2 \leq 20$ Recommended	$\varepsilon_{r\ nominal} \leq 16$ Recommended
$0 \leq Frequency \leq F_{max}$	$1 \leq \varepsilon_{r\ nominal}$ Required
$(100*\lvert\varepsilon_r-\varepsilon_{r\ nominal}\rvert)/\varepsilon_{r\ nominal} \leq 10\%$ Recommended	$(100*\lvert\varepsilon_r-\varepsilon_{r\ nominal}\rvert)/\varepsilon_{r\ nominal} \leq 20\%$ Required

F_{max}: The frequency limits of this model are dynamic with respect to the dimensions of the discontinuity. This dynamic frequency limit is displayed to the user via warning messages for the relative size, and dielectric in use. Importantly, this recommended frequency limit will change as a function of the largest width in the discontinuity for a given substrate definition. The frequency limit warns the user that at least one of the transmission lines constructing the discontinuity is approaching the frequency limit where multiple modes can propagate. See the section marked "Upper Frequency Limitations" in the General Discussion of the X-models.

Implementation Details

This model was developed under research performed at NI AWR Corporation. The details of the implementation are considered proprietary in nature.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See "Cell Options Dialog Box: Layout Tab " for Cell Options dialog box **Layout** tab details.

See "The Layout Process File (LPF)" for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

Recommendations for Use

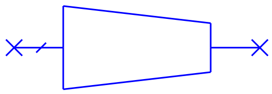
To create new tables for substrate dielectric constant different from those supplied with Microwave Office user must set ErNom = Er = needed-value-of-dielectric-constant, set AutoFill = 1 and simulate. Allow several hours (actual time depends on the computer that runs Microwave Office) for generating tables.

In exchange for speed increase, small errors resulting from the interpolation should be expected and the range of the input parameters have been restricted.

Discontinuity models function most accurately when attached to lines that match their corresponding edges. Directly connecting discontinuity models to one another reduces their accuracy.

Tapered Microstrip Line (Closed Form): MTAPER

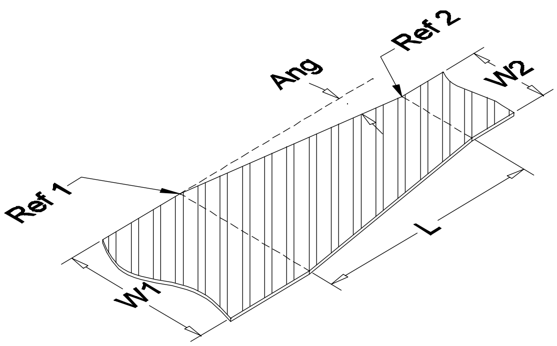
Symbol



Summary

MTAPER models a segment of inhomogeneous microstrip line. The width of microstrip tapers smoothly between ports. This model allows linear and exponential width tapers.

Topology



Parameters

MTAPER$ is a Microstrip iCell and has no W1 or W2 parameters. See "Intelligent Cells (iCells)" for more information.

Name	Description	Unit Type	Default
ID	Name		MT1
W1	Conductor width @ node 1	Length	W[a]
W2	Conductor Width @ node 2	Length	W[a]
L	Length of taper	Length	L[a]
MSUB	Substrate definition	Text	MSUB#[b]
Taper	Linear/Exponential switch		Linear
Method	Default/Interpolation switch		Default

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See Microwave Office Layout Guide for details.

[b]Modify only if schematic contains multiple substrates. See the "Using Elements With Model Blocks" for details.

Parameter Details

Taper. This parameter provides two options. The Linear option provides linear taper, that is, an outline of the strip is a trapezoid. The Exponential option provides exponential taper, that is, an outline of the strip is trapezoid-like with longitudinal sides made with concave exponents.

Method. This parameter provides two options. When the Default option is selected, the model breaks the tapered conductor into multiple constant width sections and calculates transmission line characteristics for each section. When the Interpolation option is selected, the model breaks the tapered conductor into a small number of constant width sections and creates lookup tables for transmission line parameters using width as an argument. Transmission line characteristics for multiple sections are evaluated by means of interpolation in these tables. The Interpolation option speeds up MTAPER and may be beneficial for tuning and optimization. Note that interpolation may slightly degrade accuracy, so user discretion is advised.

Also note that selecting Method=Interpolation has no effect on time-domain simulations.

Parameter Restrictions and Recommendations

$0.05 \leq W_{1\ or\ 2}/H \leq 20$ Recommended	$\varepsilon_r \leq 16.0$ Recommended
$T/W_{min} \leq 0.5$ Recommended	$1 \leq \varepsilon_r$ Required
$T/H \leq 0.5$ Recommended	Tand ≥ 0 Required
Rho ≤ 100 Recommended	Ang ≤ 45 degs or $L \geq \|W1 - W2\|/2$

Implementation Details

This model is constructed out of a cascaded series of constant width Microstrip transmission lines. The number of sections used is frequency-dependent and is constant as a function of the length divided by the guided wavelength. This model assumes a Quasi-TEM mode of propagation and incorporates the effects of dielectric and conductive losses and dispersion.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See "Cell Options Dialog Box: Layout Tab " for Cell Options dialog box **Layout** tab details.

See "The Layout Process File (LPF)" for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

Recommendations for Use

The layout cell has a local parameter NUM_EXP_POINT that is valid if Taper=Exponential. This parameter specifies the number of points used for drawing a curved outline of exponential taper. The default value of NUM_EXP_POINT is 50.

References

This is an NI AWR Developed Model. The references for the straight section Microstrip line follow:

[1] E. Hammerstad and O. Jensen, IEEE MTT-S International Microwave Symposium Digest, p.407, 1980.

[2] I. J. Bahl and D. K. Trivedi, "A Designer's Guide to Microstrip Line," Microwaves, p. 174, May, 1977.

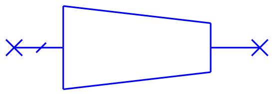
[3] S. March, "Microstrip Packaging: Watch the Last Step," Microwaves, p. 83, Dec., 1981.

[4] R. Pucel, D. J. Masse, and C. P. Hartwing, IEEE Trans. Microwave Theory Tech., Vol. MTT 16, p. 342, 1968.

[5] G. Wells and P. Pramanick, Int. J. Microwave and mmWave Computer-Aided Design, Vol. 5, p. 287, 1995.

Tapered Microstrip Line Synthesized to Match (Closed Form): MTAPER2

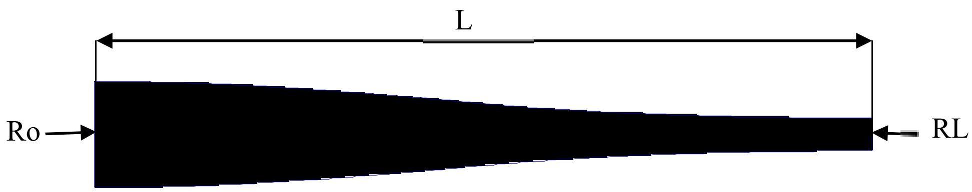
Symbol



Summary

MTAPER2 provides synthesis of three types of matching lines with tapered distribution of characteristic impedance: Exponential, Triangular, and Klopfenstein. The width of microstrip tapers smoothly between ports. Synthesis of optimal taper shape is initiated by setting parameter L (taper length) to zero.

Topology



Parameters

Name	Description	Unit Type	Default
ID	Name		MT1
Fp	Lower passband edge	Frequency	1 GHz
Ro	Input impedance @ node 1	Resistance	50 Ohm
RL	Load impedance @ node 2	Resistance	100 Ohm
L	Line length	Length	0 (if zero replaced by synthesized value)
Rmax	Absolute value of maximum reflection coefficient (voltage)		0.1
Taper	Switch between characteristic impedance distribution types (Exponential, Triangular, Klopfenstein)		Exponential
Nseg	Number of straight microstrip segments per wavelength		30
*W_min	Minimal allowed conductor width	Length	5 um
*W_max	Maximal allowed conductor width	Length	5000 um
MSUB	Substrate definition	Text	MSUB#[a]

[a]Modify only if schematic contains multiple substrates. See the "Using Elements With Model Blocks" for details.

** indicates a secondary parameter*

Parameter Details

Ro, RL. Desired input impedance (real) Ro and specified load impedance (real) RL.

Taper. Provides three options: Exponential, Triangular, and Klopfenstein. These options specify the type of characteristic impedance distribution along the line (see [1], [2]).

W_min, W_max. Used to verify feasibility of specified values of parameters Ro and RL. W_min and W_max might be respectively minimal and maximal widths allowable by the device fabrication technology or any other user-specified values. MTAPER2 uses MSUB parameters to calculate characteristic impedances Zmin(W_max) and Zmax(W_min). If either Ro<Zmin or RL>Zmax then MTAPER2 interrupts the simulation and issues an error message with recommendations for how to change Ro and/or RL.

Parameter Restrictions and Recommendations

1. Ro must be less than RL (Ro<RL). If Ro>RL then MTAPER2 swaps Ro and RL and issues a warning.
2. Ro should not be too close to RL. If RL-R0<0.1 ohm then MTAPER2 sets RL=Ro+0.1 and issues a warning.
3. To obtain the minimal length reflection coefficient, Rmax must be less than 0.5log(RL/Ro). If this is not true, MTAPER2 interrupts the simulation and issues an error message prompting you to review parameters Rmax, Ro, and RL.

Implementation Details

This model is constructed of a cascaded series of constant width microstrip transmission lines. It assumes a quasi-TEM mode of propagation and incorporates the effects of dielectric, conductive losses, and dispersion.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See “Cell Options Dialog Box: Layout Tab ” for Cell Options dialog box **Layout** tab details.

See “The Layout Process File (LPF)” for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

Recommendations for Use

Note that MTAPER2 calculates the minimal (optimal) length of line only after you set L=0. A zero value for the L parameter acts as a command to recalculate L and to update the layout. When you set L=0 the model calculates minimal length, updates the L parameter value at the schematic (replaces the 0 value with calculated length), and updates the layout. All of this is achieved before simulation starts. A simulation run uses the calculated minimal L or any non-zero value of L you enter. NI AWR recommends setting L=0 after updating any parameter value.

Remember to reset L=0 after switching types of characteristic impedance distribution with the Taper parameter.

Rmax is used only if you set L=0 to calculate minimal length. For a non-zero length MTAPER2 recalculates Rmax.

Note that synthesis of Klopfenstein taper does not provide start and end conductor widths that exactly correspond to the specified Ro and RL. This is by design (see [2]). Smaller ratios RL/Ro and smaller Rmax generate start and end widths that provide a better match to Ro and RL.

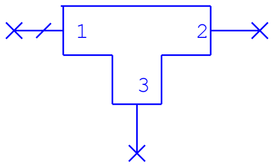
References

[1] R.W. Klopfenstein, "A transmission Line Taper of Improved Design," Proc. IRE, vol. 44, January 1956, pp. 31-35

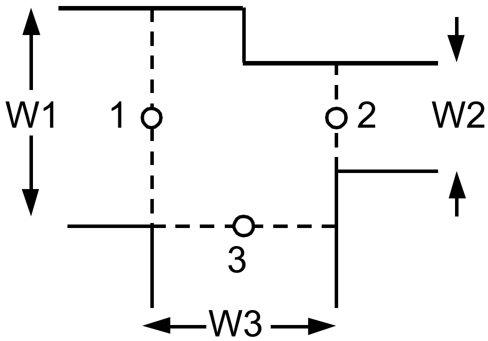
[2] D.M. Pozar, Microwave Engineering, ch. 5, sect. 5.8, 4th ed. New York: Wiley, 2012

Microstrip Tee - Junction (Closed Form): MTEE

Symbol



Topology



Parameters

Name	Description	Unit Type	Default
ID	Name	Text	TL1
W1	Conductor width @ node 1	Length	W[a]
W2	Conductor width @ node 2	Length	W[a]
W3	Conductor width @ node 3	Length	W[a]
MSUB	Substrate name	Text	MSUB[b]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See the Microwave Office Layout Guide for details.

[b]Modify only if schematic contains multiple substrates. See "Using Elements With Model Blocks" for details.

Implementation Details

MSUB indicates the name of the MSUB element to use for substrate characteristics. If blank, a default is used. The "through" line is connected to nodes 1 and 2. Node 3 is the "side" line. The reference planes for the junction are the ends of the respective lines; that is, the reference planes for ports 1 and 2 are the two sides of the line connected to node 3. Similarly, the reference plane for node 3 is the side of the widest of the lines connected to ports 1 and 2. The step in width between nodes 1 and 2 must occur at the midpoint of the width of port 3, and the "through" lines must have the same center line.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See "Cell Options Dialog Box: Layout Tab " for Cell Options dialog box **Layout** tab details.

See "The Layout Process File (LPF)" for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

Recommendations for Use

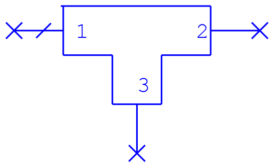
Discontinuity models function most accurately when attached to lines that match their corresponding edges. Directly connecting discontinuity models to one another reduces their accuracy.

Reference

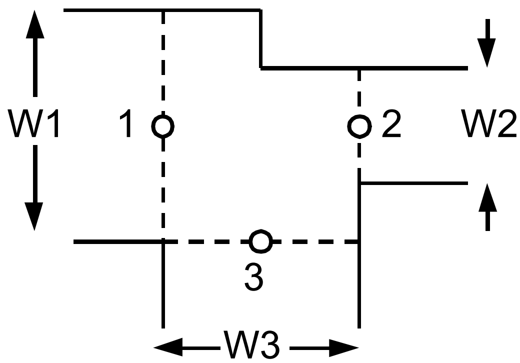
[1] E. Hammerstad, IEEE MTT-S Symp. Dig., p. 54, 1981.

Microstrip T- Junction (EM Based): MTEEX

Symbol



Topology



Parameters

MTEEX$ is a Microstrip iCell and has no parameters. See "Intelligent Cells (iCells)" for more information.

Name	Description	Unit Type	Default
ID	Element ID	Text	MT1
W1	Conductor width @ node 1	Length	W[a]
W2	Conductor width @ node 2	Length	W[a]
W3	Conductor width @ node 3	Length	W[a]
MSUB	Substrate definition	Text	MSUB[b]
*AutoFill	AutoFill database if not equal to 0	Integer	0

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. The intelligent version of this model automatically assumes these widths are equal to the attached models at that node. See the Microwave Office Layout Guide for details.
[b]Modify only if schematic contains multiple substrates. See "Using Elements With Model Blocks" for details.

** indicates a secondary parameter*

Implementation Details

This model implements an X-model of the microstrip T-junction. For a more detailed discussion of the X-models see " EM-based Models (X-models) " . This model does not include the effects of dielectric, conductor or radiation losses, nor does it account for the effect of conductor thickness.

Independent Parameters:	W1, W2, W3, W4 and Frequency
Scalable Parameters:	MSUB H- Substrate Height

Fixed Parameters:	MSUB Er Nominal
Statistical Parameters:	Er

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See "Cell Options Dialog Box: Layout Tab " for Cell Options dialog box **Layout** tab details.

See "The Layout Process File (LPF)" for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

Restrictions

$0.25 \le W_{widest}/H \le 4$ Required	$\varepsilon_{r\ nominal} \le 16$ Recommended
$0.2 \le W_{others}/W_{widest} \le 1$	$1 \le \varepsilon_{r\ nominal}$ Required
$0 \le \text{Frequency} \le F_{max}$ Recommended	$(100*\|\varepsilon_r-\varepsilon_{r\ nominal}\|)/\varepsilon_{r\ nominal} \le 10\%$ Recommended
	$(100*\|\varepsilon_r-\varepsilon_{r\ nominal}\|)/\varepsilon_{r\ nominal} \le 20\%$ Required

W_{widest} = The largest dimension of W1,W2,W3,W4

W_{others} = W1,W2,W3,W4 excluding W_{widest}

F_{max}= The frequency limits of this model are dynamic with respect to the dimensions of the discontinuity.

This dynamic frequency limit is displayed to the user via warning messages for the relative size, and dielectric in use. Importantly, this recommended frequency limit will change as a function of the largest width in the discontinuity for a given substrate definition. The frequency limit warns the user that at least one of the transmission lines constructing the discontinuity is approaching the frequency where multiple modes can propagate. See the section marked "Upper Frequency Limitations" in the General Discussion of the X-models.

Recommendations for Use

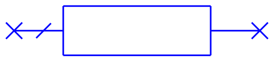
Discontinuity models function most accurately when attached to lines that match their corresponding edges. Directly connecting discontinuity models to one another reduces their accuracy.

References

This model was developed under research performed at NI AWR Corporation. The details of the implementation are considered proprietary in nature.

(Obsolete) Meander Microstrip Line (Closed Form): MTRACE

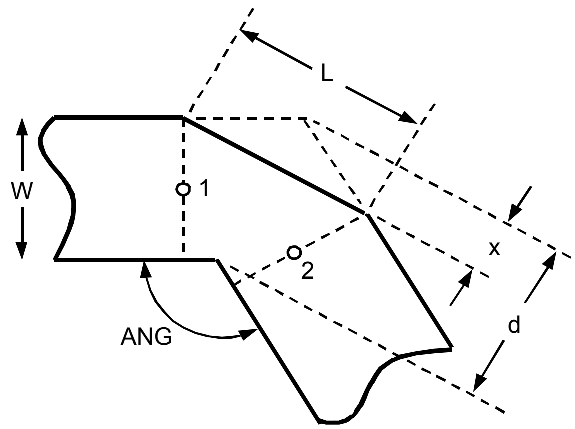
Symbol

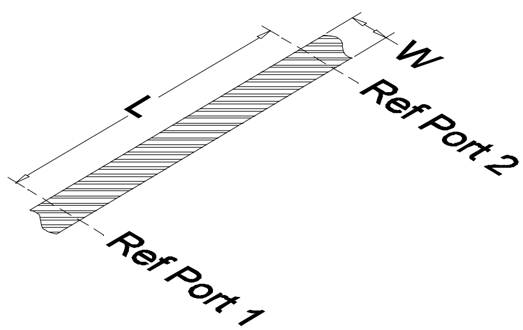


Summary

This element is OBSOLETE and is replaced by the Microstrip Meander Line 2 (Closed Form) (MTRACE2) element.

Topology





Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	TL1
W	Conductor width	Length	W[a]
L	Center line length	Angle	L[a]
Btype	Bend type:(1)Unmitered(2)mitered(3)opt miter		2
M	Miter for type=m		0
MSUB	Substrate definition	Text	MSUB[b]
*DB	Bend position array	Length	

Name	Description	Unit Type	Default
*RB	Bend angle array (degrees)		0

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See Microwave Office Layout Guide for details.

[b]Modify only if schematic contains multiple substrates. See the "Using Elements With Model Blocks" for details.

** indicates a secondary parameter*

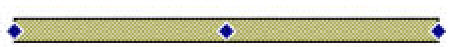
Implementation Details

The MTRACE element is a microstrip line that can be routed by graphically editing the element in the layout view. The model of the element is constructed by cascading microstrip bends, MBENDs, and microstrip lines, MLINs. The Btype parameter for the element specifies the type of MBEND used.

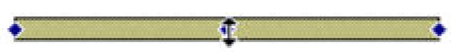
Btype	Description	Element
1	Unmitered	MBEND M=0
2	Arbitrary Miter	MBEND
3	Optimal Miter	MBENDA

See MBEND and MLIN for details about the models.

In general, graphical editing of the MTRACE element is accomplished using mouse commands. Double-clicking on the element activates blue diamonds that are used to manipulate the bends and lines.

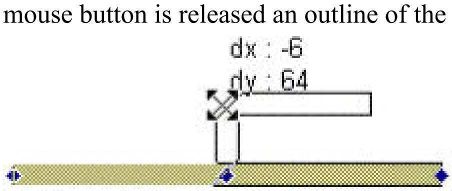


On one straight segment there are three diamonds. The outside diamonds control the length of the segment while the middle diamond controls bend manipulation. The diamonds are manipulated by placing the mouse over the diamond to activate an arrow cursor.

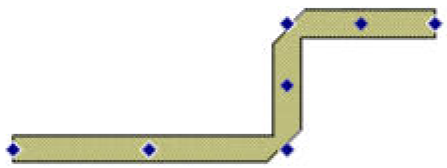


To move the diamond, click it and hold down the mouse button while dragging the cursor to another position. Release the mouse button when you reach the desired position. This series of command is abbreviated as Select Drag&Drop.

To add a bend, select the MTRACE element in the layout view. Select Drag&Drop the middle diamond. Before the mouse button is released an outline of the new MTRACE displays.

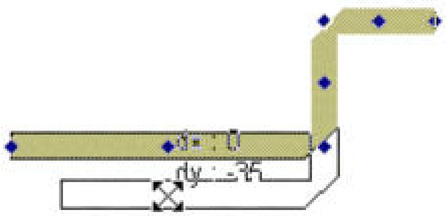


The completed bend is shown in the following figure.

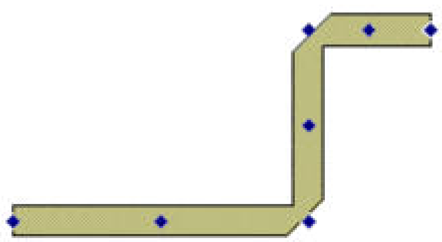


There are three options when moving a bend to a new position while keeping the overall length constant. The first two options refer to moving a bend on one of the end segments. The third option refers to moving a middle segment between two bends.

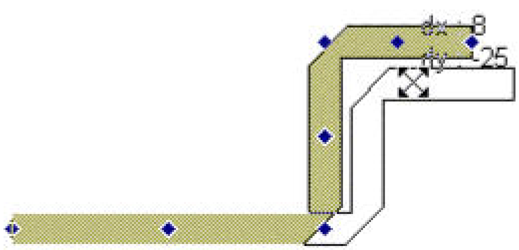
Option 1: When moving an end segment, the two segments attached to the bend being moved take up the slack in length while the other segment is kept constant. Select Drag&Drop the middle diamond of an end segment. An outline of the new bend position displays.



The following is the new bend position with overall length constant.

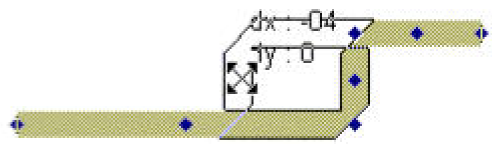


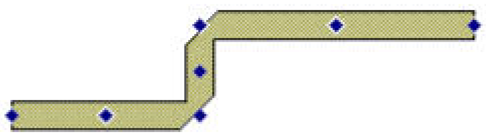
Option 2: Hold down Ctrl + Shift, and then Select Drag&Drop the middle diamond of an end segment. This option is the opposite of Option 1. The slack in the length is taken up by the segments not being moved. Note that the end segment that is moving is constant in length while the other two segments have changed.



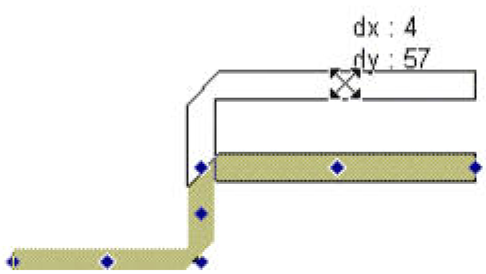
The new bend position for Option 2 is shown below.

Option 3: When moving a middle segment between two bends, the middle segment always stays constant while the end segments vary in length. Use the same commands as in Option 1 to move a middle segment.

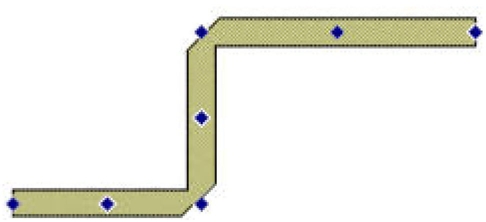




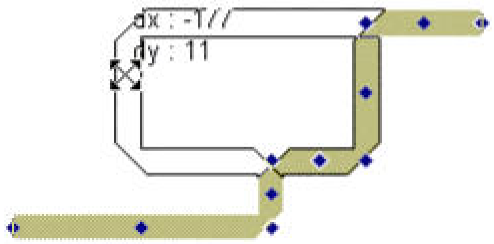
Moving a bend position while changing the overall length can only be done by moving the end segments of the MTRACE when there are two bends or less in the layout. When changing the length of an MTRACE and moving a bend position, the segment being moved remains at a constant length while the other segment attached to the bend varies in length. To move a bend and change the length, press the Shift key and Select Drag&Drop the middle diamond of the end segment. Note the change in length of the middle segment while the end segment has a constant length.



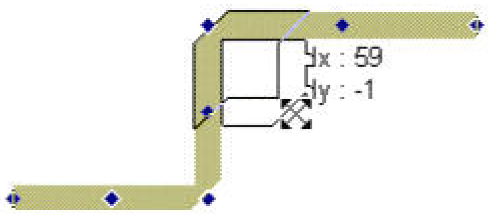
The following is the new change in length and bend position.



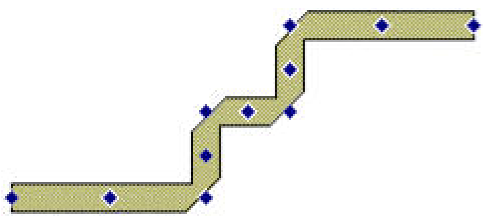
When there are more than two bends in the MTRACE you can change the bend position and the length in a "trombone" action. Use the Shift key and Select Drag&Drop procedure on an interior segment of the MTRACE. Note that the middle section length remains constant while the other two segments vary in length.



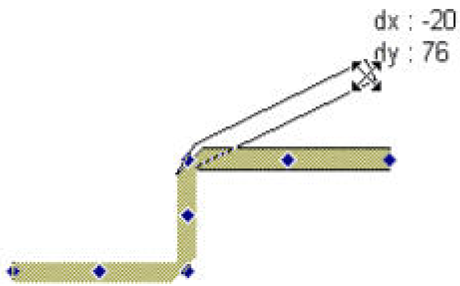
To add a bend after the initial bend has been added, press the Ctrl key and Select Drag&Drop the middle diamond between the two bends. The outline of the new bend is shown as follows.



The following is the final MTRACE with the new bend.



To rotate the end segment of an MTRACE. Press the **Ctrl** key and Select Drag&Drop the end of an MTRACE that has at least one bend in it.



When grid snap is on, the angle of rotation will only be placed on a Snap angle increment as specified in the Layout Options dialog box **Layout** tab in the **Rotation Snap** option. (Select Options > Layout Options to open the Layout Options dialog box.)

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

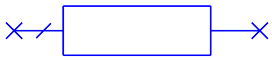
1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See "Cell Options Dialog Box: Layout Tab " for Cell Options dialog box **Layout** tab details.

See "The Layout Process File (LPF)" for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

Microstrip Meander Line 2 (Closed Form): MTRACE2

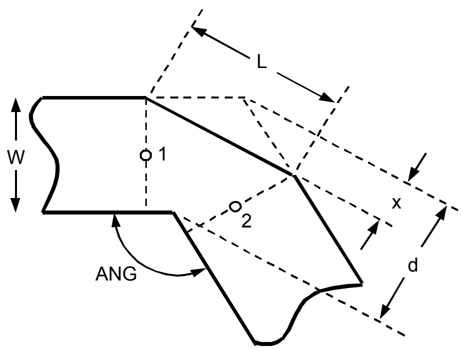
Symbol

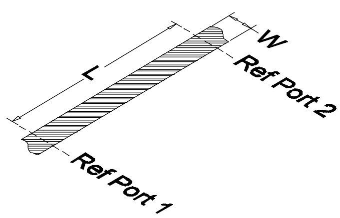


Summary

MTRACE2 improves on the MTRACE element as described in the "Implementation Details" section. MTRACE2 should be used in future projects in place of MTRACE, which is now classified as obsolete.

Topology





Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	TL1
W	Conductor width	Length	W[a]
L	Center line length	Angle	L2
Btype	Bend type: (1) Unmitered (2) Mitered (3) Opt Miter		2
M	Miter length-to-line width ratio		1.0
MSUB	Substrate definition	Text	SUB[b]
*DB	Bend position array	Length	{0}[c]
*RB	Bend angle array (degrees)		{0}4
*RC	Reclaim mode	Vector text (pull-down)	From back

[a]User-modifiable default. Modify by editing the *default.lpf* file in the root installation directory. See "Default Values" for details.

[b]Modify only if the schematic contains multiple substrates. See “Using Elements With Model Blocks” for details.
[c]Vector values are determined dynamically in the layout view of the schematic.

** indicates a secondary parameter*

Parameter Details

Btype. Bend type. For an 'optimally mitered' bend type, the substrate height must be known in order to calculate the optimal miter. If a substrate cannot be associated with the element, a default value of 100 um is used to calculate the optimal miter.

M. Miter length-to-line width ratio. This differs from the obsolete MTRACE element, for which M represented the miter fraction. Using a miter length-to-width ratio allows for miter fractions that are different at each bend (if their angles are different). This more closely resembles optimal miters and eliminates rendering problems when the bend angle is extremely shallow.

This parameter is ignored if BType = 3 (optimally mitered).

RC. Reclaim mode. During graphical editing, this parameter defines from which segment(s) slack is added or removed from the trace.

- From back (default)
- Other options to be added in future versions.

RB and DB. These parameters are meant to be used internally by the model. *Manually editing the RB and DB parameters is not recommended*; the MTRACE2 element should only be changed by graphically editing the layout cell.

Parameter Restrictions and Recommendations

1. $0.0 \le M \le 4.0$.
2. $0.02 \le W/H \le 20$ recommended
3. $T/W \le 0.7$ recommended
4. $T/H \le 0.5$ recommended
5. $\varepsilon_r \le 16$ recommended
6. $1 \le \varepsilon_r$ required
7. Tand ≥ 0 required
8. $0 \le$ Rho ≤ 1000 required
9. $0 \le$ Rho ≤ 100 recommended

Implementation Details

MTRACE2 Improvements

MTRACE2 is an improvement to the MTRACE element.

The following summarizes the MTRACE problems MTRACE2 fixed:

1. For the 'optimally mitered' bend type (BType = 3), the layout did not actually draw an optimal miter; a miter fraction of 0.7 was always used. However, it was being electrically modelled correctly.

2. For a non-90 degree bend, the miter fraction used to electrically model the bend was incorrect.
3. For multilayer traces (e.g. most MMIC traces), the offsets were not handled correctly along non-90 deg bends.
4. If the lengths of the end sections were too short, the miter fractions were changed to accommodate this, e.g. they did not represent the user-specified miter fraction.

Additional Information

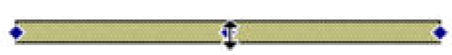
MTRACE2 is a microstrip line that can be routed by graphically editing the element in the layout view. The model of the element is constructed by cascading microstrip bends, mitered bends, and microstrip lines, MLINs. The Btype parameter for the element specifies the type of mitered bend used.

Btype	Description	Element
1	Unmitered	MBEND with M=0
2	Arbitrary Miter	MBEND
3	Optimal Miter	MBENDA

See MBEND and MLIN for details.

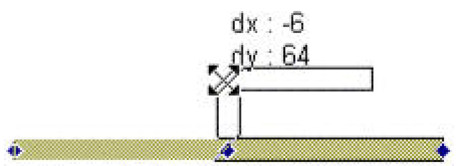
In general, graphical editing of the MTRACE2 element is accomplished with the mouse. Double-clicking the element activates blue diamonds that you use to manipulate the bends and lines.

On one straight segment there are three diamonds. The outside diamonds control the length of the segment while the middle diamond controls bend manipulation. The diamonds are manipulated by placing the mouse over the diamond to activate an arrow cursor.

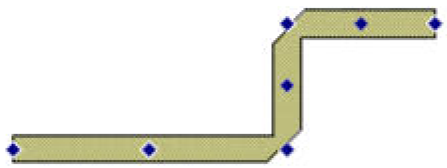


To move the diamond, click it and hold down the mouse button while dragging the cursor to another position. Release the mouse button when you reach the desired position. This series of movements is abbreviated as "Select Drag&Drop".

To add a bend, select the MTRACE2 element in the Layout View. Select Drag&Drop the middle diamond. Before you release the mouse button an outline of the new MTRACE2 displays.

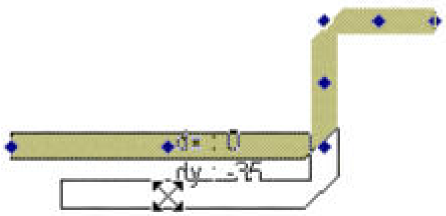


The completed bend is shown in the following figure.

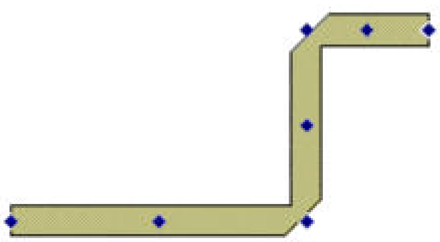


There are three options when moving a bend to a new position while keeping the overall length constant. The first two options refer to moving a bend on one of the end segments. The third option refers to moving a middle segment between two bends.

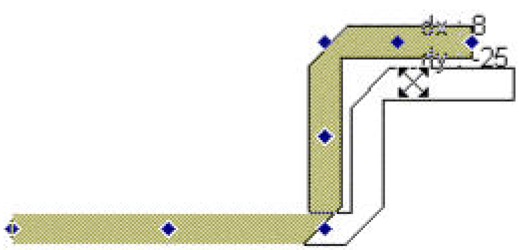
Option 1: When moving an end segment, the two segments attached to the bend being moved take up the slack in length while the other segment is kept constant. Select Drag&Drop the middle diamond of an end segment. An outline of the new bend position displays.



The following is the new bend position with overall length constant.

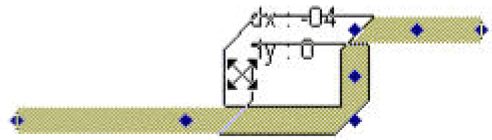


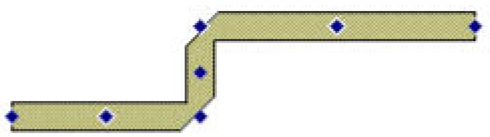
Option 2: Press **Ctrl + Shift**, and then Select Drag&Drop the middle diamond of an end segment. This option is the opposite of Option 1. The slack in the length is taken up by the segments not being moved. Note that the end segment that is moving is constant in length while the other two segments have changed.



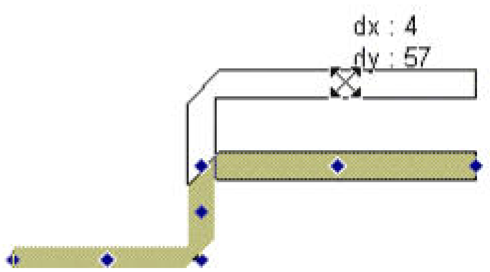
The new bend position for Option 2 is shown in the following figure.

Option 3: When moving a middle segment between two bends, the middle segment always stays constant while the end segments vary in length. Use the same commands as in Option 1 to move a middle segment.

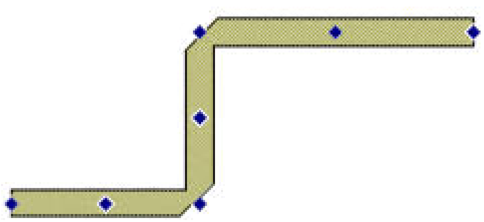




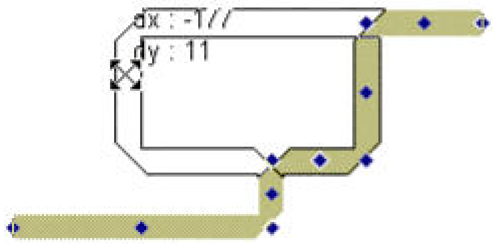
Moving a bend position while changing the overall length can only be accomplished by moving the end segments of MTRACE2 when there are two bends or less in the layout. When changing the length of an MTRACE2 and moving a bend position, the segment being moved remains at a constant length while the other segment attached to the bend varies in length. To move a bend and change the length, press the **Shift** key and Select Drag&Drop the middle diamond of the end segment. Note the change in length of the middle segment while the end segment has a constant length.



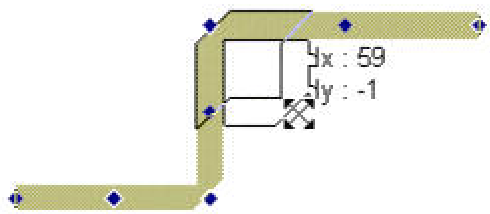
The following is the new change in length and bend position.



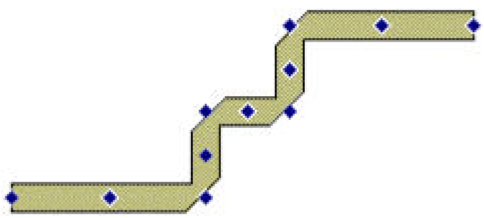
When there are more than two bends in MTRACE2 you can change the bend position and the length in a "trombone" action. Press the **Shift** key and Select Drag&Drop an interior segment of the MTRACE2. Note that the middle section length remains constant while the other two segments vary in length.



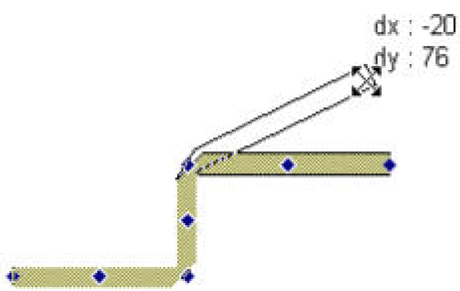
To add a bend after the initial bend is added, press the **Ctrl** key and Select Drag&Drop the middle diamond between the two bends. The outline of the new bend is shown in the following figure.



The following is the final MTRACE2 with the new bend.



To rotate the end segment of an MTRACE2 press the **Ctrl** key and Select Drag&Drop the end of an MTRACE2 that has at least one bend in it..



When the grid snap is on, the angle of rotation is only placed on a Snap angle increment as specified in the Layout Options dialog box **Layout** tab in **Rotation Snap** . (Choose **Options > Layout Options** to open the Layout Options dialog box.).

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.

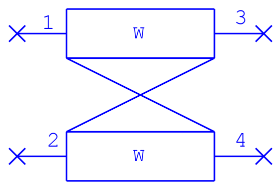
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See "Cell Options Dialog Box: Layout Tab " for Cell Options dialog box **Layout** tab details.

See "The Layout Process File (LPF)" for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

Microstrip Differential Line (Closed Form): MTRACED

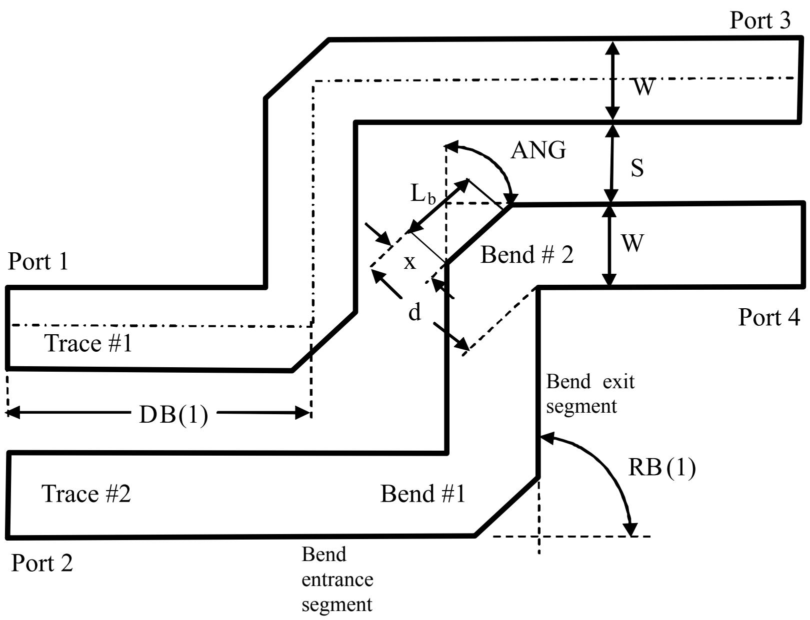
Symbol



Summary

MTRACED is a set of cascaded segments of edge-coupled microstrip lines and microstrip bends arranged at angles relative to each other. You can route this line graphically by editing the element in the Layout View. This line may be beneficial for design of differential transmission lines.

Topology



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	TL1
W	Conductor width	Length	W[a]
L	Center line length	Length	L1
Btype	Bend type: (1) "Unmitered", (2) "Mitered", or (3) "Opt Miter"		2
M	Miter length-to-width ratio (L/W)		1.0
MSUB	Substrate definition	Text	SUB[b]

Name	Description	Unit Type	Default
*DB	Bend position array	Length	{0}[c]
*RB	Bend angle array (degrees)	Angle	{0}[c]
S	Gap width conductors 1&&2	Length	L[a]
Method	Technique used to compute p.u.l parameters of Tx lines		"Closed Form"

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See the "Default Values" for details.

[b]Modify only if the schematic contains multiple substrates. See "Using Elements With Model Blocks" for details.

[c]Vector values are determined dynamically in the Layout View of the schematic.

** indicates a secondary parameter*

Parameter Details

L. Total length of the line measured along the central line of trace #1 (see the "Topology" section).

Btype. Bend type. For an "Opt miter" bend type, the substrate height must be known in order to calculate the optimal miter. If a substrate cannot be associated with the element, a default value of 100 um is used to calculate the optimal miter.

M. Miter length-to-line width ratio (Lb/W, see the "Topology" section). Using a miter length-to-width ratio allows for miter fractions (defined as x/d, see the "Topology" section) that are different at each bend (if their angles are different). This more closely resembles optimal miters and eliminates rendering problems when the bend angle is extremely shallow.

This parameter is ignored if BType = 3 ("Opt miter").

RB and DB. RB contains central line lengths of trace segments between bends along trace #1 starting from the entrance segment of bend #1. Note that the length of the last segment is equal to the difference between total line length L and the sum of all DB entries. DB contains the angle positions of the exit segment (see the "Topology" section) for each bend relative to the horizontal axis. These parameters are for internal use by the model. *Manually editing the RB and DB parameters is not recommended*; you should only change the MTRACED element by graphically editing the layout cell.

Parameter Restrictions and Recommendations

1. $0.0 \le M \le 4.0$.
2. It is preferable to have layout option "Rotation Snap" set to 90°. This will warrant correct layout. Other settings of "Rotation Snap" are also acceptable but they may make layout prone to distortions.

Implementation Details

MTRACED is a microstrip line that you can route by graphically editing the element in the Layout View. The model of the element is constructed by cascading microstrip bends, microstrip lines, and edge-coupled microstrip lines. The Btype parameter specifies the type of mitered bend used.

In general, you can graphically edit the MTRACED element using mouse commands. Double-clicking the element activates blue diamonds you can use to manipulate the bends and lines.

See the MTRACE2 documentation for a detailed description of the graphic editing process.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

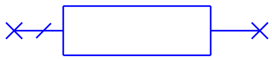
1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See "Cell Options Dialog Box: Layout Tab " for Cell Options dialog box **Layout** tab details.

See "The Layout Process File (LPF)" for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

(Obsolete) Multi-Layer Meander Microstrip Line (Configurable): MTRACEM

Symbol



Summary

There is no active replacement for this OBSOLETE element. MTRACEM is a user-configurable, multi-layer meander trace. This model is similar to the single-layer MTRACE model, except that the electrical models used for the transmission lines and discontinuities, as well as how their layout cells are rendered, are user-defined using an Element Process Utility. A multilayer substrate, NSUB, must be present in the project to use this model. Contact NI AWR for more information about how to set up a customized Element Process Utility.

This model is available with MWO-228, MWO-328, and ANO-xx9 configurations only.

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	TL1
NSUB	Substrate definition	Text	NSUB#[a]
*NC	Network node count		2
*XY	XY-point array		{0}
*FR	From index array		{1}
*WS	Segment width array		{40}
*LT	Segment line type array		{0}
*Impl	Implemented		1
*LM Type	Line model type (vector - length equal to number of lines)		{0}
*DM Type	Discontinuity model types (vector - length equal to number of discontinuities)		{0}
*ARG1	Model parameter 1 - see Help (vector		{0}
*ARG2	Model parameter 2 - see Help (vector)		{0}
*ARG3	Model parameter 3 - see Help (vector)		{0}
*ARG4	Model parameter 4- see Help (vector)		{0}
*ModLvl	Net modeling level		Level 3 -Distributed (tx lines only)
*EmExt	EM extractor		EM Extractor

[a]Modify only if schematic contains multiple substrates. See the “Using Elements With Model Blocks” for details.

** indicates a secondary parameter*

Parameter Restrictions and Recommendations

There are additional parameters associated with this model that are not included in the previous table. These parameters display on the **Parameters** tab of the Element Options dialog box and are set and used by the model; they are not intended for user editing.

Implementation Details

MTRACEM is a "smart" aggregate model whose aggregated elements are assembled dynamically. Based on the particular configuration, MTRACEM automatically figures out which models to aggregate. For example, MTRACEM can figure out whether a straight section of transmission line should be considered microstrip, stripline, embedded microstrip, etc. and aggregate the appropriate transmission line model. Similarly, it can determine whether a discontinuity should be a via, bend, taper, etc. and aggregate the appropriate discontinuity model. MTRACEM uses two things to determine what a model should be: the NSUB substrate definition and the Element Process Utility that is specified by the ProcUtil parameter of the NSUB. Contact NI AWR for information about setting up a customized Element Process Utility. You can also see the NSUB documentation for more information.

MTRACEM must always start and end with a transmission line section. Also, line sections and discontinuities must alternate (for example, line-discontinuity-line-discontinuity-line).

Layout

This element is considered obsolete. You should use iNets and the ACE simulator to obtain the same functionality.

Recommendations for Use

The following steps must be complete before you can use MTRACEM correctly:

1. Place an MTRACEM element in the schematic.
2. Place an NSUB substrate in the schematic (or in the Global Definitions window).
3. Specify the Element Process Utility by setting the "ProcUtil" parameter on the NSUB.
4. Set the remaining NSUB parameters so that they correspond to the specified process utility (ideally, the MTRACEM is used in conjunction with a process library. In this case, an NSUB can be set up and included with the library as an XML component, so no user set-up is necessary- the XML NSUB element can simply be dropped into the schematic and used as is).
5. Import the LPF file associated with the Element Process Utility.

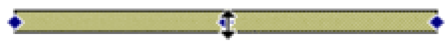
If any of these steps are not complete, error messages may result during electrical simulation and the layout may not render correctly.

Graphical Editing:

The following information is also found in the Microwave Office Layout Guide .

In general, you use the mouse to edit multilayer trace elements. Double-clicking an element activates blue diamonds that manipulate the bends and lines.

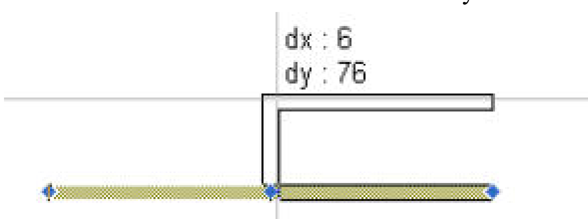
On one straight segment there are three diamonds. The outside diamonds control the length of the segment and the middle diamond controls bend manipulation. The diamonds are manipulated by placing the mouse over the diamond to activate an arrow cursor.



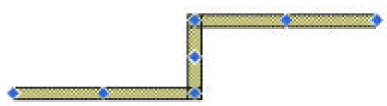
To move the diamond, click and drag the arrow cursor to another position and then release the mouse button.

Add a Bend

To add a bend to any line segment, select the multilayer trace element in the Layout View. While holding down the **Shift** key, click and drag the middle diamond to another position and then release the mouse button. Before you release the mouse button an outline of the new multilayer trace displays.

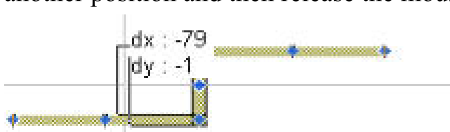


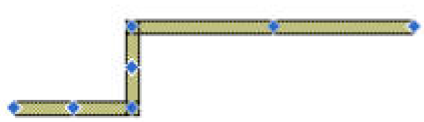
The following figure shows the completed bend.



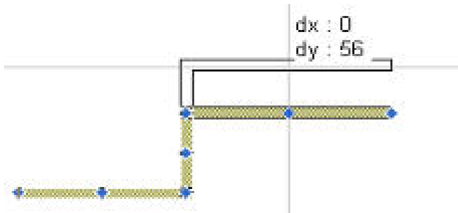
Moving a Line Segment

When moving a segment in a multiple segment trace, the length of the segment being moved always stays constant while the other segments may vary in length. Click and drag the middle diamond of the segment between the two bends to another position and then release the mouse button. The following two figures illustrate moving an interior segment.

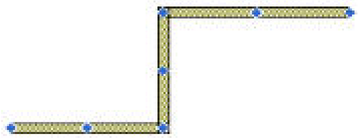




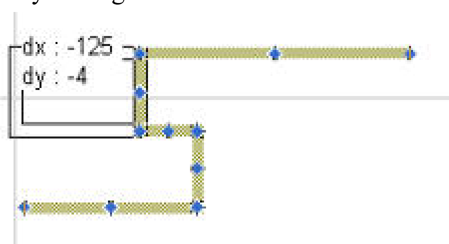
The following two figures illustrate moving an end segment.

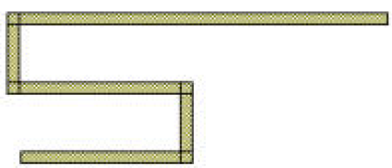


The following figure shows the new length and bend position.



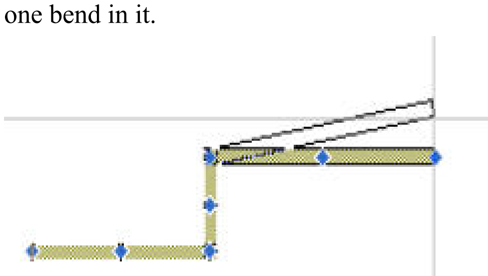
When there are more than two bends in the multilayer trace you can change the bend position and the length in a "trombone" action. Simply click and drag the middle diamond of an interior segment to another position and then release the mouse button. Note in the following two figures that the middle section length remains constant while the other two segments vary in length.

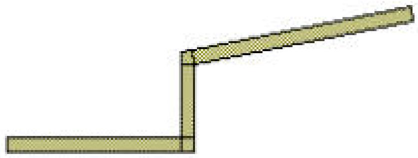




Rotate a Segment

To rotate the end segment of a multilayer trace, press **Ctrl** and click and drag the end of a multilayer trace that has at least one bend in it.

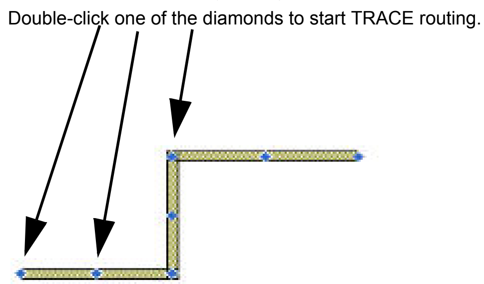




When grid snap is on, the angle of rotation is only placed on a Snap angle increment as specified on the Layout Options dialog box **Layout** tab in the **Rotation Snap** option. (Choose **Options > Layout Options** to open the Layout Options dialog box.)

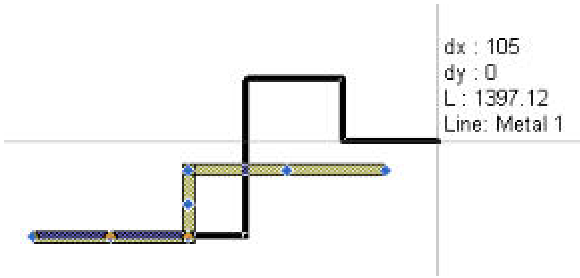
Multilayer Trace Routing

You can route the multilayer trace layout cell using the TRACE routing command. The TRACE routing command routes the multilayer trace from a center line path that is drawn with the mouse (or by using coordinate entry). The TRACE routing starts by double-clicking the multilayer trace to put it into edit mode. When the multilayer trace is in edit mode, the grab diamonds are visible (NOTE: For an iNet, the only diamonds that are not eligible for TRACE routing are the diamonds that display at a junction of three or more segments).

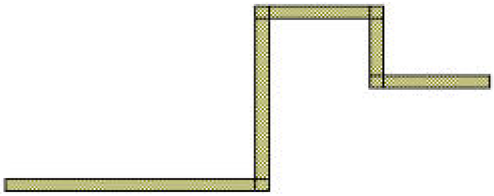


Multilayer TRACE Routing at the Start Point

If the TRACE routing is started at the beginning of a path, a center line path can be drawn as shown in the following figure. If you press the **Tab** key or **Space** bar while drawing the center line, a Coordinate Entry dialog box opens to allow coordinate entry of the center line points. You can also use gravity points when drawing the center line. You draw the center line by clicking at each of the desired vertex points, and complete it by double-clicking for the last point, or clicking on the same point twice.

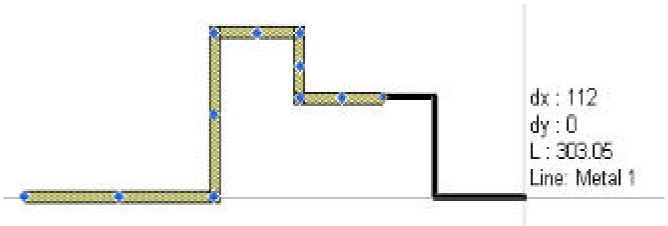


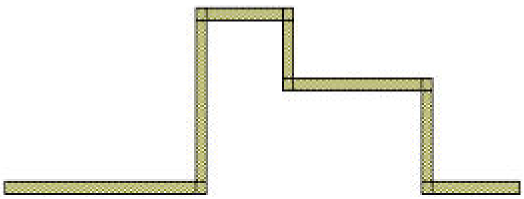
After the center line path drawing is complete, the multilayer trace element routes itself to the center line path. The routing is not always exactly on the center line because there are other constraints that may need to be satisfied. Note that performing a route at the starting path point completely replaces the old trace.



Multilayer TRACE Routing at the End Point

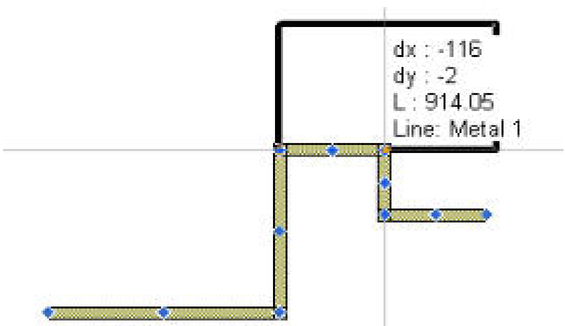
If the TRACE routing is started at the end of a path, the center line path that is drawn is appended to the existing path.

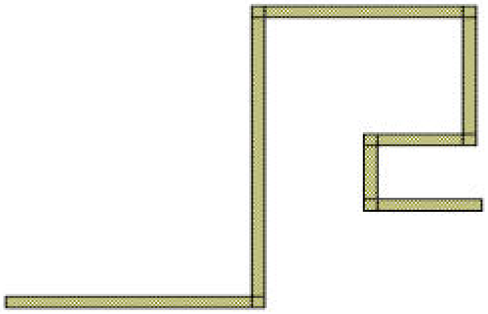




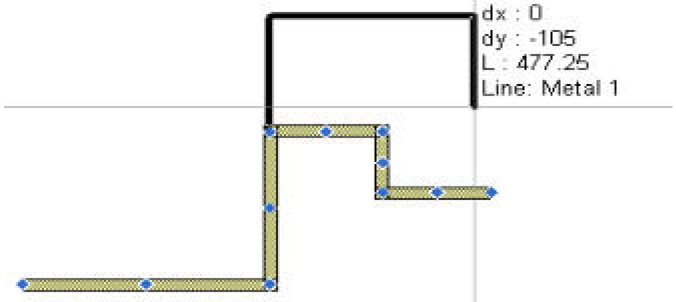
Multilayer TRACE Routing at an Interior Point

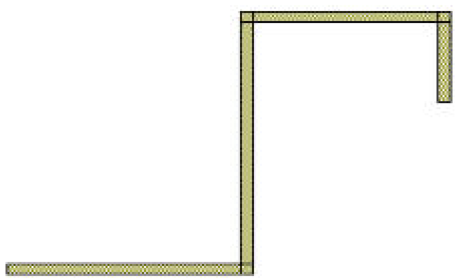
If the TRACE routing is started at an interior point, one of two things occurs, depending on where the routing ends. If the routing ends at another existing diamond, the portion of the trace between the starting and ending diamonds is replaced with the path specified by the new routing. This allows for replacing interior segments without affecting other segments of the trace.





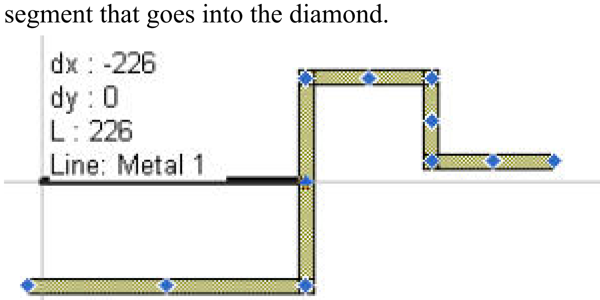
If the routing ends at a point that is not already part of the existing trace, the portion of the trace after the starting diamond is replaced with the path specified by the new routing. This allows for replacing a trace past a certain point without affecting the beginning segments of the trace.

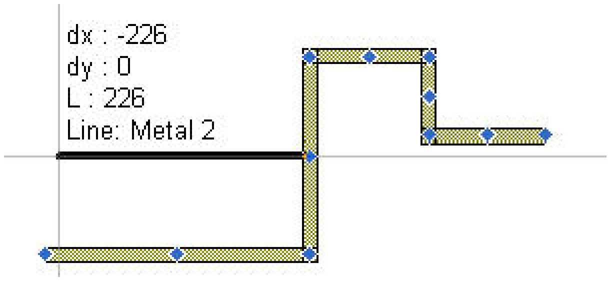


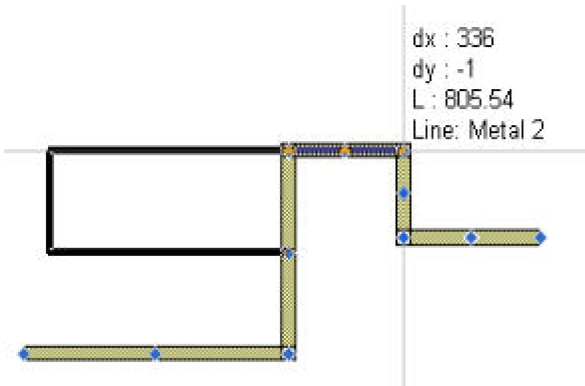


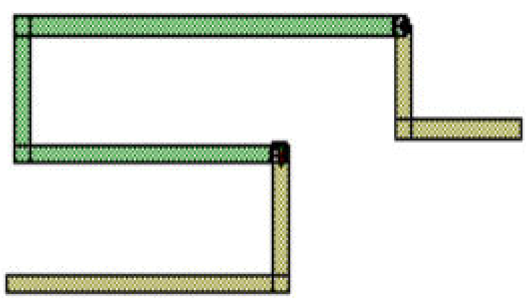
Multilayer TRACE Routing on Multiple Line Types

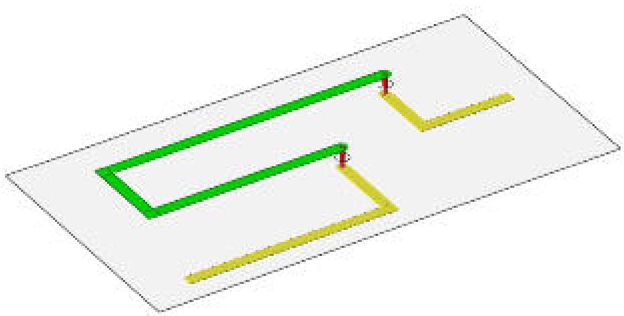
The multilayer trace can be routed to multiple line types. When you double-click a diamond to start the routing, note that the cursor displays delta x, delta y, length, and line type. To change the line type while routing, press the **Shift** key, then use the mouse wheel to scroll through all of the possible line types. If the routing begins at a diamond that is located at an intersection of two different line types, the line type that initially displays next to the cursor is the line type of the segment that goes into the diamond.











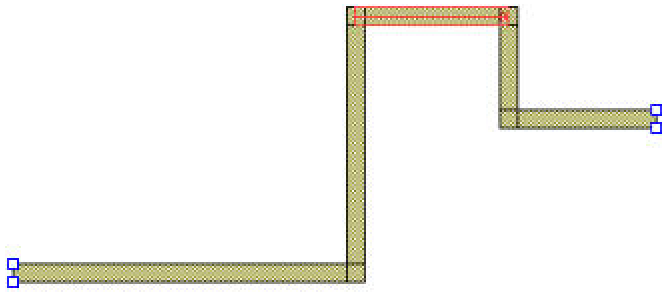
The 3D view of the layout shows that after routing on multiple line types, the proper transitions (vias in this case) are added.

Multilayer TRACE Manual Editing

You can edit the components that make up the multilayer trace. To display the dialog box for editing, right-click on the layout object in the Layout View and choose **Shape Properties** to display the Cell Options dialog box.

The **Lines** tab displays all of the individual line segments that make up the trace or net in a grid format. Each row of the grid represents a single line segment and its properties, including line type, line width, line length, and model type. **Model Type** is the type of model to use to electrically model the particular line segment. For example, if an Element Process Utility specifies two possibilities for a line segment, such as a closed-form model and an EM-based model, you can select which one to use.

In the **Discontinuity** column each line segment is paired with the discontinuity that exists at the *end* of the line segment. To see the direction for a particular segment, click on the line segment row in the grid and observe the layout. A red arrow displays over the line segment and points from the start to end. The following figure shows the arrow that displays on the fourth line segment when you click the fourth row in the grid:

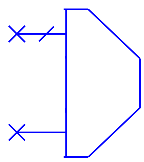


To edit the properties of a discontinuity at the end of a line segment, click on the button in the **Discontinuity** column for that particular line segment to display the Discontinuity Properties dialog box.

Here you can edit the parameters associated with the discontinuity. The discontinuity model type is similar to the line segment model type previously discussed, except that it applies to a discontinuity instead of a line segment.

U-turn (180-degrees) Microstrip Bend: MUBEND

Symbol

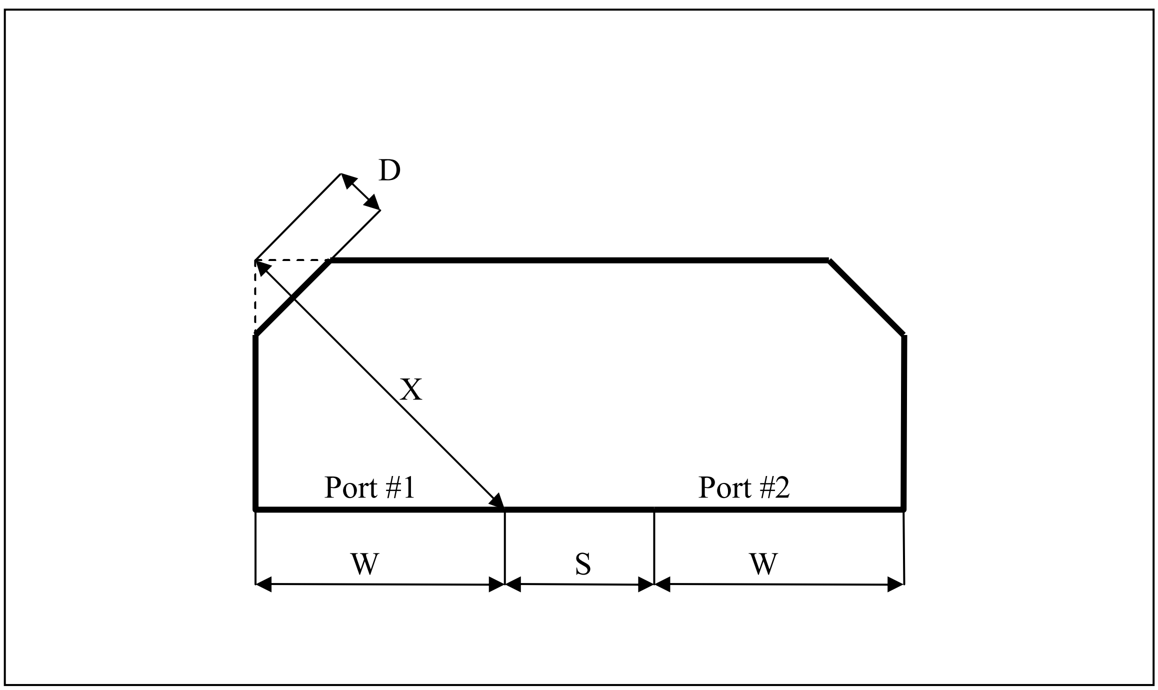


Summary

MUBEND models an interconnect between the ends of two adjacent coupled microstrip transmission lines. It comprises two right-angle mitered microstrip bends connected with a transmission line. Its primary use is as a component of hairpin microstrip filters.

MUBEND$ is the corresponding intelligent cell (iCell). An iCell model is identical to its non-iCell equivalent with the following exception: Certain dimension-related parameters are not explicitly user-specified; rather, they are automatically and dynamically determined by the dimensions of the attached elements. See the "Intelligent Cells (iCells)" for a detailed discussion of how to use iCells, their advantages, and their limitations.

Topology



Parameters

Name	Description	Unit Type	Default
ID	Name	Text	TL1
W	Width of bends	Length	W[a]
S	Gap between bends	Length	S[a]
M	Bends miter	Scalar	0
MSUB	Substrate definition	Text	MSUB#[b]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See the Microwave Office Layout Guide for details.

[b]Modify only if the schematic contains multiple substrates. See the "Using Elements With Model Blocks" for details.

Parameter Details

M. Miter M is defined as D/X (see "Topology") and its value cannot exceed 0.5.

Parameter Restrictions and Recommendations

1. Note that the maximum value of miter M is 0.5. This model issues an error if M>0.5. This cap is installed to avoid overlapping of bends if the value of parameter S is too small.
2. Parameter W refers to both ports of MUBEND; MUBEND is intended to connect microstrip lines of equal width.

Implementation Details

This model is implemented as a series connection of right-angle mitered microstrip bend, microstrip line, and another right-angle mitered microstrip bend.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See "Cell Options Dialog Box: Layout Tab " for Cell Options dialog box **Layout** tab details.

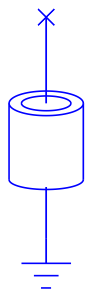
See "The Layout Process File (LPF)" for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

Recommendations for Use

This model is intended to provide a convenient means for designing coupled line structures like hairpin filters. You can use it, however, with any type of microstrip lines. The only limitation is a requirement of equal width of attached lines.

Cylindrical One Port Via with Microstrip Pad (Closed Form): MVIA1P

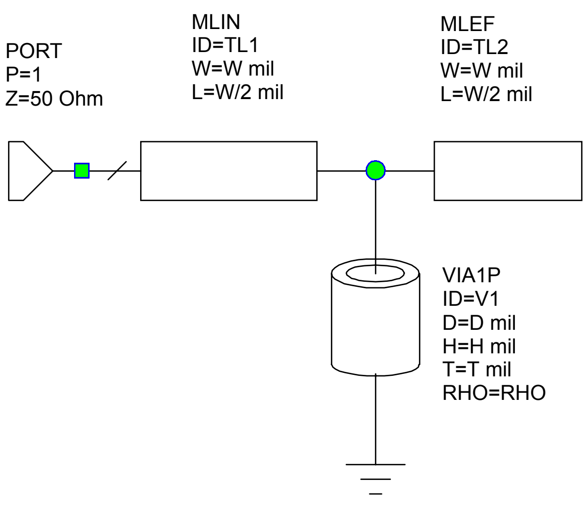
Symbol



Summary

MVIA1P models a microstrip short at the end of a transmission line. It is an aggregate model comprised of a transmission line and a transmission line open-end effect, with a via to ground at the junction of the two transmission line elements. The result is a square pad with a via in the middle.

Equivalent Circuit



Parameters

Name	Description	Unit Type	Default
ID	Element ID		V1
D	Hole diameter	um	40
H	Substrate thickness	Length	LPF default
T	Metal thickness	Length	LPF default
W	Pad width	Length	LPF default

Name	Description	Unit Type	Default
RHO	Metal bulk resistivity normalized to gold		1
*SNAME	Structure name from LPF file, empty string uses VIA, DEFINE, BEGIN from LPF file.		
MSUB	Substrate definition		

** indicates a secondary parameter*

Parameter Restrictions and Recommendations

Parameter W should match the width of the transmission line connected to MVIA1P. Otherwise, you can use any of the MSTEP family of junctions to model the change in line width.

Implementation Details

MVIA1P is implemented as a series combination of MLIN and MLEF, with VIA1P at the transmission line junction.

Layout

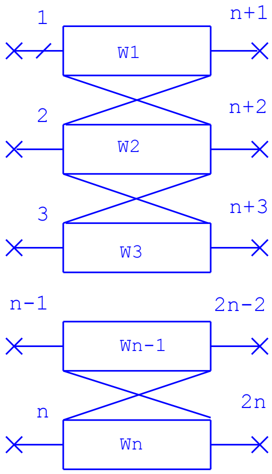
VIA models draw according to the SNAME parameter setting on the via, as configured in the LPF. If this parameter is left blank the via definition in the LPF is used to define the via layers. See "Via Definitions" for details. This typically works well if there is only one via type in the design. If you need to have different via layers, the best approach is to create structures in the LPF for each type, then set the SNAME parameter to match the structure name in the LPF. See "Structure Type Definitions" for more information.

Recommendations for Use

When using this model, keep in mind the physical nature of the square pad surrounding the via. You will effectively add a length to the targeted transmission line equal to W/2, which is the distance the signal sees in the MVIA1P before encountering the via. The element connected to MVIA1P should, therefore, be shortened by W/2 if electrical length is critical.

Multiple Edge Coupled Microstrip Lines (EM Quasi-Static) (X=3 to 16): MXCLIN

Symbol



Summary

This set of circuit components models sections of multiple edge coupled microstrip lines on a single-layer dielectric substrate. The model name includes the number of coupled lines (>=3): M3CLIN, M4CLIN and so on, up to M16CLIN. A backing ground plane is always present.

Parameters

Name	Description	Unit Type	Default
ID	Name	Text	TL1
W1	Width of microstrip line	Length	W1
W2	Width of microstrip line	Length	W1
Wn, n=3..XX - number of lines	Width of microstrip line	Length	W1
S1	Spacing between lines	Length	W1
Sn, n=2..XX - number of lines	Spacing between lines	Length	W[a]
L	Physical length	Length	L[a]
Acc	Accuracy parameter		1
MSUB	Substrate name	Text	MSUB[b]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory.

[b]If only one MSUB is present in the schematic, this substrate is automatically used. If multiple MSUB substrate definitions are present, you must specify which to use.

Implementation Details

Model implementation is based on the EM Quasi-Static technique described in [1] [10–325]. It accounts for losses in metal and in substrate dielectric. Dispersion is partly included.

These circuit components model sections of multiple edge coupled microstrip lines. M3CLIN models three edge coupled microstrip lines, M4CLIN - four, M5CLIN - five, M6CLIN - six, and so on up to M16CLIN. Two edge coupled microstrip lines are modeled by M2CLIN; Help for this model is provided separately.

The parameters W1 ..WX (strip widths), S1.. SX (gaps between strips), and L (line length) are dimensions entered in the default length units. Here X stands for the number of coupled lines that is different for each model.

The Acc parameter is the accuracy parameter (1<=Acc<=10). The default value for Acc is 1. If Acc is less than 1 or greater than 10 it is set automatically to 2. A larger value of Acc increases the density of mesh used in computations. Accuracy of model parameters may gain slightly from increasing Acc at the expense of a noticeable growth of computation time. Generally, a good trade-off between accuracy and computation time is to set Acc to 1.

The MSUB parameter specifies the microstrip substrate element, which defines additional cross-sectional parameters of the transmission line. If blank, a default is used.

These components do not impose restrictions on conductor thickness (the thickness may be zero, positive, or negative). Negative thickness means that the conductor is recessed into the substrate.

The components account for losses in metal and in substrate dielectric. Dispersion is not included.

NOTE: The implementation of EM Quasi-Static models heavily relies on the involved numerical algorithms. This may lead to the noticeable increase of computation time for some schematics that employ many models of this kind.

Layout

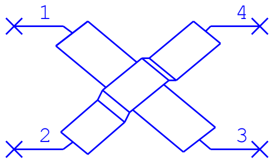
This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

References

[1] M.B. Bazdar, A.R. Djordjevic, R.F. Harrington, and T.K. Sarkar, "Evaluation of quasi-static matrix parameters for multiconductor transmission lines using Galerkin's method," IEEE Trans. Microwave Theory Tech., vol. MTT-42, July 1994, pp. 1223-1228.

Cross-over of Two of Orthogonal Microstrip Lines (EM Quasi-Static): MXOVER

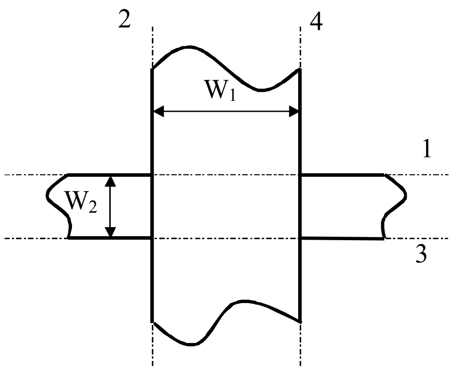
Symbol



Summary

MXOVER models a region where two microstrip lines overlap. Model implies that microstrip lines are orthogonal, are in general of unequal width and situated on adjacent dielectric layers.

Topology



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	TL1
W1	Top conductor width	Length	W[a]
W2	Bottom conductor width	Length	W[a]
Acc	Accuracy parameter		1
MSUB2	Substrate Definition	Text	MSUB1[b]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See the Microwave Office Layout Guide for details.

[b]Modify only if schematic contains multiple substrates. See "Using Elements With Model Blocks" for details.

Parameter Details

Acc. The accuracy parameter. The default value for Acc is 1. If Acc is less than 1 or greater than 10 it is set automatically to 2.

Parameter Restrictions and Recommendations

1. Accuracy parameter A is limited to $1 \le \text{Acc} \le 10$. Larger value of Acc increases density of mesh used in computations. Accuracy of model parameters may gain slightly from increasing Acc at the expense of noticeable growth of computation time. As a rule of thumb, good trade-off between accuracy and computation time is to set Acc to 1.
2. This model does not impose restrictions on the conductor thickness (thickness may be zero, positive, or negative). If thickness parameter T (see parameters of MPSUB) is negative conductors are recessed into the underlying dielectric layer.
3. Substrate parameters Tand1, Tand2 (loss tangents for upper and lower layers) should not exceed 3.0; substrate parameter Sig2 (bulk conductance of the lower layer) shouldn't exceed 3 S/m.

Implementation Details

Model implementation is based on EM Quasi-Static technique described in [1] [10–327]. Polarization loss (Tand1,Tand2) are allowed in both substrate layers. Conductive loss is allowed in lower layer only. Dispersion is partly included

To decrease the calculation time for schematics that contain several crossovers cache is implemented for this model. It means that during the first evaluation of schematic the most time consuming intermediate parameters for each crossover instance are being stored in memory cache. Each crossover model checks this cache looking for its duplicate. Duplicate crossovers copy the appropriate parameters from memory cache saving substantially on their recalculation.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

Recommendations for Use

It is implied that microstrip lines are located on two-layered substrate. The upper strip is on the top surface of the upper layer; the lower strip is on the top surface of the lower layer. Reference planes for ports are shown on Topology in dot-and-dash lines.

This element may be used to model crossover on semiconductor substrate insulated with oxide or another semiconductor. However, model impose limitations on allowed level of polarization and conductive losses, that is, for instance, only high resistance silicon (HRS) substrates may be modeled.

NOTE: The implementation of EM Quasi-Static models relies heavily on the involved numerical algorithms. This may lead to a noticeable increase in simulation time for schematics that employ many such models.

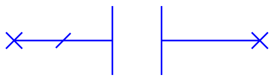
If the thickness of any layer is too small in comparison with the thickness of another layer, simulation time may also noticeably grow. However, reuse of cached crossovers alleviates this problem.

References

[1] M.B. Bazdar, A.R. Djordjevic, R.F. Harrington, and T.K. Sarkar, "Evaluation of quasi-static matrix parameters for multiconductor transmission lines using Galerkin's method," IEEE Trans. Microwave Theory Tech., vol. MTT-42, July 1994, pp. 1223-1228.

Cross-over of Two Microstrip Lines (Static Solution): MXOVER2

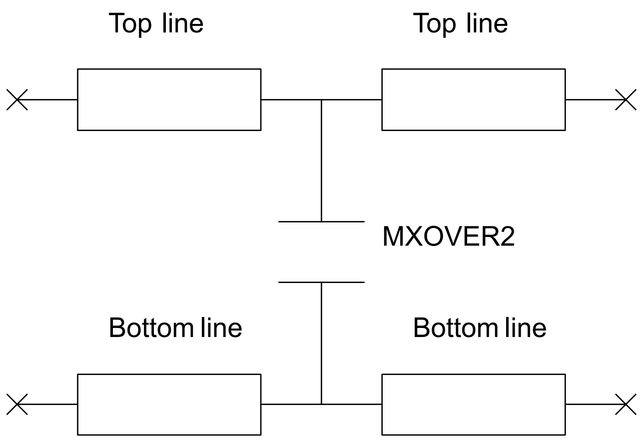
Symbol



Summary

MXOVER2 models a coupling between two microstrip lines located on different layers of dielectric stack and intersecting at an angle (not parallel). MXOVER2 is a two-port element, representing coupling between lines as lumped capacitance. You can specify a dielectric stack via the GMSUB substrate element and set the numbers of dielectric layers carrying coupled microstrip lines.

Equivalent Circuit



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	XV1
WT	Top conductor width	Length	W[a]
WB	Bottom conductor width	Length	W[a]
ANG	Angle between conductors (≥ 30 degrees)	Angle	1
CLT	Layer number of top conductor	Scalar	1
CLB	Layer number of bottom conductor	Scalar	2
GMSUB	Substrate Definition	Text	GMSUB1[b]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See the Microwave Office Layout Guide for details.

[b]Modify only if schematic contains multiple substrates. See "Using Elements With Model Blocks" for details.

Parameter Details

CLT, CLB.. Parameters CLT and CLB set the numbers of dielectric layers (specified at GMSUB) that carry microstrip conductors on their top. If the GMSUB Cover parameter is set to "Metallic Cover" and the top microstrip line is on top of the dielectric stack, then CLT=1 (the dielectric layer under the cover is not included in the count). If the GMSUB GND parameter is set to "Suspended Substrate" and the bottom microstrip line is on the bottom of the dielectric stack, then CLB=N+1, where N is the number of layers in the dielectric stack specified by the GMSUB N parameter.

T, Tand, Rho, SW, SWRight, SigmaC, Sigma, RhoV. MXOVER2 does not use these GMSUB parameters, however, the same substrate may be in use by other schematic elements, so these parameters may still need correct settings.

Parameter Restrictions and Recommendations

1. Parameter ANG must be in the range $30 \leq ANG \leq 150$ degrees.
2. Parameter CLT must exceed parameter CLB at least by 1.

Implementation Details

Model implementation is based on static Green's function technique in conjunction with approximation Green's function in a spectral domain. Capacitance is evaluated using method of moments. Polarization and conductive losses are not included.

To decrease the calculation time for schematics that contain several crossovers, cache is implemented for this model. During the first evaluation of schematic capacitances for each crossover, instances are stored in memory cache and saved to the hard drive. Each crossover model in every new schematic/project checks this cache looking for its duplicate. Duplicate crossovers copy the crossover capacitances from memory cache, saving substantial recalculation time.

Layout

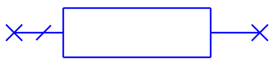
This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

Recommendations for Use

The implementation of this element relies on the involved numerical algorithms. This may lead to an increase in simulation time for schematics that employ many differing instances of MXOVER2.

Single Line on Stratified Homogeneous Substrate (EM Quasi-Static): PC1LIN

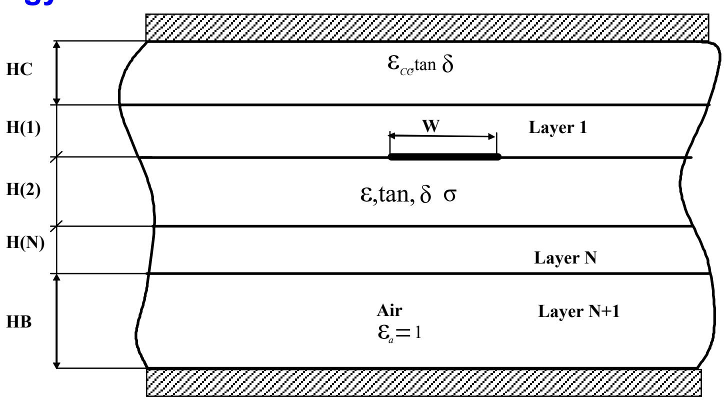
Symbol



Summary

PC1LIN models section of single microstrip line arranged within unshielded or shielded (upper cover is optional) stratified homogeneous (PCB/LTCC/HTCC) substrate. Substrate layers have arbitrary heights and are made of the same material; optionally the substrate may be suspended. Backing ground plane is always present. PC1LIN can account for presence of optional metallic side walls in conjunction with metallic cover.

Topology



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	TL1
W	Width of conductor	Length	W[a]
CL1	Number of layer containing conductor		1
L	Conductor length	Length	L[a]
Acc	Accuracy parameter		1
PCSUB	Substrate Definition	Text	PCSUB1[b]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See the Microwave Office Layout Guide for details.

[b]Modify only if schematic contains multiple substrates. See "Using Elements With Model Blocks" for details.

Parameter Details

PCSUB. Multilayer substrate parameters are listed in PCSUB model description. Substrate is comprised of layered PCB (/LTCC/HTCC) substrate (it implies that all layers are made of the same material but have various heights), optional metallic cover, backing ground plane and optional air layers separating substrate from backing ground plane (suspended substrate) and from cover (covered substrate).Optional metallic side walls are available in conjunction with metallic cover. Optional top and bottom air layers are specified by means of corresponding switches included as PCSUB parameters (see PCSUB model description). PCSUB implies that thickness of conductors on each layer is specified via vector parameter T. All conductors on the same layer have the same thickness. This thickness may vary between layers. Each dielectric layer must have corresponding entry in vector T even if this specific layer does not carry conductors. Model checks if the length of vector T is equal to the number of dielectric layers that might carry conductors.

CL1. Parameter CL1 specifies the number of layer that carries the conductor atop of it. It means that if conductor protrudes upward into layer #m (that is, it takes some space from layer #m) but lays onto the layer #m-1, parameter CL1 must be set to value m-1. If the conductor has negative thickness and is recessed into the layer #m (that is, it takes some space from layer #m), parameter CL1 must be set to value m.

Acc. The parameter Acc is the accuracy parameter. The default value for Acc is 1. If Acc is less than 1 or greater than 10 it is set automatically to 2.

Parameter Restrictions and Recommendations

1. Accuracy parameter A is limited to $1 \le Acc \le 10$. Larger value of Acc increases density of mesh used in computations. Accuracy of model parameters may gain slightly from increasing Acc at the expense of noticeable growth of computation time. As a rule of thumb, good trade-off between accuracy and computation time is to set Acc to 1.
2. Number of dielectric layers must include only those layers that may carry conductors on their top surface. Layers are enumerated from top to bottom. Note that if switch parameter Cover is set to "Metallic Cover" the topmost layer adjacent to cover is not included in count; likewise, if substrate switch parameter Gnd is set to "Suspended Substrate" the bottom air layer is not included in the count. Actually, layer number 0 may be displayed in error messages, e.g. in case when substrate is covered and thickness of conductor atop the layer #1 exceeds 95% of thickness of layer located above layer #1, message refer to this "undercover" layer as layer #0.
3. This model does not impose restrictions on the conductor thickness (thickness may be zero, positive, or negative). Negative thickness means that the conductor is recessed into the substrate. User supplies conductor thickness for each layer that may carry conductors (see description of vector parameter T in PCSUB documentation). See also below paragraph 4 regarding thickness of conductor in case of suspended substrate. Proceed with caution setting fictitious thicknesses for layers that do not actually carry conductors: excessive thickness may cause error message if it exceeds 95% of thickness of adjacent dielectric layer. To keep on the safe side you may set these thicknesses to zero.
4. If substrate is suspended, (N+1)th entry of thicknesses vector T (see substrate parameters) refers to conductor that is expected to be located at the bottom surface of layer N; model automatically assigns negative thickness to this conductor. It means that such a conductor extends downwards from the bottom surface of the substrate.

Implementation Details

Model implementation is based on EM Quasi-Static technique described in [1] [10–332]. It accounts for losses in metal and in substrate dielectric. Dispersion is partly included

Layout

The project LPF must contain a structure with a predefined name for the dielectric layer that carries a conductor. See “Structure Type Definitions” for more information on adding structures to LPF files. The structure name must be based on the template ML_LINE_X, where X is equal to the number of the dielectric layer specified with GMSUB (for example, ML_LINE_2 or ML_LINE_23). The conductor is comprised of material layers described in the structure ML_LINE_X.

If a structure with the corresponding name is not found, the name of the missing structure is drawn on the error layer.

Note that the number of dielectric layers (and X in ML_LINE_X) defined in GMSUB cannot exceed 30.

Note also that the layout cell assigns to the conductor the linetype index equal to the value of parameter CL1, that is equal to the number of the dielectric layer assigned to the conductor.

Recommendations for Use

NOTE: The implementation of EM Quasi-Static models relies heavily on the involved numerical algorithms. This may lead to a noticeable increase in simulation time for schematics that employ many such models.

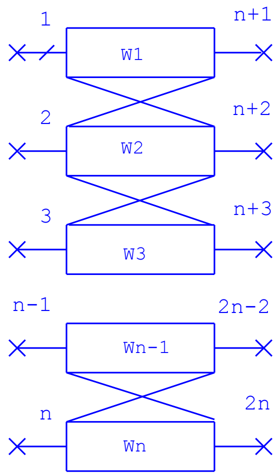
If thickness of any layer is too small in comparison with the thickness of another layer, simulation time may also noticeably grow.

References

[1] M.B. Bazdar, A.R. Djordjevic, R.F. Harrington, and T.K. Sarkar, "Evaluation of quasi-static matrix parameters for multiconductor transmission lines using Galerkin's method," IEEE Trans. Microwave Theory Tech., vol. MTT-42, July 1994, pp. 1223-1228.

Multiple Coupled Lines on Stratified Homogeneous Substrate (n=2 to 10) (EM Quasi-Static): PCnCLIN

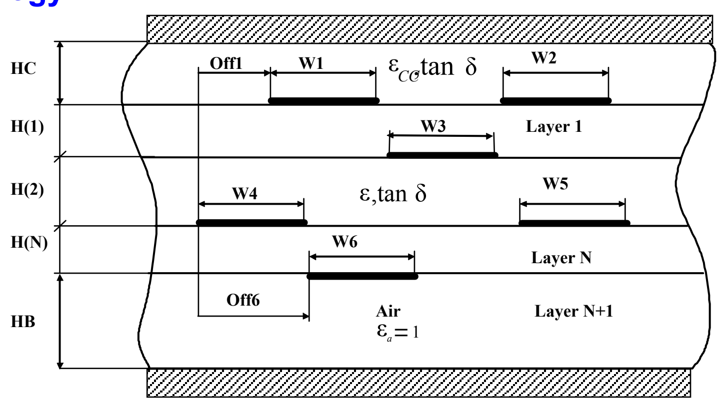
Symbol



Summary

PCnCLIN models a section of several (2 to 10) edge and/or offset broadside coupled microstrip lines arranged within unshielded or shielded (optional upper cover) stratified homogeneous (PCB/LTCC/HTCC) substrate. Substrate layers have arbitrary heights and are made of the same material; optionally the substrate may be suspended. A backing ground plane is always present. PCnCLIN models can account for the presence of optional metallic side walls in conjunction with a metallic cover.

Topology



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	TL1
W1	Width of conductor #1	Length	W[a]

Name	Description	Unit Type	Default
Wn, n=2..xx-number of lines	Width of conductor #n	Length	W[a]
Offs1	Offset of conductor #1	Length	W[a]
Offsn, n=2..xx-number of lines	Offset of conductor #n	Length	W[a]
CL1	Number of layer containing conductor #1		1
CLn, n=2..xx-number of lines	Number of layer containing conductor #n		1
L	Conductor length	Length	L[a]
Acc	Accuracy parameter		1
PCSUB	Substrate Definition	Text	PCSUB1[b]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See the Microwave Office Layout Guide for details.

[b]Modify only if schematic contains multiple substrates. See “Using Elements With Model Blocks” for details.

Parameter Details

PCSUB. Multilayer substrate parameters are listed in the PCSUB model description. The substrate is comprised of layered PCB/LTCC/HTCC substrate (this implies that all layers are made of the same material but have various heights), an optional metallic cover, backing ground plane and optional air layers separating the substrate from the backing ground plane (suspended substrate) and from the cover (covered substrate). Optional metallic side walls are available in conjunction with the metallic cover. Optional top and bottom air layers are specified by means of corresponding switches included as PCSUB parameters (see PCSUB). PCSUB implies that the thickness of conductors on each layer is specified via vector parameter T. All conductors on the same layer have similar thickness; this thickness may vary between layers. Each dielectric layer must have a corresponding entry in vector T even if this specific layer does not carry conductors. PCnCLIN checks if the length of vector T is equal to the number of dielectric layers that might carry conductors.

Acc. The accuracy parameter. The default value for Acc is 1. If Acc is less than 1 or greater than 10 it is set automatically to 2.

Offn. A relative horizontal offset of the conductor #n left edge from the left edge of the left-most conductor. The offset of the left-most conductor (or conductors) might be set to zero. Offn must always be positive.

CLn. Specifies the number of the layer that carries the conductor #n on top of it. If the conductor protrudes upward into layer #m (it takes some space from layer #m) but lays onto the layer #m-1, CLn must be set to the value m-1. If the conductor has negative thickness and is recessed into the layer #m (it takes some space from layer #m), CLn must be set to value m.

Parameter Restrictions and Recommendations

1. Accuracy parameter A is limited to $1 \le Acc \le 10$. A larger value of Acc increases the density of mesh used in computations. The accuracy of model parameters may gain slightly from increasing Acc at the expense of a noticeable growth of computation time. Generally, a good trade-off between accuracy and computation time is to set Acc to 1.
2. This model does not impose restrictions on the conductor thickness (the thickness may be zero, positive, or negative). A negative thickness indicates that the conductor is recessed into the substrate. You supply conductor thickness for each layer that may carry conductors (see the description of the vector parameter T in the PCSUB documentation). It is implied that all conductors located on top of the same dielectric layer have the same thickness. See number 7 below regarding the thicknesses of conductors in the case of a suspended substrate. Be cautious when setting fictitious

thicknesses for layers that do not actually carry conductors. Excessive thickness may prompt an error message if it exceeds 95% of the thickness of the adjacent dielectric layer. You can set these thicknesses to zero to prevent errors.

3. The Offn parameter has an upper limit. If for any pair of conductors the ratio Offn/W (W is the largest width in the pair) exceeds 200, PCnCLIN issues an error warning.
4. Conductors are numbered from left to right and from top to bottom (see "Topology"). It is assumed that conductors are located on top of dielectric layers.
5. The numbering of conductors displayed in error messages may vary. Note that conductor numbers in messages such as "Offset of conductor 3 must be greater than zero" is always the conductor order number according to paragraph 4 above. However, certain error messages that mention layer number refer to the relative conductor number on the corresponding layer. For example, "Thickness of strip #1 on layer #6 exceeds 95% of thickness of layer #5" implies that the left-most strip (#1 on layer 6) on the dielectric layer 6 is almost as thick as the upper adjacent layer 5.
6. The number of dielectric layers must include only those layers that may carry conductors on their top surface. Layers are numbered from top to bottom. Note that if the switch parameter Cover is set to "Metallic Cover" the top-most layer adjacent to the cover is not included in the count. Similarly, if the substrate switch parameter Gnd is set to "Suspended Substrate", the bottom air layer is not included in the count. Actually, layer number 0 may be displayed in error messages (when the substrate is covered and the thickness of conductors atop the layer #1 exceeds 95% of the thickness of the layer located above layer #1), this message refers to this "undercover" layer as layer #0.
7. If the substrate is suspended, the (N+1)th entry of thicknesses vector T (see the substrate parameters) refers to conductors that are expected to be arranged over the bottom surface of layer N. PCnCLIN automatically assigns a negative thickness to these conductors. This indicates that they are extended downwards from the bottom surface of the substrate (see conductors 6 and 7 in "Topology").

Implementation Details

Model implementation is based on the EM Quasi-Static technique described in [1] [10–336]. It accounts for losses in the metal and in the substrate dielectric. Dispersion is partly included.

Layout

The project LPF must contain some structures with predefined names for each dielectric layer populated with conductors (note that every conductor has the number of the assigned dielectric layer specified in the respective parameter CLn, where n is the conductor number). See "Structure Type Definitions" for more information on adding structures to LPF files. Structure names must be based on the template ML_LINE_X, where X is equal to the number of the dielectric layer. All conductors that belong to layer X are comprised of material layers described in ML_LINE_X.

For example, if model GMCLIN defines seven conductors, substrate GMSUB defines eight dielectric layers, and conductors are distributed over dielectric layers in accordance with the following table:

Conductor #	CL value
1	1
2	2
3	3
4	3
5	3
6	6
7	6

From the table you see that conductor #1 sits on dielectric layer #1, conductor #2 sits on dielectric layer #2, conductors #3, #4, and #5 are on layer #3, and conductors #6 and #7 sit on layer #6. This arrangement indicates that only layers #1, 2, 3, and 6 are populated with conductors and you need to specify structures ML_LINE_1, ML_LINE_2, ML_LINE_3, and ML_LINE_6.

If a structure with the corresponding name is not found, the name of the missing structure is drawn on the error layer.

Note that the number of dielectric layers (and X in ML_LINE_X) defined in GMSUB cannot exceed 30.

Note also that the layout cell assigns to each conductor #n the linetype index equal to the value of the corresponding parameter CLn, that is equal to the number of the dielectric layer assigned to conductor #n.

Recommendations for Use

See this section in GMnCLIN for details. The section also includes usage examples.

NOTE: The implementation of EM Quasi-Static models relies heavily on the involved numerical algorithms. This may lead to a noticeable increase in simulation time for schematics that employ many such models.

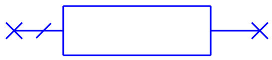
If the thickness of any layer is too small in comparison with the thickness of another layer, simulation time may also noticeably grow.

References

[1] M.B. Bazdar, A.R. Djordjevic, R.F. Harrington, and T.K. Sarkar, "Evaluation of quasi-static matrix parameters for multiconductor transmission lines using Galerkin's method," IEEE Trans. Microwave Theory Tech., vol. MTT-42, July 1994, pp. 1223-1228.

(Obsolete) RFIC Microstrip Line (FEM Quasi-Static): SIG1LNX

Symbol



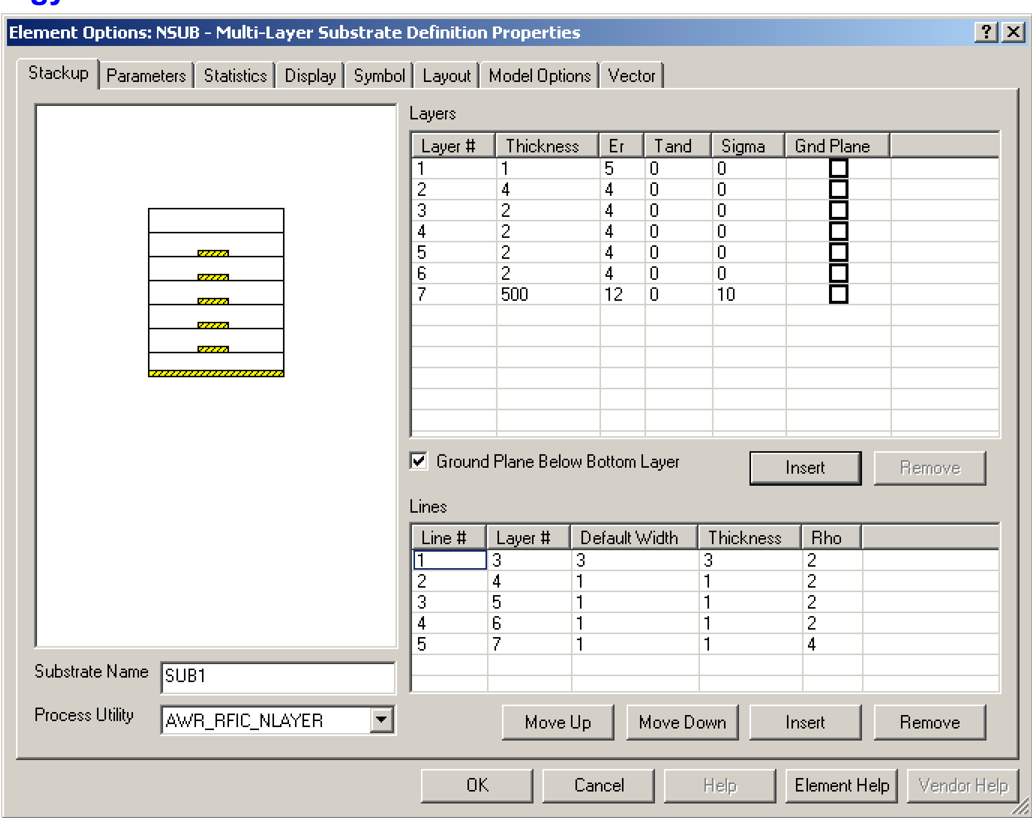
Summary

There is no active replacement for this OBSOLETE element. SIG1LNX models a section of single microstrip line (signal trace) with a conductor strip placed onto the surface of a dielectric layer incorporated into the multilayer dielectric stack (substrate definition NSUB). This model assumes the presence of infinite lateral grounds (backing ground is also modeled) to account for additional return paths in the on-chip environment.

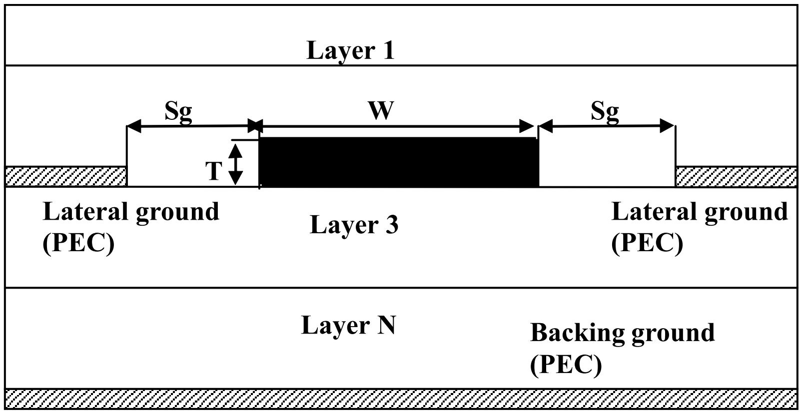
This model is constructed as an X-model (Table Based Interpolation) using the results of an EM 2-D quasi-static cross-sectional analysis based on the Finite Element Method (FEM). For a more detailed discussion of X-models, see [" EM-based Models (X-models) "]. The Quasi-Static FEM technique provides an accurate solution in the presence of a conducting/lossy dielectric stack, at the expense of a longer processor time spent on 2D meshing and solving large systems of linear equations. SIG1LNX gains a computational speed increase due to the table-based interpolation.

SIG1LNX is intended to work within iNets.

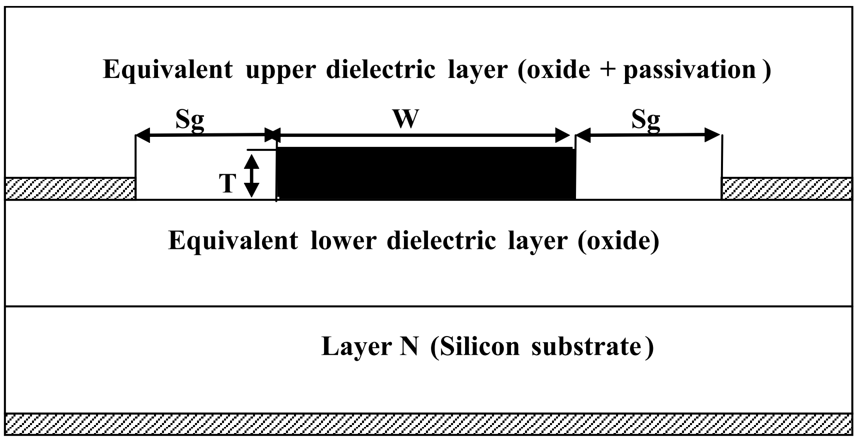
Topology



SIG1LNX topology is primarily that of NSUB. The first tab of the NSUB Element Options dialog box allows you to create/edit dielectric stackup as well as assign default line parameters to selected dielectric layers. Placing NSUB in the Global Definitions folder (in the Project Browser) provides access to NSUB from both schematics and layout environments.



Equivalent Topology



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	TL1
W	Width of line	Length	W[a]
TL	Line number (assigned in NSUB properties)		1
*Sg	Spacing between lateral ground and conductor	Length	W[a]
L	Line length	Length	L[a]
NSUB	Substrate definition	Text	See[b]
*IsHDL	Switch defines if highly doped layer (HDL) is in place. Choices: No/Yes		No
*H_HDL	Thickness of highly doped layer (HDL)	Length	2 microns
*Cond_HDL	Bulk conductance of highly doped layer (HDL)	S/m	700

Name	Description	Unit Type	Default
*AutoFill	AutoFill DataBase if not equal to 0		0

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory.

[b]Modify only if the schematic contains multiple substrates.

** indicates a secondary parameter*

Parameter Details

W. Conductor width is an independent parameter. This is the actual (modeled) line width; it is not related or associated in any way with the default line width found in NSUB properties (see the following TL parameter details).

TL. The line number. The TL parameter defines the height position of the modeled line in the dielectric stack as well as the material properties of the conductor metal.

The line number must be selected from the "Line #" column of the "Lines" table located on the Stackup tab in the NSUB Element Options dialog box (see "Topology"). You can view NSUB properties by double-clicking the NSUB element on a schematic, or by right-clicking the model name in the bottom window of the Element Browser and choosing Properties. The Insert, Move up, Move Down, and Remove buttons allow you to create as many lines as needed and place them on top of any layer from a previously created stackup. You create a "Layers" stackup filling table (just above the "Lines" table in the dialog box) by editing cell content as needed and using the Insert and Remove buttons.

SIG1LNX uses TL to get conductor thickness and bulk relative conductance of conductor metal (normalized to gold) from the corresponding cells of the "Lines" table (the model ignores the Default Width column). SIG1LNX also gets the layer number from the same table: SIG1LNX calculates the stackup height (measured up from the stackup bottom) of the top surface of this layer and takes it as stack height of the line conductor bottom.

Certain lines might be located on layers that are off limits for SIG1LNX (see "Parameter Restrictions").

Sg. The secondary parameter. Sg is an independent parameter that defines the offsets of lateral infinitely wide perfect conducting (PEC) grounds (see "Topology"). You can tweak Sg to model the impact of the actual grounding environment.

NSUB. Substrate parameters are listed in the NSUB description. Note that all layers can have arbitrary loss tangent and bulk conductance.

Autofill. A hidden input which allows you to specify that the entire interpolation table should be filled automatically at the current values. To instigate this filling process, this parameter should be set equal to 1. During normal operation, this parameter should be set equal to zero. You can access the hidden parameter by double- clicking on the schematic element.

IsHDL. Parameter IsHDL may take two values: either No or Yes. Default "No" means that no additional (to stack of NSUB) layer is present; "Yes" means that additional layer (presumably, highly doped Si) takes some height down from the top of the bottom layer (it is implied that last layer is a silicon substrate). This layer is not included in dielectric stack defined by NSUB.

H_HDL. Parameter H_HDL represents thickness of additional highly doped layer. If switch IsHDL is set to "No" (default) SIG1LNX ignores this parameter.

Cond_HDL. Parameter Cond_HDL represents bulk conductance of additional highly doped layer in Siemens/meter. If switch IsHDL is set to "No" (default) SIG1LNX ignores this parameter.

Parameter Restrictions and Recommendations

1. SIG1LNX implies that the ratio W/H lies within a predefined range of $0.05 \le W/H \le 5$ and that the ratio Sg/H lies within a predefined range of $0.05 \le S_g/H \le 10$ (H is the total thickness of all dielectric layers stacked above the bottom layer). Outside of this range, this model extrapolates output parameters and issues a warning.
2. The total thickness of all layers above the line conductor and the total thickness of all layers below the line conductor are fixed parameters; the same as the average dielectric constant and average loss tangent. The relative conductor bulk resistance Rho and the conductor thickness T are also fixed parameters. Microwave Office provides pre-generated tables for several typical values of SIG1LNX fixed parameters. Changes to any fixed parameter may start the automatic filling process (if Autofill is set to 1), the length of which may vary. You can change any fixed parameter to create corresponding tables.
3. The change of line number (TL) causes autofill only if the new line is located on a different stackup layer.
4. SIG1LNX limits the smallest conductor thickness to 0.1 micron due to possible meshing problems.
5. SIG1LNX does not accept lines located directly on the surface of the bottom layer, nor lines located on top of the upper layer of the dielectric stack.
6. A very large spread of thicknesses among layers that comprise the dielectric stack may cause the creation of very large meshes (for example, the number of nodes may exceed 40,000; common meshes contain 2,000 - 5,000 nodes). These huge meshes may slow down autofill and even cause numerical failures. The average autofill times (less than 30 minutes on 2.8 GHz P4) are reached for substrates 100-600 microns thick, and total thickness of silicon oxide 5-15 microns. The thickness of oxide above the conductor and below the conductor should be about 1 micron. It is only the total thickness of the layers above and the total thickness of the layers below the conductor that matter, because actual meshing is applied to this equivalent topology (see "Equivalent Topology").
7. The actual frequency range for which tables are generated is linked to the first cut-off frequency of the modeled line. This frequency is approximately evaluated as the cut-off frequency of the same line immersed into an oxide stack over a PEC surface. To provide generation of tables valid in the frequency range of a 0-100 GHz model, the total height of oxide above silicon substrate should be in the 5-20 micron range.

Implementation Details

SIG1LNX implies that only the bottom layer (substrate) of a dielectric stack has non-zero conductance. Non-zero conductance of any layer above the bottom layer is ignored. Losses of all layers both above and below the line conductor are averaged; the same as dielectric constants. Actually, SIG1LNX reduces a general NSUB structure to a thick lossy conductor sandwiched between two dielectric (presumably, oxide) layers above the third (presumably, silicon substrate) layer (see "Equivalent Topology").

The solution and implementation of FEM quasi-static technology is made possible due to partial use of the FEMM (Finite Element Method Magnetic) code developed by David Meeker [1] [10–341]. Meshing for FEMM is provided by the Triangle program developed by Jonathan Shewchuk [2] [10–341].

This X-model was developed under research performed at NI AWR Corporation. The full set of details of the implementation are considered proprietary in nature.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

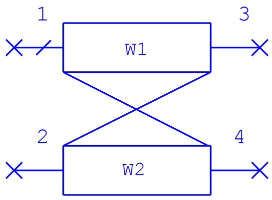
References

[1] David Meeker home page: http://femm.foster-miller.net/~dmeeker/ Follow this link for information: http://femm.foster-miller.net/Archives/readme.htm FEMM Manual: http://femm.foster-miller.net/Archives/doc/manual.pdf

[2] Jonathan Richard Shewchuk. Triangle. A Two-Dimensional Quality Mesh Generator and Delaunay Triangulator. Follow this link for information and download: http://www-2.cs.cmu.edu/~quake/triangle.html

(Obsolete) RFIC Edge-Coupled Symmetric Microstrip Lines (FEM Quasi-Static): SIG2LNX

Symbol



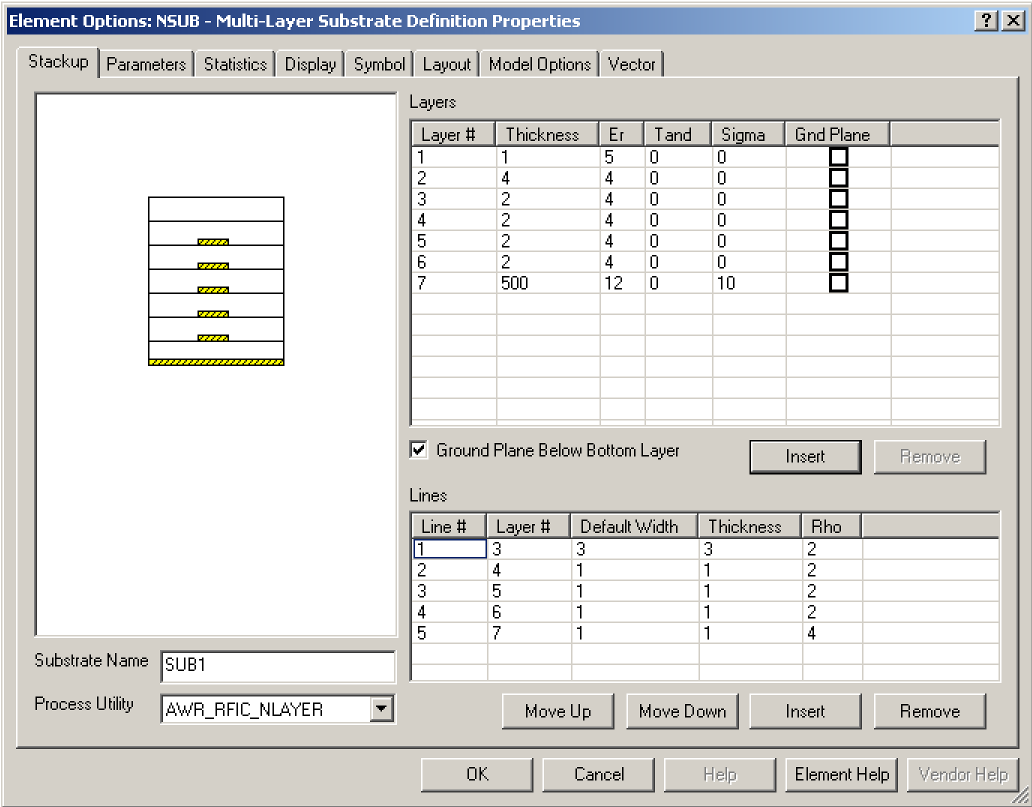
Summary

There is no active replacement for this OBSOLETE element. SIG2LNX models a section of two symmetric edge-coupled microstrip lines (signal traces) with a conductor strips placed onto the surface of a dielectric layer incorporated into the multilayer dielectric stack (substrate definition NSUB). This model assumes the presence of infinite lateral grounds (backing ground is also modeled) to account for additional return paths in the on-chip environment.

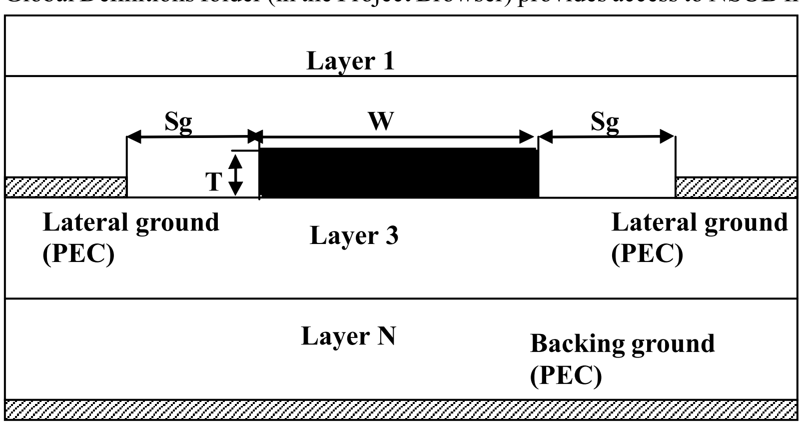
This model is constructed as an X-model (Table Based Interpolation) using the results of an EM 2-D quasi-static cross-sectional analysis based on the Finite Element Method (FEM). For a more detailed discussion of X-models, see " EM-based Models (X-models) " . The Quasi-Static FEM technique provides an accurate solution in the presence of a conducting/lossy dielectric stack, at the expense of a longer processor time spent on 2D meshing and solving large systems of linear equations. SIG2LNX gains a computational speed increase due to the table-based interpolation.

SIG2LNX is intended to work within iNets.

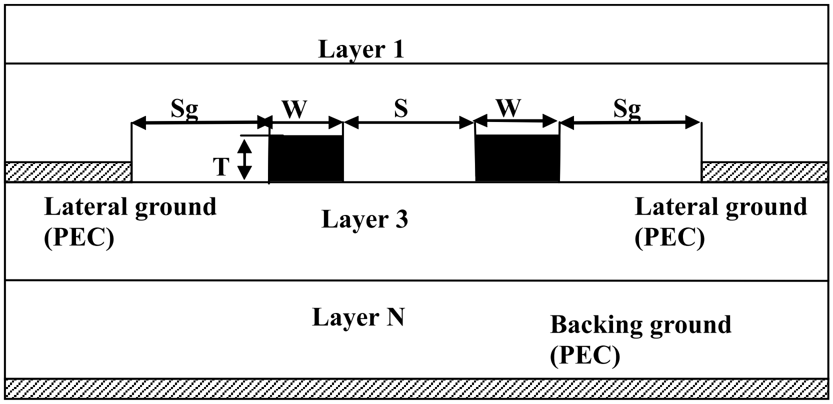
Topology



SIG2LNX topology is primarily that of NSUB. The first tab of the NSUB Element Options dialog box allows you to create/edit dielectric stackup as well as assign default line parameters to selected dielectric layers. Placing NSUB in the Global Definitions folder (in the Project Browser) provides access to NSUB from both schematics and layout environments.



Equivalent Topology



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	TL1
W	Width of lines	Length	W[a]
S	Spacing between lines	Length	W[a]
TL	Line number (assigned in NSUB properties)		1
*Sg	Spacing between lateral ground and conductor	Length	W[a]
L	Line length	Length	L[a]
NSUB	Substrate definition	Text	See[b]
*IsHDL	Switch defines if highly doped layer (HDL) is in place. Choices: No/Yes		No
*H_HDL	Thickness of highly doped layer (HDL)	Length	2 microns
*Cond_HDL	Bulk conductance of highly doped layer (HDL)	S/m	700
*AutoFill	AutoFill DataBase if not equal to 0		0

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory.
[b]Modify only if the schematic contains multiple substrates.

** indicates a secondary parameter*

Parameter Details

W. Model implies that conductors have equal width. Conductor width W is an independent parameter. This is the actual (modeled) line width; it is not related or associated in any way with the default line width found in NSUB properties (see the following TL parameter details).

S. Spacing between conductors edges.Spacing S is an independent parameter. This is the actual (modeled) line width; it is not related or associated in any way with the default line width found in NSUB properties (see the following TL parameter details).

TL. The line number. The TL parameter defines the height position of the modeled line in the dielectric stack as well as the material properties of the conductor metal. Both conductors are located at the same height position and have similar material properties.

The line number must be selected from the "Line #" column of the "Lines" table located on the Stackup tab in the NSUB Element Options dialog box (see "Topology"). You can view NSUB properties by double-clicking the NSUB element on a schematic, or by right-clicking the model name in the bottom window of the Element Browser and choosing Properties. The Insert, Move up, Move Down, and Remove buttons allow you to create as many lines as needed and place them on top of any layer from a previously created stackup. You create a "Layers" stackup filling table (just above the "Lines" table in the dialog box) by editing cell content as needed and using the Insert and Remove buttons.

SIG2LNX uses TL to get conductor thickness and bulk relative conductance of conductor metal (normalized to gold) from the corresponding cells of the "Lines" table (the model ignores the Default Width column). SIG2LNX also gets the layer number from the same table: SIG2LNX calculates the stackup height (measured up from the stackup bottom) of the top surface of this layer and takes it as stack height of the line conductor bottom.

Certain lines might be located on layers that are off limits for SIG2LNX (see "Parameter Restrictions").

Sg. The secondary parameter. Parameter Sg is an independent parameter. It defines the offsets of lateral infinitely wide perfect conducting (PEC) grounds (see "Topology"). You can tweak Sg to model the impact of the actual grounding environment.

NSUB. Substrate parameters are listed in the NSUB description. Note that all layers can have arbitrary loss tangent and bulk conductance.

Autofill. A hidden input which allows you to specify that the entire interpolation table should be filled automatically at the current values. To instigate this filling process, this parameter should be set equal to 1. During normal operation, this parameter should be set equal to zero. You can access the hidden parameter by double- clicking on the schematic element.

IsHDL. Parameter IsHDL may take two values: either No or Yes. Default "No" means that no additional (to stack of NSUB) layer is present; "Yes" means that additional layer (presumably, highly doped Si) takes some height down from the top of the bottom layer (it is implied that last layer is a silicon substrate). This layer is not included in dielectric stack defined by NSUB.

H_HDL. Parameter H_HDL represents thickness of additional highly doped layer. If switch IsHDL is set to "No" (default) SIG1LNX ignores this parameter.

Cond_HDL. Parameter Cond_HDL represents bulk conductance of additional highly doped layer in Siemens/meter. If switch IsHDL is set to "No" (default) SIG1LNX ignores this parameter.

Parameter Restrictions and Recommendations

1. SIG2LNX implies that the ratio W/H lies within a predefined range of 0.05≤W/H≤5 and ratio S/H lies within a predefined range of 0.01≤S/H≤5 Ratio Sg/H lies within a predefined range of $0.05 \le S_g/H \le 10$ (H is the total thickness of all dielectric layers stacked above the bottom layer). Outside of this range, this model extrapolates output parameters and issues a warning.
2. The total thickness of all layers above the line conductor and the total thickness of all layers below the line conductor are fixed parameters; the same as the average dielectric constant and average loss tangent. The relative conductor bulk resistance Rho and the conductor thickness T are also fixed parameters. Microwave Office provides pre-generated tables for several typical values of SIG2LNX fixed parameters. Changes to any fixed parameter may start the automatic filling process (if Autofill is set to 1), the length of which may vary. You can change any fixed parameter to create corresponding tables.
3. The change of line number (TL) causes autofill only if the new line is located on a different stackup layer.
4. SIG2LNX limits the smallest conductor thickness to 0.1 micron due to possible meshing problems.

5. SIG2LNX does not accept lines located directly on the surface of the bottom layer, nor lines located on top of the upper layer of the dielectric stack.
6. A very large spread of thicknesses among layers that comprise the dielectric stack may cause the creation of very large meshes (for example, the number of nodes may exceed 40,000; common meshes contain 2,000 - 5,000 nodes). These huge meshes may slow down autofill and even cause numerical failures. The average autofill times (less than 30 minutes on 2.8 GHz P4) are reached for substrates 100-600 microns thick, and total thickness of silicon oxide 5-15 microns. The thickness of oxide above the conductor and below the conductor should be about 1 micron. It is only the total thickness of the layers above and the total thickness of the layers below the conductor that matter, because actual meshing is applied to this equivalent topology (see "Equivalent Topology").
7. The actual frequency range for which tables are generated is linked to the first cut-off frequency of the modeled line. This frequency is approximately evaluated as the cut-off frequency of the same line immersed into an oxide stack over a PEC surface. To provide generation of tables valid in the frequency range of a 0-100 GHz model, the total height of oxide above silicon substrate should be in the 5-20 micron range.

Implementation Details

SIG2LNX implies that only the bottom layer (substrate) of a dielectric stack has non-zero conductance. Non-zero conductance of any layer above the bottom layer is ignored. Losses of all layers both above and below the line conductor are averaged; the same as dielectric constants. Actually, SIG2LNX reduces a general NSUB structure to a thick lossy conductor sandwiched between two dielectric (presumably, oxide) layers above the third (presumably, silicon substrate) layer (see "Equivalent Topology").

The solution and implementation of FEM quasi-static technology is made possible due to partial use of the FEMM (Finite Element Method Magnetic) code developed by David Meeker [1] [10–346]. Meshing for FEMM is provided by the Triangle program developed by Jonathan Shewchuk [2] [10–346].

This X-model was developed under research performed at NI AWR Corporation. The full set of details of the implementation are considered proprietary in nature.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

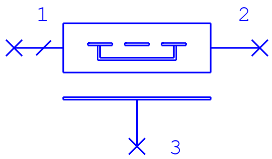
References

[1] David Meeker home page: http://femm.foster-miller.net/~dmeeker/ Follow this link for information: http://femm.foster-miller.net/Archives/readme.htm FEMM Manual: http://femm.foster-miller.net/Archives/doc/manual.pdf

[2] Jonathan Richard Shewchuk. Triangle. A Two-Dimensional Quality Mesh Generator and Delaunay Triangulator. Follow this link for information and download: http://www-2.cs.cmu.edu/~quake/triangle.html

Single Shielded CMOS Line (EM Quasi-Static): SIGCMOS

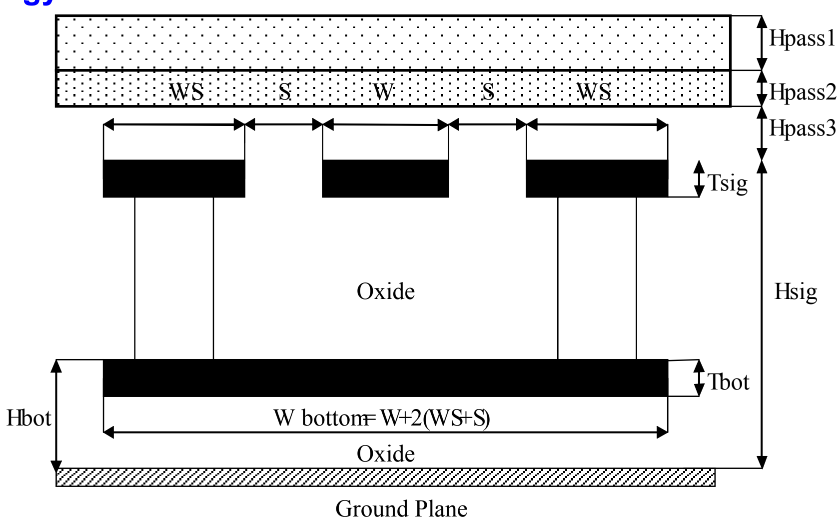
Symbol

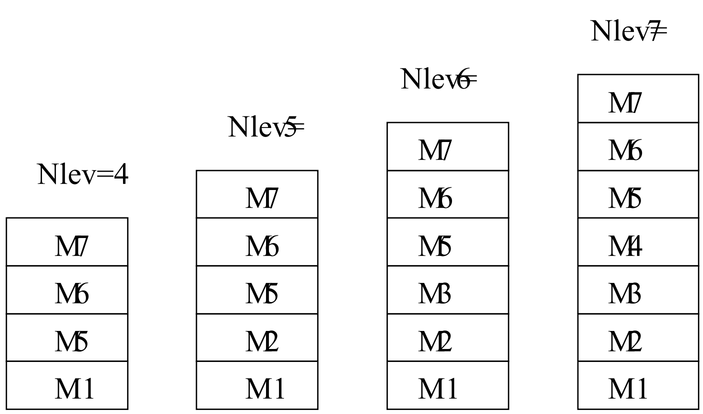


Summary

SIGCMOS models single metallic strip of rectangular cross-section located inside the dielectric stack in presence of side and bottom shielding strips. Model allows to remove/install side shields. The structure of model parameters is tailored to standard CMOS process.

Topology





Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	TL1
W	Width of Signal Conductor	Length	W[a]
WS	Width of Side Shield Conductor	Length	5 microns
S	Spacing between Signal and Side Shield Conductors	Length	5 microns
IsSide	Side Shield is in Place No/Yes		Yes
SigLev	Signal Conductor is on level M7/M6/..M2		M7
BShldLev	Bottom Shield Conductor is on level M6/M5/..M1		M6
Nlev	Number of Metal Levels		7
L	Conductor Length	Length	L[a]
Acc	Accuracy parameter		1
*ErOxide	Dielectric Constant of Silicon Oxide		4.1
*TdOxide	Loss Tangent of Silicon Oxide		0
*ErPass1	Dielectric Constant of First Top Passivation Layer		3.4
*TdPass1	Loss Tangent of First Top Passivation Layer		0
*ErPass2	Dielectric Constant of Second Top Passivation Layer		7.0
*TdPass2	Loss Tangent of Second Top Passivation Layer		0
*Hpass1	Height (Thickness) of First Top Passivation Layer	Length	2.0
*Hpass2	Height (Thickness) of Second Top Passivation Layer (Nitride)	Length	0.5
*Hpass3	Height (Thickness) of Third Top Passivation Layer (Oxide)	Length	1.0
*Hm7	Stack Height of M7Metal Top	Length	17.5
*Hm6	Stack Height of M6 Metal Top	Length	11.5
*Hm5	Stack Height of M5 Metal Top	Length	5.5
*Hm4	Stack Height of M4 Metal Top	Length	4.5
*Hm3	Stack Height of M3 Metal Top	Length	3.5
*Hm2	Stack Height of M2 Metal Top	Length	2.5
*Hm1	Stack Height of M1 Metal Top	Length	1.5
*Tm7	Thickness of M7Metal	Length	3.0
*Tm6	Thickness of M6 Metal	Length	3.0
*Tm5	Thickness of M5 Metal	Length	0.5
*Tm4	Thickness of M4 Metal	Length	0.5
*Tm3	Thickness of M3 Metal	Length	0.5
*Tm2	Thickness of M2 Metal	Length	0.5
*Tm1	Thickness of M1 Metal	Length	0.5
*Rho	Bulk Resistance of Metallization Relative to Gold		1.47

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See Microwave Office Layout Guide for details.

** indicates a secondary parameter*

Parameter Details

IsSide. If this switch is set to Yes side shielding conductors are in place; setting it to No removes side shields and leaves a signal conductor and bottom shield only.

SigLev. This switch allows user to attribute particular metal level to signal conductor and side shields. Placing conductors on certain level provides assignment of Hsig (see Topology) to preset value from available set of stack heights of metal levels; Tsig (see Topology) also gets assigned to predefined thickness value. Actual values of Hsig, Tsig may also be affected by parameter Nlev (see Nlev explanations below).

BShldLev. This switch allows user to attribute particular metal level to the bottom shield conductor. Placing conductors on certain level provides assignment of Hbot (see Topology) to preset value from available set of stack heights of metal levels; Tbot (see Topology) also gets assigned to predefined thickness value. Actual values of Hsig, Tsigmay may also be affected by parameter Nlev (see Nlev explanations below). Model implements sanity checks and checks availability of selected metal levels.

Nlev. Nlev provides actual number of metal levels used in particular CMOS process. Model implies that maximum available number of metal layers is 7 and 4<=Nlev<=7 (content of metal stack for each value of Nlev is shown on Topology). It also implies that elevation of level M7 over M6 and elevation of M6 over M5 is constant for all values of Nlev (in most CMOS processes M7 and M6 are relatively thick metal layers used for power buses and passive high Q elements). Model also implies that all stack heights specified as model parameters are heights of top surfaces of each metal measured from ground plane (see Topology). Thus, changing Nlev changes actual value of Hm5 and actual values of Hm6, Hm7 because any setting Nlev<7 removes one or several metal levels below M5 from the stack (see Topology). Nlev also limits availability of metal level selections done via SigLev and BShldLev. For example, if Nlev=4 than only M7, M6, and M5 are available for SigLev and, correspondingly, only M6, M5, and M4 are available for BshldLev. The table below lists all feasible combinations of metal levels and demonstrates how actual Hm5, Hm6, and Hm7 are evaluated for each value of Nlev.

Nlev	Allowed SigLev	Allowed BShldLev	Actual Hm5	Actual Hm6	Actual Hm7
4	M7, M6, M5	M6, M5, M1	*Hm2	*Hm2+D6	*Hm2+D7
5	M7, M6, M5, M2	M6, M5, M2, M1	*Hm3	*Hm3+D6	*Hm3+D7
6	M7, M6, M5, M3,M2	M6, M5, M3, M2, M1	*Hm4	*Hm3+D6	*Hm4+D7
7	M7, M6, M5, M4, M3, M2	M6, M5, M4, M3, M2, M1	*Hm5	*Hm3+D6	*Hm5+D7

where D6=*Hm6 -*Hm5 and D7=*Hm7 -*Hm5. Asterisk refers to model parameters values.

ErPass1, TdPass1, ErPass2, TdPass2. Model implies that CMOS process protects stack of interconnects with three passivation layers. ErPass1, TdPass1, ErPass2, TdPass2 are material parameters of top two passivation layers counting from top. The third from top passivation layer is silicon oxide.

Hpass1, Hpass2, Hpass3. These parameters are thicknesses (not stack heights) of top three passivation layers.

m1..Hm7. CMOS process specific stack heights of metal levels. Hm1..Hm7 are referenced to the ground plane and measured up to the top of the corresponding metal layer.These parameters are secondary (hidden by default) and must be set to process specific values before modeling.

Tm1..Tm7. CMOS process specific metal thicknesses of metal levels. These parameters are secondary (hidden by default) and must be set to process specific values before modeling.

Rho. Model implies that the same metal is used at all metal levels. Rho is a ratio of metal bulk resistance to bulk resistance of gold. Default value is provided for aluminum.

Parameter Restrictions and Recommendations

1. Available selections of SigLev and BShldLev are limited to those feasible for the specified Nlev; these restriction may cause error messages if user selects SigLev /BSldLev incompatible with the available set of metal levels (see table on Topology).
2. The following restrictions must be met: Tbot<0.95Hbot Tsig<0.95(Hsig-Hbot) If they are violated model issues corresponding error messages.
3. Ground plane location. Model does not account for substrate properties because it is implied that bottom shield effectively isolates signal conductor from substrate influence. Thus, ground plane might be placed at the oxide/substrate boundary.

Implementation Details

Shielding conductors are implied to be connected by vias all the way along the line length and attached to a single port 3. This connection is ideal and vias are not modeled.

Model does not account for silicon substrate properties because it is implied that bottom shield effectively isolates signal conductor from substrate influence.

Width of bottom shield is affected by parameters WS, S even if IsSig=No. Wbottom is always W+2(WS+S) so WS and S might be used to control Wbottom.

Model implementation is based on EM Quasi-Static technique described in [1] [10–350]. It accounts for losses in metal and in substrate dielectric. Dispersion is partly included

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Recommendations for Use

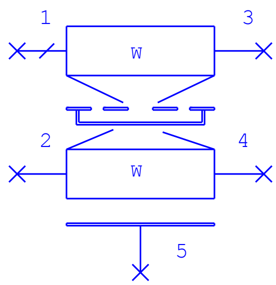
NOTE: The implementation of EM Quasi-Static models relies heavily on the involved numerical algorithms. This may lead to a noticeable increase in simulation time for schematics that employ many such models.

References

[1] M.B. Bazdar, A.R. Djordjevic, R.F. Harrington, and T.K. Sarkar, "Evaluation of quasi-static matrix parameters for multiconductor transmission lines using Galerkin's method," IEEE Trans. Microwave Theory Tech., vol. MTT-42, July 1994, pp. 1223-1228.

Coupled Shielded CMOS Line (EM Quasi-Static): SIGCMOS2

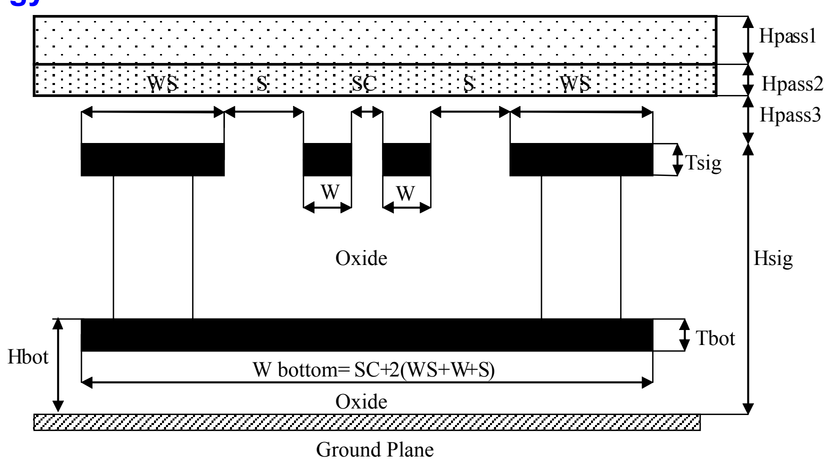
Symbol

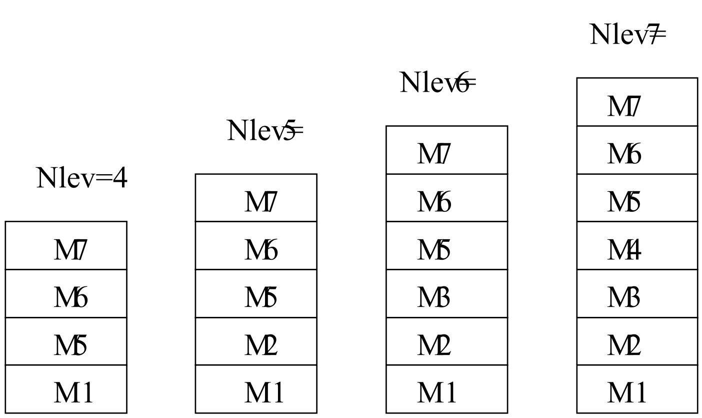


Summary

SIGCMOS2 models two symmetric edge coupled metallic strips of rectangular cross-section located inside the dielectric stack in presence of side and bottom shielding strips. Model allows to remove/install side shields. The structure of model parameters is tailored to standard CMOS process.

Topology





Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	TL1
W	Width of Signal Conductor	Length	5 microns
WS	Width of Side Shield Conductor	Length	5 microns
SC	Spacing between Signal Conductors	Length	5 microns
S	Spacing between Signal and Side Shield Conductors	Length	5 microns
IsSide	Side Shield is in Place No/Yes		Yes
SigLev	Signal Conductor is on level M7/M6/..M2		M7
BShldLev	Bottom Shield Conductor is on level M6/M5/..M1		M6
Nlev	Number of Metal Levels		7
L	Conductor Length	Length	1000 microns
Acc	Accuracy parameter		1
*ErOxide	Dielectric Constant of Silicon Oxide		4.1
*TdOxide	Loss Tangent of Silicon Oxide		0
*ErPass1	Dielectric Constant of First Top Passivation Layer		3.4
*TdPass1	Loss Tangent of First Top Passivation Layer		0
*ErPass2	Dielectric Constant of Second Top Passivation Layer		7.0
*TdPass2	Loss Tangent of Second Top Passivation Layer		0
*Hpass1	Height (Thickness) of First Top Passivation Layer	Length	2.0
*Hpass2	Height (Thickness) of Second Top Passivation Layer (Nitride)	Length	0.5
*Hpass3	Height (Thickness) of Third Top Passivation Layer (Oxide)	Length	1.0
*Hm7	Stack Height of M7Metal Top	Length	17.5
*Hm6	Stack Height of M6 Metal Top	Length	11.5
*Hm5	Stack Height of M5 Metal Top	Length	5.5
*Hm4	Stack Height of M4 Metal Top	Length	4.5
*Hm3	Stack Height of M3 Metal Top	Length	3.5
*Hm2	Stack Height of M2 Metal Top	Length	2.5
*Hm1	Stack Height of M1 Metal Top	Length	1.5
*Tm7	Thickness of M7Metal	Length	3.0
*Tm6	Thickness of M6 Metal	Length	3.0
*Tm5	Thickness of M5 Metal	Length	0.5
*Tm4	Thickness of M4 Metal	Length	0.5
*Tm3	Thickness of M3 Metal	Length	0.5
*Tm2	Thickness of M2 Metal	Length	0.5
*Tm1	Thickness of M1 Metal	Length	0.5
*Rho	Bulk Resistance of Metallization Relative to Gold		1.47

** indicates a secondary parameter*

Parameter Details

IsSide. If this switch is set to Yes side shielding conductors are in place; setting it to No removes side shields and leaves two signal conductors and bottom shield only.

SigLev. This switch allows user to attribute particular metal level to signal conductors and side shields. Placing conductors on certain level provides assignment of Hsig (see Topology) to preset value from available set of stack heights of metal levels; Tsig (see Topology) also gets assigned to predefined thickness value. Actual values of Hsig, Tsig may also be affected by parameter Nlev (see Nlev explanations below).

BShldLev. This switch allows user to attribute particular metal level to bottom shield conductor. Placing conductors on certain level provides assignment of Hbot (see Topology) to preset value from available set of stack heights of metal levels; Tbot (see Topology) also gets assigned to predefined thickness value. Actual values of Hsig, Tsigmay may also be affected by parameter Nlev (see Nlev explanations below). Model implements sanity checks and checks availability of selected metal levels.

Nlev. Nlev provides actual number of metal levels used in particular CMOS process. Model implies that maximum available number of metal layers is 7 and 4<=Nlev<=7 (content of metal stack for each value of Nlev is shown on Topology). It also implies that elevation of level M7 over M6 and elevation of M6 over M5 is constant for all values of Nlev (in most CMOS processes M7 and M6 are relatively thick metal layers used for power buses and passive high Q elements). Model also implies that all stack heights specified as model parameters are heights of top surfaces of each metal measured from ground plane (see Topology). Thus, changing Nlev changes actual value of Hm5 and actual values of Hm6, Hm7 because any setting Nlev<7 removes one or several metal levels below M5 from the stack (see Topology). Nlev also limits availability of metal level selections done via SigLev and BShldLev. For example, if Nlev=4 than only M7, M6, and M5 are available for SigLev and, correspondingly, only M6, M5, and M4 are available for BshldLev. The table below lists all feasible combinations of metal levels and demonstrates how actual Hm5, Hm6, and Hm7 are evaluated for each value of Nlev.

Nlev	Allowed SigLev	Allowed BShldLev	Actual Hm5	Actual Hm6	Actual Hm7
4	M7,M6,M5	M6,M5,M1	*Hm2	*Hm2+D6	*Hm2+D7
5	M7,M6,M5,M2	M6,M5,M2,M1	*Hm3	*Hm3+D6	*Hm3+D7
6	M7,M6,M5,M3,M2	M6,M5,M3,M2,M1	*Hm4	*Hm3+D6	*Hm4+D7
7	M7,M6,M5,M4,M3,M2	M6,M5,M4,M3,M2,M1	*Hm5	*Hm3+D6	*Hm5+D7

where D6=*Hm6 -*Hm5 and D7=*Hm7 -*Hm5. Asterisk refers to model parameters values.

ErPass1, TdPass1, ErPass2, TdPass2. Model implies that CMOS process protects stack of interconnects with three passivation layers. ErPass1, TdPass1, ErPass2, TdPass2 are material parameters of top two passivation layers counting from top. The third from top passivation layer is silicon oxide.

Hpass1, Hpass2, Hpass3. These parameters are thicknesses (not stack heights) of top three passivation layers.

Hm1..Hm7. CMOS process specific stack heights of metal levels. Hm1..Hm7 are referenced to the ground plane and measured up to the top of the corresponding metal layer.These parameters are secondary (hidden by default) and must be set to process specific values before modeling.

Tm1..Tm7. CMOS process specific metal thicknesses of metal levels. These parameters are secondary (hidden by default) and must be set to process specific values before modeling.

Rho. Model implies that the same metal is used at all metal levels. Rho is a ratio of metal bulk resistance to bulk resistance of gold. Default value is provided for aluminum.

Parameter Restrictions and Recommendations

1. Available selections of SigLev and BShldLev are limited to those feasible for the specified Nlev; these restriction may cause error messages if user selects SigLev /BSldLev incompatible with the available set of metal levels (see table on Topology).
2. If SC and/or S are very large compared to W or WS model issues a warning that for very large gaps between conductors model accuracy may degrade.
3. The following restrictions must be met: Tbot<0.95Hbot Tsig<0.95(Hsig-Hbot) If they are violated model issues corresponding error messages.
4. Ground plane location. Model does not account for substrate properties because it is implied that bottom shield effectively isolates signal conductor from substrate influence. Thus, ground plane might be placed at the oxide/substrate boundary

Implementation Details

Shielding conductors are implied to be connected by vias all the way along the line length and attached to a single port 5. This connection is ideal and vias are not modeled.

Model does not account for silicon substrate properties because it is implied that bottom shield effectively isolates signal conductor from substrate influence.

Width of bottom shield is affected by parameters WS, S even if IsSig=No. Wbottom is always SC+2(WS+W+S) so WS and S might be used to control Wbottom.

Model implementation is based on EM Quasi-Static technique described in [1] [10–354]. It accounts for losses in metal and in substrate dielectric. Dispersion is partly included

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

Recommendations for Use

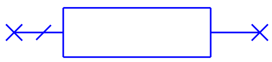
NOTE: The implementation of EM Quasi-Static models relies heavily on the involved numerical algorithms. This may lead to a noticeable increase in simulation time for schematics that employ many such models.

References

[1]. Bazdar, A.R. Djordjevic, R.F. Harrington, and T.K. Sarkar, "Evaluation of quasi-static matrix parameters for multiconductor transmission lines using Galerkin's method," IEEE Trans. Microwave Theory Tech., vol. MTT-42, July 1994, pp. 1223-1228.

(Obsolete) Single CMOS Microstrip Line Over Conducting Substrate (EM Based): SIGCMOSX

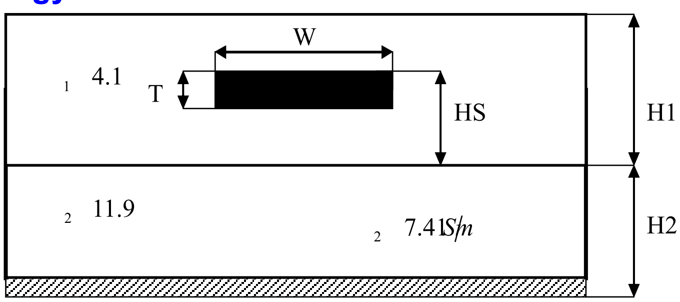
Symbol



Summary

There is no replacement for this OBSOLETE element. SIGCMOSX models section of single microstrip line arranged inside the layer of silicon oxide over the conducting silicon substrate. This model is constructed as an EM Based (Table Based Interpolation) model using the results of a 2D Finite Element Method analysis. Applicability of this model is limited to several predefined configurations.

Topology



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	TL1
W	Width of conductor	Length	W[a]
HS	Stack height of signal conductor above boundary oxide/substrate		W[a]
L	Line length	Length	L[a]
MSUB2	Substrate Definition	Text	MSUB21[b]
*AutoFill	AutoFill dataBase if not equal to 0		0

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See Microwave Office Layout Guide for details.

[b]Modify only if schematic contains multiple substrates. See the "Using Elements With Model Blocks" for details.

** indicates a secondary parameter*

Parameter Details

MSUB2. Two-layer substrate parameters are listed in MSUB2 model description. Note that all MSUB2 parameters are fixed parameters (see Implementation Details section below) and must be set by user in accordance with the table in the section Parameter Restrictions and Recommendations).

W. Parameter W is independent parameter, that is, in can be set to any value within the specified limits (see section Parameter Restrictions and Recommendations).

HS. Stack height of metal level the conductor belongs to is a fixed parameter. It must be set to one of the predefined values listed in the table in the section Parameter Restrictions and Recommendations.

AutoFill. Secondary parameter which in vast majority of EM Based models controls the initiation of the Autofill operation; in these models AutoFill allows the user to specify that the entire interpolation table should be filled automatically at the current values. To instigate this filling process, this parameter should be set to one (1). During normal operation, this parameter should be set to zero (0). Access to the hidden parameter can be accomplished by double-clicking on the schematic element. This model keeps Autofill just for consistency sake: the current version of SIGCMOSX does not accept AutoFill=1 so Autofill must always be equal to zero.

Parameter Restrictions and Recommendations

1. Substrate parameters T, Sig2 and model parameter HS must belong to one of five predefined sets presented in the following table:

Set No	T	HS	Sig2
1	0.31 micron	1.11 micron	7.41 S/m
2	0.31 micron	3.75 micron	7.41 S/m
3	1.25 micron	9 micron	7.41 S/m
4	4.0 micron	13.65 micron	7.41 S/m
5	4.0 micron	13.65 micron	0

2. The rest of substrate parameters must be set to the following values:
   - H1=15 microns
   - H2 = 6 microns
   - Er1 = 4.1
   - Er2 = 11.9
   - Tand1 = 0
   - Tand2 = 0
   - Rho = 1.19
3. Model parameter W should be within 1.5 microns <W<28.5 microns. Model extrapolates results outside this interval.
4. Frequency range is recommended 0.5 GHz<f<106 GHz. Model extrapolates results outside this interval.
5. No limitations on parameter L.

Implementation Details

This model is constructed as an EM-Based model (table Based Interpolation) using the results of a 2-D cross-sectional analysis based upon the Finite Elements Method. For a more detailed discussion of the EM-Based models see " EM-based Models (X-models) " (Element Catalog Supplementary Information).

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

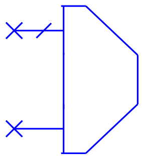
Recommendations for Use

NOTE: Default values of substrate parameters are not acceptable for this model; after placing the new substrate instance on the schematic user must set substrate parameters to the allowed values.

This model was developed under research performed at NI AWR Corporation. The full set of details of the implementation are considered proprietary in nature.

U-turn (180-degrees) Stripline Bend: SUBEND

Symbol

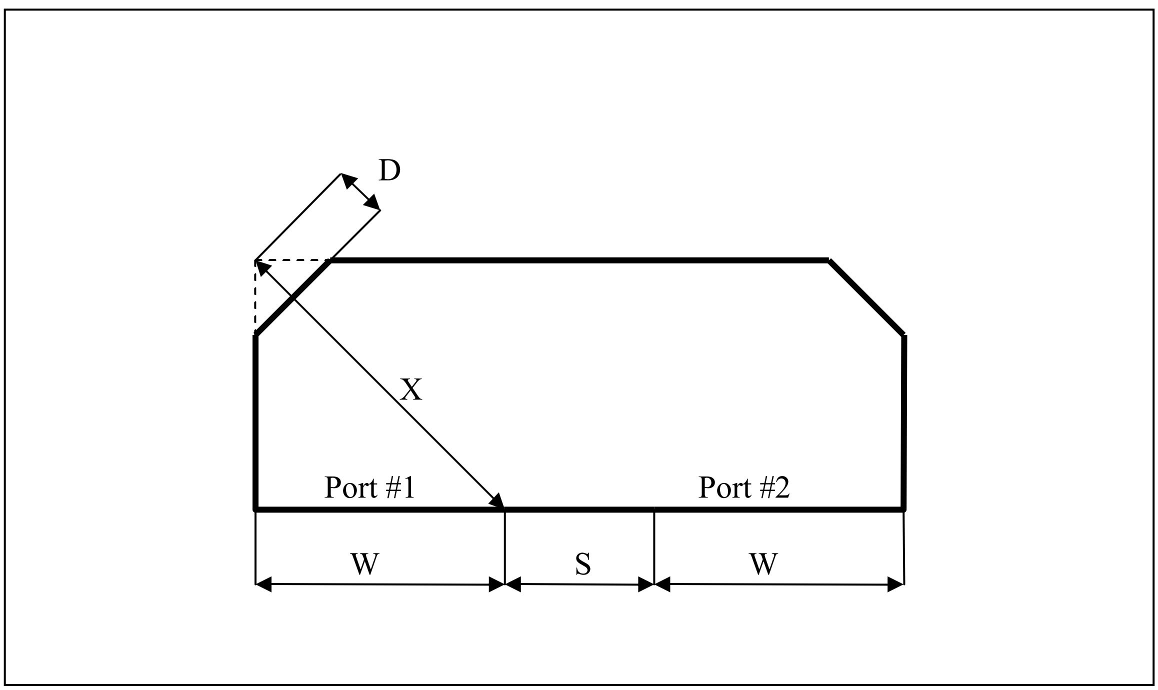


Summary

SUBEND models an interconnect between the ends of two adjacent coupled stripline transmission lines. It comprises two right-angle non-mitered stripline bends connected with a transmission line. Its primary use is as a component of hairpin stripline filters.

SUBEND$ is the corresponding intelligent cell (iCell). An iCell model is identical to its non-iCell equivalent with the following exception: Certain dimension-related parameters are not explicitly user-specified; rather, they are automatically and dynamically determined by the dimensions of the attached elements. See the "Intelligent Cells (iCells)" for a detailed discussion of how to use iCells, their advantages, and their limitations.

Topology



Parameters

Name	Description	Unit Type	Default
ID	Name	Text	TL1
W	Width of bends	Length	W[a]
S	Gap between bends	Length	S[a]
M	Bends miter (not used by this model)	Scalar	0
SSUB	Substrate definition	Text	SSUB#[b]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See the Microwave Office Layout Guide for details.

[b]Modify only if the schematic contains multiple substrates. See the "Using Elements With Model Blocks" for details.

Parameter Details

M. Miter M is defined as D/X (see "Topology") and its value cannot exceed 0.5. Note that current version of SUBEND does not support non-zero values of M (bends are non-mitered) and model always assumes M=0.

Parameter Restrictions and Recommendations

1. This model assumes that M=0 and ignores parameter M. However, the layout cell reflects changes in parameter M.
2. Parameter W refers to both ports of SUBEND; SUBEND is intended to connect striplines of equal width.

Implementation Details

This model is implemented as a series connection of right-angle non-mitered stripline bend, stripline, and another right-angle non-mitered stripline bend.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See "Cell Options Dialog Box: Layout Tab " for Cell Options dialog box **Layout** tab details.

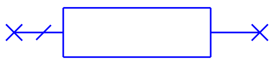
See "The Layout Process File (LPF)" for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

Recommendations for Use

This model is intended to provide a convenient mean for designing coupled line structures like hairpin filters. You can use it, however, with any type of striplines. The only limitation is a requirement of equal width of attached lines.

Coated Microstrip Line with Trapezoidal Cross-section and Solder Mask Overlay - 3D EM Cell: TCMLIN_EM

Symbol

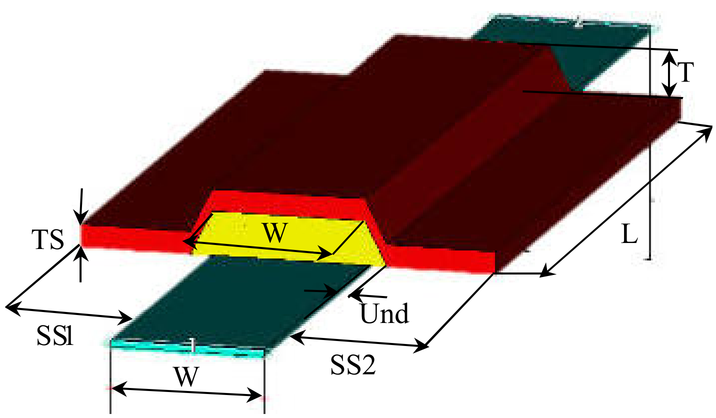


Summary

TCMLIN_EM is a model of a segment of microstrip line with an optional trapezoidal cross-section and optional dielectric coating representing a solder mask overlay. You can select the shape of cross-section, setting the sign and value of the cross-sectional undercut. You can use this model to explore the impact of etching/plating tolerances on the electrical performance of PCB/IC, simultaneously accounting for the effect of the solder mask. Multiple instances of TCMLIN_EM can be used to create a set of coated coupled microstrip lines.

This element is for use in 3D EM documents only. It is intended to create a 3D parameterized cell for a multiplate chip capacitor for 3D EM analysis. See "Using 3D EM Elements " for details on using 3D pCells with Analyst.

Topology



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	IC1
W	Width of conductor (top side)	Length	100 um
L	Length of conductor	Length	1000 um
T	Conductor thickness	Length	10 um
Und	Etching undercut	Length	15 um
Rho	Bulk resistance of conductor metal normalized to gold		1.0
TS	Solder mask thickness	Length	10 um
SS1	Solder mask left shoulder	Length	50 um
SS2	Solder mask right shoulder	Length	50 um
ErS	Relative dielectric constant of solder mask		3

Name	Description	Unit Type	Default
TandS	Loss tangent of solder mask		1e-3
IsSolderMask	Switch "No solder mask/Solder mask overlay"		No solder mask
*DIE_NAME	Filling dielectric name for 3D EM cell		""

** indicates a secondary parameter*

Parameter Details

W. Specifies the top side of a cross-sectional trapezoid (see the "Topology" section).

Und. Specifies how the bottom side of a cross-sectional trapezoid differs from the top side. If Und>0 (see the "Topology" section), then the bottom side equals W + 2*Und; if Und<0, then the bottom side equal W - 2*|Und| where |Und| is an absolute value.

SS1, SS2. Shoulders of solder mask overlay SS1 and SS2 are measured from the downward projection of the trapezoid top side onto the bottom side (see the "Topology" section).

Parameter Restrictions and Recommendations

1. Minimal allowable value of W, T, and TS is equal to 0.1 micron.
2. TCMLIN_EM sets the lower cap on SS1 and SS2 equal to Und if Und >0
3. You can use the DIE_NAME (hidden) parameter to assign different dielectric materials to multiple instances of TCMLIN_EM.

Layout

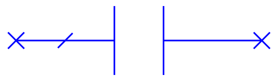
The layout for this cell has hard-coded model layers. When you first use this layout cell, a layer named "TCMLIN_EM" is added to your drawing layer and model layer list (if they are not already there). Using the model layer mapping, you can assign these layers to draw on any drawing layer.

3D EM Layout

This element has a layout cell specifically for Analyst 3D EM layouts. See "Using 3D EM Elements " for details on using 3D pCells with Analyst.

Thin Film Capacitor with Step Discontinuity (Closed Form): TFC2

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Name	Text	TFC1
W	Width	Length	W[a]
L	Length	Length	L[a]
T	Dielectric thickness	Length	T[a]
ER	Dielectric constant		Er[a]
RHO	Metal bulk resistivity normalized to gold (NOT USED IN THIS MODEL)		Rho[a]
TAND	Dielectric loss tangent		0
*W1	Conductor width @ node 1	Length	W[a]
*W2	Conductor width @ node 2	Length	W[a]
MSUB	Substrate definition	Text	MSUB#[b]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory.

[b]If only one MSUB is present in the schematic, this substrate is automatically used. If multiple MSUB substrate definitions are present, you must specify which to use.

** indicates a secondary parameter*

Implementation Details

This circuit component models a thin film capacitor.

The parameters W (width), L (length), and T (thickness) are dimensions entered in the default length units.

RHO is a metal bulk resistivity (normalized to gold - see MSUB) of capacitor plates (the current implementation of TFC2 ignores this parameter). Parameters ER and TAND apply to the dielectric between capacitor plates.

This model accounts for fringing capacitance.

Layout

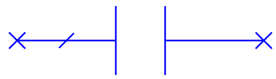
The layout for this cell has hard-coded model layers. When you first use this layout cell, layers named "Metal0", "Metal1", and "MIM" are added to your drawing layer and model layer list (if they are not already there). Using the model layer mapping, you can assign these layers to draw on any drawing layer. The full size of the capacitor is drawn on the "Metal0" layer. The "MIM" is drawn inset 3um on each side from "Metal0" and the "Metal1" layer is drawn inset 5um from "Metal0". You can control these offsets. If you want more control, you can use the TFCM or TFC elements.

References

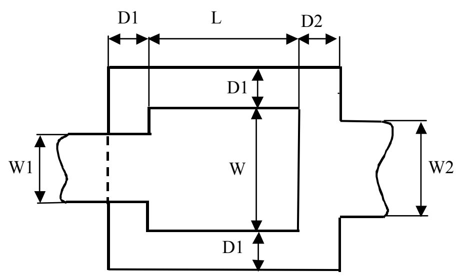
[1] Hurt, J. and Mohr, C. "A Computer-Aided Design System for Hybrid Circuits", *Components, Hybrids, and Manufacturing Technology*, Vol. 3, Issue 4, Dec.1980, pp. 525-535. See also, IEEE Trans. on Components, Packaging, and Manufacturing Technology, Parts A, B, C.

Thin Film Capacitor for MMIC: TFCM

Symbol



Topology



Parameters

Name	Description	Unit Type	Default
ID	Device ID	Text	TL1
W	Capacitor width	Length	W[a]
L	Capacitor length	Length	W[a]
C	Computed capacitance	Capacitance	0
W1	Conductor width @ node 1(bottom plate)	Length	W[a]
W2	Conductor width @ node 2(bottom plate)	Length	W[a]
D1	Top plate inset from bottom plate	Length	2um
D2	Top plate inset at bottom connection	Length	2um
CA	Capacitance per area (Farads/meter 2)		0
RS	Series resistance	Resistance	0
MSUB	Substrate definition	Text	MSUB[b]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory.

[b]If only one MSUB is present in the schematic, this substrate is automatically used. If multiple MSUB substrate definitions are present, the user must specify.

Implementation Details

This circuit component models a thin film MMIC capacitor. The parameters W, L, W1, W2, D1, D2 are dimensions entered in the default length units. CA is a capacitance per area, must be entered only in Farads/sq. meter. RS is a series resistance, must be entered in resistance units. The parameter MSUB specifies the substrate element, which defines additional cross sectional parameters and substrate characteristics. If blank, a default is used. This model accounts for the width steps at the bottom and top connections and phase shift due to the component's final electrical length. This component features a special synthesis ability. It means that the computed capacitance is permanently displayed on the schematic and is automatically updated in response to modification of affecting parameters.

Layout

The TFCM layout object obtains its drawing layer information from the LPF file that is loaded for the project. The drawing layers are defined within a block that begins with $CAP_DEFINE_BEGIN and $CAP_DEFINE_END, respectively (see "The Layout Process File (LPF)" for more information on working with LPF files). If this block is not defined, the capacitor is drawn on the Default layer (if the Default layer is not defined, it is drawn on the first layer that is defined in Layer Setup).

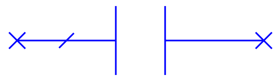
When SNAME is specified, the structure is identified in a $STRUCT_TYPE_BEGIN section of the LPF file. When SNAME is empty, the TFCMP layout cell uses the $CAP_DEFINE_BEGIN section of the LPF file, as previous.

When defining $CAP_DEFINE_BEGIN or $STRUCT_TYPE_BEGIN in the LPF file, there are flags that control the layout cell. The following are flags for control of faces and area pins for each layer:

- Flag 0: default behavior, no change
- Flag 1: face 1 on layer
- Flag 2: face 2 on layer
- Flag 3: face 1 and 2 on layer
- Flag 5: area pin on layer, face 1
- Flag 6: area pin on layer, face 2

Polygon Thin Film Capacitor for MMIC (Closed Form): TFCMP

Symbol



Summary

TFCMP models a thin film MMIC capacitor. The shape of the capacitor can be any non-self-intersecting polygon.

Parameters

Name	Description	Unit Type	Default
ID	Device ID	Text	TL1
C	Computed capacitance (read-only)	Capacitance	0.0 pF
CA	Capacitance per area (F/m^2)		0.0
RS	Series resistance	Resistance	0.0 ohm
MSUB	Substrate definition	Text	MSUB[a]
*XY	XY-point array	Length	W[b]
*SNAME	Structure name from LPF file. Empty string uses CAP_DEFINE_BEGIN from LPF file.		
CP	Capacitance per perimeter length (Farads/meter). Empty value means 0.0.		0.0

[a]Modify only if the schematic contains multiple substrates. See “Using Elements With Model Blocks” for details.

[b]User-modifiable default. Modify by editing the *DEFAULT.LPF* file in the root installation directory. See “Default Values” for details.

** indicates a secondary parameter*

Parameter Restrictions and Recommendations

1. The polygon that defines the shape of the capacitor must not be self-intersecting.
2. SNAME is for layout and has no effect on the electrical performance.

Implementation Details

The capacitance is computed as a sum of two capacitances. The first capacitance is obtained by multiplying the value of the CA parameter (in Farads per square meter) by the area of the polygon; the second capacitance is obtained by multiplying the value of the CP parameter (in Farads per meter) by the perimeter of the polygon. The computed capacitance is permanently displayed on the schematic as a read-only parameter and is automatically updated in response to modification of the CA and CP parameters or the polygon shape in the layout.

Layout

The TFCMP layout object obtains its drawing layer information from the LPF file that is loaded for the project. The drawing layers are defined within a block that begins with $CAP_DEFINE_BEGIN and $CAP_DEFINE_END, respectively (see “The Layout Process File (LPF)” for more information on working with LPF files). If this block is not defined, the capacitor is drawn on the Default layer (if the Default layer is not defined, it is drawn on the first layer that is defined in Layer Setup).

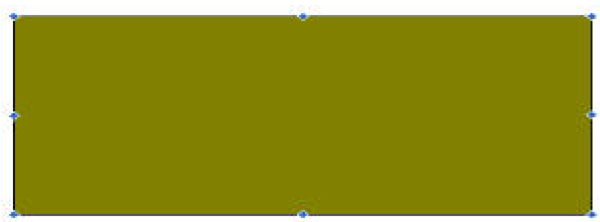
When SNAME is specified, the structure is identified in a $STRUCT_TYPE_BEGIN section of the LPF file. When SNAME is empty, the TFCMP layout cell uses the $CAP_DEFINE_BEGIN section of the LPF file, as previous.

When defining $CAP_DEFINE_BEGIN or $STRUCT_TYPE_BEGIN in the LPF file, there are flags that control the layout cell. The following are flags for control of faces and area pins for each layer:

- Flag 0: default behavior, no change
- Flag 1: face 1 on layer
- Flag 2: face 2 on layer
- Flag 3: face 1 and 2 on layer
- Flag 5: area pin on layer, face 1
- Flag 6: area pin on layer, face 2

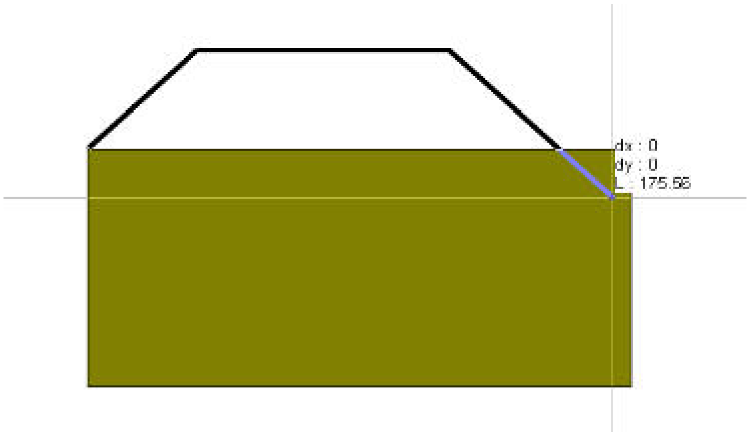
The shape of the polygon is modified in the layout by graphically drawing the outline of the polygon using the mouse. The following figures illustrate how to reshape a rectangular capacitor into a hexagon.

Double-click the capacitor layout object to activate the black diamond drag handles that allow you to manipulate the polygon.

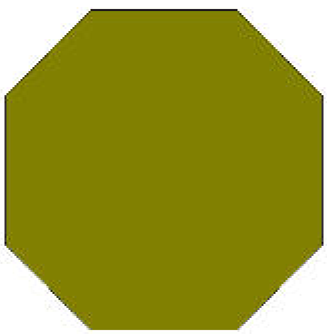


Double-click on one of the diamonds at a vertex to activate the reshape command.

Use the mouse to trace out the desired shape by clicking at each vertex.



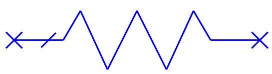
Double-click to finish the polygon.



Connecting objects can connect to any side of the polygon.

(Obsolete) Thin Film Resistor Distributed Lossy Line Model (Closed Form): TFR

Symbol



Summary

This element is OBSOLETE and is replaced by the Thin Film Resistor Distributed Lossy Line Model (Closed Form) (TFR2) element.

Parameters

Name	Description	Unit Type	Default
ID	Name	Text	TL1
W	Resistor width	Length	W[a]um
L	Resistor length	Length	L[a]um
RS	Sheet (surface) resistivity		50
F	Frequency for scaling sheet resistivity	Frequency	0
MSUB	Substrate definition	Text	MSUB[b]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory.

[b]If only one MSUB is present in the schematic, this substrate is automatically used. If multiple MSUB substrate definitions are present, you must specify which to use.

Implementation Details

This circuit component models a thin film resistor as a lossy microstrip line with the complex characteristic impedance.

The parameters W (width), and L (length) are dimensions entered in the default length units. The parameter MSUB specifies microstrip substrate element, which defines additional cross sectional parameters of the transmission line. If blank, a default is used.

If parameter F is set to zero, then sheet resistance RS stays constant with respect to frequency. This holds true if thickness of the resistive film that makes resistor body is less than skin depth.

If thickness of resistive film exceeds skin depth the sheet resistance RS may be scaled with frequency in accordance with the formula:

$$\mathrm{RS}(f)) = \mathrm{RS}(F)\sqrt{\frac{f}{F}}$$

User can activate this scaling by setting $F \neq 0$. In this case F should be evaluated by user in advance as

$$F_{GHz} = \frac{25.33}{(\sigma \cdot 10^{-7}) \cdot t_{microns}^2}$$

where σ - bulk conductivity of the resistive film in S/m and t - thickness of the resistive film in microns. If F is set in accordance with this formula scaling of RS is implemented only for F >f; for f<F RS stays constant.

Note that you must enter the actual value of parameter F in default frequency units.

For most cases frequency F is very large (for example, hundreds gigahertz) and setting F=0 is the best choice.

The component accounts for losses in metal and in substrate dielectric. Dispersion is included.

Layout

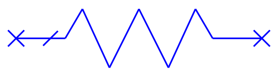
The layout for this cell has hard-coded model layers. When you first use this layout cell, layers named "NiCr" and "Metal1" are added to your drawing layer and model layer list (if they are not already there). Using the model layer mapping, you can assign these layers to draw on any drawing layer. The resistor material is drawn on the "NiCr" model layer and the resistor pad is drawn on the "Metal1" model layer. The metal overlap of the resistor is hard-coded as 3 um and the resistor material extends 3 um under the metal pad, therefore, the metal pad length is 6 um and the width is the width of the resistor plus 6 um.

References

[1] E. Hammerstad and O. Jensen, "Accurate Models for Microstrip Computer-Aided Design," IEEE MTT-S International Microwave Symposium Digest, 1980, pp. 407-409.

Thin Film Resistor Distributed Lossy Line Model (Closed Form): TFR2

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Name	Text	TL1
W	Resistor width	Length	W[a]um
L	Resistor length	Length	L[a] um
RS	Sheet (surface) resistivity		50
F	Frequency for scaling sheet resistivity	Frequency	0
MSUB	Substrate definition	Text	MSUB[b]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory.

[b]If only one MSUB is present in the schematic, this substrate is automatically used. If multiple MSUB substrate definitions are present, you must specify which to use.

Implementation Details

This circuit component models a thin film resistor as a lossy microstrip line with the complex characteristic impedance. TFR2 is intended to replace the TFR model. The TFR model has a pad overlap of the resistive material by 3um on each side, so the layout draws 6um smaller than actually specified. This is a small error in PCB or other similar processes, but the error is large for integrated circuit processes. The new model draws the resistive material the exact length specified in the model. Additional layout only parameters control how the pad for the resistors draw.

The parameters W (width), and L (length) are dimensions entered in the default length units. The MSUB parameter specifies microstrip substrate element, which defines additional cross-sectional parameters of the transmission line. If blank, a default is used.

If parameter F is set to zero, then sheet resistance RS stays constant with respect to frequency. This is true if the thickness of the resistive film that makes the resistor body is less than skin depth.

If the thickness of the resistive film exceeds skin depth, the sheet resistance RS may be scaled with frequency in accordance with the formula:

$$\mathrm{RS}(f)) = \mathrm{RS}(F)\sqrt{\frac{f}{F}}$$

You can activate this scaling by setting F≠0. In this case you should evaluate F in advance as

$$F_{GHz} = \frac{25.33}{(\sigma \cdot 10^{-7}) \cdot t^2_{microns}}$$

where σ - bulk conductivity of the resistive film in S/m and t - thickness of the resistive film in microns. If F is set in accordance with this formula, scaling of RS is implemented only for F >f; for f<F RS stays constant.

Note that you must enter the actual value of parameter F in default frequency units.

Generally, frequency F is very large (for example, hundreds of gigahertz) and setting F=0 is the best choice.

TFR2 accounts for losses in metal and in substrate dielectric. Dispersion is included.

Layout

The layout for this cell has its model layers hard-coded. To draw the layout, you must add layers named "NiCr" and "Metal1" to your model layer list (if they don't already exist). After these are defined in the model layer, you can assign them to draw on any draw layer. The resistor material is drawn on the "NiCr" model layer and the resistor pad is drawn on the "Metal1" model layer.

For more information on making new model layers, see "Layer Mapping".

This model has several layout only parameters that control how the pad draws. You access these parameters by selecting the item in the layout, right-clicking and choosing **Shape Properties**, then clicking the **Parameters** tab.

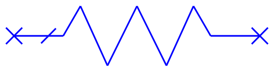
Name	Description	Unit Type	Default
PADL	Total length of the overlap pad.	Layout	6um
PADEXT	Pad width extension, draws this much wider than the W of the resistor on each side.	Layout	3um
PADOV	Total length that resistor material overlaps the pad.	Layout	3um

References

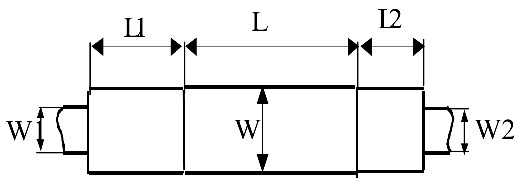
[1] E. Hammerstad and O. Jensen, "Accurate Models for Microstrip Computer-Aided Design," IEEE MTT-S International Microwave Symposium Digest, 1980, pp. 407-409.

Thin Film Resistor for MMIC (Closed Form): TFRM

Symbol



Topology



Parameters

Name	Description	Unit Type	Default
ID	Device ID	Text	TL1
W	Resistor width	Length	W[a]
L	Resistor length	Length	L[a] um
R	Computed resistance	Resistance	0
W1	Conductor width @ node 1	Length	W[a]
W2	Conductor width @ node 2	Length	W[a]
L1	Resistor input pad length	Length	W[a]
L2	Resistor output pad length	Length	W[a]
RS	Sheet resistivity (ohms/square)		1
RC	Contact resistance per width (ohms*meter)		0.002
F	Frequency for scaling resistance	Frequency	0
MSUB	Substrate definition	Text	MSUB[b]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory.

[b]If only one MSUB is present in the schematic, this substrate is automatically used. If multiple MSUB substrate definitions are present, you must specify which to use.

Implementation Details

This circuit component models a thin film MMIC resistor. A thin film resistor is treated as a lossy microstrip line with the complex characteristic impedance.

The W, L, L1, and L2 parameters are dimensions entered in the default length units. W1 and W2 are the width of the lines connecting to this model. These parameters do not change the width of the resistor pads; the pads are modeled with the W of the resistor. RS is a sheet resistance of resistor body and must be entered in ohms per square units. RC is an ohmic resistance per width of contact pads and must be entered in ohms*meters units.

If the F parameter is set to zero, then sheet resistance RS stays constant with respect to frequency. This is true if the thickness of the resistive film that makes the resistor body is less than skin depth.

If the thickness of the resistive film exceeds the skin depth, the sheet resistance RS may be scaled with frequency in accordance with the formula:

$$\mathrm{RS}(f)) = \mathrm{RS}(F)\sqrt{\frac{f}{F}}$$

You can activate this scaling by setting F≠0. In this case, you should evaluate F in advance as

$$F_{GHz} = \frac{25.33}{(\sigma \cdot 10^{-7}) \cdot t^2_{microns}}$$

where σ - bulk conductivity of the resistive film in S/m and t - thickness of the resistive film in microns. If F is set in accordance with this formula, scaling of RS is implemented only for F >f; for f<F RS stays constant.

Note that you must enter the actual value of the F parameter in default frequency units.

Typically, frequency F is very large (for example, hundreds of gigahertz) and setting F=0 is the best choice.

The MSUB parameter specifies the substrate element, which defines additional cross sectional parameters and substrate characteristics. If blank, a default is used. This model accounts for the width steps at the connections, phase shift due to the component's final electrical length, losses in metal and in substrate dielectric. Dispersion is included. This component features a special synthesis ability; the computed DC resistance is permanently displayed on the schematic and is automatically updated in response to the modification of affecting parameters.

Layout

The drawing layers for this element are controlled by settings in the .LPF file loaded into the program. Since this element is used for MMIC processing, two different types of layouts are possible depending on the value of sheet resistance (RS) set in the model. For RS less than 50, the layout layers are specified by the following block in the .LPF file:

```
$STRUCT_TYPE_BEGIN "Tan Resistor"
     ! -> Layer    offset         minWidth       flags
     "Tantalum"    -2e-06         4e-06          1
     "Thick Metal"      0         5e-06          0
$STRUCT_TYPE_END
```

Any layers and offsets can be set for this structure, but it must have the name "Tan Resistor".

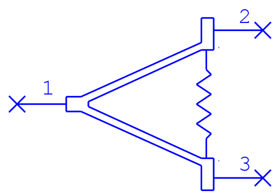
For RS greater than or equal to 50, the layout layers are specified by the following block in the .LPF file:

```
$STRUCT_TYPE_BEGIN "Mesa Resistor"
     ! -> Layer    offset         minWidth       flags
     "Mesa"        -6e-06         5e-06          0
     "Source Drain"0              5e-06          0
     "Thick Metal" -2e-06         5e-06          0
     "Nitride Etch"-4e-06         4e-06          0
$STRUCT_TYPE_END
```

Any layers and offset can be set for this structure, but it must be named "Mesa Resistor"

Wilkinson Equal Power Split Divider Chip Resistor Aggregate: WILKE1

Symbol



Summary

WILKE1 models Wilkinson equal power split divider/combiner based on microstrip line models (closed form). Model implies that load resistor is frequency-independent chip resistor.

Parameters

Name	Description	Unit Type	Default
ID	Name	Text	TL1
W	Feeding line width at ports 1,2, and 3	Length	W[a]
W1	Width of branched lines	Length	W[a]
L1	Length of branched lines	Length	L[a]
R	Resistor impedance	ohm	100
MSUB	Substrate definition	Text	MSUB1[b]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See the Microwave Office Layout Guide for details.

[b] Modify only if schematic contains multiple substrates. See "Using Elements With Model Blocks" for details.

Parameter Restrictions and Recommendations

1. Choice of parameter W1 should provide characteristic impedance of branched lines that is √2 times greater than the characteristic impedance of the feeding lines
2. Length of branched lines L1 should be about quarter wavelength of microstrip line having width W1.
3. Impedance of load resistor R should be twice the characteristic impedance of feeding lines at ports 1,2, and 3

Implementation Details

WILKE1 uses closed form models MLIN (microstrip line) and MTEE (T-junction of microstrip lines), so all limitations of these models are applicable to WILKE1.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Recommendations for Use

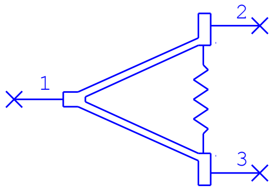
To provide equal power split, matching and maximum isolation between branched lines parameters must comply with the above recommendations. However, user may set parameters to values of his choice or use them for circuit optimization.

References

[1] Bahl,I.J., and Bhartia P., Microwave solid state circuit design, John Wiley & Sons, N.Y, 1988, pp. 214-222

Wilkinson Equal Power Split Divider Thin Film Resistor Aggregate: WILKE2

Symbol



Summary

WILKE2 models equal Wilkinson power split divider/combiner based on microstrip line models (closed form). Model implies that load resistor is frequency-dependent thin film resistor.

Parameters

Name	Description	Unit Type	Default
ID	Name	Text	TL1
W	Feeding line width at ports 1,2, and 3	Length	W[a]
W1	Width of branched lines	Length	W[a]
L1	Length of branched lines	Length	L[a]
WR	Thin film resistor width	Length	W[a]
LR	Thin film resistor length	Length	L[a]
RS	Sheet (surface) resistivity of thin film resistor	ohm/square	100
FR	Frequency for scaling sheet resistivity	Frequency	0
MSUB	Substrate definition	Text	MSUB1[b]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See the Microwave Office Layout Guide for details.

[b]Modify only if schematic contains multiple substrates. See "Using Elements With Model Blocks" for details.

Parameter Restrictions and Recommendations

1. Choice of parameter W1 should provide characteristic impedance of branched lines that is √2 times greater than the characteristic impedance of the feeding lines
2. Length of branched lines L1 should be about quarter wavelength of microstrip line having width W1.
3. Impedance of load resistor R should be twice the characteristic impedance of feeding lines at ports 1,2, and 3
4. See recommendations how to choose frequency for scaling sheet resistivity in TFR model description. For most cases setting F=0 is the best choice.

Implementation Details

WILKE2 uses closed form models MLIN (microstrip line) and MTEE (T-junction of microstrip lines), so all limitations of these models are applicable to WILKE2.

Load resistor is modeled with TFR model. This model represents thin film resistor as a lossy microstrip line with the complex characteristic impedance.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Recommendations for Use

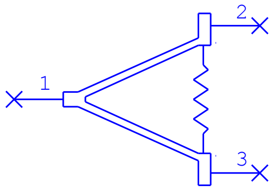
To provide equal power split, matching and maximum isolation between branched lines parameters must comply with the above recommendations. However, user may set parameters to values of his choice or use them for circuit optimization.

References

[1] Bahl,I.J., and Bhartia P., Microwave solid state circuit design, John Wiley & Sons, N.Y, 1988, pp. 214-222

Wilkinson Unequal Power Split Divider Chip Resistor Aggregate: WILKN1

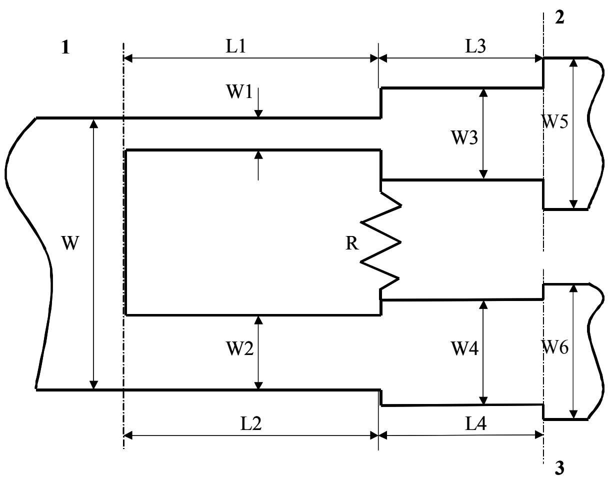
Symbol



Summary

WILKN1 models Wilkinson unequal power split divider/combiner based on microstrip line models (closed form). Model implies that load resistor is frequency-independent chip resistor.

Topology



Parameters

Name	Description	Unit Type	Default
ID	Name	Text	TL1
W	Feeding line width at port 1	Length	W[a]
W1	Width of branched line to port 2	Length	W[a]
W2	Width of branched line to port 3	Length	W[a]
W3	Width of matching line at port 2	Length	W[a]
W4	Width of matching line at port 3	Length	W[a]
W5	Feeding line width ap port 2	Length	W[a]
W6	Feeding line width ap port 3	Length	W[a]
L1	Length of branched line to port 2	Length	L[a]

Name	Description	Unit Type	Default
L2	Length of branched lIne to port 3	Length	L[a]
L3	Length of matching line to port 2	Length	L[a]
L4	Length of matching line to port 3	Length	L[a]
R	Resistor impedance	ohm	50
MSUB	Substrate definition	Text	MSUB1[b]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See the Microwave Office Layout Guide for details.

[b]Modify only if schematic contains multiple substrates. See "Using Elements With Model Blocks" for details.

Parameter Restrictions and Recommendations

1. Choice of parameters L1, L2, L3, and L4 should provide quarter wavelength of microstrip lines having width W1, W2, W3, and W4.

Implementation Details

WILKN1 uses closed form models MLIN (microstrip line) and MTEE (T-junction of microstrip lines), so all limitations of these models are applicable to WILKN1.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Recommendations for Use

WILKN1 incorporates matching lines L3, L4 and width steps at all ports (see Topology). To provide additional matching at port 1 user may choose to add matching line and width step (e.g. MLIN and MSTEP models) before port 1.

Feeding lines at ports may have different characteristic impedances. This is provided by proper choice of width parameters W, W5, and W6 (see Topology).

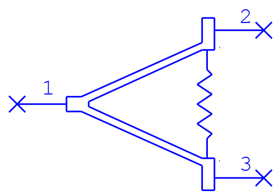
To ensure specified power split, matching and maximum isolation between branched lines parameters may be derived in accordance with recommendations in Reference [1] [10–380]. However, user may set parameters to values of his choice or use them for circuit optimization.

References

[1] Bahl,I.J., and Bhartia P., Microwave solid state circuit design, John Wiley & Sons, N.Y, 1988, pp. 214-222

Wilkinson Unequal Power Split Divider Thin Film Resistor Aggregate: WILKN2

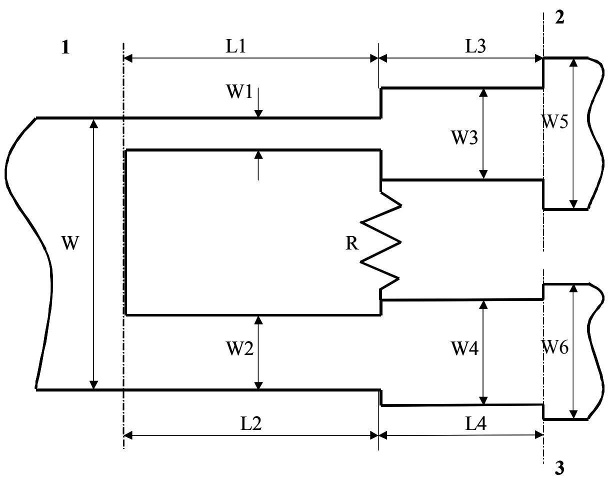
Symbol



Summary

WILKN2 models Wilkinson unequal power split divider/combiner based on microstrip line models (closed form). Model implies that load resistor is frequency-dependent thin film resistor.

Topology



Parameters

Name	Description	Unit Type	Default
ID	Name	Text	TL1
W	Feeding line width at port 1	Length	W[a]
W1	Width of branched line to port 2	Length	W[a]
W2	Width of branched line to port 3	Length	W[a]
W3	Width of matching line at port2	Length	W[a]
W4	Width of matching line at port 3	Length	W[a]
W5	Feeding line width at port 2	Length	W[a]
W6	Feeding line width a port 3	Length	W[a]
L1	Length of branched line to port 2	Length	L[a]

Name	Description	Unit Type	Default
L2	Length of branched lIne to port 3	Length	L[a]
L3	Length of matching line to port 2	Length	L[a]
L4	Length of matching line to port 3	Length	L[a]
WR	Thin film resistor width	Length	W[a]
LR	Thin film resistor length	Length	L[a]
RS	Sheet (surface) resistivity of thin film resistor	ohm/square	100
FR	Frequency for scaling sheet resistivity	Frequency	0
MSUB	Substrate definition	Text	MSUB1[b]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See the Microwave Office Layout Guide for details.

[b]Modify only if schematic contains multiple substrates. See "Using Elements With Model Blocks" for details.

Parameter Restrictions and Recommendations

1. Choice of parameters L1, L2, L3, and L4 should provide quarter wavelength of microstrip lines having width W1, W2, W3, and W4.
2. See recommendations how to choose frequency for scaling sheet resistivity in TFR model description. For most cases setting F=0 is the best choice.

Implementation Details

WILKN2 uses closed form models MLIN (microstrip line) and MTEE (T-junction of microstrip lines), so all limitations of these models are applicable to WILKN2. Load resistor is modeled with TFR model. This model represents thin film resistor as a lossy microstrip line with the complex characteristic impedance.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Recommendations for Use

WILKN2 incorporates matching lines L3, L4 and width steps at all ports (see Topology). To provide additional matching at port 1 user may choose to add matching line and width steps (e.g. MLIN and MSTEP models) in front of port 1.

Feeding lines at ports may have different characteristic impedances. This is provided by proper choice of width parameters W, W5, and W6 (see Topology).

To ensure specified power split, matching and maximum isolation between branched lines parameters may be derived in accordance with the recommendations in Reference [1] [10–382]. However, user may set parameters to values of his choice or use them for circuit optimization.

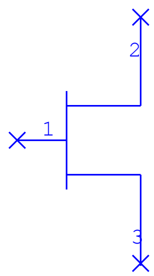
References

[1] Bahl,I.J., and Bhartia P., Microwave solid state circuit design, John Wiley & Sons, N.Y, 1988, pp. 214-222

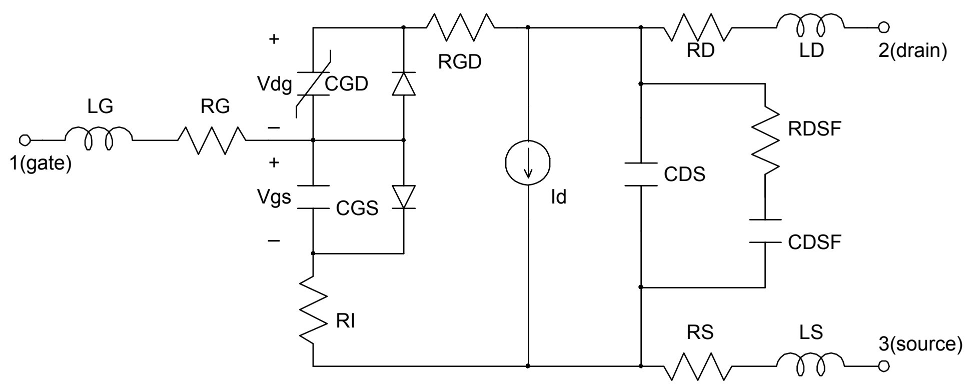
Nonlinear

Angelov HEMT Model: ANGELOV

Symbol



Equivalent Circuit



Parameters

Name	Description	Unit Type	Default
ID	Device ID	Text	AF1
P1	I/V polynomial coefficient		1
P2	I/V polynomial coefficient		0
P3	I/V polynomial coefficient		0
B1	Soft breakdown polynomial coefficient		0
B2	Soft breakdown polynomial coefficient		0
VDB	Subthreshold voltage parameter	Voltage	1 V
VPK	Gate voltage at peak Gm	Voltage	-1 V
ALPHA	Drain I/V knee parameter		1
LAMBDA	Drain-source resistance parameter		0
IPK	Current at peak Gm	Current	50 mA
TAU	Gate-drain time delay	Time	0 ns
CGS0	Gate-source capacitance parameter	Capacitance	0 pF

Name	Description	Unit Type	Default
P10	Gate-source capacitance polynomial coef.		0
P11	Gate-source capacitance polynomial coef.		0
P20	Gate-source capacitance polynomial coef.		0
P21	Gate-source capacitance polynomial coef.		0
CGD0	Gate-drain capacitance parameter	Capacitance	0 pF
P30	Gate-drain capacitance polynomial coef.		0
P31	Gate-drain capacitance polynomial coef.		0
P40	Gate-drain capacitance polynomial coef.		0
P41	Gate-drain capacitance polynomial coef.		0
P400	Gate-drain capacitance polynomial coef.		0
ISG	Gate diode current parameter	Current	1e-17 mA
NG	Gate diode ideality factor		1
BVG	Gate breakdown voltage	Voltage	1e+06 V
RG	Gate resistance	Resistance	1 ohm
RS	Source resistance	Resistance	1 ohm
RI	Intrinsic resistance	Resistance	1 ohm
RD	Drain resistance	Resistance	1 ohm
RGD	Gate-drain resistance	Resistance	0.01 ohm
RDSF	RF drain-source resistance	Resistance	300 ohm
CDSF	Capacitance that determines Rds break frequency	Capacitance	0 pF
CDS	Drain-source capacitance	Capacitance	0 pF
TNOM	Temperature	Temperature	27 DegC
LS	Source inductance	Inductance	0 nH
LG	Gate inductance	Inductance	0 nH
LD	Drain inductance	Inductance	0 nH
AFAC	Gate width scale factor		1
NFING	Number of fingers scale factor		1

Implementation Details

I/V Characteristic

$$I_d = I_{\mathrm{pk}} \cdot \mathrm{AFAC} \cdot (1 + \tanh(\Psi))(1 + \lambda \cdot V_d)\tanh(\alpha \cdot V_d)$$

where

$$\Psi = P1 \cdot V_{\mathrm{gf}} + P2 \cdot V_{\mathrm{gF}}^2 + P3 \cdot V_{\mathrm{gF}}^3$$

Breakdown is modeled by modifying V_g as follows:

$$V_{\mathrm{gr}} = V_g + B1(V_d - V_{\mathrm{db}}) + B2(V_d - V_{\mathrm{db}})^2 - \mathrm{VPK}$$

The B1 and B2 terms are not in the original model. They promote a simpler expression for soft breakdown.

Capacitances

$$C_{\mathrm{gs}} = \mathrm{AFAC} \cdot \mathrm{CGS0}(1 + \tanh(\varphi_1))(1 + \tanh(\varphi_2))$$

$$\varphi_1 = P10 + P11 \cdot V_g$$

$$\varphi_2 = P20 + P21 \cdot V_{\mathrm{dg}}$$

$$C_{\mathrm{gd}} = \mathrm{AFAC} \cdot \mathrm{CGD0}(1 + \tanh(\varphi_3))(1 - P400 \cdot \tanh(\varphi_4))$$

$$\varphi_3 = P30 + P31 \cdot V_g$$

$$\varphi_4 = P40 + P41 \cdot V_{\mathrm{dg}}$$

Charge Functions

$$Q_{\mathrm{gs}} = \mathrm{AFAC} \cdot \mathrm{CGS0}(1 + \tanh(\varphi_2))[V_g + \frac{1}{P11}\log(\cosh(\varphi_1))]$$

$$Q_{\mathrm{gd}} = \mathrm{AFAC} \cdot \mathrm{CGD0}(1 + \tanh(\varphi_3))[V_{\mathrm{dg}} - \frac{P400}{P41}\log(\cosh(\varphi_4))]$$

Gate-to-drain and gate-to-source diodes are modeled by PNIV (diode junction) elements.

Parameter Scaling

$$\mathrm{RGG} = \frac{\mathrm{RG} \cdot \mathrm{AFAC}}{\mathrm{NFING}^2}$$

$$\mathrm{RDD} = \frac{\mathrm{RD}}{\mathrm{AFAC}}$$

$$\mathrm{RSS} = \frac{\mathrm{RS}}{\mathrm{AFAC}}$$

$$\mathrm{RII} = \frac{\mathrm{RI}}{\mathrm{AFAC}}$$

$$\mathrm{RDSFF} = \frac{\mathrm{RDSF}}{\mathrm{AFAC}}$$

$$\mathrm{CDSS} = \mathrm{CDS} \cdot \mathrm{AFAC}$$

$$\mathrm{CDSFF} = \mathrm{CDSF} \cdot \mathrm{AFAC}$$

NOTES:

1. V_{dg} and V_{gs} polarity are chosen to be consistent with Angelov's report. (See References.)

2. The parasitic gate, source, and drain inductances are not scaled. These represent the inductive parasitics of pads, on-chip interconnects, and bond wires which, in general, do not scale in any predictable way.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

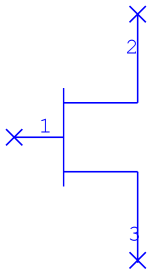
References

[1] I. Angelov, "Chalmers Nonlinear HEMT and MESFET Model Extraction Procedure, Part A," Chalmers University of Technology, Dept. of Microwave Technology, Report no. 25, July, 1996.

[2] I. Angelov, "Chalmers Nonlinear HEMT and MESFET Model Extraction Procedure, Part B," Chalmers University of Technology, Dept. of Microwave Technology, Report no. 26, November, 1996.

Second Generation Angelov (Chalmers) Model: ANGELOV2

Symbol

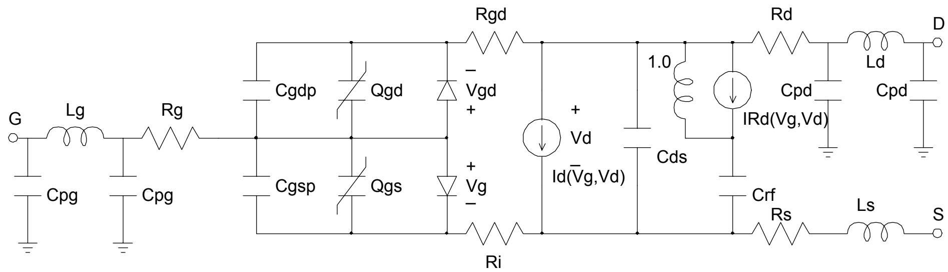


Summary

This is a further development of the well known Angelov/Zirath/Rorsman, or Chalmers, model. It is especially good for use with HEMT devices, but is also useful for MESFETs. Angelov2 includes improved capacitance and breakdown modeling, as well as modeling of thermal effects.

Although this model is similar in many ways to the original Angelov model, it is not identical, and, in general, parameters for the earlier model cannot be used in this one.

Equivalent Circuit



Parameters

Parameter	Description	Units	Default Value
ID	Device ID	Text	AF1
*IPK	Current at peak Gm	ma	50
*P1	I/V polynomial coefficient		1.0
*P2	I/V polynomial coefficient		0
*P3	I/V polynomial coefficient		0
*P4	I/V polynomial coefficient		0
*P5	I/V polynomial coefficient		0
*P6	I/V polynomial coefficient		0
*B1	P1 term		0

Parameter	Description	Units	Default Value
*B2	P1 term		3.0
*VPKS	Gate voltage at peak Gm in sat	Voltage	-0.2 V
*VPK0	Gate voltage at peak Gm near 0 Vds		-0.4 V
*ALPHR	Drain I/V knee parameter		2.0
*ALPHS	Drain I/V knee parameter at saturation		0
*LAMBDA	Drain-source resistance parameter		0
*LAMSB	Surface breakdown parameter		0
*VTR	Threshold voltage for breakdown	Voltage	5.0
*VSB2	Surface breakdown parameter		0.0
*TAU	Gate-drain time delay	Time	0
*CGS0	Gate-source capacitance parameter	Capacitance	0
*CGSP	Linear part of Cgs	Capacitance	0
*PC10	Gate-source capacitance polynomial coef.		0
*PC11	Gate-source capacitance polynomial coef.		1.0
*PC110	Polynomial coef. to model peaked Cgs		0
*PC111	Polynomial coef. to model peaked Cgs		0
*ADIV	Term to model peaked Cgs (1= no peaking)		1
*PC20	Gate-source capacitance polynomial coef.		0
*PC21	Gate-source capacitance polynomial coef.		0.5
*CGD0	Gate-drain capacitance parameter	Capacitance	0
*CGDP	Linear part of Cgd	Capacitance	0
*PC30	Gate-drain capacitance polynomial coef.		0
*PC31	Gate-drain capacitance polynomial coef.		0.5
*PC40	Gate-drain capacitance polynomial coef.		0
*PC41	Gate-drain capacitance polynomial coef.		1.0
*CDS0	Fixed drain-source capacitance (not scaled)	Capacitance	0
*CDSW	Scalable drain-source capacitance	Capacitance	0
*CPG	Gate-pad parasitic capacitance (not scaled)	Capacitance	0
*CPD	Drain-pad parasitic capacitance (not scaled)	Capacitance	0
*ISG	Gate diode current parameter	Current	10^{-20} A
*NG	Gate diode ideality factor		1.0
*RG	Gate resistance	Resistance	1.0 ω
*RS	Source resistance	Resistance	1.0 ω
*RI	Intrinsic resistance	Resistance	1.0 ω
*RD	Drain resistance	Resistance	1.0 ω
*RGD	Gate-drain resistance	Resistance	1.0 ω
*RCW	RF drain-source resistance parameter	Resistance	300 ω

Parameter	Description	Units	Default Value
*CRF	Capacitance that determines Rds break frequency	Capacitance	10^{-6} F
*LS	Source inductance (not scaled)	Inductance	0.0
*LG	Gate inductance (not scaled)	Inductance	0.0
*LD	Drain inductance (not scaled)	Inductance	0.0
*RTH	Thermal resistance C/W		0.1
*TEX	Temperature at which parameters were extracted	Temperature	25 C
*TEMP	Baseplate temperature	Temperature	25 C
*TAU_TH	Thermal time constant	Time	1 mS
*TCIPK	Thermal IDS IPK coefficient		0
*TCP1	Thermal IDS P1 coefficient		0
*TCCGS0	Thermal CGS0 coefficient		0
*TCCGD0	Thermal CGD0 coefficient		0
*TCRCW	Thermal RCW coefficient		0
*TCCRF	Thermal CRF coefficient		
*DTMAX	Maximum temperature increase (C) in self-heating		
*TMIN	Minimum device temperature		
AFAC	Gate-width scale factor		1.0
NFING	Number of fingers scale factor		1.0

** indicates a secondary parameter*

Implementation Details

This is a further development of the well known Angelov/Zirath/Rorsman, or Chalmers, model. It is especially good for use with HEMT devices, but is also useful for MESFETs. Angelov2 includes improved capacitance and breakdown modeling, as well as modeling of thermal effects.

Although this model is similar in many ways to the original Angelov model, it is not identical, and, in general, parameters for the earlier model cannot be used in this one.

Equations:

Below are the equations for this model.

Drain current

The drain current is given by

$$I_d(V_g, V_d) = \mathrm{IPK}(1 + \tanh(\Psi))\tanh(\alpha V_d)(1 + \mathrm{LAMBDA} \cdot V_d)$$

where

$$\Psi = P_{1m}(V_g - V_{\mathrm{pk}}) + P2(V_g - V_{\mathrm{pk}})^2 + P3(V_g - V_{\mathrm{pk}})^3 + \cdot \cdot \cdot$$

and

$$\alpha = \text{ALPHR} + \text{ALPHS}(1 + \tan(\Psi))$$

$$P_{1m} = P1(1 + B1 / \cosh^2(B2 \cdot V_d))$$

$$V_{\text{pk}} = \text{VPK0} + (\text{VPKS} - \text{VPK0})\tanh(\text{ALPHS} \cdot V_d)$$

Many of the above parameters are related directly to the peak current and transconductance of the device. See the references for the original Angelov model for further information.

Soft Breakdown

When breakdown is included, the above expressions are modified as

$$I_d(V_g, V_d) = IPK(1 + \tanh(\Psi)) \tanh(\alpha V_d)(1 + LAMBDA \cdot V_d + LAMSB \cdot e^{V_{dg} - VTR})$$

$$V_{\text{pk}} = \text{VPK0} + (\text{VPKS} - \text{VPK0})\tanh(\text{ALPHA} \cdot V_d) - \text{VSB2} \cdot (V_{\text{dg}} - \text{VTR})^2$$

Capacitances

The model uses a capacitance formulation to determine the reactive gate-to-source and gate-to-drain currents. This approach allows the model to be consistent with time-domain (SPICE) formulations. In this implementation, the gate charge is formulated as a single function of gate and drain voltages, so the gate current is

$$I_g = \frac{\partial Q_g}{\partial V_g}\frac{\text{dV}_g}{\text{dt}} + \frac{\partial Q_g}{\partial V_{\text{gd}}}\frac{\text{dV}_{\text{gd}}}{\text{dt}}$$

The first term in the above equation represents gate-to-source current, and the second, gate-to-drain current. This approach simplifies parameter extraction, because no transcapacitances are needed and the charge derivatives are identical to the small-signal capacitance.

The resulting capacitance functions are given by the following expressions:

$$C_{\text{gs1}}(V_g, V_d) = \text{ADIV} \cdot \text{CGS0}(1 + \tanh(\text{PC20} + \text{PC21} \cdot V_d)) \cdot (1 \cdot 0 + \tanh(\text{PC10} + \text{PC11} \cdot V_g))$$

$$C_{\text{gs2}}(V_g, V_d) = (1 - \text{ADIV})\text{CGS0}(1 + \tanh(\text{PC20} + \text{PC21} \cdot V_d)) \cdot (1 \cdot 0 + \tanh(\text{PC110} + \text{PC111} \cdot V_g))$$

$$C_{\text{gs}}(V_g, V_d) = \frac{\partial Q_g}{\partial V_g} + C_{\text{gs1}} + C_{\text{gs2}}$$

and

$$C_{\text{gd}}(V_g, V_d) = \frac{\partial Q_g}{\partial V_{\text{gd}}} = \text{CGD0}(1 + \tanh(\text{PC30} + \text{PC31} \cdot V_d)) \cdot (1 \cdot 0 + \tanh(\text{PC40} + \text{PC41} \cdot V_{\text{gd}}))$$

The parameter ADIV is used to account for a peak and decrease in capacitance, with increasing gate bias voltage, that sometimes is observed in pHEMTs. In most cases, ADIV = 1 and the expressions are symmetrical. It is important to recognize that this capacitance formulation is entirely different from the original Angelov model, and the parameters of the original model are not transferable to this one.

Drain Dispersion

The model uses a simple approach to account for nonlinear, frequency-sensitive drain-to-source resistance, often called drain dispersion. The drain-to-source resistance is described by a current source having the following I/V characteristic:

$$I_{\mathrm{Rd}}(V_g V_d) = \frac{(1 + \tanh(\Psi))}{\mathrm{RCW}} V_d$$

where

$$\Psi$$

is given by (2). The parameter RCW, which has units of resistance, is only approximately the drain-to-source resistance in current saturation; it is best viewed as a model parameter used to fit the measured small-signal drain-source resistance.

This element has a large inductor in parallel to provide a return for dc currents generated in the element and to force its dc voltage to be zero under all conditions.

Thermal Model

The thermal model modifies the parameters IPK, P1, CGS0, CGD0, RCW, and CRF as follows:

$$\mathrm{CGS0} \rightarrow \mathrm{CGS0} \cdot (1 + \mathrm{TCCGS0} \cdot \Delta T)$$

$$\mathrm{CGD0} \rightarrow \mathrm{CGD0} \cdot (1 + \mathrm{TCCGD0} \cdot \Delta T)$$

$$\mathrm{IPK} \rightarrow \mathrm{IPK} \cdot (1 + \mathrm{TCIPK} \cdot \Delta T)$$

$$P1 \rightarrow P1 \cdot (1 + \mathrm{TCP1} \cdot \Delta T)$$

$$\mathrm{RCW} \rightarrow \mathrm{RCW} \cdot (1 - \mathrm{TCRCW} \cdot \Delta T)$$

$$\mathrm{CRF} \rightarrow \mathrm{CRF} \cdot (1 - \mathrm{TCCRF} \cdot \Delta T)$$

where

$$\Delta T = T - \mathrm{TEX}$$

and T is the instantaneous temperature, determined from an electrothermal equivalent circuit. The electrothermal circuit consists of a thermal resistance and capacitance. The total power dissipation in the device, not just the drain dissipation, is used to determine temperature. The temperature coefficients are chosen so that, in most cases, they will be positive quantities; however, occasionally one or more may be negative.

RCW and CRF. The temperature dependence of RCW and CRF requires some explanation. In general, RCW (the RF drain-to-source resistance) decreases with temperature so, for accurate power-amplifier analysis, its temperature dependence should be included. Although CRF is a nonphysical component, it models the transition between the DC and RF drain-to-source resistance regions. This transition increases in frequency as temperature increases, so CRF should be made temperature dependent when necessary to model this phenomenon. In circuits where there are no frequency components near the transition frequency (which is usually on the order of 1 KHz to 1 MHz), this phenomenon need not be modeled, so TCCRF can be set to zero.

The temperature increase, ΔT, is calculated from an electrothermal equivalent circuit consisting of a thermal resistance, RTH, and a capacitance, C. C is calculated from the thermal time constant, TAU_TH. = RTH AzA C. ΔT is limited to 300 C in a numerically acceptable manner. The power used to calculate ΔT includes all RF and DC power dissipated in the device, not just the drain dissipation.

AWR Extensions

In the original model, ΔT was temperature in Celsius degrees. In the AWR implementation, it is as shown above. Setting MDLFLAG=0 changes this to the original formulation, in effect setting TEX = 0 C.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

References

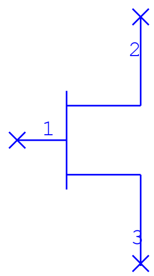
There are no references for this model. This model was implemented before publishing. The above information comes from personal communication from Prof. Angelov. The breakdown modeling is a joint effort of I. Angelov and S. Maas.

For further information about the model, contact NI AWR Technical Support.

(Obsolete) Advanced Compatible Angelov HEMT Model: ANGELOV2C

This version of the Angelov/Zirath/Rorsman (Chalmers) model is designed to be compatible with other implementations. This model is similar to the ANGELOV2 model.

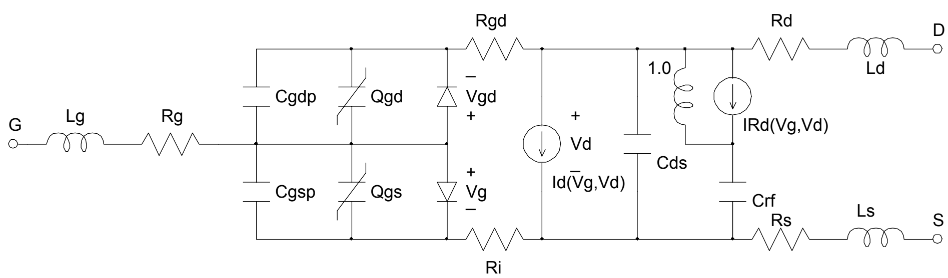
Symbol



Summary

This element is OBSOLETE and is replaced by the Advanced Compatible Angelov HEMT Model (ANGELOV2C2) element.

Equivalent Circuit



Parameters

Parameter	Description	Units	Default Value
ID	Device ID	Text	AC1
Idsmod	Ids model selector	Integer	0
Igmod	Igs/Igd model selector	Integer	0
Capmod	Capacitance model selector	Integer	2
*Ipk0	Current at maximum transconductance	Current	0.050
*Vpks	Gate voltage at maximum transconductance	Voltage	-0.2
*Dvpks	Delta gate voltage at peak transconductance	Voltage	0.2
*P1	Coefficient for channel current		1.0
*P2	Coefficient for channel current		0

Parameter	Description	Units	Default Value
*P3	Coefficient for channel current		0
*Alphar	Saturation parameter		0.1
*Alphas	Saturation parameter		1.0
*Vkn	Knee voltage parameter	Voltage	0.8
*Lambda	Channel length modulation term		0
*Lambda1	Channel length modulation term		0
*Lvg	Coefficient for Lambda		0
*B1	Coefficient for P1 dependence on Vds		0
*B2	Coefficient for P1 dependence on Vds		3.0
*Lsb0	Soft breakdown parameter		0
*Vtr	Threshold voltage term for breakdown	Voltage	20
*Vsb2	Surface breakdown parameter	Voltage	0
*Cds	Drain-to-source capacitance (linear)	Capacitance	0
*Cgspi	Gate-to-source capacitance term	Capacitance	0
*Cgs0	Gate-to-source capacitance coefficient	Capacitance	0
*Cgdpi	Gate-to-drain capacitance term	Capacitance	0
*Cgd0	Gate-to-drain capacitance coefficient	Capacitance	0
*Cgdpe	External gate-to-drain capacitance	Capacitance	0
*P10	Capacitance polynomial coefficient		0
*P11	Capacitance polynomial coefficient		1.0
*P20	Capacitance polynomial coefficient		0
*P21	Capacitance polynomial coefficient		0.2
*P30	Capacitance polynomial coefficient		0
*P31	Capacitance polynomial coefficient		0.2
*P40	Capacitance polynomial coefficient		0
*P41	Capacitance polynomial coefficient		1.0
*P111	Capacitance polynomial coefficient		0
*Ij	Gate forward conductance saturation current	Current	5e-4
*Pg	Gate dc current parameter		0.0
*Ne	Gate current ideality factor		1.4
*Vjg	Gate dc current parameter	Voltage	0.7
*Rg	Gate series resistance	Resistance	0
*Rd	Drain series resistance	Resistance	0
*Ri	Gate-to-source intrinsic series resistance	Resistance	0
*Rs	Source series resistance	Resistance	0
*Rgd	Gate-to-drain series resistance	Resistance	0
*Lg	Gate series inductance (not scaled with AFAC)	Inductance	0

Parameter	Description	Units	Default Value
*Ld	Drain series inductance (not scaled with AFAC)	Inductance	0
*Ls	Source series inductance (not scaled with AFAC)	Inductance	0
*Tau	Time delay	Time	0
*Rcmin	Minimum value of Rc	Resistance	1.0e-3
*Rcin	Resistance of frequency-dependent output conductance	Resistance	0.0
*Crfin	Capacitance of frequency-dependent output conductance	Capacitance	0.0
*Rth	Thermal resistance	Resistance	0.0
*Cth	Thermal capacitance	Capacitance	0.0 1 [11–15]
*Tcipk0	Temperature coefficient for Ipk0 parameter		0.0
*Tcp1	Temperature coefficient for P1 parameter		0.0
*Tccgs0	Temperature coefficient for Cgs0 parameter		0.0
*Tccgd0	Temperature coefficient for Cgd0 parameter		0.0
*Tclsb0	Temperature coefficient for Lsb0 parameter		0.0
*Tcrc	Temperature coefficient for Crc parameter		0.0
*Tccrf	Temperature coefficient for Crf parameter		0.0
*Tnom	Parameter measurement temperature	Temperature	25 C
Selft	Self-heating flag	Boolean	False
Noimod	Noise model selector	Integer	0
*NoiseR	Gate noise coefficient		0.5
*NoiseP	Drain noise coefficient		1.0
*NoiseC	Gate-drain noise correlation coefficient		0.9
*Fnc	Flicker noise corner	Frequency	0.0
*Kf	Flicker noise constant		0.0
*Af	Flicker noise exponent		1.0
*Ffe	Flicker noise frequency exponent	Frequency	1.0
*Tg	Gate noise temperature	Temperature	25 C
*Td	Drain noise temperature coefficient	Temperature	25 C
*Tdl	Drain noise temperature coefficient		0.1
*Tmn	Noise fitting term		1.0
*Klf	Flicker noise coefficient		0.0
*Fgr	Generation-recombination noise corner frequency	Frequency	0.0
*Np	Flicker noise frequency exponent		1.0
*Lw	Effective gate noise width	Length	0.1 mm
*DTMAX	Maximum temperature increase (C) in self-heating	Temperature	300
*TMIN	Minimum device temperature	Temperature	10
AFAC	Gate-width scale factor		1.0
NFING	Number of gate fingers scale factor		1.0

This default value is preserved for compatibility, but it is a poor choice. If Rth is nonzero, Cth should be given a large value to prevent convergence problems.

** indicates a secondary parameter*

Implementation Details

ANGELOV2C is a version of the ANGELOV2 model that is compatible with other implementations. This model is not identical to the original ANGELOV2 model and should be used only where compatibility is needed.

A few characteristics of the model are unique to NI AWR. The model scales according to gate width and number of gate fingers in the usual manner (see other FET models for examples). The default values of AFAC and NFING provide compatibility with other implementations. The temperature of the device is limited by DTMAX and TMIN during harmonic-balance iterations to prevent ill-conditioning when self-heating is used. You should set these in such a way that it is impossible, for example, for temperature coefficients to make a parameter (say, a resistance) change sign at these extreme temperatures.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

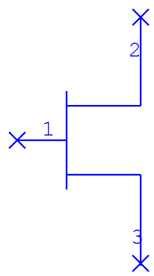
References

There are no references for this model. This model was implemented before publishing. The above information comes from personal communication from Prof. Angelov.

For further information about the model, contact NI AWR Technical Support.

Advanced Compatible Angelov HEMT Model: ANGELOV2C2

Symbol

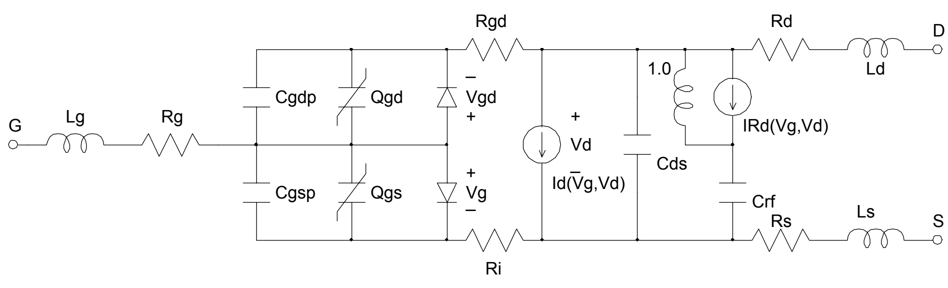


Summary

ANGELOV2C2 provides a single entry point to several versions of the well known Angelov/Zirath/Rorsman or Chalmers model and versions derived from it[1] [11–22]. It currently supports four different versions. The first version corresponds to the Chalmers model, with all its recent additions focusing on supporting GaN FETs[2] [11–23], as defined in version 1.7 of angelov_gan.va. The second and third versions are implementations of ADS's Angelov model, as described by the equations in the ADS documentation. The third version of the model is a refinement of the second version and it incorporates further modifications to increase compatibility with ADS's version of the model. The third version is recommended when you are interested in reproducing the behavior found in ADS's Angelov model. The fourth version corresponds to version 1.12 of the Chalmers model, as defined in version 1.12 of angelov_gan.va. These four modes of operation are controlled by the COMPAT parameter "angelov_gan 1.6", "ADS", "ADS2", or "angelov_gan 1.12" options, respectively. This model is conceived for compatibility. Compatibility in the Chalmers modes of operation ("angelov_gan 1.6" and "angelov_gan 1.12") is guaranteed with any other model based on the same Verilog-A definitions.

The Angelov model is especially good for use with HEMT devices, but is also useful for MESFETs. It includes improved capacitance and breakdown modeling, as well as modeling of thermal effects and noise. In addition, the Chalmers version of the model is particularly suitable for modeling GaN FETs. Although ANGELOV2C2 is similar in many ways to its predecessors ANGELOV and ANGELOV2, it is not identical, and in general, you cannot use parameters for the earlier models in this model.

Equivalent Circuit



Parameters

Parameter	Description	Units	Default Value
ID	Device ID	Text	AC1

Parameter	Description	Units	Default Value
Idsmod	Ids model selector	Integer	0
Igmod	Igs/Igd model selector	Integer	0
Capmod	Capacitance model selector	Integer	2
*Ipk0	Current at maximum transconductance	Current	0.050A
*Vpks	Gate voltage at maximum transconductance	Voltage	-0.2V
*Dvpks	Delta gate voltage at peak transconductance	Voltage	0.2V
*P1	Polynomial coefficient for channel current		If COMPAT="ADS" or "ADS2", 1.0 If COMPAT="angelov_gan 1.6" or "angelov_gan 1.12", 0.8
*P2	Polynomial coefficient for channel current		0
*P3	Polynomial coefficient for channel current		0
*Alphar	Saturation parameter		0.1
*Alphas	Saturation parameter		1.0
*Vkn	Knee voltage parameter	Voltage	If COMPAT="ADS" or "ADS2", 0.8 If COMPAT="angelov_gan 1.6" or "angelov_gan 1.12", 4.0
*Lambda	Channel length modulation parameter		If COMPAT="ADS" or "ADS2", 0.0 If COMPAT="angelov_gan 1.6" or "angelov_gan 1.12", 0.001
*Lambda1	Channel length modulation parameter		0
*Lvg	Coefficient for Lambda parameter		0
*B1	Coefficient for P1 dependence on Vds		If COMPAT="ADS" or "ADS2", 0.0 If COMPAT="angelov_gan 1.6" or "angelov_gan 1.12", 0.1
*B2	Coefficient for P1 dependence on Vds		If COMPAT="ADS" or "ADS2", 3.0 If COMPAT="angelov_gan 1.6" or "angelov_gan 1.12", 4.0

Parameter	Description	Units	Default Value
*Lsb0	Soft breakdown parameter		0
*Vtr	Threshold voltage term for breakdown	Voltage	If COMPAT="ADS" or "ADS2", 20.0 If COMPAT="angelov_gan 1.6" or "angelov_gan 1.12", 50.0
*Vsb2	Surface breakdown parameter	Voltage	0
*Cds	Drain-to-source capacitance (linear)	Capacitance	0
*Cgspi	Gate-to-source capacitance term	Capacitance	0
*Cgs0	Gate-to-source capacitance coefficient	Capacitance	0
*Cgdpi	Gate-to-drain capacitance term	Capacitance	0
*Cgd0	Gate-to-drain capacitance coefficient	Capacitance	0
*Cgdpe	External gate-to-drain capacitance	Capacitance	0
*P10	Capacitance polynomial coefficient		0
*P11	Capacitance polynomial coefficient		1.0
*P20	Capacitance polynomial coefficient		0
*P21	Capacitance polynomial coefficient		0.2
*P30	Capacitance polynomial coefficient		0
*P31	Capacitance polynomial coefficient		0.2
*P40	Capacitance polynomial coefficient		0
*P41	Capacitance polynomial coefficient		1.0
*P111	Capacitance polynomial coefficient		0
*Ij	Gate forward conductance saturation current	Current	0.5mA
*Pg	Gate dc current parameter		
*Ne	Gate current ideality factor		
*Vjg	Gate dc current parameter	Voltage	

Parameter	Description	Units	Default Value
*Rg	Gate series resistance	Resistance	If COMPAT="ADS" or "ADS2", 0.0 If COMPAT="angelov_gan 1.6" or "angelov_gan 1.12", 0.05
*Rd	Drain series resistance	Resistance	If COMPAT="ADS" or "ADS2", 0.0 If COMPAT="angelov_gan 1.6" or "angelov_gan 1.12", 0.05
*Ri	Gate-to-source intrinsic series resistance	Resistance	If COMPAT="ADS" or "ADS2", 0.0 If COMPAT="angelov_gan 1.6" or "angelov_gan 1.12", 0.05
*Rs	Source series resistance	Resistance	If COMPAT="ADS" or "ADS2", 0.0 If COMPAT="angelov_gan 1.6" or "angelov_gan 1.12", 0.05
*Rgd	Gate-to-drain series resistance	Resistance	If COMPAT="ADS" or "ADS2", 0.0 If COMPAT="angelov_gan 1.6" or "angelov_gan 1.12", 0.05
*Lg	Gate series inductance	Inductance	0
*Ld	Drain series inductance	Inductance	0
*Ls	Source series inductance	Inductance	0
*Tau	Time delay	Time	0
*Rcmin	Minimum value of Rc	Resistance	1.0e3 ohm
*Rc	Resistance of frequency-dependent output conductance	Resistance	1.0e4 ohm
*Crf	Capacitance of frequency-dependent output conductance	Capacitance	0
*Rcin	Resistance of frequency-dependent input conductance		1.0e5 ohm
*Crfin	Capacitance of frequency-dependent output conductance	Capacitance	0.0

Parameter	Description	Units	Default Value
*Rdel	Resistance of frequency-dependent input conductance		1.0e3 ohm
*Cdel	Capacitance of frequency-dependent output conductance	Capacitance	0.0
*Kbgate	Polynomial coeff Kbgate for freq dep input cond [1/V]		
*Rth	Thermal resistance	Resistance	If COMPAT="ADS" or "ADS2", 0.0 If COMPAT="angelov_gan 1.6" or "angelov_gan 1.12", 0.001
*Cth	Thermal capacitance	Capacitance	If COMPAT="ADS" or "ADS2", 0.0 If COMPAT="angelov_gan 1.6" or "angelov_gan 1.12", 0.0001
*Tcipk0	Linear temperature coefficient for Ipk0 parameter		If COMPAT="ADS" or "ADS2", 0.0 If COMPAT="angelov_gan 1.6" or "angelov_gan 1.12", -0.002
*Tcp1	Linear temperature coefficient for P1 parameter		If COMPAT="ADS" or "ADS2", 0.0 If COMPAT="angelov_gan 1.6" or "angelov_gan 1.12", -0.002
*Tccgs0	Linear temperature coefficient for Cgs0 parameter		If COMPAT="ADS" or "ADS2", 0.0 If COMPAT="angelov_gan 1.6" or "angelov_gan 1.12", 0.002
*Tccgd0	Linear temperature coefficient for Cgd0 parameter		If COMPAT="ADS" or "ADS2", 0.0 If COMPAT="angelov_gan 1.6" or "angelov_gan 1.12", 0.002
*Tclsb0	Linear temperature coefficient for Lsb0 parameter		0.0
*Tcrc	Linear temperature coefficient for Rc parameter		0.0
*Tccrf	Linear temperature coefficient for Crf parameter		0.0

Parameter	Description	Units	Default Value
*Tcrs	Linear temperature coefficient for Rs parameter		0.0
*Tnom	Parameter measurement temperature	Temperature	25 C
*Temp	Simulation temperature	Temperature	25 C
Selft	Self-heating flag (Off, On)		Off
Noimod	Noise model selector	Integer	If COMPAT="ADS" or "ADS2", 0 If COMPAT="angelov_gan 1.6" or "angelov_gan 1.12", 1
*NoiseR	Gate noise coefficient		0.5
*NoiseP	Drain noise coefficient		1.0
*NoiseC	Gate-drain noise correlation coefficient		0.9
*Fnc	Flicker noise corner	Frequency	0.0
*Kf	Flicker noise constant		0.0
*Af	Flicker noise exponent		1.0
*Ffe	Flicker noise frequency exponent	Scalar	1.0e-9GHz
*Tg	Gate noise temperature	Temperature	25 C
*Td	Drain noise temperature	Temperature	25 C
*Td1	Drain noise temperature coefficient		0.1
*Tmn	Noise fitting term		1.0
*Klf	Flicker noise coefficient		If COMPAT="ADS" or "ADS2", 0.0 If COMPAT="angelov_gan 1.6" or "angelov_gan 1.12", 1.0e14
*Fgr	Generation-recombination noise corner frequency	Frequency	If COMPAT="ADS" or "ADS2", 0.0 If COMPAT="angelov_gan 1.6" or "angelov_gan 1.12", 6.0e4
*Np	Flicker noise frequency exponent		If COMPAT="ADS" or "ADS2", 1.0 If COMPAT="angelov_gan 1.6" or "angelov_gan 1.12", 0.3

Parameter	Description	Units	Default Value
*Lw	Effective gate noise width	Length	If COMPAT="ADS" or "ADS2", 0.01 If COMPAT="angelov_gan 1.6" or "angelov_gan 1.12", 1.0e-4
*MULT	Scaling factor		1.0
*NFLAG	Noise flag (Off, On)		On
*Ebd	Surface breakdown model parameter		0.2
*Kbdgate	Gate breakdown parameter		1.0
*Pbdg	Gate breakdown exponent		0.5
*Rd2	Variable Drain ohmic resistance	Resistance	0.0
*Vbdgd	Gate drain breakdown voltage	Voltage	20.0
*Vbdgs	Gate source breakdown voltage	Voltage	5.0
*P222	Polynomial coefficient for capacitance		0.0
COMPAT	Compatibility flag (Chalmers, ADS)		ADS2
*m	Junction capacitance coefficient		0.5

** indicates a secondary parameter*

Implementation Details

Most of this model's parameters originate in ADS's Angelov model or in Chalmer's Verilog-A definition of it. Only a few parameters are unique to the ANGELOV2C2 model: MULT, COMPAT and NFLAG. MULT is a scaling factor. COMPAT allows switching between the three modes of operation, "Chalmers", "ADS", and "ADS2". NFLAG is used to enable/disable the noise of this model. Unlike the comprehensive parameter list in the "Parameters" section, only those parameters corresponding to a given COMPAT value are displayed with the element. Parameters appearing without a default value in the "Parameters" section are calculated parameters, or the default value depends on the setting of the compatibility flag. Although the model supports self-heating in all of its modes of operation, the thermal node is only externally available when COMPAT is set to "angelov_gan 1.12".

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

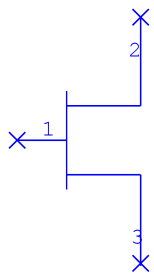
References

[1] I. Angelov, H. Zirath, and, N. Rorsman "Validation of a Nonlinear Transistor Model by Power Spectrum Characteristics of HEMT's and MESFET's", IEEE Trans. on Microwave Theory Tech., pp. 1046-1052, May 1995

[2] I. Angelov, M. Thorsell, K. Andersson, A. Inoue, K. Yamanaka, and, H. Noto "On the Large Signal Evaluation and Modeling of GaN FET", IEICE Transactions, pp. 1225-1233. 2010

AURIGA_M6: AURIGA_M6

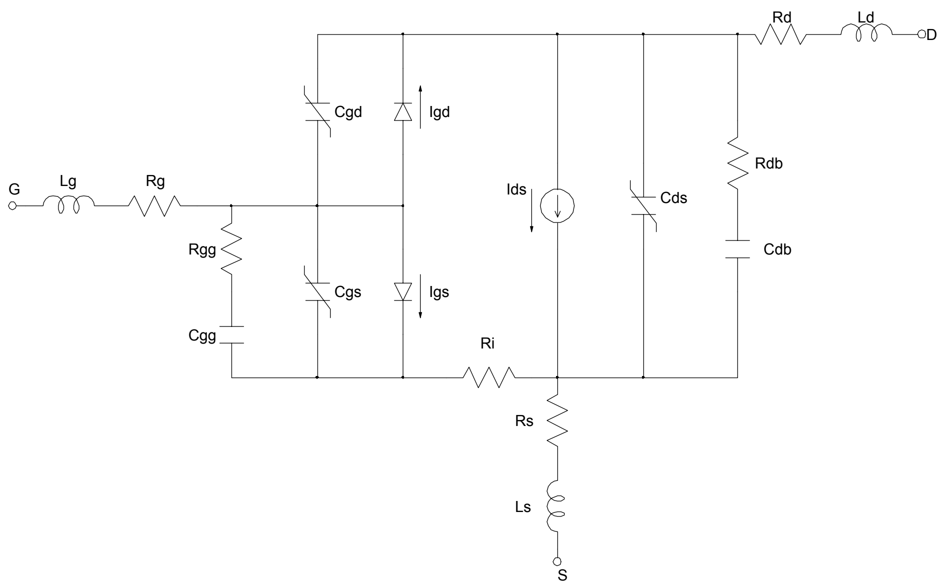
Symbol



Summary

The AURIGA_M6 model provides advanced characterization for Depletion or Enhancement mode MESFETs, GaNHEMTs, and other variations of the MESFET family. Its enhanced IV and capacitor modeling observes the Charge Conservation rule. AURIGA_M6 becomes symmetric at Vds = 0 when used for SWITCH modelling. It is accurate under a wide range of operating conditions, including passive applications (switches, mixers), and high voltage applications (power amplifiers).

Topology



Parameters

Name	Description	Unit Type	Default
ID	Device ID	Text	MF1
*SWX	Amplifier or Switch mode		0
*IK	Breakdown current	Current	1.00E-03

Name	Description	Unit Type	Default
*Vb0	Breakdown voltage	Voltage	500
*Vgs9	Cds coef		0
*c4	Cds-Vds slope	Capacitance	0
*a3	Cds-Vds slope exponent		-0.03
*a4	Cds-Vds slope exponent		0
*c3	Cds-Vds slope	Capacitance	9.50E-13
*a5	Cds-Vds slope exponent		0
*Cgd	Cgd constant		1.00E-13
*Cg1	Cgs at peak	Capacitance	1.27E-12
*Cg0	Cgs at pinch off	Capacitance	2.16E-12
*Cg2	Cgs at Vgs max	Capacitance	9.56E-12
*K4	Cgs slope (0,1) to Vdg		0.1
*K1	Cgs slope (0,1) to Vgs		0.87
*K6	Cgs slope (1,2) to Vdg		0.1
*K2	Cgs slope (1,2) to Vgs		2.474
*K3	Cgs swing (0,1) to Vdg		2
*K5	Cgs swing (1,2) to Vdg		2
*E	Constant port of gate I/V exponent		1.1
*Rdrain	Convergence factor	Resistance	1.00E+06
*Cdb	Dispersion Capacitance	Capacitance	1.00E-11
*Vgs8	Dispersion Resistance factor		0.3
*LD	Drain Inductance	Inductance	1.96E-11
*RD	Drain Resistance	Resistance	0.63
*IDSS	Drain saturation current at zero Vgs	Current	0.204
*alpha2	Drain Source Dispersion Resistance	Resistance	6
*alpha3	Drain Source Dispersion Resistance	Resistance	0
*beta2	Drain Source Dispersion Resistance	Resistance	1.5
*beta3	Drain Source Dispersion Resistance	Resistance	0
*Cds	Drain-Source Capacitance	Capacitance	4.10E-14
*AFAB	Exponential coef of GD diode		35
*AFAG	Exponential coef of GS diode		30.6
*K7	Gate Charge factor		5
*K8	Gate Charge factor		0.3
*Vgd1	Gate Charge factor		2.5
*LG	Gate Inductance	Inductance	4.53E-11
*RG	Gate Resistance	Resistance	1.3
*T	Gate-drain time delay	Seconds	3.25E-12

Name	Description	Unit Type	Default
AFAC	Gate-width scale factor		1
*x1	Ids slope adjustments at high Vds		1.5
*x2	Ids slope adjustments at high Vds		0.5
*x3	Ids slope adjustments at high Vds		60
*x4	Ids slope adjustments at high Vds		60
*Tcoef	IDSS Temperature coef		-0.00112
AL	Inductance scale factor		1
*beta	Input Dispersive Loss factor		1.5
*alpha	Input Dispersive Loss factor		6
*Vgs7	Input Dispersive Loss factor		0.3
*Cgg	Input Loss Capacitance	Capacitance	1.00E-11
*Ri	Intrinsic Resistance	Resistance	0
*M	IV Frequency Calculation factor		0.055
*P	IV Frequency Calculation factor		5
*R	IV Frequency Calculation factor		3
*Vgs3	IV Frequency Calculation factor		0.8
*FRX	IV Frequency Calculation mode		0
*Vgs2	IV function (Vgsd)	Voltage	0.34
*C	IV function (Vgsd) factor		4
*D	IV function (Vgsd) factor		0.28
*B	KG		2
*y1	Knee current cupression (f)		10
*y2	Knee current cupression (f)		0
*y3	Knee current cupression (f)		1
*y4	Knee current cupression (f)		-1
*aa	Knee current cupression (fVgsi)		0
*bb	Knee current cupression (fVgsi)		1
*cc	Knee current cupression (fVgsi)		0
*VDD	Knee current cupression drain voltage	Voltage	0.5
*Tnom	Nomial Temperature	Temperature	297
Temp	Operational Temperature	Temperature	297
*N	Pinch off voltage function at large Vds	Voltage	0.5
*GAM	Pinch-off slope		-0.02
*VP	Pinch-off voltage	Voltage	-1.1
*ds	Residual slope of drain I/V in saturated region		0.23
*Rb0	Resistance for breakdown Current	Resistance	20
*TcoefR	Rs, Rd Temperature coef		0.0113

Name	Description	Unit Type	Default
*IB0	Saturation current of GD diode	Current	2.80E-16
*IG0	Saturation current of GS diode	Current	5.00E-17
*SL	Slope of drain I/V in linear region	NOne	0.56
*G	Slope of drain I/V in saturated region (a)		1.7
*H	Slope of drain I/V in saturated region (a)		1.8
*q	Slope of drain I/V in saturated region (SS)		1
*SK	Slope of drain I/V in saturated region (SS)		1
*Vgs1	Slope of drain I/V in saturated region (SS)		1.1
*LS	Source Inductance	Inductance	1.76E-11
*Lsdb	Source PRL Inductance	Inductance	1.00E-06
*Rsdb	Source PRL Resistance	Resistance	0
*RS	Source Resistance	Resistance	0.914251
*Vgs4	Vgs between Cgs0 and Cgs1	Voltage	-0.969
*Vgs5	Vgs between Cgs1 and Cgs2	Voltage	-0.207
*Vgss	Vgs offset	Voltage	0
*x5	Xsl		0
*x6	Xsl		1

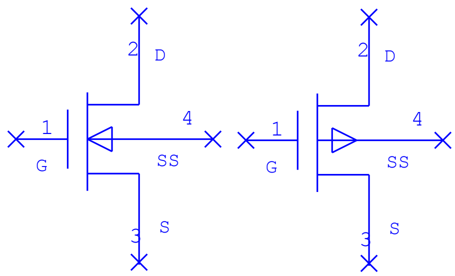
** indicates a secondary parameter*

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

BSIM3 MOSFET Model: BSIM3

Symbol



Summary

NI AWR's BSIM3 element is the successor of the BSIM3V322 and BSIM3V322P elements. Unlike BSIM3V322 and BSIM3V322P, which implement a single version of the BSIM3 model, version 3.22, BSIM3 model supports versions 3.2, 3.21, 3.22, 3.23 and 3.24.

BSIM3 supports HSPICE Automatic Model Selection. Automatic Model Selection, also known as binning, allows for the definition of MOSFET parameter sets over specific ranges of lengths and widths. This is accomplished by implementing the BSIM3 element in terms of block (data) and instance (model) elements, BSIM3d and BSIM3m, respectively. Parameters L and W, along with few other parameters are placed in instance element, BSIM3m, whereas the remaining BSIM3 parameters are located in the BSIM3d block element. Parameters in the block element are, in general, arrays of data. The data arrays corresponding to parameters LMIN, LMAX, WMIN and WMAX define intervals, which in turn, determine what range of L and W a specific entry in a parameter array corresponds to. As a result, setting the values of L and W in a BSIM3m element, automatically selects the appropriate values from the BSIM3d block element.

Parameters

Name	Description	Binnable	Unit Type	Default
ID	Element ID		Text	BS1
L	Gate Length		Length	0.5e-6m
W	Gate Width		Length	10e-6m
*XL	Length bias accounts for masking and etching effects		Length	
*XW	Width bias accounts for masking and etching effects		Length	
*PD	Drain diffusion perimeter		Length	
*PS	Source diffusion perimeter		Length	
*AD	Drain area, m^2			
*AS	Source area, m^2			

Name	Description	Binnable	Unit Type	Default
*NRD	No. of squares in drain area			
*NRS	No. of squares in source area			
*TEMP	Device temperature		Temperature	300.15K
*TNOM	Extraction temperature		Temperature	300.15K
*VERSION	Model version			3.22
*MOBMOD	Mobility model selector			1
*CAPMOD	Capacitance model selector			0
*PARAMCHK	Parameter value check			0
*UCBUNITS	1: UCB units for doping & temp; 0: All MKS (legacy)			2
*NQSMOD	Non Quasi Static model flag			0
*NOIMOD	Noise model selector			
*NFLAG	Noise flag			Noise On
*VTH0	Threshold V @ Vbs=0		Voltage	
*VFB	Flat-band voltage		Voltage	
*K1	First-order body effect coef			
*K2	Second-order body effect coef			
*K3	Narrow width coef			80
*K3B	Body-effect coef of K3			0
*W0	Narrow width param			2.5e-6
*NLX	Lateral nonuniform doping param			1.74e-7
*VBM	Minimum bulk-substrate bias for body effect calculation		Voltage	-3V
*DVT0	First coef of short channel effect on Vth			2.2
*DVT1	Second coef of short channel effect on Vth			0.53
*DVT2	Body-bias coef of short channel effect on Vth			-0.032

Name	Description	Binnable	Unit Type	Default
*DVT0W	1st coef of narrow width effect on Vth (m^-1)			0
*DVT1W	2nd coef of narrow width effect on Vth (m^-1)			5.3e6
*DVT2W	Body coef of narrow width effect on Vth			-0.032
*U0	Mobility (cm^2/Vsec)			
*UA	First-order mobility degradation coef (m/V)			2.25e-9
*UB	Second-order mobility degradation coef [(m/V)^2]			5.87e-19
*UC	Body effect mobility degradation coef			
*VSAT	Sat velocity (project length units/sec)			8.0e4
*A0	Bulk charge effect coef for channel length			1
*AGS	Gate bias coef of ABULK (V^-1)			0
*B0	Bulk charge effect coef for channel width			0
*B1	Bulk charge effect width offset			0
*KETA	Body bias coef of bulk charge effect (V^-1)			-0.047
*A1	First nonsaturation Effect Param (V^-1)			0
*A2	Second nonsaturation factor			1
*RDSW	Parasitic resistance per unit width			0
*PRWB	Body effect coef of RDSW			0
*PRWG	Gate-bias effect coef of RDSW			0
*WR	Width offset from Weff for Rds calc			1
*WINT	Width offset fitting param from IV w/out bias			0

Name	Description	Binnable	Unit Type	Default
*LINT	Width offset fitting prarm from IV w/out bias			0
*DWG	Coef of Weff gate dependence (m/V)			0
*DWB	Coef of Weff body dependence (m/V^0.5)			0
*VOFF	Offset voltage in subthreshold region			-0.08
*NFACTOR	Subthreshold swing factor			1
*ETA0	DIBL coef in subthreshold region			0.08
*ETAB	Body coef for subthreshold DIBL effect (1/V)			-0.07
*DSUB	DIBL coef exponent in subthreshold region			
*CIT	Interface trap capacitance (F/m^2)			0
*CDSC	DS channel coupling cap (F/m^2)			2.4e-4
*CDSCB	Body bias sensitivity of CDSC (F/Vm^2)			0
*CDSCD	Drain sensitivity of CDSC (F/Vm^2)			0
*PCLM	Channel length modulation			1.3
*PDIBLC1	First output res DIBL param			0.39
*PDIBLC2	Second output res DIBL param			0.0086
*PDIBLCB	Body effect DIBL correction param (V^-1)			0
*DROUT	L dependence DIBL correction in Rout			0.56
*PSCBE1	First substrate current body effect param (V/m)			4.24e8
*PSCBE2	2nd substrate current body effect (m/V)			1e-5

Name	Description	Binnable	Unit Type	Default
*PVAG	Gate dependence of Early voltage			0
*DELTA	Effective Vds param			0.01
*NGATE	Poly gate doping (cm^-3)			0
*ALPHA0	First param of impact ionization current (m/V)			0
*ALPHA1	Isub parameter for length scaling (1/V)			0
*BETA0	Second param of impact ionization current			30
*RSH	Source-drain sheet res per sq			0
*JSW	Sidewall sat current (A/m)			0
*JS	S/D jcn saturation current per area (A/m^2)			1e-4
*IJTH	S/D to bulk diode limiting current (Not implemented)			0.1
*XPART	Charge-partitioning selector			1
*CGSO	S-G overlap cap per length (F/m)			
*CGDO	D-G overlap cap per length (F/m)			
*CGBO	B-G overlap cap per length (F/m)			
*CJ	Bottom jcn cap per area (F/m^2)			5e-4
*MJ	CJ grading coef			0.5
*MJSW	S&D side jcn grading coef			0.33
*CJSW	S&D side jcn cap (Proj cap units/m^2)			5e-10
*CJSWG	Source drain/gate sidewall jcn cap coef (Proj cap units/m)			
*MJSWG	CJSWG grading coef			

Name	Description	Binnable	Unit Type	Default
*PBSW	CJSW built-in voltage			1
*PB	S&D Bottom jcn built-in voltage			1
*PBSWG	Source drain/gate sidewall jcn cap built-in voltage			
*CGSL	Light doped S-G region overlap (F/m)			0
*CGDL	Light doped D-G region overlap (F/m)			0
*CKAPPA	Coef for light-doped region			0.6
*CF	Fringing field capacitance (F/m)			
*CLC	Const term for short-channel model			0.1e-6
*CLE	Exp term for short-channel model			0.6
*DLC	Length offset fitting param for CV			
*DWC	Width offset fitting param for CV			
*VFBCV	Flat-band voltage param for CAPMOD=0 only			-1
*NOFF	CV param in vgsteff for weak to strong inversion			1
*VOFFCV	CV param in vgsteff for weak to strong inversion			0
*ACDE	Exponential coef for charge thickness in CAPMOD=3			1
*MOIN	Coef for gate-bias dependent surface potential			15
*ELM	Elmore constant for the channel			5
*WL	Coef of length dependence for width offset			0

Name	Description	Binnable	Unit Type	Default
*WLN	Power of length dependence for width offset			1
*WW	Coef of width dependence for width offset			0
*WWN	Power of width dependence for width offset			1
*WWL	Coef of length & width cross term for width offset			0
*LL	Coef of length dependence for length offset			0
*LLN	Power of length dependence for length offset			1
*LW	Coef of width dependence for length offset			0
*LWN	Power of width dependence for length offset			1
*LWL	Coef of length and width cross term for length offset			0
*LLC	Coef of length dependence foe CV length offset			
*LWC	Coef of width dependence for CV length offset			
*LWLC	Coef of length and width dependence for CV length offset			
*WLC	Coef of length dependence for CV width offset			
*WWC	Coef of width dependence for CV width offset			

Name	Description	Binnable	Unit Type	Default
*WWLC	Coef of length & width dependence for CV width offset			
*UTE	Mobility temp exponent			-1.5
*KT1	Temp coef for threshold voltage			-0.11
*KT1L	Channel length dependence of Vth coef (Vm)			0
*KT2	Body bias coef of Vth			0.022
*UA1	Temp coef of UA			4.31e-9
*UB1	Temp coef of UB			7.61e-18
*UC1	Temp coef of UC			
*AT	Temp coef of saturation velocity			3.3e4
*PRT	Temp coef of RDSW			0
*NJ	Emission coef of junction			1
*XTI	Junction current temp exponent coef			3
*TPB	Temp coef of PB			0
*TPBSW	Temp coef of PBSW			0
*TPBSWG	Temp coef of PBSWG			0
*TCJ	Temp coef fo CJ			0
*TCJSW	Temp coef of CJSW			0
*TCJSWG	Temp coef of CJSWG			0
*NOIA	(Noise parameter A)			
*NOIB	(Noise parameter B)			
*NOIC	(Noise parameter C)			
*EM	(Saturation electric field parameter)			4.1e7
*AF	Flicker noise exponent			1
*EF	Flicker noise frequency exponent			1
*KF	Flicker noise coefficient			0
*TOX	Oxide thickness (m)			1.5e-8
*TOXM	TOX at which params were extracted (m)			

Name	Description	Binnable	Unit Type	Default
*XJ	Junction depth (m)			1.5e-7
*GAMMA1	Body-effect coef near the surface			
*GAMMA2	Body-effect coef in the bulk			
*NCH	Channel doping concentration (cm^-3)			1.7e17
*NSUB	Substrate doping concentration (cm^-3)			
*VBX	VBS at which the depletion region equals XT			
*XT	Doping depth			
*LMIN	Minimum channel length for which the model is valid		Length	0m
*LMAX	Maximum channel length for which the model is valid		Length	1m
*WMIN	Minimum channel width for which the model is valid		Length	0m
*WMAX	Maximum channel width for which the model is valid		Length	1m
*BINUNIT	Units of size parameters in binning approach			1
*MULT	Gate-width linear scale factor			1
*AGS	Length/ width/ product	Yes		0
*AT	Length/ width/ product	Yes		0
*KETA	Length/ width/ product	Yes		0
*A0	Length/ width/ product	Yes		0
*B0	Length/ width/ product	Yes		0
*U0	Length/ width/ product	Yes		0
*UA	Length/ width/ product	Yes		0
*UB	Length/ width/ product	Yes		0
*UC	Length/ width/ product	Yes		0
*UC1	Length/ width/ product	Yes		0
*VSAT	Length/ width/ product	Yes		0

Name	Description	Binnable	Unit Type	Default
*VTH0	Length/ width/ product	Yes		0
*K1	Length/ width/ product	Yes		0
*K2	Length/ width/ product	Yes		0
*K3	Length/ width/ product	Yes		0
*K3B	Length/ width/ product	Yes		0
*W0	Length/ width/ product	Yes		0
*VOFF	Length/ width/ product	Yes		0
*VOFFCV	Length/ width/ product	Yes		0
*CIT	Length/ width/ product	Yes		0
*ETA0	Length/ width/ product	Yes		0
*ETAB	Length/ width/ product	Yes		0
*PCLM	Length/ width/ product	Yes		0
*PDIBLC2	Length/ width/ product	Yes		0
*PSCBE1	Length/ width/ product	Yes		0
*KT1	Length/ width/ product	Yes		0
*KT2	Length/ width/ product	Yes		0
*UTE	Length/ width/ product	Yes		0
*UA1	Length/ width/ product	Yes		0
*UB1	Length/ width/ product	Yes		0
*CGSL	Length/ width/ product	Yes		0
*CGDL	Length/ width/ product	Yes		0
*RDSW	Length/ width/ product	Yes		0
*XJ	Length/ width/ product	Yes		0
*NGATE	Length/ width/ product	Yes		0
*NSUB	Length/ width/ product	Yes		0
*NCH	Length/ width/ product	Yes		0
*NLX	Length/ width/ product	Yes		0
*DVT0	Length/ width/ product	Yes		0
*DVT1	Length/ width/ product	Yes		0
*DVT2	Length/ width/ product	Yes		0
*DVT0W	Length/ width/ product	Yes		0
*DVT1W	Length/ width/ product	Yes		0
*DVT2W	Length/ width/ product	Yes		0
*DSUB	Length/ width/ product	Yes		0
*VBM	Length/ width/ product	Yes		0
*B1	Length/ width/ product	Yes		0
*A1	Length/ width/ product	Yes		0

Name	Description	Binnable	Unit Type	Default
*A2	Length/ width/ product	Yes		0
*PRWG	Length/ width/ product	Yes		0
*PRWB	Length/ width/ product	Yes		0
*WR	Length/ width/ product	Yes		0
*NFACTOR	Length/ width/ product	Yes		0
*CDSC	Length/ width/ product	Yes		0
*CDSCD	Length/ width/ product	Yes		0
*CDSCB	Length/ width/ product	Yes		0
*PDIBLC1	Length/ width/ product	Yes		0
*PDIBLCB	Length/ width/ product	Yes		0
*DROUT	Length/ width/ product	Yes		0
*PSCBE2	Length/ width/ product	Yes		0
*PVAG	Length/ width/ product	Yes		0
*DELTA	Length/ width/ product	Yes		0
*ALPHA0	Length/ width/ product	Yes		0
*ALPHA1	Length/ width/ product	Yes		0
*BETA0	Length/ width/ product	Yes		0
*CKAPPA	Length/ width/ product	Yes		0
*CF	Length/ width/ product	Yes		0
*CLC	Length/ width/ product	Yes		0
*CLE	Length/ width/ product	Yes		0
*VFBCV	Length/ width/ product	Yes		0
*DWG	Length/ width/ product	Yes		0
*DWB	Length/ width/ product	Yes		0
*KT1L	Length/ width/ product	Yes		0
*PRT	Length/ width/ product	Yes		0
*GAMMA1	Length/ width/ product	Yes		0
*GAMMA2	Length/ width/ product	Yes		0
*VBX	Length/ width/ product	Yes		0
*XT	Length/ width/ product	Yes		0
*VFB	Length/ width/ product	Yes		0
*NOFF	Length/ width/ product	Yes		0
*ACDE	Length/ width/ product	Yes		0
*MOIN	Length/ width/ product	Yes		0
*TLEV	Temperature model selector			1

Name	Description	Binnable	Unit Type	Default
*TLEVC	Capacitance temperature model selector			1
*EG	Energy gap at 0 K/n0.0 < EG			1.16
*GAP1	First bandgap correction factor			0
*GAP2	Secod bandgap correction factor			0
*PHI	Surface potential			
*PTC	PHI temperature coefficient			0
*DCAP	Capacitance model selector			0
*IMAX	Maximum device current (for improving convergence)			1000
*ACM	Area calculation method			
*CALCACM	HSPICE-style area and perimeter calculations			0
*HDIF	Length of heavy doped diffusion contact		Length	
*LD	Lateral diffusion into channel		Length	
*LDIF	Length of light diffusion region		Length	
*RDC	Additional drain contact resistance			
*RSC	Additional source contact resistance			
*RD	Drain resistance			
*RS	Source resistance			
*WMLT	Channel width shrink factor			
*LMLT	Channel length shrink factor			
*NLEV	Noise model selector			
*GDSNOI	Channel thermal noise coefficient			
*COMPAT	Model copatibility selector			1

Name	Description	Binnable	Unit Type	Default
*SA0	Reference distance from OD edge to poly		Length	
*SB0	Reference distance from OD edge to poly		Length	
*SA	Distance from OD edge to poly		Length	
*SB	Distance from OD edge to poly		Length	
*WLOD	Stress effect length parameter		Length	
*KU0	Stress effect mobility degradation/ enhancement coefficient		Length	
*STK2	VTH0 related K2 shift factor		Length	
*STETA0	VTH0 releated ETA0 shift factor		Length	
*LKU0	Length dependence of KU0			
*WKU0	Width dependence of KU0			
*PKU0	Product dependence of KU0			
*TKU0	Temperature parameter for U0 stress effect			
*LLODKU0	Length parameter for U0 stress effect			
*WLODKU0	Width parameter for U0 stress effect			
*LLODVTH	Length parameter for VTH stress effect			
*WLODVTH	Width parameter for VTH stress effect			
*KVTH0	Threshold shift parameter for stress effect			
*LKVTH0	Length dependence of KVTH0			
*WKVTH0	Width dependence of KVTH0			
*PKVTH0	Product dependence of KVTH0			

Name	Description	Binnable	Unit Type	Default
*LODK2	Stress effect K2 shift modification factor			
*LODETA0	Stress effect ETA0 shift modification factor			
*KVSAT	Stress effect saturation velocity degradation/ enhancement parameter			
*ELM	Length/ width/ product	Yes		
TYPE	Device Type			1
*FLKMOD	Spice Flicker noise model selector			0
*LDDR	Length of Drain Diffusion Region		Length	
*LSDR	Length of Souce Diffusion Region		Length	

Operating Points

The following letter pairs have been used to identify the NL branches: ds, bs, bd, gs and gd. Here g,d,s abd b correspond to the gate, drain, source and substrate terminals, respectively. These are used to identify branch related operating point information, i.e., branch voltages, currents, etc.

Parameter	Description
vth (Voltage)	Threshold voltage
vdsat (Voltage)	Drain-source saturation voltage
gm (Conductance)	Common-source transconductance
gds (Conductance)	Common-source output conductance
gmbs (Conductance)	Body-transconductance
Isub(Current)	Substrate current
cgg (Capacitance)	$\frac{\partial Q_g}{\partial V_g}$, intrinsic charge.
cgb (Capacitance)	$\frac{\partial Q_g}{\partial V_b}$, intrinsic charge.
cgd (Capacitance)	$\frac{\partial Q_g}{\partial V_d}$, intrinsic charge.
cgs (Capacitance)	$\frac{\partial Q_g}{\partial V_s}$, intrinsic charge.

Parameter	Description
cbg (Capacitance)	$\frac{\partial Q_b}{\partial V_g}$, intrinsic charge.
cbb (Capacitance)	$\frac{\partial Q_b}{\partial V_b}$, intrinsic charge.
cbd (Capacitance)	$\frac{\partial Q_b}{\partial V_d}$, intrinsic charge.
cbs (Capacitance)	$\frac{\partial Q_b}{\partial V_s}$, intrinsic charge.
cdg (Capacitance)	$\frac{\partial Q_d}{\partial V_g}$, intrinsic charge.
cdb (Capacitance)	$\frac{\partial Q_d}{\partial V_b}$, intrinsic charge.
cdd (Capacitance)	$\frac{\partial Q_d}{\partial V_d}$, intrinsic charge.
cds (Capacitance)	$\frac{\partial Q_d}{\partial V_s}$, intrinsic charge.
csg (Capacitance)	$\frac{\partial Q_s}{\partial V_g}$, intrinsic charge.
csb (Capacitance)	$\frac{\partial Q_s}{\partial V_b}$, intrinsic charge.
csd (Capacitance)	$\frac{\partial Q_s}{\partial V_d}$, intrinsic charge.
css (Capacitance)	$\frac{\partial Q_s}{\partial V_s}$, intrinsic charge.
cjd (Capacitance)	Drain-substrate junction capacitance
cjs (Capacitance)	Source-substrate junction capacitance
pwr (Power)	Dissipated power
gmoverid (None)	Gm/Ids
ro (Resistance)	Common-source output resistance
Cgs (Capacitance)	$\frac{\partial Q_g}{\partial V_s}$
Cgd (Capacitance)	$\frac{\partial Q_g}{\partial V_d}$

Parameter	Description
Cgtot (Capacitance)	$\frac{\partial Q_g}{\partial V_g}$
Cbtot (Capacitance)	$\frac{\partial Q_b}{\partial V_b}$
Cdtot (Capacitance)	$\frac{\partial Q_d}{\partial V_d}$
Cstot (Capacitance)	$\frac{\partial Q_s}{\partial V_s}$
ft (Frequency)	Unity small-signal current-gain frequency

.

Implementation Details

Parameter TYPE controls whether the device is N, or, P channel; which is reflected by the device symbol. BSIM3 is based on the source code released by the BSIM Research Group at UC Berkeley. However, many parameters have been added to support HSPICE and Spectre enhancements to Berkeley's original model. These extensions, among others, include a variety of ways to compute junction areas and perimeters. HSPICE Area Calculation Methods (ACM) 0, 2, 3,10, 12 and 13 are all supported.

The NFLAG and COMPAT parameters are unique to the NI AWR implementation. Parameter NFLAG allows to turn the device noise on, or, off. COMPAT, the compatibility selection flag, takes three different values: HSPICE, AWR and SPECTRE. The HSPICE and Spectre values emulate HSPICE and Spectre implementations of the BSIM3 model, respectively. Parameters for which there is no default value listed in the table above, obtain their value from formulas. In many instances the provided default value depends on the compatibility flag setting.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

References

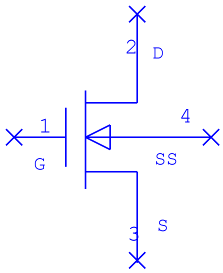
[1] W. Liu et al., BSIM3V3.2.2 MOSFET Model User's Manual, Dept. of Electrical Engineering and Computer Sciences, University of California, Berkeley CA 94720, USA, 1999.

[2] Y. Cheng and C. Hu, MOSFET Modeling & BSIM3 User's Guide, Norwell, MA: Kluwer, 1999.

[3] William Liu, MOSFET Models for Spice Simulation, Including BSIM3v3 and BSIM4, Wiley-IEEE, February 2001.

(Obsolete) Berkeley BSIM3V322 MOSFET ver. 3.2.2: BSIM3V322

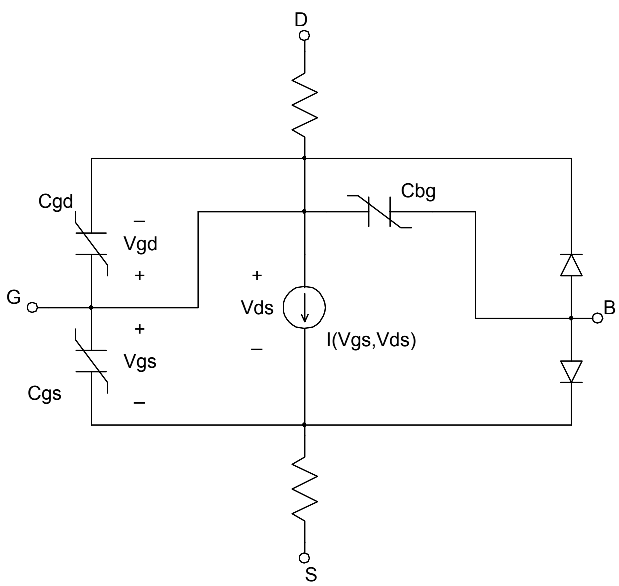
Symbol



Summary

This element is OBSOLETE and is replaced by the BSIM3 MOSFET Model (BSIM3) element.

Equivalent Circuit



Parameters

Name	Description	Binnable?	Unit Type	Default
ID	Element ID		Text	BS1
L	Gate Length		Length	0.5um
W	Gate Width		Length	10um
XL	Length bias accounts for masking and etching effects		Length	

Name	Description	Binnable?	Unit Type	Default
XW	Width bias accounts for masking and etching effects		Length	
PD	Drain diffusion perimeter		Length	
PS	Source diffusion perimeter		Length	
AD	Drain area (m^2)			
AS	Source area (m^2)			
NRD	No. of squares in drain area			
NRS	No. of squares in source area			
TNOM	Parameter Extraction temperature		Temperature	27C
TEMP	Device temperature		Temperature	27C
VERSION	Model version (must be 3.1 or 3.22)			
MOBMOD	Mobility model selector			1
CAPMOD	Capacitance model selector			0
PARAMCHK	Parameter value check			0
UCBUNITS	1: UCB units for doping & temp; 0: All MKS			2
NQSMOD	Non Quasi Static model Flag			0
NOIMOD	Supported only in HSPICE simulations			2: BSIM-Flicker BSIM-channel
NFLAG	Noise flag			Noise On
VTH0	Threshold V @ Vbs=0	YES	V	
VFB	Flat-band voltage	YES	V	
K1	First order body effect coefficient	YES		
K2	Second order body effect coefficient	YES		
K3	Narrow width coefficient	YES		80.0
K3B	Body effect coefficient of k3	YES		0.0
W0	Narrow width parameter	YES		2.5e-6
NLX	Lateral non-uniform doping parameter	YES		1.74e-7
VBM	Max. applied body bias in Vth calculation	YES	V	-3.0V
DVT0	First coefficient of short-channel effect on Vth	YES		2.2
DVT1	Second coefficient of short-channel effect on Vth	YES		0.53
DVT2	Body-bias coefficient of short-channel effect on Vth	YES		-0.032
DVT0W	First coefficient of narrow width effect on Vth (m^ -1)	YES		0.0
DVT1W	Second coefficient of narrow width effect on Vth (m^ -1)	YES		5.3e6
DVT2W	Body-bias coefficient of narrow width effect Vth	YES		-0.032
U0	Mobility	YES		670

Name	Description	Binnable?	Unit Type	Default
UA	First-order mobility degradation coefficient (m/V)	YES		2.25E-9
UB	Second-order mobility degradation coefficient [(m/V)^2]	YES		5.87E-19
UC	Body effect mobility degradation coefficient	YES		
VSAT	Saturation velocity (project length units/sec)	YES		8.0E4
A0	Bulk charge effect coefficient for channel length	YES		1.0
AGS	Gate bias coefficient of ABULK (V^-1)	YES		0.0
B0	Bulk charge effect coefficient for channel width	YES		0.0
B1	Bulk charge effect width offset	YES		0.0
KETA	Body-bias coefficient of bulk charge effect (V^-1)	YES		-0.047
A1	First non-saturation effect parameter (V^-1)	YES		0.0
A2	Second non-saturation factor	YES		1.0
RDSW	Parasitic resistance per unit width	YES		0.0
PRWB	Body effect coefficient of RDSW	YES		0
PRWG	Gate bias effect coefficient of RDSW	YES		0
WR	Width Offset from Weff for Rds calculation	YES		1.0
WINT	Width offset fitting parameter from I-V without bias			0.0
LINT	Width offset fitting parameter from I-V without bias			0.0
DWG	Coefficient of Weff's gate dependence (m/V)	YES		0.0
DWB	Coefficient of Weff' body dependence (m/V^0.5)	YES		0.0
VOFF	Offset voltage in the subthreshold region	YES		-0.08
NFACTOR	Subthreshold swing factor	YES		1.0
ETA0	DIBL coefficient in subthreshold region	YES		0.08
ETAB	Body coefficient for subthreshold DIBL effect (1/V)	YES		-0.07
DSUB	DIBL coefficient exponent in subthreshold region	YES		
CIT	Interface trap capacitance (F/m^2)	YES		0.0
CDSC	Drain/Source to channel coupling capacitance (F/Vm^2)	YES		2.4E-4
CDSCB	Body-bias sensitivity of CDSC (F/Vm^2)	YES		0.0
CDSCD	Drain-bias sensitivity of CDSC (F/Vm^2)	YES		0.0
PCLM	Channel length modulation	YES		1.3
PDIBLC1	First output resistance DIBL parameter	YES		0.39
PDIBLC2	Second output resistance DIBL parameter	YES		0.0086
PDIBLCB	Body-effect DIBL correction parameters (V^-1)	YES		0.0
DROUT	L dependence DIBL correction in Rout	YES		0.56

Name	Description	Binnable?	Unit Type	Default
PSCBE1	First substrate current body effect parameter (V/m)	YES		4.24E8
PSCBE2	Second substrate current body effect parameter (m/V)	YES		1.0E-5
PVAG	Gate dependence of Early voltage	YES		0.0
DELTA	Effective Vds parameter	YES		0.01
NGATE	Poly gate doping (cm^-3)	YES		0
ALPHA0	The first parameter of impact ionization current (m/V)	YES		0
ALPHA1	Isub parameter for length scaling (1/V)	YES		0.0
BETA0	The second parameter of impact ionization current	YES		30
RSH	Source drain sheet resistance per square			0.0
JSSW	Side wall saturation current (A/m)			0.0
JS	Source drain junction saturation current per area (A/m^2)			1.0E-4
IJTH	Source drain to bulk diode limiting current (Not implemented)			0.1
XPART	Charge partitioning selector			0.0
AFAC	Area scale factor			1.0
CGS0	Source-gate overlap capacitance per length (F/m)			
CGD0	Drain-gate overlap capacitance per length (F/m)			
CGB0	Gate bulk overlap capacitance per unit length (F/m)			0.0
CJ	Bottom junction capacitance per area (F/m^2)s			5.0e-4
MJ	CJ grading coefficient			0.5
MJSW	Source/Drain side junction grading coefficient			0.33
CJSW	Source/Drain side junction capacitance (Proj cap units/m^2)			5.E-10
CJSWG	Source/Drain gate side wall junction capacitance coefficient (Proj cap units/m)			
MJSWG	CJSWG grading coefficient			
PBSW	CJSW built-in voltage			1.0
PB	Source/Drain bottom junction built-in voltage			1.0
PBSWG	Source/Drain gate side wall junction built-in voltage			
CGS1	Light doped source-gate region overlap (F/m)			0.0
CGD1	Light doped drain-gate region overlap (F/m)			0.0
CKAPPA	Coefficient for lightly doped region	YES		0.6
CF	Fringing field capacitance (F/m)	YES		
CLC	Constant term for the short channel model	YES		0.1E-6

Name	Description	Binnable?	Unit Type	Default
CLE	Exponential term for the short channel model	YES		0.6
DLC	Length offset fitting parameter from C-V			
DWC	Width offset fitting parameter from C-V			
VFBCV	Flat-band voltage parameter (for capMod=0 only)	YES		-1
NOFF	CV parameter in Vgsteff, for weak to strong inverion	YES		1.0
VOFFCV	CV parameter in Vgsteff, for week to strong inversion	YES		0.0
ACDE	Exponential coefficient for charge thickness in CAPMOD=3	YES		1.0
MOIN	Coefficient for the gate-bias dependent surface potential	YES		15.0
ELM	Elmore constant of the channel			5
WL	Coefficient of length dependence for width offset			0.0
WLN	Power of length dependence of width offset			1.0
WW	Coefficient of width dependence for width offset			0.0
WWN	Power of width dependence of width offset			1.0
WWL	Coefficient of length and width cross term for width offset			0.0
LL	Coefficient of length dependence for length offset			0.0
LLN	Power of length dependence for length offset			1.0
LW	Coefficient of width dependence for length offset			0.0
LWN	Power of width dependence for length offset			1.0
LWL	Coefficient of length and width cross term for length offset			0.0
LLC	Coefficient of length dependence for CV length offset			
LWC	Coefficient of width dependence for CV length offset			
LWLC	Coefficient of length and width dependence for CV length offset			
WLC	Coefficient of length dependence for CV width offset			
WWC	Coefficient of width dependence for CV width offset			
WWLC	Coefficient of length and width dependence for CV width offset			
UTE	Mobility temperature exponent	YES		-1.5
KT1	Temperature coefficient for threshold voltage	YES		-0.11
KT1L	Temperature coefficient for threshold voltage	YES		0.0

Name	Description	Binnable?	Unit Type	Default
KT2	Body-bias coefficient of Vth	YES		0.022
UA1	Temperature coefficient for UA	YES		4.31E-9
UB1	Temperature coefficient for UB	YES		-7.61E-18
UC1	Temperature coefficient for UC	YES		
AT	Temperature coefficient of saturation velocity	YES		3.3E4
PRT	Temperature coefficient for RDSW	YES		0.0
NJ	Emission coefficient of junction			1.0
XTI	Junction current temperature exponent coefficient (Not implemented)			3.0
TPB	Temperature coefficient of PB (Not implemented)			0.0
TPBSW	Temperature coefficient of PBSW (Not implemented)			0.0
TPBSWG	Temperature coefficient of PBSWG (Not implemented)			0.0
TCJ	Temperature coefficient of CJ (Not implemented)			0.0
TCJSW	Temperature coefficient of CJSW (Not implemented)			0.0
TCJSWG	Temperature coefficient of CJSWG (Not implemented)			0.0
NOIA	Noise parameter A			
NOIB	Noise parameter B			
NOIC	Noise parameter C			
EM	Saturation electric field parameter			4.1e7
AF	Flicker noise exponent			1.0
EF	Flicker noise frequency exponent			1.0
KF	Flicker noise coefficient			0.0
TOX	Oxide thickness			1.5e-8
TOXM	Tox at which parameters are extracted (m)			
XJ	Junction Depth (m)	YES		1.5e-7
GAMMA1	Body-effect coefficient near the surface	YES		
GAMMA2	Body-effect coefficient in the bulk	YES		
NCH	Channel doping concentration (cm^-3)	YES		
NSUB	Substrate doping concentration (cm^-3)	YES		
VBX	VBS at which the depletion region width equals XT	YES		
XT	Doping depth	YES		
TLEV	Temperature model selector			1
TLEVC	Capacitance temperature model selector			1

Name	Description	Binnable?	Unit Type	Default
EG	Energy gap at 0 K			1.16
GAP1	First bandgap correction factor			0.0
GAP2	Second bandgap correction factor			0.0
PHI	Surface potential			
PTC	PHI temperature coefficient			0.0
LMIN	(Not Implemented)			0.0
LMAX	(Not Implemented)			1.0
WMIN	(Not Implemented)			0.0
WMAX	(Not Implemented)			1.0
BINUNIT	Units of size parameters in binning approach			1 (microns)
DCAP	Capacitance model selector			1
IMAX	Maximum device current		Current	1e6 mA
ACM	Area calculation method			
CALCACM	HSPICE-style area and perimeter calculations			0
HDIF	Length of heavy doped diffusion contact		Length	
LD	Lateral diffusion into channel		Length	
LDIF	Length of light diffusion region		Length	
RDC	Additional drain contact resistance			
RSC	Additional source contact resistance			
RD	Drain resistance			
RS	Source resistance			
WMLT	Channel width shrink factor			
LMLT	Channel length shrink factor			
NLEV	Noise model selector			
GDSNOI	Channel thermal noise coefficient			
COMPAT	Model compatibility selector: HSPICE, AWR, or, Spectre.			AWR

Implementation Details

BSIM3V322 is the Microwave Office implementation of version 3.2.2 of the BSIM3 model, developed at the University of California, Berkeley (UCB), as an industry-wide standard MOSFET model. The model conforms to the description in the defining document, Ref. [1] (except for obvious errors; see Special Implementation Notes). Additional information on the BSIM model, although not up-to-date, can be found in Ref. [2] and [3].

The MW Office implementation attempts to be as close to the UCB SPICE implementation as possible. However, the following parameters have been added to support HSPICE enhancements to Berkeley's original model: PHI, XL, XW, AFAC (M in HSPICE), TLEV, TLEVC, PTC, EG, GAP1, GAP2, DCAP, ACM, CALCACM, HDIF, LD, LDIF, RDC, RSC, RD, RS, WMLT, LMLT, NLEV, and GDSNOI.

The NFLAG and COMPAT parameters are unique to the MW Office implementation. The flag NFLAG allows to turn the device noise on and off. COMPAT, the compatibility selection flag, can take three different values: HSPICE, AWR and SPECTRE. The HSPICE and Spectre values emulate HSPICE and Spectre implementations of the BSIM3, version 3.2.2 model, respectively. Parameters LMIN, WMAX,WMIN, and LMAX are not implemented or mapped to HSPICE.

This model is mapped to HSPICE as a NMOS M-device with parameters LEVEL, and VERS set to 49,and 3.22, respectively. Noise, binning and HSPICE Area Calculation Methods (ACM values of 0,2,10 and 12) are supported by the model. Those parameters supporting binning are indicated in the table above.

CALCULATED DEFAULTS: BSIM3 has a set of rather involved conditions for calculating the default values of certain parameters. These calculated defaults are used when a value is "not given"; i.e., not provided in the list of parameters. In a netlist simulator, like SPICE, it is a simple matter to neglect to provide a parameter; however, in a graphics-oriented user interface, all parameters are automatically listed. Thus, in Microwave Office, "not given" is equivalent to "leaving the field empty". The default values of the following parameters are calculated within the model in case their corresponding fields are left empty.

Parameter	Details of the Calculated Default Value
VTH0	Calculated from VFB.
VFB	Calculated from VTH0 if VTH0 is specified; otherwise, VFB = -1.0
K1, K2	Calculated from GAMMA1 AND GAMMA2, if specified.
GAMMA1, GAMMA2	Calculated from C_{ox} and NCH.
NCH	Calculated from GAMMA1 and Cox; if these are not given, defaults to 1.7e23 m-3.
VBX	Calculated from NCH and XT.
CGS0	Calculated from DLC. If DLC is not given or results in a negative capacitance, calculated from XJ and C_{ox}.
CGD0	Calculated from DLC. If DLC is not given or results in a negative capacitance, calculated from XJ and C_{ox}.
CGB0	Calculated from DWC and C_{ox}.
CJSWG	CJSW.
PBSWG	PBSW
MJSWG	MJSW
CF	Calculated from TOX
DLC	WINT
DWC	WINT
UC	mobMod=1,2: -4.65e-11mobMod=3: -0.0465
DSUB	DROUT
LLC	LL
LWC	LW
LWLC	LWL
WLC	WL
WWC	WW

Parameter	Details of the Calculated Default Value
WWLC	WWL
UC1	mobMod=1, 2:-5.6E-11mobMod=3: -0.056
TOXM	TOX
NSUB	6.0E16 cm^3
VBX	Calculated from PHI, NCH and XT.
XT	1.55E-7 m
PHI	Computed from NCH.
XL	0.0 m
XW	0.0 m
AS	0.0 m^2
AD	0.0 m^2
PS	0.0 m
PD	0.0 m
NRD	1.0
NRS	1.0
U0	(NMOS) 670(PMOS) 250
NOIA	(NMOS) 1e20(PMOS) 9.9e18
NOIB	(NMOS) 5e4(PMOS) 2.4e3
NOIC	(NMOS) -1.4e-12(PMOS) 1.4e-12
ACM	10
HDIF	0.0 m
LD	0.0 m
LDIF	0.0 m
RDC	0.0
RSC	0.0
RD	0.0
RS	0.0
WMLT	1.0
LMLT	1.0
GDSNOI	1.0

NOTE: C_{ox} cannot be specified independently in BSIM3, as it can in other MOSFET models. C_{ox} is calculated as $C_{OX} = \varepsilon_{\delta X} / TOX$ where $\varepsilon_{\delta X} = 3.9\varepsilon_0$

UNITS: Ref. [1] indicates that this model uses an unfortunate choice of units for the input parameters. Most units are in the MKS system, but a few, which are traditionally in other units, are entered in those units. The UCBUNITS parameter allowed the user to circumvent this problem, by making him /her choose a value for this parameter, which indicated whether the units were defined as in [1] (UCBUNITS=1) or are all MKS (UCBUNITS=0). The UCBUNITS parameter has no meaning anymore and has become read only for compatibility purposes. Parameters TEMP, TNOM, U0, NSUB, NGATE and NCH, whose value depended on the setting of UCBUNITS, work differently now. Parameters TEMP and TNOM are interpreted according to the project units setting. Parameter NCH is always interpreted in cm^{-3}. The units corresponding to parameters U0, NSUB, NGATE are assessed from the corresponding parameter value. NSUB and NGATE are interpreted in cm^{-3} if their values are smaller than 1e20 and 1e23, respectively, and in m-3 otherwise. U0 is considered in cm2/vs if it is greater than 1, and in m2/vs otherwise.

Most units in BSIM3 are treated as scalar values; i.e., they are not converted to project units. This is slightly different from most MW Office models; it is done because most parameters, in this very complex model, are not expected to be entered or manipulated by the circuit designer. A very few parameters that may be manipulated, such as L and W, do have project units. See the table of parameters to determine the units of any parameter.

Equations

Equations for the BSIM3V322 model are too extensive to be repeated here. Please see Ref. [1] for this information.

Operating Points

The following letter pairs have been used to identify the NL branches: ds, bs, bd, gs and gd. Here g,d,s abd b correspond to the gate, drain, source and substrate terminals, respectively. These are used to identify branch related operating point information, i.e., branch voltages, currents, etc

Parameter	Description
vth (Voltage)	Threshold voltage
vdsat (Voltage)	Drain-source saturation voltage
gm (Conductance)	Common-source transconductance
gds (Conductance)	Common-source output conductance
gmbs (Conductance)	Body-transconductance
Isub(Current)	Substrate current
cgg (Capacitance)	$\frac{\partial Q_g}{\partial V_g}$, intrinsic charge.
cgb (Capacitance)	$\frac{\partial Q_g}{\partial V_b}$, intrinsic charge.
cgd (Capacitance)	$\frac{\partial Q_g}{\partial V_d}$, intrinsic charge.
cgs (Capacitance)	$\frac{\partial Q_g}{\partial V_s}$, intrinsic charge.
cbg (Capacitance)	$\frac{\partial Q_b}{\partial V_g}$, intrinsic charge.
cbb (Capacitance)	$\frac{\partial Q_b}{\partial V_b}$, intrinsic charge.
cbd (Capacitance)	$\frac{\partial Q_b}{\partial V_d}$, intrinsic charge.

Parameter	Description
cbs (Capacitance)	$\frac{\partial Q_b}{\partial V_s}$, intrinsic charge.
cdg (Capacitance)	$\frac{\partial Q_d}{\partial V_g}$, intrinsic charge.
cdb (Capacitance)	$\frac{\partial Q_d}{\partial V_b}$, intrinsic charge.
cdd (Capacitance)	$\frac{\partial Q_d}{\partial V_d}$, intrinsic charge.
cds (Capacitance)	$\frac{\partial Q_d}{\partial V_s}$, intrinsic charge.
csg (Capacitance)	$\frac{\partial Q_s}{\partial V_g}$, intrinsic charge.
csb (Capacitance)	$\frac{\partial Q_s}{\partial V_b}$, intrinsic charge.
csd (Capacitance)	$\frac{\partial Q_s}{\partial V_d}$, intrinsic charge.
css (Capacitance)	$\frac{\partial Q_s}{\partial V_s}$, intrinsic charge.
cjd (Capacitance)	Drain-substrate junction capacitance
cjs (Capacitance)	Source-substrate junction capacitance
pwr (Power)	Dissipated power
gmoverid (None)	Gm/Ids
ro (Resistance)	Common-source output resistance
Cgs (Capacitance)	$\frac{\partial Q_g}{\partial V_s}$
Cgd (Capacitance)	$\frac{\partial Q_g}{\partial V_d}$
Cgtot (Capacitance)	$\frac{\partial Q_g}{\partial V_g}$
Cbtot (Capacitance)	$\frac{\partial Q_b}{\partial V_b}$
Cdtot (Capacitance)	$\frac{\partial Q_d}{\partial V_d}$

Parameter	Description
Cstot (Capacitance)	$\frac{\partial Q_s}{\partial V_s}$
ft (Frequency)	Unity small-signal current-gain frequency

Special Implementation Notes

Note that much of the information in Ref. [1] is invalid, either because of typographical errors, or simple disagreement with the actual UC Berkeley (UCB) SPICE implementation of the model. In all cases, the SPICE source-code implementation is the standard. The equations in Ref. [1] have been painstakingly checked against the SPICE BSIM3V3.2.2 source code, and where discrepancies between the documentation and code existed, the SPICE source code has been viewed as the correct version. Conversely, we have identified possible errors in the SPICE source code, but in all cases these were minor and rarely will affect calculations. They have been reported to UC Berkeley and may be addressed in a future revision of the model. Berkeley's original implementations also suffered from discontinuities in the derivatives of various charge elements at the vds=0 boundary. This problem has been corrected in the current implementation.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

References

Ref. [1] is the "official" UCB description of this model. It has a large number of errors, frequently serious ones. Ref. [2] presents a good treatment of the model, but it also contains a few typographical errors and does not cover the later versions. Ref. [3] is current, very accurate and has served as a guide for this implementation. Ref. [1] is available on the internet; see http://www-device.eecs.berkeley.edu/~bsim3/get.html/.

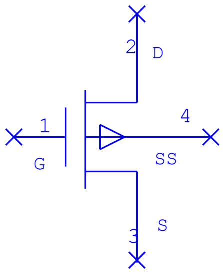
[1] W. Liu et al., BSIM3V3.2.2 MOSFET Model User's Manual, Dept. of Electrical Engineering and Computer Sciences, University of California, Berkeley CA 94720, USA, 1999.

[2] Y. Cheng and C. Hu, MOSFET Modeling & BSIM3 User's Guide, Norwell, MA: Kluwer, 1999.

[3] William Liu, MOSFET Models for Spice Simulation, Including BSIM3v3 and BSIM4, Wiley-IEEE, February 2001.

(Obsolete) BSIM3 Ver. 3.2.2 P-Channel MOSFET Model: BSIM3V322P

Symbol



Summary

This element is OBSOLETE and is replaced by the BSIM3 MOSFET Model (BSIM3) element.

Equivalent Circuit

This model is the same as the BSIM3V322 model, except that the orientation of the currents in all the voltage controlled current sources comprising the model are reversed and the threshold voltage has opposite sign.

Parameters

Parameters are identical to those of BSIM3V322 except for the default values of the following parameters.

Name	Description	Unit Type	Default
U0	Mobility (cm/V sec or m/V)		250
NOIA	Noise parameter A		9.9e18
NOIB	Noise parameter B		2.4e3
NOIC	Noise parameter C		1.4e-12

Implementation Details

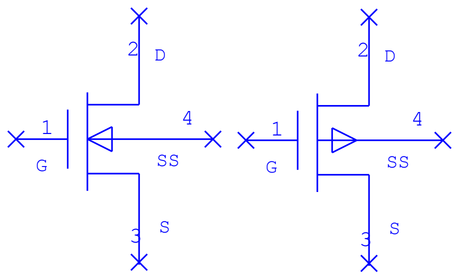
Except for its P-nature, this model is equivalent to its N-channel counterpart (BSIM3V322). This model is mapped into HSPICE as a PMOS M-device with parameters LEVEL, ACM and VERS set to 49, 10 and 3.22, respectively.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

BSIM4 MOSFET Model: BSIM4

Symbol



Summary

NI AWR's BSIM4 element is based on the UC Berkeley BSIM4 model. Versions 4.4 and 4.5 are supported. The BSIM4 model explicitly addresses many issues in modeling sub-0.13 micron technology and RF high-speed circuit simulation. The BSIM4 model has major improvements and additions over BSIM3v3, some of which follow:

- Accurate new model of the intrinsic input resistance for both RF, high-frequency analog and high-speed digital applications
- Flexible substrate resistance network for RF modeling
- New accurate models for channel thermal noise and noise partitioning for the induced gate noise
- Non-Quasi-Static (NQS) model that is consistent with the Rg-based RF model
- Accurate gate direct tunneling model
- Comprehensive and versatile geometry-dependent parasitics model for various source/drain connections and multi-finger devices
- Asymmetrical and bias-dependent source/drain resistance, either internal or external to the intrinsic MOSFET at user discretion
- Quantum mechanical charge-layer-thickness model for both IV and CV
- More accurate mobility model for predictive modeling
- Gate-induced drain leakage (GIDL) current model
- Improved unified flicker (1/f) noise model, which is smooth over all bias regions and considers the bulk charge effect
- Different diode IV and CV characteristics for source and drain junctions

Parameters

Name	Description	Binning	Unit Type	Default
ID	Element ID		Text	M1
TYPE	Device type			1
*TNOM	Parameter extraction temperature		Temperature	294.15DegK
*TEMP	Device temperature		Temperature	294.15DegK
*NFLAG	Noise flag			1
*MULT	Number of devices in parallel			1

Name	Description	Binning	Unit Type	Default
*W	Channel width (m)			5e-6
*L	Channel length (m)			5e-6
*NF	Number of device fingers			1
*MIN	Minimum number of device fingers			0
*AS	Area of source diffusion (m^2)			
*AD	Area of drain diffusion (m^2)			
*PS	Perimeter of source diffusion (m)			
*PD	Perimeter of drain diffusion (m)			
*NRS	Number of squares of source diffusion			1
*NRD	Number of squares of drain diffusion			1
*SA	Distance between OD edge to poly of one side			0
*SB	Distance between OD edge to poly of the other side			0
*SD	Distance between neighbor fingers			0
*RGEOMOD	Diffusion resistance and contact model flag			0
*MOBMOD	Mobility model selector			0
*BINUNIT	Units of size parameters in binning approach			1
*PARAMCHK	Parameter value check flag			1
*CAPMOD	Intrinsic charge model			2
*DIOMOD	Diode model selector			1
*RDSMOD	Bias-dependent D/S model selector			0
*TRNQSMOD	Transient NQS flag			0
*ACNQSMOD	AC NQS flag			0
*RBODYMOD	Rbody flag			0
*RGATEMOD	Rgate flag			0
*PERMOD	Perimeter model selector			1
*GEOMOD	geometry flag			0
*FNOIMOD	Flicker noise model selector			1
*TNOIMOD	Thermal noise model selector			0
*IGCMOD	Gate-to-channel tunneling model selector			0
*IGBMOD	Gate-to-substrate tunneling model selector			0
*TEMPMOD	Temperature mode selector			0
*TOXREF	Nominal get oxide thickness for gate dielectric tunneling current model only			30e-10
*TOXE	Electrical gate oxide thickness (m)			30e-10
*TOXP	Electrical gate oxide thickness (m)			
*TOXM	Toxe at which parameters were extracted (m)			

Name	Description	Binning	Unit Type	Default
*DTOX	Difference between electrical and physical gate oxide thickness (m)			0
*EPSROX	Gate dielectric constant			3.9
*CDSC	Source/drain and channel coupling capacitance (F/m^2)			2.4e-4
*CDSCB	Body-bias dependence of cdsc (F/m^2*v)			0
*CDSCD	Drain-bias dependence of cdsc (F/m^2*v)			0
*CIT	Interface trap parameter for subthreshold swing (F/m^2)			0
*NFACTOR	Subthreshold swing coefficient			1
*XJ	Source/drain junction depth (m)			0.15e-6
*VSAT	Carrier saturation velocity at tnom (m/s)			8.0e4
*A0	Nonuniform depletion width effect coefficient			1
*AGS	Gate-bias dependence of Abulk (F/m^2*V)			0
*A1	No-saturation coefficient			0
*A2	No-saturation coefficient			1
*AT	Temperature coefficient for vsat (m/s)			3.3e4
*KETA	Body-bias coefficient for non-uniform depletion width effect (1/V)			-0.047
*NSUB	Substrate doping concentration (cm^-3)			6e16
*NDEP	Channel doping concentration (cm^-3)			1.7e17
*NSD	Source-drain dropping concentration (cm^-3)			1e20
*NGATE	Poly-gate doping concentration (cm^-3)			0
*GAMMA1	Body-effect coefficient near the surface			
*GAMMA2	Body-effect coefficient in the bulk			
*VBX	Threshold voltage transition body voltage			
*VBM	Maximum applied body voltage			-3
*XT	Doping depth (m)			1.55e-7
*K1	Body-effect coefficient			
*KT1	Temperature coefficient for threshold voltage (V)			-0.11
*KT1L	Temperature coefficient for threshold voltage (v*m)			0
*KT2	Temperature coefficient for threshold voltage			0.022
*K2	Charge-sharing parameter			
*K3	Narrow width coefficient			80
*K3B	Narrow width coefficient			0
*LPE0	Lateral non-uniform doping at Vbs=0			1.74e-7
*LPEB	Lateral non-uniform doping effect on K1			0

Name	Description	Binning	Unit Type	Default
*DVTP0	First coefficient of drain-incduced Vth shift for long-channel pocket devices (m)			0
*DVTP1	Second coefficient of drain-induced Vth shift for long-channel pocket devices (1/V)			0
*W0	Narrow width coefficient			2.5e-6
*DVT0	First coefficient of short-channel effects			2.2
*DVT1	Second coefficient of short-channel effects			0.53
*DVT2	Body-bias coefficient of short-channel effects (1/V)			-0.032
*DVT0W	First coefficient of narrow-width effects			0
*DVT1W	Second coefficient of narrow-width effects (1/m)			5.3e6
*DVT2W	Body-bias coefficient of narrow-width effects			-0.032
*DROUT	DIBL effect on output resistance coefficient			0.56
*DSUB	DIBL effect in subthreshold region			
*VTH0	Threshold voltage at zero body bias for long-channel devices			
*EU	Exponent for mobility degradation of mobmod=2			
*UA	First-order mobility reduction coefficient (m/v)			
*UA1	Temperature coefficient for ua (m/v)			1e-9
*UB	Second-order mobility reduction coefficient (m^2/v^2)			1e-19
*UB1	Temperature coefficient for ub (m^2/v^2)			-1e-18
*UC	Body-bias dependence of mobility (m/v^2)			
*UC1	Temperature coefficient for uc (m/v^2)			
*U0	Low-field surface mobility at tnom (cm^2/V*S)			
*UTE	Mobility temperature exponent			-1.5
*VOFF	Threshold voltage offset (V)			-0.08
*VOFFL	Channel-length dependence of Voff (V)			0
*MINV	Vgsteff fitting parameter for moderate inversion condition			0
*FPROUT	Effect of pocket implant on Rout degradation (V/m^0.5)			0
*PDITS	Effect of pocket implant on Rout degradation (1/V)			0
*PDITSD	Channel-length of drain-induced Vth shift on Rout (1/V)			0
*PDITSL	Channel-length of drain-induced Vth shift on Rout (1/m)			0
*DELTA	Effective drain voltage smoothing parameter (v)			0.01

Name	Description	Binning	Unit Type	Default
*RDSW	Zero bias LDD resistance per unit width for RDSMOD=0			200
*RDSWMIN	LDD resistance per unit width at high Vgs and zero Vbs for RDSMOD=0			0
*RDWMIN	LDD resistance per unit width at high Vgs and zero Vbs for RDSMOD=1			0
*RSWMIN	LDD resistance per unit width at high Vgs and zero Vbs for RDSMOD=1			0
*RDW	Zero bias LDD resistance per unit width for RDSMOD=1			100
*RSW	Zero bias LDD resistance per unit width for RDSMOD=1			100
*PRWG	Gate-effect coefficient for Rds (1/V)			1
*PRWB	Body-effect coefficient for Rds (v^-0.5)			0
*PRT	Temperature coefficient for Rds (ohm*m)			0
*ETA0	DIBL coefficient subthreshold region			0.08
*ETAB	Body-bias dependence of et0 (1/V)			-0.07
*PCLM	Channel length modulation coefficient			1.3
*PDIBL1	First coefficient of drain-induced barrier lowering			0.39
*PDIBL2	Second coefficient of drain-induced barrier lowering			0.0086
*PDIBLB	Body-effect coefficient for DIBL (1/V)			0
*PSCBE1	First coefficient of substrate current body effect (v/m)			4.24e8
*PSCBE2	Second coefficient of substrate current body effect (m/v)			1e-5
*PVAG	Gate dependence of Early voltage			0
*WR	Width offset for parasitic resistance			1
*DWG	Gate-bias dependence of channel width (m/v)			0
*DWB	Body-bias dependence of channel width (m/v^0.5)			0
*B0	Bulk charge coefficient due to narrow width effect (m)			0
*B1	Bulk charge coefficient due to narrow width effect (m)			0
*ALPHA0	Substrate current impact ionization coefficient (m/v)			0
*ALPHA1	Substrate current impact ionization coefficient (1/V)			0
*AGIDL	Pre-exponential coefficient for GIDL (1/ohm)			0
*BGIDL	Exponential coefficient for GIDL (v)			2.3e9

Name	Description	Binning	Unit Type	Default
*CGIDL	Exponential coefficient for GIDL (V^3)			0.5
*PHIN	Non-uniform vertical doping effect on surface potential			0
*EGIDL	Fitting parameter for band bending for GIDL (V)			0.8
*AIGC	Parameter for Igcs and Igcd			
*BIGC	Parameter for Igcs and Igcd			
*CIGC	Parameter for Igcs and Igcd (1/V)			
*AIGSD	Parameter for Igs and Igd			
*BIGSD	Parameter for Igs and Igd			
*CIGSD	Parameter for Igs and Igd (1/V)			
*AIGBACC	Parameter for Igb in accumulation			0.43
*BIGBACC	Parameter for Igb in accumulation			0.054
*CIGBACC	Parameter for Igb in accumulation (1/V)			0.075
*AIGBINV	Parameter for Igb in inversion			0.35
*BIGBINV	Parameter for Igb in inversion			0.03
*CIGBINV	Parameter for Igb in inversion (1/V)			0.006
*NIGC	Source/drain overlap length for Igs and Igd			1
*NIGBINV	Parameter for Igb in inversion			3
*NIGBACC	Parameter for Igb in accumulation			1
*NTOX	Exponent for the gate oxide ratio			1
*EIGBINV	Parameter for Igb in inversion (V)			1.1
*PIGCD	Vds dependence of Igcs and Igcd			1
*POXEDGE	Factor for the gate oxide thickness in source/drain overlap regions			1
*XRCRG1	Parameter for distributed channel-resistance effect for NQS models			12
*XRCRG2	Parameter to account for the excess channel diffusion resistance for NQS models			1
*LAMBDA	Velocity overshoot coefficient (m/s)			0
*VTL	Thermal velocity (m/s)			
*XN	Velocity back scattering coefficient			3
*LC	Velocity back scattering coefficient (m)			5e-9
*TNOIA	Coefficient of channel-length dependence of total channel thermal noise			1.5
*TNOIB	Coefficient of channel-length dependence of total channel thermal noise			3.5
*RNOIA	Thermal noise coefficient A			0.577
*RNOIB	Thermal noise coefficient B			0.5164

Name	Description	Binning	Unit Type	Default
*NTNOI	Noise factor for short-channel devices for TNOIMOD=0 only			1
*VFBSDOFF	Flatband voltage offset parameter (V)			0
*LINTNOI	Length reduction parameter offset (m)			0
*SAREF	Reference distance between od edge to poly of one side (m)			1e-6
*SBREF	Reference distance between od edge to poly of the other side (m)			1e-6
*WLOD	Length parameter for stress effect (m)			0
*KU0	Mobility degradation/ enhancement coefficient for stress effect (m)			0
*KVSAT	Saturation velocity degradation/ enhancement parameter for stress effect			0
*KVTH0	Threshold shift parameter for stress effect (v*m)			0
*TKU0	Temperature coefficient of ku0			0
*LLODKU0	Length parameter for u0 stress effect			0
*WLODKU0	Width parameter for u0 stress effect			0
*LLODVTH	Length parameter for Vth stress effect			0
*WLODVTH	Width parameter for Vth stress effect			0
*LKU0	Length dependence of ku0			0
*WKU0	Width dependence of ku0			0
*PKU0	Cross-term dependence of ku0			0
*LKVTH0	u0 length dependence of kvth0			0
*WKVTH0	u0 width dependence of kvth0			0
*PKVTH0	Cross-term dependence of kvth0			0
*STK2	k2 shift factor related to vth0 change (m)			0
*LODK2	k2 shift modification factor for stress effect			1
*STETA0	eta0 shift factor related to vth0 change (m)			0
*LODETA0	eta0 shift modification factor for stress effect			1
*BETA0	Substrate current impact ionization exponent (1/V)			30
*IJTHDFWD	Limiting current in forward bias region (A) (Not implemented)			
*IJTHSFWD	Limiting current in forward bias region (A) (Not implemented)			0.1
*IJTHDREV	Limiting current in reverse bias region (A) (Not implemented)			
*IJTHSREV	Limiting current in reverse bias region (A) (Not implemented)			0.1
*XJBVD	Limiting current in forward bias region			

Name	Description	Binning	Unit Type	Default
*XJBVS	Limiting current in forward bias region			1
*BVD	Breakdown voltage (V)			
*BVS	Breakdown voltage (V)			10
*JTSS	Bottom trap-assisted saturation current density (A/m)			0
*JTSD	Bottom trap-assisted saturation current density (A/m)			
*JTSSWS	STI sidewall trap-assisted saturation current density (A/m^2)			0
*JTSSWD	STI sidewall trap-assisted saturation current density (A/m^2)			
*JTSSWGS	Gate-edge sidewall trap-assisted saturation current density (A/m)			0
*JTSSWGD	Gate-edge sidewall trap-assisted saturation current density (A/m)			
*NJTS	Non ideality factor for JTSS and JTSD			20
*NJTSSW	Non ideality factor for JTSSWS and JTSSWD			20
*NJTSSWG	Non-ideality factor for JTSSWGS and JTSSWGD			20
*XTSS	Power dependence of JTSS on temperature			0.02
*XTSD	Power dependence of JTSD on temperature			
*XTSSWS	Power dependence of JTSSWS on temperature			0.02
*XTSSWD	Power dependence of JTSSWD on temperature			
*XTSSWGS	Power dependence of JTSSWGS on temperature			0.02
*XTSSWGD	Power dependence of JTSSWGD on temperature			
*TNJTS	Temperature coefficient for NJTS			0
*TNJTSSW	Temperature coefficient for NJTSSW			0
*TNJTSSWG	Temperature coefficient for NJTSSWG			0
*VTSS	Bottom trap-assisted voltage dependent parameter			10
*VTSD	Bottom trap-assisted voltage dependent parameter			
*VTSSWS	STI sidewall trap-assisted voltage dependent parameter			10
*VTSSWD	STI sidewall trap-assisted voltage dependent parameter			
*VTSSWGS	Gate-edge sidewall trap-assisted voltage dependent parameter			10
*VTSSWGD	Gate-edge sidewall trap-assisted voltage dependent parameter			
*VFB	Flat-band voltage			

Name	Description	Binning	Unit Type	Default
*GBMIN	Conductance in parallel with each of the five substrate resistances to avoid potential numerical instability			1e-12
*RBDB	Resistance connected between db and b nodes			50
*RBPB	Resistance connected between b and b nodes			50
*RBSB	Resistance connected between sb and b nodes			50
*RBPS	Resistance connected between b and s nodes			50
*RBPD	Resistance connected between b and d nodes			50
*CGSL	Overlap capacitance between gate and lightly-doped source region			0
*CGDL	Overlap capacitance between gate and lightly-doped drain region (F/m)			0
*CKAPPAS	Coefficient of bias-dependent overlap capacitance for the source side			0.6
*CKAPPAD	Coefficient of bias-dependent overlap capacitance for the source side (V)			
*CF	Coefficient of bias-dependent overlap capacitance for the source side (F/m)			
*CLC	Constant term for the short channel model (m)			0.1e-6
*CLE	Intrinsic capacitance fitting parameter			0.6
*DWC	Delta W for capacitance model (m)			
*DLC	Delta L for capacitance model (m)			
*XW	Width variation due to masking etching (m)			0
*XL	Length variation due to masking and etching (m)			0
*DLCIG	Source/drain overlap length for lgs and lgd (m)			
*DWJ	Offset of the S/D junction width			
*VFBCV	Flat-band voltage for capmod=0			-1
*ACDE	Exponential coefficient for charge thickness in CAPMOD=2 for accumulation and depletion regions (m/v)			1
*MOIN	Exponential coefficient for charge thickness for accumulation and depletion regions (1/V)			15
*NOFF	Transition parameter			1
*VOFFCV	CV parameter in VgsteffCV for weak to strong inversion (V)			0
*DMCG	Distance from S/D contact center to the gate edge (m)			0
*DMCI	Distance from S/D contact center to the isolation edge in the channel-length direction (m)			

Name	Description	Binning	Unit Type	Default
*DMDG	Distance from S/D contact center to the gate edge (m)			0
*DMCGT	DMCG of test structures (m)			0
*XGW	Distance from the gate contact to the channel edge (m)			0
*XGL	Offset of the gate length due to variations in patterning (m)			0
*RSHG	Gate electrode diffusion sheet resistance			0.1
*NGCON	Number of gate contacts			1
*TCJ	Temperature coefficient for cj (1/C)			0
*TPB	Temperature coefficient for pb (V/C)			0
*TCJSW	Temperature coefficient for cjsw (1/C)			0
*TPBSW	Temperature coefficient for pbsw (V/C)			0
*TCJSWG	Temperature coefficient for cjswg (1/C)			0
*TPBSWG	Temperature coefficient for pbswg (V/C)			0
*CDSC	Length, width, product	yes		
*CDSCB	Length, width, product	yes		
*CDSCD	Length, width, product	yes		
*CIT	Length, width, product	yes		
*NFACTOR	Length, width, product	yes		
*XJ	Length, width, product	yes		
*VSAT	Length, width, product	yes		
*A0	Length, width, product	yes		
*AGS	Length, width, product	yes		
*A1	Length, width, product	yes		
*A2	Length, width, product	yes		
*AT	Length, width, product	yes		
*KETA	Length, width, product	yes		
*NSUB	Length, width, product	yes		
*NDEP	Length, width, product	yes		
*NSD	Length, width, product	yes		
*NGATE	Length, width, product	yes		
*GAMMA1	Length, width, product	yes		
*GAMMA2	Length, width, product	yes		
*VBX	Length, width, product	yes		
*VBM	Length, width, product	yes		
*XT	Length, width, product	yes		

Name	Description	Binning	Unit Type	Default
*K1	Length, width, product	yes		
*KT1	Length, width, product	yes		
*KT1L	Length, width, product	yes		
*KT2	Length, width, product	yes		
*K2	Length, width, product	yes		
*K3	Length, width, product	yes		
*K3B	Length, width, product	yes		
*LPE0	Length, width, product	yes		
*LPEB	Length, width, product	yes		
*DVTP0	Length, width, product	yes		
*DVTP1	Length, width, product	yes		
*W0	Length, width, product	yes		
*DVT0	Length, width, product	yes		
*DVT1	Length, width, product	yes		
*DVT2	Length, width, product	yes		
*DVT0W	Length, width, product	yes		
*DVT1W	Length, width, product	yes		
*DVT2W	Length, width, product	yes		
*DROUT	Length, width, product	yes		
*DSUB	Length, width, product	yes		
*VTH0	Length, width, product	yes		
*UA	Length, width, product	yes		
*UA1	Length, width, product	yes		
*UB	Length, width, product	yes		
*UB1	Length, width, product	yes		
*UC	Length, width, product	yes		
*UC1	Length, width, product	yes		
*U0	Length, width, product	yes		
*UTE	Length, width, product	yes		
*VOFF	Length, width, product	yes		
*MINV	Length, width, product	yes		
*FPROUT	Length, width, product	yes		
*PDITS	Length, width, product	yes		
*PDITSD	Length, width, product	yes		
*DELTA	Length, width, product	yes		
*RDSW	Length, width, product	yes		
*RDW	Length, width, product	yes		

Name	Description	Binning	Unit Type	Default
*RSW	Length, width, product	yes		
*PRWB	Length, width, product	yes		
*PRWG	Length, width, product	yes		
*PRT	Length, width, product	yes		
*ETA0	Length, width, product	yes		
*ETAB	Length, width, product	yes		
*PCLM	Length, width, product	yes		
*PDIBL1	Length, width, product	yes		
*PDIBL2	Length, width, product	yes		
*PIDBLB	Length, width, product	yes		
*PSCBE1	Length, width, product	yes		
*PSCBE2	Length, width, product	yes		
*PVAG	Length, width, product	yes		
*WR	Length, width, product	yes		
*DWG	Length, width, product	yes		
*DWB	Length, width, product	yes		
*B0	Length, width, product	yes		
*B1	Length, width, product	yes		
*ALPHA0	Length, width, product	yes		
*ALPHA1	Length, width, product	yes		
*BETA0	Length, width, product	yes		
*AGIDL	Length, width, product	yes		
*BGIDL	Length, width, product	yes		
*CGIDL	Length, width, product	yes		
*PHIN	Length, width, product	yes		
*EGIDL	Length, width, product	yes		
*AIGC	Length, width, product	yes		
*BIGC	Length, width, product	yes		
*CIGC	Length, width, product	yes		
*AIGSD	Length, width, product	yes		
*BIGSD	Length, width, product	yes		
*CIGSD	Length, width, product	yes		
*AIGBACC	Length, width, product	yes		
*BIGBACC	Length, width, product	yes		
*CIGBACC	Length, width, product	yes		
*AIGBINV	Length, width, product	yes		
*BIGBINV	Length, width, product	yes		

Name	Description	Binning	Unit Type	Default
*CIGBINV	Length, width, product	yes		
*NIGC	Length, width, product	yes		
*NIGBINV	Length width, product	yes		
*NIGBACC	Length, width, product	yes		
*NTOX	Length, width, product	yes		
*EIGBINV	Length, width, product	yes		
*PIGCD	Length, width, product	yes		
*POXEDGE	Length, width, product	yes		
*XRCRG1	Length, width, product	yes		
*XRCRG2	Length, witch, product	yes		
*LAMBDA	Length, width, product	yes		
*VTL	Length, width, product	yes		
*XN	Length, width, product	yes		
*VFBSDOFF	Length, width, product	yes		
*EU	Length, width, product	yes		
*VFB	Length, width, product	yes		
*CGSL	Length, width, product	yes		
*CGDL	Length, width, product	yes		
*CKAPPAS	Length, width, product	yes		
*CKAPPAD	Length, width, product	yes		
*CF	Length, width, product	yes		
*CLC	Length, width, product	yes		
*CLE	Length, width, product	yes		
*VFBCV	Length, width, product	yes		
*ACDE	Length, width, product	yes		
*MOIN	Length, width, product	yes		
*NOFF	Length, width, product	yes		
*VOFFCV	Length, width, product	yes		
*CGSO	Non LDD region source-gate overlap capacitance per unit channel width (F/m)			
*CGDO	Non LDD region drain-gate overlap capacitance per unit channel width			
*CGBO	Non LDD region substrate-gate overlap capacitance per unit channel width (F/m)			
*XPART	Charge partition number. Use 0.0 for 40/60, 0.5 for 50/50, or 1.0 for 0/100			0
*RSH	Source/drain diffusion sheet resistance			0

Name	Description	Binning	Unit Type	Default
*JSS	Bottom junction reverse saturation current density (A/m^2)			1e-4
*JSWS	Isolation-edge sidewall reverse saturation current density (A/m)			0
*JSWGS	Gate-edge sidewall reverse saturation current density (A/m)			0
*PBS	Bottom junction built-in potential (V)			1
*MJS	Bulk junction bottom grading coefficient			0.5
*PBSWS	Isolation-edge sidewall junction built-in potential (V)			1
*MJSWS	Isolation-edge sidewall junction capacitance grading coefficient			0.33
*CJS	Zero bias bottom junction capacitance per unit area (F/m^2)			5e-4
*CJSWS	Zero-bias junction sidewall capacitance density (F/m)			5e-10
*NJS	Bulk-Source junction emission coefficient			1
*PBSWGS	Gate-edge sidewall junction built-in potential (V)			
*MJSWGS	Gate-edge sidewall junction grading coefficient			
*CJSWGS	Zero-bias gate-side junction capacitance density (F/m)			
*XTIS	Bulk-Source junction saturation current temperature exponent			3
*JSD	Bottom junction reverse saturation current density (A/m^2)			
*JSWD	Isolation-edge sidewall reverse saturation current density (A/m)			
*JSWGD	Gate-edge sidewall reverse saturation current density (A/m)			
*PBD	Bottom junction built-in potential (V)			
*MJD	Bulk junction bottom grading coefficient			
*PBSWD	Isolation-edge sidewall junction built-in potential (V)			
*MJSWD	Isolation-edge sidewall junction capacitance grading coefficient			
*CJD	Zero bias bottom junction capacitance per unit area (F/m^2)			
*CJSWD	Zero-bias junction sidewall capacitance density (F/m)			
*NJD	Bulk-Source junction emission coefficient			
*PBSWGD	Gate-edge sidewall junction built-in potential (V)			

Name	Description	Binning	Unit Type	Default
*MJSWGD	Gate-edge sidewall junction grading coefficient			
*CJSWGD	Zero-bias gate-side junction capacitance density (F/m)			
*XTID	Bulk-Source junction saturation current temperature exponent			
*LINT	Lateral diffusion for one side (m)			0
*LL	Length dependence of delta L			0
*LLC	Length dependence of delta L for CV			
*LLN	Length exponent of delta L			1
*LW	Width dependence of delta L			0
*LWC	Width dependence pf delta L for CV			
*LWN	Width exponent for delta L			1
*LWL	Area dependence of delta L			0
*LWLC	Area dependence of delta L for CV			
*LMIN	Minimum channel length for which the model is valid (m)			0
*LMAX	Maximum channel length for which the model is valid (m)			1
*WINT	Width reduction for one side (m)			0
*WL	Length dependence of delta W			0
*WLC	Length dependence of delta W for CV			
*WLN	Length exponent of delta W			1
*WW	Width dependence of delta W			0
*WWC	Width dependence of delta W for CV			
*WWN	Width exponent of delta W			1
*WWL	Area dependence of delta W			0
*WWLC	Area dependence of delta W for CV			
*WMIN	Minimum channel width for which the model is valid (m)			0
*WMAX	Maximum channel width for which the model is valid (m)			1
*NOIA	Flicker noise parameter A			
*NOIB	Flicker noise parameter B			
*NOIC	Flicker noise parameter C			8.75e9
*EM	Saturation field (V/m)			4.1e7
*EF	Flicker noise frequency exponent			1
*AF	Flicker noise exponent			1
*KF	Flicker noise coefficient			0

Name	Description	Binning	Unit Type	Default
*COMPAT	Model compatibility selector			1
VERSION	Model version			4.4
*SCA	Integral of the first distribution function for scattered well dopant			0.0
*SCB	Integral of the second distribution function for scattered well dopant			0.0
*SCC	Integral of the third distribution function for scattered well dopant			0.0
*SC	Distance to a single well edge			0.0
*DELVTO	Zero bias threshold voltage variation			0.0
*UD	Coulomb scattering factor of mobility	yes		1e14
*UD1	Temperature coefficient of ud	yes		0.0
*UP	Channel length linear factor of mobility	yes		0.0
*LP	Channel length exponential factor of mobility	yes		1e-8
*TVFBSDOFF	Temperature parameter for vfbsdoff	yes		0.0
*TVOFF	Temperature parameter for voff	yes		0.0
*RBPS0	Body resistance RBPS scaling			50
*RBPSL	Body resistance RBPS L scaling			0.0
*RBPSW	Body resistance RBPS W scaling			0.0
*RBPSNF	Body resistance RBPS NF scaling			0.0
*RBPD0	Body resistance RBPD scaling			50
*RBPDL	Body resistance RBPD L scaling			0.0
*RBPDW	Body resistance RBPD W scaling			0.0
*RBPDNF	Body resistance RBPD NF scaling			0.0
*RBPBX0	Body resistance RBPBX scaling			100
*RBPBXL	Body resistance RBPBX L scaling			0.0
*RBPBXW	Body resistance RBPBX W scaling			0.0
*RBPBXNF	Body resistance RBPBX NF scaling			0.0
*RBPBY0	Body resistance RBPBY scaling			100
*RBPBYL	Body resistance RBPBY L scaling			0.0
*RBPBYW	Body resistance RBPBY W scaling			0.0
*RBPBYNF	Body resistance RBPBY NF scaling			0.0
*RBSBX0	Body resistance RBSBX scaling			100
*RBSBY0	Body resistance RBSBY scaling			100
*RBDBX0	Body resistance RBDBX scaling			100
*RBDBY0	Body resistance RBDBY scaling			100
*RBSDBXL	Body resistance RBSDBX L scaling			0.0

Name	Description	Binning	Unit Type	Default
*RBSDBXW	Body resistance RBSDBX W scaling			0.0
*RBSDBXNF	Body resistance RBSDBX NF scaling			0.0
*RBSDBYL	Body resistance RBSDBY L scaling			0.0
*RBSDBYW	Body resistance RBSDBY W scaling			0.0
*RBSDBYNF	Body resistance RBSDBY NF scaling			0.0
*WEB	Coefficient for SCB			0.0
*WEC	Coefficient for SCC			0.0
*KVTH0WE	Threshold shift factor for well proximity effect	yes		0.0
*K2WE	K2 shift factor for well proximity effect	yes		0.0
*KU0WE	Mobility degradation factor for well proximity effect	yes		0.0
*SCREF	Reference distance to calculate SCA, SCB and SCC			1e-6
*WPEMOD	Flag for WPE model (WPEMOD=1 to activate this model			0.0

Operating Points

The following letter pairs are used to identify the NL branches: ds, bs, bd, gs and gd. Here, g, d, s and b correspond to the gate, drain, source and substrate terminals, respectively. These are used to identify branch related operating point information such as branch voltages and currents.

Parameter	Description
vth (Voltage)	Threshold voltage
vdsat (Voltage)	Drain-source saturation voltage
gm (Conductance)	Common-source transconductance
gds (Conductance)	Common-source output conductance
gmbs (Conductance)	Body-transconductance
cgg (Capacitance)	$\frac{\partial Q_g}{\partial V_g}$, intrinsic charge.
cgb (Capacitance)	$\frac{\partial Q_g}{\partial V_b}$, intrinsic charge.
cgd (Capacitance)	$\frac{\partial Q_g}{\partial V_d}$, intrinsic charge.
cgs (Capacitance)	$\frac{\partial Q_g}{\partial V_s}$, intrinsic charge.
cbg (Capacitance)	$\frac{\partial Q_b}{\partial V_g}$, intrinsic charge.

Parameter	Description
cbb (Capacitance)	$\frac{\partial Q_b}{\partial V_b}$, intrinsic charge.
cbd (Capacitance)	$\frac{\partial Q_b}{\partial V_d}$, intrinsic charge.
cbs (Capacitance)	$\frac{\partial Q_b}{\partial V_s}$, intrinsic charge.
cdg (Capacitance)	$\frac{\partial Q_d}{\partial V_g}$, intrinsic charge.
cdb (Capacitance)	$\frac{\partial Q_d}{\partial V_b}$, intrinsic charge.
cdd (Capacitance)	$\frac{\partial Q_d}{\partial V_d}$, intrinsic charge.
cds (Capacitance)	$\frac{\partial Q_d}{\partial V_s}$, intrinsic charge.
csg (Capacitance)	$\frac{\partial Q_s}{\partial V_g}$, intrinsic charge.
csb (Capacitance)	$\frac{\partial Q_s}{\partial V_b}$, intrinsic charge.
csd (Capacitance)	$\frac{\partial Q_s}{\partial V_d}$, intrinsic charge.
css (Capacitance)	$\frac{\partial Q_s}{\partial V_s}$, intrinsic charge.
cjd (Capacitance)	Drain-substrate junction capacitance
cjs (Capacitance)	Source-substrate junction capacitance
pwr (Power)	Dissipated power

Implementation Details

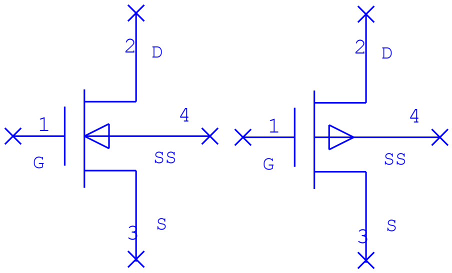
The TYPE parameter controls whether the device is N, or P channel, which is reflected by the device symbol. The NFLAG and COMPAT parameters are unique to NI AWR's implementation. The NFLAG parameter allows you to turn the device noise on or off. COMPAT, the compatibility selection flag, takes three different values: HSPICE, AWR, and SPECTRE. The HSPICE and Spectre values emulate HSPICE and Spectre implementations of the BSIM4 model, respectively. Parameters without default values listed in the previous table get their values from formulas. In many instances, the provided default value depends on the compatibility flag setting. The VERSION parameter allows selecting between versions 4.4 and 4.5 of BSIM4.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

BSIMS6 MOSFET Model: BSIM6

Symbol



Summary

NI AWR's BSIM6 element implements the latest bulk MOSFET model from the BSIM Group. "The model provides excellent accuracy compared to measured data in all regions of operation. It features model symmetry valued for analog and RF applications while maintaining the strong support and performance of the BSIM model valued for all applications since 1996." BSIM6 is Verilog-A based and implements version 1.0.

See the BSIM6 model website[1] [11–77] for complete documentation and Verilog-A definition of component models.

Parameters

Only primary parameters are shown. Secondary parameters follow the BSIM6 model specification/standard.

Name	Description	Binnable	Unit Type	Default
ID	Element ID		Text	X2
VERSION	Version selector			1.0
TYPE	Device type			NMOS
TNOM	Temperature at which params were extracted		Temperature	27
TEMP	Ambient temperature		Temperature	_TEMP
GEOMOD	Geo dependent parasitics model			0
RGEOMOD	Geometry-dependent source/drain resistance			0
COVMOD	Use Bias-independent Overlap Capacitances			0
RDSMOD	Internal s/d resistance model			0
GIDLMOD	Turn off GIDL Current			0
RGATEMOD	Gate resistance model selector			0
RBODYMOD	Distributed body R model			0

Name	Description	Binnable	Unit Type	Default
IGCMOD	Turn off Igc, Igs and Igd			0
IGBMOD	Turn off Igb			0
TNOIMOD	Thermal noise model selector			0
NF	Number of fingers			1
PERMOD	Whether PS/PD (when given) include gate-edge perimeter			1
MINZ	Minimize either D or S			0
NGCON	Number of gate contacts			1

Operating Points

Operating point information is identical to that found in the Verilog-A definition of the model.

Implementation Details

This model is only supported in APLAC measurements. The TYPE parameter controls whether the device is NMOS or PMOS. The extraction and simulation temperatures are controlled using the TNOM and TEMP parameters, respectively. Parameter default and truncation values are identical to those found in the Verilog-A definition of the model.

Layout

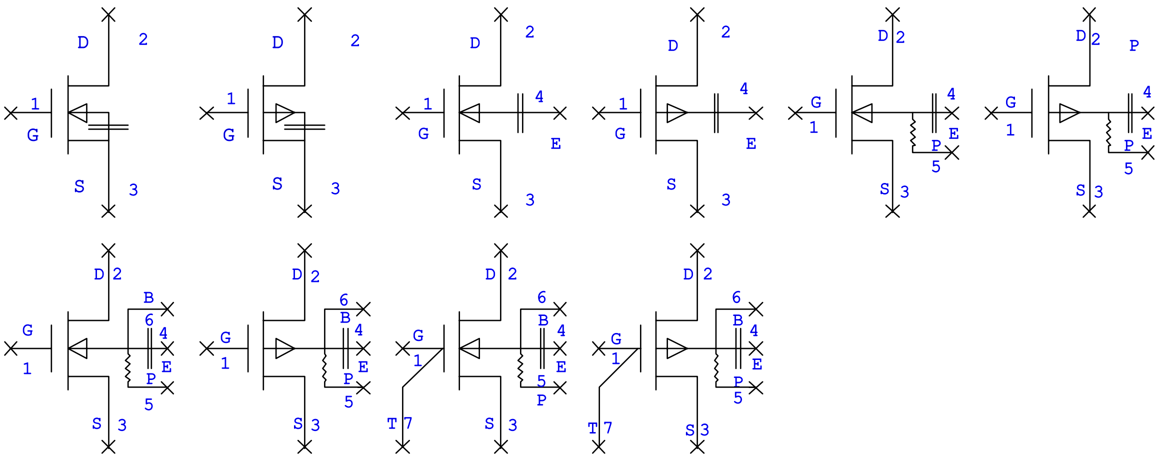
This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

References

[1] https://awrcorp.com/support/help.aspx?id=55

BSIMSOI MOSFET Model: BSIMSOI

Symbol



Summary

NI AWR's BSIMSOI element is a MOSFET model for SOI (Silicon-On-Insulator) circuit design. This model is formulated on top of the BSIM3 framework. It shares the same basic equations with the bulk model so that the physical nature and smoothness of BSIM3v3 are preserved. BSIMSOI is Verilog-A based and implements version 4.4.

See the BSIM model website[1] [11–79] for complete documentation and Verilog-A definition of component models.

Parameters

Only primary parameters are shown. Secondary parameters follow the BSIMSOI model specification/standard.

Name	Description	Binnable	Unit Type	Default
ID	Element ID		Text	X1
VERSION	Version selector			4.4
TYPE	Device type			NMOS
TERMS	Number of terminals exposed			g d s e p
TNOM	Temperature at which params were extracted		Temperature	27
TEMP	Ambient temperature		Temperature	_TEMP

Operating Points

You can access operating point information, as defined by the active Verilog-A based component model.

Implementation Details

This model is only supported in APLAC measurements.

The TYPE parameter controls whether the device is NMOS or PMOS and the TERMS parameter controls which terminals are exposed. You can select five different values: "g d s", "g d s e", "g d s e p", "g d s e p b", and "g d s e p b t". The terminal names are the same as those used in the Verilog-A definition of the model. The current setting of any of these parameters is reflected by the device symbol. The extraction and simulation temperatures are controlled using the TNOM and TEMP parameters, respectively. Parameter default and truncation values are identical to those found in the Verilog-A definition of the model.

Layout

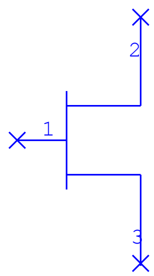
This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

References

[1] http://www-device.eecs.berkeley.edu/bsim/

Curtice FET Model (IV Only): CFET

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Name used to identify the FET	Text	F1
A0	A0 coefficient		0.5
A1	A1 coefficient		0.3
A2	A2 coefficient		-0.05
A3	A3 coefficient		-0.03
Beta	Beta		0
Gamma	Gamma		1
Tau	Time delay		0
Vdo	Vdo		1
Vt	Vt		-2.5

Implementation Details

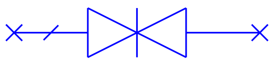
CFET is a Curtice FET model consisting only of the controlled current source. It is intended for testing and model building. The CURTICE model should be used for ordinary analysis.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

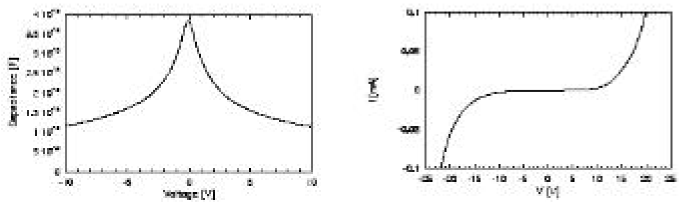
Chalmers Heterostructure Barrier Varactor Diode Model: CHAL_HBV

Symbol

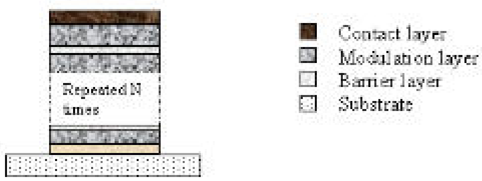


Summary

The HBV diode is a unipolar device which consists of a symmetric layer structure. An undoped high band gap material (barrier) is sandwiched between two moderately n-doped, low band gap materials. The barrier prevents electron transport through the structure. When the diode is biased a depleted region builds up, causing a nonlinear capacitance voltage characteristic. For more information about the HBV's, see [1] [11–84] .

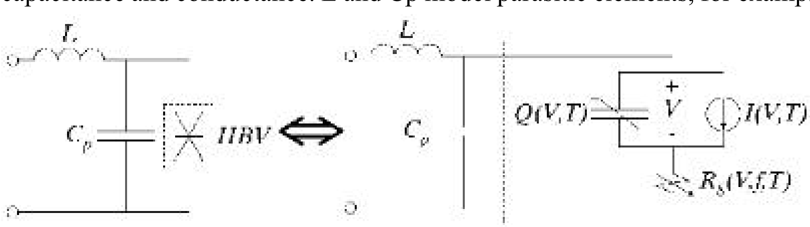


An epitaxial layer structure of an N-barrier HBV is shown in the following figure.



Equivalent Circuit

The model of the HBV consists of a parasitic series resistance in series with a shunt- connected voltage dependent capacitance and conductance. L and Cp model parasitic elements, for example the airbridge and pad-to-pad capacitance.



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	1
A	Device area	um^2	HBV1
b	Barrier thickness	nm	0.02
s	Undoped spacer layer thickness	nm	0.0035
epsb	Dielectric constant in the barrier material	F/m	9.6e-11
epsd	Dielectric constant in the modulation layer	F/m	1.13e-10

Name	Description	Unit Type	Default
1	Length of the modulation layer	nm	250
N	Number of barriers		4
Nd	Doping concentration in the modulation layer	m^-3	8e22
Rs	Parasitic series resistance	Ω	10
T	Device temperature	K	26.85
a	An empirical constant (I-V)	A/(m^2K^2)	170
E0	An empirical constant (I-V)	V/m	4.2e6
Fib	An empirical constant (I-V)	eV	0.17
Cp	Parasitic pad-to-pad capacitance	pF	0
L	Series inductance (to model airbridge inductance)	nH	0

Implementation Details

C-V Characteristic

An accurate quasi-empirical expression for the C-V characteristic of a homogeneously doped HBV is described in [2] [11–84].

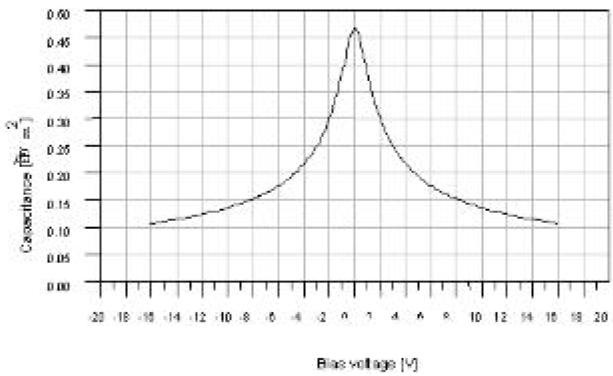
The voltage across the nonlinear capacitor is expressed as a function of its charge as:

$$V(Q) = N(\frac{\mathrm{bQ}}{\varepsilon_b A} + 2\frac{\mathrm{sQ}}{\varepsilon_d A} + \mathrm{sign}(Q)[\frac{Q^2}{2\mathrm{qN}_d\varepsilon_d A^2} + \frac{4\mathrm{KT}}{q}(1 - e^{-\frac{|Q|}{2L_D \mathrm{AqN}_d}})])$$

where Q is the charge stored in the HBV and L_D is the extrinsic Debye length:

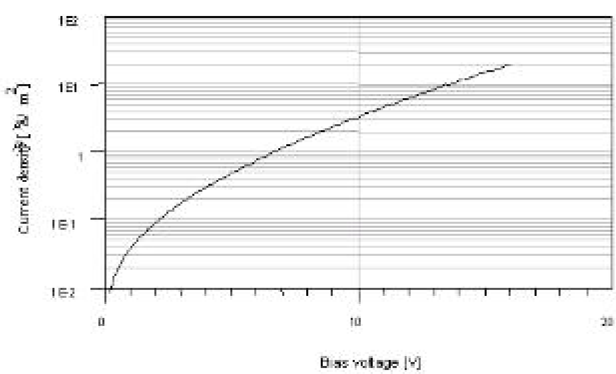
$$L_D = \sqrt{\frac{\mathrm{KT}\varepsilon_d}{q^2 N_d}}$$

The C-V characteristic for an HBV is symmetrical and the following figure shows the C-V characteristic for a typical HBV with the Chalmers model.



I/V Characteristic

The following figure shows the current density versus applied voltage for a typical AlGaAsHBV.



The current through the heterojunction barrier as a function of temperature and voltage can be described as:

$$I(E_b) = A \cdot a \cdot T^2 \cdot \sinh(\frac{E_b}{E_0}) \cdot e^{-(\varphi_b/\mathrm{KT})}$$

where E_b is the electric field in the barrier and is expressed as a function of the voltage across the capacitor [2] as:

$$E_b(V) = \mathrm{sign}(V) \cdot q\mathrm{Nd}\frac{b\varepsilon_d + 2s\varepsilon_b}{\varepsilon_b^2}[\sqrt{1 + \frac{2\varepsilon_d\varepsilon_b^2 \mid V \mid}{\mathrm{NqN}_d(b\varepsilon_d + 2s\varepsilon_b)^2}} - 1]$$

This IV-model is accurate when the thermionic emission dominates. To neglect the influence of leakage current, set a=0.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

References

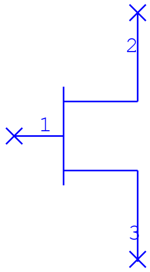
[1] J. Stake, "Varactors," in Modern Microwave and RF Handbook, M.Golio, Ed.: CRC press, 2001.

[2] L.Dillner, J.Stake, and E.Kollberg, "Modeling of the heterostructure barrier varactor diode," presented at Int. Semiconductor Device Symp., Charlottesville, December 1997.

[3] J.Stake, L.Dillner, S.H.Jones, C.M.Mann, J.Thornton, J.R.Jones, W.L.Bishop, and E.L.Kollberg, "Effects of self-heating on planar heterostructure barrier varactor diodes," IEEE Transaction on Electron Devices, Vol.45, No.11, pp.2298-2303, 1998

(Obsolete) Curtice Cubic Nonlinear FET Model: CURTICE

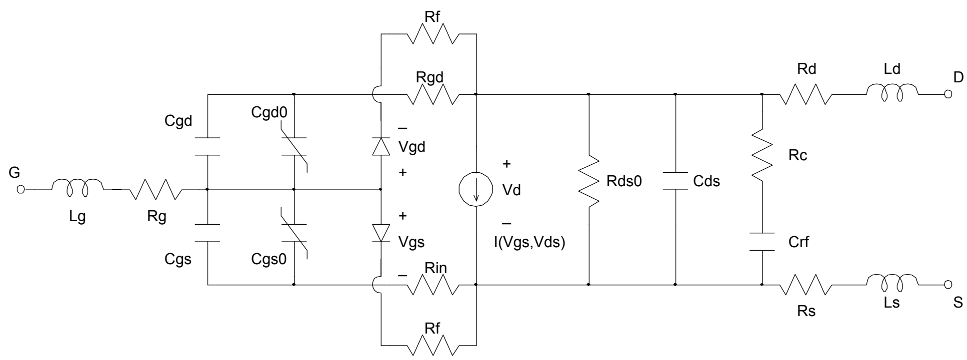
Symbol



Summary

This element is OBSOLETE and is replaced by the Curtice Cubic Nonlinear FET Model (CURTICE3) element.

Equivalent Circuit



Parameters

Name	Description	Unit Type	Default
ID	Device ID	Text	CF1
*BETA	Subthreshold conduction parameter		0
*GAMMA	Drain I/V knee parameter		2
*VOUTO	Vds at which subthreshold effects begin	Voltage	3 V
*VTO	Pinch-off voltage	Voltage	-1 V
*A0	Gate I/V polynomial coefficient		0.065
*A1	Gate I/V polynomial coefficient		0.0140
*A2	Gate I/V polynomial coefficient		0.013
*A3	Gate I/V polynomial coefficient		0
*TAU	Gate-drain time delay	Time	0 ns

Name	Description	Unit Type	Default
*R1	(Not implemented)	Resistance	0.001 ohm
*R2	(Not implemented)	Resistance	0.001 ohm
*VBO	Gate-channel breakdown voltage (pos.)	Voltage	1e+06 V
*VBI	Gate-source capacitance built-in voltage	Voltage	1 V
*RF	Gate-diode forward-bias resistance	Resistance	1e+06 ohm
*IS	Gate-conduction diode current parameter	Current	1e-11 mA
*N	Gate-conduction diode ideality factor		1
*RDS	RF drain-source resistance	Resistance	1e+06 ohm
*CRF	Capacitance that sets the RF Rds break freq	Capacitance	0 pF
*RD	Drain resistance	Resistance	0.001 ohm
*RG	Gate resistance	Resistance	0.001 ohm
*RS	Source resistance	Resistance	0.001 ohm
*RIN	Intrinsic resistance	Resistance	0.001 ohm
*CGSO	Gate-source capacitance at 0V	Capacitance	0 pF
*CGDO	Gate-drain capacitance at 0V	Capacitance	0 pF
*FC	Gate-capacitance linearization parameter		0.5
*CDS	Drain-source capacitance	Capacitance	0 pF
*CGS	Fixed gate-source capacitance	Capacitance	0 pF
*CGD	Fixed gate-drain capacitance	Capacitance	0 pF
*TNOM	Temperature	Temperature	27 DegC
*LAMBDA	DC drain-source resistance parameter		0
*RGD	Gate-drain resistance	Resistance	0.001 ohm
*RDSO	Constant drain-source resistance	Resistance	1e+06 hm
*LG	Gate inductance	Inductance	0 nH
*LS	Source inductance	Inductance	0 nH
*LD	Drain inductance	Inductance	0 nH
*P	Noise par: P		
*Tg	Noise par: gate noise temp		
*KF	Flicker noise coefficient		
*AF	Flicker noise exponent		
*FFE	Flicker noise freq. exponent		
*NFLAG	Noise model		
AFAC	Gate-width scale factor		1
NFING	Number of gate fingers scale factor		1

** indicates a secondary parameter*

NOTES:

1. Use MKS units for parameters that have Scalar units. These are not adjusted to the default units of the project.
2. Parasitic inductances are not scaled.

Implementation Details

I/V Characteristic

The equations for the Curtice model are as follows:

$$I_d = \mathrm{AFAC}(A0 + A1 \cdot V_1 + A2 \cdot V_1^2 + A3 \cdot V_1^3) \cdot (1 + \mathrm{LAMBDA} \cdot V_{\mathrm{ds}})\mathrm{Tanb}(\mathrm{GAMMA} \cdot V_{\mathrm{ds}})$$

where

$$V_1 = V_{\mathrm{gs}}(t - \mathrm{TT})(1 - \mathrm{BETA}(V_{\mathrm{ds}} - \mathrm{VOUTO}))$$

V_1 is limited to $V_1 > \mathrm{VTO}$ in the following manner:

$$V_1 \rightarrow \mathrm{VT0} + 0 \cdot 5(V_1 - \mathrm{VTO} + \sqrt{(V_1 - \mathrm{VTO})^2 + \delta}\ )$$

where δ = 0.0025 VTO. This equation guarantees that the transconductance decreases smoothly to zero below pinchoff.

The form of these equations does not guarantee that the channel pinches off at V_1 = VTO. If the drain I/V function does not pinch off properly, porr convergence may result. In MW Office, the A0 and A1 parameters are modified to guarantee proper pinchoff characteristics when the original values do not do so. This may result in differences between MW Office's results and other simulators using the Curtice model.

Capacitance

The nonlinear gate-to-source and gate-to-drain capacitances, C_{gs0} and C_{gd0}, are modeled as PNCAP elements. See PNCAP for the equations.

Gate Diodes

The gate-to-source and gate-to-drain diodes are modeled as PNIV elements. The equations are listed in the description of SDIODE.

Parameter Scaling

$$\mathrm{RG} \rightarrow \frac{\mathrm{RG} \cdot \mathrm{AFAC}}{(\mathrm{NFING})^2}$$

$$\mathrm{RD} \rightarrow \frac{\mathrm{RD}}{\mathrm{AFAC}}$$

$$\mathrm{RS} \rightarrow \frac{\mathrm{RS}}{\mathrm{AFAC}}$$

$$R_{\mathrm{in}} \rightarrow \frac{R_{\mathrm{in}}}{\mathrm{AFAC}}$$

$$R_{gd} \rightarrow \frac{R_{gd}}{\text{AFAC}}$$

$$R_{rf} \rightarrow \frac{R_{rf}}{\text{AFAC}}$$

$$R_{ds} \rightarrow \frac{R_{ds}}{\text{AFAC}}$$

$$R_{ds0} \rightarrow \frac{R_{ds0}}{\text{AFAC}}$$

$$C_{rf} \rightarrow C_{rf} \cdot \text{AFAC}$$

$$C_{ds} \rightarrow C_{ds} \cdot \text{AFAC}$$

$$C_{gs} \rightarrow C_{gs} \cdot \text{AFAC}$$

$$C_{gd} \rightarrow C_{gd} \cdot \text{AFAC}$$

The drain current and nonlinear capacitances are scaled in proportion to AFAC on exit from the subroutines that calculate them. Their parameters are not scaled individually.

Parasitic inductances LG, LS, and LD are not scaled.

Layout

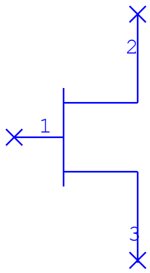
This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

Reference

[1] Curtice, W. R., "A Nonlinear GaAs FET Model for Use in the Design of Output Circuits for Power Amplifiers," IEEE Trans. Microwave Theory Tech., Vol. MTT-33, 1985, p. 1383.

Curtice Quadratic FET Model: CURTICE2

Symbol



Summary

CURTICE2 implements the Curtice-Quadratic GaAsFET model as described in [1] [11–91]. Contact NI AWR Support for additional information: `awr.support@ni.com`.

Parameters

Name	Description	Unit Type	Default
ID	Device ID	Text	CF1
*IDSMOD	Ids Model (not used)	Scaler	1
*VTO	Pinch-off voltage	Voltage	-2.0
*BETA	Transconductance	Scaler	1.0E-4
*LAMBDA	Channel length modulation	Scaler	0
*ALPHA	I/V knee parameter	Scaler	2.0
*TAU	Transit time under the gate	Time	0
*TNOM	Temperature at which model parameters were determined	Temperature	25
*IDSTC	Ids Temperature coefficient	Scaler	0
*VTOTC	VTO temperature coefficient	Scaler	0
*BETATCE	Drain current exponent temperature coefficient	Scaler	0
*RIN	Gate-to-source resistance	Resistance	1E-3
*RF	Gate-to-source forward bias resistance	Resistance	1E6
*GSCAP	0=none, 1=linear, 2=pn junction	Scaler	1
*CGS	Gate-source zero-bias capacitance	Capacitance	0
*CGD	Gate-drain zero-bias capacitance	Capacitance	0
*RGD	Gate-to-drain resistance	Resistance	1E-3
*GDCAP	0=none, 1=linear, 2=pn junction	Scaler	1
*FC	Depletion cap linearization parameter	Scaler	0.5
*RD	Drain resistance	Resistance	1E-3
*RG	Gate resistance	Resistance	1E-3
*RS	Source resistance	Resistance	1E-3

Name	Description	Unit Type	Default
*LD	Drain inductance	Inductance	0
*LG	Gate inductance	Inductance	0
*LS	Source inductance	Inductance	0
*CDS	Drain-source capacitance	Capacitance	0
*CRF	Capacitance that sets the RF Rds break freq	Capacitance	0
*RC	RF drain-source resistance	Resistance	1.0E6
*GSFWD	0=none, 1=linear, 2=diode	Voltage	1
*GSREV	0=none, 1=linear, 2=diode	Voltage	0
*GDFWD	0=none, 1=linear, 2=diode	Voltage	0
*GDREV	0=none, 1=linear, 2=diode	Voltage	1
*R1	Approximate breakdown resistance	Resistance	1E6
*R2	Resistance relating breakdown voltage to channel current	Resistance	1E-3
*VBI	Gate-source capacitance built-in voltage	Voltage	0.85
*VBR	Breakdown voltage	Voltage	1.0E6
*VJR	Breakdown potential	Voltage	1.0
*IS	Gate-conduction diode current parameter	Current	1.0E-14
*IR	Gate reverse saturation current	Current	1.0E-14
*IMAX	(Not implemented)	Current	0
*XTI	Temperature term for saturation current	Scaler	3.0
*EG	Energy gap for temperature effect on IS	Scaler	1.11
*N	Gate-conduction diode ideality factor	Scaler	1.0
*FNC	1/F noise corner	Scaler	1.0
*R	Gate noise param	Capacitance	0.5
*P	Drain noise param	Capacitance	1.0
*C	Noise correlation	Capacitance	0.9
*TAUMDL	(Not implemented)	Time	0
*TEMP	Device Temperature	Temperature	25
AFAC	Gate-width scale factor	Scaler	1.0
NFING	Number of gate fingers scale factor	Scaler	1.0

** indicates a secondary parameter*

Implementation Details

Most parameters are consistent with the model description in [1] [11–91]. There are some parameters, however, that are unique to NI AWR's implementation of the model: The AFAC and NFING parameters are used to specify the device geometry and correspond to the area and number of fingers, respectively.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

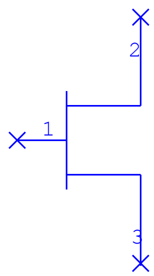
References

[1] W. R. Curtice, A MESFET model for use in the design of GaAs integrated circuits, IEEE Trans Microwave Theory Tech , vol. MTT-28, pp. 448-456, May 1980.

[2] P. Antognetti and G. Massobrio, Semiconductor Device Modeling with SPICE, New York: McGraw-Hill, 1988.

Advanced Curtice Quadratic FET Model with Noise: CURTICE2ADV

Symbol



Summary

CURTICE2ADV implements the Curtice-Quadratic GaAsFET model [1] [11–94] with all of the extensions found in ADS's Advanced Curtice2 Model. Contact NI AWR Support for additional information: `awr.support@ni.com`.

Parameters

Name	Description	Unit Type	Default
ID	Device ID	Text	CF1
*IDSMOD	Ids Model (not used)	Scaler	1
*VTO	Pinch-off voltage	Voltage	-2.0
*BETA	Transconductance	Scaler	1.0E-4
*LAMBDA	Channel length modulation	Scaler	0.0
*ALPHA	I/V knee parameter	Scaler	2.0
*TAU	Transit time	Time	0
*TAUMDL	(Not implemented)	Time	0
*TNOM	Temperature at which model parameters were determined	Temperature	25
*IDSTC	Ids Temperature coefficient	Scaler	0
*UCRIT	Transconductance degradation coefficient	Scaler	0
*VGEXP	Vg-VTO exponent	Scaler	2.0
*GAMDS	Pinch-off coefficient for Vds	Scaler	-0.01
*VTOTC	VTO temperature coefficient	Scaler	0
*BETATCE	Beta exponent temperature coefficient	Scaler	0
*RGS	Gate to source resistance	Resistance	1E-3
*RF	Gate to source forward bias resistance	Resistance	1E6
*GSCAP	0=none, 1=linear, 2=pn junction	Scaler	1
*CGS	Gate-source zero-bias capacitance	Capacitance	0
*CGD	Gate-drain zero-bias capacitance	Capacitance	0
*GDCAP	0=none, 1=linear, 2=pn junction	Scaler	1
*FC	Depletion capacitance linearization parameter	Scaler	0.5

Name	Description	Unit Type	Default
*RGD	Gate to drain resistance	Resistance	1E-3
*RD	Drain resistance	Resistance	1E-3
*RG	Gate resistance	Resistance	1E-3
*RS	Source resistance	Resistance	1E-3
*LD	Drain inductance	Inductance	0
*LG	Gate inductance	Inductance	0
*LS	Source inductance	Inductance	0
*CDS	Drain to source capacitance	Capacitance	0
*RC	RF drain to source resistance	Resistance	1.0E6
*CRF	Capacitance that sets the RF Rds break freq	Capacitance	0
*GSFWD	0=none, 1=linear, 2=diode	Voltage	1
*GSREV	0=none, 1=linear, 2=diode	Voltage	0
*GDFWD	0=none, 1=linear, 2=diode	Voltage	0
*GDREV	0=none, 1=linear, 2=diode	Voltage	1
*R1	Breakdown resistance	Resistance	1E6
*R2	Breakdown voltage to channed current resistance	Resistance	1E-3
*VBI	Gate to source built-in voltage	Voltage	0.85
*VBR	Breakdown voltage	Voltage	1.0E6
*VJR	Breakdown potential	Voltage	1.0
*IS	Diode reverse saturation current	Current	1.0E-14
*IR	Gate reverse saturation current	Current	1.0E-14
*IMAX	(Not implemented)	Current	0.0
*XTI	Temperature term for saturation current	Scaler	3.0
*EG	Energy gap for temperature scaling of IS	Scaler	1.11
*N	Diode ideality factor	Scaler	1.0
*FNC	1/F noise corner	Frequency	0.0
*R	Gate noise param	Scaler	0.5
*P	Drain noise param	Scaler	1.0
*C	Noise correlation	Scaler	0.9
*TEMP	Device Temperature	Temperature	25
AFAC	Gate width scale factor	Scaler	1.0
NFING	Number of gate fingers scale factor	Scaler	1.0
*NFLAG	Noise Model	Scaler	Noise Off

** indicates a secondary parameter*

Implementation Details

This model is based on ADS's publicly available model documentation, and as such, it has a parameter set and behavior consistent with ADS's original model. Most parameters are consistent with ADS's implementation of the model. There are some parameters, however, that are unique to NI AWR's implementation of the model: The NFLAG parameter allows enabling/disabling the noise sources of the model. The AFAC and NFING parameters are used to specify the device geometry and correspond to the area and number of fingers, respectively.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

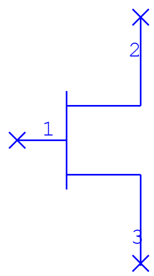
References

[1] W. R. Curtice, A MESFET model for use in the design of GaAs integrated circuits, IEEE Trans Microwave Theory Tech , vol. MTT-28, pp. 448-456, May 1980.

[2] P. Antognetti and G. Massobrio, Semiconductor Device Modeling with SPICE, New York: McGraw-Hill, 1988.

Curtice Cubic Nonlinear FET Model: CURTICE3

Symbol



Summary

The CURTICE3 model implements the Curtice-Cubic GaAsFET model as described in [1] [11–96]. Contact NI AWR Support for additional information: awr.support@ni.com.

Parameters

Name	Description	Unit Type	Default
ID	Device ID	Text	CF1
*BETA	Subthreshold conduction parameter	Scaler	0
*GAMMA	Drain I/V knee parameter	Scaler	2
*VOUTO	Vds at which subthreshold effects begin	Voltage	3
*VTO	Pinch-off voltage	Voltage	-1.0
*A0	Gate I/V polynomial coefficient	Scaler	0.065
*A1	Gate I/V polynomial coefficient	Scaler	0.104
*A2	Gate I/V polynomial coefficient	Scaler	0.013
*A3	Gate I/V polynomial coefficient	Scaler	-0.026
*TAU	Gate-drain time delay	Time	0
*R1	Approximate breakdown resistance	Resistance	1E6
*R2	Resistance relating breakdown voltage to channel current	Resistance	0
*VBO	Gate-channel breakdown voltage (pos.)	Voltage	1.0E6
*VBI	Gate-source capacitance built-in voltage	Voltage	1.0
*RF	Gate-diode forward-bias resistance	Resistance	1E6
*IS	Gate-conduction diode current parameter	Current	1.0E-14
*N	Gate-conduction diode ideality factor	Scaler	1.0
*RDS	RF drain-source resistance	Resistance	1.0E6
*CRF	Capacitance that sets the RF Rds break freq	Capacitance	0
*RD	Drain resistance	Resistance	1E-3
*RG	Gate resistance	Resistance	1E-3
*RS	Source resistance	Resistance	1E-3

Name	Description	Unit Type	Default
*RIN	Intrinsic resistance	Resistance	1E-3
*CGSO	Gate-source capacitance at 0V	Capacitance	0
*CGDO	Gate-drain capacitance at 0V	Capacitance	0
*FC	Gate-capacitance linearization parameter	Scaler	0.5
*CDS	Drain-source capacitance	Capacitance	0
*CGS	Fixed gate-source capacitance	Capacitance	0
*CGD	Fixed gate-drain capacitance	Capacitance	0
*TNOM	Temperature	Temperature	26.85
*LAMBDA	DC drain-source resistance parameter	Scaler	0
*RGD	Gate-drain resistance	Resistance	1E-3
*RDSO	Constant drain-source resistance	Resistance	1.0E6
*LG	Gate inductance	Inductance	0
*LS	Source inductance	Inductance	0
*LD	Drain inductance	Inductance	0
*P	Noise par: P	Scaler	2
*Tg	Noise par: gate noise temp	Temperature	16.85
*KF	Flicker noise coefficient	Scaler	0
*AF	Flicker noise exponent	Scaler	1
*FFE	Flicker noise freq. exponent	Scaler	1
*NFLAG	Noise model	Scaler	AWR1
AFAC	Gate-width scale factor (for COMPAT=AWR)	Scaler	1
NFING	Number of gate fingers scale factor	Scaler	1
*FIXUP	disable/enable standard fixup	Scaler	enable
COMPAT	Compatibility selector	Scaler	AWR

** indicates a secondary parameter*

Implementation Details

Most parameters are consistent with the model description in [1] [11–96]. There area some parameters, however, that are unique to NI AWR's implementation of the model. The NFLAG parameter allows enabling/disabling the noise sources of the model. You can access ADS's extensions to the model by choosing "ADS" as the COMPAT setting.

References

[1] W. R. Curtice and M. Ettenberg, A nonlinear GaAsFET model for use in the design of output circuits for power amplifiers, IEEE Trans of Microwave Theory Tech , vol. MTT-33, pp. 1383-1394, Dec. 1985.

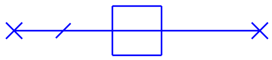
[2] P. Antognetti and G. Massobrio, Semiconductor Device Modeling with SPICE, New York: McGraw-Hill, 1988.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

DC Short and RF Open: DCSHORT

Symbol



Implementation Details

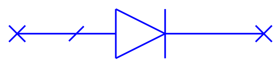
Short at DC, open otherwise.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Simple Diode Model: DIODE

Symbol



Summary

NI AWR's DIODE element is a simple diode model. It is conceived as a successor for both DIODE1 and DIODE2, while also incorporating the behavior in the Agilent ADS Pin Diode model.

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	SD1
lo	Reverse saturation current	Current	1e-14A
*MULT	Scaling factor		1
Nu	Bottom ideality factor		1
T	Temperature	Temperature	25
*IMAX	Maximum device current (for improving convergence)	Current	1000A
*R	Series resistance	Resistance	0.001 ohm
*C	Zero-voltage bottom junction capacitance	Capacitance	0
*CFLAG	Noise model. Changing this value changes parameters for this model.		Linear Capacitance
*VJ	Bottom built-in voltage	Voltage	0.8
*M	Bottom junction grading coefficient		0.5
*FC	Bottom depletion capacitance linearization parameter		0.5
*NFLAG	Noise model. Changing this value changes parameters for this model.		Noise off
*KF	Flicker noise coefficient		0.0
*AF	Flicker noise exponent		1.0
*FFE	Flicker noise frequency exponent		1.0
*SSFLAG	Small signal model flag. Changing this value changes parameters for this model.		Auto differentiation
*Vi	I-Region forward bias voltage drop	Voltage	0.0
*Un	Electron mobility cm^2/(VxS)		900
*Wi	I-region width	Length	1.0e-6
*Rr	I-region reverse bias resistance	Resistance	1.0e-6
*Cmin	P-I-N punch-through capacitance	Capacitance	0.0
*Tau	Ambipolar lifetime within I region	Time	1.0e-7

Operating Points

The letter "d" is used to identify the only NL branch of this model. Consequently, "Vd" identifies the voltage across this branch and "Id" its current.

Parameter	Description
pwr (Power)	Dissipated power
rs (Resistance)	Parasitic resistance
cd (Capacitance)	Diode capacitance
gd (Conductance)	Diode conductance

Implementation Details

DIODE in its simplest mode of operation behaves as DIODE1. However, unlike DIODE1 or DIODE2, DIODE supports noise modeling, enabled by setting the NFLAG parameter to **on**. DIODE includes thermal, Flicker, and shot noise sources. The capacitance of DIODE is linear (the same as in DIODE2). This is the default behavior. It is possible, however, to enable the depletion capacitance model by setting the CFLAG parameter to **depletion**. Depletion capacitance parameters VJ, M, and FC display accordingly. While in the depletion capacitance mode of operation, you can choose between two distinctly different small signal models which correspond to the **auto-differentiation** and **specialized** settings of the SSFLAG parameter. In **auto-differentiation**, the small signal model is obtained by straight-forward differentiation of the large signal expressions. In **specialized**, the Agilent ADS Pin Diode small signal model is employed. The Vi, Un, Wi, Rr, Cmin, and Tau parameters apply to the latter small signal model and only show when the SSFLAG parameter enables it.

Layout

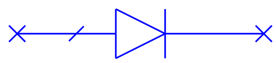
This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

References

P. Antognetti and G. Massobrio, Semiconductor Device Modeling with SPICE, New York: McGraw-Hill, 1988.

Simple Diode Model: DIODE1

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Name used to identify the diode	Text	D1
Nu	Ideality factor		1.2
T	Temperature	Temperature	22 DegC
Io	Saturation current	Current	1e-06 mA

Implementation Details

Implements an ideal resistive diode junction having the I-V characteristic

$$I(V) = I_0(e^{\frac{qV}{\mathrm{Nu} \cdot K \cdot T}} - 1)$$

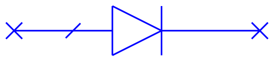
where q is electron charge and K is Boltzmann's constant.

Layout

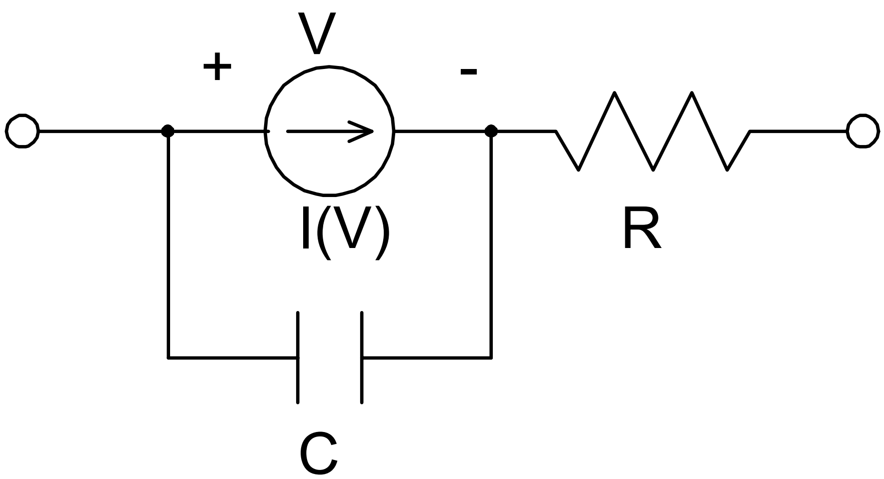
This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Physical Diode Model: DIODE2

Symbol



Equivalent Circuit



Parameters

Name	Description	Unit Type	Default
ID	Name used to identify the diode	Text	D1
Nu	Ideality factor		1.2
T	Temperature	Temperature	22 DegC
Io	Saturation current	Current	1e-06 mA
R	Series resistance	Resistance	1 ohm
C	Junction capacitance	Capacitance	1 pF

Implementation Details

Implements an ideal diode having the I/V characteristic

$$I(V) = I_0(e^{\frac{qV}{\mathrm{Nu} \cdot K \cdot T}} - 1)$$

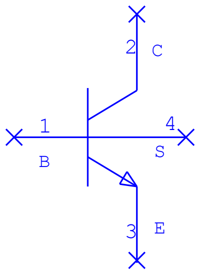
where q is electron charge and K is Boltzmann's constant. Series resistance R and parallel junction capacitance C are linear.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

(Obsolete) EEBJT2 Bipolar Junction Transistor Model : EEBJT2

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Device ID		EB1
*ISF	Saturation current	mA	9.53e-12
*ISR	Saturation current	mA	1.01e-11
*NF	Fwd ideality factor		1
*VAF	Fwd Early voltage	V	1000000000
*IKF	Fwd current knee	mA	1e15
*ISE	BC leakage current param	mA	0
*NE	BE leakage ideality factor		1.5
*NR	Rev ideality factor		1
*VAR	Rev Early voltage	V	1000000000
*IKR	Rev current knee	mA	1e15
*ISC	BC leakage current param	mA	0
*NC	BC leakage ideality factor		2
*IBIF	BE base current param	mA	1.48e-13
*NBF	BE base current ideality factor		1.06
*IBIR	BC base current param	mA	6.71e-13
*NBR	BC base current ideality factor		1.04
*RB	Base resistance	ohm	0.01
*RE	Emitter resistance	ohm	0.01
*RC	Collector resistance	ohm	0.01
*CJE	CJ0 for BE junction	pF	0
*VJE	BE built-in potential	V	0.75
*MJE	BE grading coefficient		0.33
*TF	Fwd transit time	ns	0
*XTF	Coefficient for bias dependence of TF		0

Name	Description	Unit Type	Default
*VTF	Voltage for VBC dependence of TF	V	1000000000
*ITF	High-current parameter for TF	mA	0
*PTF	Excess phase param; must be degrees		0
*CJC	CJ0 for BC junction	pF	0
*VJC	BC built-in potential	V	0.75
*MJC	BC grade coefficient		0.33
*XCJC	Fraction of CBC to internal node		1
*TR	Rev transit time	ns	0
*CJS	CJ0 for substrate capacitance	pF	0
*VJS	Built-in potential for substrate cap	V	0.75
*MJS	Substrate cap grading coefficient		0.33
*XTB	Thermal scaling parameter		0
*EG	Thermal scaling parameter		1.11
*XTI	Thermal scaling parameter		3
*KF	Flicker noise coefficient		0
*AF	Flicker noise exponent		1
*FFE	Flicker noise frequency exponent		1
*FC	Coefficient for forward-bias depletion		0.5
*TYPE	(Not implemented)		0
*NK	(Not implemented)		0
*TEXT	Temperature at which params were extracted	DegC	25
*TNOM	Temperature	DegC	25
*KB	Burst noise coefficient		0
*AB	Burst noise exponent		1
*FB	Burst noise cutoff frequency		1
*ISS	Collector-substrate current param	mA	0
*NS	Collector-substrate ideality factor		1
*TS	Collector-substrate diode storage time	ns	0
*LB	Base inductance	nH	1e-6
*LE	Emitter inductance	nH	0
*LC	Collector inductance	nH	0
*AFAC	Area scale factor		1
*NFLAG	1=noise is on; 0=noise is turned off		0

** indicates a secondary parameter*

Implementation Details

The EEBJT2 model evolved from the Gummel-Poon and other earlier models. As a result EEBJT2 still maintains lots of similarities with its predecessors and can be conveniently described by simply stating its differences respect to element GBJT. With the exception of the equations listed below elements EEBJT2 and GBJT are identical. As the these equations show, EEBJT2 has completely dropped the basic assumption that the non-leakage base-emitter current can be related to the collector-emitter current by a simple constant. Also, the base resistance is now linear and there is no splitting of the base-collector junction capacitance. These two facts translate in a simpler model with generally better convergence properties than SGPM.

$$I_{be1} = IBIF \cdot \left(\exp\left(\frac{V_{be}}{NBF \cdot V_T}\right) - 1.0\right) \quad (1)$$

$$I_{bc1} = IBIR \cdot \left(\exp\left(\frac{V_{bc}}{NBR \cdot V_T}\right) - 1.0\right) \quad (2)$$

$$I_{cf} = ISF \cdot \left(\exp\left(\frac{V_{be}}{NF \cdot V_T}\right) - 1.0\right) \quad (3)$$

$$I_{cr} = ISR \cdot \left(\exp\left(\frac{V_{bc}}{NR \cdot V_T}\right) - 1.0\right) \quad (4)$$

$$R_b = r_B \quad (5)$$

$$XCJC = 1 \quad (6)$$

Layout

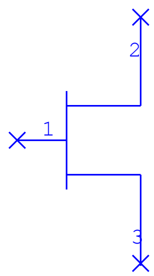
This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

References

This model was developed under research performed at NI AWR Corporation. The full set of details of the implementation are considered proprietary in nature.

EEHEMT: EEHEMT

Symbol



Summary

EEHEMT implements ADS's EEsof Scalable Nonlinear HEMT model (EE_HEMT1). Contact NI AWR Support for additional information: `awr.support@ni.com`.

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	EEHEMT1
*Vto	Threshold voltage, zero bias	Voltage	-1.435
*Gamma	Threshold constant	Scaler	0.03492
*Vgo	Gate-source voltage for maximum gm	Voltage	-0.8118
*Vdelt	Not used	Voltage	15.05e-09
*Vch	Gate-source voltage where Gamma has no effect	Voltage	0.1
*Gmmax	Peak gm	Conductance	0.08942
*Vdso	Output voltage where Vo has no effect	Voltage	2
*Vsat	Drain-source voltage for current saturation	Voltage	1.128
*Kapa	Output conductance	Scaler	133.2e-18
*Peff	Power for self-heating, channel to backside	Power	3.841
*Vtso	Onset voltage, subthreshold	Voltage	-10
*Is	Reverse saturation current, gate diode junction	Current	2.816e-12
*N	Gate diode ideality factor	Scaler	1.568
*Ris	Channel resistance, source end	Resistance	0.587
*Rid	Channel resistance, drain end	Resistance	0.001
*Tau	Transit time delay, gate	Time	1.018e-12
*Cdso	Capacitance, drain-source	Capacitance	34.37e-15
*Rdb	Resistance of dispersion source	Resistance	2000
*Cbs	Capacitance, trapping-state	Capacitance	60e-12
*VtoAC	AC threshold voltage, zero bias	Voltage	-1.413
*GammaAC	AC threshold constant	Scaler	0.03512

Name	Description	Unit Type	Default
*VdeltAC	Not used	Voltage	1
*GmmaxAC	AC peak gm	Conductance	0.08815
*KapaAC	AC output conductance	Scaler	133.2e-12
*PeffAC	Power for self-heating, channel to backside, AC	Power	63.69
*VtsoAC	Onset voltage, subthreshold, AC	Voltage	-10
*Gdbm	Idb conductance at Vo = Vdsm	Conductance	0.00142
*Kdb	Controls Vds dependence of Idb	Scaler	0.08882
*Vdsm	Voltage where Idb = 0	Voltage	1
*C11o	Maximum capacitance for Vds=Vdso and Vdso>Deltds	Capacitance	174.1e-15
*C11th	Minimum capacitance for Vds=Vdso	Capacitance	75.36e-15
*Vinfl	Inflection voltage in C11-Vgs characteristic	Voltage	-1.233
*Deltgs	Transition voltage, C11th to C11o	Voltage	0.4244
*Deltds	Transition voltage, linear to saturation region	Voltage	0.5673
*Lambda	C11-Vds characteristic slope	Scaler	0.05150
*C12sat	Transcapacitance for Vgs=Vinfl and Vds>Deltds	Capacitance	18.41e-15
*Cgdsat	Capacitance, gate drain for Vds>Deltds	Capacitance	20.39e-15
*Kbk	Breakdown current coefficient	Scaler	0.03
*Vbr	Drain-gate breakdown voltage	Voltage	15
*Nbr	Breakdown current exponent	Scaler	2
*Idsoc	Open channel drain-source current	Current	0.193
*Rd	Contact resistance, drain	Resistance	2.398
*Rs	Contact resistance, source	Resistance	2.015
*Rg	Metallization resistance, gate	Resistance	3.074
*UgwRef	Reference unit gate width	Length	75e-06
*NfgRef	Reference number of gate fingers	Scaler	2
*Vco	Voltage, gm compression for Vds=Vdso	Voltage	-0.4962
*Vba	Voltage, gm compression tail-off	Voltage	5
*Vbc	Transition voltage, gm roll-off to tail-off	Voltage	0.4131
*Mu	Adds Vds dependence to gm compression onset	Scaler	1.332e-18
*Deltgm	Slope of gm compression characteristic	Scaler	0.09378
*DeltgmAC	AC slope of gm compression characteristic	Scaler	0.07629
*Alpha	Transition voltage, gm saturation to compression	Voltage	0.07436
*Kmod	Model number	Scaler	1
*Kver	Version number	Scaler	1005
*Rgtc	Linear temperature coefficient for Rg	Scaler	0
*Rdtc	Linear temperature coefficient for Rd	Scaler	0
*Rstc	Linear temperature coefficient for Rs	Scaler	0

Name	Description	Unit Type	Default
*Vtotc	Linear temperature coefficient for Vtso	Scaler	0
*Gmmaxtc	Linear temperature coefficient for Gmmax	Scaler	0
*Xti	Saturation current temperature exponent	Scaler	3.0
*Vinfltc	Linear temperature coefficient for Vinfl	Scaler	0
*Gammatc	Linear temperature coefficient for Gamma	Scaler	0
*VtoACtc	Linear temperature coefficient for VtoAC	Scaler	0
*GmmaxACtc	Linear temperature coefficient for GmmaxAC	Scaler	0
*GammaACtc	Linear temperature coefficient for GammaAC	Scaler	0
*Tnom	Parameter measurement temperature	Temperature	25
*Temp	Temperature of environment	Temperature	25
*Ld	Drain inductance (not scaled)	Inductance	30.29e-12
*Ls	Source inductance (not scaled)	Inductance	8.163e-12
*Lg	Gate inductance (not scaled)	Inductance	45.85e-12
UgwNew	New unit gate width	Length	75e-06
NfgNew	New number of gate fingers	Scaler	2
*Lshunt	Shunt inductance parallel to Idb (not scaled)	Inductance	1e-3
*NOISE	Noise model off or on	Scaler	Noise off
*COMPAT	AWR model or ADS model	Scaler	AWR

** indicates a secondary parameter*

Implementation Details

This model is based on ADS's publicly available model documentation, and as such, it has a parameter set and behavior consistent with ADS's original model. The model has been enhanced to support fully symmetrical behavior, which cannot be found in ADS's original model. You can access this behavior by choosing "AWR" as the COMPAT setting.

Layout

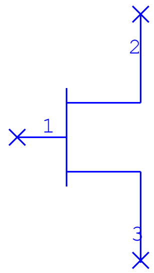
This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

References

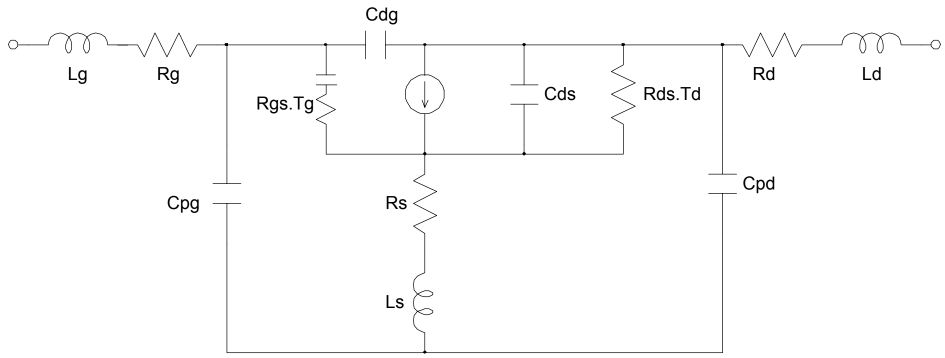
[1] P. Antognetti and G. Massobrio, Semiconductor Device Modeling with SPICE, New York: McGraw-Hill, 1988.

Field Effect Transistor Noise Model (Pospieszalski's): FETN

Symbol



Equivalent Circuit



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	FN1
TA	Ambient temperature of the chip	Temperature	24 DegC
TG1	Constant temperature of RGS	Temperature	-98 DegC
TG2	Frequency-dependent temperature of RGS	Temperature	1727 DegC
FN	Frequency-dependent multiplier	Frequency	1e-09 GHz
TD	Temperature of RDS	Temperature	1727 DegC
*GM	FET transconductance	Conductance	0.1 S
*CGS	Gate to source capacitance	Capacitance	0.25 pF
*RGS	Gate to source resistance	Resistance	4 ohm
*RDS	Drain to source resistance	Resistance	300 ohm
*TAU	FET transit time	Time	0 ns
*CDS	Drain to source capacitance	Capacitance	0.05 pF
*CDG	Drain to gate capacitance	Capacitance	0.05 pF

Name	Description	Unit Type	Default
*RS	Source resistance	Resistance	2 ohm
*RD	Drain resistance	Resistance	1 ohm
*RG	Gate resistance	Resistance	1 ohm
*LS	Source inductance	Inductance	0.01 nH
*LD	Drain inductance	Inductance	0.1 nH
*LG	Gate inductance	Inductance	0.2 nH
*CPG	Gate pad capacitance	Capacitance	0.01 pF
*CPD	Drain pad capacitance	Capacitance	0.01 pF

** indicates a secondary parameter*

Layout

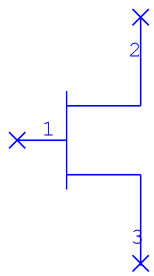
This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

References

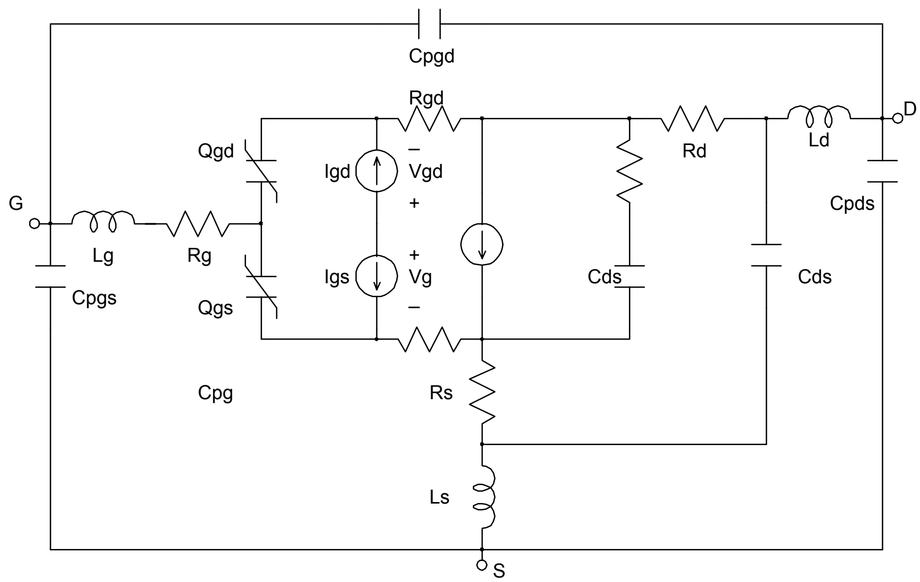
[1] Marian W. Pospieszalski, "Modeling of Noise Parameters of MESFET's and MODFET's and Their Frequency and Temperature Dependence", IEEE Transactions on Microwave Theory and Techniques, Vol. 37, No. 9, pp. 1340 - 1350, September 1989.

Fujii FET Model: FUJII

Symbol



Equivalent Circuit



Parameters

Name	Description	Unit Type	Default
ID	Device ID	Text	FF1
*P1	Gate I/V polynomial coefficient		1
*P2	Gate I/V polynomial coefficient		0
*P3	Gate I/V polynomial coefficient		0
*P4	Gate I/V polynomial coefficient		0
*P5	Gate I/V polynomial coefficient		0
*P6	Gate I/V polynomial coefficient		0
*VPK0	Gate voltage at peak Gm	Voltage	0 V

Name	Description	Unit Type	Default
*IPK	Current at peak Gm	Current	50 mA
*DEL1	Drain I/V polynomial coefficient		0
*DEL2	Drain I/V polynomial coefficient		0
*DEL3	Drain I/V polynomial coefficient		0
*DEL4	Drain I/V polynomial coefficient		0
*DEL5	Drain I/V polynomial coefficient		0
*DEL6	Drain I/V polynomial coefficient		0
*GAMMA	Drain I/V knee parameter		0
*ALP0	Drain I/V knee parameter		1
*ALP1	Drain I/V knee parameter		0
*LAMBDA0	Drain-source resistance parameter		0
*LAMBDA1	Drain-source resistance parameter		0
*TAU	Gate-drain time delay	Time	0 ns
*CGS0	Gate capacitance parameter (per mm of width)	Capacitance	0.25 pF
*CF	Gate capacitance term	Capacitance	0.15 pF
*GCG	Gate capacitance term	Capacitance	0.2 pF
*DCG	Gate capacitance range term	Voltage	2.7 V
*DKGS	Gate capacitance knee term	Voltage	3 V
*IG0F	Gate diode forward current parameter	Current	1e-17 mA
*ALPGF	Gate diode forward exponential parameter		40
*PB1	Gate breakdown polynomial coefficient		1
*PB2	Gate breakdown polynomial coefficient		0
*PB3	Gate breakdown polynomial coefficient		0
*PB4	Gate breakdown polynomial coefficient		0
*PB5	Gate breakdown polynomial coefficient		0
*PB6	Gate breakdown polynomial coefficient		0
*VGSPK	Gate voltage peak for breakdown	Voltage	-1 V
*IG0R	Gate diode breakdown current parameter	Current	0.001 mA
*ALPGR	Gate diode breakdown exponential parameter		40
*BVGDPK	Drain-to-gate breakdown voltage	Voltage	1e+06 V
*RG	Gate resistance	Resistance	1 ohm
*RS	Source resistance	Resistance	1 ohm
*RGS	Intrinsic resistance	Resistance	1 ohm
*RD	Drain resistance	Resistance	1 ohm
*RGD	Gate-drain resistance	Resistance	1 ohm
*RDSF	RF drain-source resistance	Resistance	300 ohm
*CDSF	Capacitance that determines Rds break frequency	Capacitance	0 pF

Name	Description	Unit Type	Default
*CDS	Drain-source capacitance	Capacitance	0 pF
*CPDS	Drain-source pad capacitance (not scaled)	Capacitance	0 pF
*CPGS	Gate-source pad capacitance (not scaled)	Capacitance	0 pF
*CPGD	Gate-drain pad capacitance (not scaled)	Capacitance	0 pF
*LS	Source inductance	Inductance	0.001 nH
*LG	Gate inductance	Inductance	0.001 nH
*LD	Drain inductance	Inductance	0.001 nH
W	Gate width, mm (affects only Cgs & Cgd)		1
AFAC	Gate width scale factor		1
NFING	Number of fingers scale factor		1

** indicates a secondary parameter*

Implementation Details

This models offers a capacitance function whose parameters are relatively easy to extract, an improved breakdown model, and an improved I/V model. It is loosely based on the Angelov (Chalmers) model. This model is designed to reproduce the I/V derivatives correctly; thus, with proper parameter extraction, it should be useful for intermodulation analysis.

Drain current

The drain current is given by

$$I_d(V_g,\ V_d) = \mathrm{AFAC}(F_1(V_g,\ V_d) + F_2(V_d))$$

where

$$F_1(V_g,\ V_d) = \mathrm{IPK}(1 + \tanh(\Psi))\tanh(\alpha V_d)(1 + \lambda V_d)$$

$$F_2(V_d) = D_1 V_d + D_2 V_d^2 + \ldots$$

$$\Psi = P1(V_g - V_{\mathrm{pk}}) + P2(V_g - V\mathrm{pk})^2 + P3(V_g - V_{\mathrm{pk}})^3 + \ldots$$

$$\alpha = \mathrm{ALPH0} + \mathrm{ALPH1} \cdot V_g$$

$$\lambda = \mathrm{LAMBDA0} + \mathrm{LAMBDA1} \cdot V_g$$

$$V_{\mathrm{pk}} = \mathrm{VPK0} + \mathrm{GAMMA} \cdot V_d$$

The above expressions are defined for $V_d > 0$. When $V_d < 0$,

$$I_d(V_g,\ V_d) = \mathrm{AFAC}(-(F_1(V_{\mathrm{gd'}} - V_d) + F_2(-V_d))$$

Breakdown

The current sources Igs and Igd account for both gate rectification and breakdown. The expressions for both sources are identical. In forward conduction ($V_g > 0$ or $V_{gd} > 0$), the current is given by the diode junction equation:

$$I_{gs} = \mathrm{IG0} \cdot (\exp(\mathrm{ALPHAG} \cdot V_g) - 1)$$

$$I_{gd} = \mathrm{IG0} \cdot (\exp(\mathrm{ALPHAG} \cdot V_{gd}) - 1)$$

The breakdown current ($-V_{gd} > V_{bd}$ or $-V_{gs} > V_{bg}$) is given by

$$I_{gs} = -\mathrm{IG0}R \cdot (\exp(\mathrm{ALPHGR} \cdot (-V_{gs} - V_{bg})) - 1)$$

$$I_{gd} = -\mathrm{IG0}R \cdot (\exp(\mathrm{ALPHGR} \cdot (-V_{gd} - V_{bd})) - 1)$$

where V_{bg}, V_{bd} are positive quantities and

$$V_{bg} = \mathrm{BVGDPK} \cdot (1 + \Psi_d)$$

$$V_{bd} = \mathrm{BVGDPK} \cdot (1 + \Psi_g)$$

$$\Psi_d = \mathrm{PG1}(V_g - \mathrm{VGSPK}) + \mathrm{PG2}(V_g - \mathrm{VGSPK})^2 + \mathrm{PG3}(V_g - \mathrm{VGSPK})^3 + \ldots$$

$$\Psi_g = \mathrm{PG1}(V_{gd} - \mathrm{VGSPK}) + \mathrm{PG2}(V_{gd} - \mathrm{VGSPK})^2 + \mathrm{PG3}(V_{gd} - \mathrm{VGSPK})^3 + \ldots$$

Capacitances

The gate-to-source and gate-to-drain charge functions are symmetrical. They are given by the following expressions:

$$Q_{gs}(V_g, V_d) = W \cdot \mathrm{CGS0}[V_g + \frac{C_{fs}}{A_g}\log(\cosh(A_g(V_g - \mathrm{DCG} + \mathrm{DCG} \cdot \tanh(D_{ks}V_d))))] - W \cdot \mathrm{CGS0}[\frac{C_{fs}}{A_g}\log(\cosh(A_g(-\mathrm{DCG} + \mathrm{DCG} \cdot \tanh(D_{ks}V_d))))]$$

$$Q_{gd}(V_g, V_d) = W \cdot \mathrm{CGS0}[V_{gd} + \frac{C_{fs}}{A_g}\log(\cosh(A_g(V_{gd} - \mathrm{DCG} + \mathrm{DCG} \cdot \tanh(D_{ks}V_d))))] - W \cdot \mathrm{CGS0}[\frac{C_{fs}}{A_g}\log(\cosh(A_g(-\mathrm{DCG} + \mathrm{DCG} \cdot \tanh(D_{ks}V_d))))]$$

$$A_g = \frac{\mathrm{GCG}}{\mathrm{CGS0} \cdot C_{fs}}$$

$$C_{fs} = \frac{\mathrm{CGS0} - \mathrm{CF}}{\mathrm{CGS0}}$$

Because of the symmetry, one function can be used for both charges.

Scaling

Resistances, channel current, and linear capacitances are scaled in the standard manner:

Resistances are scaled as 1/AFAC;

Gate resistance, RG, is scaled as AFAC/NFING^2;

Capacitances are scaled in proportion to AFAC.

Scaling of the nonlinear channel capacitance, C_g, is different from other MW Office models.

CG Scaling

W is the device width in mm, and CGS0 is the capacitance per mm. This capacitance is proportional to W; it is not scaled by AFAC. If CGS0 is provided as the total capacitance of a particular device, it is correct simply to enter that capacitance for CGS0 and set W = 1 for the standard device. Then, to scale the device, set W = AFAC. W affects no other model parameters.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

References

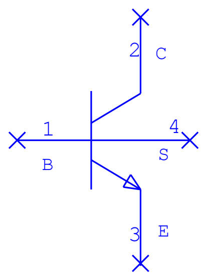
[1] K. Fujii, Y. Hara, F. M. Ghannouchi, T. Yakabe, and H. Yabe, "A Nonlinear GaAs FET Model Suitable for Active and Passive MM-Wave Applications," IEICE Trans., vol. E83-A, no. 2, p. 228, Feb., 2000.

[2] K. Fujii, K. Ogawa, and Y. Takano, "The Consideration of Voltage-Controlled Charge Sources Controlled by Two Voltage Sources for a GaAs MESFET Large-Signal Model," IEIEC Trans., vol. J80-C-I, no. 9, p. 414, Sept. 1997.

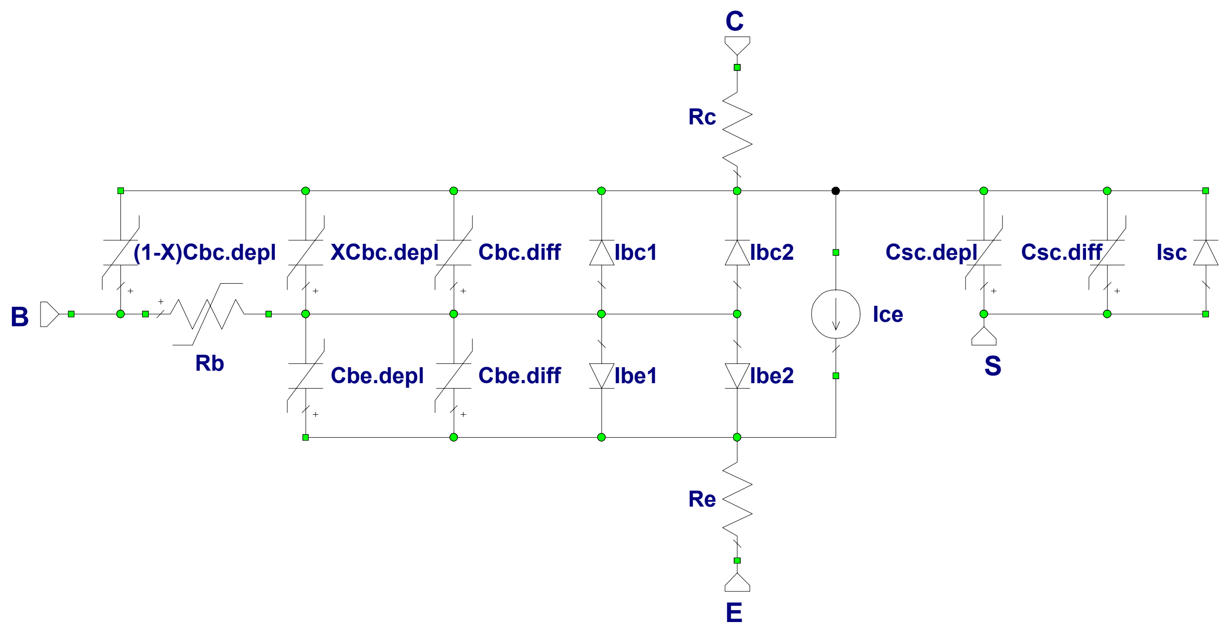
[3] K. Fujii, Y. Hara, T. Yakabe and H. Yabe, "Accurate Modeling for Drain Breakdown Current of GaAs MESFETs," IEEE Trans. Microwave Theory Tech., vol. 47, no. 4, p. 516, April, 1999.

Gummel-Poon NPN BJT: GBJT

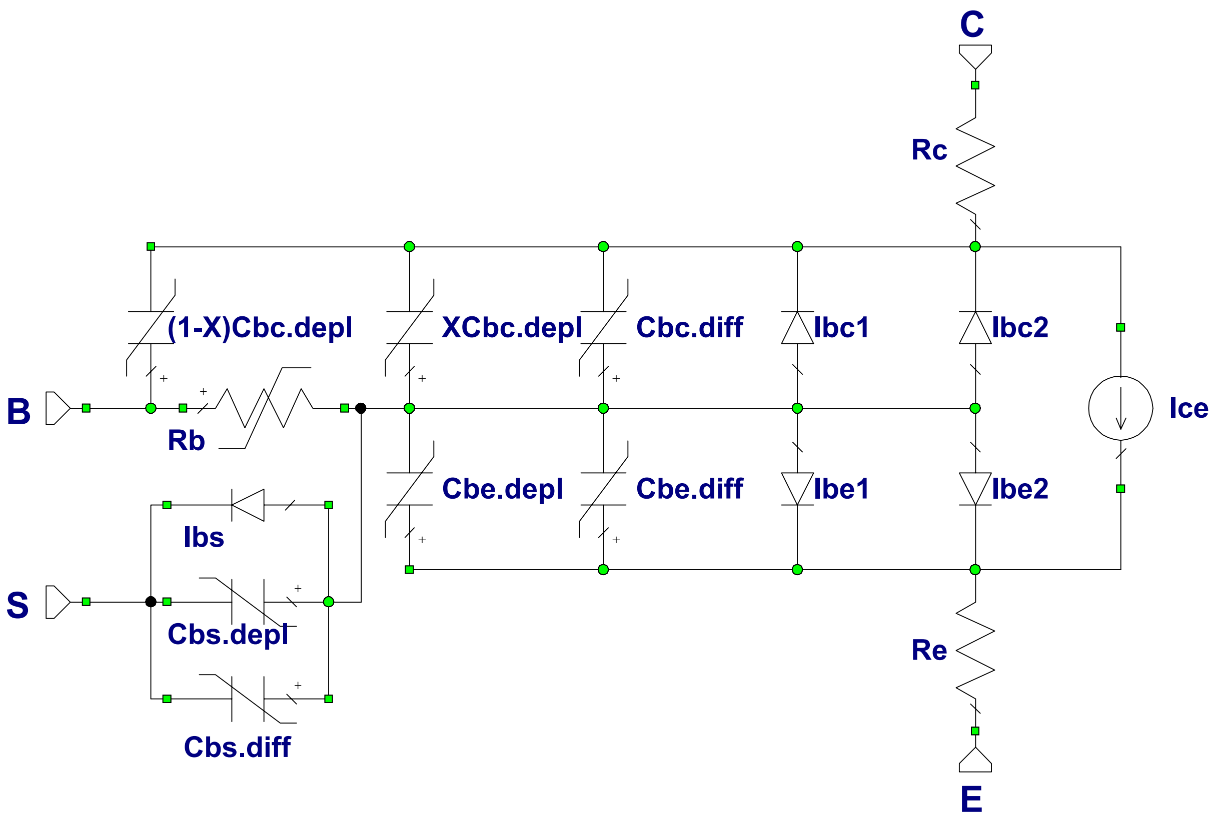
Symbol



Equivalent Circuit (Vertical)



Equivalent Circuit (Lateral)



Parameters

Name	Description	Unit Type	Default
ID	Device ID	Text	GP1
*IS	Saturation current	Current	1e-13 mA
*IBE	Rev BE saturation current	Current	
*IBC	Rev BC saturation current	Current	
*BF	Fwd current gain		100
*NF	Fwd ideality factor		1.0
*VAF	Fwd early voltage	Voltage	1e+09 V
*IKF	Fwd current knee	Current	1e+15 mA
*ISE	BE leakage current param	Current	0 mA
*NE	BE leakage ideality factor		1.5
*BR	Rev current gain		1.0
*NR	Rev ideality factor		1.0
*VAR	Rev early voltage	Voltage	1e+09 V

Name	Description	Unit Type	Default
*IKR	Rev current knee	Current	1e+15 mA
*ISC	BC leakage current param	Current	0 mA
*NC	BC leakage ideality factor		2.0
*RB	Base resistance	Resistance	0.01 ohm
*IRB	Current where RB falls halfway to min	Current	0 mA
*RBM	Minimum base resistance	Resistance	0.01 ohm
*RE	Emitter resistance	Resistance	0.01 ohm
*RC	Collector resistance	Resistance	0.01 ohm
*CJE	CJ0 for BE junction	Capacitance	0 pF
*VJE	BE built-in potential	Voltage	0.75 V
*MJE	BE grading coefficient		0.33
*TF	Fwd transit time	Time	0 ns
*XTF	Coef for bias dependence of TF		0
*VTF	Voltage for VBC dependence of TF	Voltage	1e+09 V
*ITF	High-current parameter for TF	Current	0 mA
*PTF	Excess phase param; must be degrees		0
*CJC	CJ0 for BC junction	Capacitance	0 pF
*VJC	BC built-in potential	Voltage	0.75V
*MJC	BC grading coefficient		0.33
*XCJC	Fraction of CBC to internal node		1.0
*TR	Rev transit time	Time	0 ns
*CJS	CJ0 for substrate capacitance	Capacitance	0 pF
*VJS	Built-in potential for substrate cap	Voltage	0.75V
*MJS	Substrate cap grading coeff		0.33
*XTB	Temperature scaling term for Beta		0
*EG	Energy gap at T=TNOM		1.11
*XTI	Temperature scaling term		3.0
*KF	Flicker noise coefficient		0
*AF	Flicker noise exponent		1.0
*FFE	Flicker noise frequency exponent		1.0
*FC	Coef for fwd-bias depletion		0.5
*NKF	Exponent for high-current Beta roll-off		0.5
*TNOM	Nominal temperature (parameter extraction)	Temperature	27 DegC
*TEMP	Simulation temperature	Temperature	27 DegC
*KB	Burst noise coefficient		0
*AB	Burst noise exponent		1.0
*FB	Burst noise cutoff frequency		1.0

Name	Description	Unit Type	Default
*ISS	Collector-substrate current param	Current	0 mA
*NS	Collector-substrate ideality factor		1.0
*TS	Collector-substrate diode storage time	Time	0 ns
*LB	Base inductance (Not supported anymore)	Inductance	1e-06 nH
*LE	Emitter inductance (Not supported anymore)	Inductance	0 nH
*LC	Collector inductance (Not supported anymore)	Inductance	0 nH
*CBCP	External BC constant capacitance	Capacitance	0 pF
*CBEP	External BE constant capacitance	Capacitance	0 pF
*CCSP	External CS constant capacitance	Capacitance	0 pF
*AREA	Emitter area scale factor		
*AREAB	Base area scale factor		
*NFLAG	Noise model: on, or, off.		on
*M	Multiplicity Parameter		1
*TLEV	I/V temperature model (0-3)		0
*TLEVC	Capacitance temperature model (0-3)		0
*CTC	Temperature coefficient for CJC		0
*CTE	Temperature coefficient for CJE		0
*CTS	Temperature coefficient for CJS		0
*TIKF1	First-order temperature coefficient for IKF		0
*TIKF2	Second-order temperature coefficient for IKF		0
*TIKR1	First-order temperature coefficient for IKR		0
*TIKR2	Second-order temperature coefficient for IKR		0
*TIRB1	First-order temperature coefficient for IRB		0
*TIRB2	Second-order temperature coefficient for IRB		0
*TIS1	First-order temperature coefficient for IS		0
*TIS2	Second-order temperature coefficient for IS		0
*TISC1	First-order temperature coefficient for ISC		0
*TISC2	Second-order temperature coefficient for ISC		0
*TISE1	First-order temperature coefficient for ISE		0
*TISE2	Second-order temperature coefficient for ISE		0
*TISS1	First-order temperature coefficient for ISS		0
*TISS2	Second-order temperature coefficient for ISS		0
*TRB1	First-order temperature coefficient for RB		0
*TRB2	Second-order temperature coefficient for RB		0
*TRC1	First-order temperature coefficient for RC		0
*TRC2	Second-order temperature coefficient for RC		0
*TRE1	First-order temperature coefficient for RE		0

Name	Description	Unit Type	Default
*TRE2	Second-order temperature coefficient for RE		0
*TRS1	First-order temperature coefficient for RS		0
*TRS2	Second-order temperature coefficient for RS		0
*TRM1	First-order temperature coefficient for RM		0
*TRM2	Second-order temperature coefficient for RM		0
*TVJE	Temperature coefficient for VJE		0
*TVJC	Temperature coefficient for VJC		0
*TVJS	Temperature coefficient for VJS		0
*TBF1	First-order temperature coefficient for BF		0
*TBF2	Second-order temperature coefficient for BF		0
*TBR1	First-order temperature coefficient for BR		0
*TBR2	Second-order temperature coefficient for BR		0
*TVAF1	First-order temperature coefficient for TVAF		0
*TVAF2	Second-order temperature coefficient for TVAF		0
*TVAR1	First-order temperature coefficient for TVAR		0
*TVAR2	Second-order temperature coefficient for TVAR		0
*TITF1	First-order temperature coefficient for ITF		0
*TITF2	Second-order temperature coefficient for ITF		0
*TTF1	First-order temperature coefficient for TF		0
*TTF2	Second-order temperature coefficient for TF		0
*TTR1	First-order temperature coefficient for TR		0
*TTR2	Second-order temperature coefficient for TR		0
*TNF1	First-order temperature coefficient for NF		0
*TNF2	Second-order temperature coefficient for NF		0
*TNR1	First-order temperature coefficient for NR		0
*TNR2	Second-order temperature coefficient for NR		0
*TNE1	First-order temperature coefficient for NE		0
*TNE2	Second-order temperature coefficient for NE		0
*TNC1	First-order temperature coefficient for NC		0
*TNC2	Second-order temperature coefficient for NC		0
*TNS1	First-order temperature coefficient for NS		0
*TNS2	Second-order temperature coefficient for NS		0
*TMJE1	First-order temperature coefficient for MJE		0
*TMJE2	Second-order temperature coefficient for MJE		
*TMJC1	First-order temperature coefficient for MJC		0
*TMJC2	Second-order temperature coefficient for MJC		0
*TMJS1	First-order temperature coefficient for MJS		0

Name	Description	Unit Type	Default
*TMJS2	Second-order temperature coefficient for MJS		0
*GAP1	First bandgap correction factor		0
*GAP2	Second bandgap correction factor		0
*DCAP	Capacitance model selector.		1
*RBMOD	Base resistance model selector: SPICE2, or, linear.		SPICE2
*SUBS	Transistor geometry: vertical, or, lateral.		vertical
*UPDATE	Charge model equation selector: SPICE2, or, alternate.		SPICE2
*IMAX	Maximum device current	Current	1000000 mA
*COMPAT	Model compatibility selector: HSPICE, AWR, or, Spectre.		HSPICE
*SSFLAG	Small signal model flag: autodifferentiation, or, specialized.		autodifferentiation
*AREAC	Collector area scale factor		

** indicates a secondary parameter*

Operating Points

The following letter pairs have been used to identify the NL branches: be, bc, ce, bs, sc, and bb. Here be, bc and ce correspond to the intrinsic base-emitter, base-collector and collector-emitter branches, respectively. bs and sc correspond to the branches formed by connecting the intrinsic base or collector nodes to the substrate terminal. bb denotes the branch where the nonliner base resistor is located. The positive node of this branch is connected to the linear base resistor. These names are used to identify branch related operating point information, i.e., branch voltages, currents, etc.

Parameter	Description
pwr (W)	Power dissipation
betadc (A/A)	Ratio of DC collector current to DC base current.
betaac (A/A)	Small-signal common-emitter current gain.
gm (S)	Common-emitter transconductance.
rpi (ohm)	Common-emitter input resistance.
ro (ohm)	Common-emitter output resistance.
rx (ohm)	Parasitic base resistance.
cpi (F)	Common-emitter input capacitance.
cmu (F)	Common-base output capacitance.
cmux (F)	External common-base output capacitance.
csub (F)	Substrate capacitance.
ft**(Hz)	Unity small-signal current-gain frequency.
rc (ohm)	Parasitic collector resistance.
re (ohm)	Parasitic emitter resistance.
rb (ohm)	Parasitic base resistance. Same as rx.

Implementation Details

GBJT started as an implementation of the G-device level 1 of SPICE2. It has progressively been enhanced to the point that it includes now all HSPICE extensions to the original bipolar junction model, together with some other enhancements by AWR. Accordingly, this model is mapped into HSPICE as a G-device with parameter LEVEL set to 1.

Parameters, NFLAG, SSFLAG, RBMOD, and COMPAT, are unique to the Microwave Office implementation of the bipolar junction transistor. NFLAG's behavior is quite obvious and deserves no further comment. **COMPAT**, the compatibility selection flag, can take three different values: HSPICE, AWR and SPECTRE. The HSPICE and Spectre values are used to emulate the behavior of the G-device level 1 and bjt model in HSPICE and Spectre, respectively. **RBMOD** allows the user to replace the complex SPICE2 non-linear base resistor model by a linear one. The **SSFLAG** has effect in linear simulations only. It enables the user to toggle between the complete small signal model, obtained by automatic differentiation of the large signal expressions, and the simpler "specialized" model, as prescribed by SPICE2, where some of the derivatives are neglected or have been modified.

Except for (1) complete collector-to-substrate diode parameters, CJS, VJS, MJS, ISS, NS, and TS, and (2) the ability to replace the non-linear base resistor, by a linear one, the equations for the Gummel-Poon model are implemented exactly as in HSPICE. Terminal inductances LB, LC, and LE are not part of the model anymore. They have to be added externally. The substrate terminal can be left open-circuited if desired.

The static model equations are presented below.

$$I_{be1} = \frac{IS}{\beta_f} \cdot \left(\exp\left(\frac{V_{be}}{NF \cdot V_T}\right) - 1.0\right) \tag{7}$$

$$I_{be2} = ISE \cdot \left(\exp\left(\frac{V_{be}}{NE \cdot V_T}\right) - 1.0\right) \tag{8}$$

$$I_{bc1} = \frac{IS}{\beta_r} \cdot \left(\exp\left(\frac{V_{bc}}{NR \cdot V_T}\right) - 1.0\right) \tag{9}$$

$$I_{bc2} = ISC \cdot \left(\exp\left(\frac{V_{bc}}{NC \cdot V_T}\right) - 1.0\right) \tag{10}$$

$$I_{cf} = IS \cdot \left(\exp\left(\frac{V_{be}}{NF \cdot V_T}\right) - 1.0\right) \tag{11}$$

$$I_{cr} = IS \cdot \left(\exp\left(\frac{V_{bc}}{NR \cdot V_T}\right) - 1.0\right) \tag{12}$$

$$I_{ce} = \frac{(I_{cf} - I_{cr})}{q_b} \tag{13}$$

$$q_b = \frac{q_1}{2} \cdot \left[1 + (1 + 4q_2)^{n_k}\right] \tag{14}$$

$$q_1 = \frac{1}{(1 - \frac{V_{bc}}{V_{af}} - \frac{V_{be}}{V_{ar}})} \tag{15}$$

$$q_2 = \frac{I_{be1}}{I_{kf}} + \frac{I_{bc1}}{I_{kr}} \tag{16}$$

$$R_b = r_{BM} + 3 \cdot (r_B - r_{BM}) \cdot \left(\frac{\tan z - z}{z\tan^2 z}\right) \quad (17)$$

$$z = \frac{-1 + \sqrt{1 + 144 I_B / \pi^2 I_{rB}}}{(24/\pi)\sqrt{I_B / I_{rB}}} \quad (18)$$

The location of these current sources and resistance can be easlily identified in either diagram above. The capacitances in the same diagram follow the capacitance equations presented for elements PNCAP and PNDCAP, with C, Vj, Cd, Ij, CJ0, VJ, FC, M, and, TF, replaced with Cbe.depl, Vbe, Cbe.diff, Ibe1, CJE, VJE, FC, MJE, and, Tf for the base-emitter and Cbc.depl, Vbc, Cbc.diff, Ibc1, CJC, VJC, FC, MJC, and, TR base-collector junction capacitances, respectively. The base-collector depletion capacitance Cbc.depl is divided into extrinsic-base and intrinsic-base fractions with parameter XCJC.

The effective forward transit time Tf of the base-emitter diffusion capacitance is a funtion of parameters TF, XTF, ITF, and, VTF and of the current Icf. The reverese transit tim Tr of the base-collector capacitance is specified by parameter TR and has no dependency on other parameters. An additional delay is introduced to model the distributd phenomena if parameter PTF is specified.

A complete treatment of the model can be found at [1] [11–124].

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Reference

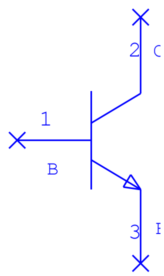
[1] P. Antognetti and G. Massobrio, Semiconductor Device Modeling with SPICE, New York: McGraw-Hill, 1988.

[2] H. K. Gummel and H. C. Poon, "An Integral Charge Control Model of Bipolar Transistors," Bell Sys. Tech. J., vol. 49, p. 827, May/June, 1970.

[3] H. C. de Graaf and F. M. Klaasen, Compact Transistor Modelling for Circuit Design, New York: Springer-Verlag, 1990.

Gummel-Poon 3-Terminal NPN BJT: GBJT3

Symbol



Equivalent Circuit

This element is identical to the four-terminal Gummel-Poon NPN BJT model, GBJT, except that its substrate terminal is connected internally to the emitter terminal. Parameters associated to the substrate have no effect.

Parameters

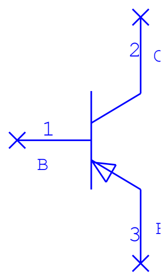
Parameters are identical to those of GBJT.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Gummel-Poon 3-Terminal PNP BJT: GBJT3_PNP

Symbol



Equivalent Circuit

Same as GBJT3, except that all source and diode currents are reversed.

Parameters

Parameters are identical to those of GBJT. Parameters associated to the substrate have no effect.

Implementation Details

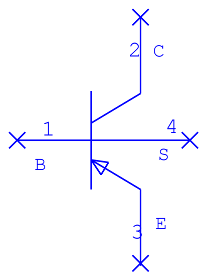
Except for the PNP structure, this model is equivalent to GBJT3, the three-terminal Gummel-Poon NPN BJT model.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Gummel-Poon PNP BJT: GBJT_PNP

Symbol



Equivalent Circuit

Same as GBJT, except that all diode and source currents are reversed.

Parameters

Parameters are identical to those of GBJT.

Implementation Details

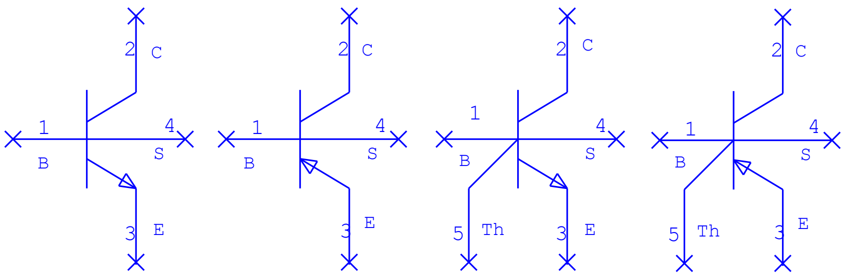
Except for the PNP structure, this model is equivalent to GBJT, the Gummel-Poon NPN BJT model.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

UCSD HBT Model: HBT

Symbol



Summary

NI AWR's HBT (Hetero-junction Bipolar Transistor) element implements the HBT model, developed by UCSD as part of the ARPA High Speed Circuit Design Program. The UCSD HBT model, like the Berkeley BSIM3 model, was conceived as an "industry standard" SPICE model. This model may be described as a revised version of the standard Gummel-Poon model; where some portions of the model have been enhanced, or, modified to support features not supported by the original Gummel-Poon model; while the remaining has been kept unchanged. Support for breakdown and self heating are among the most important new features introduced by this model. Complete documentation for this model is available on the UCSD web site[1] [11–135]. NI AWR's HBT model also supports Agilent's extensions to the original UCSD HBT model.

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	UH1
*NFLAG	Noise Model		Noise On
*MULT	Number of devices in parallel		1
*AREA	Device area factor (Does not scale parasitic inductors or capacitors)		1
*TNOM	Reference (extraction) temperature	Temperature	26.85 C
*TEMP	Ambient temperature	Temperature	26.85 C
*TBP	Base-plate temperature	Temperature	
*SELFT	Self-Heating flag		off
*EXT_FLAG	ADS-Extensions flag		Disabled
*RE	Emitter resistance	Resistance	
*RCI	Intrinsic collector resistance	Resistance	
*RCX	Extrinsic collector resistance	Resistance	
*RBI	Intrinsic base resistance	Resistance	
*RBX	Extrinsic base resistance	Resistance	
*IS	Collector-Emitter current: Forward collector saturation current	Current	

Name	Description	Unit Type	Default
*NF	Collector-Emitter current: Forward collector current ideality factor		
*ISR	Collector-Emitter current: Reverse emitter saturation current	Current	
*NR	Collector-Emitter current: Reverse emitter current ideality factor		
*ISH	Base-Emitter current: Ideal base-emitter current	Current	
*NH	Base-Emitter current: Ideal base-emitter current ideality factor		
*ISE	Base-Emitter current: Non-ideal base-emitter current	Current	
*NE	Base-Emitter current: Non-ideal base-emitter current ideality factor		
*ISRH	Base-Collector current: Ideal base-collector saturation current	Current	
*NRH	Base-Collector current: Ideal base-collector current ideality factor		
*ISC	Base-Collector current: Non-ideal base-collector saturation current	Current	
*NC	Base-Collector current: Non-ideal base-collector current ideality factor		
*ABEL	Base-Emitter current: Portion of base-emitter current allocated to extrinsic region		
*VAF	Forward Early voltage	Voltage	
*VAR	Reverse Early voltage	Voltage	
*ISA	Base-emitter heterojunction saturation current (BE barrier effects)	Current	
*NA	Base-emitter heterojunction ideality factor		
*ISB	Base-collector heterojunction current (BC barrier effects)	Current	
*NB	Base-collector heterojunction ideality factor		
*IKDC1	Soft knee effect: Transition width in Ic of q3 function	Current	
*IKDC2INV	Soft knee effect: slope of q3 function (1/Amp)		
*IKDC3	Soft knee effect: Critical current at which soft knee effect occurs at Vcb=VJC	Current	
*VKDCINV	Soft knee effect: Transition width of Vcb (in inverse voltage) (1/Volt)		
*NKDC	Soft knee effect: maximum value of q3		
*GKDC	Soft knee effect: Exponent of q3 factor in base current		
*IK	High current roll-off in Beta	Current	
*CJE	Base-emitter capacitance: zero-bias capacitance	Capacitance	
*VJE	Base-emitter capacitance: built- in voltage	Voltage	
*MJE	Base-emitter capacitance: grading factor (0.1<MJE)		
*CEMAX	Base-emitter capacitance: maximum value in forward bias	Capacitance	

Name	Description	Unit Type	Default
*VPTE	Base-emitter capacitance: punchthrough voltage	Voltage	
*MJER	Base-emitter capacitance: grading factor beyond punchthrough		
*ABEX	Base-emitter capacitance: ratio between extrinsic and total base-emitter regions		
*CJC	Base-collector capacitance: zero-bias capacitance	Capacitance	
*VJC	Base-collector capacitance: built-in voltage	Voltage	
*MJC	Base-collector capacitance: grading factor (0.1<MJC)		
*CCMAX	Base-collector capacitance: maximum value in forward bias	Capacitance	
*VPTC	Base-collector capacitance: punchthrough voltage	Voltage	
*MJCR	Base-collector capacitance: grading factor beyond punchthrough		
*ABCX	Ratio between extrinsic and total base-collector regions		
*TFB	Base transit time: delay through the base	Time	
*FEXTB	Base transit time: fraction of base transit time charge allocated to the base-collector junction		
*TFC0	Collector transit time: low current transit time	Time	
*TCMIN	Collector transit time: high current transit time	Time	
*ITC	Collector transit time: midpoint in collector current between TFC0 and TCMIN	Current	0.001A
*ITC2	Collector transit time: transition width in collector current between TFC0 and TCMIN	Current	0.001A
*VTC0IVN	Collector transit time: rate of change of TFC0 with Vcb (in inverse saturation)		
*VTR0	Collector transit time: transition width in Vcb to VMX0 (models velocity saturation)	Voltage	
*VMX0	Collector transit time: maximum Vcb for TFC0 (models velocity saturation)	Voltage	
*VTCMININV	Collector transit time: rate of change of TCMIN with Vbc (in inverse voltage) (1/Volt)		
*VTRMIN	Collector transit time: transition width in Vcb to VMXMIN (models velocity saturation)	Voltage	
*VMXMIN	Collector transit time: maximum Vcb for TCMIN (models velocity saturation)	Voltage	
*VTCINV	Collector transit time: rate of change of ITC with Vcb (in inverse voltage) (1/Volt)		
*VTC2INV	Collector transit time: rate of change of ITC2 with Vcb (in inverse voltage) (1/Volt)		
*FEXTC	Collector transit time: fraction of the collector transit time charge allocated to the base-collector junction		
*TKRK	Kirk effect: Kirk effect delay time	Time	

Name	Description	Unit Type	Default
*IKRK	Kirk effect: critical current for Kirk effect at Vbc=0	Current	
*IKRKTR	Kirk effect: transition width to prevent Ikirk=0 (use sparingly)	Current	
*VKRK	Kirk effect: rate of change of IKRK with Vcb	Voltage	
*VKRK2INV	Kirk effect: rate of change of TKRK with Vcb (in inverse voltage) (1/Volt)		
*GKRK	Kirk effect: exponent of kirk effect delay		
*VKTR	Kirk effect: transition width in Vcb to VKMX	Voltage	
*VKMX	Kirk effect: Maximum Vcb for TKRK	Voltage	
*FEXKE	Kirk effect: fraction of the Kirk effect charge allocated ti the base-collector junction		
*TR	Reverse transit time	Time	
*CPCE	Parasitic/Fringing collector-emitter capacitance	Capacitance	
*CPBE	Parasitic/Fringing base-emitter capacitance	Capacitance	
*CPBC	Parasitic/Fringing base-collector capacitance	Capacitance	
*LPC	Parasitic collector inductance	Inductance	
*LPB	Parasitic base inductance	Inductance	
*LPE	Parasitic emitter inductance	Inductance	
*XRB	Temperature exponent for RBI and RBX		
*XRC	Temperature exponent for RCI and RCX		
*XRE	Temperature exponent for RE		
*TVJE	Rate of change in temperature of VJE (Volt/K)		
*TVPE	Rate of change in temperature of VPTE (Volt/K)		
*TVJC	Rate of change in temperature of VJC (Volt/K)		
*TVPC	Rate of change in temperature of VPTC (Volt/K)		
*EGE	Effective emitter bandgap parameter	Voltage	
*TNF	Rate of change in temperature of NF		
*XTIS	Temperature exponent for IS		
*XTIH	Temperature exponent for ISH		
*XTIE	Temperature exponent for ISE		
*EGC	Effective collector bandgap parameter	Voltage	
*TNR	Rate of change in temperature of NR		
*XTIR	Temperature exponent for ISR		
*XTIC	Temperature exponent for ISC		
*XTIRH	Temperature exponent for ISRH		
*XTIK3	Temperature exponent for IKDC3		
*EAA	Temperature dependence of ISA	Voltage	
*EAB	Temperature dependence of ISB	Voltage	

Name	Description	Unit Type	Default
*XTFB	Temperature exponent for TFB		
*XTCMIN	Temperature exponent for TCMIN		
*XTFC0	Temperature exponent for TFC0		
*XTITC	Temperature exponent for ITC		
*XTITC2	Temperature exponent for ITC2		
*XTTKRK	Temperature exponent for TKRK		
*XTIKRK	Temperature exponent for IKRK		
*XTVKRK	Temperature exponent for VKRK		
*RTH1	Thermal resistance #1 (Kelvin/Watt)		
*CTH1	Thermal capacitance #1 (Sec-Amp/Volt)		
*XTH1	Temperature exponent for Rth1		
*RTH2	Thermal resistance #2 (Kelvin/Watt)		
*CTH2	Thermal capacitance #2 (Sec-Amp/Volt)		
*XTH2	Temperature exponent for Rth2		
*KF	Flicker noise coefficient		
*AF	Flicker noise exponent		
*FFE	Flicker noise frequency exponent		
*KB	Burst noise coefficient		
*AB	Burst noise exponent		
*FB	Burst noise corner frequency	Frequency	
*IMAX	Explosion current	Current	
*BKDN	Breakdown current flag		Disabled
*BF	Forward ideal current gain		
*BR	Reverse ideal current gain		
*ISEX	Saturation current for emitter leakage diode	Current	
*NEX	Ideality factor for emitter leakage diode		
*ISCX	Saturation current for extrinsic BC junction current	Current	
*NCX	Ideality factor for extrinsic BC junction current		
*FA	Factor to specify avalanche voltage		
*BVC	C-B breakdown voltage (BVcb0); positive	Voltage	
*NBC	Exponent for BC multiplication factors vs. voltage		
*ICS	Saturation value for collector-substrate current	Current	
*NCS	Ideality factor for collector-substrate current		
*REX	Extrinsic emitter leakage diode series resistance	Resistance	
*CEMIN	Min value for intrinsic BC capacitance	Capacitance	
*FCE	Factor for BE capacitance approximation near Vbi		
*CCMIN	Min valuer of intrinsic BC capacitance	Capacitance	

Name	Description	Unit Type	Default
*FC	Factor for BC capacitance approximation near Vbi		
*CJCX	Extrinsic BC capacitance at zero voltage	Capacitance	
*VJCX	Extrinsic BC capacitance Vbi	Voltage	
*MJCX	Extrinsic base-collector capacitance: grading factor (0.1<MJCX)		
*CXMIN	Minimum extrinsic BC capacitance	Capacitance	
*XCJC	Factor for partitioning BC capacitance		
*CJS	Substrate depletion capacitance at zero voltage	Capacitance	
*VJS	Built-in potential for substrate capacitance	Voltage	
*MJS	Substrate capacitance: grading factor (0.1<MJS)		
*TBEXS	Excess BE transit time	Time	
*TBCXS	Excess BC transit time	Time	
*ICRIT0	Critical current for junction capacitance	Current	
*VTC	Characteristic voltage for TFC	Voltage	
*TRX	Reverse storage time for extrinsic BC diode	Time	
*FEX	Excess phase factor		
*RTH	Thermal resistance	Resistance	
*CTH	Thermal capacitance	Capacitance	
*XTI	Exponent for IS temperature dependence		
*XTB	Exponent for beta temperature dependence		
*TNE	Coefficient for NE temperature dependence		
*TNC	Coefficient for NC temperature dependence		
*TNEX	Coefficient for NEX temperature dependence		
*EG	Activation energy for IS temperature dependence	Voltage	
*EAE	Activation energy for ISE temperature dependence	Voltage	
*EAC	Activation energy for ISC temperature dependence	Voltage	
*EAX	Activation energy for ISEX temperature dependence	Voltage	
*XREX	Coefficient for REX temperature dependence		
*TVJCX	Coefficient for VJCX temperature dependence		
*TVJS	Coefficient for VJS temperature dependence		
*XTTF	Coefficient for TF temperature dependence		
*XRT	Coefficient for RTH temperature dependence		
*DTMAX	Maximum temperature rise above heatsink in Celsius degrees		100
*COMPAT	Model compatibility selector		AWR

** indicates a secondary parameter*

Implementation Details

Since the model supports both the UCSD original model and Agilent's extensions, the model has a pretty extensive parameter set. Some parameters are shared by both modes of operation, and, some are exclusive to a particular one. Agilent's extensions are enabled by choosing the appropriate setting of the EXT_FLAG parameter. The SELFT parameter controls whether self-heating simulation is supported or not. The self-heating terminal is shown or hidden, accordingly. The extraction and device temperatures are controlled using the TNOM and TEMP parameters, respectively. The temperature parameter TBP (Temperature of the Base Plate) enables partial self-heating simulation. This type of operation is not enabled when the model supports the Agilent extensions. Finally, the TYPE parameter controls whether the device is a NPN, or, PNP.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

Recommendations for Use

CAUTIONARY NOTE. Whenever self-heating effects do not need to be considered, set the SELFT parameter to "off". This removes unused self-heating associated branches from the model, resulting in faster execution time. The DTMAX parameter limits the value of self-heating induced temperature changes during simulation. You should keep this value as small as possible, provided it does not affect the simulation results.

Operating Points

The following letter pairs have been used to identify the NL branches: bei, bci, ci, e, bx, cx, bex1, bex2sc, bex, ce, bi, bcxx, and bcx. These are used to identify branch related operating point information, i.e., branch voltages, currents, etc.

Parameter	Description
pwr (Power)	Dissipated power
Ibk (Current)	Breakdown current
dice_dvbe (Conductance)	Intrinsic $\frac{\partial \text{Ice}}{\partial \text{Vbe}}$
dice_dvbc (Conductance)	Intrinsic $\frac{\partial \text{Ice}}{\partial \text{Vbc}}$
dibe_dvbe (Conductance)	Intrinsic $\frac{\partial \text{Ibe}}{\partial \text{Vbe}}$
dibe_dvbc (Conductance)	Intrinsic $\frac{\partial \text{Ibe}}{\partial \text{Vbc}}$
dqbe_dvbe (Capacitance)	Intrinsic $\frac{\partial \text{Qbe}}{\partial \text{Vbe}}$
dqbe_dvbc (Capacitance)	Intrinsic $\frac{\partial \text{Qbe}}{\partial \text{Vbc}}$
dqbc_dvbe (Capacitance)	Intrinsic $\frac{\partial \text{Qbc}}{\partial \text{Vbe}}$

Parameter	Description
dqbc_dvbc (Capacitance)	Intrinsic $\frac{\partial Qbc}{\partial Vbc}$
cbcx (Capacitance)	XB-C junction capacitance
cbcxx (Capacitance)	EXTB-C junction capacitance
csc	Capacitance
temp (Temperature)	Temperature
Qbb	Intrinsic base terminal charge
Qcc	Intrinsic collector terminal charge

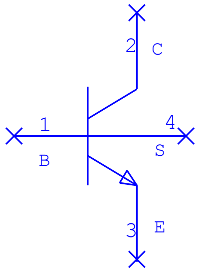
.

References

[1] http://hbt.ucsd.edu/

(Obsolete) UCSD HBT Model: HBT_UCSD

Symbol

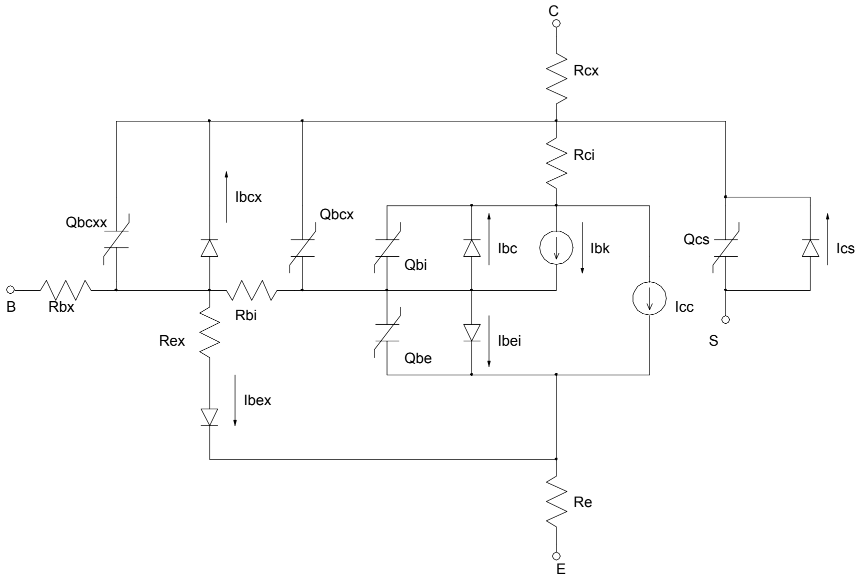


Summary

This element is OBSOLETE and is replaced by the UCSD HBT Model (HBT) element with the EXT_FLAG parameter set to "Disabled". The UCSD HBT model, like the Berkeley BSIM3 model, is designed as an "industry standard" SPICE model. Its basic concepts are the same as the Gummel-Poon BJT model; capacitances are a combination of diffusion and depletion components, for example, but the expressions are more complex. This model includes breakdown and self-heating.

Documentation for this model is available on the UCSD web site. See [1] [11–141] for further information.

Equivalent Circuit



Parameters

Name	Description	Unit Type	Default
ID	Element ID		UH#1
SELFT	Flag; 0=no self-heating (T=TOP); 1=self-heating is included		0

Name	Description	Unit Type	Default
BKDN	Flag; 0=no breakdown; 1=breakdown is included		0
TBP	Base-plate temperature	Temperature	26.85 C
TNOM	Temperature at which model parameters were extracted	Temperature	26.85 C
TOP	Best-guess operating temperature	Temperature	26.85 C
IS	Saturation current	Current	10^{-22}
NF	Forward ideality factor		1.0
NR	Reverse ideality factor		1.0
ISA	Collector EB barrier limiting current	Current	10^{13}
NA	Collector EB barrier ideality factor		2.0
ISB	Collector BC barrier limiting current	Current	10^{13}
NB	Collector BC barrier ideality factor		2.0
VAF	Forward early voltage	Voltage	1000.0
VAR	Reverse early voltage	Voltage	1000.0
IK	Forward current knee	Current	10^{13}
BF	Forward ideal current gain		10^{4}
BR	Reverse ideal current gain		10^{4}
ISE	Saturation value for nonideal forward base current	Current	10^{-27}
NE	Ideality factor for nonideal forward base current		2.0
ISEX	Saturation current for emitter leakage diode	Current	10^{-27}
NEX	Ideality factor for emitter leakage diode		2.0
ISC	Saturation value for intrinsic BC current	Current	10^{-27}
NC	Ideality factor for intrinsic BC current		2.0
ISCX	Saturation current for extrinsic BC junction current	Current	10^{-27}
NCX	Ideality factor for extrinsic BC junction current		2.0
FA	Factor to specify avalanche voltage		0.9
BVC	C-B breakdown voltage (BVcb0); positive	Voltage	1000.0
NBC	Exponent for BC multiplication factor vs. voltage		8.0
ICS	Saturation value for collector-substrate current	Current	10^{-27}
NCS	Ideality factor for collector-substrate current		2.0
RE	Emitter resistance	Resistance	0.001
REX	Extrinsic emitter leakage diode series resistance	Resistance	0.001
RBX	Extrinsic base resistance	Resistance	0.001
RBI	Intrinsic base resistance	Resistance	0.001
RCX	Extrinsic collector resistance	Resistance	0.001
RCI	Intrinsic collector resistance	Resistance	0.001
CJE	BE depletion capacitance at zero voltage	Capacitance	2e-8
VJE	BE built-in potential	Voltage	1.6

Name	Description	Unit Type	Default
MJE	BE capacitance exponent (0.1<MJE<0.9)		0.33
CEMIN	Min value of intrinsic BC capacitance	Capacitance	1e-8
FCE	Factor for BE capacitance approximation near Vbi		0.8
CJC	BC depletion capacitance at zero voltage	Capacitance	2e-8
VJC	BC built-in potential	Voltage	1.4
MJC	BC capacitance exponent (0.1<MJC<0.9)		0.33
CCMIN	Min value of intrinsic BC capacitance	Capacitance	1e-8
FC	Factor for BC capacitance approximation near Vbi		0.8
CJCX	Extrinsic BC capacitance at zero voltage	Capacitance	2e-8
VJCX	Extrinsic BC capacitance Vbi	Voltage	1.4
MJCX	Extrinsic BC capacitance exponent (0.1<MJCX<0.9)		0.33
CXMIN	Minimum extrinsic BC capacitance	Capacitance	1e-8
XCJC	Factor for partitioning BC capacitance		1.0
CJS	Substrate depletion capacitance at zero voltage	Capacitance	0.0
VJS	Built-in potential for substrate capacitance	Voltage	1.4
MJS	Substrate capacitance exponent (0.1<MJS<0.9)		0.5
TFB	Base transit time	Time	0.0
TBEXS	Excess BE transit time	Time	0.0
TBCXS	Excess BC transit time	Time	0.0
TFC0	Collector forward transit time	Time	0.0
ICRIT0	Critical current for junction capacitance	Current	1000000
ITC	Characteristic current for TFC	Current	0.0
ITC2	Characteristic current for TFC	Current	0.0
VTC	Characteristic voltage for TFC	Current	1000.0
TKRK	Forward transit time for Kirk effect	Time	0.0
VKRK	Characteristic voltage for Kirk effect	Voltage	1000.0
IKRK	Characteristic current for Kirk effect	Current	1000000
TR	Reverse storage time for intrinsic BC diode	Time	0.0
TRX	Reverse storage time for extrinsic BC diode	Time	0.0
FEX	Excess phase factor		0.0
RTH	Thermal resistance	Resistance	10^{-8}
CTH	Thermal capacitance	Capacitance	1e6
KFN	BE flicker noise constant		0.0
AFN	BE flicker noise exponent for current		1.0
BFN	BE flicker noise exponent for frequency		1.0
XTI	Exponent for IS temperature dependence		2.0
XTB	Exponent for beta temperature dependence		2.0

Name	Description	Unit Type	Default
TNE	Coefficient for NE temperature dependence		0.0
TNC	Coefficient for NC temperature dependence		0.0
TNEX	Coefficient for NEX temperature dependence		0.0
EG	Activation energy for IS temperature dependence		1.5
EAA	Activation energy for ISA temperature dependence		0.0
EAB	Activation energy for ISB temperature dependence		0.0
EAE	Activation energy for ISE temperature dependence		0.0
EAC	Activation energy for ISC temperature dependence		0.0
EAX	Activation energy for ISEX temperature dependence		0.0
XRE	Coefficient for RE temperature dependence		0.0
XREX	Coefficient for REX temperature dependence		0.0
XRB	Coefficient for RB temperature dependence		0.0
XRC	Coefficient for RC temperature dependence		0.0
TVJE	Coefficient for VJE temperature dependence		0.0
TVJCX	Coefficient for VJCX temperature dependence		0.0
TVJC	Coefficient for VJC temperature dependence		0.0
TVJS	Coefficient for VJS temperature dependence		0.0
XTITC	Coefficient for ITC temperature dependence		0.0
XTITC2	Coefficient for ITC2 temperature dependence		0.0
XTTF	Coefficient for TF temperature dependence		0.0
XTTKRK	Coefficient for TKRK temperature dependence		0.0
XTVKRK	Coefficient for VKRK temperature dependence		0.0
XTIKRK	Coefficient for IKRK temperature dependence		0.0
XRT	Coefficient for RTH temperature dependence		0.0
DTMAX	Maximum temperature rise above heatsink in Celsius degrees	Temperature	1000.0 C
AFAC	Area scale factor		1.0

Parameter Details

DTMAX. A temperature limit used to avert numerical difficulties in the self-heating model. The Microwave Office implementation also includes a minimum temperature fixed at 100 K (-173 C). You cannot change the minimum temperature.

TOP and TBP. The estimated device temperature is not listed in the tables in [1] [11–141] because, in SPICE, the global temperature parameter is used for this quantity. In Microwave Office, however, TOP is included in the model's parameter list, and therefore can be different from the temperature of other parts of the circuit. TOP is your best estimate of device temperature in operation. It is used for parameters that depend on temperature only weakly, or have only a minor effect in the model. Including self-heating for such parameters increases the computational overhead of the model, for very little improvement in accuracy. TOP is essentially the same as the temperature parameter in conventional SPICE models, which employ thermal scaling but not self-heating.

TBP is the baseplate temperature, which you specify. [1] [11–141] indicates that you should set this value by applying a voltage to a fifth, thermal node in the HBT. Microwave Office does not use this approach, because treating it as a model parameter is simpler and less error-prone.

CJx and CxMIN. Defaults have been changed to $2\cdot10^{-20}$ F for the parameters CJE, CJC, and CJCX; and to $1*10^{-20}$ F for the parameters CEMIN, CCMIN, and CXMIN. The default values of zero, listed in [1] [11–141], cause divide-by-zero errors.

HB_UCSD contains terms of the form (CJx/CxMIN)(1/MJx), where MJx is typically in the range of 0.33 to 0.50. With incorrect values for these parameters, it is easy to "crash" such equations. These errors are very difficult to trap without restricting the use of reasonable parameter values, so you should be aware of this problem and choose parameter values accordingly.

RTH and CTH. HBT_UCSD provides for a nonlinear thermal resistance, RTH, which is a function of temperature. This formulation makes the temperature increase, normally the dependent quantity, a function of itself. This could lead to numerical problems, poor convergence, instability, or other unusual behavior. Although Microwave Office attempts to make its models conform to the defining documentation (so SPICE and Microwave Office calculations give the same results), in this case, Microwave Office makes RTH a function of TOP, instead of instantaneous device temperature.

BF and BR. The default values of BF and BR (10^4) specified in [1] [11–141] present little problem in most practical cases, but if the base is biased with a current source (as might happen in the initial stages of a design), and BF/BR use the default values, bad convergence and strange results are possible. You should be careful to set these parameters to reasonable values, or make certain that BE leakage currents account for base current correctly.

Parameter Restrictions and Recommendations

If you exceed the following parameter limits, the parameter is modified as necessary to prevent a numerical error. In general, physically reasonable values are always acceptable.

1. CEMIN > 0.01 CJE.
2. CCMIN > 0.01 CJC.
3. 0.1 < MJx < 0.9, where MJx is any of the parameters beginning with MJ.
4. FC and FCE must be in the range (0.001, 0.999). If either is outside this range, it is set to 0.5.
5. 0.01 < FA < 0.99.
6. Device temperature must be greater than 100 K (-173 C).

Implementation Details

Breakdown and Self-heating

HBT_UCSD includes collector-to-base breakdown and self-heating effects. You should exercise caution in making use of these effects. Breakdown is a strong nonlinearity which can cause convergence difficulties in a harmonic-balance simulator. Reference [1] [11–141] warns that breakdown modeling can cause poor convergence even in time-domain simulators, and [2] [11–141] gives some of the reasons. Self-heating also can introduce negative-resistance effects that cause difficult conversion in harmonic-balance simulation.

You can easily switch these effects on and off with the BKDN and SELFT flag parameters. If SELFT=0, the device temperature is set to TOP, and unless TOP=TNOM, thermal scaling is still performed.

Scaling

Scaling is relatively simple:

All charges are scaled in proportion to AFAC;

All resistors are scaled inversely with AFAC;

All currents are scaled in proportion to AFAC.

The model equations are too extensive to include here. See reference [1] [11–141] for further information.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

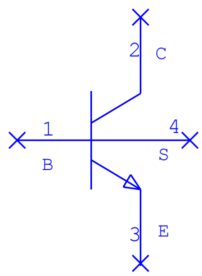
References

[1] UCSD Electrical Engineering Dept., High-Speed Devices Group, HBT Modeling, rev. 9.001a, http://hbt.ucsd.edu, March, 2000.

[2] S. Maas, "Ill Conditioning in Self Heating FET Models," IEEE Microwave and RF Component Letters, March, 2002.

Anholt HBT Model: HBTRA

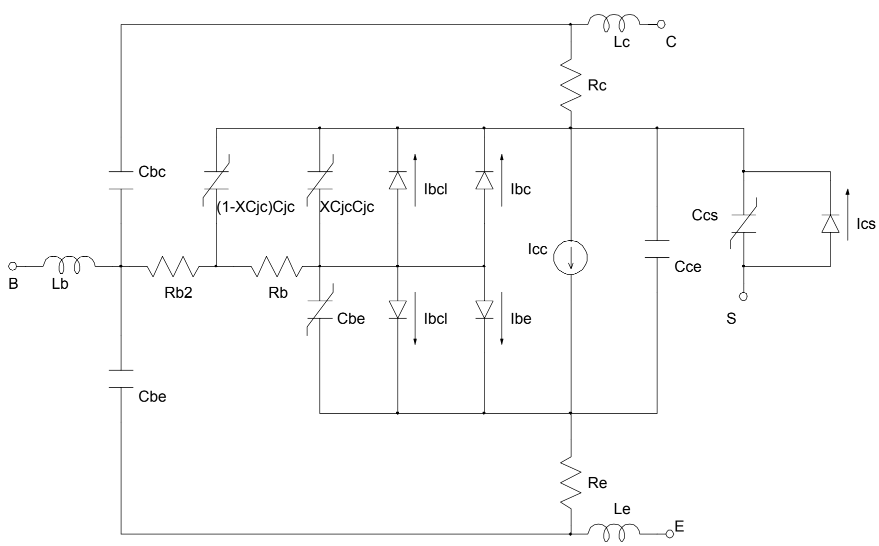
Symbol



Summary

The Anholt HBT model is a modification of the well known Gummel-Poon model. Although it uses a similar topology, its expressions for temperature effects and transit time are very different. The model includes self-heating and accounts for nonreciprocal behavior.

Equivalent Circuit



Parameters

Name	Description	Unit Type	Default
ID	Device ID	Text	HA1
*ISF	Forward saturation current	Current	1e-13
*ISR	Reverse saturation current	Current	1e-13
*BF	Forward current gain		100.0
*NF	Forward ideality factor		1.0

Name	Description	Unit Type	Default
*VAF	Forward early voltage	Voltage	$1.0x10^9$
*IKF	Forward current knee	Current	1e12
*ISE	BE leakage current param	Current	0.0
*NE	BE leakage ideality factor		1.5
*BR	Reverse current gain		1.0
*NR	Reverse ideality factor		1.0
*VAR	Reverse early voltage	Voltage	$1.0x10^9$
*IKR	Reverse current knee	Current	10e12
*ISC	BC leakage current param	Current	0.0
*NC	BC leakage ideality factor		2.0
*CJE	CJ0 for BE junction	Capacitance	0.0
*VJE	BE built-in voltage	Voltage	0.75
*MJE	BE capacitance exponent		0.50
*TB0	Constant part of the base transit time	Time	0.0
*TB1	Variable part of the base transit time	Time	0.0
*NTB	Exponent for the base transit time		5.0
*TC0	Collector transit time	Time	0.0
*BVS	Saturation velocity parameter for TC		$1.0x10^9$
*ITF	High-current parameter for the transit time		0.0
*PTF	Excess phase param; must be in degrees	Degrees	0.0
*CJC	CJ0 for BC junction	Capacitance	0.0
*VJC	BC built-in voltage	Voltage	0.75
*MJC	BC capacitance exponent		0.50
XCJC	Fraction of Cbc connected to the internal node		1.0
TR	Reverse transit time	Time	0.0
*CJS	CJ0 for substrate capacitance	Capacitance	0.0
*VJS	Built-in voltage of the substrate capacitance	Voltage	0.75
*MJS	Substrate capacitance exponent		0.50
*RB	Base resistance	Resistance	0.01
*RB2	Extrinsic base resistance	Resistance	0.01
*RE1	Emitter resistance	Resistance	0.01
*RC2	Collector resistance	Resistance	0.01
*CBE	Extrinsic BE capacitance	Capacitance	0.0
*CBC	Extrinsic BC capacitance	Capacitance	0.0
*CCE	Extrinsic CE capacitance	Capacitance	0.0
*LB	Base inductance (not scaled)	Inductance	0.0
*LE	Emitter inductance (not scaled)	Inductance	0.0

Name	Description	Unit Type	Default
*LC	Collector inductance (not scaled)	Inductance	0.0
*FC	Linearization term for forward-bias depletion capacitance		0.5
*ISS	Collector-substrate current param	Current	0.0
*NS	Collector-substrate ideality factor		1.0
*TS	Collector-substrate diode storage time	Time	0.0
*TNOM	Temperature at which parameters were extracted	Temperature	26.85
*TBP	Baseplate temperature	Temperature	26.85
*XTI	Temperature exponent for diode currents		0.0
*ABF	BF temperature coefficient		0.0
*ABR	BR temperature coefficient		0.0
*ANF	NF temperature coefficient		0.0
*ANR	NR temperature coefficient		0.0
*ANE	NE temperature coefficient		0.0
*ANC	NC temperature coefficient		0.0
*ANS	NS temperature coefficient		0.0
*VSF	Barrier height for ISF thermal modeling	Voltage	0.0
*VSR	Barrier height for ISR thermal modeling	Voltage	0.0
*VSE	Barrier height for ISE thermal modeling	Voltage	0.0
*VSC	Barrier height for ISC thermal modeling	Voltage	0.0
*VSS	Barrier height for ISS thermal modeling	Voltage	0.0
*ARB	RB temperature coefficient		0.0
*ARB2	RB2 temperature coefficient		0.0
*ARE	RE temperature coefficient		0.0
*ARE1	RE1 temperature coefficient		0.0
*ARC2	RC2 temperature coefficient		0.0
*ACJE	CJE temperature coefficient		0.0
*ACJC	CJC temperature coefficient		0.0
*AVJE	VJE temperature coefficient		0.0
*AVJC	VJC temperature coefficient		0.0
*ATB0	TB0 temperature coefficient		0.0
*ATB1	TB1 temperature coefficient		0.0
*ATC0	TB1 temperature coefficient		0.0
*AITF	ITF temperature coefficient		0.0
*RTH	Thermal resistance	Capacitance	0.001
*CTH	Thermal capacitance	Capacitance	1.0
*AFAC	Area scale factor		1.0

** indicates a secondary parameter*

Parameter Details

1. Certain parameters (especially *IKF, IKR, VAF,* and *VAR*) are seldom needed in HBTs. These are included to allow the model to be used for conventional BJTs as well as HBTs. It is essential to note that including such parameters does not necessarily make the model equivalent to the SPICE Gummel-Poon model.
2. Default parameter values for this model may not be the same as those of the SPICE Gummel-Poon model (GBJT). Be sure to be aware of differences when porting existing Gummel-Poon model parameters to this model.
3. Be careful specifying temperature coefficients. It is possible to create nonphysical parameters if they are too great. This is especially true of the NF, NE, NR, and NC parameters.

Implementation Details

SCALING

Resistances, capacitances, and currents are scaled in the usual manner: all resistors are scaled as 1/AFAC; junction currents and all capacitors are scaled in proportion to AFAC. Inductances LB, LE, and LC are not scaled.

EQUATIONS

The model is fundamentally the same as the Gummel-Poon model used in SPICE, with the addition of allowance for nonreciprocal behavior, self-heating, and a new expression for transit time as a function of collector voltage and current. The model also includes common extensions over the expressions in the references.

COLLECTOR CURRENT

The collector current expression in the Gummel-Poon model has been modified to allow for nonreciprocal behavior of HBT devices. The SPICE parameter IS has been replaced by two parameters, ISF and ISR, that describe forward and reverse conduction, respectively. The current expression is

$$I_{\mathrm{cc}} = \frac{\mathrm{ISF}}{Q_b}\left(\exp\left(\frac{qV_{\mathrm{be}}}{\mathrm{NF}.K.T}\right) - 1\right) - \frac{\mathrm{ISR}}{Q_b}\left(\exp\left(\frac{qV_{\mathrm{be}}}{\mathrm{NR}.K.T}\right) - 1\right)$$

Q_b is the normalized base charge, as used in the Gummel-Poon model; q is electron charge, K is Boltzmann's constant, and T is temperature in Kelvin degrees. Note that T varies with power dissipation in the device. *ISF* and *ISR* are also functions of temperature (see below).

TRANSIT TIME

The expression for transit time is

$$\tau_f = \mathrm{TB0} + \mathrm{TB1} \cdot \exp\left(\frac{-\mathrm{ITF}}{I_c} - \left(\frac{\mathrm{ITF}}{I_c}\right)^{\mathrm{NTB}}\right) + \frac{\mathrm{TC0}}{1+\mathrm{BVS}}\sqrt{1 - \frac{V_{\mathrm{be}}}{\mathrm{VJC}}}$$

(When $V_{bc} > \mathrm{VJC}$, the square-root term in is corrected in the same manner as similar terms in depletion capacitances; the method is identical to the one used in SPICE.) The base-to-emitter diffusion charge is

$$q_{\mathrm{diff}} = \tau_f \frac{\mathrm{ISF}}{Q_b}\left(\exp\left(\frac{\mathrm{qV_{be}}}{\mathrm{NF} \cdot K \cdot T}\right) - 1\right)$$

Reverse diffusion charge is included in the model; it is part of the "inner" base-to-collector capacitance. In reverse operation, the transit time is simply TR.

EXCESS PHASE

The *PTF* parameter is an awkward way of specifying a simple time delay in the base-to-emitter voltage. *PTF* is defined as in the SPICE Gummel-Poon model. It must be given in degrees; then, the delay represented by this quantity is

$$\tau_{\mathrm{PTF}} = \mathrm{PTF} \cdot \mathrm{TB0} \cdot \frac{\pi}{180}$$

THERMAL SCALING

Saturation currents (*ISF, ISR, ISE, ISC, ISS*) are scaled as follows:

$$\mathrm{ISX}(T) = \mathrm{ISX}(\mathrm{TNOM})\left(\frac{T}{\mathrm{TNOM}}\right)^{\mathrm{XTI}}\exp\left(\frac{\mathrm{VSX}}{V_t}\left(\frac{T}{\mathrm{TNOM}} - 1\right)\right)$$

where *ISX* is one of the parameters above, and *VSX* is the corresponding voltage parameter.

Other parameters are scaled as

$$P(T) = P(\mathrm{TNOM}) \cdot (1 + \mathrm{AP}(T - \mathrm{TNOM}))$$

where *P* is any of the thermally scaled parameters and AP is the corresponding temperature coefficient. (Be careful using this capability; see Note 3 [11–145], above.) *T* is the instantaneous temperature, which includes effects of self-heating.

SELF-HEATING MODEL

The model uses a conventional electrothermal self-heating model. *RTH* is the thermal resistance in degrees C per watt of power dissipation. CTH is the thermal capacitance, defined as

$$\mathrm{CTH} = \frac{\tau_{\mathrm{TH}}}{\mathrm{RTH}}$$

where t_{TH} is the thermal time constant of the device. *CTH* should never be set to zero; otherwise, the device temperature T cycles at the RF frequency. When the time constant is long compared to a period of the excitation waveform, the device temperature *T* is

$$T = P_{\mathrm{diss}} \cdot \mathrm{RTH} + \mathrm{TBP}$$

The power dissipation, P_{diss}, includes all power dissipated in the device: DC sources at both the base and collector, as well as RF dissipation.

TEMPERATURE LIMITS

To prevent numerical difficulties, the device temperature, *T*, is limited to values greater than 100 K. An abrupt limit at 100 K would introduce convergence difficulties; therefore, the limiting function smooths the temperature in the vicinity of 100 K. This smoothing function introduces a small error throughout the entire temperature range; the maximum error, which occurs at 100 K, is approximately 7 K. Above 200K, the error is less than 0.5 K.

EXTENSIONS

The expression for t_f is somewhat different from those in the references. This modified expression, which is common in industry, provides better accuracy and convergence in harmonic-balance simulators.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

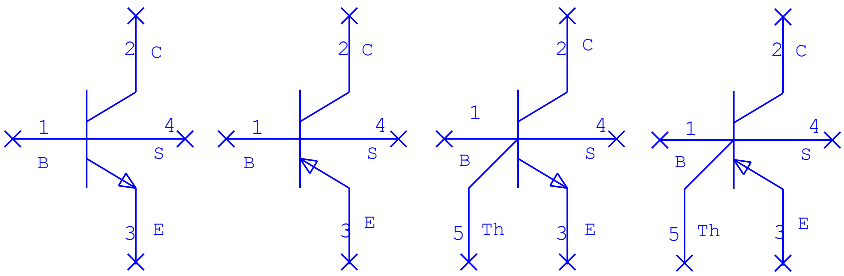
References

[1] R. Anholt et al., "HBT Model Parameter Extractor for SPICE and Harmonic Balance Simulators," IEEE MTT-S International Microwave Symposium Digest, 1994, p. 1257.

[2] R. Anholt, Electrical and Thermal Characterization of MESFETs, HEMTs, and HBTs, Artech House, Norwood, MA, 1995.

HICUM Level 0 BJT Model: HICUM_L0

Symbol



Summary

HICUM_L0 implements the HICUM Level 0 model, which is a simplified version of the HICUM Level 2 model. HICUM_L0 is based on the Verilog-A description of the HICUM Level 0 version 1.12 model, which is available at http://www.iee.et.tu-dresden.de/iee/eb/hic_new/hic_source.html. In terms of complexity and associated computational effort this model is comparable to the well known SPICE-Gummel-Poon-Model (SGPM). However, HICUM Level 0 has some salient features and differences from SGPM which are worth mentioning: a more accurate transit time formulation, a better collector-emitter-voltage dependence of high current behavior, a more physics-based internal base resistance description, support for weak avalanche effects, and separate components for back injection current into the emitter and collector. Parameter temperature dependences, noise, and self-heating are also supported.

Parameters

Name	Description	Unit Type	Default
ID	Device ID	Text	HCM_L0_1
*TNOM	Temperature at which params were extracted	DegC	27
TEMP	Ambient temperature	DegC	_TEMP
MULT	Number of devices in parallel		1
*IS	Modified saturation current	mA	1e-13
*MCF	Non-ideality collector current coefficient		1
*MCR	Non-ideality emitter current coefficient		1
*VEF	Forward normalization voltage	V	1000000
*IQF	Forward DC high-injection roll-off current	mA	1000000000
*IQR	Inverse DC high injection roll-off current	mA	1000000000
*IQFH	Forward DC high current correction	mA	1000000000
*TFH	Forward DC high current correction		1000000
*IBES	BE saturation current	mA	1e-15
*MBE	BE non-ideality factor		1
*IRES	BE recombination saturation current	mA	0
*MRE	BE recombination non-ideality factor		2
*IBCS	BC saturation current	mA	0

Name	Description	Unit Type	Default
*MBC	BC non-ideality factor		1
*CJE0	Zero-bias BE depletion capacitance	pF	1e-8
*VDE	BE built-in Voltage	V	0.9
*ZE	BE exponent factor		0.5
*AJE	BE ratio of maximum to zero-bias voltage		2.5
*T0	Low-current transit time at Vbc=0	ns	0
*DT0H	Base width modulation transit time contribution	ns	0
*TBVL	SCR width modulation transit time contribution	ns	0
*TEF0	Storage time in neutral emitter	ns	0
*GTE	Exponent factor for emitter transit time		1
*THCS	Saturation time at high current densities	ns	0
*AHC	Smoothing factor for current dependence		0.1
*TR	Inverse operation storage time	ns	0
*RCI0	Low-field collector resistance under emitter	ohm	150
*VLIM	Voltage dividing ohmic and saturation regions	V	0.5
*VPT	Push-through voltage	V	100
*VCES	Saturation voltage	V	0.1
*CJCI0	Internal BC zero-bias depletion capacitance	pF	1e-8
*VDCI	Internal BC built-in voltage	V	0.7
*ZCI	Internal BC exponent factor		0.333
*VPTCI	Internal BC punch-through voltage	V	100
*CJCX0	External BC zero-bias depletion capacitance	pF	1e-8
*VDCX	External BC built-in voltage	V	0.7
*ZCX	External BC exponent factor		0.333
*VPTCX	External BC punch-through voltage	V	100
*FBC	Internal/external capacitance split factor		1
*RBI0	Internal base resistance value at zero-bias	ohm	0
*VR0E	Base resistance forward normalization voltage	V	2.5
*VR0C	Base resistance reverse normalization voltage	V	1000000
*FGEO	Base resistance geometric factor		0.656
*RBX	External base series resistance	ohm	0
*RCX	External collector series resistance	ohm	0
*RE	Emitter series resistance	ohm	0
*ITSS	Substrate transistor transfer saturation current	mA	0
*MSF	Substrate transistor transfer current non-ideality factor		1
*ISCS	CS saturation current	mA	0
*MSC	CS non-ideality factor		1

Name	Description	Unit Type	Default
*CJS0	Zero-bias CS depletion capacitance	pF	1e-8
*VDS	CS built-in voltage	V	0.3
*ZS	CS exponent factor		0.3
*VPTS	CS punch-through voltage	V	100
*CBCPAR	Emitter-base overlap capacitance	pF	0
*CBEPAR	Collector-base oxide capacitance	pF	0
*EAVL	BC avalanche current exponent factor		0
*KAVL	BC avalanche current prefactor		0
*KF	Flicker noise coefficient		0
*AF	Flicker noise exponent factor		2
*VGB	Bandgap voltage	V	1.2
*VGE	Effective emitter bandgap-voltage	V	1.17
*VGC	Effective collector bandgap-voltage	V	1.17
*VGS	Effective substrate bandgap-voltage	V	1.17
*F1VG	Coefficient K1 in T-dependent bandgap equation (V/K)		-0.000102377
*F2VG	Coefficient K2 in T-dependent bandgap equation (V/K)		0.00043215
*ALT0	First-order TC of forward transit time		0
*KT0	Second-order TC of forward transit time		0
*ZETACT	Exponent coefficient in transfer current temperature dependence		3
*ZETABET	Exponent coefficient in BE junction current temperature dependence		3.5
*ZETACI	TC of epi_collector diffusivity		0
*ALVS	Relative TC of saturation drift velocity		0
*ALCES	Relative TC of Vce		0
*ZETARBI	TC of internal base resistance		0
*ZETARBX	TC of external base resistance		0
*ZETARCX	TC of collector external resistance		0
*ZETARE	TC of emitter resistance		0
*ALKAV	TC of KAVL		0
*ALEAV	TC of EAVL		0
*ALB	Relative TC of forward current gain		0
SELFT	Self-heating support		off
*RTH	Thermal resistance	ohm	0
*CTH	Thermal capacitance	pF	0
FLSH	Flag for self-heating calculation		COMPLETE
TYPE	Device type		NPN

** indicates a secondary parameter*

Implementation Details

The TYPE parameter controls whether the device is NPN or PNP. The SELFT parameter enables/disables the self-heating modeling capabilities. The current setting of either parameter is immediately reflected by the device symbol. The FLSH, CTH, and RTH parameters are only visible when self-heating is enabled. The FLSH parameter determines how the dissipated power calculation is performed.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Operating Points

Parameter	Description
Igc (Current)	Avalanche current
gmu (Conductance)	Feedback transconductance
gm (Conductance)	Transconductance
go	Output conductance
ro	Output resistance
rx (Resistance)	Base spreading resistance
cmu (Capacitance)	Feedback capacitance
cpi (Capacitance)	Base capacitance
gpi(Resistance)	Base conductance
rpi(Resistance)	Base resistance
dic_dvbe(Conductance)	
dic_dvbc (Conductance)	
dib_dvbe (Conductance)	
dib_dvbc (Conductance)	
rbi (Resistance)	Resistance in bb branch
rci (Resistance)	Resistance in cc branch
rbp (Resistance)	Resistance in bbp branch
cjc (Capacitance)	Capacitance in bc branch
cje (Capacitance)	Capacitance in be branch
cbex (Capacitance)	Capacitance in bex branch
cbcx (Capacitance)	Capacitance in bcx branch
cbep (Capacitance)	Capacitance in bep branch
cbcp (Capacitance)	Capacitance in bcp branch
betaac	AC current amplification
betadc	DC current amplification

Parameter	Description
temp (Temperature)	Simulation temperature
ft (Frequency)	Cutoff frequency approximation
pwr (Power)	Dissipated power

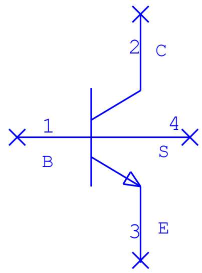
.

References

This model was developed under research performed at NI AWR Corporation. The full set of details of the implementation are considered proprietary in nature.

(Obsolete) HICUM Level 2 NPN Transistor: HICUM_L2

Symbol



Summary

This element is OBSOLETE and is replaced by the HICUM Level 2 Version 2.22 BJT Model (HICUM_L2A) element. HICUM Level 2 is a semi-physical compact bipolar transistor model. HICUM is based on an extended and generalized Integral Charge Control Relation (ICCR). However, in contrast to the original Gummel_Poon model and as well as its variants, in HICUM the ICCR concept is applied consistently without simplifications and additional fitting parameters. Quantities like depletion capacitances, transit times and associated charges, which determine the dynamic transistor behavior, are considered as basic quantities of the model. The important physical and electrical effects taken into account by HICUM_L2 include: high current effects, emitter current crowding, two and three dimensional collector current spreading, temperature dependence and self-heating (HICUM_L2_SH and HICUM_L2_SH_P only), weak avalanche breakdown, tunneling in the base-emitter junction, bandgap differences, substrate transistor, etc.

Compared to the Spice Gummel-Poon model the equivalent circuit of HICUM contains two additional circuit nodes. Furthermore, the transfer current is not an explicit function of branch voltages an its computation relies on an internal Newton solver. Due to this additional complexity, simulations using the HICUM_L2 model are in general slower than those employing the GBJT model. As a result, HICUM_L2 should be used only where the need for accuracy justifies its complexity, and where parameter libraries are available.

Parameters

Name	Description	Unit Type	Default
ID	Device ID		Q1
*TNOM	Temperature at which parameters were extracted	DegC	25
*TEMP	Device Temperature	DegC	25
*C10	GICCR constant (AC)		2e-30
*QPO	Zero-bias hole charge (C)		2e-14
*MCF	Forward ideality factor		1
*ICH	High-current correction factor		1e20
*HFC	Collector minority charge weighting factor in HBTs		1
*HFE	Emitter minority charge weighting factor in HBTs		1
*HJCI	B-C depletion charge weighting factor in HBTs		1
*HJEI	B-E depletion charge weighting factor in HBTs		1
*CJEI0	Internal B-E zero-bias depletion capacitance	pF	0

Name	Description	Unit Type	Default
*VDEI	Internal B-E built-in potential		0.9
*ZEI	Internal B-E grading coefficient		0.5
*ALJEI	Ratio of maximum to zero-bias value of internal B-E capacitance		2.5
*CJCI0	Internal B-C zero-bias depletion capacitance	pF	0
*VDCI	Internal B-C built-in potential	V	0.7
*ZCI	Internal B-C grading coefficient		0.4
*VPTCI	Internal B-C punch through voltage	V	1e20
*T0	Low-current forward transit time	ns	0
*DT0H	B-C width modulation time constant	ns	0
*TBVL	Low voltage carrier jam time constant	ns	0
*TEFO	Neutral emitter storage time	ns	0
*GTFE	Neutral emitter storage time current dependence exponent		1
*THCS	High current saturation time constant		0
*ALHC	Base and collector current dependence smoothing factor		0.0
*FTHC	Base and collector partitioning factor		0
*ALIT	Factor for additional delay time of minority charge		0
*ALQF	Factor for additional delay time of transfer current		0
*RCI0	Low field internal collector resistance	ohm	150
*VLIM	ohmic and saturation regime boundary voltage	V	0.4
*VPT	Collector punch-through voltage	V	3
*VCES	Internal C-E saturation voltage	V	0.1
*TR	Storage time for inverse operation	ns	0
*IBEIS	Internal B-E saturation current	mA	1e-15
*MBEI	Internal B-E current ideality factor		1
*IREIS	Internal B-E recombination saturation current	mA	1e-27
*MREI	Internal B-E recombination current ideality factor		2
*IBCIS	Internal B-C saturation current	mA	1e-13
*MBCI	Internal B-C current ideality factor		1
*FAVL	Avalanche current factor		0
*QAVL	Exponent factor for avalanche current		0
*RBI0	Zero-bias internal base resistance	ohm	0
*FDQR0	B-E and B_C space charge modulation correction factor		0
*FGEO	Emitter current crowding geometry factor		0.73
*FQI	Internal to minority charge ratio		0.9055
*LATB	Scaling factor for collector minority charge (emitter width direction)		3.765

Name	Description	Unit Type	Default
*LATL	Scaling factor for collector minority charge (emitter length direction)		0.342
*CJEP0	Peripheral B-E zero-bias depletion capacitance	pF	0.00207
*VDEP	Peripheral B-E built in potential	V	1.05
*ZEP	Peripheral B-E grading coefficient		0.4
*ALJEP	Maximum to zero-bias peripheral B-E capacitance ratio		2.4
*IBEPS	Peripheral B-E saturation current	mA	3.72e-18
*MBEP	Peripheral B-E current ideality factor		1.015
*IREPS	Peripheral B-E recombination saturation current	mA	1e-27
*MREP	Peripheral B-E recombination current ideality factor		2
*IBETS	B-E tunneling saturation current	mA	0
*ABET	Exponent factor for tunneling current		40
*CJCX0	External B-C zero-bias depletion capacitance	pF	0.005393
*VDCX	External B-C built in potential	V	0.7
*ZCX	External B-C grading coefficient		0.333
*VPTCX	External B-C punch through voltage	V	100
*CCOX	B-C overlap capacitance	pF	0.00297
*FBC	Partitioning factor for external B-C capacitance		0.1526
*IBCXS	External B-C saturation current	mA	4.39e-17
*MBCX	External B-C current ideality factor		1.03
*CEOX	B-E isolation capacitance	pF	0.00113
*RBX	External base series resistance	ohm	0
*RE	Emitter series resistance	ohm	0
*RCX	External collector series resistance	ohm	0
*ITSS	Substrate transistor saturation current	mA	0
*MSF	Forward ideality factor of substrate transfer current		1
*MSR	Reverse ideality factor of substrate transfer current		1
*TSF	Substrate forward transit time	ns	0
*ISCS	Saturation current of C-S diode	mA	0
*MSC	Ideality factor of C-S diode		1
*CJS0	C-S zero-bias depletion capacitance	pF	0.0364
*VDS	C-S built-in potential	V	0.6
*ZS	C-S grading coefficient		0.447
*VPTS	C-S punch-through voltage	V	1000
*RSU	Substrate series resistance	ohm	1e-5
*CSU	Substrate series capacitance	pF	0
*KF	Flicker noise coefficient		1.43e-8

Name	Description	Unit Type	Default
*AF	Flicker noise exponent factor		1
*FFE	Flicker noise frequency exponent		1
*KRBI	Internal base resistance noise factor		1
*VGB	Bandgap-voltage extrapolated to 0K	V	1.17
*ALB	Forward current gain relative temperature coefficient		0.0063
*ALT0	First-order relative temperature coefficient of T0		0
*KT0	Second-order relative temperature coefficient of T0		0
*ZETACI	RCI temperature exponent		1.6
*ALVS	Saturation velocity relative temperature coefficient		0.001
*ALCES	VCES relative temperature coefficient		0.0004
*ZETARBI	RCI temperature exponent		0.588
*ZETARBX	RBX temperature exponent		0.206
*ZETARCX	RCX temperature exponent		0.223
*ZETARE	RE temperature exponent		0
*ALFAV	Relative temperature coefficient for FAVL		8.25e-5
*ALQAV	Relative temperature coefficient for QAVL		0.000196
*RTH	Thermal resistance	ohm	0
*CTH	Thermal capacitance	pF	0
*FCRBI	Ratio of HF shunt to total internal capacitance		1
*NFLAG	Noise analysis flag		on
*MULT	Scale factor		1
*FBCS	External B-C capacitance partitioning factor (not implemented)		-1
*IS	Ideal Saturation Current	mA	-1000
*ZETACX	Epi-layer temperature exponent factor (not implemented)		0.223
*SSFLAG	Small signal model flag: autodifferentiation, or, specialized.		specialized

** indicates a secondary parameter*

Operating Points

The following letter sets have been used to identify the NL branches: ce, be, bc, bb, bex, bcx, sc, and bsx. Here be, bc and ce correspond to the intrinsic base-emitter, base-collector and collector-emitter branches, respectively. bb denotes the branch connecting the intrinsic and extrinsic base nodes. bex and bcx correspond to the branches formed by connecting the emitter or collector nodes to the extrinsic base node. Finally, sc and bsx refer to the branches connecting the substrate node to the collector or extrinsic base nodes, respectively. These names are used to identify branch related operating point information, i.e., branch voltages, currents, etc.

Parameter	Description
ibet (A)	Tunneling current.

Parameter	Description
iavl (A)	Avalanche current.
qjei (Coul)	Internal base-emitter space charge.
cjei (F)	Internal base-emitter depletion capacitance.
qjep (Coul)	Peripheral base-emitter space charge.
cjep (F)	Peripheral base-emitter depletion capacitance.
qjci (Coul)	Internal base-collector space charge.
cjci (F)	Internal base-collector depletion capacitance.
qjcp (A)	Qjcp.
cjcp (A)	Cjcp.
qjcx (A)	Qjcx.
cjcx (A)	Cjcx.
cjcp (A)	Cjcp.
qjcx (A)	Qjcx.
cjcx (A)	Cjcx.
qjs (Coul)	Substrate space charge.
cjs (F)	Substrate capacitance.
tf (s)	Transit Time.
qf (Coul)	Minority charge.
qdei (F)	Internal base-emitter diffusion charge.
cdei (F)	Internal base-emitter diffusion capacitance.
qdci (F)	Internal base-collector diffusion charge.
cdci (F)	Internal base-collector diffusion capacitance.
qp (Coul)	Hole charge.
itf (A)	Forward Transfer current.
itr (A)	Reverse Transfer current.
it (A)	Transfer current.
sfb	Sfb variable.
srb	Srb variable.
sfc	Sfc variable.
src	Src variable.
gm (S)	Common-emitter transconductance.
si (S)	Common emitter output conductance.
rbi (ohm)	Internal base resistance.
pwr (W)	Power dissipation.
temp (C)	Temperature.
ft (Hz)	Unity small-signal current-gain frequency.
rpi (ohm)	Common-emitter input resistance.

Parameter	Description
ro (ohm)	Common-emitter output resistance.
rx (ohm)	Parasitic base resistance.
cpi (F)	Common-emitter input capacitance.
cmu (F)	Common-base output capacitance.

Implementation Details

Except for subtle differences in the implementation of the Non Quasi Static effects, HICUM_L2 has been implemented following the HICUM Level 2 original model definition[1] [11–158] striclty. HICUM_L2 model is mapped into HSPICE as a NPN G-device with parameter LEVEL set to 8.

Parameters NFLAG and SSFLAG, are unique to the Microwave Office implementation of HICUM Level 2. **NFLAG**'s behavior is quite obvious and deserves no further comment. The **SSFLAG** has effect in linear simulations only. It enables the user to toggle between the "specialized" model, where some of the derivatives are neglected or have been modified, as prescribed in the defining document [1] [11–158], and the complete small signal model, obtained by automatic differentiation of the original large signal expressions.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

Restrictions

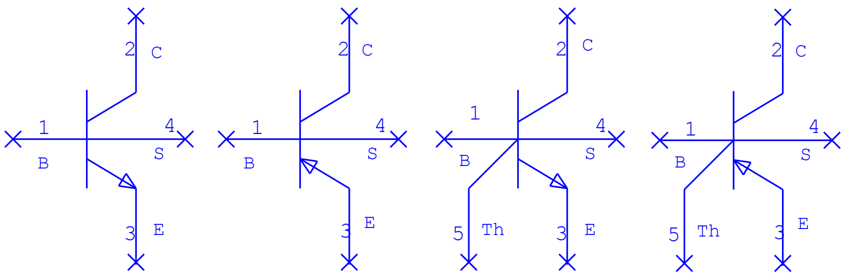
1. Parameters RTH and CTH are used to characterize self-heating effects. They are not supported in HICUM_L2. These parameters are supported in HICUM_L2_SH and HICUM_L2_SH_P devices only.
2. Parameters ZETACX and FBCS are HSPICE extensions to the model definition and are included for HSPICE compatibility purposes only. They have no effect in Harmonic balance simulations.
3. For the purpose of Harmonic Balance simulations only, the low-current forward transit time, T0, replaces the total forward time, t_f, in the expressions used for the modeling of NQS effects.

Reference

[1] Michael Schroter, "HICUM, a scalable physics-based compact bipolar transistor model", http://www.iee.et.tu-dresden.de/~schroter/Models/hicman.pdf, December 2000

HICUM Level 2 Version 2.22 BJT Model: HICUM_L2A

Symbol



Summary

HICUM_L2A was conceived as a replacement for models HICUM_L2, HICUM_L2_P, HICUM_L2_SH, and HICUM_L2_SH_P. HICUM_L2A is based on the Verilog-A description of the HICUM Level 2 Version 2.22 model, which is available on the web. The HICUM Level 2 model is a semi-physical compact bipolar transistor model. HICUM is based on an extended and generalized Integral Charge Control Relation (ICCR). However, in contrast to the original Gummel-Poon model and its variants, in HICUM the ICCR concept is applied consistently without simplifications or additional fitting parameters. Quantities like depletion capacitances, transit times and associated charges, which determine the dynamic transistor behavior, are considered as basic quantities of the model.

The important physical and electrical effects taken into account by HICUM_L2A include: high current effects, emitter current crowding, 2- and 3-dimensional collector current spreading, temperature dependence and self-heating (SELFT on), weak avalanche breakdown, tunneling in the base-emitter junction, bandgap differences, substrate transistor, etc. Compared to the SPICE Gummel-Poon model, the equivalent circuit of HICUM contains two additional circuit nodes. Furthermore, the transfer current is not an explicit function of branch voltages and its computation relies on an internal Newton solver. Due to this additional complexity, simulations using the HICUM_L2A model are generally slower than those employing the GBJT model. As a result, you should use HICUM_L2A only where the need for accuracy justifies its complexity, and where parameter libraries are available.

Parameters

Name	Description	Unit Type	Default
ID	Device ID		HCM L0 1
*TNOM	Temperature at which params were extracted	DegC	27
TEMP	Ambient temperature	DegC	_TEMP
MULT	Number of devices in parallel		1
*C10	GICCR constant		2e-30
*QP0	Zero-bias hole change		2e-14
*ICH	High-current correction for 2D and 3D effects		0
*HFE	Emitter minority charge weighting factor in HBTs		1
*HFC	Collector minority charge weighting factor in HBTs		1
*HJEI	B-E depletion charge weighting factor in HBTs		1
*HJCI	B-C depletion charge weighting factor in HBTs		1
*IBEIS	Internal B-E saturation current	mA	1e-15

Name	Description	Unit Type	Default
*MBEI	Internal B-E current ideality factor		1
*IREIS	Internal B-E recombination saturation current	mA	0
*MREI	Internal B-E recombination current ideality factor		2
*IBEPS	Peripheral B-E saturation current	mA	0
*MBEP	Peripheral B-E current ideality factor		1
*IREPS	Peripheral B-E recombination saturation current	mA	0
*MREP	Peripheral B-E recombination current ideality factor		2
*MCF	Non-ideality factor for III-V HBTs		1
*TBHREC	Base-current recombination time constant at B-C barrier for high forward injection		0
*IBCIS	Internal B-C saturation current	mA	1e-13
*MBCI	Internal B-C current ideality factor		
*IBCXS	External B-C saturation current	mA	0
*MBCX	External B-C current ideality factor		1
*IBETS	B-E tunneling saturation current	mA	0
*ABET	Exponent factor for tunneling current		40
*TUNODE	Specifies the base node connection for the tunneling current		1
*FAVL	Avalanche current factor		0
*QAVL	Exponent factor for avalanche current		0
*ALFAV	Relative TC for FAVL		0
*ALQAV	Relative TC for QAVL		0
*RBI0	Zero bias internal base resistance	ohm	0
*RBX	External base series resistance	ohm	0
*FGEO	Factor for geometry dependence of emitter current crowding		0.6557
*FDQR0	Correction factor for modulation by B-E and B-C space charge layer		0
*FCRBI	Ratio of HF shunt to total internal capacitance (lateral NQS effect)		0
*FQI	Ratio of internal to total minority charge		1
*RE	Emitter series resistance	ohm	0
*RCX	External collector series resistance	ohm	0
*ITSS	Substrate transistor transfer saturation current	mA	0
*MSF	Forward ideality factor of substrate transfer current		1
*ISCS	C-S diode saturation current	mA	0
*MSC	Ideality factor of C-S diode current		1
*TSF	Transit time for forward operation of substrate transistor	ns	0
*RSU	Substrate series resistance	ohm	0

Name	Description	Unit Type	Default
*CSU	Substrate shunt capacitance	pF	0
*CJEI0	Internal B-E zero-bias depletion capacitance	pF	1e-8
*VDEI	Internal B-E built-in potential	V	0.9
*ZEI	Internal B-E grading coefficient		0.5
*AJEI	Ratio of maximum to zero-bias value of internal B-E capacitance		2.5
*CJEP0	Peripheral B-E zero-bias depletion capacitance	pF	1e-8
*VDEP	Peripheral B-E built-in potential	V	0.9
*ZEP	Peripheral B-E grading coefficient		0.5
*AJEP	Ratio of maximum to zero-bias value of peripheral B-E capacitance		2.5
*CJCI0	Internal B-C zero-bias depletion capacitance	pF	1e-8
*VDCI	Internal B-C built-in potential	V	0.7
*ZCI	Internal B-C grading coefficient		0.4
*VPTCI	Internal B-C punch-through voltage	V	100
*CJCX0	External B-C zero-bias depletion capacitance	pF	1e-8
*VDCX	External B-C built-in potential	V	0.7
*ZCX	External B-C grading coefficient		0.4
*VPTCX	External B-C punch-through voltage	V	100
*FBCPAR	Partitioning factor of parasitic B-C cap		0
*FBEPAR	Partitioning factor of parasitic B-E cap		1
*CJS0	C-S zero-bias depletion capacitance	pF	0
*VDS	C-S built-in potential	V	0.6
*ZS	C-S grading coefficient		0.5
*VPTS	C-S punch-through voltage	V	100
*T0	Low current forward transit time in VBC=0V	ns	0
*DTOH	Time constant for base and B-C space charge layer width modulation	ns	0
*TBVL	Time constant for modelling carrier jam at low VCE	ns	0
*TEF0	Neutral emitter storage time	ns	0
*GTFE	Exponent factor for current dependence of neutral emitter storage time		1
*THCS	Saturation time constant at high current densities	ns	0
*AHC	Smoothing factor for current dependence of base and collector transit time		0.1
*FTHC	Partitioning factor for base and collector portion		0
*RCI0	Internal collector resistance at low electric field	ohm	150
*VLIM	Voltage separating ohmic and saturation velocity regime	V	0.5

Name	Description	Unit Type	Default
*VCES	Internal C-E saturation voltage	V	0.1
*VPT	Collector punch-through voltage	V	0
*TR	Storage time for inverse operation	ns	0
*CBEPAR	Total parasitic B-E capacitance	pF	0
*CBCPAR	Total parasitic B-C capacitance	pF	0
*ALQF	Factor for additional delay time of minority charge		0
*ALIT	Factor for additional delay time of transfer current		0
*FLNQS	Flag for turning on and off of vertical NQS effect		0
*KF	Flicker noise coefficient		0
*AF	Flicker noise exponent factor		2
*CFBE	Flag for determining where to tag the flicker noise source		(-1)
*LATB	Scaling factor for collector minority charge in direction of emitter width		0
*LATL	Scaling factor for collector minority charge in direction of emitter length		0
*VGB	Bandgap voltage extrapolated to 0 K	V	1.17
*ALT0	First order relative TC of parameter T0		0
*KT0	Second order relative to TC of parameter T0		0
*ZETACI	Temperature exponent for RCI0		0
*ALVS	Relative TC of saturation drift velocity		0
*ALCES	Relative TC of VCES		0
*ZETARBI	Temperature exponent for internal base resistance		0
*ZETARBX	Temperature exponent of external base resistance		0
*ZETARCX	Temperature exponent for external collector resistance		0
*ZETARE	Temperature exponent of emitter resistance		0
*ZETACX	Temperature exponent of mobility in substrate transistor transit time		0
*VGE	Effective emitter bandgap voltage	V	1.17
*VGC	Effective collector bandgap voltage	V	1.17
*VGS	Effective substrate bandgap voltage	V	1.17
*F1VG	Coefficient K1 in T-dependent band-gap equation		(-1.023770E-004)
*F2VG	Coefficient K2 in T-dependent band-gap equation		0.00043215
*ZETACT	Exponent coefficient in transfer current temperature dependence		3
*ZETABET	Exponent coefficient in B-E junction current temperature dependence		3.5
*ALB	Relative TC of forward current gain for V2.1 model		0
SELFT	Self-heating support		off

Name	Description	Unit Type	Default
*RTH	Thermal resistance	ohm	0
*CTH	Thermal capacitance	pF	0
FLSH	Flag for self-heating calculation		COMPLETE
TYPE	Device type		NPN
*FLCOMP	Flag for compatibility with v2.1 model (0=v2.1)		0

** indicates a secondary parameter*

Implementation Details

The TYPE parameter controls whether the device is NPN or PNP. The SELFT parameter is used to enable/disable the Self-Heating modeling capabilities. The current setting of either parameter is immediately reflected by the device symbol. The FLSH, CTH and RTH parameters are only visible when Self-Heating is enabled. The FLSH parameter determines how the dissipated power calculation is performed.

Layout

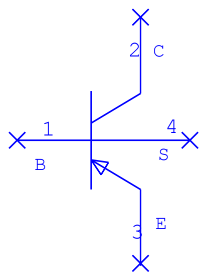
This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

References

This model was developed under research performed at AWR Corporation. The full set of details of the implementation are considered proprietary in nature.

(Obsolete) HICUM Level 2 PNP Transistor: HICUM_L2_P

Symbol



Summary

This element is OBSOLETE and is replaced by the HICUM Level 2 Version 2.22 BJT Model (HICUM_L2A) element. This is a PNP implementation of the HICUM Level 2, version 2.1 transistor. It is identical to HICUM_L2, except for being of PNP-type. As a result all diode and source currents and charges are reversed.

Parameters

Parameters are identical to those of HICUM_L2.

Implementation Details

Same as its NPN counterpart, this model has self-heating disabled. Accordingly, parameters RTH and CTH are disabled. See HICUM_L2 for information on parameters and implementation.

Layout

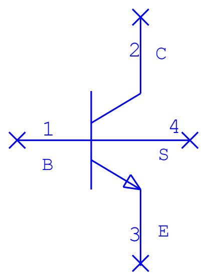
This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Operating Points

Operating point information is the same as in HICUM_L2.

(Obsolete) HICUM Level 2 NPN Transistor with Self-Heating: HICUM_L2_SH

Symbol



Summary

This element is OBSOLETE and is replaced by the HICUM Level 2 Version 2.22 BJT Model (HICUM_L2A) element. This is an NPN implementation of the HICUM Level 2, version 2.1 transistor with support for self-heating.

Implementation Details

With the exception of parameters CTH and RTH, which support self-heating, all other parameters behave as in HICUM_L2. See HICUM_L2 for information on parameters and implementation.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Recommendations for Use

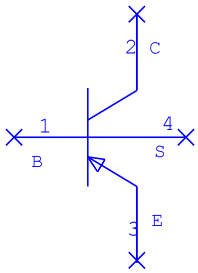
Due to the additional complexity associated with modeling self-heating, and additional NL branch and recomputation of all temperature compensation equations each time the model is evaluated, this model is slower than HICUM_L2. Consequently, we recommend using this model only when self-heating needs to be accounted for.

Operating Points

Operating point information is the same as in HICUM_L2.

(Obsolete) HICUM Level 2 PNP Transistor with Self-Heating: HICUM_L2_SH_P

Symbol



Summary

This element is OBSOLETE and is replaced by the HICUM Level 2 Version 2.22 BJT Model (HICUM_L2A) element. This is a PNP implementation of the HICUM Level 2, version 2.1 transistor with support for self-heating.

Implementation Details

With the exception of the CTH and RTH parameters which support self-heating, all other parameters behave as in HICUM_L2. See HICUM_L2 for information on parameters and implementation.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Recommendations for Use

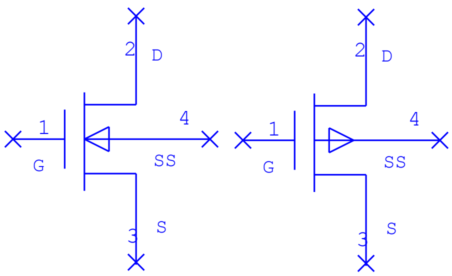
Due to the additional complexity associated with modeling self-heating, (one additional NL branch and recomputation of all temperature compensation equations each time the model is evaluated), this model is slower than HICUM_L2_P. Consequently, we recommend using this model only when self-heating must be accounted for.

Operating Points

Operating point information is the same as in HICUM_L2.

Hiroshima University STARC IGFET Model: HISIM

Symbol



Summary

NI AWR's HISIM element implements Hiroshima University's STARC IGFET Model. HISIM is Verilog-A based and implements version 251.

HISIM is the first complete surface-potential-based MOSFET model for circuit simulation based on the drift-diffusion approximation. This modeling approach relies on a unified description of device characteristics for all bias conditions. All new phenomena such as short-channel and reverse-short-channel effects are included in the surface potential calculations, causing modifications resulting from the features of these advanced technologies.

See the model website[1] [11–168] for complete documentation and Verilog-A definition of component models.

Parameters

Only primary parameters are shown. Secondary parameters follow the model specification/standard.

Name	Description	Binnable	Unit Type	Default
ID	Device ID		Text	M1
TYPE	Device Type			NMOS
TNOM	Parameter extraction temperature		Temperature	27
TEMP	Ambient temperature		Temperature	_TEMP
MULT	Number of devices in parallel			1

Operating Points

You can access operating point information, as defined by the active Verilog-A based component model.

Implementation Details

The TYPE parameter controls whether the device is NMOS or PMOS; its setting is reflected by the device symbol. The extraction and simulation temperatures are controlled using the TNOM and TEMP parameters, respectively. The MULT parameter determines the number of devices in parallel. The remaining parameter default and truncation values are identical to those found in the Verilog-A definition of the model.

Layout

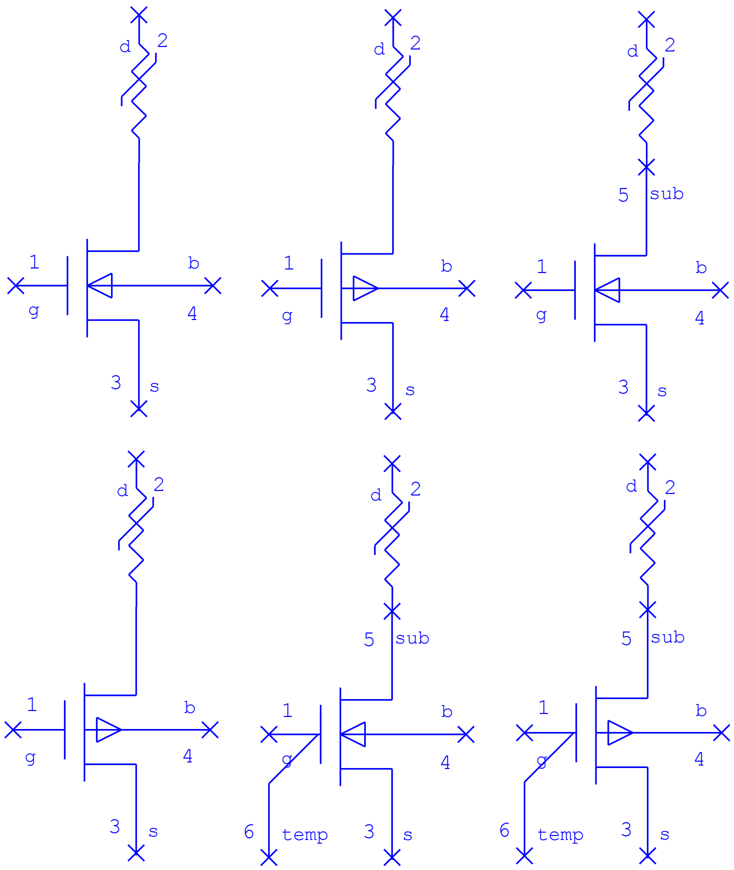
This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

References

[1] http://home.hiroshima-u.ac.jp/usdl/HiSIM2/

Hiroshima University STARC IGFET Model with High Voltage Extensions: HISIM_HV

Symbol



Summary

NI AWR's HISIM_HV element implements Hiroshima University's STARC IGFET Model with High Voltage Extensions. This model extends the application range of the HISIM model to two types of commonly used structures, LDMOS (laterally diffused MOS), and HVMOS (symmetrical High Voltage MOS) devices. The most important features of LDMOS/HVMOS devices, which set them apart from conventional MOSFET devices, originate in the drift region. This additional region enables the device to sustain high voltages. The basic modeling approach is taken from the HISIM2 model. Additional equations for capturing the drift-region effects are included.

HISIM_HV works as a single entry point to a family of Verilog-A defined component models. The appropriate Verilog-A defined component is selected according to the version and number of exposed terminals you specify. Version 2.1.0 is the latest version this model supports.

See the model website[1] [11–170] for complete documentation and Verilog-A definition of component models.

Parameters

Only primary parameters are shown. Secondary parameters follow the model specification/standard.

Name	Description	Binnable	Unit Type	Default
ID	Device ID		Text	M1
VERSION	Version selector			2.01
TYPE	Device Type			NMOS
TERMS	Number of terminals exposed			g d s b
TNOM	Parameter extraction temperature		Temperature	27DegC
TEMP	Simulation temperature		Temperature	_TEMP
MULT	Number of devices in parallel			1

Operating Points

You can access operating point information, as defined by the active Verilog-A based component model.

Implementation Details

The TYPE parameter controls whether the device is NMOS or PMOS. The TERMS parameter controls which terminals are exposed. You can select three different values: "g d s b", "g d s b sub", and "g d s b sub temp". The terminal names are the same as those used in the Verilog-A definition of the model. The current setting of any of these parameters is reflected by the device symbol. The VERSION parameter sets the version as "1.2.0" or "2.1.0". The extraction and simulation temperatures are controlled using the TNOM and TEMP parameters, respectively. The number of devices in parallel is determined by the MULT parameter. Parameter default and truncation values are identical to those found in the Verilog-A definition of the model.

Layout

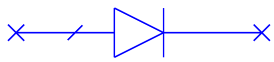
This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

References

[1] http://home.hiroshima-u.ac.jp/usdl/HiSIM_HV/

Philips JUNCAP Model: JUNCAP

Symbol



Summary

The JUNCAP model is appropriate for characterizing the behavior of diodes formed by the source, drain, or well-to-bulk junctions in MOST devices. The model is limited to the case of reverse biasing of these junctions. The currents contributions from the sidewall, bottom, and gate-edge-junctions are considered separately. This allows to include the effects resulting from differences in the junction doping profiles. Accordingly, the equivalent circuit is comprised of three diode-like current sources in parallel, one for each of the junctions. Both the diffusion and the generation currents are considered in the model, each with its own temperature and voltage dependence.

Parameters

Name	Description	Unit Type	Default
ID	Device ID	Text	PN1
*TNOM	Parameter extraction temperature	DegC	25
*TEMP	Device temperature	DegC	25
*MULT	Number of devices in parallel		1
*AB	Diffusion area (m^2)		1e-12
*LS	Side-wall length which is not under gate	um	1
*LG	Side-wall length under gate	um	1
*VR	Parameter value reference voltage	V	0
*JSGBR	Bottom saturation current density due to electron-hole generation (A m^-2)		0.001
*JSDBR	Bottom saturation current density due to diffusion from back-contact (A m^-2)		0.001
*JSGSR	Side-wall saturation current density due to electron-hole generation (A m^-2)		0.001
*JSDSR	Side-wall saturation current density due diffusion from back-contact (A m^-2)		0.001
*JSGGR	Gate-edge saturation current density due to electron-hole generation (A m^-2)		0.001
*JSDGR	Gate-edge saturation current density due diffusion from back-contact (A m^-2)		0.001
*NB	Bottom forward current emission coefficient		1
*NS	Side-wall forward current emission coefficient		1
*NG	Gate-edge forward current emission coefficient		1
*VB	Reverse breakdown voltage	V	0.9
*CJBR	Bottom junction capacitance (Fm^-2)		1e-12

Name	Description	Unit Type	Default
*CJSR	Side-wall junction capacitance (Fm^-2)		1e-12
*CJGR	Gate-edge junction capacitance (Fm^-2)		1e-12
*VDBR	Bottom junction diffusion voltage	V	1
*VDSR	Side-wall junction diffusion voltage	V	1
*VDGR	Gate-edge junction diffusion voltage	V	1
*PB	Bottom junction grading coefficient		0.4
*PS	Side-wall junction grading coefficient		0.4
*PG	Gate-edge junction grading coefficient		0.4

** indicates a secondary parameter*

Parameter Details

The default and restrictions on the range of parameter values is in full compliance with the model definition.

Implementation Details

This model is mapped into HSPICE as a LEVEL 4 D-device. For the complete set of equations the interested reader is referred to Ref [2].

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

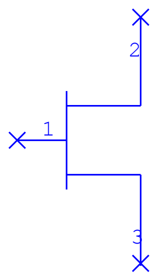
References

[1] Philips Semiconductor web site, http://www.semiconductors.philips.com/Philips_Models/additional/juncap/.

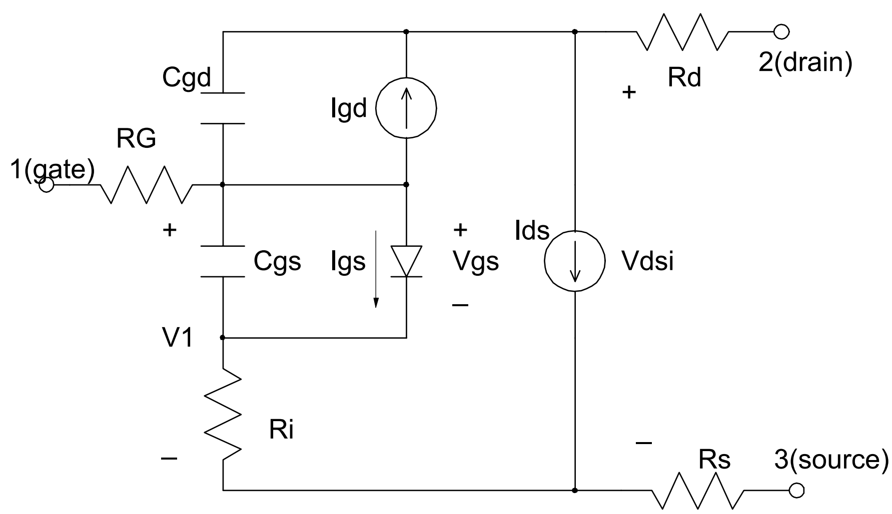
[2] Philips Semiconductor web site, http://www.semiconductors.philips.com/Philips_Models/additional/m_juncap/.

Modified Materka FET Model: MATRK

Symbol



Equivalent Circuit



Parameters

Name	Description	Unit Type	Default
ID	Device ID	Text	MF1
*IDSS	Drain saturation current at zero Vgs	Current	100 mA
*VP	Pinch-off voltage	Voltage	-2 V
*GAM	Pinch-off slope parameter		0
*E	Constant part of gate I/V exponent		2
*KE	Dependence of power law on Vgs		0
*SL	Slope of drain I/V in linear region		0.15
*KG	Drain Vgs dependence in linear region		0
*SS	Slope of drain I/V in saturation		0
*T	Gate-drain time delay	Time	0 ns

Name	Description	Unit Type	Default
*IG0	Saturation current of GS diode	Current	0 mA
*AFAG	Exponential coef of GS diode		38.7
*IB0	Saturation current of GD diode	Current	0 mA
*AFAB	Exponential coef of GD diode		38.7
*VBC	Breakdown voltage	Voltage	1e+06 V
*R10	Intrinsic resistance	Resistance	0.001 ohm
*KR	(Not implemented)		0
*C10	Cgs at zero voltage	Capacitance	0 pF
*K1	Inverse of built-in voltage for G-D cap.		1.25
*C1S	Constant part of Cgs	Capacitance	0 pF
*CF0	Cgd at zero voltage	Capacitance	0 pF
*KF	Inverse of built-in voltage for G-D cap.		1.25
*RS	Source resistance	Resistance	0.001 ohm
*RG	Gate resistance	Resistance	0.001 ohm
*RD	Drain resistance	Resistance	0.001 ohm
*P	Noise par: P		2
*Tg	Noise par: gate noise temp	Temperature	16.85
*NKF	Flicker noise coefficient		0
*AF	Flicker noise exponent		1
*FFE	Flicker noise freq. exponent		1
*NFLAG	Noise model		AWR1
*NFING	No. of fingers scale factor		1
AFAC	Gate-width scale factor		1

** indicates a secondary parameter*

Implementation Details

I/V Characteristic

$$I_{ds} = \mathrm{AFAC}\cdot\mathrm{IDSS}\left(1+\mathrm{SS}\frac{V_{dsi}}{\mathrm{IDSS}}\right)\left(1-\frac{V_{gsi}(t-T)}{\mathrm{VPO}+\mathrm{GAM}\cdot V_{dsi}}\right)^{\mathrm{EXPON}}\cdot\tanh\left(\frac{\mathrm{SL}\cdot V_{dsi}}{\mathrm{IDSS}\cdot(1-\mathrm{KG}\cdot V_{gss}(t-T))}\right)$$

where

$$\mathrm{EXPON} = E+\mathrm{KE}\cdot V_{gsi}(t-T)$$

$$I_{gs} = \mathrm{IG0}\left(e^{\mathrm{AFAG}\cdot V_{gsi}}-1\right)$$

$$I_{gd} = \mathrm{IB0}(e^{\mathrm{AFAG} \cdot V_{dsi} - V_i - \mathrm{VBC}} - 1)$$

$$R_i = \begin{cases} R10(1 - \mathrm{KR} \cdot V_{gsi}) & \mathrm{KR} \cdot V_{gsi} < 1.0 \\ 0 & \mathrm{KR} \cdot V_{gsi} \geq 1.0 \end{cases}$$

Capacitance

$$C_{gs} = \begin{cases} \dfrac{C10}{\sqrt{1 - K1 \cdot V_{gsi}}} + C1S & K1 \cdot V_{gsi} < 0.8 \\ C10\sqrt{5} + C1S & K1 \cdot V_{gsi} \geq 0.8 \end{cases}$$

$$C_{gd} = \begin{cases} \dfrac{\mathrm{CF0}}{\sqrt{1 - \mathrm{KF}(V_1 - V_{dsi})}} & \mathrm{KF} \cdot (V_1 - V_{dsi}) < 0.8 \\ \mathrm{CF0}\sqrt{5} & \mathrm{KF} \cdot (V_1 - V_{dsi}) \geq 0.8 \end{cases}$$

Parameter Scaling

$$\mathrm{RG} \rightarrow \frac{\mathrm{RG} \cdot \mathrm{AFAC}}{\mathrm{NFING}^2}$$

$$\mathrm{RD} \rightarrow \frac{\mathrm{RD}}{\mathrm{AFAC}}$$

$$\mathrm{RS} \rightarrow \frac{\mathrm{RS}}{\mathrm{AFAC}}$$

$$R10 \rightarrow \frac{R10}{\mathrm{AFAC}}$$

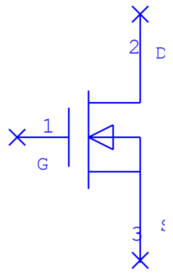
C_{gs}, C_{gd}, and the currents I_{ds}, I_{gs}, and I_{gd} are scaled in proportion to AFAC.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

Motorola Electrothermal MOSFET Model: MET_LDMOS

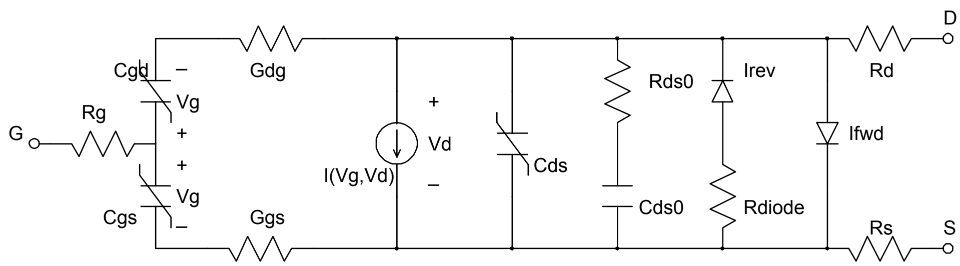
Symbol



Summary

The MET_LDMOS model was developed by Motorola specifically to model LDMOS devices. It includes self-heating.

Equivalent Circuit



Parameters

The following parameters are included in the MET_LDMOS model implemented in the XML library:

Name	Description	Unit Type	Default
ID	Element ID	Text	LD1
*RG_0	Gate resistance at 0C	Resistance	1 ohm
*RG_1	Gate resistance temp coef	Resistance	0.001 ohm
*RS_0	Source resistance at 0C	Resistance	0.1 ohm
*RS_1	Source resistance temp coef	Resistance	0.0001 ohm
*RD_0	Drain resistance at 0C	Resistance	1.5 ohm
*RD_1	Drain resistance temp coef	Resistance	0.0015 ohm
*VTO_0	Threshold voltage at 0C	Voltage	3.5 V
*VTO_1	VT temp coef	Voltage	-0.001 V
*GAMMA	IDS parameter		-0.02
*VST	Subthreshold slope parameter	Voltage	0.15 V

Name	Description	Unit Type	Default
*BETA_0	IDS BETA parameter at 0C		0.2
*BETA_1	BETA temp coef		-0.0002
*LAMBDA	IDS parameter		-0.0025
*VGEXP	IDS parameter		1.1
*ALPHA	IDS parameter		1.5
*VK	IDS parameter	Voltage	7 V
*DELTA	IDS parameter	Voltage	0.9 V
*VBR_0	Breakdown V at 0C and Vgs=0	Voltage	75 V
*VBR_1	VBR temp coef	Voltage	0.01 V
*K1	Breakdown parameter		1.5
*K2	Breakdown parameter		1.15
*M1	Breakdown parameter		9.5
*M2	Breakdown parameter		1.2
*M3	Breakdown parameter		0.001
*BR	Reverse Ids parameter		0.5
*RDIODE_0	Reverse diode series res at 0C	Resistance	0.5 ohm
*RDIODE_1	Rev diode temp coef	Resistance	0.001 ohm
*ISR	Rev diode leakage	Current	1e-10 mA
*NR	Rev diode ideality		1
*VTO_R	Rev diode threshold voltage	Voltage	3 V
*RTH	Thermal resistance (deg C/W)	Resistance	10 ohm
*RDSO	Fixed drain-source resistance	Resistance	1e+05 ohm
*CDSO	Cap in series with Rds0	Capacitance	1e+09 pF
*GGS	Gate-source conductance	Conductance	100S
*GGD	Gate-drain conductance	Conductance	100S
*TAU	Gate transit time	Time	0.001 ns
*TNOM	Temp at which parameters were measured	Temperature	25 DegC
*TSNK	Heat-sink (baseplate) temp	Temperature	25 DegC
*CGST	CGS temp coef		0.001
*CDST	CDS temp coef		0.001
*CGDT	CGD temp coef		0
*CTH	Thermal capacitance		1
*KF	Flicker noise coefficient		0
*AF	Flicker noise exponent		1
*FFE	Flicker noise frequency exponent		1
*N	Forward diode ideality factor		1
*ISS	Forward diode leakage		1e-13

Name	Description	Unit Type	Default
*CGS1	Cgs parameter 1	Capacitance	2 pF
*CGS2	Cgs parameter 2	Capacitance	1 pF
*CGS3	Cgs parameter 3	Voltage	-4 V
*CGS4	Cgs parameter 4	Capacitance	1 pF
*CGS5	Cgs parameter 5		0.25
*CGS6	Cgs parameter 6		3.5
*CGD1	Cgd parameter 1	Capacitance	0.4 pF
*CGD2	Cgd parameter 2	Capacitance	0.1 pF
*CGD3	Cgd parameter 3		0.1
*CGD4	Cgd parameter 4	Voltage	4 V
*CDS1	Cds parameter 1	Capacitance	1 pF
*CDS2	Cds parameter 2	Capacitance	1.5 pF
*CDS3	Cds parameter 3		0.1
*AREA	Area scale factor		1
*N_FING	Gate finger scale factor (inverse of the usual def.)		1
*NFLAG	Noise model		MET_LDMOS

** indicates a secondary parameter*

Implementation Details

The MET_LDMOS model is a new LDMOS model, developed by W. Curtice in cooperation with Motorola (see References). The model includes self-heating, using an electrothermal equivalent circuit.

The model has a number of nice features. The nonlinear capacitances are continuous functions of a single control voltage, eliminating difficulties with discontinuities. Similarly, the drain I/V characteristic is a continuous, infinitely differentiable function. These characteristics are responsible, in part, for the model's robust convergence characteristics.

NOTE

NI AWR has implemented two changes from the description in [2] [11–179]: (1) the elements RDS0 and CDS0 have been included; and (2) to prevent nonphysical thermal cycling, the default value of CTH is 1.0 F instead of zero.

The definition of N_FING is unusual; it is the ratio of the number of gates of the standard device to the number of gates of the scaled device:

$$N\_\mathrm{FING} = \frac{\mathrm{NGates}_{\mathrm{std}}}{\mathrm{NGates}_{\mathrm{scaled}}}$$

N_FING is the inverse of the gate-scaling parameter used in most of the other models. The definition of AREA is the conventional one, however:

$$\mathrm{AREA} = \frac{\mathrm{NGates}_{\mathrm{scaled}}}{\mathrm{NGates}_{\mathrm{std}}}$$

Equations

The model equations are too extensive to be repeated here; Ref. [2] [11–179] is the standard documentation for the model.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

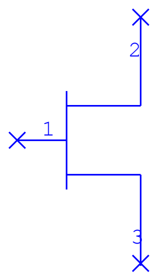
References

[1] W. Curtice, J. Pla, D. Bridges, T. Liang, and E. Shumate, "A New Dynamic Electro-Thermal Nonlinear Model for Silicon RF LDMOS FETs," IEEE International Microwave Symposium Digest of Papers, 1999.

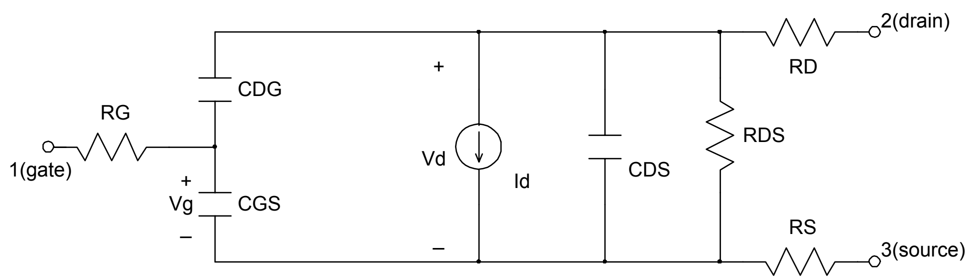
[2] Motorola's Electro Thermal (MET) LDMOS Model, Motorola Internal Report. (No identifying numbers or date; available by download from http://mot-sps.com/models/ldmos//ldmosmodels.html.)

Maas-Neilsen FET Model: MNFET

Symbol



Equivalent Circuit



Parameters

Name	Description	Unit Type	Default
ID	FET Identification	Text	F11
*RG	Gate resistance	Resistance	1 ohm
*RD	Drain resistance	Resistance	1 ohm
*RDS	Drain to source resistance	Resistance	300 ohm
*RS	Source resistance	Resistance	1 ohm
*CGS	Gate to source capacitance	Capacitance	0.1 pF
*CDG	Gate to drain capacitance	Capacitance	0.1 pF
*CDS	Drain to source resistance	Capacitance	0.1 pF
*VT	Threshold (pinchoff) voltage	Voltage	-1 V
*VF	Maximum gate voltage	Voltage	0.5 V
*A0	Drain current coefficient A0		0.032
*A1	Drain current coefficient A1		0.05
*A2	Drain current coefficient A2		0.05
*A3	Drain current coefficient A3		0.05
*A4	Drain current coefficient A4		0.05

Name	Description	Unit Type	Default
*GAMMA	Drain voltage coefficient		1

** indicates a secondary parameter*

Implementation Details

I/V Characteristic

For

$$V_g \leq \mathrm{VT}$$

$$I_d = 0.0$$

For

$$V_g > \mathrm{VT}$$

$$I_d = (A0 \cdot X_{\mathrm{rad}} + A1 \cdot \sin(X_{\mathrm{rad}}) + A2 \cdot \sin(2X_{\mathrm{rad}}) + A3 \cdot \sin(3X_{\mathrm{rad}}) + A4 \cdot \sin(4X_{\mathrm{rad}})) \cdot \tanh(\Gamma \cdot V_d)$$

where

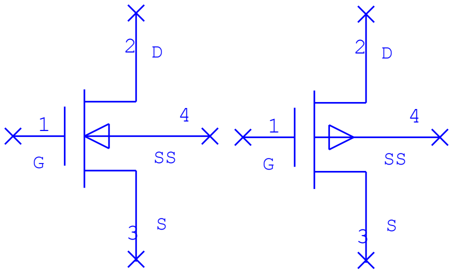
$$X_{\mathrm{rad}} = \pi \frac{(V_g - \mathrm{VT})}{(\mathrm{VF} - \mathrm{VT})}$$

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Philips Level 1102 MOSFET Model: MOS11

Symbol



Summary

NI AWR's MOS11 element implements the entire family of Philips MOS model 11 level 1102 devices, i.e., mos1102e, mos1102et, mos11020, mos11020t, mos11021 and mos11021t. Element MOS11 is based on the Philips SiMKit 2.2.

MOS Model 11 was developed by Philips as the successor of MOS Model 9. The development goals for MOS Model 11 were:

Suitable for digital, analog and RF

Suitable for modern and future CMOS technologies

Physics based

Number of parameters comparable to MOS Model 9

Simulation time comparable to MOS Model 9

Simple parameter extraction

The best reference for the model equations is the Philips website.[1] [11–199]

Philips MOS model 11 level 1102, is an updated version of Level 1101, which in turn replaced MOS model 11 Level 1100. Level 1102 uses the same parameter set and parameter scaling relations as Level 1101, but uses slightly different model equations. The surface potential is calculated iteratively using a second-order Newton-Raphson procedure, resulting in a more accurate description of surface potential. In addition, a more physical and simpler velocity saturation expression is used, and as a consequence the saturation voltage expression has changed slightly as well. Source-bulk and drain-bulk junction diodes are not accounted for in the model and need to be added externally. The model supports Self-Heating and accounts for thermal noise, induced gate noise and their correlation.

Parameters

Name	Description	Binnable	Unit Type	Default
ID	Element ID		Text	
LEVEL	Parameters and clipping selector			0
SELFT	Self-Heating flag			Off

Name	Description	Binnable	Unit Type	Default
*TNOM	Parameter extraction temperature		Temperature	21DegK
*TEMP	Device temperature		Temperature	_TEMP
*NFLAG	Noise flag			Noise On
*GATENOISE	Flag for in/exclusion of introduced gate thermal noise			Gate Thermal Noise Excluded
*MULT	Number of devices in parallel			1
*LVAR	Difference between the actual and the programmed poly-silicon gate length			0
*LAP	Effective channel length reduction per side			4e-8
*WVAR	Difference between the actual and the programmed field-oxide opening			0
*STVFB	Coefficient of temperature dependence of VFB			0.5e-3
*KOR	Body effect coefficient for then reference transistor			0.5
*SLKO	Coefficient for then length dependence of KO			0.0
*SL2KO	Second coefficient of then length dependence of KO			0.0
*SWKO	Coefficient of the width dependence of KO			0.0
*PHIBR	Surface potential at strong inversion			0.95
*STPHIB	Coefficient of the temperature dependency of PHIB			-8.50e-4
*SLPHIB	Coefficient of the lenngth dependence of PHIB			0.0

Name	Description	Binnable	Unit Type	Default
*SL2PHIB	Second coefficient of then length dependence of PHIB			0.0
*SWPHIB	Coefficient of the width dependence of PHIB			0.0
*BETSQ	Gain factor for an infinite square transistor			
*ETABETR	Exponent of the temperature dependence of the gain factor			
*SLETABET	Coefficient of length dependence of ETABETR			0.0
*FBET1	Relative mobility decrease due to first lateral profile			0.0
*LP1	Characteristic length of first lateral profile			0.0
*FBET2	Relative mobility decrease due to second lateral profile			0.0
*LP2	Characteristic length of second lateral profile			0.8e-6
*THESRR	Coefficient of the mobility reduction due to surface roughness scattering			
*ETASR	Exponent of the temperature dependence of THESR			
*SWTHESR	Coefficient of the width dependence of THESR			
*THEPHR	Coefficient of the mobility reduction due to phonon scattering			
*ETAPH	Exponent of the temperature dependence of THEPH			

Name	Description	Binnable	Unit Type	Default
*SWTHEPH	Coefficient of the width dependence of THEPH			0.0
*ETAMOBR	Effective field parameter for dependence on depletion/inversion charge			
*STETAMOB	Coefficient of the temperature dependence of ETAMOB			0.0
*SWETAMOB	Coefficient of the width dependence of ETAMOB			0.0
*NU	Exponent of field dependence of mobility model			2.0
*NUEXP	Exponent of the temperature dependence of parameter NU			
*THERR	Coefficient of the series resistance			
*SWTHER	Coefficient of the width dependence of THER			0.0
*THESATR	Velocity saturation parameter due to optical/acoustic phonon scattering			
*SLTHESAT	Coefficient of length dependence of THESAT			1.0
*THESATEXP	Exponent of length dependence of THESAT			1.0
*SWTHESAT	Coefficient of the width dependence of THESAT			0.0
*THETHR	Coefficient of self-heating			
*THETHEXP	Exponent of the length dependence of THETH			1.0

Name	Description	Binnable	Unit Type	Default
*SWTHETH	Coefficient of the width dependence of THETH			0.0
*SDIBLO	Drain-induced barrier lower parameter			1.0e-4
*SDIBLEXP	Exponent of the length dependence of BDIBL			1.35
*MOO	Parameter for short-channel subthreshold slope			0.0
*MOR	Parameter for short-channel subthreshold slope per unit length			0.0
*MOEXP	Exponent of the length dependence of MO			1.34
*SSFR	Static feedback parameter			6.25e-3
*SLSSF	Coefficient of the length dependence of SSF			1.0
*SWSSF	Coefficient of the width dependence of SSF			0.0
*ALPR	Factor of the channel length modulation			0.01
*SLALP	Coefficient of the length dependence of ALP			1.0
*ALPEXP	Exponent of the length dependence of ALP			1.0
*SWALP	Coefficient of the width dependence of ALP			0.0
*LMIN	Minimum effect channel length in technology, used for calculation of smoothing factor m			0.12e-6
*A1R	Factor of the weak-avalanche current			6.0
*STA1	Coefficient of the temperature dependence of A1			0.0

Name	Description	Binnable	Unit Type	Default
*SLA1	Coefficient of the length dependence of A1			0.0
*SWA1	Coefficient of the width dependence of A1			0.0
*A2R	Exponent of the weak-avalanche current			38.0
*SLA2	Coefficient of the length dependence of A2			0.0
*SWA2	Coefficient of the width dependence of A2			0.0
*A3R	Factor of the drain-source voltage above which weak-avalanche occur			1.0
*SLA3	Coefficient of the length dependence of A3			0.0
*SWA3	Coefficient of the width dependence of A3			0.0
*IGINVR	Gain factor for the intrinsic gate tunnelling current in inversion			0.0
*BINV	Probability factor for intrinsic gate tunnelling current in inversion			
*IGACCR	Gain factor for intrinsic gate tunnelling current in inversion			0.0
*BACC	Probability factor in intrinsic gate tunnelling current in accumulation			48.0
*VFBOV	Flat-band voltage for the source/drain overlap extensions			0.0
*IGOVR	Gain factor for source/drain overlap gate tunnelling current			0.0

Name	Description	Binnable	Unit Type	Default
*AGIDLR	Gain factor for gate-induced leakage current			0.0
*STBGIDL	Coefficient of the temperature dependence of BGIDL			-3.368e-4
*CGIDL	Factor for the lateral field dependence of the gate-induced leakage current			0.0
*COL	Gate overlap capacitance for a channel width of 1 um			3.20e-16
*NT	Thermal noise coefficient			1.65e-20
*NFAR	First coefficient of the flicker noise for a channel area of 1 um^2			
*NFBR	Second coefficient of the flicker noise of a channel area of 1 um^2			
*NFCR	Third coefficient of the flicker noise for a channel area of 1 um^2			
*WOT	Effective channel width reduction per side			0
*KO	Coefficient for the geometry independent, length dependence, width dependence and length times width dependence.	Yes		0.0
*PHIB	Coefficient for the geometry independent, length dependence, width dependence and length times width dependence.	Yes		0.0
*BET	Coefficient for the geometry independent, length dependence, width dependence and length times width dependence.	Yes		0.0

Name	Description	Binnable	Unit Type	Default
*THESR	Coefficient for the geometry independent, length dependence, width dependence and length times width dependence.	Yes		0.0
*THEPH	Coefficient for the geometry independent, length dependence, width dependence and length times width dependence.	Yes		0.0
*ETAMOB	Coefficient for the geometry independent, length dependence, width dependence and length times width dependence.	Yes		0.0
*THER	Coefficient for the geometry independent, length dependence, width dependence and length times width dependence.	Yes		0.0
*THER1	Numerator of gate voltage dependent part of series resistance			0
*THER2	Denominator of gate voltage dependent part of series resistance			1
*THESAT	Coefficient for the geometry independent, length dependence, width dependence and length times width dependence.	Yes		0.0
*THETH	Coefficient for the geometry independent, length dependence, width dependence and length times width dependence.	Yes		0.0
*SDIBL	Coefficient for the geometry independent, length dependence, width dependence and	Yes		0.0

Name	Description	Binnable	Unit Type	Default
	length times width dependence.			
*MO	Coefficient for the geometry independent, length dependence, width dependence and length times width dependence.	Yes		0.0
*SSF	Coefficient for the geometry independent, length dependence, width dependence and length times width dependence.	Yes		0.0
*ALP	Coefficient for the geometry independent, length dependence, width dependence and length times width dependence.	Yes		2.5e-2, 0.0
*VP	Characteristic voltage of channel-length modulation			0.05
*MEXP	Coefficient for the geometry independent, length dependence, width dependence and length times width dependence.	Yes		5.0e-2, 0.0
*A1	Coefficient for the geometry independent, length dependence, width dependence and length times width dependence.	Yes		0.0
*A2	Coefficient for the geometry independent, length dependence, width dependence and length times width dependence.	Yes		0.0
*A3	Coefficient for the geometry independent, length dependence, width dependence and length times width dependence.	Yes		0.0

Name	Description	Binnable	Unit Type	Default
*IGINV	Coefficient for the geometry independent, length dependence, width dependence and length times width dependence.	Yes		0.0
*BINV	Coefficient for the geometry independent, length dependence, width dependence and length times width dependence.	Yes		0.0
*IGACC	Coefficient for the geometry independent, length dependence, width dependence and length times width dependence.	Yes		0.0
*BACC	Coefficient for the geometry independent, length dependence, width dependence and length times width dependence.	Yes		4.8e+1, 0.0
*IGOV	Coefficient for the geometry independent, length dependence, width dependence and length times width dependence.	Yes		0.0
*AGIDL	Coefficient for the geometry independent, length dependence, width dependence and length times width dependence.	Yes		0.0
*BGIDL	Coefficient for the geometry independent, length dependence, width dependence and length times width dependence.	Yes		0.0
*CGIDL	Coefficient for the geometry independent, length dependence, width dependence and	Yes		0.0

Name	Description	Binnable	Unit Type	Default
	length times width dependence.			
*COX	Coefficient for the geometry independent, length dependence, width dependence and length times width dependence.	Yes		0.0
*CGDO	Coefficient for the geometry independent, length dependence, width dependence and length times width dependence.	Yes		0.0
*CGSO	Coefficient for the geometry independent, length dependence, width dependence and length times width dependence.	Yes		0.0
*NFA	Coefficient for the geometry independent, length dependence, width dependence and length times width dependence.	Yes		0.0
*NFB	Coefficient for the geometry independent, length dependence, width dependence and length times width dependence.	Yes		0.0
*NFC	Coefficient for the geometry independent, length dependence, width dependence and length times width dependence.	Yes		0.0
*TVFB	Coefficient for the geometry independent, length dependence, width dependence and length times width dependence.	Yes		5.0e-4, 0.0
*TPHIB	Coefficient for the geometry independent, length dependence,	Yes		-8.5e-4, 0.0

Name	Description	Binnable	Unit Type	Default
	width dependence and length times width dependence.			
*TETABET	Coefficient for the geometry independent, length dependence, width dependence and length times width dependence.	Yes		0.0
*TETASR	Coefficient for the geometry independent, length dependence, width dependence and length times width dependence.	Yes		0.0
*TETAPH	Coefficient for the geometry independent, length dependence, width dependence and length times width dependence.	Yes		0.0
*TETAMOB	Coefficient for the geometry independent, length dependence, width dependence and length times width dependence.	Yes		0.0
*TNUEXP	Coefficient for the geometry independent, length dependence, width dependence and length times width dependence.	Yes		0.0
*TETAR	Coefficient for the geometry independent, length dependence, width dependence and length times width dependence.	Yes		0.0
*TETASAT	Coefficient for the geometry independent, length dependence, width dependence and length times width dependence.	Yes		0.0
*TA1	Coefficient for the geometry independent,	Yes		0.0

Name	Description	Binnable	Unit Type	Default
	length dependence, width dependence and length times width dependence.			
*TBGIDL	Coefficient for the geometry independent, length dependence, width dependence and length times width dependence.	Yes		-3.638e-4, 0.0
*VFB	Flat-band voltage at reference temperature			-1.05
*KO	Body-effect factor			5.0e-01
*PHIB	Surface potential at the onset of strong inversion			9.5e-01
*ETABET	Exponent of the temperature dependence of the gain factor			
*THESR	Mobility degradation parameter due to surface roughness scattering			
*THESAT	Velocity saturation parameter due to optical/acoustic phonon scattering			
*ETHASAT	Exponent of the temperature dependence of THESAT			
*THETH	Coefficient of self-heating			
*MO	Parameter for (short-channel) subthreshold slope			0.0
*SSF	Static-feedback parameter			
*MEXP	Smoothing factor			5.0
*IGINV	Gain factor for intrinsic gate tunnelling current in inversion			0.0

Name	Description	Binnable	Unit Type	Default
*IGACC	Gain factor for intrinsic gate tunnelling current in accumulation			0.0
*IGOV	Gain factor for source/drain overlap tunnelling current			0.0
*AGIDL	Gain factor for gate-induced leakage current			0.0
*BGIDL	Probability factor for gate-induced drain leakage current at reference temperature			41.0
*COX	Oxide capacitance for the intrinsic channel (*mult)			
*NFA	First coefficient for the flicker noise			
*NFB	Second coefficient for the flicker noise			
*NFC	Third Coefficient of the flicker noise			
*BET	Gain factor			
*THEPH	Mobility degradation parameter due to phonon scattering			
*ETAMOB	Effective field parameter for dependence on depletion charge			
*THER	Coefficient of series resistance			
*ETAR	Exponent of the temperature dependence of THER			
*SDIBL	Drain-induced barrier lowering parameter			
*ALP	Factor of channel length modulation			2.5e-02
*A1	Factor of the weak-avalanche current			
*A2	Exponent of the weak-avalanche current			

Name	Description	Binnable	Unit Type	Default
*A3	Factor of the drain-source voltage above which weak-avalanche occurs			
*THESR	Mobility degradation parameter due to surface roughness scattering			
*ETASAT	Exponent of the temperature dependence of THESAT			
*BGIDL	Probability factor for gate-induced drain leakage current at reference temperature			41
*TOX	Thickness of gate oxide layer			3.2e-9
*RTH	Thermal resistance			300
*CTH	Thermal capacitance			3e-9
*KPINV	Inverse of body-effect factor of the poly-silicon gate			0
*ETAR	Exponent of the temperature dependence of THER			
*KOV	Body-effect factor for the Source/Drain overlap extensions			2.5
*L	Drawn channel length in the layout. Scale set by option scale.			2e-6
*W	Drawn channel width in the layout. Scale set by option scale.			1e-5
*ATH	Temperature coefficient of the thermal resistance			0
TYPE	Device Type			1

Operating Points

The following letter pairs have been used to identify the NL branches: ds, bs, bd, gs, gb and gd. Here g,d,s abd b correspond to the gate, drain, source and substrate terminals, respectively. These are used to identify branch related operating point information, i.e., branch voltages, currents, etc

Parameter	Description
cgg (Capacitance)	$\frac{\partial Q_g}{\partial V_g}$, intrinsic charge.
cgb (Capacitance)	$\frac{\partial Q_g}{\partial V_b}$, intrinsic charge.
cgd (Capacitance)	$\frac{\partial Q_g}{\partial V_d}$, intrinsic charge.
cgs (Capacitance)	$\frac{\partial Q_g}{\partial V_s}$, intrinsic charge.
cbg (Capacitance)	$\frac{\partial Q_b}{\partial V_g}$, intrinsic charge.
cbb (Capacitance)	$\frac{\partial Q_b}{\partial V_b}$, intrinsic charge.
cbd (Capacitance)	$\frac{\partial Q_b}{\partial V_d}$, intrinsic charge.
cbs (Capacitance)	$\frac{\partial Q_b}{\partial V_s}$, intrinsic charge.
cdg (Capacitance)	$\frac{\partial Q_d}{\partial V_g}$, intrinsic charge.
cdb (Capacitance)	$\frac{\partial Q_d}{\partial V_b}$, intrinsic charge.
cdd (Capacitance)	$\frac{\partial Q_d}{\partial V_d}$, intrinsic charge.
cds (Capacitance)	$\frac{\partial Q_d}{\partial V_s}$, intrinsic charge.
csg (Capacitance)	$\frac{\partial Q_s}{\partial V_g}$, intrinsic charge.
csb (Capacitance)	$\frac{\partial Q_s}{\partial V_b}$, intrinsic charge.
csd (Capacitance)	$\frac{\partial Q_s}{\partial V_d}$, intrinsic charge.
css (Capacitance)	$\frac{\partial Q_s}{\partial V_s}$, intrinsic charge.
cgdo (Capacitance)	Gate-drain overlap capacitance

Parameter	Description
cgso (Capacitance)	Gate-source overlap capacitance
pwr (Power)	Dissipated power
temp (Temperature)	Simulation temperature
vto (Voltage)	Zero-bias threshold voltage
vts (Voltage)	Threshold voltage including back-bias effects
vth (Voltage)	Threshold voltage including back-bias and drain-bias effects
vgt (Voltage)	Effective gate drive including back-bias and drain-bias effects
vdss (Voltage)	Drain saturation voltage at actual bias
vsat (Voltage)	Saturation limit
vearly (Voltage)	Equivalent Early voltage
gm (Conductance)	Transconductance
gmb (Conductance)	Substrate transconductance
gds (Conductance)	Output conductance
weff (Length)	Effective channel width for geometrical models
leff (Length)	Effective channel length for geometrical models
rout (Resistance)	Small signal output resistance
fknee (Frequency)	Cross-over frequency above which white noise is dominant
fug (Frequency)	Unity gain frequency
u	Transistor gain
keff	Body effect parameter
beff	Gain factor
sqrtsfw	Input-referred RMS white noise voltage density
sqrtsff	Input-referred RMS white noise voltage density at 1kHz

.

Implementation Details

Parameter TYPE controls whether the device is N, or, P channel. The user can select the desired model level using parameter LEVEL, which allows selecting between physical geometrical scaling rules (11020), binning geometrical scaling rules (11021), and the electrical model (1102e). Unlike the comprehensive parameter list above, only those parameters corresponding to a given level are displayed with the element. Parameter SELFT controls whether Self-Heating simulation is supported, or, not. The extraction and simulation temperatures correspond to parameters TNOM and TEMP, respectively; instead of TR and DTA. Parameter default and truncation values are identical as those employed by Philips. Operating point information also follows Philip's prescription closely.

Layout

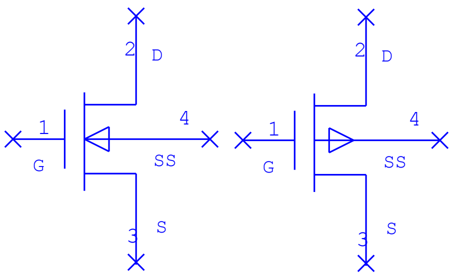
This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

References

[1] http://www.semiconductors.philips.com/acrobat_download/other/philipsmodels/m1102.pdf.

Philips Level 3100 MOSFET Model: MOS31

Symbol



Summary

AWR's MOS31 element implements Philips MOS model 31 level 3100 devices, i.e., mos3100 and mos3100t. AWR's model implementation is based on the SiMKit 2.2.

MOS Model 31 is a physics based transistor model to be used in circuit simulation and IC-design of analogue high-voltage applications. The model describes the electrical behavior of a junction-isolated accumulation/depletion-type MOSFET. The model is used as the drain extension of high-voltage MOS devices, like the Lateral Double-diffused MOS (LDMOS), the Vertical Double-diffused MOS (VDMOS), and the Extended MOS transistors.

The following physical effects are included in MOS Model 31:

- Accumulation and depletion underneath the gate oxide
- Depletion from the substrate (a pn-junction)
- Pinch-off effects
- Velocity saturation
- Temperature scaling

The best reference for the model equations is the Philips website.[1] [11–203]

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	M1
SELFT	Self- Heating flag		off
*TNOM	Parameter extraction temperature	Temperature	21DegK
*TEMP	Device temperature	Temperature	_TEMP
*MULT	Number of devices in parallel		1
*RON	Ohmic resistance at zero bias	Resistance	1 ohm
*RSAT	Space charge resistance at zero bias	Resistance	1 ohm
*VSAT	Critical drain-source voltage for hot carriers	Voltage	10V
*PSAT	Velocity saturation coefficient		1
*VP	Pinch off voltage at zero gate and substrate voltages	Voltage	-1V

Name	Description	Unit Type	Default
*TOX	Gate oxide thickness	Length	-1000000m
*DCH	Doping level channel (m^ -3)		1e21
*DSUB	Doping level substrate (m^ -3)		1e21
*VSUB	Substrate diffusion voltage	Voltage	0.6V
*VGAP	Bandgap voltage channel	Voltage	1.2V
*CGATE	Gate capacitance at zero bias	Capacitance	0nF
*CSUB	Substrate capacitance at zero bias	Capacitance	0nF
*TAUSC	Space charge transit time of the channel	Time	0ns
*ACH	Temperature coefficient resistivity of the channel		0
*RTH	Thermal resistance (K/W)		300
*CTH	Thermal capacitance (J/W)		3e-9
*ATH	Temperature coefficient of the thermal resistance		0
TYPE	Device Type		N

Operating Points

The following letter pairs have been used to identify the NL branches: ds, bs, bd, gs, gb and gd. Here g, d, s and b correspond to the gate, drain, source and substrate terminals, respectively. These are used to identify branch related operating point information, i.e., branch voltages, currents, etc.

Parameter	Description
cgg (Capacitance)	$\frac{\partial Q_g}{\partial V_g}$, intrinsic charge.
cgb (Capacitance)	$\frac{\partial Q_g}{\partial V_b}$, intrinsic charge.
cgd (Capacitance)	$\frac{\partial Q_g}{\partial V_d}$, intrinsic charge.
cgs (Capacitance)	$\frac{\partial Q_g}{\partial V_s}$, intrinsic charge.
cbg (Capacitance)	$\frac{\partial Q_b}{\partial V_g}$, intrinsic charge.
cbb (Capacitance)	$\frac{\partial Q_b}{\partial V_b}$, intrinsic charge.
cbd (Capacitance)	$\frac{\partial Q_b}{\partial V_d}$, intrinsic charge.
cbs (Capacitance)	$\frac{\partial Q_b}{\partial V_s}$, intrinsic charge.

Parameter	Description
cdg (Capacitance)	$\frac{\partial Q_d}{\partial V_g}$, intrinsic charge.
cdb (Capacitance)	$\frac{\partial Q_d}{\partial V_b}$, intrinsic charge.
cdd (Capacitance)	$\frac{\partial Q_d}{\partial V_d}$, intrinsic charge.
cds (Capacitance)	$\frac{\partial Q_d}{\partial V_s}$, intrinsic charge.
csg (Capacitance)	$\frac{\partial Q_s}{\partial V_g}$, intrinsic charge.
csb (Capacitance)	$\frac{\partial Q_s}{\partial V_b}$, intrinsic charge.
csd (Capacitance)	$\frac{\partial Q_s}{\partial V_d}$, intrinsic charge.
css (Capacitance)	$\frac{\partial Q_s}{\partial V_s}$, intrinsic charge.
Qg	Gate terminal charge
Qb	Bulk terminal charge
Qd	Drain terminal charge
Qs	Source terminal charge
pwr (Power)	Dissipated power
temp (Temperature)	Simulation temperature
u	Transistor gain
Rout (Resistance)	Small signal output resistance
Vearly (Voltage)	Equivalent Early voltage
gm (Conductance)	Transconductance
gmb (Conductance)	Bulk transconductance
gds (Conductance)	Output conductance
Iohm (Current)	Drain-source current excluding velocity saturation
Ihc (Current)	Critical current for velocity saturation
Vp (Voltage)	Channel pinch-off voltage

.

Implementation Details

The TYPE parameter controls whether the device is N, or, P channel. The extraction and simulation temperatures are controlled using the TNOM and TEMP parameters, respectively; instead of TR and DTA. The SELFT parameter controls whether self-heating modeling is enabled or not. Parameter default and truncation values are identical to those employed by Philips. Operating point information also follows Philip's prescription closely.

Layout

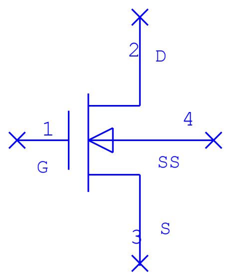
This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

References

[1]http://www.semiconductors.philips.com/acrobat_download/other/philipsmodels/m3100.pdf

Philips Level 903 Geometric N MOSFET Model: MOS9

Symbol



Summary

MOS9 is an implementation of the Philips MOS model 9 level 903 device. The model is intended for the simulation of circuit behavior with emphasis on analogue applications. It provides an excellent description of the electrical characteristics in all relevant regions of transistor operation. The underlying model is based on the gradual-channel approximation with a number of first order corrections for small size effects, which allows to describe transistor behavior over a wide range of channel lengths and widths using just one parameter set. The consistency is maintained by using the same carrier-density and electrical-field expression in the calculation of all model quantities.

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	M1
*TNOM	Parameter extraction temperature	DegC	21
*TEMP	Device temperature	DegC	21
*NFLAG	Noise flag		Noise On
*LER	Effective channel length of reference transistor	um	1.1
*WER	Effective channel width of reference transistor	um	20
*LVAR	Difference between actual and programmed poly-silicon gate lengths	um	-0.22
*LAP	Effective channel length reduction per side	um	0.1
*WVAR	Difference between actual and programmed filed-oxide opening	um	-0.025
*WOT	Effective channel width reduction per side	um	0
*VTOR	Threshold at zero back-bias of reference transistor	V	.73
*STVTO	Threshold voltage temperature of coefficient (VK^-1)		-0.0012
*SLVTO	Threshold voltage length dependence first coefficient (Vm)		-1.35e-7
*SL2VTO	Threshold voltage length dependence second coefficient (Vm^2)		0
*SL3VTO	Threshold voltage length dependence third coefficient	V	
*SWVTO	Threshold voltage width dependence coefficient (Vm)		1.3e-7
*KOR	Low back-bias body factor (V^0.5)		0.65

Name	Description	Unit Type	Default
*SLKO	Low back-bias body factor length dependence first coefficient (V^0.5m)		-1.3e-7
*SL2KO	Low back-bias body factor length dependence second coefficient (V^0.5m^2)		
*SWKO	Low back-bias body factor width dependence coefficient (V^0.5m)		2e-9
*KR	High back-bias body factor (V^0.5)		0.11
*SLK	High back-bias body factor length dependence first coefficient (V^0.5m)		-2.8e-7
*SLK2	High back-bias body factor length dependence second coefficient (V^0.5m^2)		
*SWK	High back-bias body factor width dependence coefficient (V^0.5m)		2.75e-7
*PHIBR	Strong inversion surface potential	V	0.65
*VSBXR	Transition voltage for the dual-k factor model	V	0.66
*SLVSBX	VSBXR length dependence coefficient (Vm)		0
*SWVSBX	VSBXR width dependence coefficient (Vm)		-6.75e-7
*BETSQ	Gain factor for the infinite square transistor (AV^.2)		8.3e-5
*ETABET	Gain factor temperature dependence exponent		1.6
*LP1	Characteristic length of first profile	um	
*FBET1	Relative mobility decrease due to first profile		
*LP2	Characteristic length of second profile	um	
*FBET2	Relative mobility decrease due to second profile		
*THE1R	Gate induced field mobility reduction coefficient (V^-1)		0.19
*STTHE1R	THE1R temperature dependence coefficient (V^-1K^-1)		0
*SLTHE1R	THE1R length dependence coefficient (V^-1m)		1.4e-7
*STLTHE1	THE1R length dependence temperature coefficient (V^-1mK^-1)		0
*GTHE1	THE1R scaling rule selector 0:old, 1:new		
*SWTHE1	THE1R width dependence coefficient (V^-1m)		-5.8e-8
*WDOG	Characteristic drawn gate width, below which dogboning appears	um	0
*FTHE1	THE1R geometry dependence coefficient		
*THE2R	Back-bias mobility reduction coefficient (V^-.5)		0.012
*STTHE2R	THE2R temperature dependence coefficient (V^-.5K^-1)		0
*SLTHE2R	THE2R length dependence coefficient (V^-0.5m)		-3.3e-8
*STLTHE2	THE2R length dependence temperature coefficient (V^-0.5mK^-0.5)		0
*SWTHE2	THE2R width dependence coefficient (V^-0.5m)		3e-8

Name	Description	Unit Type	Default
*THE3R	Lateral field mobility reduction coefficient (V^-1)		0.145
*STTHE3R	THE3R temperature dependence coefficient (V^-1k^-1)		-0.00066
*SLTHE3R	THE3R length dependence coefficient (V^-1m)		1.85e-7
*STLTHE3	THE3R length dependence temperature coefficient (V^-1mK^-1)		-6.2e-10
*SWTHE3	THE3R width dependency coefficient (V^-1m)		2e-8
*GAM1R	Drain induced threshold shift for large gate drive coefficient (V^(1-nDS))		0.145
*SLGAM1	GAM1R length dependence coefficient (V^(1-nDS)m)		1.6e-7
*SWGAM1	GAM1R width dependence coefficient (V^(1-nDS)m)		-1e-8
*ETADSR	GAM1R Vds dependence exponent		0.6
*ALPR	Channel length modulation factor		0.003
*ETAALP	ALPR length dependence exponent		0.15
*SLALP	ALPR length dependence coefficient (m^nALPHA)		-0.00565
*SWALP	ALPR width dependence coefficient		0.00167
*VPR	Channel length modulation characteristic voltage	V	0.34
*GAMOOR	Zero gate drive drain induced threshold shift coefficient		0.018
*SLGAMOO	GAMOOR first length dependence coefficient (m^2)		2e-14
*SL2GAMOO	GAMOOR second length dependence coefficient		
*ETAGAMR	GAMOOR back-bias dependence exponent		2
*MOR	Subthreshold slope factor		0.5
*STMO	MOR temperature dependence coefficient (K^1)		0
*SLMO	MOR length dependence coefficient (M^0.5)		0.00028
*ETAMR	MOR back-bias dependence exponent		2
*ZET1R	ZET1R length dependence exponent		0.42
*ETAZET	ZET1R length dependence exponent		0.5
*SLZET1	ZET1R length dependence coefficient (m^nGAMMA)		-0.39
*VSBTR	MOR and GAMOOR Vsb dependence limiting voltage	V	2.1
*SLVSBT	VSVTR length dependence coefficient (Vm)		-4.4e-6
*A1R	Weak-avalanche current factor		6
*STA1	A1R temperature dependence coefficient (K^-1)		0
*SLA1	A1R length dependence coefficient	um	1.3
*SWA1	A1R width dependence coefficient	um	3
*A2R	Weak-avalanche current exponent	V	38
*SLA2	A2R length dependence coefficient (Vm)		1e-6
*SWA2	A2R width dependence coefficient (Vm)		2e-6
*A3R	Weak-avalanche drain-source voltage factor		0.65

Name	Description	Unit Type	Default
*SLA3	A3R length dependence coefficient	um	-0.55
*SWA3	A1R width dependence coefficient	um	0
*TOX	Gate-oxide layer thickness	um	0.025
*COL	Gate overlap capacitance per unit channel width (Fm^-1)		3.2e-10
*NTR	Thermal noise coefficient (J)		2.44e-20
*NFMOD	Flicker noise model selector. 0:old, 1:new		0
*NFR	Flicker noise coefficient (V^2)		7e-11
*NFAR	First flicker noise coefficient (V^-1m^-4)		
*NFBR	Second Flicker noise coefficient (V^-1m^-2)		
*NFCR	Third Flicker noise coefficient (V^-1)		
*L	Lay-out drawn channel length	um	1.5
*W	Lay-out drawn channel width	um	20
*MULT	Number of devices in parallel		1
*THE3MOD	Flag for THE3R clipping		

** indicates a secondary parameter*

The default and restrictions on the range of parameter values is in full compliance with the model definition.

Operating Point Information

The following letter pairs have been used to identify the NL branches: ds, bs, bd, gs and gd

Parameter	Description
pwr (W)	Power.
gm (S)	Transconductance $\frac{\partial \text{ids}}{\partial \text{igs}}$.
gmb (S)	Bulk transconductance $\frac{\partial \text{ids}}{\partial \text{vbs}}$.
gds (S)	Output conductance $\frac{\partial \text{ids}}{\partial \text{vds}}$.
vgt1 (V)	Threshold voltage including backbias effects.
vgt2 (V)	Effective gate drive including backbias and drain effects.
vds1 (V)	Saturation limit.
vdss1 (V)	Saturation voltage at actual bias.
cgg (F)	$\frac{\partial Q_g}{\partial V_g}$
cgb (F)	$\frac{\partial Q_g}{\partial V_b}$

Parameter	Description
cgd (F)	$\frac{\partial Q_g}{\partial V_d}$
cgs (F)	$\frac{\partial Q_g}{\partial V_s}$
cbg (F)	$\frac{\partial Q_b}{\partial V_g}$
cbb (F)	$\frac{\partial Q_b}{\partial V_b}$
cbd (F)	$\frac{\partial Q_b}{\partial V_d}$
cbs (F)	$\frac{\partial Q_b}{\partial V_s}$
cdg (F)	$\frac{\partial Q_d}{\partial V_g}$
cdb (F)	$\frac{\partial Q_d}{\partial V_b}$
cdd (F)	$\frac{\partial Q_d}{\partial V_d}$
cds (F)	$\frac{\partial Q_d}{\partial V_s}$
csg (F)	$\frac{\partial Q_s}{\partial V_g}$
csb (F)	$\frac{\partial Q_s}{\partial V_b}$
csd (F)	$\frac{\partial Q_s}{\partial V_d}$
css (F)	$\frac{\partial Q_s}{\partial V_s}$

. Here g,d,s abd b correspond to the gate, drain, source and substrate terminals, respectively. These are used to identify branch related operating point information, i.e., branch voltages, currents, etc.

Implementation Details

This model is mapped into HSPICE as a NMOS M-device with parameters LEVEL set to 50, which corresponds to the Philips MOS model 9 level 902 device, whose parameter set is a subset of the one of the level 903 device. Changing any of the parameters not supported by HSPICE into something different than its default value will result in a warning being issued by the simulator. For the complete set of equations the interested reader is referred to Ref. [1] [11–209]

Layout

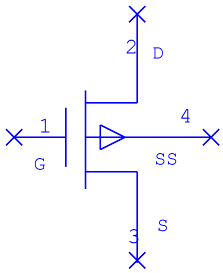
This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

References

[1] R.M.D.A. Velghe, D.B.M. Klaassen, and F.M. Klaassen, "Unclassified Report NL-UR 003/94".

Philips Level 903 Geometric P MOSFET Model: MOS9P

Symbol



Implementation Details

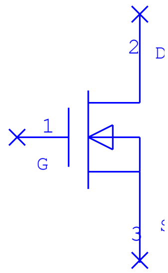
This is a P-channel implementation of the Philips MOS model 9 level 903 device. It is identical to MOS9, except for being of P-type. As a result all diode and source currents and charges are reversed. See MOS9 for information on its parameters and implementation.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

(Obsolete) SPICE Level 1 N MOSFET Model: MOSN1

Symbol



Summary

This element is OBSOLETE and is replaced by the SPICE Level 1 3-Terminal N MOSFET (MOSN1A) element.

Implementation Details

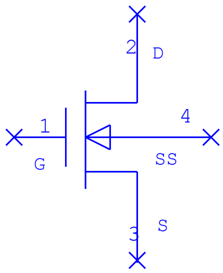
This model is identical to MOSN1_4 except that the body (substrate) terminal is connected to the source internally. See the MOSN1_4 documentation for parameters and implementation.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

(Obsolete) SPICE Level 1 4-Terminal N MOSFET Model: MOSN1_4

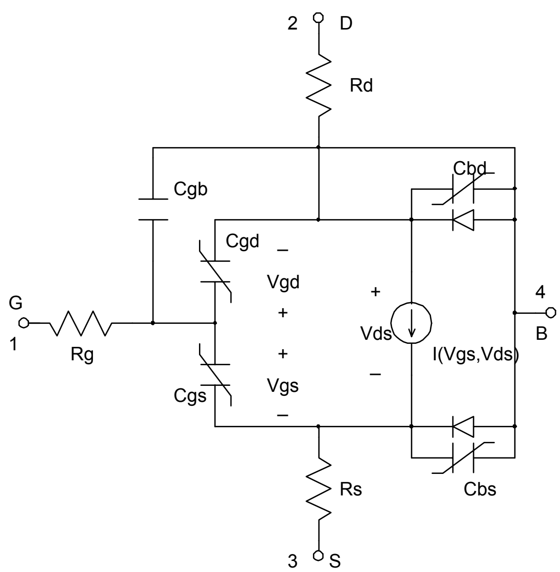
Symbol



Summary

This element is OBSOLETE and is replaced by the SPICE Level 1 N MOSFET (MOSN1_4A) element.

Equivalent Circuit



Parameters

Name	Description	Unit Type	Default
ID	Device ID	Text	M1N1
*VTO	Threshold voltage at zero bias	Voltage	1 V
*KP	Transconductance parameter		$2*10^{-5}$
*GAMMA	Bulk-effect parameter		0
*PHI	Surface potential	Voltage	0.6 V
*LAMBDA	Channel-length modulation parameter		0

Name	Description	Unit Type	Default
*RD	Drain resistance	Resistance	0.001 ohm
*RS	Source resistance	Resistance	0.001 ohm
*CBD	Bulk-drain capacitance at zero bias	Capacitance	0 pF
*CBS	Bulk-source capacitance at zero bias	Capacitance	0 pF
*PB	Bulk junction built-in voltage	Voltage	0.8 V
*IS	Bulk junction current parameter	Current	10^{-11} mA
*CGSO	Gate-source overlap capacitance per meter of gate width	Capacitance	0 pF
*CGDO	Gate-drain overlap capacitance per meter of gate width	Capacitance	0 pF
*CGBO	Gate-bulk overlap capacitance per meter of gate length	Capacitance	0 pF
*RSH	Drain & source diffusion sheet resistance	Resistance	0.001 ohm
*CJ	Bulk bottom capacitance per square meter at zero bias	Capacitance	0 pF
*MJ	Bulk junction grading parameter		0.5
*CJSW	Bulk junction periphery capacitance per meter at zero bias	Capacitance	0 pF
*MJSW	Bulk junction periphery capacitance grading parameter		0.33
*JS	Bulk junction saturation current per square meter	Current	0 mA
*TOX	Oxide thickness	Length	0.1 um
*NSUB	Bulk doping density (cm^-3)		0
*NSS	Surface state density (cm^-2)		0
*TPG	Gate material flag; metal=0, same=-1, diff=+1		0
*LD	Lateral diffusion length	Length	0 um
*UO	Mobility (cm^2/v*s); must not be zero if TOX=0		600
*FC	Depletion capacitance linearization parameter		0.5
*TYPE	(Not implemented)	Text	NMOS
*RG	Gate resistance	Resistance	0.001 ohm
*RDS	Drain-source resistance	Resistance	1e+06 ohm
*N	Bulk PN ideality factor		1
*TT	Bulk PN storage time	Time	0 ns
*TNOM	Temperature	Temperature	27 DegC
*ER	Substrate dielectric constant		11.7
*EOX	Oxide dielectric constant		3.78
*NI	Substrate intrinsic carrier concentration (cm^-3)		$1.45*10^{10}$
*KF	Drain flicker (1/f) noise coefficient		0
*AF	Drain flicker (1/f) noise exponential term		1
*FFE	Drain flicker (1/f) noise frequency exponent		1
L	Gate length	Length	100 um
W	Gate width	Length	100 um
AS	Source diffusion bottom area m^2		0

Name	Description	Unit Type	Default
AD	Drain diffusion bottom area, m^2		0
PS	Source diffusion perimeter	Length	0 um
PD	Drain diffusion perimeter	Length	0 um
NRD	Number of squares in drain area (for RD)		0
NRS	Number of squares in source area (for RS)		0

** indicates a secondary parameter*

Implementation Details

Parameter Entry

As with the SPICE MOSFET model, parameters can be entered either as electrical parameters or as process parameters. TOX is used as a flag to determine whether process or electrical parameters are used. If process parameters are entered, the program calculates macroscopic parameters; the following rules determine how the parameters are used:

If TOX is zero, VTO, KP, GAMMA, and PHI are used directly, and C_{ox} is calculated from

$$C_{ox} = \frac{KP}{UO}$$

In this case, if UO is zero, $C_{ox} = 0$.

If TOX is not zero, C_{ox} is calculated. Then,

if KP=0, IP is computed;

if NSUB > 0 and PHI = 0, PHI is computed;

if NSUB > 0 and GAMMA = 0, GAMMA is computed;

if NSUB > 0 and VTO = 0, VTO is computed.

I/V Characteristic

The equations for the I/V characteristic are identical to those in SPICE. See [1] for complete information.

Gate Capacitance

The SPICE Level 1 MOSFET model uses the Meyer capacitance model. This model has many problems; see MOSN3, and its references, for more information on its limitations. Instead we have implemented a simple division scheme to divide the gate-to-channel charge between the gate-to-source and gate-to-drain capacitances.

The gate-source capacitance in normal operation, Q_{gs}, is

$$Q_{gs} = 0.5C_{ox}(L - 2 \cdot \mathrm{LD})\mathrm{WV}_{gs}(1 + \tanh(V_{ds}/(\mathrm{VTR})))$$

where V_{gs} is the gate-to-source voltage, V_{ds} is the drain-to-source voltage, and Cox is the oxide capacitance. The hyperbolic tangent provides a smooth transition from normal to inverse operation. The gate-to-drain charge is

$$Q_{gd} = 0.5C_{ox}(L - 2 \cdot \mathrm{LD})\mathrm{WV}_{gd}(1 + \tanh(-V_{ds}/(\mathrm{VTR})))$$

where V_{gd} is the gate-to-drain voltage. Note that the same expression can be used for both of these charges.

Drain-Bulk and Source-Bulk Diodes

The diodes are modeled by PNIV elements. See PNIV for the equations. The sidewall capacitance and bottom capacitance of the drain and source diffusions are modeled separately by PNCAP and PNDCAP elements, respectively.

These parasitic capacitances are calculated exactly as in SPICE. See [1] and the documentation for the SDIODE element for complete information.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

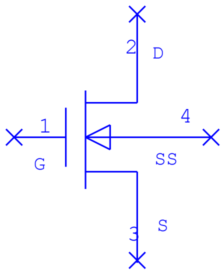
Reference

[1] P. Antognetti and G. Massobrio, Semiconductor Device Modeling with SPICE, New York: McGraw-Hill, 1988.

[2] D. P. Foty, MOSFET Modeling with SPICE: Principles and Practice, Prentice Hall, Upper Saddle River, NJ, USA, 1997.

SPICE Level 1 N MOSFET: MOSN1_4A

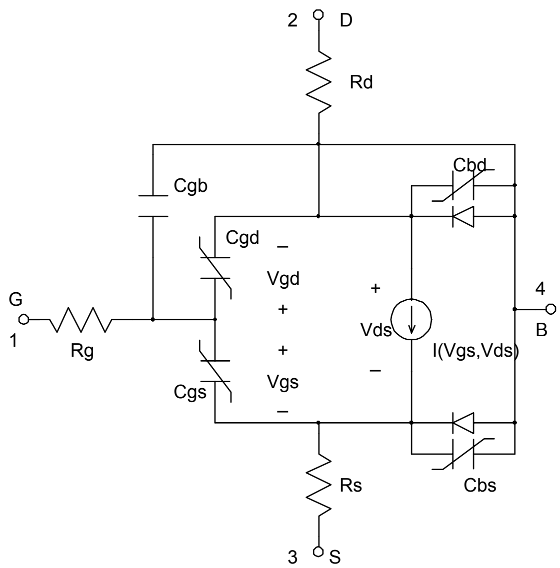
Symbol



Summary

This is an N-channel implementation of the SPICE2 MOST (Metal-Oxide-Semiconductor Transistor) LEVEL1 device. The original Meyer capacitance model has been replaced by BSIM1's charge conserving capacitance model. Further enhancements provide support for the most popular Area Calculation Methods, ACM=0 and ACM=2.

Equivalent Circuit



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	M1
*VTO	Threshold voltage at zero bias	V	
*KP	Transconductance Parameter		
*GAMMA	Bulk-effect parameter		

Name	Description	Unit Type	Default
*PHI	Surface potential	V	
*LAMBDA	Channel-length modulation parameter		0
*RD	Drain resistance	ohm	
*RS	Source resistance	ohm	
*CBD	Bulk-drain capacitance at zero bias	pF	
*CBS	Bulk-source capacitance at zero bias	pF	
*PB	Bulk junction built-in voltage	V	0.8
*IS	Bulk Junction current parameter		1e-14
*CGSO	Gate-source overlap capacitance per meter of gate width		0
*CGDO	Gate-drain overlap capacitance per meter of gate width		0
*CGBO	Gate-bulk overlap capacitance per meter of gate length		0
*RSH	Drain & source diffusion sheet resistance		
*CJ	Bulk bottom capacitance per square meter at zero bias		
*MJ	Bulk junction grading parameter		0.5
*CJSW	Bulk junction periphery capacitance per meter at zero bias		0
*MJSW	Bulk junction periphery capacitance grading parameter		0.33
*JS	Bulk junction saturation current per square meter		0
*TOX	Oxide thickness	um	0.1
*NSUB	Bulk doping density (cm^-3)		1e15
*NSS	Surface state density (cm^-2)		0
*TPG	Gate material flag; metal=0, same =-1, diff=+1		1
*LD	Lateral diffusion length	um	0
*U0	Mobility (cm^2/v*s)		
*FC	Depletion capacitance linearization parameter		0
*N	Bulk PN ideality factor		1
*NS	Sidewall PN ideality factor		1
*TNOM	Parameter extraction temperature	DegC	26.85
*TEMP	Device Temperature	DegC	26.85
*KF	Drain flicker (1/F) noise coefficient		0
*AF	Drain flicker (1/f) noise exponential term		1
*FFE	Drain flicker (1/f) noise frequency exponent		1
*L	Gate length	um	100
*W	Gate Width	um	100
*AS	Source diffusion bottom area m^2		
*AD	Drain diffusion bottom area, m^2		
*PS	Source diffusion perimeter	um	
*PD	Drain diffusion perimeter	um	

Name	Description	Unit Type	Default
*NRD	Number of squares in drain area (for RD)		1
*NRS	Number of squares in source area (for RS)		1
*XPART	Drain/source channel charge partition in saturation for charge models		1
*NFLAG	Noise Model		Noise On
*CAPMOD	Capacitance model selector		BSIM1
*MULT	Number of devices in parallel		1
*ACM	Area calculation method		0
*HDIF	Length of heavily doped diffusion	um	0
*WMLT	Width diffusion layer shrink reduction factor		1
*LDIF	Length of lightly doped diffusion	0	um
*SCALM	Model scaling factor		1
*XL	Length of variation due to masking and etching	um	0
*DEL	Channel length reduction on both sides	um	0
*XW	Width variation due to masking and etching	um	0
*LMLT	Length diffusion layer shrink reduction factor		1
*WD	Lateral diffusion length		1
*JSSW	Bulk junction saturation current per meter		0

** indicates a secondary parameter*

Operating Point Information

The following letter pairs have been used to identify the NL branches: ds, bs, bd, gs and gd

Parameter	Description
gm (S)	Transconductance $\frac{\partial \text{ids}}{\partial \text{igs}}$.
gmbs (S)	Bulk transconductance $\frac{\partial \text{ids}}{\partial \text{vbs}}$.
gds (S)	Output conductance $\frac{\partial \text{ids}}{\partial \text{vds}}$.
vth (V)	Threshold voltage.
vdsat (V)	Saturation voltage.
cgg (F)	$\frac{\partial Q_g}{\partial V_g}$, intrinsic charge.
cgb (F)	$\frac{\partial Q_g}{\partial V_b}$, intrinsic charge.

Parameter	Description
cgd (F)	$\frac{\partial Q_g}{\partial V_d}$, intrinsic charge.
cgs (F)	$\frac{\partial Q_g}{\partial V_s}$, intrinsic charge.
cbg (F)	$\frac{\partial Q_b}{\partial V_g}$, intrinsic charge.
cbb (F)	$\frac{\partial Q_b}{\partial V_b}$, intrinsic charge.
cbd (F)	$\frac{\partial Q_b}{\partial V_d}$, intrinsic charge.
cbs (F)	$\frac{\partial Q_b}{\partial V_s}$, intrinsic charge.
cdg (F)	$\frac{\partial Q_d}{\partial V_g}$, intrinsic charge.
cdb (F)	$\frac{\partial Q_d}{\partial V_b}$, intrinsic charge.
cdd (F)	$\frac{\partial Q_d}{\partial V_d}$, intrinsic charge.
cds (F)	$\frac{\partial Q_d}{\partial V_s}$, intrinsic charge.
csg (F)	$\frac{\partial Q_s}{\partial V_g}$, intrinsic charge.
csb (F)	$\frac{\partial Q_s}{\partial V_b}$, intrinsic charge.
csd (F)	$\frac{\partial Q_s}{\partial V_d}$, intrinsic charge.
css (F)	$\frac{\partial Q_s}{\partial V_s}$, intrinsic charge.
cjd (F)	Drain-bulk junction capacitance.
cjs (F)	Source-bulk junction capacitance.
pwr (W)	Power at operating point.
gmoverid (1/V)	Gm/Ids.
betaeff	Effective beta

Parameter	Description
Isub(A)	Substrate current
ro (ohm)	Common-source output resistance.
Cgs (F)	$-\frac{\partial Q_g}{\partial V_s}$
Cgd (F)	$-\frac{\partial Q_g}{\partial V_d}$
Cgtot	$\frac{\partial Q_g}{\partial V_g}$
Cbtot	$\frac{\partial Q_b}{\partial V_b}$
Cdtot	$\frac{\partial Q_d}{\partial V_d}$
Cstot	$\frac{\partial Q_s}{\partial V_s}$
ft (Hz)	Unity small-signal current-gain frequency.

Here g, d, s, and b correspond to the intrinsic gate, drain, source and substrate nodes, respectively. These are used to identify branch related operating point information, i.e., branch voltages, currents, etc.

Parameter Details

As with SPICE2 MOST models, parameters can be entered either as electrical parameters or as process parameters, the former used in computing the prior whenever possible. MOSN1_4A accepts empty strings for most parameters.

ACM. The ACM (Area Calculation Method) parameter is used to select the type of diode model used for the MOSFET bulk diodes. In ACM=0 the junction area and perimeter are used in determining the capacitance and reverse saturation current. ACM=2 uses HPSICE-style MOS diodes. This method supports both lightly-doped and heavily doped diffusions, which are described by means of parameters LD, LDIF and HDIF.

CAPMOD. This parameter is used to select the capacitance model. Two models are available: BSIM1 and Meyer's capacitance models. Meyer's capacitance model is only supported in HSPICE simulations.

Implementation Details

This model is mapped into HSPICE as a NMOS M-device with parameters LEVEL and CAPMOD set to 1 and 13, respectively. CAPMOD equal to 13 invokes the BSIM1 charge model.

Equations

The MOS1_4A intrinsic model combines LEVEL1 MOST static and BSIM1 Large-Signal Charge models. The equations for both models are described in detail in Ref. [1]. The small signal model is obtained by linearization of the static and charge large signal model.

Drain-Bulk and Source-Bulk JUNCTIONS

The current and charge contributions from the sidewall and bottom surfaces of each of these junctions are considered separately. These parasitic-diode capacitances and currents are calculated exactly as in SPICE2. See [1] for complete information.

Layout

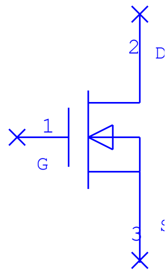
This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

References

[1] P. Antognetti and G. Massobrio, Semiconductor Device Modeling with SPICE, New York: McGraw-Hill, 1988.

SPICE Level 1 3-Terminal N MOSFET: MOSN1A

Symbol



Implementation Details

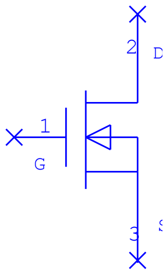
This is an implementation of the SPICE 2 MOST (Metal-Oxide-Semiconductor Transistor) LEVEL1 device, where the body (substrate) and source terminals have been connected internally. The original Meyer capacitance model has been replaced by BSIM1's charge conserving capacitance model. Further enhancements provide support for the most popular Area Calculation Methods, ACM=0 and ACM=2. See MOSN1_4A for information on its parameters and implementation.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

(Obsolete) SPICE Level 2 N MOSFET Model: MOSN2

Symbol



Summary

There is no replacement for this OBSOLETE element.

Parameters

Name	Description	Unit Type	Default
ID	Device ID	Text	M2P3
L	Gate length	um	100
W	Gate width	um	100
AS	Source diffusion bottom area m^2		0
AD	Drain diffusion bottom area, m^2		0
PS	Source diffusion perimeter	um	0
PD	Drain diffusion perimeter	um	0
NRD	Number of squares in drain area (for RD)		0
NRS	Number of squares in source are (for RS)		0
Delta	Narrow width factor to adjust threshold		0
VMAX	Maximum carrier drift velocity		0
NEFF	Total channel charge coefficient		1
XJ	Metallurgical junction depth		1e-6
UCRIT	Critical electric field for mobility		1000
UTRA	Transverse field coefficient		0
UEXP	Exponential coefficient for mobility		0
NFLAG	Noise Model		Spice Model
*VTO	Threshold voltage at zero bias	V	-0.13388
*KP	Transconductance parameter		2e-5
*GAMMA	Bulk-effect parameter		0.5273
*PHI	Surface potential	V	0.57984
*LAMBDA	Channel-length modulation parameter		0

Name	Description	Unit Type	Default
*RD	Drain resistance	ohm	0.001
*RS	Source resistance	ohm	0.001
*CBD	Bulk-drain capacitance at zero bias	pF	0
*CBS	Bulk-source capacitance at zero bias	pF	0
*PB	Bulk junction built-in voltage	V	0.8
*IS	Bulk junction current parameter	mA	1e-11
*CGSO	Gate-source overlap capacitance per meter of gate width	pF	0
*CGDO	Gate-drain overlap capacitance per meter of gate width	pF	0
*CGBO	Gate-bulk overlap capacitance per meter of gate width	pF	0
*RSH	Drain&Source diffusion sheet resistance	ohm	0.001
*CJ	Bulk bottom capacitance per square meter at zero bias	pF	0
*MJ	Bulk junction grading parameter		0.5
*CJSW	Bulk junction periphery capacitance per meter at zero bias	pF	0
*MJSW	Bulk junction periphery capacitance grading parameter		0.33
*JS	Bulk junction saturation current per square meter	mA	0
*TOX	Oxide thickness	um	0.1
*NSUB	Bulk doping density (cm^-3)		1e15
*NSS	Surface state density (cm^-2)		0
*TPG	Gate material flag; metal=0, same=-1, diff-+1		1
*LD	Lateral diffusion length	um	0
*WD	Lateral diffusion length along width	um	0
*UO	Mobility (cm^2/v*s); must not be zero if TOX=0		250
*FC	Depletion capacitance linearization parameter		0.5
*TYPE	(Not implemented)		PMOS
*RG	Gate resistance	ohm	0.001
*RDS	Drain-source resistance	ohm	100000000
*N	Bulk PN ideality factor		1
*TT	Bulk PN storage time	ns	0
*TNOM	Parameter extraction temperature	Degc	26.85
*TEMP	Device temperature	DegC	26.85
*ER	Substrate dielectric constant		11.7
*EQX	Oxide dielectric constant		3.9
*NI	Substrate intrinsic carrier concentration (cm^-3)		14500000000
*KF	Drain flicker (1/f) noise coefficient		0
*AF	Drain flicker (1/f) noise exponential term		1
*FFE	Drain flicker (1/f) noise frequency exponent		1

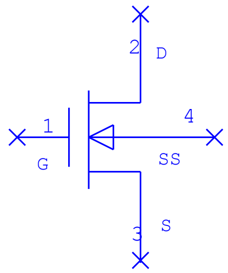
** indicates a secondary parameter*

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

(Obsolete) SPICE Level 2 4-Terminal N MOSFET Model: MOSN2_4

Symbol



Summary

There is no replacement for this OBSOLETE element. This is the 4-port version of the MOSN2 model.

Parameters

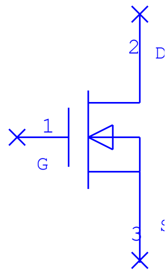
Parameters are identical to those of MOSN2.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

(Obsolete) SPICE Level 3 N MOSFET Model: MOSN3

Symbol



Summary

This element is OBSOLETE and is replaced by the SPICE Level 3 3-Terminal N MOSFET (MOSN3A) element.

Implementation Details

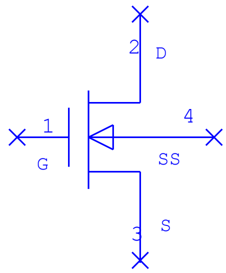
This model is identical to MOSN3_4 except that the body (substrate) terminal is connected to the source internally. See the MOSN3_4 documentation for parameters and implementation.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

(Obsolete) SPICE Level 3 4-Terminal N MOSFET Model: MOSN3_4

Symbol



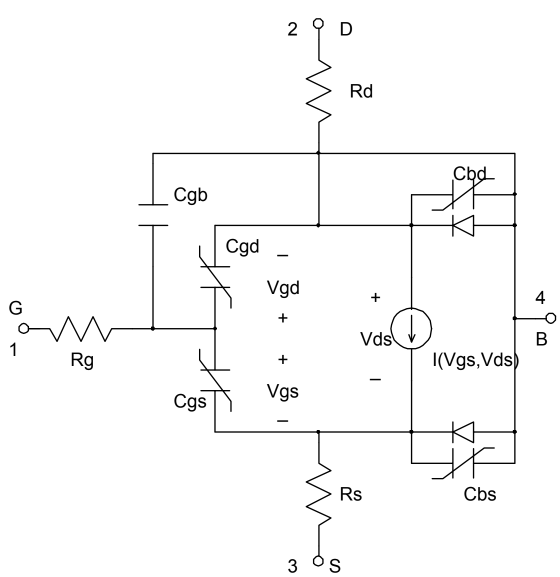
Summary

This element is OBSOLETE and is replaced by the SPICE Level 3 N MOSFET (MOSN3_4A) element. MOSN3_4 is an implementation of the Level 3 MOSFET model developed at the University of California, Berkeley, and used in SPICE2 and SPICE3. The I/V equations are identical to those in SPICE, but the capacitance functions have been modified to eliminate serious discontinuities and implementation problems in harmonic-balance analysis.

MOSN3 and MOSN3_4 differ only in the substrate (bulk) terminal, terminal 4 in the following figure. MOSN3 is a three-terminal model.

The MOSN3and MOSN3_4 models are symmetrical.

Equivalent Circuit



Parameters

Name	Description	Unit Type	Default
ID	Device ID	Text	M3N1
VTO	Threshold voltage at zero bias	Voltage	0
KP	Transconductance parameter		0
GAMMA	Bulk-effect parameter		0
PHI	Surface potential	Voltage	0
RD	Drain resistance	Resistance	0
RS	Source resistance	Resistance	0
CBD	Bulk-drain capacitance at zero bias	Capacitance	0
CBS	Bulk-source capacitance at zero bias	Capacitance	0
PB	Bulk junction built-in voltage	Voltage	0.8
IS	Bulk junction current parameter	Current	10^{-14}
CGSO	Gate-source overlap capacitance per meter of gate width	Capacitance	0
CGDO	Gate-drain overlap capacitance per meter of gate width	Capacitance	0
CGBO	Gate-bulk overlap capacitance per meter of gate length	Capacitance	0
RSH	Drain & source diffusion sheet resistance	Resistance	0
CJ	Bulk bottom capacitance per square meter at zero bias	Capacitance	0
MJ	Bulk junction grading parameter		0.5
CJSW	Bulk junction periphery capacitance per meter at zero bias	Capacitance	0
MJSW	Bulk junction periphery capacitance grading parameter		0.33
JS	Bulk junction saturation current per square meter	Current	0
TOX	Oxide thickness		0.1 um
NSUB	Bulk doping density (cm^-3); must be provided		0
NSS	Surface state density (cm^-2)		0
NFS	Fast surface state density (cm^-2)		0.0
TPG	Gate material flag; aluminum=0, same=-1, diff=+1		+1
XJ	Metallurgical junction depth	Length	0
LD	Lateral diffusion (L)	Length	0
WD	Lateral diffusion (W)	Length	0
UO	Mobility (cm^2/V s); must not be zero if TOX=0	cm^2/Vs	600
VMAX	Max drift velocity		0
KF	Flicker noise (1/f noise) coefficient)		0
AF	Flicker noise (1/f noise) exponential term		0
FFE	Flicker noise (1/f noise) frequency exponent		1.0
FC	Depletion capacitance linearization parameter		0.5
XQC	Coefficient of channel charge share		0.5

Name	Description	Unit Type	Default
DELTA	Width effect on threshold V		0
THETA	Mobility modulation		0
ETA	Static feedback		0
KAPPA	Saturation field factor		0.2
TYPE	(Not implemented)		NMOS
RG	Gate resistance	Resistance	0
RDS	Drain-source resistance	Resistance	10^6
N	Bulk PN ideality factor		1
TT	Bulk PN storage time	Time	0
TNOM	Temperature	Temperature	27 C
ER	Substrate dielectric constant		11.7
EOX	Oxide dielectric constant		3.78
NI	Substrate intrinsic carrier concentration (cm^{-3})	cm^{-3}	$1.45*10^{10}$
L	Gate length	Length	100 um
W	Gate width	Length	100 um
AS	Source diffusion bottom area m^2		0
AD	Drain diffusion bottom area, m^2		0
PS	Source diffusion perimeter	Length	0
PD	Drain diffusion perimeter	Length	0
NRD	Number of squares in drain area (for RD)		0
NRS	Number of squares in source area (for RS)		0

Parameter Details

The Level 3 MOSFET model follows the SPICE implementation. Differences include the following:

1. Some of the model's default values, listed in SPICE manuals, are not really valid. In particular, the defaults for KP and PHI are calculated; they are not fixed quantities as the documentation sometimes states.
2. Units in the MW Office implementation of the model may not be the same as SPICE or other implementations. If model parameters are entered from a library, all parameters are in MKS units and there should be no problems.
3. The Berkeley SPICE3 Level 3 MOSFET model does not include the parameter WD, so it does not modify the gate width; i.e., Weff = W in all cases. We include WD for compatibility with other implementations that do use WD. The latter use the conventional correction, Weff = W - 2 WD.
4. The Berkeley SPICE3 implementation uses the Meyer capacitance model. This model is severely flawed. It does not conserve charge, has discontinuities that affect both accuracy and convergence, and its integration in SPICE3 is theoretically incorrect. For these reasons, many simulators have implemented the Ward-Dutton model instead of the Meyer model. We also have chosen to do so. This may introduce differences between SPICE3 and Microwave Office calculations. For further information on the Ward-Dutton model and on problems of the Meyer model, see Ref. [4] [11–231]. The Ward-Dutton model is also described in [3] [11–231], which is the standard reference in the SPICE3 MOSFET model, but the Ward-Dutton model apparently is not implemented in Berkeley SPICE3.

5. The Level 3 MOSFET model is an old one, developed before there was a sophisticated understanding of the limitations of such models, and suffers from many problems. Most of these are associated with the capacitance model. In some cases, negative gate capacitances can exist, and there can be singularities, at some bias settings. The I/V model has discontinuous derivatives, which also can cause convergence difficulties under some conditions, especially when the device switches between the linear and saturation regions.

Because of these problems, the various implementations of the Level 3 MOSFET model have been customized and are often somewhat different. Wherever discrepancies have occurred, we have attempted to match the HSPICE implementation. If the HSPICE implementation is unclear, we use the Berkeley SPICE3 functions.

Scaling

Equations for the MOSN3 model are too extensive to repeat here. See the references for further information.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

References

Refs. [1] [11–231] and [2] [11–231] have multiple errors, but may be useful for general information. Ref. [3] [11–231] is the standard for this model, but it implies that the Ward-Dutton capacitance model is used. Ref. [4] [11–231] presents a good treatment of the model, but it contains typographical errors.

[1] P. Antognetti and G. Maasobrio, Semiconductor Device Modeling with SPICE, McGraw-Hill, New York, 1988.

[2] D. A. Divekar, FET Modeling for Circuit Simulation, Kluwer, Boston, 1988.

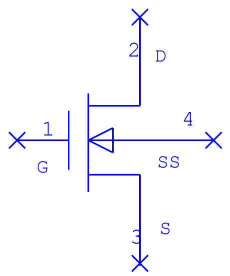
[3] A. Vladimirescu and S. Liu, "The Simulation of MOS Integrated Circuits Using SPICE2," Memorandum no. UCB M80/7, Electronics Research Laboratory, University of California, Berkeley, Feb., 1980.

[4] D. P. Foty, MOSFET Modeling with SPICE: Principles and Practice, Prentice Hall, Upper Saddle River, NJ, USA, 1997.

[5] S. Liu, "A Unified CAD Model for MOSFETs," University of California, Berkeley, Memorandum no. UCB/ERL M81/31, 20 May, 1981.

SPICE Level 3 N MOSFET: MOSN3_4A

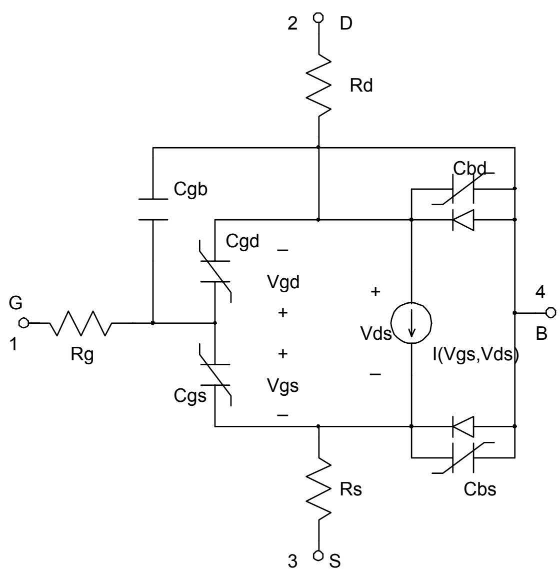
Symbol



Summary

This is a N-channel implementation of the SPICE2 MOST (Metal-Oxide-Semiconductor Transistor) LEVEL3 device. The original Meyer capacitance model has been replaced by BSIM1's charge conserving capacitance model. Further enhancements provide support for the most popular Area Calculation Methods, ACM=0 and ACM=2, and impact ionization.

Equivalent Circuit



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	M1
*VTO	Threshold voltage at zero bias	V	
*KP	Transconductance Parameter		

Name	Description	Unit Type	Default
*GAMMA	Bulk-effect parameter		
*PHI	Surface potential	V	
*RD	Drain resistance	ohm	
*RS	Source resistance	ohm	
*CBD	Bulk-drain capacitance at zero bias	pF	
*CBS	Bulk-source capacitance at zero bias	pF	
*PB	Bulk junction built-in voltage	V	0.8
*IS	Bulk Junction current parameter		1e-14
*CGSO	Gate-source overlap capacitance per meter of gate width		0
*CGDO	Gate-drain overlap capacitance per meter of gate width		0
*CGBO	Gate-bulk overlap capacitance per meter of gate length		0
*RSH	Drain & source diffusion sheet resistance		
*CJ	Bulk bottom capacitance per square meter at zero bias		
*MJ	Bulk junction grading parameter		0.5
*CJSW	Bulk junction periphery capacitance per meter at zero bias		0
*MJSW	Bulk junction periphery capacitance grading parameter		0.33
*JS	Bulk junction saturation current per square meter		0
*TOX	Oxide thickness	um	0.1
*NSUB	Bulk doping density (cm^-3)		1e15
*NSS	Surface state density (cm^-2)		0
*TPG	Gate material flag; metal=0, same =-1, diff=+1		1
*LD	Lateral diffusion length	um	0
*U0	Mobility (cm^2/v*s)		
*FC	Depletion capacitance linearization parameter		0
*N	Bulk PN ideality factor		1
*NS	Sidewall PN ideality factor		
*TNOM	Parameter extraction temperature	DegC	26.85
*TEMP	Device Temperature	DegC	26.85
*KF	Drain flicker (1/F) noise coefficient		0
*AF	Drain flicker (1/f) noise exponential term		1
*FFE	Drain flicker (1/f) noise frequency exponent		1
*L	Gate length	um	100
*W	Gate Width	um	100
*AS	Source diffusion bottom area m^2		
*AD	Drain diffusion bottom area, m^2		
*PS	Source diffusion perimeter	um	
*PD	Drain diffusion perimeter	um	

Name	Description	Unit Type	Default
*NRD	Number of squares in drain area (for RD)		1
*NRS	Number of squares in source area (for RS)		1
*XPART	Drain/source channel charge partition in saturation for charge models		1
*NFLAG	Noise Model		Noise On
*CAPMOD	Capacitance model selector		BSIM1
*MULT	Number of devices in parallel		1
*ACM	Area calculation method		0
*ETA	Static feedback (V^-1)		0
*DELTA	Width effect on threshold voltage		0
*NFS	Surface fast state density (cm^-2 V^-1)		0
*THETA	Mobility modulation (V^-1)		0
*VMAX	Max drift velocity (cm s^-1)		0
*KAPPA	Saturation field factor (V^-1)		0.2
*XJ	Metallurgical junction depth	um	0
*HDIF	Length of heavily doped diffusion	um	0
*METO	Metal overlap in fringing field (um)		0
*WD	Lateral diffusion length	um	0
*SCALM	Model scaling factor		1
*LMLT	Length diffusion layer shrink reduction factor		1
*XL	Length variation due to masking and etching	um	0
*DEL	Channel length reduction on both sides	um	0
*WMLT	Width diffusion layer shrink reduction factor		1
*XW	Width variation due to masking and etching	um	0
*ALPHA	Impact ionization current coefficient (V^-1)		0
*LALPHA	Alpha length sensitivity (umV^-1)		0
*WALPHA	Alpha width sensitivity (umV^-1)		0
*VCR	Critical voltage	V	0
*LVCR	VCR length sensitivity *umV)		0
*WVCR	VCR width sensitivity (umV)		0
*IIRAT	Portion of impact ionization current associated with the source		0
*RDC	Drain resistance due to contact resistance	ohm	0
*RSC	Source resistance due to contact resistance	ohm	0
*LDIF	Length of lightly doped diffusion	um	0
*JSSW	Bulk junction saturation current per meter		0

** indicates a secondary parameter*

Operating Point Information

The following letter pairs have been used to identify the NL branches: ds, bs, bd, gs and gd

Parameter	Description
gm (S)	Transconductance $\frac{\partial \text{ids}}{\partial \text{igs}}$.
gmbs (S)	Bulk transconductance $\frac{\partial \text{ids}}{\partial \text{vbs}}$.
gds (S)	Output conductance $\frac{\partial \text{ids}}{\partial \text{vds}}$.
vth (V)	Threshold voltage.
vdsat (V)	Saturation voltage.
cgg (F)	$\frac{\partial Q_g}{\partial V_g}$, intrinsic charge.
cgb (F)	$\frac{\partial Q_g}{\partial V_b}$, intrinsic charge.
cgd (F)	$\frac{\partial Q_g}{\partial V_d}$, intrinsic charge.
cgs (F)	$\frac{\partial Q_g}{\partial V_s}$, intrinsic charge.
cbg (F)	$\frac{\partial Q_b}{\partial V_g}$, intrinsic charge.
cbb (F)	$\frac{\partial Q_b}{\partial V_b}$, intrinsic charge.
cbd (F)	$\frac{\partial Q_b}{\partial V_d}$, intrinsic charge.
cbs (F)	$\frac{\partial Q_b}{\partial V_s}$, intrinsic charge.
cdg (F)	$\frac{\partial Q_d}{\partial V_g}$, intrinsic charge.
cdb (F)	$\frac{\partial Q_d}{\partial V_b}$, intrinsic charge.
cdd (F)	$\frac{\partial Q_d}{\partial V_d}$, intrinsic charge.

Parameter	Description
cds (F)	$\frac{\partial Q_d}{\partial V_s}$, intrinsic charge.
csg (F)	$\frac{\partial Q_s}{\partial V_g}$, intrinsic charge.
csb (F)	$\frac{\partial Q_s}{\partial V_b}$, intrinsic charge.
csd (F)	$\frac{\partial Q_s}{\partial V_d}$, intrinsic charge.
css (F)	$\frac{\partial Q_s}{\partial V_s}$, intrinsic charge.
cjd (F)	Drain-bulk junction capacitance.
cjs (F)	Source-bulk junction capacitance.
pwr (W)	Power at operating point.
gmoverid (1/V)	Gm/Ids.
betaeff	Effective beta
Isub(A)	Substrate current
ro (ohm)	Common-source output resistance.
Cgs (F)	$-\frac{\partial Q_g}{\partial V_s}$
Cgd (F)	$-\frac{\partial Q_g}{\partial V_d}$
Cgtot	$\frac{\partial Q_g}{\partial V_g}$
Cbtot	$\frac{\partial Q_b}{\partial V_b}$
Cdtot	$\frac{\partial Q_d}{\partial V_d}$
Cstot	$\frac{\partial Q_s}{\partial V_s}$
ft (Hz)	Unity small-signal current-gain frequency.

. Here g,d,s abd b correspond to the intrinsic gate, drain, source and substrate nodes, respectively. These are used to identify branch related operating point information, i.e., branch voltages, currents, etc.

Parameter Details

As with SPICE2 MOST models, parameters can be entered either as electrical parameters or as process parameters, the former used in computing the prior whenever possible. MOSN3_4A accepts empty strings for most parameters.

ACM. The ACM (Area Calculation Method) parameter is used to select the type of diode model used for the MOSFET bulk diodes. In ACM=0 the junction area and perimeter are used in determining the capacitance and reverse saturation current. ACM=2 uses HPSICE-style MOS diodes. This method supports both lightly-doped and heavily doped diffusions, which are described by means of parameters LD, LDIF and HDIF.

CAPMOD. This parameter is used to select the capacitance model. Two models are available: BSIM1 and Meyer's capacitance models. Meyer's capacitance model is only supported in HSPICE simulations.

Implementation Details

This model is mapped into HSPICE as a NMOS M-device with parameters LEVEL and CAPMOD set to 1 and 13, respectively. CAPMOD equal to 13 invokes the BSIM1 charge model.

Equations

The MOS3_4A intrinsic model combines LEVEL3 MOST static and BSIM1 Large-Signal Charge models. The equations for both models are described in detail in Ref. [1]. The small signal model is obtained by linearization of the static and charge large signal model.

Drain-Bulk and Source-Bulk JUNCTIONS

The current and charge contributions from the sidewall and bottom surfaces of each of these junctions are considered separately. These parasitic-diode capacitances and currents are calculated exactly as in SPICE2. See [1] for complete information.

Layout

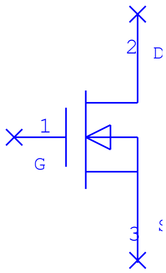
This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

References

[1] P. Antognetti and G. Massobrio, Semiconductor Device Modeling with SPICE, New York: McGraw-Hill, 1988.

SPICE Level 3 3-Terminal N MOSFET: MOSN3A

Symbol



Implementation Details

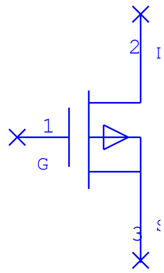
This is an implementation of the SPICE2 MOST (Metal-Oxide-Semiconductor Transistor) LEVEL3 device, where the body (substrate) and source terminals have been connected internally. The original Meyer capacitance model has been replaced by BSIM1's charge conserving capacitance model. Further enhancements provide support for the most popular Area Calculation Methods, ACM=0 and ACM=2, and impact ionization. See MOSN3_4A for information on its parameters and implementation.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

(Obsolete) SPICE Level 1 P MOSFET Model: MOSP1

Symbol



Summary

This element is OBSOLETE and is replaced by the SPICE Level 1 3-Terminal P MOSFET (MOSP1A) element.

Implementation Details

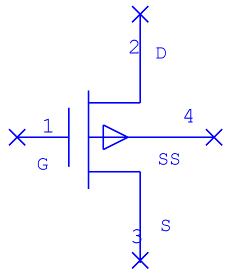
This is a P-channel implementation of the level-1 MOSFET, and is identical to MOSN1 except for the P channel. See MOSN1_4 for parameters and implementation details.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

(Obsolete) SPICE Level 1 4-Terminal P MOSFET Model: MOSP1_4

Symbol



Summary

This element is OBSOLETE and is replaced by the SPICE Level 1 P MOSFET (MOSP1_4A) element.

Implementation Details

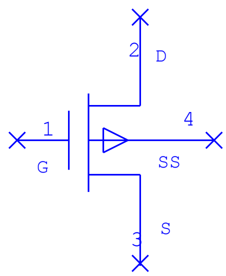
This is a P-channel implementation of the 4-terminal level-1 MOSFET, and is identical to MOSN1_4 except for the P channel. See MOSN1_4 for information on its parameters and implementation.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

SPICE Level 1 P MOSFET: MOSP1_4A

Symbol



Implementation Details

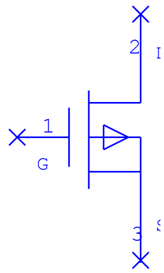
This is a P-channel implementation of the SPICE2 MOST (Metal-Oxide-Semiconductor Transistor) LEVEL1 device. The original Meyer capacitance model has been replaced by BSIM1's charge conserving capacitance model. Further enhancements provide support for the most popular Area Calculation Methods, ACM=0 and ACM=2. It is identical to MOSN1_4A, except for being of P-type. As a result all diode and source currents and charges are reversed. See MOSN1_4A for information on its parameters and implementation.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

SPICE Level 1 3-Terminal P MOSFET: MOSP1A

Symbol



Implementation Details

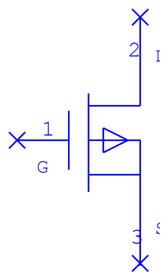
This is a P-channel implementation of the SPICE2 MOST (Metal-Oxide-Semiconductor Transistor) LEVEL1 device, where the body (substrate) and source terminals have been connected internally. The original Meyer capacitance model has been replaced by BSIM1's charge conserving capacitance model. Further enhancements provide support for the most popular Area Calculation Methods, ACM=0 and ACM=2. It is identical to MOSN1A, except for being of P-type. As a result all diode and source currents and charges are reversed. See MOSN1_4A for information on its parameters and implementation.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

(Obsolete) SPICE Level 2 P MOSFET Model: MOSP2

Symbol



Summary

There is no replacement for this OBSOLETE element.

Parameters

Name	Description	Unit Type	Default
ID	Device ID	Text	M2P3
L	Gate length	um	100
W	Gate width	um	100
AS	Source diffusion bottom area m^2		0
AD	Drain diffusion bottom area, m^2		0
PS	Source diffusion perimeter	um	0
PD	Drain diffusion perimeter	um	0
NRD	Number of squares in drain area (for RD)		0
NRS	Number of squares in source are (for RS)		0
Delta	Narrow width factor to adjust threshold		0
VMAX	Maximum carrier drift velocity		0
NEFF	Total channel charge coefficient		1
XJ	Metallurgical junction depth		1e-6
UCRIT	Critical electric field for mobility		1000
UTRA	Transverse field coefficient		0
UEXP	Exponential coefficient for mobility		0
NFLAG	Noise Model		Spice Model
*VTO	Threshold voltage at zero bias	V	-0.13388
*KP	Transconductance parameter		2e-5
*GAMMA	Bulk-effect parameter		0.5273
*PHI	Surface potential	V	0.57984
*LAMBDA	Channel-length modulation parameter		0

Name	Description	Unit Type	Default
*RD	Drain resistance	ohm	0.001
*RS	Source resistance	ohm	0.001
*CBD	Bulk-drain capacitance at zero bias	pF	0
*CBS	Bulk-source capacitance at zero bias	pF	0
*PB	Bulk junction built-in voltage	V	0.8
*IS	Bulk junction current parameter	mA	1e-11
*CGSO	Gate-source overlap capacitance per meter of gate width	pF	0
*CGDO	Gate-drain overlap capacitance per meter of gate width	pF	0
*CGBO	Gate-bulk overlap capacitance per meter of gate width	pF	0
*RSH	Drain&Source diffusion sheet resistance	ohm	0.001
*CJ	Bulk bottom capacitance per square meter at zero bias	pF	0
*MJ	Bulk junction grading parameter		0.5
*CJSW	Bulk junction periphery capacitance per meter at zero bias	pF	0
*MJSW	Bulk junction periphery capacitance grading parameter		0.33
*JS	Bulk junction saturation current per square meter	mA	0
*TOX	Oxide thickness	um	0.1
*NSUB	Bulk doping density (cm^-3)		1e15
*NSS	Surface state density (cm^-2)		0
*TPG	Gate material flag; metal=0, same=-1, diff-+1		1
*LD	Lateral diffusion length	um	0
*WD	Lateral diffusion length along width	um	0
*UO	Mobility (cm^2/v*s); must not be zero if TOX=0		250
*FC	Depletion capacitance linearization parameter		0.5
*TYPE	(Not implemented)		PMOS
*RG	Gate resistance	ohm	0.001
*RDS	Drain-source resistance	ohm	100000000
*N	Bulk PN ideality factor		1
*TT	Bulk PN storage time	ns	0
*TNOM	Parameter extraction temperature	Degc	26.85
*TEMP	Device temperature	DegC	26.85
*ER	Substrate dielectric constant		11.7
*EQX	Oxide dielectric constant		3.9
*NI	Substrate intrinsic carrier concentration (cm^-3)		14500000000
*KF	Drain flicker (1/f) noise coefficient		0
*AF	Drain flicker (1/f) noise exponential term		1
*FFE	Drain flicker (1/f) noise frequency exponent		1

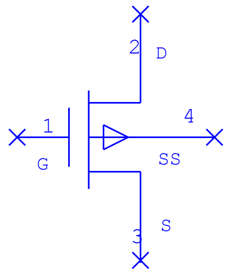
** indicates a secondary parameter*

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

(Obsolete) SPICE Level 2 4-Terminal P MOSFET Model: MOSP2_4

Symbol



Summary

There is no replacement for this OBSOLETE element. This is the 4-port version of the MOSP2 model.

Parameters

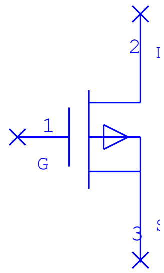
Parameters are identical to those of MOSP2.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

(Obsolete) Spice Level 3 P MOSFET Model: MOSP3

Symbol



Summary

This element is OBSOLETE and is replaced by the SPICE Level 3 3-Terminal P MOSFET (MOSP3A) element.

Implementation Details

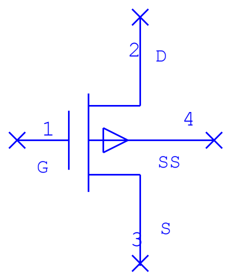
This is a P-channel implementation of the level-3 MOSFET, and is identical to MOSN3 except for the P channel. See MOSN3_4 for information on its parameters and implementation.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

(Obsolete) Spice Level 3 4-terminal P MOSFET Model: MOSP3_4

Symbol



Summary

This element is OBSOLETE and is replaced by the SPICE Level 3 P MOSFET (MOSP3_4A) element.

Implementation Details

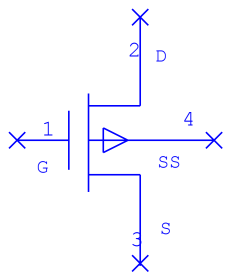
This is a P-channel implementation of the 4-terminal level-3 MOSFET, and is identical to MOSN3_4 except for the P channel. See MOSN3_4for information on its parameters and implementation.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

SPICE Level 3 P MOSFET: MOSP3_4A

Symbol



Implementation Details

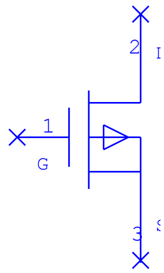
This is a P-channel implementation of the SPICE2 MOST (Metal-Oxide-Semiconductor Transistor) LEVEL3 device. The original Meyer capacitance model has been replaced by BSIM1's charge conserving capacitance model. Further enhancements provide support for the most popular Area Calculation Methods, ACM=0 and ACM=2, and impact ionization. It is identical to MOSN3_4A, except for being of P-type. As a result all diode and source currents and charges are reversed. See MOSN3_4A for information on its parameters and implementation.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

SPICE Level 3 3-Terminal P MOSFET: MOSP3A

Symbol



Implementation Details

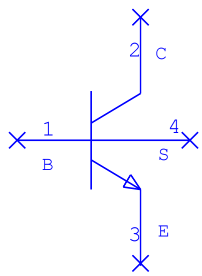
This is a P-channel implementation of the SPICE2 MOST (Metal-Oxide-Semiconductor Transistor) LEVEL3 device, where the body (substrate) and source terminals have been connected internally. The original Meyer capacitance model has been replaced by BSIM1's charge conserving capacitance model. Further enhancements provide support for the most popular Area Calculation Methods, ACM=0 and ACM=2, and impact ionization. It is identical to MOSN3A, except for being of P-type. As a result all diode and source currents and charges are reversed. See MOSN3_4A for information on its parameters and implementation.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

(Obsolete) Mextram 503 NPN BJT Model: MXTR503N

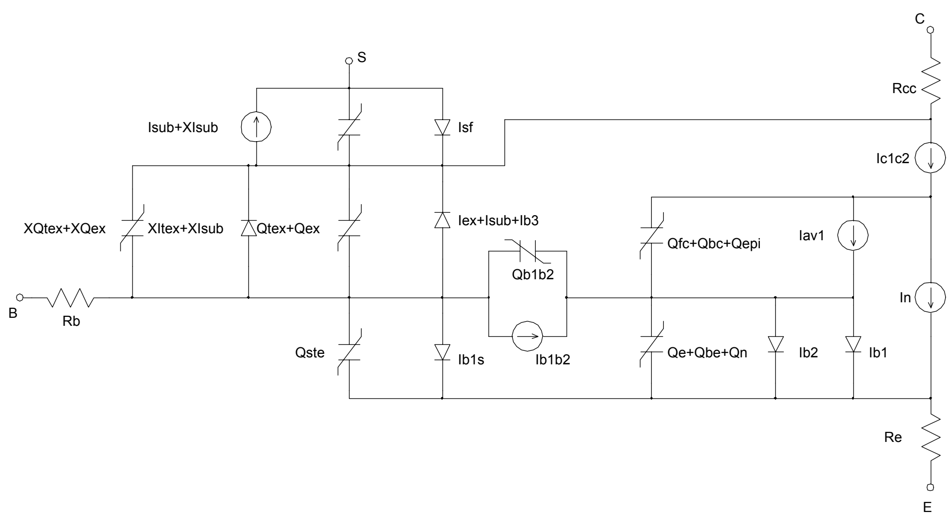
Symbol



Summary

There is no replacement for this OBSOLETE element. The Mextram model was developed as a general-purpose BJT model for use throughout the electronics industry. Mextram was developed at Philips, and the Microwave Office model is a full implementation of Philips' Mextram 503 model. Mextram models a large variety of effects that are not included in the SPICE Gummel-Poon model; these include weak avalanche effects, hard- and quasi-saturation, hot-carrier effects in the epilayer, and substrate current. It includes improved temperature scaling, but not self-heating.

Equivalent Circuit



Parameters

Name	Description	Unit Type	Default
ID	Element ID		MX1
*NPN	NPN (flag, information only)		1
*PNP	PNP (flag, information only)		0
*Release	Release (information only)		503

Name	Description	Unit Type	Default
EXMOD	Flag for extended modeling of reverse current gain		1.0
*EXPHI	(Not implemented)		0.0
EXAVL	Flag for extended modeling of avalanche currents		0.0
*IS	Collector-to-emitter saturation current	Amperes	5e-14
*BF	Forward current gain		140.0
*XIBI	Fraction of ideal base current from sidewall		0.0
*IBF	Saturation current of the non-ideal forward base current	Amperes	2e-14A
*VLF	Cross-over voltage of the non-ideal forward base current	Voltage	0.5V
*IK	High-injection knee current	Amperes	15.0e-3A
*BRI	Ideal reverse current gain		16.0
*IBR	Saturation current of the non-ideal base current	Amperes	8e-15A
*VLR	Cross-over voltage of the non-ideal reverse base current	Voltage	0.5V
*XEXT	Parameter dependency of VBC1		0.5
*QBO	Base charge at zero bias		1.2e12
*ETA	Factor of the built-in field base)		4.0
*AVL	Weak avalanche parameter		50.0
*EFI	Electric field intercept		0.7
*IHC	Critical current for hot carriers	A	3.0e-3
*RCC	Constant part of the collector resistance	Resistance	25.0 ohm
*RCV	Resistance of the unmodulated epilayer	Resistance	750.0 ohm
*SCRCV	Space charge resistance of the epilayer	Resistance	1000.0 ohm
*SFH	Current spreading factor epilayer	Resistance	0.6 ohm
*RBC	Constant part of the base resistance	Resistance	50.0 ohm
*RBV	Variable part of the base resistance at zero bias	Resistance	100.0 ohm
*RE	Emitter series resistance	Resistance	2.0 ohm
*TAUNE	Minimum delay time of neutral and emitter charge	Conductance	0.3e-9s
*MTAU	Non-ideality factor of the neutral and emitter charge		1.18
*CJE	Zero bias BE depletion capacitance	Faraday	0.25e-12F
*VDE	BE built-in voltage	Voltage	0.9V
*PE	BE grading coefficient		0.33
*XCJE	Reaction of BE capacitance to the sidewall		0.5
*CJC	Zero bias BC depletion capacitance	Faraday	0.13e-12F
*VDC	BC built-in voltage	Voltage	0.6V
*PC	BC grading coefficient		0.4
*XP	Constant part of CJC		0.2
*MC	Collector current modulation coefficient		0.5
*XCJC	Fraction of BE capacitance under emitter		0.1

Name	Description	Unit Type	Default
*TREF	Reference (extraction) temperature	Celsius	25°C
*DTA	Device temperature rise above ambient in degrees C	Celsius	0.0°C
*TAMB	Ambient (baseplate) temperature	Celsius	25°C
*VGE	Emitter Bandgap	Voltage	1.01V
*VGB	Base Bandgap	Voltage	1.18V
*VGC	Collector Bandgap	Voltage	1.205V
*VGJ	EB Bandgap	Voltage	1.1V
*VI	Ionization voltage base dope	Voltage	0.4V
*NA	Maximum base dope (per cm^3)		3e17
*ER	VLF and VLR temp coefficient		0.002
*AB	Base resistance temp coefficient		1.35
*AEPI	Epilayer temp coefficient		2.15
*AEX	Extrinsic base temp coefficient		1.0
*AC	Buried layer temp coefficient		0.4
*KF	Base current 1/f noise		2e-16
*KFN	Nonideal base 1/f noise		2e-16
*AF	1/f noise exponent		1.0
*ISS	base-substrate saturation current	Amperes	6.0e-16A
*IKS	Knee current of the substrate	Amperes	0.005e-3A
*CJS	Zero-voltage substrate capacitance	Faraday	1.0e-12F
*VDS	CS built-in voltage	Voltage	0.5V
*PS	CS grading coefficient		0.33
*VGS	Substrate Bandgap	Voltage	1.15V
*AS	=AC for closed buried layer; =AEPI for open buried layer.		2.15
*NFLAG	Noise flag; 1=ON, 0=OFF		0
MULT	Number of devices in parallel		1

** indicates a secondary parameter*

Parameter Details

Parameter default values are those given in Philips' documentation [2] [11–254] and are generally respected throughout the user community. Those defaults, however, are representative of RF or Microwave devices.

Parameter Restrictions and Recommendations

The range of many of the model parameters is restricted. In spite of extensive error-trapping in the Microwave Office implementation, it is possible that some parameter errors may not be trapped. See [2] [11–254] for details on such restrictions.

Implementation Details

This model is mapped into HSPICE as a NPN G-device with parameters LEVEL and VERS set to 6 and 503, respectively. The complete set of equations is too complex to be listed here. The user should consult the references for specific information. Ref. [1] [11–254] contains technical background information and theory of the model, and the documents available from Philips' web site [2] [11–254] define the model in detail.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Recommendations for Use

Mextram is an advanced model. It should be used only where the need for accuracy justifies its complexity, and where parameter libraries are available. The Gummel-Poon model, GBJT, is recommended for simple applications.

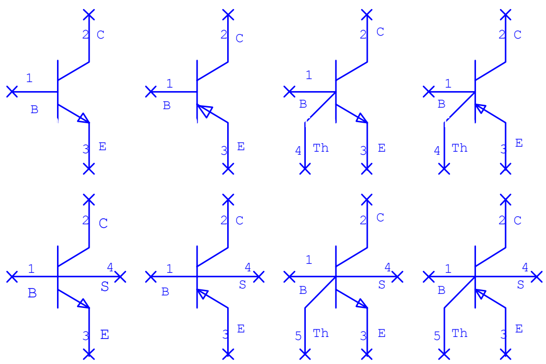
References

[1] H. C. de Graaf and F. M. Klaassen, Compact Transistor Modelling for Circuit Design, Springer-Verlag, Vienna, 1990.

[2] Philips Semiconductor web site, http://www.semiconductors.com/Philips_Models/documentation/mextram

(Obsolete) Mextram 504 BJT Model: MXTR504

Symbol



Summary

NI AWR's MXTR504 element is the successor of the MXTR504N and MXTR504P elements, and is replaced by MXTRM. MXTR504 is the based on the SiMKit 2.2. It implements the entire family of Philips Mextram level 504 devices, i.e., bjt504, bjtd504, bjt504t, and bjtd504t.

Philips Mextram model gives an excellent description of vertical bipolar transistors in all kinds of processes, amongst which are modern SiGe processes and robust HV processes. It is very efficient in modeling the lowly doped collector epilayer of a bipolar transistor where effects like velocity saturation, base widening, Kirk effect, and impact ionization play an important role. Effects resulting from the presence of Germanium in the base are also modeled. Furthermore, it contains a full description of the extrinsic regions of a transistor, including substrate current and capacitance. Mextram has formulations for temperature scaling and is easily scalable over geometry. Mextram, level 504 supports full self-heating. The best reference for the model equations is the Philips website.[1] [11–260]

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	MX1
*NPN	NPN (flag; information only)		1
*PNP	PNP (flag; information only)		0
*LEVEL	Release (information only)		504
*MULT	Number of devices in parallel		1.0
*TNOM	Reference (extraction) temperature	Temperature	25DegC
*TEMP	Ambient (baseplate) temperature	Temperature	_TEMP
*EXMOD	Flag for extended modeling of reverse current gain		1
*EXPHI	Flag for distributed high frequency effects in transient		1
*EXAVL	Flag for extended modeling of avalanche currents		0
*NFLAG	Noise model		1
*IS	Collector-to-emitter saturation current	Current	22.0e-18A

Name	Description	Unit Type	Default
*IK	High-injection knee current	Current	0.1A
*VER	Reverse Early voltage	Voltage	2.5V
*VEF	Forward Early voltage	Voltage	44.0V
*BF	Ideal Forward current gain		215
*IBF	Saturation current of nonideal forward base current	Current	2.7e-15A
*MLF	Non-ideality factor of the non-ideal forward base current		2.0
*XIBI	Fraction of ideal base current form sidewall		0.0
*BRI	Ideal reverse current gain		7
*IBR	Saturation current of nonideal reverse base current	Current	1.0e-15A
*VLR	Crossover voltage of nonideal reverse base current	Voltage	0.2V
*XEXT	Part of Iex, Qex, Qtex and Isub depending on Vbc1 instead of Vb1c1		0.63
*WAVL	Epilayer thickness used in weak-avalanche model	Length	1.1e-6m
*VAVL	Voltage determining curvature of avalanche current	Voltage	3.0V
*SFH	Current-spreading factor of avalanche model	Resistance	0.3 ohm
*RE	Emitter resistance	Resistance	5 ohm
*RBC	Constant part of the base resistance	Resistance	23 ohm
*RBV	Variable part of base resistance at zero bias	Resistance	18 ohm
*RCC	Constant part of the collector resistance	Resistance	12 ohm
*RCV	Resistance of the unmodulated epilayer	Resistance	150 ohm
*SCRCV	Space-charge resistance of the epilayer	Resistance	1250 ohm
*IHC	Critical current for hot carriers	Current	0.004A
*AXI	Smoothness parameter for the onset of quasi-saturation		0.3
*CJE	Zero-bias BE depletion capacitance	Capacitance	73e-15F
*VDE	BE built-in voltage	Voltage	0.95V
*PE	BE grading coefficient		0.4
*XCJE	Fraction of BE capacitance to the sidewall		0.4
*CBEO	Emitter-base overlap capacitance	Capacitance	0.0F
*CJC	Zero-bias BC depletion capacitance	Capacitance	78e-15F
*VDC	BC built-in voltage	Voltage	0.68V
*PC	BC grading coefficient		0.5
*XP	Constant part of CJC		0.35
*MC	Collector current modulation coefficient		0.50
*XCJC	Fraction of BE capacitance under emitter		0.032
*CBCO	Collector-base overlap capacitance	Capacitance	0.0F
*MTAU	Nonideality factor of neutral and emitter charge		1.0
*TAUE	Minimum transit time of stored emitter charge	Time	2.0e-12s

Name	Description	Unit Type	Default
*TAUB	Transit time of stored base charge	Time	4.2e-12s
*TEPI	Transit time of stored epilayer charge	Time	41.0e-12s
*TAUR	Transit time of reverse extrinsic stored base charge	Time	520.0e-12s
*DEG	Bandgap difference over the base		0.0
*XREC	Pre-factor of the recombination part of Ib1		0.0
*AQBO	Temperature coefficient of the zero base charge		0.3
*AE	Temperature coefficient of the resistivity of the emitter		0.0
*AB	Base resistance temp coefficient		1.0
*AEPI	Epilayer temp coefficient		2.50
*AEX	Extrinsic base temp coefficient		0.62
*AC	Buried layer temp coefficient		2.0
*DVGBF	Band-gap voltage difference of forward current gain	Voltage	50e-3V
*DVGBR	Band-gap voltage difference of reverse current gain	Voltage	45e-3V
*VGB	Base bandgap	Voltage	1.17V
*VGC	Collector bandgap	Voltage	1.18V
*VGJ	EB bandgap	Voltage	1.15V
*DVGTE	Band-gap voltage difference of emitter stored charge	Voltage	0.05V
*AF	1/f noise exponent		2
*KF	Base current 1/f noise		20e-12
*KFN	Nonideal base 1/f noise		20e-12
*ISS	Base-substrate saturation current	Current	48e-18A
*IKS	Knee current of the substrate per	Current	250.0e-6A
*CJS	Zero-voltage substrate capacitance	Capacitance	315e-15F
*VDS	CS built-in voltage	Voltage	0.62V
*PS	CS grading coefficient		0.34
*VGS	Substrate bandgap	Voltage	1.2V
*AS	AS=AC for closed buried layer; AS=AEPI for open buried layer		1.58
*RTH	Thermal resistance		300
*CTH	Thermal capacitance		3.0e-9
*TH_NODE	Thermal node visibility		0
SUBS_NODE	Substrate node visibility		1
TYPE	Device type		1

Implementation Details

The TYPE parameter controls whether the device is NPN, or PNP. The SUBS_NODE parameter is used to enable/disable substrate support. Finally, the SELFT parameter is used to enable/disable the self-heating modeling capabilities. The current setting of any of these parameters is immediately reflected by the device symbol. The extraction and simulation temperatures are controlled using parameters, TNOM and TEMP, respectively; instead of TR and DTA. Parameter default and truncation values are identical to those employed by Philips. Operating point information also follows Philip's prescription closely.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Operating Points

The following letter pairs have been used to identify the NL branches: bb1, cc1, ee1, sc1, b1b2, b2e1, b2c2, and, c1c2. These are used to identify branch related operating point information, i.e., branch voltages, currents, etc.

Parameter	Description
In (Current)	Main current
Ib1 (Current)	Ideal forward base current
SIb1 (Current)	Ideal side-wall base current
Ib2 (Current)	Non-ideal forward base current
Ib3 (Current)	Non-ideal reverse base current
Iex (Current)	Extrinsic reverse base current
xIex (Current)	Extrinsic reverse base current
Iavl (Current)	Avalanche current
Irbc (Current)	Current through constant base resistance
Ircc (Current)	Current through constant collector resistance
Ire (Current)	Current through emitter resistance
Iqs (Current)	Current at onset of quasi-saturation
Isub (Current)	Substrate current
XIsub (Current)	Substrate current
Isf (Current)	Substrate failure current
grcvy (Conductance)	Conductance of the epilayer current
grcvz (Conductance)	Conductance of the epilayer current
gx (Conductance)	Forward transconductance
gy (Conductance)	Reverse transconductance
gz (Conductance)	Reverse transconductance
gmux (Conductance)	Early effect on avalanche current limiting
gmuy (Conductance)	Conductance of avalanche current
gmuz (Conductance)	Conductance of avalanche current
sgpi (Conductance)	Conductance side-wall b-e junction
gpix (Conductance)	Conductance floor b-e junction
gpiy (Conductance)	Early-effect on recombination base current
gpiz (Conductance)	Early-effect on recombination base current

Parameter	Description
grbvx (Conductance)	Early-effect on base resistance
grbvy (Conductance)	Early-effect on base resistance
grbvz (Conductance)	Early-effect on base resistance
gm (Conductance)	Transconductance
gout (Conductance)	Output conductance
gmu (Conductance)	Feedback transconductance
gmuex (Conductance)	Conductance extrisic b-c junction
gs (Conductance)	Conductance parasitic pnp transistor
xgs (Conductance)	Conductance parasitic pnp transistor
gsf(Conductance)	
rbv (Resistance)	Base resistance
re (Resistance)	Emitter resistance
rbc (Resistance)	Constant base resistance
rcc (Resistance)	Constant collector resistance
rpi (Resistance)	
rb (Resistance)	Base resistance
scbe (Capacitance)	Capacitance sidewall b-e junction
cbex (Capacitance)	Capacitance floor b-e junction
cbey (Capacitance)	Early-effect on b-e diffusion charge
cbez (Capacitance)	Early-effect on b-e diffusion charge
cbcx (Capacitance)	Early-effect on b-c diffusion charge
cbcy (Capacitance)	Capacitance floor b-c junction
cbcz (Capacitance)	Capacitance floor b-c junction
cb1b2 (Capacitance)	Capacitance AC current crowding
cb1b2x (Capacitance)	Capacitance AC current crowding
cb1b2y (Capacitance)	Capacitance AC current crowding
cb1b2z (Capacitance)	Capacitance AC current crowding
cbe (Capacitance)	Base-emitter capacitance
cbc (Capacitance)	Base-collector capacitance
XCbcex (Capacitance)	Capacitance extrinsic b-c junction
Cbcex (Capacitance)	Capacitance extrinsic b-c junction
Cts (Capacitance)	Capacitance s-c junction
XiWepi	Thickness of injection layer
betaac	AC current amplification
betadc	DC current amplification
Temp (Temperature)	Simulation temperature
vb2c2star	Physical value of internal base-collector bias

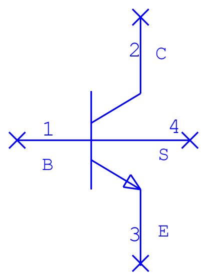
Parameter	Description
Q	
Qte	Base-emitter depletion charge
SQte	Sidewall base-emitter depletion charge
Qbe	Base-emitter diffusion charge
Qbc	Base-collector diffusion charge
Qtc	Base-collector depletion charge
Qepi	Epilayer diffusion charge
Qb1b2	AC current crowding charge
Qtex	Extrinsic base-collector depletion charge
XQtex	Extrinsic base-collector depletion charge
Qex	Extrinsic base-collector diffusion charge
XQex	Extrinsic base-collector diffusion charge
Qts	Collector-substrate depletion charge
ft (Frequency)	Cutoff frequency approximation
pwr (Power)	Dissipated power

References

[1] http://www.semiconductors.philips.com/Philips_Models/bipolar/mextram/

(Obsolete) Mextram 504 (Nonlinear BJT Model): MXTR504N

Symbol

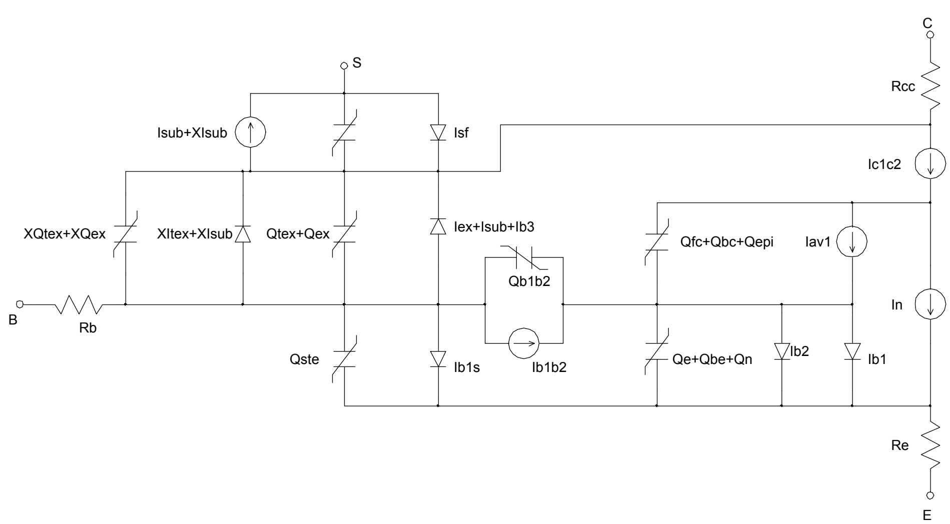


Summary

This element is OBSOLETE and is replaced by the Mextram 504 BJT Model (MXTR504) element. Mextram 504 is Philips' most recent vertical bipolar transistor model. Compared to Mextram 503, this model has improved descriptions of transistor characteristics and easier parameter extraction. The improved transistor characteristics are achieved by changing some of the model's formulations. The resulting equations are much smoother, such that first-order and higher derivatives are better. The complete set of equations and derivation are found in references [1] and [2], respectively. Parameter extraction is improved by decreasing parameter interdependence, without losing the physical basis of the model. As a result, the number of parameters has increased from 62 to 71.

MXTR504N is a full implementation of Philips' Mextram 504 model. It includes self-heating, which adds complexity to the model and increases the computation time. Due to the model's improved smoothness, tests show that MXTR504, with self-heating disabled, converges in general better than MXTR503.

Equivalent Circuit



Parameters

Name	Description	Unit Type	Default
ID	Element ID		MX1
*NPN	NPN (flag, information only)		1
*PNP	PNP (flag, information only)		0
*LEVEL	Release (Information only)		504
*EXMOD	Flag for extended modeling of reverse current gain		1.0
*EXPHI	Flag for distributed high frequency effects in transient simulations		1
*EXAVL	Flag for extended modeling of avalanche currents		0.0
*IS	Collector-to-emitter saturation current	Amperes	5e-017A
*BF	Forward current gain		215
*XIBI	Fraction of ideal base current from sidewall		0.0
*IBF	Saturation current of the non-ideal forward base current	Amperes	2e-14A
*VER	Reverse Early voltage	Voltage	2.5V
*VEF	Forward Early voltage	Voltage	44V
*MLF	Non-ideality factor of the non-ideal forward base current		2
*VLF	Cross-over voltage of the non-ideal forward base current	Voltage	0.5V
*IK	High-injection knee current	Amperes	15.0e-3A
*BRI	Ideal reverse current gain		7
*IBR	Saturation current of the non-ideal base current	Amperes	1e-12
*VLR	Cross-over voltage of the non-ideal reverse base current	Voltage	0.2
*XEXT	Parameter dependency of VBC1		0.63
*WAVL	Epilayer thickness used in weak-avalanche model		1.1
*VAVL	Voltage determining curvature of avalanche current	Voltage	3V
*QBO	Base charge at zero bias		1.2e12
*ETA	Factor of the built-in field base)		4.0
*AVL	Weak avalanche parameter		50.0
*EFI	Electric field intercept		0.7
*IHC	Critical current for hot carriers	Amperes	4
*AXI	Smoothness parameter for the onset of quasi-saturation		0.3
*RCC	Constant part of the collector resistance	Resistance	25.0 ohm
*RCV	Resistance of the unmodulated epilayer	Resistance	150
*SCRCV	Space charge resistance of the epilayer	Resistance	1250
*SFH	Current spreading factor epilayer	Resistance	0.6 ohm
*RBC	Constant part of the base resistance	Resistance	23
*RBV	Variable part of the base resistance at zero bias	Resistance	18

Name	Description	Unit Type	Default
*RE	Emitter series resistance	Resistance	2.0 ohm
*TAUNE	Minimum delay time of neutral and emitter charge	Conductance	0.3e-9s
*MTAU	Non-ideality factor of the neutral and emitter charge		1.18
*TAUE	Minimum transit time of stored emitter charge	Time	0.002ns
*TAUB	Transit time of stored base charge	Time	0.0042ns
*TEPI	Transit time of stored epilayer charge	Time	0.041ns
*TAUR	Transit time of reverse extrinsic stored base charge	Time	0.52ns
*CJE	Zero bias BE depletion capacitance	Faraday	0.25e-12F
*VDE	BE built-in voltage	Voltage	0.9V
*PE	BE grading coefficient		0.33
*XCJE	Reaction of BE capacitance to the sidewall		0.5
*CBE0	Emitter-base overlap capacitance		0
*CJC	Zero bias BC depletion capacitance	Faraday	0.13e-12F
*VDC	BC built-in voltage	Voltage	0.6V
*PC	BC grading coefficient		0.4
*XP	Constant part of CJC		0.2
*MC	Collector current modulation coefficient		0.5
*XCJC	Fraction of BE capacitance under emitter		0.1
*CBC0	Collector-base overlap capacitance		0
*TNOM	Reference (extraction) temperature	Celsius	25°C
*TEMP	Ambient (baseplate) temperature	Celsius	25°C
*VGE	Emitter Bandgap	Voltage	1.01V
*VGB	Base Bandgap	Voltage	1.18V
*VGC	Collector Bandgap	Voltage	1.205V
*VGJ	EB Bandgap	Voltage	1.1V
*VI	Ionization voltage base dope	Voltage	0.4V
*NA	Maximum base dope (per cm^3)		3e17
*ER	VLF and VLR temp coefficient		0.002
*DEG	Bandgap difference over the base		0
*XREC	Pr-factor of the recombination part of IB1		0
*AQB0	Temperature coefficient of the zero bias base charge		0.3
*AE	Temperature coefficient of the resistivity of the emitter		0
*AB	Base resistance temp coefficient		1.35
*AEPI	Epilayer temp coefficient		2.15
*AEX	Extrinsic base temp coefficient		1.0
*AC	Buried layer temp coefficient		4.0
*DVGBF	Band-gap voltage difference of forward current gain	Voltage	0.05V

Name	Description	Unit Type	Default
*DVGBR	Band-gap voltage difference of reverse current gain	Voltage	0.045V
*DVGTE	Band-gap voltage difference of emitter stored charge	Voltage	0.05V
*KF	Base current 1/f noise		2e-6
*KFN	Nonideal base 1/f noise		2e-6
*AF	1/f noise exponent		1.0
*ISS	base-substrate saturation current	Amperes	6.0e-16A
*IKS	Knee current of the substrate	Amperes	0.005e-3A
*CJS	Zero-voltage substrate capacitance	Faraday	1.0e-12F
*VDS	CS built-in voltage	Voltage	0.5V
*PS	CS grading coefficient		0.33
*RTH	Thermal resistance		300
*CTH	Thermal capacitance		3e-9
*VGS	Substrate Bandgap	Voltage	V
*AS	=AC for closed buried layer; =AEPI for open buried layer.		2.15
*NFLAG	Noise flag; 1=ON, 0=OFF		
*MULT	Number of devices in parallel		1

** indicates a secondary parameter*

Parameter Details

Parameter default values correspond to HSPICE default values and correspond to those listed as test parameters in Philips' documentation [1].

Parameter Restrictions and Recommendations

The range of many of the model parameters is restricted. In spite of extensive error-trapping in the Microwave Office implementation, some parameter errors may not be trapped. See [1] [11–265] for details on such restrictions.

Implementation Details

This model is mapped into HSPICE as a NPN G-device with parameters LEVEL and VERS set to 6 and 504, respectively. The complete set of equations is too complex to be listed here. Please consult the references for specific information. Ref. [1] [11–265] defines the model in detail and ref. [2] [11–265] contains the derivation of the model.

Recommendations for Use

Like Mextram 503, Mextram 504 is an advanced model and should be used only where the need for accuracy justifies its complexity, and where parameter libraries are available. Care must be exercised when self-heating is enabled.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

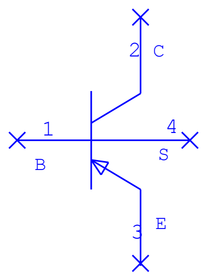
References

[1] http://www.semiconductors.philips.com/acrobat/other/philipsmodels/b504.pdf

[2] http://www-us.semiconductors.philips.com/acrobat/other/philipsmodels/newsflash/nlur2002806.pdf:

(Obsolete) Mextram 504 PNP BJT: MXTR504P

Symbol



Summary

This element is OBSOLETE and is replaced by the Mextram 504 BJT Model (MXTR504) element.

Equivalent Circuit

Same as MXTR504N, except that all diode and current and charge sources are reversed.

Parameters

Parameters are identical to those of MXTR504N.

Implementation Details

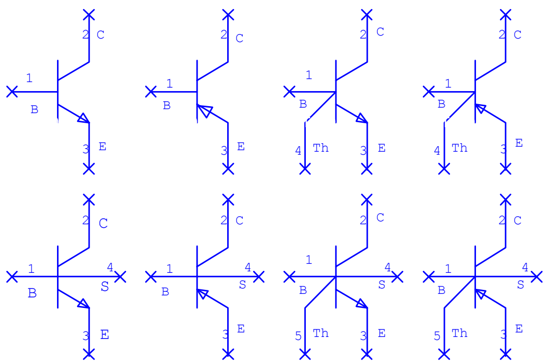
Except for the PNP structure, this model is equivalent to MXTR504N, the Mextram bipolar transistor model, level 504.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Mextram 504 BJT Model (504.10): MXTRM

Symbol



Summary

NI AWR's MXTRM element is the successor of MXTR504, and implements the entire family of Philips Mextram level 504 devices (bjt504, bjtd504, bjt504t, and bjtd504t). MXTRM works as a single entry point to a growing family of Verilog-A defined component models. The appropriate Verilog-A defined component is selected depending on what you specify, for example, version, self-heating support, or substrate support. Version 504.10 is the latest version supported by this model.

The Philips Mextram model provides an excellent description of vertical bipolar transistors in all types of processes, including modern SiGe processes and robust HV processes. It is very efficient in modeling the lowly-doped collector epilayer of a bipolar transistor where effects like velocity saturation, base widening, Kirk effect, and impact ionization play an important role. Effects resulting from the presence of Germanium in the base are also modeled. Furthermore, it contains a full description of the extrinsic regions of a transistor, including substrate current and capacitance. Mextram has formulations for temperature scaling and is easily scalable over geometry. Mextram level 504 supports full self-heating.

See the Philips model website[1] [11–268] for complete documentation and Verilog-A definition of component models.

Parameters

Only primary parameters are shown. Secondary parameters follow the Mextram 504 specification/standard.

Name	Description	Unit Type	Default
ID	Element ID	Text	MX1
VERSION	Model version		504.10
TYPE	Device type		npn
SELFT	Self-heating support		off
SUBS_NODE	Substrate node support		on
TNOM	Reference (extraction) temperature	Temperature	25DegC
TEMP	Simulation temperature	Temperature	_TEMP
MULT	Number of devices in parallel		1.0

Operating Points

You can access operating point information, as defined by the active Verilog-A based component model.

Implementation Details

The TYPE parameter controls whether the device is NPN or PNP, and the SUBS_NODE parameter enables/disables (on/off) substrate support. The SELFT parameter enables/disables the self-heating modeling capabilities. The current setting of any of these parameters is reflected by the device symbol. The extraction and simulation temperatures are controlled using the TNOM and TEMP parameters respectively; instead of TR and DTA. Parameter default and truncation values are identical to those employed by Philips.

Layout

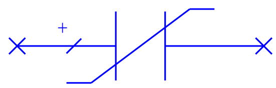
This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

References

[1] http://mextram.ewi.tudelft.nl/

(Obsolete) Nonlinear Capacitor: NLCAP

Symbol



Summary

This element is OBSOLETE and is replaced by the Nonlinear Capacitor (NLCAPA) element.

Parameters

Name	Description	Unit Type	Default
ID	Device ID	Text	C1
*SC	Scale factor		1
*VTH	Threshold voltage	Voltage	-1e+08 V
*C0	Polynomial coefficient	Capacitance	1 pF
*C1	Polynomial coefficient	Capacitance	0 pF
*C2	Polynomial coefficient	Capacitance	0 pF
*C3	Polynomial coefficient	Capacitance	0 pF
*C4	Polynomial coefficient	Capacitance	0 pF
*C5	Polynomial coefficient	Capacitance	0 pF
*C6	Polynomial coefficient	Capacitance	0 pF
*C7	Polynomial coefficient	Capacitance	0 pF
*C8	Polynomial coefficient	Capacitance	0 pF
*C9	Polynomial coefficient	Capacitance	0 pF
*C10	Polynomial coefficient	Capacitance	0 pF
*C11	Polynomial coefficient	Capacitance	0 pF
*C12	Polynomial coefficient	Capacitance	0 pF
*C13	Polynomial coefficient	Capacitance	0 pF
*C14	Polynomial coefficient	Capacitance	0 pF
*C15	Polynomial coefficient	Capacitance	0 pF

** indicates a secondary parameter*

Parameter Details

SCALING

Although all the Cn parameters except C0 technically do not have units of capacitance, they are treated as if they do. For example, if C2 = 0.1 and project units are pF, it is evaluated as $0.1 \cdot 10^{-12}$. This is done to simplify parameter extraction.

The charge Q(V) is scaled in proportion to SC.

Implementation Details

$$C(V) = \mathrm{SC}(C0 + C1 \cdot V + C2 \cdot V^2 + \ldots + C15 \cdot V^{15})$$

$$Q(V) = \mathrm{SC}(C0 \cdot V + \frac{1}{2}C1 \cdot V^2 + \frac{1}{3} \cdot V^3 + \ldots + \frac{1}{16}C15 \cdot V^{16})$$

NOTE:

If V < VTH,

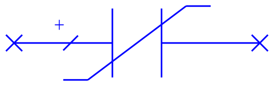
$$Q(V) = Q(\mathrm{VTH}) + C(\mathrm{VTH})(V - \mathrm{VTH})$$

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

Nonlinear Capacitor: NLCAPA

Symbol



Summary

NLCAPA is a replacement for NLCAP. It also provides all of the functionality found in Spectre's capacitor, or HSPICE's C device.

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	C1
*L	Capacitor length	Length	0
*W	Capacitor width	Length	0
*AREA	Capacitor area		
*PERIM	Capacitor perimeter		
MULT	Multiplicity factor		1
*TC1	Linear temperature coefficient of capacitance		0
*TC2	Quadratic temperature coefficient of capacitance		0
*TNOM	Parameter extraction temperature	Temperature	26.85DegC
*TEMP	Device temperature	Temperature	_TEMP
C	Capacitance	Capacitance	
*ETCH	Width narrowing due to etching per side	Length	0
*COEFFS	Vector of polynomial coefficients		
*CJ	Bottom capacitance density		
*CJSW	Sidewall capacitance density		
*THICK	Dielectric thickness		
*DI	Relative dielectric constant		

Operating Points

The letter pair pn identifies the only NL branch of this model. Consequently, Vpn and Ipn identify voltage and current of this branch, respectively.

Parameter	Description
cap (Capacitance)	Element capacitance

Implementation Details

Several differences are noticed when NLCAPA is compared with its predecessor NLCAP: polynomial coefficients are entered as a vector, COEFF={C0,C1,...}, instead of as individual parameters; there is temperature compensation support, in terms of parameters TC1 and TC2; and; the capacitance may be entered directly, or, calculated it from the physical length and width of the capacitor. In either case, the capacitance may be a function of temperature, or, applied voltage.

The capacitance is given by

$$C(V) = C \times (1 + C0 \cdot V + C1 \cdot V^2 + ...)$$

And the corresponding charge by

$$Q(V) = C \times \left(V + \frac{1}{2}C0 \cdot V^2 + \frac{1}{3}C1 \cdot V^3 + ...\right)$$

Layout

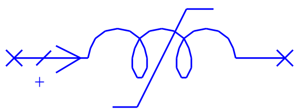
This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

Recommendations for Use

NOTE: NLCAPA is implemented as a nonlinear device in harmonic balance simulations, independently of whether the polynomial truly describes a nonlinear device. As a result, from a performance standpoint, it is advisable to limit the use of this element to those cases where the desired functionality cannot be implemented by a simpler linear element, such as CAP.

(Obsolete) Nonlinear Inductor: NLIND

Symbol



Summary

This element is OBSOLETE and is replaced by the Nonlinear Inductor (NLINDA) element.

Parameters

Name	Description	Unit Type	Default
ID	Device ID	Text	L1
*SC	Scale factor		1
*ITH	Threshold current	Current	-1e+11 mA
*L0	Polynomial coefficient	Inductance	1 nH
*L1	Polynomial coefficient	Inductance	0 nH
*L2	Polynomial coefficient	Inductance	0 nH
*L3	Polynomial coefficient	Inductance	0 nH
*L4	Polynomial coefficient	Inductance	0 nH
*L5	Polynomial coefficient	Inductance	0 nH
*L6	Polynomial coefficient	Inductance	0 nH
*L7	Polynomial coefficient	Inductance	0 nH
*L8	Polynomial coefficient	Inductance	0 nH
*L9	Polynomial coefficient	Inductance	0 nH
*L10	Polynomial coefficient	Inductance	0 nH
*L11	Polynomial coefficient	Inductance	0 nH
*L12	Polynomial coefficient	Inductance	0 nH
*L13	Polynomial coefficient	Inductance	0 nH
*L14	Polynomial coefficient	Inductance	0 nH
*L15	Polynomial coefficient	Inductance	0 nH

** indicates a secondary parameter*

Parameter Details

SCALING

Although all the Ln parameters except L0 technically do not have units of inductance, they are treated as if they do. For example, if L2 = 0.1 and project units are nH, it is evaluated as $0.1 \cdot 10^{-9}$ H. This is done to simplify parameter extraction.

The flux $\Phi(V)$ is scaled in proportion to SC.

Implementation Details

The inductance is given by

$$L(I) = \mathrm{SC}(L0 + L1 \cdot I + L2 \cdot I^2 + \ldots + L15 \cdot I^{15})$$

And the corresponding flux by

$$\Phi(I) = \mathrm{SC}(L0 \cdot I + \frac{1}{2}L1 \cdot I^2 + \frac{1}{3}L2 \cdot I^3 + \ldots + \frac{1}{16}L15 \cdot I^{16})$$

If I < ITH,

$$\Phi(I) = \Phi(\mathrm{ITH}) + L(\mathrm{ITH})(I - \mathrm{ITH})$$

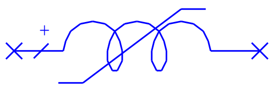
Node 1 is the positive terminal.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Nonlinear Inductor: NLINDA

Symbol



Summary

NLINDA is a replacement for NLIN. It also provides all the functionality found in Spectre's inductor.

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	L1
*L	Inductance	Inductance	0
*R	Resistance	Resistance	0
*MULT	Multiplicity factor		1
*TC1	Linear temperature coefficient of resistance		0
*TC2	Quadratic temperature coefficient of resistance		0
TNOM	Parameter extraction temperature	Temperature	26.85DegC
TEMP	Device temperature	Temperature	_TEMP
*COEFFS	Vector of polynomial coefficients		
*NFLAG	Noise flag		On
*KF	Flicker (1/f) noise coefficient		0
*AF	Flicker (1/f) noise exponential term		2

Operating Points

The letter pair pn identifies the only NL branch of this model. Consequently, Vpn and Ipn identify voltage and current of this branch, respectively.

Parameter	Description
ind (Inductance)	Element inductance

Implementation Details

Several differences are noticed when NLINDA is compared with its predecessor NLIND: polynomial coefficients are entered as a vector, COEFF = {L0,L1,L2,...}, instead of as individual parameters; NLINDA accounts for noise, both thermal and Flicker; parameter R is used to model the inductor's series resistance, and is temperature dependent:

R = R * (1 + TC1*(TEMP-TNOM)^2 + TC2*(TEMP-TNOM)^2).

In the following equations, I is the current into pin 1 (marked on the symbol), as measured in Amperes.

The inductance is given by

$$L(I) = L \times (1 + L0 \cdot I + L1 \cdot I^2 + ...)$$

The corresponding flux is given by

$$\Phi(I) = L \times (I + \frac{1}{2}L0 \cdot I^2 + \frac{1}{3}L1 \cdot I^3 + ...)$$

Layout

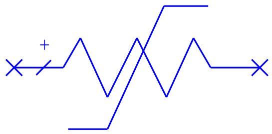
This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Recommendations for Use

NOTE: NLINDA is implemented as a nonlinear device in harmonic balance simulations, independently of whether the polynomial truly describes a nonlinear device. As a result, from a performance standpoint, it is advisable to limit the use of this element to those cases where the desired functionality cannot be implemented by a simpler linear element.

(Obsolete) Nonlinear Resistor: NLRES

Symbol



Summary

This element is OBSOLETE and is replaced by the Nonlinear Resistor (NLRESA) element.

Parameters

Name	Description	Unit Type	Default
ID	Device ID	Text	R1
*SC	Scale factor		1
*VTH	Threshold voltage	Voltage	-1e+08 V
*A0	Polynomial coefficient		0
*A1	Polynomial coefficient		0.01
*A2	Polynomial coefficient		0
*A3	Polynomial coefficient		0
*A4	Polynomial coefficient		0
*A5	Polynomial coefficient		0
*A6	Polynomial coefficient		0
*A7	Polynomial coefficient		0
*A8	Polynomial coefficient		0
*A9	Polynomial coefficient		0
*A10	Polynomial coefficient		0
*A11	Polynomial coefficient		0
*A12	Polynomial coefficient		0
*A13	Polynomial coefficient		0
*A14	Polynomial coefficient		0
*A15	Polynomial coefficient		0

** indicates a secondary parameter*

Parameter Details

SCALING

The current I(V) is scaled in proportion to SC.

Implementation Details

$$I(V) = \mathrm{SC}(A0 + A1 \cdot V + A2 \cdot V^2 + ... + A15 \cdot V^{15})$$

$$G(V) = \mathrm{SC}(A1 + 2A2V + ... + 15A15V^{14})$$

When V < VTH, the resistor is linear and has the I/V relation

$$I(V) = I(\mathrm{VTH}) + G(\mathrm{VTH})(V - \mathrm{VTH})$$

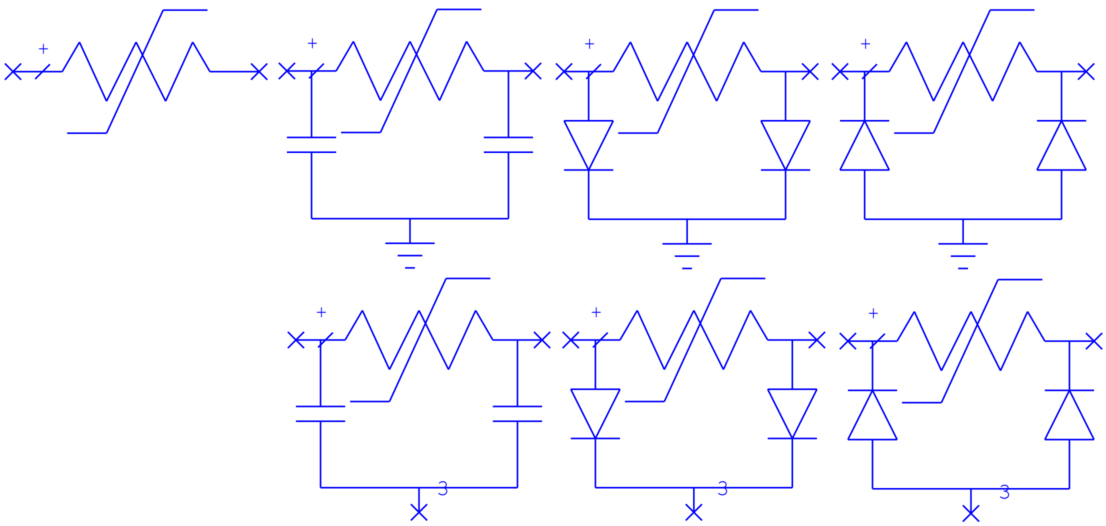
Node 1 is the positive terminal.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Nonlinear Resistor: NLRESA

Symbol



Summary

NI AWR's NLRESA element is a replacement for NLRES. It also provides most of the functionality found in Spectre's resistor and phy_res elements, and in HSpice's Wire RC model. This device has three different modes of operation: wire, physical resistor, and nonlinear resistor. In the wire mode of operation, NLRESA supports most of the functionality available in both Spectre's resistor and in HSpice's Wire RC model. In the phy_res mode of operation, NLRESA implements Spectre's phy_res element. In the nonlinear resistor mode (default), NLRESA behaves very much as its predecessor NLRES. The topology underlying each mode of operation is reflected by the device symbol.

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	R1
TYPE	Device type		2
SUBTYPE	Substrate type		0
*R	Resistance	Resistance	
*MULT	Multiplicity factor		1
*NFLAG	Noise flag		On
*L	Resistor length	Length	
*W	Resistor width	Length	
*TC1	Linear temperature coefficient of resistance		0
*TC2	Quadratic temperature coefficient of resistance		0
*TNOM	Parameter extraction temperature	Temperature	26.85DegC
*TEMP	Device temperature	Temperature	_TEMP
*C	Capacitance	Capacitance	

Name	Description	Unit Type	Default
*TC1C	Linear temperature coefficient of capacitance		0
*TC2C	Quadratic temperature coefficient of capacitance		0
*RSH	Sheet resistance		
*ETCH	Width narrowing due to etching per side	Length	0
*ETCHL	Length narrowing due to etching per side	Length	0
*ETCHC	Width narrowing due to etching per side for capacitances	Length	
*ETCHLC	Length narrowing due to etching per side for capacitances	Length	
*COEFFS	Vector of polynomial coefficients		
*NONLINFORM	Form of the nonlinear resistance		0
*CJ	Bottom capacitance		
*CJSW	Sidewall capacitance		
*FC	Junction bottom capacitor forward-bias threshold		0.5
*FCSW	Junction sidewall capacitor forward-bias threshold		0.5
*MJ	Junction bottom grading coefficient		0.5
*MJSW	Junction sidewall grading coefficient		0.33e-13
*PB	Junction bottom built-in potential		0.8
*PBSW	Junction sidewall built-in potential		0.8
*THICK	Dielectric thickness		
*DI	Relative dielectric constant		
*IS	Saturation current	Current	
*JS	Saturation current density		
*N	Emission coefficient		1
*CRATIO	Capacitance ratio		
*KF	Flicker (1/f) noise coefficient		0
*AF	Flicker (1/f) noise exponential term		2
*THIRD_TERMINAL	Third terminal connection		Grounded

Operating Points

The following letter pairs have been used to identify the NL branches: pn, p0 and n0. Here pn, p0 and n0 correspond to the branches defined between the positive and negative terminals, positive and ground terminals, and negative and ground terminals, respectively. These names are used to identify branch related operating point information, i.e., branch voltages, currents, etc.

Parameter	Description
gpn (Conductance)	Main branch conductance
gp0 (Conductance)	Positive terminal to ground conductance
gn0 (Conductance)	Negative terminal to ground conductance
cp0 (Capacitance)	Positive terminal to ground capacitance

Parameter	Description
cn0 (Capacitance)	Negative terminal to ground capacitance

Implementation Details

The TYPE parameter allows you to choose between three different modes of operation: wire, physical resistor, and, non-linear resistor (resistor). The SUB_TYPE parameter, available in the physical resistor mode of operation only, allows you to choose the substrate type. Possible values are n, p or poly. Finally, the THIRD_TERMINAL parameter, available in the physical resistor and wire modes of operation only, controls whether the third terminal is grounded, or can be externally accessed.

In the nonlinear resistor mode (default), NLRESA behaves very much as its predecessor NLRES, but there are some differences worth mentioning: polynomial coefficients are entered as a vector, COEFF={C0, C1, C2, ...}, instead of as individual parameters; and; the non-linearity described by the coefficients can be either a resistance, or, a conductance. Also, NLRESA accounts for noise, both thermal and Flicker.

When NONLINFORM is set to conductance the resistive branch of the element has following I/V relationship:

$$I(V) = \frac{1}{R} \times \left(V + \frac{1}{2}C0 \cdot V^2 + \frac{1}{3}C1 \cdot V^3 + ...\right)$$

$$G(V) = \frac{1}{R} \times (1 + C0 \cdot V + C1 \cdot V^2 + ...)$$

When NONLINFORM is set to resistance, it has the following form:

$$G(V) = \frac{1}{R} \times (1 + C0 \cdot V + C1 \cdot V^2 + ...)$$

Layout

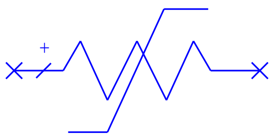
This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

Recommendations for Use

NOTE: NLRESA is implemented as a nonlinear device in harmonic balance simulations, independently of whether the polynomial truly describes a nonlinear device. As a result, from a performance standpoint, it is advisable to limit the use of this element to those cases where the desired functionality cannot be implemented by a simpler linear element, such as RES or REST.

Table-Specified Nonlinear Resistor: NLRESTAB

Symbol



Summary

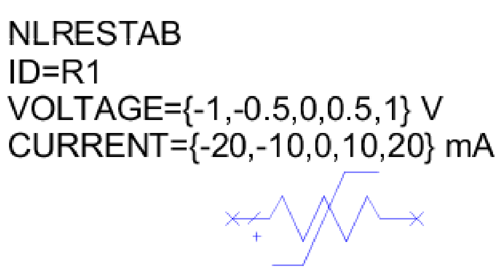
NLRESTAB is a table-specified nonlinear resistor that uses spline interpolation of the specified I(V) curve with linear extrapolation outside of the specified voltage range. It is essential to specify a sufficient voltage range so that linear extrapolation does not affect the final result. This way the extrapolation will only be used during Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	R1
Voltage	Vector of voltage values (independent variable)	Voltage	{-1, -0.5, 0, 0.5, 1} V
Current	Vector of current values	Current	{-0.02, -0.01, 0, 0.01, 0.02} A

Implementation Details

NLRESTAB is similar to NLRES but it uses a spline-interpolated I(V) curve rather than a polynomial. Spline interpolation is the only interpolation method available as it preserves continuity of the first derivative that is a mandatory requirement for all models usable with the HB simulator. The element uses a natural spline (defined by the conditions I''(V1)=0, I''(Vn)=0). The extrapolation outside of the range of the specified voltage values is linear using the values and derivatives at the end points.

HSPICE translation limits the number of points in the table to a maximum of 98.

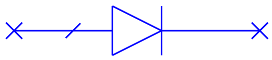


Layout

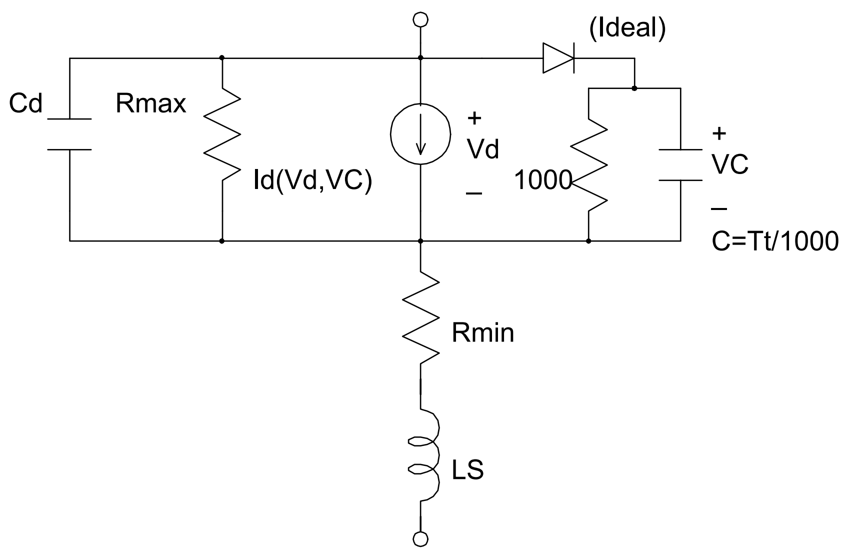
This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

PIN Diode: PINDD

Symbol



Equivalent Circuit



Parameters

Name	Description	Unit Type	Default
ID	Device ID	Text	P1
*IS	Reverse saturation current	Current	1e-11 mA
*N	Ideality factor		1
*TT	Storage time	Time	0 ns
*GD	Parameter for junction resistance		100
*KD	Current exponential		0.9
*CD	RF junction capacitance	Capacitance	0 pF
*RMIN	Minimum junction resistance	Resistance	0.001 ohm
*RMAX	Maximum junction resistance	Resistance	1e+06 ohm
*LS	Series inductance	Inductance	1e-10 nH
*T	Temperature	Temperature	27 DegC
*AFAC	Area scale factor		1

** indicates a secondary parameter*

Implementation Details

If $I_d>0$,

$$I(V_d, V_c) = G_d I_d^{K_d} V_d + I_d$$

If $I_d<0$,

$$I(V_d, V_c) = -G_d (I_d)^{K_d} V_d + I_d$$

where I_d is

$$I_d = I_s(\exp(\frac{\mathrm{qV}_c}{\mathrm{NKT}}) - 1)$$

This exponential function is linearized at high currents to prevent numerical overflow. The current $I(V_d, V_c)$ is scaled in proportion to area.

Layout

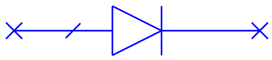
This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Reference

This model was developed internally at NI AWR.

Caverly PIN Diode Model: PINDRC

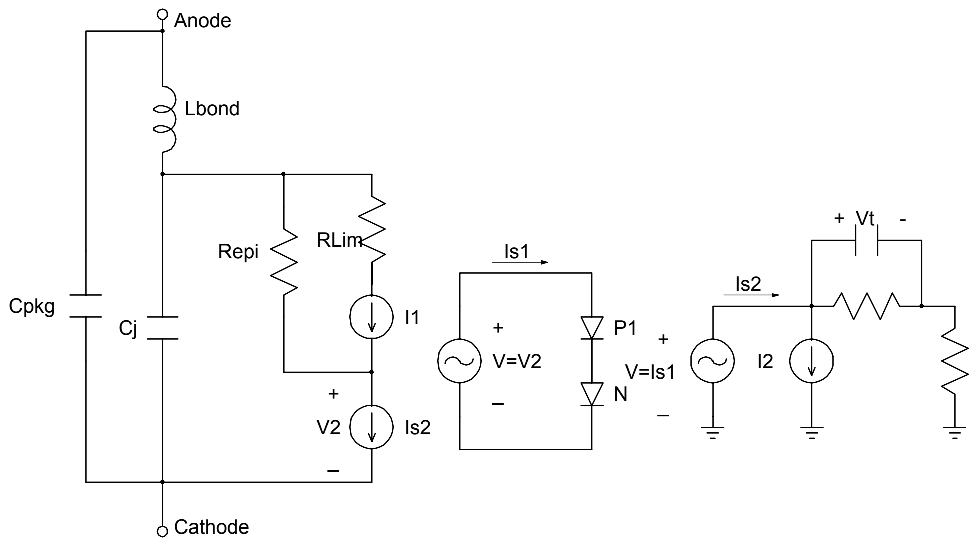
Symbol



Summary

PINDRC is an implementation of the Caverly PIN diode model [1] [11–287]. This advanced model is strongly recommended for all types of PIN diode circuit design.

Equivalent Circuit



Parameters

Name	Description	Unit Type	Default
ID	Device ID	Text	P1
*IS	Reverse saturation current	Current	1e-6 mA
*IKNEE	Knee param for current dependent tau	Current	1e6 mA
*N	Ideality factor		1
*RLIM	Minimum series resistance	Resistance	0.001 ohm
*REPI	Epi leakage resistance	Resistance	1000 ohm
*CJ	Reverse capacitance	Capacitance	0.1 pF
*CPKG	Package capacitance	Capacitance	0.1 pF
*TAU	Storage time	Time	57 ns
*W	I region width in micrometers		6.0 um

Name	Description	Unit Type	Default
*B	Ratio of electron to hole mobility		3.0
*LBOND	Bond wire inductance	Inductance	0.1 nH

** indicates a secondary parameter*

Restrictions

REPI should not be made too large because of the possibility of the spurious "zero solution," in which the diode looks like an open circuit regardless of the voltage across it. Usually, REPI can be made fairly low (~1KΩ) without any effect on the simulation results. A low value of REPI also facilitates convergence, especially in self-biased limiter circuits. Since I_1 represents a relatively low resistance, a large value of REPI should never be necessary. Reducing the Absolute Error limit (Circuit Options, Harmonic Balance tab) may also prevent zero solutions.

N is not the ideality factor of the PIN diode, as measured at DC, but a parameter from which the diode's ideality factor can be derived. See the detailed explanation, below.

If W is not provided by the diode manufacturer, it can be calculated from τ and the RF resistance curve. See below.

Implementation Details

This is an advanced PIN diode model that includes a description of the I-region charge that is accurate for both large- and small-signal cases. The model is recommended over PINDD.

Equations

Aside from the diode junctions, which are represented by standard PNIV elements, there are only two nonlinear elements in the model, I_1 and I_2. I_1 accounts for the nonlinear series resistance; it is given by

$$I_1 = \frac{2V_1V_\tau}{V_m}$$

where

$$V_m = \frac{W^2}{0.1\tau}$$

I_2 accounts for the current-dependent storage time. It is given by

$$I_2 = \frac{V_2^2}{\text{IKNEE}}$$

The voltages V_1, V_2, and V_t are shown in the figure.

The diodes represent the PI junction and the IN junction. Their current parameters are both IS, but their ideality factors differ. They are, respectively, N_{PI} and N_{IN}, given by

$$N_{\text{PI}} = \frac{2N}{(1+B)}$$

$$N_{\mathrm{IN}} = \frac{2\mathrm{NB}}{(1+B)}$$

The resulting I/V characteristic for the complete PIN diode, consisting of the PI and NI junctions in series, is a conventional diode I/V characteristic having the ideality factor

$$N_{\mathrm{PIN}} = 2N$$

The default value of B, for silicon diodes, is 3.0; this probably should not be changed in most cases.

Parameters

The critical parameters for a forward-biased PIN diode are W and τ. If other parameters are unknown, the defaults can be used. W and τ are usually given in manufacturers' data sheets; if W is not given, it can be found from curves of the RF resistance, R_d, as a function of DC current, I_d. The relation is

$$R_d = \frac{W^2}{\mu(1+1/B)\tau I_d}$$

where μ is the electron mobility (~1200 cm^2/Vs in silicon). At low currents, the RF resistance is inversely proportional to Id. At currents above IKNEE, τ decreases with increased Id and the R_d-I_d curve flattens somewhat. τ is the most fundamental of PIN diode parameters. If τ is not given in the data sheet, find a new diode supplier.

In reverse bias, the model treats the junction as a constant capacitance, CJ. This value should be provided by the manufacturer.

The parameters IS and N describe DC conduction in the same way as a Schottky diode. These can be extracted in the conventional manner from a DC I/V plot.

CPKG and LBOND are package parasitics. If a chip diode is used, LBOND represents the bond-wire inductance and CPKG should be set to zero.

Layout

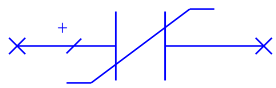
This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

References

[1] R. Caverly et al., "Spice Modeling of Microwave and RF Control Diodes," Midwest Symposium on Circuits and Systems Digest of Papers, 2000.

Diode Junction Capacitance: PNCAP

Symbol



Summary

PNCAP is a PN or Schottky junction depletion capacitance. It is identical to the model used in SPICE for diodes and the depletion components of BJT capacitance.

Parameters

Name	Description	Unit Type	Default
ID	Device ID	Text	PC1
*CJ0	Zero-voltage junction capacitance	Capacitance	0 pF
*VJ	Junction potential	Voltage	1 V
M	Grading parameter		0.5
*FC	Depletion capacitance linearization parameter		0.5
AFAC	Area scaling factor		1

** indicates a secondary parameter*

Implementation Details

The charge-voltage characteristic Q(Vj), where Vj is junction voltage, is identical to that used in SPICE to model depletion capacitances. It is given by the following equations, which result in a linearly with junction voltage when Vj > FC VJ. This prevents the problem of

$$C(V_j) \rightarrow \infty$$

$$V_j = \mathrm{VJ}$$

:

$$Q(V_j) = \begin{cases} \dfrac{\mathrm{VJ} \cdot \mathrm{CJ0}}{1-M}\left(1 - \dfrac{V_j}{\mathrm{VJ}}\right)^{1-M} & V_j \le V_{\mathrm{th}} \\ \mathrm{CJ0} \cdot F1 + \dfrac{\mathrm{CJ0}}{F2}\left(F3 \cdot V_j + \dfrac{M \cdot V_j^2}{2 \cdot \mathrm{VJ}} + F4\right) & V_j > V_{\mathrm{th}} \end{cases}$$

The capacitance-voltage characteristic $C(V_j)$ is found by differentiating the above equations:

$$C(V_j) = \begin{cases} \mathrm{CJ0}\left(1 - \dfrac{V_j}{\mathrm{VJ}}\right)^{-M} & V_j \le V_{\mathrm{th}} \\ \dfrac{\mathrm{CJ0}}{F2}\left(F3 + \dfrac{M \cdot V_j}{\mathrm{VJ}}\right) & V_j > V_{\mathrm{th}} \end{cases}$$

where

$$V_{th} = FC \cdot VJ$$

$$F1 = \frac{VJ}{1-M}(1-(1-FC)^{1-M})$$

$$F2 = (1-FC)^{1+M}$$

$$F3 = 1 - FC(1+M)$$

$$F4 = FC \cdot VJ(FC(1-\frac{M}{2})-1)$$

When 0.999 < M < 1.001, it is offset to the closest end of this range.

Layout

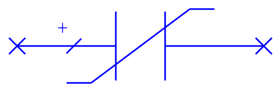
This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Reference

[1] P. Antognetti and G. Massobrio, Semiconductor Device Modeling with SPICE, New York: McGraw-Hill, 1988.

Diode Junction Capacitance With Diffusion: PNDCAP

Symbol



Summary

PNDCAP models a PN junction depletion and diffusion capacitance. It is identical to the model used in SPICE for diodes and the capacitance components of BJTs.

Parameters

Name	Description	Unit Type	Default
ID	Device ID	Text	PD1
*CJ0	Zero-voltage junction capacitance	Capacitance	0 pF
*VJ	Junction potential	Voltage	1 V
*M	Grading parameter		0.5
*FC	Depletion capacitance linearization parameter		0.5
*TT	Storage time	Time	0 ns
*IS	Reverse saturation current	Current	1e-11 mA
*N	Ideality factor		1
*T	Temperature	Temperature	27 DegC
AFAC	Area scaling factor		1

** indicates a secondary parameter*

NOTE: Although junction current parameters must be entered to calculate diffusion capacitance, this model does not include junction current.

Implementation Details

The depletion charge expression is identical to that of PNCAP. Additionally, PNDCAP includes a diffusion charge component, given by the following:

$$Q_d = \mathrm{TT} \cdot I_j$$

where

$$I_j = \mathrm{IS}(\exp(\frac{q \cdot V_j}{\mathrm{NKT}} - 1)$$

In the above equations, q is electron charge, K is Boltzmann's constant, and T is temperature in Kelvins. The total charge is the sum of the depletion and diffusion components.

Layout

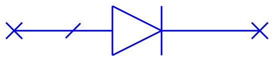
This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Reference

[1] P. Antognetti and G. Massobrio, Semiconductor Device Modeling with SPICE, New York: McGraw-Hill, 1988.

Diode Resistive Junction No Parasitics: PNIV

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Device ID	Text	PN1
*IS	Reverse saturation current	Current	1e-11 mA
*N	Ideality factor		1
*BV	Breakdown voltage (positive)	Voltage	1e+06 V
*IKF	Forward knee current	Current	0 mA
*IKR	Reverse knee current	Current	0 mA
*IBV	Current at breakdown voltage	Current	0 mA
*T	Device temperature	Temperature	27 DegC
AFAC	Junction area scaling factor		1

** indicates a secondary parameter*

Implementation Details

The I/V characteristic is identical to that of the SDIODE element. This element does not include junction capacitance or series resistance. As a result, a smaller set of parameters controls the behavior of this device.The same as SDIODE, this device is mapped to HSPICE as a D-device with parameter LEVEL set to 1.

Layout

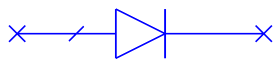
This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Reference

[1] P. Antognetti and G. Massobrio, Semiconductor Device Modeling with SPICE, New York: McGraw-Hill, 1988.

Diode, Noisy Resistive Junction, No Parasitics: PNIV_N

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Device ID	Text	PN1
*IS	Reverse saturation current	Current	1e-11 mA
*N	Ideality factor		1
*BV	Breakdown voltage (positive)	Voltage	1e+06 V
*IKF	Forward knee current	Current	0 mA
*IKR	Reverse knee current	Current	0 mA
*IBV	Current at breakdown voltage	Current	0 mA
*T	Temperature	Temperature	27 DegC
AFAC	Junction area scaling factor		1
*KF	Flicker noise coefficient		0
*AF	Flicker noise exponent		1
*FFE	Flicker noise frequency exponent		1
*KB	Burst noise coefficient		0
*AB	Burst noise exponent		1
*FB	Burst noise cutoff frequency		1
*NFLAG	Noise model		Noise On

** indicates a secondary parameter*

Implementation Details

This model is identical to PNIV, except that it includes SPICE noise sources. The noise model is identical to that of SDIODE.

Layout

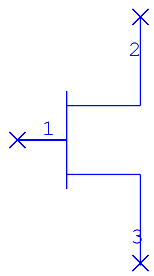
This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

References

[1] P. Antognetti and G. Massobrio, Semiconductor Device Modeling with SPICE, New York: McGraw-Hill, 1988.

Parker-Skellern FET Model: PSFET

Symbol



Summary

The Parker-Skellern FET model is an advanced MESFET model, which may also be useful for HEMTs. It features advanced modeling of dispersion phenomena and derivatives of the I/V characteristic.

Parameters

Name	Description	Unit Type	Default
ID	PS MESFET	Text	PSF1
*ACGAM	Capacitance modulation		0
*BETA	Linear-region transconductance scale		0.0001
*CGD	Zero-bias gate-drain capacitance	Capacitance	0 pF
*CGS	Zero-bias gate-source capacitance	Capacitance	0 pF
*DELTA	Thermal reduction coefficient		0
*FC	Forward bias capacitance parameter		0.5
*HFETA	High-frequency VGS feedback parameter		0
*HFE1	HFGAM modulation by vGD		0
*HFE2	HFGAM modulation by vGS		0
*HFGAM	High-frequency VGD feedback parameter		0
*HFG1	HFGAM modulation by vSG		0
*HFG2	HFGAM modulation by vDG		0
*IBD	Gate-junction breakdown current	Current	0 mA
*IS	Gate-junction saturation current	Current	1e-11 mA
*LFGAM	Low-frequency feedback parameter		0
*LFG1	LFGAM modulation by vSG		0
*LFG2	LFGAM modulation by vDG		0
*MVST	Subthreshold modulation		0
*N	Gate-junction ideality factor		1
*P	Linear-region power-law exponent		2
*Q	Saturated-region power-law exponent		2

Name	Description	Unit Type	Default
*RD	Drain ohmic resistance	Resistance	0 ohm
*RS	Source ohmic resistance	Resistance	0 ohm
*TAUD	Relaxation time for thermal reduction	Time	0 ns
*TAUG	Relaxation time for gamma feedback	Time	0 ns
*VBD	Gate-junction breakdown potential	Voltage	1 V
*VBI	Gate-junction potential	Voltage	1 V
*VST	Subthreshold potential	Voltage	0 V
*VTO	Threshold potential	Voltage	-2 V
*XC	Capacitance pinch-off reduction factor		0
*XI	Saturation-knee potential factor		1000
*Z	Knee transition parameter		0.5
*RG	Gate ohmic resistance	Resistance	0 ohm
*LG	Gate inductance	Inductance	0 nH
*LS	Source inductance	Inductance	0 nH
*LD	Drain inductance	Inductance	0 nH
*CDSS	Fixed Drain-source capacitance	Capacitance	0 pF
AFAC	Gate-width scale factor		1
NFING	Number of gate fingers scale factor		1
TEMP	Simulation temperature	Temperature	28.85 DegC
COMPAT	Compatibility selector: AWR or ADS		AWR
*CPD	Linear gate-drain capacitance	Capacitance	0 pF
*CPG	Linear gate-source capacitance	Capacitance	0 pF
*W	Device width	Length	1 m

** indicates a secondary parameter*

Implementation Details

A new COMPAT parameter is added. This flag allows toggling between two different behaviors, the behavior corresponding to the AWR implementation of the model (COMPAT=AWR) and a new one, the ADS compatibility mode of operation (COMPAT=ADS). Parameters CPD, CPG, and W are only visible in the ADS compatibility mode. Equations are too extensive to list. For more information, see Ref. [1]. [11–296]

Parameter Scaling

$$RF \rightarrow \frac{1E^{-6}}{AFAC}$$

$$RG \rightarrow \frac{RG \cdot AFAC}{NFING^2 + RF}$$

$$RS \rightarrow \frac{RS}{AFAC + RF}$$

$$RD \rightarrow \frac{RD}{AFAC + RF}$$

$$IBD \rightarrow IBD \cdot AFAC$$

$$IS \rightarrow IS \cdot AFAC$$

$$BETA \rightarrow BETA \cdot AFAC$$

$$DELTA \rightarrow \frac{DELTA}{AFAC}$$

$$CGD \rightarrow CGD \cdot AFAC$$

$$CGS \rightarrow CGS \cdot AFAC$$

$$CDSS \rightarrow CDSS \cdot AFAC$$

Layout

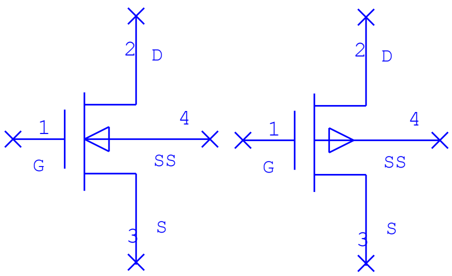
This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

References

[1] A. Parker and D. Skellern, "A Realistic Large-signal MESFET Model for SPICE," IEEE Trans. Microwave Theory Tech., pp. 1563-1571, Sept. 1997.

PSP Model Version 102.1: PSP

Symbol



Summary

PSP implements the entire family of Philips MOS model PSP level 102 devices, i.e., psp1020, psp1021 and psp102e. PSP is based on the Philips SiMKit 2.4.

The PSP model is a new compact MOSFET model, which has been jointly developed by Philips Research and Penn State University. It is a surface-potential based MOS model appropriate to model present-day and upcoming deep-submicron bulk CMOS technologies. It accounts for the following physical effects:

- mobility reduction
- velocity saturation
- DIBL gate current
- lateral doping gradient effects
- STI stress

The best reference for the model equations is the Philips website.

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	M1
LEVEL	Parameters and clipping selector		
*TNOM	Parameter extraction temperature	DegC	21
*TEMP	Device Temperature	DegC	_TEMP
*NFLAG	Noise Flag		Noise On
*MULT	Number of devices in parallel		1
TYPE	Device type		N
*A1L	Length dependence of A1		0
*A1O	Geometry independent impact-ionization pre-factor		1
*A1W	Width dependence of A1		0
*A2O	Impact-ionization exponent at TR (V_)		10
*A3L	Length dependence of A3		0

Name	Description	Unit Type	Default
*A3O	Geometry independent saturation-voltage dependence of II		0
*A3W	Width dependence of A3		0
*A4W	Width dependence of A4		0
*ABDRAIN	Bottom area of drain junction (m^2)		1e-12
*ABSOURCE	Bottom area of source junction (m^2)		1e-12
*AD	Bottom area of drain junction (m^2)		1e-12
*AGIDLW	Width dependence of GIDL pre-factor (A/V^3)		0
*ALP1L1	Length dependence of CLM enhancement factor above threshold (V)		0
*ALP1L2	Second_order length dependence of ALP1		0
*ALP1LEXP	Exponent for length of ALP1		0.5
*ALP1W	Width dependence of ALP1		0
*ALP2L1	Length dependence of CLM enhancement factor below threshold (V^-1)		0
*ALP2L2	Second_order length dependence of ALP2		0
*ALP2LEXP	Exponent for length dependence of ALP2		0.5
*ALP2W	Width dependence of ALP2		0.0005
*ALPL	Length dependence of ALP		1
*ALPLEXP	Exponent for length dependence of ALP		1
*ALPW	Width dependence of ALP		0
*AS	Bottom area of source junction (m^2)		1e-12
*AXL	Length dependence of AX		0.4
*AXO	Geometry independent linear/saturation transition factor		18
*BETW1	First higher-order width scaling coefficient of BETN		0
*BETW2	Second higher-order width scaling coefficient of BETN		0
*BGIDLO	GIDL probability factor at TR (V)		41
*CFBO	Back-bias dependence of DIBL-parameter (V^-1)		0
*CFL	Length dependent of DIBL-parameter (V^-1)		0
*CFLEXP	Exponent for length dependence of CF		0
*CFRW	Outer fringe capacitance for 1um wide channel (F)		0
*CFW	Width dependence of CF		0
*CGBOVL	Oxide capacitance for gate-bulk overlap for 1um^2 area (F)		0
*CGIDLO	Back-bias dependence of GIDL		0
*CHIBO	Tunnelling barrier height (V)		3.1
*CSL	Length dependence of CS		0
*CSLEXP	Exponent for length dependence of CS		0
*CSLW	Area dependence of CS		0

Name	Description	Unit Type	Default
*CSO	Geometry independent coulomb scattering parameter at TR		0
*CSW	Width dependence of CS		0
*CTL	Length dependence of interface states factor		0
*CTLEXP	Exponent for length dependence of interface states factor		1
*CTLW	Area dependence of interface states factor		0
*CTO	Geometry-independent interface states factor		0
*CTW	Width dependence of interface states factor		0
*DLQ	Effective channel length reduction for CV (m)		0
*DNSUBO	Effective doping bias-dependence parameter (V^-1)		0
*DPHIBL	Length dependence offset of PHIB (V)		0
*DPHIBLEXP	Exponent for length dependence of offset of PHIB		1
*DPHIBLW	Area dependence offset of PHIB (V)		0
*DPHIBO	Geometry independent coulomb scattering parameter at TR		0
*DPHIBW	Width dependence of offset of PHIB		0
*DWQ	Effective channel width reduction for CV (m)		0
*FBET1	Relative mobility decrease due to first lateral profile		0
*FBET1W	Width dependence of relative mobility decrease due to first la		0
*FBET2	Relative mobility decrease due to second lateral profile		0
*FETAO	Effective field parameter		1
*FNTO	Thermal noise coefficient		1
*FOL1	First length dependence coefficient for short channel body eff		0
*GC20	Gate current slope factor		0.375
*GC30	Gate current curvature factor		0.063
*GCOO	Gate tunneling energy adjustment		0
*IGINVLW	Gate channel current pre-factor for 1 um^2 channel area (A)		
*IGOVW	Gate overlap current pre-factor for 1 um wide channel (A)		0
*KUO	Gate overlap current pre-factor for 1 um wide channel (A)		0
*KVSAT	Saturation velocity degradation/enhancement coefficient (m)		0
*KVTHO	Threshold shift parameter (Vm)		0
*L	Design length (m)		1e-5
*LAP	Effective channel length reduction side due to lateral diffision (m)		0
*LGDRAIN	Gate-edge length of drain junction (m)		1e-6
*LGSOURCE	Gate-edge length of source junction (m)		1e-6
*LKUO	Length dependence of KUO (m^LLODKUO)		0

Name	Description	Unit Type	Default
*LKVTHO	Length dependence of KVTHO (m^LLODVTH)		0
*LLODKUO	Length parameter for UO stress effect		0
*LLODVTH	Length parameter for VTH-stress effect		0
*LODETAO	eta0 shift modification factor for stress effect		1
*LOV	Overlap length for gate/drain and gate/source overlap capacitance (m)		0
*LP1	Mobility-related characteristic length of first lateral profile (m)		1e-8
*LP1W	Width dependence of mobility-related characteristic length		0
*LP2	Mobility-related characteristic length of second lateral profile (m)		1e-8
*LPCK	Char. length of lateral doping profile (m)		1e-8
*LPCKW	Width dependence of char. length of lateral doping profile		0
*LSDRAIN	STI-edge length of drain junction (m)		1e-6
*LSSOURCE	STI-edge length of source junction (m)		1e-6
*LVARL	Length dependence of difference between actual and program		0
*LVARO	Geometry independent difference between actual programmed gate (m)		0
*LVARW	Width dependence of LVAR		0
*MUEO	Geometry independent mobility reduction coefficient at TR (m/V)		0.5
*MUEW	Width dependence of mobility reduction coefficient at TR		0
*MUNQSO	Relative mobility for NQS modelling		1
*NFALW	First coefficient of flicker noise for 1 um^2 channel area (V^-1/m^4)		8e22
*NFBLW	Second coefficient of flicker noise for 1 um^2 channel area (V^-1/m^2)		30000000
*NFCLW	Third coefficient of flicker noise for 1um^2 channel area (V^-1)		0
*NOVO	Effective doping of overlap region (m^-3)		5e25
*NPCK	Pocket doping level (m^-3)		1e24
*NPCKW	Width dependence of pocket doping NPCK due to segregation		0
*NPL	Length dependence of gate poly-silicon doping		0
*NPO	Geometry-independent gate poly silicon doping (m^-3)		1e26
*NSLPO	Effective doping bias-dependence parameter (V)		0.05
*NSUBO	Geometry independent substrate doping (m^-3)		3e23
*NSUBW	Width dependence of background doping NSUBO due to segregation		0
*PD	Perimeter of drain junction (m)		1e-6

Name	Description	Unit Type	Default
*PKUO	Cross-term dependence of KUO (m^(LLODKUO+WLODKUO))		0
*PKVTHO	Cross-term dependence of KVTHO (m^(LLODVTH+WLODVTH))		0
*PS	Perimeter of source junction (m)		1e-6
*QMC	Quantum-mechanical correction factor		1
*RBULKO	Bulk resistance between node BP and BI (ohm)		0.001
*RGO	Gate resistance		0.001
*RJUNDO	Drain-size bulk resistance between node BI and BD (ohm)		0.001
*RJUNSO	Source-side bulk resistance between node BI and BS (ohm)		0.001
*RSBO	Back-bias dependence of series resistance (V^-1)		0
*RSGO	Gate-bias dependence of series resistance (V^-1)		0
*RSW1	Source/drain series resistance for 1 um wide channel at TR (ohm)		2500
*RSW2	Higher-Order width scaling of RS		0
*RWELLO	Well resistance between node BI and B (ohm)		0.001
*SA	Distance between OD-edge to poly from one side (m)		0
*SAREF	Reference distance between OD-edge to poly from one side (m)		1e-6
*SB	Distance between OD-edge to poly from other side		0
*SBREF	Reference distance between OD-edge to poly from other side (m)		1e-6
*STA20	Temperature dependence of A2 (V)		0
*STBETL	Length dependence of temperature dependence of BETN		0
*STBETLW	Area dependence of temperature dependence of BETN		0
*STBETO	Geometry independent temperature dependence of BETN		1
*STBETW	Width dependence of temperature dependence of BETN		0
*STBGIDLO	Temperature dependence of BGIDL (V/K)		0
*STCSO	Temperature dependence of CS		0
*STEATAO	eta0 shift factor related to VTHO change (m)		0
*STIGO	Temperature dependence of IGINV AND IGOV		2
*STMEUO	Temperature dependence of MUE		0
*STRSO	Temperature dependence of RS		1
*STTHEMUO	Temperature dependence of THEMU		1.5
*STTHESATLW	Area dependence of temperature dependence of THESAT		0
*STTHESATO	Geometry independent temperature dependence of THESAT		1
*STTHESATW	Width dependence of temperature dependence of THESAT		0
*STVFBL	Length dependence of temperature dependence of VFB		0

Name	Description	Unit Type	Default
*STVFBLW	Area dependence of temperature dependence of VFB		0
*STVFBO	Geometry-independent temperature dependence of VFB (V/K)		0.0005
*STVFBW	Width dependence of temperature dependence of VFB		0
*STXCORO	Temperature dependence of XCOR		0
*SWGIDL	Flag for GIDL current		0
*SWIGATE	Flag for gate current		0
*SWIMPACT	Flag for impact ionization current		0
*SWJUNCAP	Flag for juncap		0
*SWNQS	Flag for NQS (number of collocation points)		0
*THEMUO	Mobility reduction exponent at TR		1.5
*THESATBO	Back-bias dependence of velocity saturation (V^-1)		0
*THESATGO	Gate-bias dependence of velocity saturation (V^-1)		0
*THESATL	Length dependence of THESAT(V^-1)		0.05
*THESATLEXP	Exponent for length dependence of THESAT		1
*THESATLW	Area dependence of velocity saturation parameter		0
*THESATO	Geometry independent velocity saturation parameter at TR		0
*THESATW	Width dependence of velocity saturation parameter		0
*TKUO	Temperature dependence of KUO		0
*TOXO	Gate oxide thickness (m)		2e-9
*TOXOVO	Overlap oxide thickness (m)		2e-9
*TRJ	reference temperature (C)		21
*UO	Zero-field mobility at TR (m^2/V/s)		0.05
*VFBL	Length dependence of flat-band voltage		0
*VFBLW	Area dependence of flat-band voltage		0
*VFBO	Geometry-independent flat0band voltage at TR (V)		-1
*VFBW	Width dependence of flat-band voltage		0
*VNSUBO	Effective doping bias-dependence parameter (V)		0
*VPO	CLM logarithmic dependence parameter (V)		0.05
*W	Design width (m)		1e-5
*WBET	Characteristic width for width scaling of BETN (m)		1e-9
*WKUO	Width dependence of KUO (m^WLODKUO)		0
*WKVTHO	Width dependence of KVTHO (m^WLODVTH)		0
*WLOD	Width parameter (m)		0
*WLODKUO	Width parameter for UO stress effect		0
*WLODVTH	Width parameter for VTH-stress effect		0
*WOT	Effective reduction of channel width per side due to later (m)		0

Name	Description	Unit Type	Default
*WSEG	Char. length of segregation of background doping NSUBO (m)		1e-8
*WSEGP	Char. length of segregation of pocket doping NPCK (m)		1e-8
*WVARL	Length dependence of WVAR		0
*WVARO	Geom. independent difference between actual and programmed field oxide opening (m)		0
*WVARW	Width dependence of difference between actual and programmed field oxide opening (m)		0
*XCOR	Non-universality factor (V^-1)		0
*XCORL	Length dependence of non-universality parameter		0
*XCORLW	Area dependence of non-universality parameter		0
*XCORO	Geometry independent non-universality parameter (V^-1)		0
*XCORW	Width dependence of non-universality parameter		0

** indicates a secondary parameter*

Implementation Details

The TYPE parameter controls whether the device is N or P channel. You can select the desired model level using the LEVEL parameter, which allows selecting between physical geometrical scaling rules (1020), binning geometrical scaling rules (1021), and the electrical model (102e). Unlike the comprehensive parameter list above, only those parameters corresponding to a given level are displayed with the element. The extraction and simulation temperatures correspond to the TNOM and TEMP parameters, respectively; instead of the TR and DTA parameters. Parameter default and truncation values are identical to those employed by Philips. Operating point information also follows Philip's prescription closely.

Layout

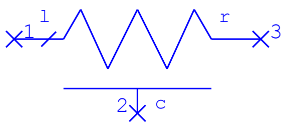
This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

References

[1] http://www.nxp.com/acrobat_download/other/philipsmodels/nl_tn2006_00546.pdf

CMC 3-terminal Resistor Model: R3_CMC

Symbol



Summary

NI AWR's R3_CMC element is a three-terminal nonlinear (diffused and polysilicon) resistor model and JFET Model. R3_CMC is Verilog-A based and implements version 1.0.

See the Compact Model Alliance website[1] [11–305] for complete documentation and Verilog-A definition of this model.

Parameters

Only primary parameters are shown. Secondary parameters follow the R3_CMC model specification/standard.

Name	Description	Binnable	Unit Type	Default
ID	Element ID		Text	X1
m	multiplicity factor		Scalar	1
w	design width of resistor body		Length	1
l	design length of resistor body		Length	1
trise	local temperature delta to ambient (before self-heating)		Scalar	0
tnom	nominal (reference) temperature			27

Operating Points

Operating point information is identical to that found in the Verilog-A definition of the model.

Implementation Details

The type parameter controls whether the substrate is p or n type. The extraction and simulation temperatures are controlled using the tnom and trise parameters, respectively. Parameter default and truncation values are identical to those found in the Verilog-A definition of the model.

Layout

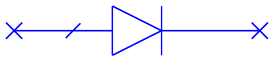
This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

References

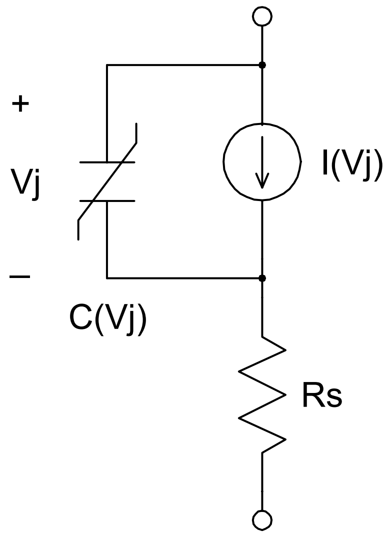
[1] https://awrcorp.com/support/help.aspx?id=56

SPICE Diode Model: SDIODE

Symbol



Equivalent Circuit



Parameters

Name	Description	Unit Type	Default
ID	Diode ID	Text	SD1
*IS	Reverse saturation current	Current	1e-11 mA
*JSW	Periphery reverse saturation current	Current	0 mA
*MULT	Scaling factor		1
*AFAC	Junction area		1
*PJFAC	Junction periphery		1
*RS	Series resistance	Resistance	0.001 ohm
*N	Ideality factor		1
*TT	Storage time	Time	0 ns
*CJ0	Zero-voltage bottom junction capacitance	Capacitance	0 pF
*CJP	Zero-voltage periphery junction capacitance	Capacitance	0 pF
*VJ	Built-in voltage	Voltage	0.8 V
*PHP	Periphery built-in voltage	Voltage	0.8 V
*M	Grading coefficient		0.5

Name	Description	Unit Type	Default
*MJSW	Periphery junction grading coefficient		0.33
*FC	Depletion capacitance linearization parameter		0.5
*FCS	Periphery depletion capacitance linearization parameter		0.5
*BV	Breakdown voltage	Voltage	1e+06 V
*IBV	Current at breakdown voltage	Current	1 mA
*IKF	Forward knee current	Current	0 mA
*IKR	Reverse knee current	Current	0 mA
*EG	Energy gap @ TNOM; default is Si		
*XTI	Temp scaling coefficient; default is Si PN		3.0
*TEXT	Temperature at which diode params were determined	Temperature	25 DegC
*T	Temperature	Temperature	25 DegC
*KF	Flicker noise coefficient		0
*AF	Flicker noise exponent		1.0
*FFE	Flicker noise frequency exponent		1.0
*KB	Burst noise coefficient		0.0
*AB	Burst noise exponent		1.0
*FB	Burst noise cutoff frequency		1.0
*NFLAG	Noise model		SPICE model
*DCAP	Capacitance model selector		1
*TLEV	I/V temperature model		0
*TLEVC	Capacitance temperature model		0
*CTA	Temperature coefficient for CJ0		0
*CTP	Temperature coefficient for CJP		0
*GAP1	First bandgap correction factor		0.000702
*GAP2	Second bandgap correction factor		1108
*TCV	Breakdown voltage temperature coefficient		0
*TM1	First-order temperature coefficient for M		0
*TM2	Second-order temperature coefficient for M		0
*TPB	Temperature coefficient for VJ		0
*TPHP	Temperature coefficient for PHP		0
*TRS	Resistance temperature coefficient		0
*TTT1	First-order temperature coefficient for TT		0
*TTT2	Second-order temperature coefficient for TT		0
*COMPAT	Compatibility selector: HPSICE		AWR
*IMAX	Maximum device current	Current	1e6 mA
*NS	Ideality factor for periphery diode		
*RSW	Sidewall series resistance	Resistance	

Name	Description	Unit Type	Default
*GLEAK	Bottom junction leakage conductive	S	
*GLEAKSW	Sidewall junction leakage conductive	S	

** indicates a secondary parameter*

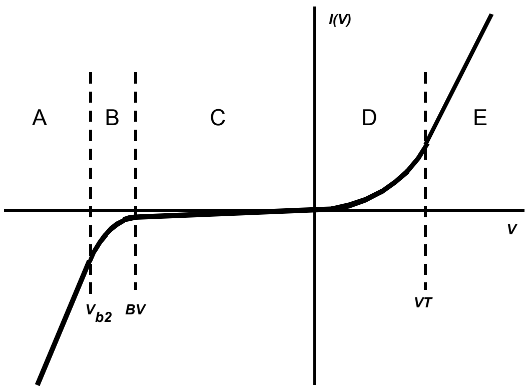
Implementation Details

SDIODE started as an implementation of the D-device level 1of SPICE2. It has progressively been enhanced to the point that it now includes most of HSPICE extensions to the original diode model. Accordingly, this model is mapped into HSPICE as a D-device with parameter LEVEL set to 1.

Two parameters, NFLAG and COMPAT, are unique to Microwave Office diode implementation. NFLAG's behavior is quite obvious and deserves no further comment. **COMPAT**, the compatibility selection flag, can take three different values: HSPICE, AWR and SPECTRE. The HSPICE and Spectre values are used to emulate the behavior of the D-device level 1 and diode model in HSPICE and Spectre, respectively. The AWR value supports NI AWR's implementation of the pn junction diode with parasitics.

I/V Characteristic

There are five operating regions, as follows:



Region A:

$$V_j < V_{b2}$$

$$I_j = -\mathrm{IS}(-\delta(\mathrm{BV} + V_j) + 1)\exp(30)$$

Region B: for

$$V_{b2} < V_j < -\mathrm{BV}$$

$$I_j = -\mathrm{IS} \cdot \exp(-\delta(\mathrm{BV} + V_j))$$

Region C: for

$$-\mathrm{BV} < V_j < 0$$

$$I_j = -\mathrm{IS}\cdot\delta\cdot V_j(1+\frac{\delta\cdot V_j}{2})$$

Region D: for

$$0 < V_j < V_{\mathrm{th}}$$

$$I_j = \mathrm{IS}\cdot(\exp(\delta V_j) - 1)$$

Region E: for

$$V_{\mathrm{th}} < V_j$$

$$I_j = \mathrm{IS}\cdot\exp(\delta V_{\mathrm{th}})\cdot(1+\delta(V_j - V_{\mathrm{th}}) + 0.5\delta^2(V_j - V_t h)^2)$$

where

$$\delta = \frac{q}{\mathrm{NKT}}$$

$$V_{\mathrm{th}} = \frac{1}{\delta}\ln(\frac{10^3}{\mathrm{IS}})$$

$$V_{b2} = \frac{-30}{\delta} - \mathrm{BV}$$

The equations are designed so that derivatives of the I/V characteristics are continuous at the boundaries.

Q/V CHARACTERISTIC

The junction capacitance is modeled as in the PNDCAP element. See PNDCAP for the equations.

NOISE MODEL

SDIODE noise comes from four uncorrelated sources: thermal (associated with the series resistance) shot, flicker and burst noise (associated with the junction). The thermal noise is described by a current source in parallel with the series resistance and with spectral density

$$S_{\mathrm{thermal}}(f) = 4\mathrm{kTG}_s$$

The shot, flicker and burst noise are described as current sources in parallel with the junction and with spectral densities:

$$S_{\mathrm{shot}}(f) = 2\mathrm{qI}_{\mathrm{DC}}$$

$$S_{\mathrm{flicker}}(f) = \mathrm{KF}\frac{I_{\mathrm{DC}}^{\mathrm{AF}}}{f^{\mathrm{FFE}}}$$

$$S_{\text{burst}}(f) = \text{KB}\frac{I_{\text{DC}}^{\text{AF}}}{1 + (\frac{f}{\text{FB}})^2}$$

k is Boltzman's constant, T is the temperature, q is the electron charge, I_{DC} is the DC component of junction current, and all spectral densities are expressed in A^2/Hz.

NOTE: This model can be used as a Schottky-barrier diode model when TT, the transit time, is set to zero.

Layout

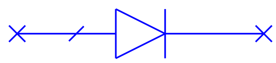
This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Reference

[1] P. Antognetti and G. Massobrio, Semiconductor Device Modeling with SPICE, New York: McGraw-Hill, 1988.

SPICE Geometric Junction Diode Model: SDIODEG

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Diode ID	Text	SDG1
*IS	Reverse saturation current	Current	1e-11 mA
*JSW	Periphery reverse saturation current	Current	0 mA
*MULT	Multiplier		1
*AFAC	Junction area		1
*PJFAC	Junction periphery		1
*SCALE	Scale factor		1
*SHRINK	Shrink factor		1
*L	Diode length	Length	0 um
*W	Diode width	Length	0 um
*XW	Masking or etching correction factor	Length	0 um
*LP	Diode length	Length	0 um
*WP	Diode width	Length	0 um
*XP	Poly masking or etching correction factor	Length	0 um
*LM	Diode length	Length	0 um
*WM	Diode width	Length	0 um
*XM	Metal masking or etching correction factor	Length	0 um
*XOI	Poly to bulk oxide thickness	Length	10,000um
*XOM	Metal to bulk oxide thickness	Length	10,000um
*RS	Series resistance	Resistance	0.001 ohm
*N	Ideality factor		1
*TT	Storage time	Time	0 ns
*CJ0	Zero-voltage bottom junction capacitance	Capacitance	0 pF
*CJP	Zero-voltage periphery junction capacitance	Capacitance	0 pF
*VJ	Bottom built-in voltage	Voltage	0.8 V
*PHP	Periphery built-in voltage	Voltage	0.8 V
*M	Bottom junction grading coefficient		0.5
*MJSW	Periphery junction grading coefficient		0.33
*FC	Bottom depletion capacitance linearization parameter		0.5
*FCS	Periphery depletion capacitance linearization parameter		0.5

Name	Description	Unit Type	Default
*BV	Breakdown voltage	Voltage	1000000 V
*IBV	Current at breakdown voltage	Current	1mA
*IKF	Forward knee current	Current	0 mA
*IKR	Reverse knee current	Current	0 mA
*EG	Energy gap @ TNOM; default is Si		1.16
*XTI	Temp scaling coefficient; default is Si PN		3
*TEXT	Temperature at which diode params were determined	Temperature	25 DegC
*T	Temperature	Temperature	25 DegC
*KF	Flicker noise coefficient		0
*AF	Flicker noise exponent		1
*FFE	Flicker noise frequency exponent		1
*KB	Burst noise coefficient		0
*AB	Burst noise exponent		1
*FB	Burst noise cutoff frequency		1
*NFLAF	Noise model		Noise On
*DCAP	Capacitance model selector		1
*TLEV	I/V temperature model		0
*TLEVC	Capacitance temperature model		0
*CTA	Temperature coefficient for CJ0		0
*CTP	Temperature coefficient for CJP		0
*GAP1	First bandgap correction factor		0.000702
*GAP2	Second bandgap correction factor		1108
*TCV	Breakdown voltage temperature coefficient		0
*TM1	First-order temperature coefficient for MJ		0
*TM2	Second-order temperature coefficient for MJ		0
*TPB	Temperature coefficient for VJ		0
*TPHP	Temperature coefficient for PHP		0
*TRS	Resistance temperature coefficient		0
*TTT1	First-order temperature coefficient for TT		0
*TTT2	Second-order temperature coefficient for TT		0
*COMPAT	Compatibility selector: HPSICE		AWR
*IMAX	Maximum device current	Current	1e6 mA
*NS	Ideality factor for periphery diode		1

** indicates a secondary parameter*

Implementation Details

SDIODEG was developed as and extension to SDIODE to support the parameters associated with pn junctions described in terms of their geometry. Accordingly, this model is mapped into HSPICE as a D-device with parameter LEVEL set to 3.

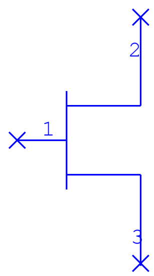
All SDIODE parameters are available in SDIODEG.

Layout

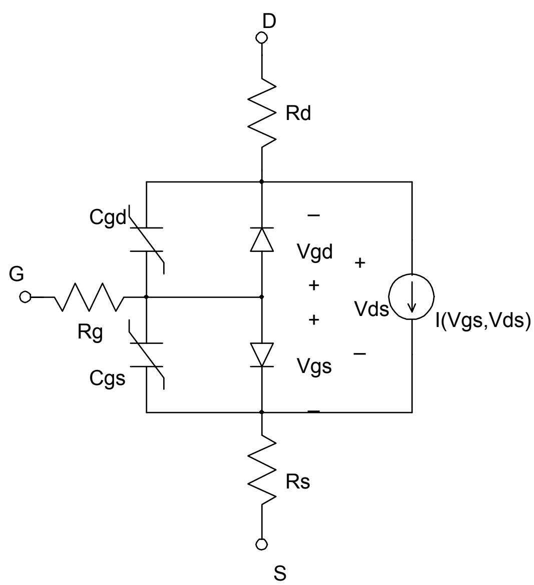
This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

SPICE JFET Model: SJFET

Symbol



Equivalent Circuit



Parameters

Name	Description	Unit Type	Default
ID	Device ID	Text	S1
*VTO	Threshold (pinch-off) voltage	Voltage	-2 V
*BETA	I/V BETA parameter		0.0001
*LAMBDA	Drain-to-source resistance parameter		0
*RD	Drain resistance	Resistance	0.001 ohm
*RS	Source resistance	Resistance	0.001 ohm
*RG	Gate resistance	Resistance	0.001 ohm
*CGS	Gate-source capacitance	Capacitance	0 pF

Name	Description	Unit Type	Default
*CGD	Gate-drain capacitance	Capacitance	0 pF
*PB	Gate built-in voltage	Voltage	1 V
*IS	Gate junction current parameter	Current	1e-11 mA
*FC	Depletion capacitance linearization parameter		0.5
*T	Temperature	Temperature	27 DegC
*KF	Flicker noise (1/f noise) coefficient		
*AF	Flicker noise (1/f noise) exponential term		
*FFE	Flicker noise (1/f noise) frequency exponent		
*AFAC	Gate-width scale factor		1
*NFLAG	Noise model	None	Spice Model
*BEX	Mobility temperature exponent		0
*MJ	Grading coefficent for G-S and G-D diodes		0.5
*N	Emission coefficent for G-S and G-D diodes		1
*TCV	Temperature compensation coefficient for VTO		0
*TNOM	Parameter extraction temperature	DegC	26.85
*XTI	Saturation current temperature exponent		
*COMPAT	Model compatibility selector		HSPICE
*LEVEL	Model level selector (default=1)		
*LAMBDA1	Gate dependence of channel length modulation parameter		
*EG	Energy gap at T=TNOM		
*GAP1	First bandgap correction factor		
*GAP2	Second bandgap correction factor		

** indicates a secondary parameter*

Implementation Details

I/V Characteristic

The JFET model is defined for both forward and reverse conduction. The equations for both cases are as follows:

Forward Operation (VDS > 0)

$$I_d = \begin{cases} 0 & V_{\mathrm{gs}} < \mathrm{VT0} \\ \beta \cdot V_{\mathrm{ds}}(2(V_{\mathrm{gs}} - \mathrm{VT0}) - V_{\mathrm{ds}})(1 + \lambda \cdot V_{\mathrm{ds}}) & V_{\mathrm{ds}} \le V_{\mathrm{gs}} - \mathrm{VT0} \\ \beta \cdot (V_{\mathrm{gs}} - \mathrm{VT0})^2(1 + \lambda \cdot V_{\mathrm{ds}}) & V_{\mathrm{ds}} > V_{\mathrm{gs}} - \mathrm{VTO} \end{cases}$$

Reverse Operation (VSD = -VDS > 0)

$$I_d = \begin{cases} 0 & V_{\text{gd}} < \text{VT0} \\ -\beta \cdot V_{\text{sd}}(2(V_{\text{gd}} - \text{VT0}) - V_{\text{sd}})(1 + \lambda \cdot V_{\text{sd}}) & V_{\text{sd}} \le V_{\text{gd}} - \text{VT0} \\ -\beta \cdot (V_{\text{gd}} - \text{VT0})^2(1 + \lambda \cdot V_{\text{sd}}) & V_{\text{sd}} > V_{\text{gd}} - \text{VTO} \end{cases}$$

Capacitance

The gate-to-source and gate-to-drain capacitances, C_{gs} and C_{gd}, are modeled as PNCAP elements. See PNCAP for the equations.

SCALING

Currents and capacitances are scaled in proportion to AFAC.

All resistances, including gate resistance, are scaled as 1/AFAC.

Gate Diodes

The gate-to-source and gate-to-drain diodes are modeled as PNIV elements. See PNIV for the equations.

NOTE: This model uses a few extra parameters that are not part of the SPICE parameter set. These are necessary for many RF applications. The default values make the model equivalent to the SPICE JFET model.

Layout

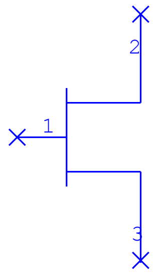
This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

Reference

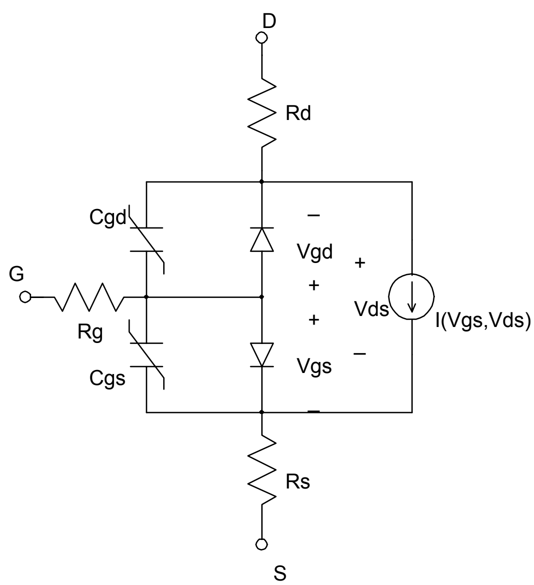
[1] P. Antognetti and G. Massobrio, Semiconductor Device Modeling with SPICE, New York: McGraw-Hill, 1988.

SPICE JFET P Model: SJFETP

Symbol



Topology



Parameters

Name	Description	Unit Type	Default
ID	Device ID	Text	S1
*VTO	Threshold (pinch-off) voltage	Voltage	-2 V
*BETA	I/V BETA parameter		0.0001
*LAMBDA	Drain-to-source resistance parameter		0
*RD	Drain resistance	Resistance	0.001 ohm
*RS	Source resistance	Resistance	0.001 ohm
*RG	Gate resistance	Resistance	0.001 ohm
*CGS	Gate-source capacitance	Capacitance	0 pF

Name	Description	Unit Type	Default
*CGD	Gate-drain capacitance	Capacitance	0 pF
*PB	Gate built-in voltage	Voltage	1 V
*IS	Gate junction current parameter	Current	1e-11 mA
*FC	Depletion capacitance linearization parameter		0.5
*T	Temperature	Temperature	27 DegC
*KF	Flicker noise (1/f noise) coefficient		
*AF	Flicker noise (1/f noise) exponential term		
*FFE	Flicker noise (1/f noise) frequency exponent		
*AFAC	Gate-width scale factor		1
*NFLAG	Noise Model		Spice Model
*BEX	Mobility temperature exponent		0
*MJ	Grading coefficent for G-S and G-D diodes		0.5
*N	Emission coefficent for G-S and G-D diodes		1
*TCV	Temperature compensation coefficient for VTO		0
*TNOM	Parameter extraction temperature	DegC	26.85
*XTI	Saturation current temperature exponent		
*COMPAT	Model compatibility selector		HSPICE
*LEVEL	Model level selector (default=1)		
*LAMBDA1	Gate dependence of channel length modulation parameter		
*EG	Energy gap at T=TNOM		
*GAP1	First bandgap correction factor		
*GAP2	Second bandgap correction factor		

** indicates a secondary parameter*

Implementation Details

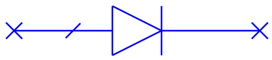
See SPICE JFET (SJFET) for equation.

Layout

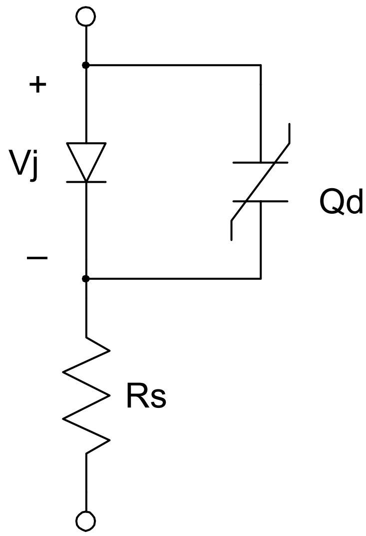
This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Zhang and Raisanen Step-Recovery Diode Model: SRD_ZR

Symbol



Equivalent Circuit



Parameters

Name	Description	Unit Type	Default
ID	Diode ID	Text	SRD1
IS	Reverse saturation current	Current	1e-11 mA
RS	Series resistance	Resistance	0.001 ohm
N	Ideality factor		1
CF	Forward capacitance	Capacitance	0
CR	Reverse-voltage capacitance	Capacitance	0
VJE	Built-in Voltage	Voltage	1.0
BV	Breakdown voltage	Voltage	1e+06 V
T	Temperature	Temperature	27 DegC
AFAC	Junction area scaling factor		1

Implementation Details

The diode I/V characteristic is identical to that of other Schottky junctions:

$$I_d = I_s(\exp(\frac{\mathrm{qV}_j}{\mathrm{NKT}}) - 1)$$

The expression uses a quadratic extrapolation at high current to prevent numerical overflow (see SDIODE).

The stored charge is given by

$$Q_d(V) = \begin{cases} C_F V_j & V_j < 0 \\ \frac{C_f - C_F}{2V_{\mathrm{je}}}(V_j + \frac{C_F V_{\mathrm{je}}}{C_f - C_F})^2 - \frac{C_F^2}{2(C_f - C_F)} V_{\mathrm{je}} & 0 \le V_j \le V_{\mathrm{je}} \\ C_f V_j - \frac{C_f - C_F}{2} & V_j > V_{\mathrm{je}} \end{cases}$$

I_d and Q_d are scaled in proportion to the area scaling factor, AFAC.

Layout

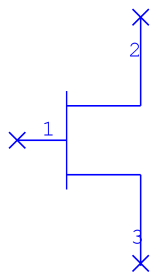
This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

References

[1] J. Zhang and A. Raisanen, "A New Model of a Step Recovery diode for CAD," IEEE MTT-S International Microwave Symposium Digest, 1995, p. 1459.

SPICE Statz FET Model: SSTATZ

Symbol



Summary

SSTATZ is a SPICE-equivalent version of the STATZ FET model. It should be used instead of the STATZ model for SPICE compatibility.

Parameters

Name	Description	Unit Type	Default
ID	Device ID	Text	SF1
*BETA	I/V Beta coefficient		0.01
*VTO	Threshold (pinch-off) voltage	Voltage	-2.5 V
*ALPHA	Drain I/V knee parameter		2
*LAMBDA	Output conductance parameter		0
*THETA	I/V parameter (b in Statz's paper)		1
*TAU	Gate-drain time delay	Time	0 ns
*VBR	Gate junction breakdown voltage	Voltage	1e+06 V
*IS	Gate diode current parameter	Current	1e-11 mA
*N	Gate diode ideality factor		1
*VBI	Gate junction built-in voltage	Voltage	1 V
*FC	Gate depletion cap. linearization parameter		0.5
*RC	RF drain-source resistance	Resistance	10^6 ohm
*CRF	Capacitance that determines Rds break frequency	Capacitance	0 pF
*RD	Drain resistance	Resistance	0.001 ohm
*RG	Gate resistance; fixed part	Resistance	0.001 ohm
*RS	Source resistance	Resistance	0.001 ohm
*RIN	Intrinsic resistance	Resistance	0.001 ohm
*CGSO	Gate-source capacitance parameter	Capacitance	0 pF
*CGDO	Gate-drain capacitance parameter	Capacitance	0 pF
*NFING	Number of fingers scale factor		1
*DELTA2	Capacitance DELTA2 parameter		0.2

Name	Description	Unit Type	Default
*CDS	Drain-source capacitance	Capacitance	0 pF
*CGS	Gate-source fixed capacitance	Capacitance	0 pF
*CGD	Gate-drain fixed capacitance	Capacitance	0 pF
*TNOM	Temperature	Temperature	27 DegC
*RGD	Gate-drain resistance	Resistance	0.001 ohm
*LS	Source inductance	Inductance	0 nH
*LG	Gate inductance	Inductance	0 nH
*LD	Drain inductance	Inductance	0 nH
*P	Noise par: P		
*Tg	Noise par: gate noise temp		
*KF	Flicker noise coefficient		
*AF	Flicker noise exponent		
*FFE	Flicker noise frequency exponent		
*NFLAG	Noise model		
AFAC	Gate width scale factor		1
VMAX	Capacitance limiting voltage	Voltage	0.5 V

** indicates a secondary parameter*

Implementation Details

The implementation is identical to Staz except for the following charge expressions:

$$Q_{gd} = \mathrm{CGS0} \cdot \frac{V_{gs} + V_{gd} - \sqrt{T1}}{2}$$

$$Q_{gs} = \begin{cases} \mathrm{CGS}[2 \cdot \mathrm{VBI}(1 - \sqrt{1 - \frac{V_{max}}{\mathrm{VBI}}}) + \frac{V_{new} - V_{max}}{\sqrt{1 - \frac{V_{max}}{\mathrm{VBI}}}}] & V_{new} \ge V_{max} \\ \mathrm{CGS}(2 \cdot \mathrm{VBI}(1 - \sqrt{1 - \frac{V_{new}}{\mathrm{VBI}}})) & V_{new} < V_{max} \end{cases}$$

where

$$V_{max} = \mathrm{FC} \cdot \mathrm{VBI}$$

$$V_{new} = \frac{1}{2}(V_{eff1} + \mathrm{VT} + \sqrt{(V_{eff1} - \mathrm{VT})^2 + (\mathrm{DELTA2})^2})$$

$$V_{eff1} = \frac{1}{2}(V_{gs} + V_{gd} + \sqrt{T1})$$

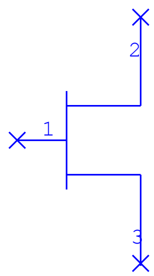
$$I = (V_{gs} - V_{gd})^2 + (\frac{1}{\mathrm{ALPHA}})^2$$

Layout

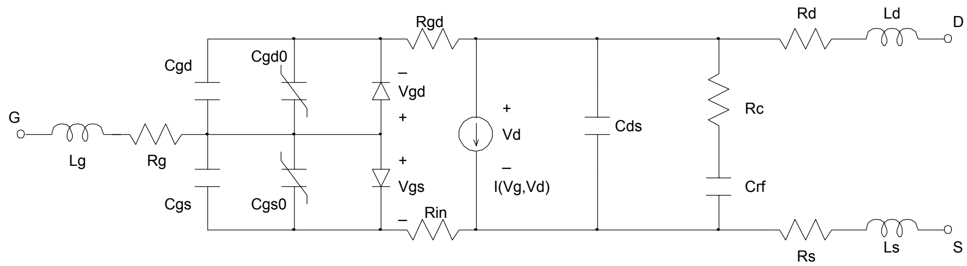
This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

STATZ FET Model: STATZ

Symbol



Equivalent Circuit



Parameters

Name	Description	Unit Type	Default
ID	Device ID	Text	SF1
*BETA	I/V Beta coefficient		0.01
*VTO	Threshold (pinch-off) voltage	Voltage	-2.5 V
*ALPHA	Drain I/V knee parameter		2
*LAMBDA	Output conductance parameter		0
*THETA	I/V parameter (b in Statz's paper)		1
*TAU	Gate-drain time delay	Time	0 ns
*VBR	Gate junction breakdown voltage	Voltage	1e+06 V
*IS	Gate diode current parameter	Current	1e-11 mA
*N	Gate diode ideality factor		1
*VBI	Gate junction built-in voltage	Voltage	1 V
*FC	Gate depletion cap. linearization parameter		0.5
*RC	RF drain-source resistance	Resistance	1e+06 ohm
*CRF	Capacitance that determines Rds break frequency	Capacitance	0 pF
*RD	Drain resistance	Resistance	0.001 ohm

Name	Description	Unit Type	Default
*RG	Gate resistance; fixed part	Resistance	0.001 ohm
*RS	Source resistance	Resistance	0.001 ohm
*RIN	Intrinsic resistance	Resistance	0.001 ohm
*CGSO	Gate-source capacitance parameter	Capacitance	0 pF
*CGDO	Gate-drain capacitance parameter	Capacitance	0 pF
*DELTA1	Capacitance DELTA1 parameter		0.3
*DELTA2	Capacitance DELTA2 parameter		0.2
*CDS	Drain-source capacitance	Capacitance	0 pF
*CGS	Gate-source fixed capacitance	Capacitance	0 pF
*CGD	Gate-drain fixed capacitance	Capacitance	0 pF
*TNOM	Temperature	Temperature	27 DegC
*RGD	Gate-drain resistance	Resistance	0.001 ohm
*LS	Source inductance	Inductance	0 nH
*LG	Gate inductance	Inductance	0 nH
*LD	Drain inductance	Inductance	0 nH
*P	Noise par: P		
*Tg	Noise par: gate noise temp		
*KF	Flicker noise coefficient		
*AF	Flicker noise exponent		
*FFE	Flicker noise frequency exponent		
*NFLAG	Noise model		
AFAC	Gate width scale factor		1
NFING	Number of fingers scale factor		1

** indicates a secondary parameter*

NOTES:

1. Use MKS units for parameters that have Scalar units. These are not adjusted to the default units of the project.
2. Gate resistance is scaled in proportion to AFAC / NFING.
3. Parasitic inductances are not scaled.

Implementation Details

I/V Characteristic

The equations for the Statz model are as follows:

$$I_d = \begin{cases} 0 & V_{gs} \le \mathrm{VT} \\ \dfrac{\beta(V_{gs}-\mathrm{VT})^2}{1+\theta(V_{gs}-\mathrm{VT})}(1-(1-\alpha\dfrac{V_{ds}}{3})^3)(1+\lambda\cdot V_{ds}) & V_{gs} > \mathrm{VT} \quad \text{and} \quad V_{ds} < \dfrac{3}{\alpha} \\ \dfrac{\beta(V_{gs}-\mathrm{VT})^2}{1+\theta(V_{gs}-\mathrm{VT})}(1+\lambda\cdot V_{\mathrm{DS}}) & V_{gs} > \mathrm{VT} \quad \text{and} \quad V_{ds} > \dfrac{3}{\alpha} \end{cases}$$

As with all other FET models, the gate voltage in the above equations is delayed by TAU.

Capacitance

The charge-voltage equations for the capacitance are given below. The C/V characteristics are found by differentiation:

$$Q_{gs} = \begin{cases} \mathrm{CGS}\left(2\cdot\mathrm{VBI}(1-\sqrt{1-\dfrac{V_{max}}{\mathrm{VBI}}})+\dfrac{V_{new}-V_{max}}{\sqrt{1-\dfrac{V_{max}}{\mathrm{VBI}}}}\right) & V_{new} \ge V_{max} \\ \mathrm{CGS}\left(2\cdot\mathrm{VBI}(1-\sqrt{1-\dfrac{V_{new}}{\mathrm{VBI}}})\right) & V_{new} < V_{max} \end{cases}$$

$$Q_{gd} = \mathrm{CGD}\cdot V_{eff2}$$

where

$$V_{max} = \mathrm{FC}\cdot\mathrm{VBI}$$

$$V_{new} = \frac{1}{2}(V_{eff1}+\mathrm{VT}+\sqrt{(V_{eff1}-\mathrm{VT})^2+(\mathrm{DELTA2})^2})$$

$$V_{eff1} = \frac{1}{2}(V_{gs}+V_{gd}+\sqrt{(V_{gs}-V_{gd})^2+(\mathrm{DELTA1})^2})$$

$$V_{eff2} = \frac{1}{2}(V_{gs}+V_{gd}-\sqrt{(V_{gs}-V_{gd})^2+(\mathrm{DELTA1})^2})$$

Gate Diodes

The gate-to-source and gate-to-drain diodes are modeled as PNIV elements. See PNIV for the equations.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Parameter Scaling

$$\mathrm{RG} \to \frac{\mathrm{RG}\cdot\mathrm{AFAC}}{\mathrm{NFING}^2}$$

$$\mathrm{RD} \to \frac{\mathrm{RD}}{\mathrm{AFAC}}$$

$$\mathrm{RS} \rightarrow \frac{\mathrm{RS}}{\mathrm{AFAC}}$$

$$R_{\mathrm{in}} \rightarrow \frac{R_{\mathrm{in}}}{\mathrm{AFAC}}$$

$$R_{\mathrm{gd}} \rightarrow \frac{R_{\mathrm{gd}}}{\mathrm{AFAC}}$$

$$R_c \rightarrow \frac{R_c}{\mathrm{AFAC}}$$

$$\mathrm{Crf} \rightarrow \mathrm{Crf} \cdot \mathrm{AFAC}$$

$$\mathrm{Cds} \rightarrow \mathrm{Cds} \cdot \mathrm{AFAC}$$

$$\mathrm{Cgs} \rightarrow \mathrm{Cgs} \cdot \mathrm{AFAC}$$

$$\mathrm{Cgd} \rightarrow \mathrm{Cgd} \cdot \mathrm{AFAC}$$

Nonlinear currents and charges are multiplied by AFAC.

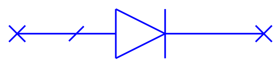
References

[1] H. Statz et al., "GaAs FET Device and Circuit Simulation in SPICE," IEEE Trans. Electron Devices, Vol. ED-34, 1987, p. 160.

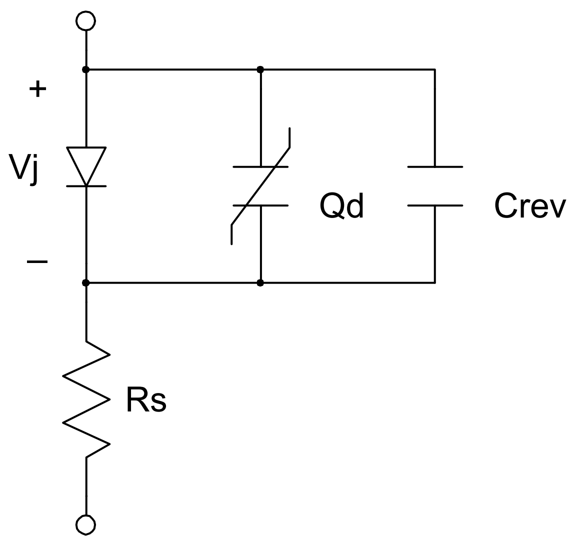
[2] D. Divekar, "Comments on 'GaAs FET Device and Circuit Simulation in SPICE'," IEEE Trans. Electron Devices, Vol. ED-34, 1987, p. 2564.

Step-Recovery Diode Model: STEPRD

Symbol



Equivalent Circuit



Parameters

Name	Description	Unit Type	Default
ID	Diode ID	Text	SRD1
IS	Reverse saturation current	Current	1e-11 mA
RS	Series resistance	Resistance	0.001 ohm
N	Ideality factor		1
TT	Storage time	Time	0 ns
CREV	Reverse-voltage capacitance	Capacitance	0 pF
BV	Breakdown voltage	Voltage	1e+06 V
T	Temperature	Temperature	27 DegC
AFAC	Junction area scaling factor		1

Implementation Details

The diode I/V characteristic is identical to that of other Schottky junctions:

$$I_d = I_s(\exp(\frac{qV_j}{NKT}) - 1)$$

The expression uses a quadratic extrapolation at high current to prevent numerical overflow (see SDIODE). The stored charge is given by

$$Q_d(V) = T_t I_d$$

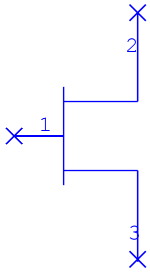
where T_t is the junction charge storage time. I_d is scaled in proportion to the area scaling factor, AFAC. This scales the stored charge Q_d. C_{rev} is a linear capacitance. It also is scaled in proportion to AFAC.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

SPICE TOM1 FET Model: STOM1

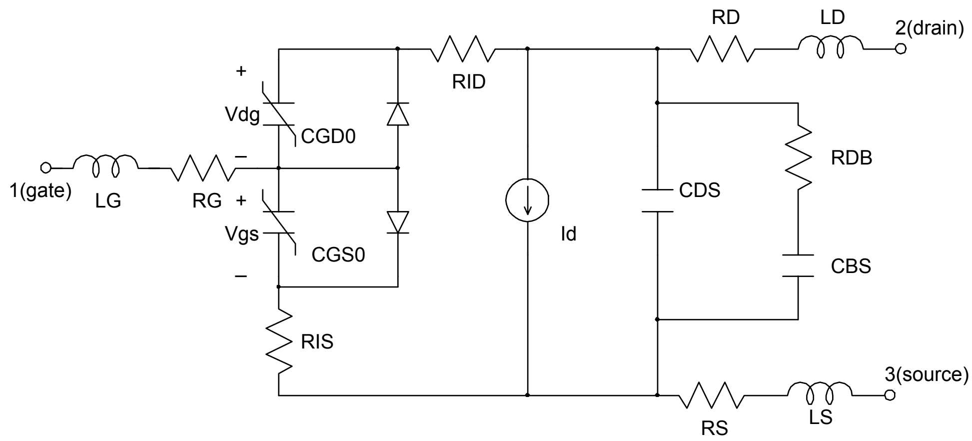
Symbol



Summary

STOM1 is a SPICE-equivalent implementation of the TOM1 model. It should be used where precise SPICE compatibility is needed.

Equivalent Circuit



Parameters

Name	Description	Unit Type	Default
ID	Device ID	Text	F1
*VTO	Nonscalable portion of Vt	Voltage	-2.5 V
*VTOSC	Scalable portion of Vt	Voltage	0 V
*ALPHA	Drain I/V knee parameter		2
*BETA	I/V Beta coefficient		0.01
*GAMMA	AC drain term		0
*GAMMADC	DC drain term		0
*Q	I/V exponent		2

Name	Description	Unit Type	Default
*DELTA	I/V DELTA term		0
*VBI	Gate junction built-in voltage	Voltage	1 V
*IS	Gate diode current parameter	Current	10[a] mA
*N	Gate diode ideality factor		1
*RIS	Gate-source intrinsic resistance	Resistance	0.001 ohm
*RID	Gate-drain resistance	Resistance	0.001 ohm
*TAU	Gate-drain time delay	Time	0 ns
*CDS	Drain-source capacitance	Capacitance	0 pF
*RDB	RF drain-source resistance	Resistance	10^{6} Ohm
*CBS	Capacitance that determines Rds break frequency	Capacitance	0 pF
*CGSO	Gate-source capacitance parameter	Capacitance	0 pF
*CGDO	Gate-drain capacitance parameter	Capacitance	0 pF
*VMAX	Capacitance limiting voltage		0.5
*DELTA2	Capacitance DELTA2 parameter		0.2
*FC	Gate depletion cap. linearization parameter		0.5
*VBR	Gate junction breakdown voltage	Voltage	10^{6} V
*RG	Gate resistance; nonscalable part	Resistance	0.001 ohm
*RD	Drain resistance	Resistance	0.001 ohm
*RS	Source resistance	Resistance	0.001 ohm
*RGMET	Gate metalization res.; scales with AFAC & NFING	Resistance	0.001 ohm
*TNOM	Temperature	Temperature	27 DegC
*LS	Source inductance	Inductance	0 nH
*LG	Gate inductance	Inductance	0 nH
*LD	Drain inductance	Inductance	0 nH
*P	Noise par: P		
*Tg	Noise par: gate noise temp		
*KF	Flicker noise coefficient		
*AF	Flicker noise exponent		
*FFE	Flicker noise frequency exponent		
*NFLAG	Noise model		
AFAC	Gate width scale factor		1
NFING	Number of fingers scale factor		1

[a]

** indicates a secondary parameter*

Implementation Details

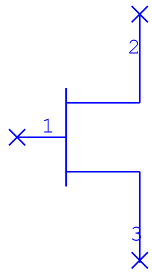
The implementation is identical toTOM1 except for the charge expressions, which are identical to SSTATZ.

Layout

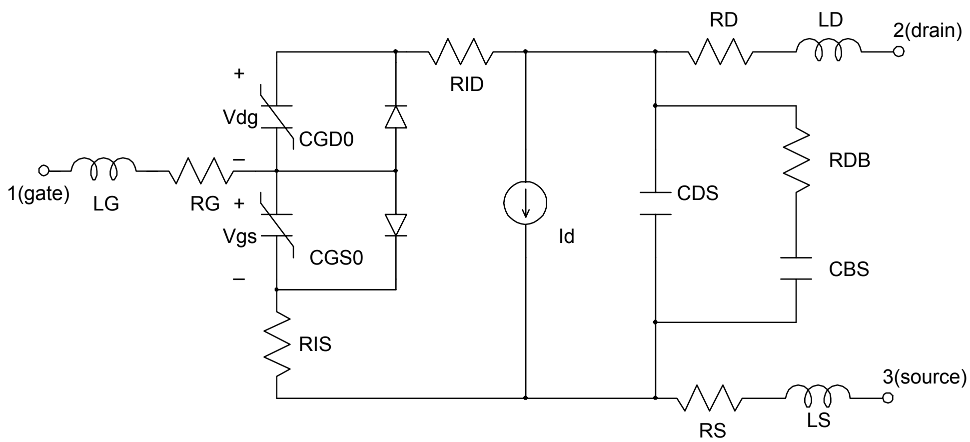
This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

TOM1 FET Model: TOM1

Symbol



Equivalent Circuit



Parameters

Name	Description	Unit Type	Default
ID	Device ID	Text	F1
*VTO	Non-scalable portion of Vt	Voltage	-2.5 V
*VTOSC	Scalable portion of Vt	Voltage	0 V
*ALPHA	Drain I/V knee parameter		2
*BETA	I/V Beta coefficient		0.01
*GAMMA	AC drain term		0
*GAMMADC	DC drain term		0
*Q	I/V exponent		2
*DELTA	I/V DELTA term		0
*VBI	Gate junction built-in voltage	Voltage	1 V
*IS	Gate diode current parameter	Current	10^{-11} mA

Name	Description	Unit Type	Default
*N	Gate diode ideality factor		1
*RIS	Gate-source intrinsic resistance	Resistance	0.001 ohm
*RID	Gate-drain resistance	Resistance	0.001 ohm
*TAU	Gate-drain time delay	Time	0 ns
*CDS	Drain-source capacitance	Capacitance	0 pF
*RDB	RF drain-source resistance	Resistance	10^6 ohm
*CBS	Capacitance that determines Rds break frequency	Capacitance	0 pF
*CGSO	Gate-source capacitance parameter	Capacitance	0 pF
*CGDO	Gate-drain capacitance parameter	Capacitance	0 pF
*DELTA1	Capacitance DELTA1 parameter		0.3
*DELTA2	Capacitance DELTA2 parameter		0.2
*FC	Gate depletion cap. linearization parameter		0.5
*VBR	Gate junction breakdown voltage	Voltage	10^6 V
*RG	Gate resistance; non-scalable part	Resistance	0.001 ohm
*RD	Drain resistance	Resistance	0.001 ohm
*RS	Source resistance	Resistance	0.001 ohm
*RGMET	Gate metallization resistance; scales with AFAC & NFING	Resistance	0.001 ohm
*TNOM	Temperature	Temperature	27 DegC
*LS	Source inductance	Inductance	0 nH
*LG	Gate inductance	Inductance	0 nH
*LD	Drain inductance	Inductance	0 nH
*P	Noise par: P		
*Tg	Noise par: gate noise temp		
*KF	Flicker noise coefficient		
*AF	Flicker noise exponent		
*FFE	Flicker noise frequency exponent		
*NFLAG	Noise model		
AFAC	Gate width scale factor		1
NFING	Number of fingers scale factor		1

** indicates a secondary parameter*

Implementation Details

I/V Characteristic

The equations for the model are as follows:

$$I_d = \frac{I_{d0}}{1 + \mathrm{DELTA} \cdot I_{d0}}$$

where

$$V_t = \mathrm{VT0} + \frac{\mathrm{VT0SC}}{\mathrm{AFAC}} - \gamma V_{\mathrm{ds}}$$

for

$$V_{\mathrm{ds}} < \frac{3}{\alpha}$$

$$I_{d0} = \beta(V_{\mathrm{gs}} - V_t)^e(1 - (1 - \frac{\alpha \cdot V_{\mathrm{ds}}}{3})^3)$$

for

$$V_{\mathrm{ds}} < \frac{3}{\alpha}$$

$$I_{d0} = \beta\left(V_{gs} - V_t\right)^e$$

V_{t0sc} is the portion of the pinch-off voltage that is scaled with device area.

I_{db} accounts for drain-current dispersion. Two components of I_d are computed, one for RF and one for DC:

$$I_{\mathrm{ds,DC}} = I_d(V_{\mathrm{gs}}, V_{\mathrm{ds}}, \mathrm{GAMMADC})$$

$$I_{\mathrm{ds,RF}} = I_d(V_{\mathrm{gs}}, V_{\mathrm{ds}}, \mathrm{GAMMA})$$

then

$$I_{\mathrm{db}} = I_{\mathrm{ds,RF}} - I_{\mathrm{ds,DC}}$$

As with all other FET models, the gate voltage in the above equations is delayed by TAU.

Capacitance

The charge-voltage equations for the capacitance, given below, are identical to those of the Statz model. The C/V characteristics are found by differentiation:

$$Q_{\mathrm{gd}} = \mathrm{CGD0} \cdot V_{\mathrm{eff2}}$$

$$Q_{\mathrm{gs}} = \begin{cases} \mathrm{CGS0}[2 \cdot \mathrm{VBI}(1 - \sqrt{1 - \frac{V_{\mathrm{max}}}{\mathrm{VBI}}}) + \frac{V_{\mathrm{new}} - V_{\mathrm{max}}}{\sqrt{1 - \frac{V_{\mathrm{max}}}{\mathrm{VBI}}}}] & V_{\mathrm{new}} > V_{\mathrm{max}} \\ \mathrm{CGS0}(2 \cdot \mathrm{VBI}(1 - \sqrt{1 - \frac{V_{\mathrm{new}}}{\mathrm{VBI}}})) & V_{\mathrm{new}} < V_{\mathrm{max}} \end{cases}$$

where

$$V_{\mathrm{max}} = \mathrm{FC} \cdot \mathrm{VBI}$$

$$V_{\text{new}} = \frac{1}{2}(V_{\text{eff1}} + \text{VT0} + \sqrt{(V_{\text{eff1}} - \text{VT})^2 + (\text{DELTA2})^2})$$

$$V_{\text{eff1}} = \frac{1}{2}(V_{\text{gs}} + V_{\text{gd}} + \sqrt{(V_{\text{gs}} - V_{\text{gd}})^2 + (\text{DELTA1})^2})$$

$$V_{\text{eff2}} = \frac{1}{2}(V_{\text{gs}} + V_{\text{gd}} - \sqrt{(V_{\text{gs}} - V_{\text{gd}})^2 + (\text{DELTA1})^2})$$

Gate Diodes

The gate-to-source and gate-to-drain diodes are modeled as PNIV elements. See PNIV and SDIODE for the equations.

Parameter Scaling

Parasitic inductances LG, LS, and LD are not scaled.

Currents and nonlinear charges are scaled in proportion to AFAC.

Other parameters are scaled as follows:

$$R_g = (\frac{\text{RGMET} \cdot \text{AFAC}}{\text{NFING}^2} + \text{RG})$$

$$\text{RD} \rightarrow \frac{\text{RD}}{\text{AFAC}}$$

$$\text{RS} \rightarrow \frac{\text{RS}}{\text{AFAC}}$$

$$\text{RIS} \rightarrow \frac{\text{RIS}}{\text{AFAC}}$$

$$\text{RID} \rightarrow \frac{\text{RID}}{\text{AFAC}}$$

$$\text{RDB} \rightarrow \frac{\text{RDB}}{\text{AFAC}}$$

$$\text{CDS} \rightarrow \text{CDS} \cdot \text{AFAC}$$

Layout

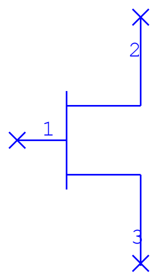
This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

Reference

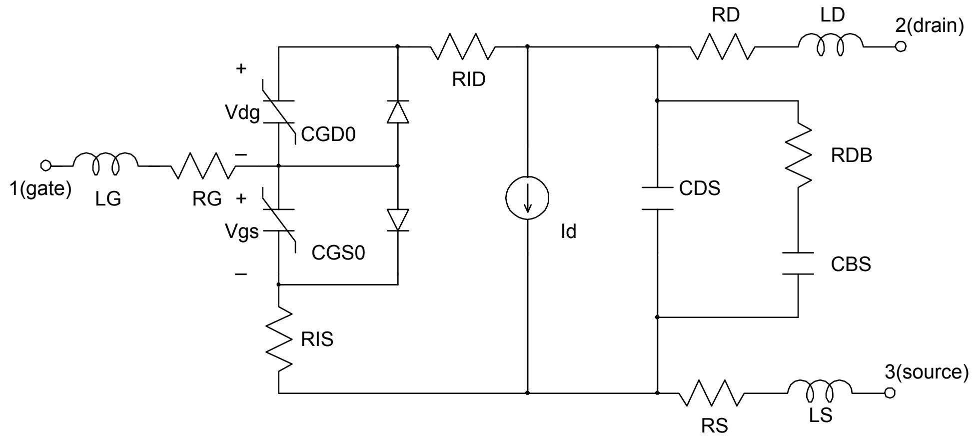
[1] A. J. McCant, G. D. McCormack, and D. H. Smith, "An Improved GaAs FET Model for SPICE," IEEE Trans. Microwave Theory Tech., Vol. MTT-38, p. 822, June, 1990.

TOM2 FET Model: TOM2

Symbol



Equivalent Circuit



Parameters

Name	Description	Unit Type	Default
ID	Device ID	Text	F1
*VTO	Threshold voltage	Voltage	-2.5 V
*ALPHA	Knee voltage parameter		2
*BETA	Transconductance parameter		0.1
*GAMMA	Threshold-shifting parameter		0
*DELTA	Output feedback parameter		0.2
*Q	Power-law parameter		2
*NG	Subthreshold slope gate parameter		0
*ND	Subthreshold slope drain pull parameter		0
*TAU	Conduction current time delay	Time	0 ns
*RG	Gate resistance	Resistance	0.001 ohm

Name	Description	Unit Type	Default
*RGSH	Gate resistance component	Resistance	0 ohm
*RD	Drain ohmic resistance	Resistance	0.001 ohm
*RS	Source ohmic resistance	Resistance	0.001 ohm
*IS	Gate diode saturation current	Current	10^{-11} mA
*N	Gate diode ideality factor		1
*VBI	Gate diode built-in potential	Voltage	1 V
*VDELTA	Capacitance transition voltage	Voltage	0.2 V
*VMAX	Gate-diode capacitance limiting voltage	Voltage	0.95 V
*CGD	Gate-drain	Capacitance	0 pF
*CGS	Gate-source	Capacitance	0 pF
*CDS	Drain-source capacitance	Capacitance	0 pF
*RIS	Gate-source intrinsic resistance	Resistance	0.01 ohm
*RID	Gate-drain resistance	Resistance	0.01 ohm
*VBR	Gate junction breakdown voltage	Voltage	10^6 V
*RDB	RF drain-source resistance	Resistance	10^6 ohm
*CBS	Capacitance that determines Rds break frequency	Capacitance	0 pF
*T	Device temperature	Temperature	27 DegC
*TNOM	Temperature at which parameters were extracted	Temperature	27 DegC
*LS	Source inductance	Inductance	0 nH
*LG	Gate inductance	Inductance	0 nH
*LD	Drain inductance	Inductance	0 nH
AFAC	Gate width scale factor (total width, not per finger)		1
NFING	Number of fingers scale factor		1
*EG	Gate diode energy gap		1.11
*XTI	Gate diode ideality factor		0
*VTOTC	VTO temperature coefficient		0
*VBITC	VBI temperature coefficient		0
*ALPHATCE	ALPHA temperature coefficient		0
*BETATCE	BETA temperature coefficient		0
*GAMMATC	GAMMA temperature coefficient		0
*TRG1	RG temperature coefficient		0
*TRD1	RD temperature coefficient		0
*TRS1	RS temperature coefficient		0
*CGDTCE	CGD temperature coefficient		0
*CGSTCE	CGS temperature coefficient		0
*P	Noise par: P		
*Tg	Noise par: gate noise temp		

Name	Description	Unit Type	Default
*KF	Flicker noise coefficient		0
*AF	Flicker noise frequency exponent		0
*FFE	Flicker noise frequency exponent		
*NFLAG	Noise model		

** indicates a secondary parameter*

Implementation Details

I/V Characteristic

The equations for the TOM2 model are as follows:

$$I_d = \frac{I_{\mathrm{ds0}}}{1 + \mathrm{DELTA} \cdot V_{\mathrm{ds}} \cdot I_{\mathrm{ds0}}}$$

$$I_{\mathrm{ds0}} = \mathrm{BETA} \cdot V_g^Q \cdot \frac{\mathrm{ALPHA} \cdot V_d}{\sqrt{1 + (\mathrm{ALPHA} \cdot V_d)^2}}$$

$$V_g = Q \cdot V_{\mathrm{st}} \cdot \mathrm{In}(\exp(\frac{V_{\mathrm{gs}} - \mathrm{VTO} + \mathrm{GAMMA} \cdot V_d}{Q \cdot V_{\mathrm{st}}}) + 1)$$

$$V_{\mathrm{st}} = (\mathrm{NG} + \mathrm{ND} \cdot V_{\mathrm{ds}}) \cdot \frac{K \cdot T}{q}$$

In the above equations, q is electron charge, K is Boltzmann's constant, and T is temperature in Kelvins.

Capacitance

The capacitance equations are the same as for the models STATZ and TOM1, except the parameters DELTA and DELTA2 are used instead of ALPHA and VDELTA. They are related as follows:

$$\mathrm{DELTA} = \frac{1}{\mathrm{ALPHA}}$$

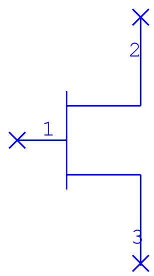
$$\mathrm{DELTA2} = \mathrm{VDELTA}$$

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

TOM3 MESFET Model: TOM3

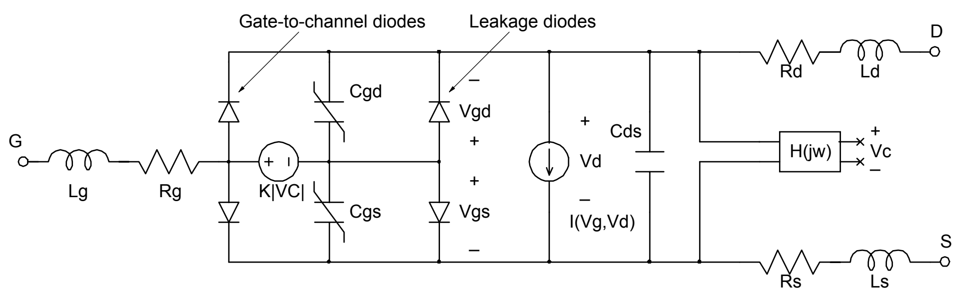
Symbol



Summary

TOM stands for TriQuint's Own Model. TOM3 is an implementation of a model developed by TriQuint, Inc., for their own foundry devices. Their models have become quite popular outside of the TriQuint foundry as well.

Equivalent Circuit



Parameters

Parameter	Description	Units	Default Value
ID	Device ID	Text	F1
*VTO	Threshold voltage	Voltage	-2.0
*ALPHA	Knee voltage parameter		2.0
*BETA	Transconductance parameter		0.1
*LAMBDA	Drain I/V slope in saturation		0.0
*GAMMA	Threshold-shifting parameter		0.0
*Q	Power-law parameter		2.0
*K	Output feedback parameter		2.0
*VST	Subthreshold slope	Voltage	1.0
*MST	Subthreshold slope drain parameter		0.0
*ILK	Leakage diode saturation current	Current	0.0

Parameter	Description	Units	Default Value
*PLK	Leakage diode potential	Voltage	1.0
*QGQH	Charge parameter		$-2.0x10^{-16}$
*QGSH	Charge parameter		$1.0x10^{-16}$
*QGDH	Charge parameter		0.0
*QGI0	Charge parameter		$1x10^{-6}$
*QGQL	Charge parameter		$5.0x10^{-16}$
*QGAG	Charge parameter		1.0
*QGAD	Charge parameter		1.0
*QGCL	Charge parameter		$2.0x10^{-16}$
*QGGB	Charge parameter		100.0
*QGG0	Charge parameter		0.0
*CDS	Drain-source capacitance	Capacitance	0.0
*IS	Gate diode saturation current	Current	$1.0x10^{-14}$
*EG	Barrier height	Voltage	0.8
*N	Gate diode ideality factor		1.0"
*XTI	Diode current temperature parameter		2.0
*TAU	Drain current time delay	Time	0.0
*VBI	(Not implemented)		
*TAU_GD	Time constant of the drain dispersion circuit	Time	$1.0x10^{-6}$
*KGAMMA	Feedback voltage source parameter		0.0
*RG	Gate resistance	Resistance	0.01
*RGSH	Gate resistance component	Resistance	0.0
*RD	Drain ohmic resistance	Resistance	0.01
*RS	Source ohmic resistance	Resistance	0.01
*LS	Source inductance	Inductance	0.0
*LG	Gate inductance	Inductance	0.0
*LD	Drain inductance	Inductance	0.0
NG	Number of gate fingers scale factor		1.0
TJ	Device temperature	Temperature	298.0
*TNOM	Reference (extraction) temperature	Temperature	298.0
*ALPHATCE	ALPHA temperature coefficient		0.0
*GAMMATC	GAMMA temperature coefficient		0.0
*VBITC	(Not implemented)		
*CGSTC	CGS temperature coefficient		0.0
*CGDTC	CGD temperature coefficient		0.0
*VTOTC	VT0 temperature coefficient		0.0
*VSTTC	VST temperature coefficient		0.0

Parameter	Description	Units	Default Value
*MSTTC	MST temperature coefficient		0.0
*VTOTC	VTO temperature coefficient		
*BETATCE	BETA temperature coefficient		0.0
*RGTC	RG temperature coefficient		0.0
*RDTC	RD temperature coefficient		0.0
*RSTC	RS temperature coefficient		0.0
*NOIS	(Not implemented)		0.0
*KF	(Not implemented)		0.0
*AF	(Not implemented)		0.0
*FCP	(Not implemented)		0.0
*SN	(Not implemented)		0.0
*NAME	(Not implemented)		0.0
W	Gate width per finger scale factor	Length	1.0

** indicates a secondary parameter*

Notes

The series RC elements in the drain, shown in [1] [11–343], do not model the "drain dispersion" effect directly. Instead, they are used only to create a transfer function for the controlled voltage source in the gate, which models the drain dispersion effect. Instead of using RC elements for this transfer function, the Microwave Office implementation uses a two-port block having the same transfer function. This results in better numerical conditioning of the model.

The model supports inverse operation.

Early documents describing this model had an error in the capacitance formulas. This error has been corrected in the Microwave Office implementation, but it appears not to have been corrected in other simulators. This error results in approximately a 10% to 20% error in Y parameters, especially Y_{11} and Y_{12}.

Be careful of default values of model parameters. Some of the parameters in the TriQuint implementation of this model are hard-coded into SPICE subcircuits, so these have no defaults; thus, there are no "standard" defaults. When porting model parameters from other simulators, do not assume that the default values of unspecified parameters are the same in both implementations.

Implementation Details

Equations

The equations for this model are too extensive to be listed here. See the References for more information

Scaling

Resistances, except R_g, are scaled as 1/AFAC.

TOM3 uses an unusual scaling formula for gate resistance. The scaled gate resistance, Rg, is

$$R_g = \mathrm{RG} \cdot \frac{W}{\mathrm{NG}} + \mathrm{RGSH} / (W \cdot \mathrm{NG})$$

RGSH is treated as if it were a sheet resistance in series with the gate. For conventional scaling, simply set RGSH = 0.

Capacitances and currents are scaled in proportion to AFAC.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

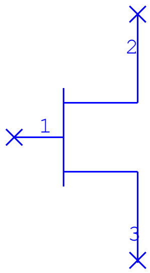
References

[1] R. B. Hallgren and D. S. Smith, "TOM3 Equations," Triquint Internal Report (unnumbered), 2 December 1999.

[2] R. B. Hallgren and P. H. Litzenberg, "TOM3 Capacitance Model: Linking Large- and Small-signal MESFET Models in SPICE," IEEE Trans. Microwave Theory Tech., vol. 47, no. 5, p. 556 (May, 1999).

TriQuint TX Modified Materka FET Model: TQ_MATRK

Symbol

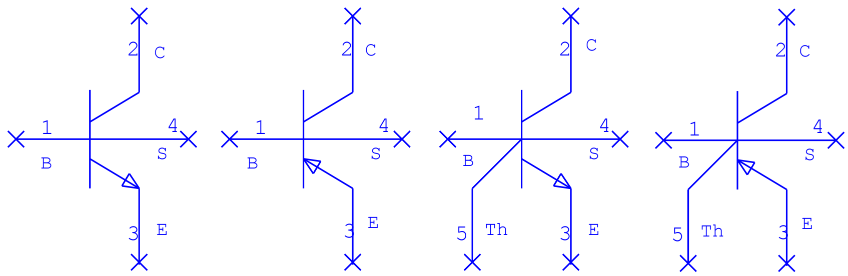


Summary

Contact NI AWR Support for additional information: `awr.support@ni.com`

VBIC BJT Model: VBIC

Symbol

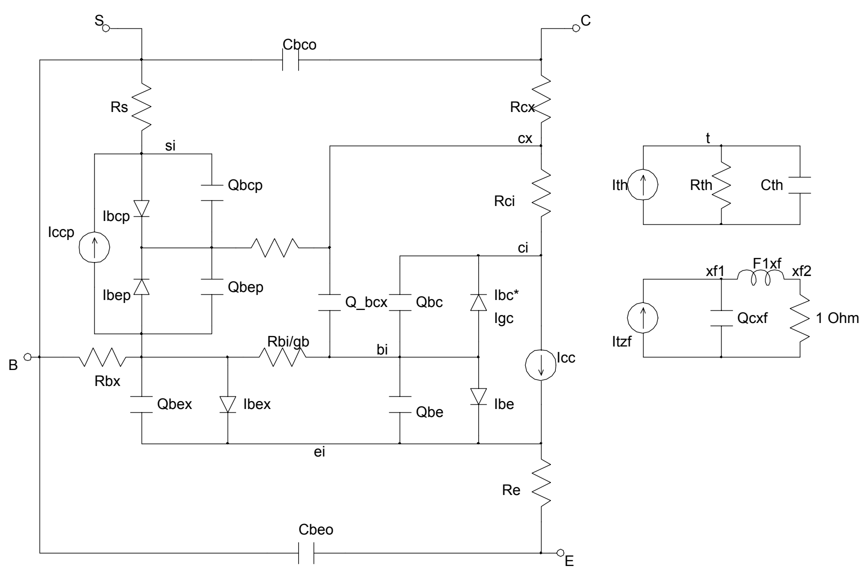


Summary

VBIC is a successor for both VBIC95N and VBIC95P, while it also provides access to the thermal node not available in the earlier implementations. Exposing the thermal node allows modeling thermal coupling among different devices.

The Vertical Bipolar Inter-Company (VBIC) model was developed to be an industry standard replacement for the SPICE Gummel-Poon (SGP) general-purpose BJT model. The VBIC model offers improved modeling of the early effect, quasi-saturation, substrate and oxide parasitics, avalanche multiplication and temperature behavior including self-heating. The improved modeling of these effects can be selectively invoked depending on the model parameters chosen. Since the VBIC model was developed to utilize the extraction techniques of the SGP model, it is possible that VBIC can degenerate to be similar to the SGP model. The MWO implementations of the VBIC model are largely based on the Saber VBIC95 Model Definition, Ver. 1.1.5, which has been in the public domain since June 1996.

Equivalent Circuit



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	VB1
*TNOM	Extraction temperature	DegC	25
*TEMP	Baseplate temperature	DegC	25
*RCX	Extrinsic Collector Resistance	ohm	0.01
*RCI	Intrinsic Collector Resistance (must be >0)	ohm	0.01
*VO	Epi Drift Saturation Voltage	V	0
*GAMM	Epi Doping Parameter		0
*HRCF	High-Current RC Factor		1
*RBX	Extrinsic Base Resistance	ohm	0.01
*RBI	Intrinsic Base Resistance (must be > 0)	ohm	0.01
*RE	Emitter Resistance	ohm	0.01
*RS	Substrate Resistance	ohm	0.01
*RBP	Parasitic Base Resistance (must be >0)	ohm	0.01
*IS	Transport Saturation Current	mA	1e-13
*NF	Forward Emission Coefficient		1
*NR	Reverse Emission Coefficient		1
*FC	Forward-Bias Junction Threshold Capacitance (0.05<FC<0.95)		0.9
*CBEO	Base-Emitter Small Signal Capacitance	pF	0
*CJE	Base-Emitter Zero-Bias Junction Capacitance	pF	0
*PE	Base-Emitter Grading Coefficient	V	0.75
*ME	Base-Emitter Junction Exponent		0.33
*AJE	Base-Emitter Capacitance Smoothing Factor		-0.5
*CBCO	Extrinsic Base-Collector Overlap Capacitance	pF	0
*CJC	Base-Collector Zero-Bias Capacitance	pF	0
*QCO	Collector Charge at Zero Bias		0
*CJEP	Extrinsic Base-Emitter Zero-Bias Capacitance	pF	0
*PC	Base-Collector Grading Coefficient	V	0.75
*MC	Base-Collector Junction Exponent		0.33
*AJC	Base-Collector Capacitance Smoothing Factor		-0.5
*CJCP	Extrinsic Base-Collector Zero-Bias Capacitance	pF	0
*PS	Collector-Substrate Grading Coefficient	V	0.75
*MS	Collector-Substrate Junction Exponent		0.33
*AJS	Collector-Substrate Capacitance Smoothing Factor		-0.5
*IBEI	Ideal Base-Emitter Saturation Current	mA	1e-15

Name	Description	Unit Type	Default
*WBE	Portion of Ibei from Vbei, (1-Wbe) is Vbex		1
*NEI	Ideal Base-Emitter Emission Coefficient		1
*IBEN	Non-Ideal Base-Emitter Saturation Current	mA	0
*NEN	Non-Ideal Base-Emitter Emission Coefficient		2
*IBCI	Ideal Base-Collector Saturation Current	mA	1e-13
*NCI	Ideal Base-Collector Emission Coefficient		1
*IBCN	Non-Ideal Base-Collector Saturation Current	mA	0
*NCN	Non-Ideal Base-Collector Emission Coefficient		2
*ISP	Parasitic Transport Saturation Current	mA	0
*WSP	Portion of Iccp from Vbep, 1-Wsp from Vbci		1
*NFP	Parasitic Forward Emission Coefficient		1
*IBEIP	Ideal Parasitic Base-Emitter Saturation Current	mA	0
*IBENP	Non-Ideal Parasitic Base-Emitter Saturation Current	mA	0
*IBCIP	Ideal Parasitic Base-Collector Saturation Current	mA	0
*NCIP	Ideal Parasitic Base-Collector Emission Coefficient		1
*IBCNP	Non-Ideal Parasitic Base-Collector Saturation Current	mA	0
*AVC1	Base-Collector Weak Avalanche Parameter 1		0
*AVC2	Base-Collector Avalanche Parameter 2		0
*NCNP	Non-Ideal Parasitic Base-Collector Emission Coefficient		2
*VEF	Forward Early Voltage	V	1000000000
*VER	Reverse Early Voltage	V	1000000000
*IKF	Forward Knee Current	mA	1000000000000
*IKR	Reverse Knee Current	mA	1000000000000
*IKP	Parasitic Knee Current	mA	1000000000000
*TF	Forward Transit Time (must be >0)	ns	0
*QTF	Variation of Tf with Base Width Modulation		0
*XTF	Coefficient of Tf Bias Dependence		0
*VTF	Coefficient of Tf Dependence on Vbc	V	0
*ITF	Coefficient of Tf Dependence on Icc	mA	0
*TR	Ideal Reverse Transit Time (must be >0)	ns	0
*TD	Forward Excess-Phase Delay Time (must be >0)	ns	0
*KFN	Flicker Noise Coefficient		0
*AFN	Flicker Noise Exponent		1
*BFN	Flicker Noise Frequency Exponent		1
*NFLAG	Noise analysis flag,0: Noise off, 1: Noise on		1
*XRE	Temperature Exponent of Emitter Resistance		0
*XRB	Temperature Exponent of Base Resistance		0

Name	Description	Unit Type	Default
*XRC	Temperature Exponent of Collector Resistance		0
*XRS	Temperature Exponent of Substrate Resistance		0
*XVO	Temperature Exponent of Vo		0
*EA	Activation Energy for Is (eV)		1.12
*EAIE	Activation Energy for Ibei (eV)		1.12
*EAIC	Activation Energy for Ibc/Ibeip (eV)		1.12
*EAIS	Activation Energy for Ibcip (eV)		1.12
*EANE	Activation Energy for Iben (eV)		1.12
*EANC	Activation Energy for Ibcn/Ibenp (eV)		1.12
*EANS	Activation Energy for Ibcnp (eV)		1.12
*XIS	Temperature Exponent of Is		3
*XII	Temperature Exponent of Ibei/Ibci/Ibeip/Ibcip		3
*XIN	Temperature Exponent of Iben/Ibcn/Ibenp/Ibcnp		3
*TNF	Temperature Coefficient of Nf		0
*TAVC	Temperature Coefficient of Avc		0
*RTH	Thermal Resistance	ohm	0
*CTH	Thermal Capacitance	pF	1000000000000
*IMAX	Explosion Current (not Implemented)	mA	1000
MULT	Area scale factor		1
*VERSION	Model Version		1.15
SELFT	Self-heating support		on
TYPE	Device type		NPN
*COMPAT	Model compatibility selector		AWR
*XRBX	Temperature exponent of extrinsic resistance RBX		0
*XRCX	Temperature exponent of extrinsic resistance RCX		0
*XRBP	Temperature exponent of extrinsic resistance RBP		0
*XIKF	Temperature exponent of IKF		0
*ISRR	Reverse saturation current factor		1.0
*XISR	Temperature exponent of ISRR		0
*DEAR	Activation energy for ISRR (eV)		0
*EAP	Activation energy for ISP (eV)		1.12
*VBBE	Base-Emitter breakdown voltage, zero means infinity		0
*TVBBE1	First temperature coefficient of VBBE		0
*TVBBE2	Second temperature coefficient of VBBE		0
*NBBE	Base-Emitter breakdown emission coefficient		1.0
*TNBBE	Temperature coefficient of NBBE		0
*IBEE	Base-Emitter breakdown current		1.0e-6

Name	Description	Unit Type	Default
*QBM	Selector for SGP qb formulation		0
*NKF	High current roll-off coefficient		0.5
*VRT	B-C reach-through limiting voltage (0 means infinity)		0
*ART	B-C reach-through limiting smoothing factor		0.1
*CCSO	Fixed collector-substrate capacitance		0

** indicates a secondary parameter*

Implementation Details

The TYPE parameter controls whether the device is NPN or PNP. The SELFT parameter is used to enable/disable the self-heating modeling capabilities. The current setting of either parameter is immediately reflected by the device symbol. The MULT and TEMP parameters replace the AFAC and TAMB parameters of the VBIC95N and VBIC95 elements, respectively. The VERSION parameter, whose default value is 1.15, is added as a provision to supporting additional versions of the model.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Operating Points

The following letter pairs identify the NL branches: ce, be, bc, bex, bb, bbp, cc, bcp, bep, and, cep. These are used to identify branch-related operating point information (branch voltages, currents, etc.).

Parameter	Description
Igc (Current)	Avalanche current
gmu (Conductance)	Feedback transconductance
gm (Conductance)	Transconductance
go	Output conductance
ro	Output resistance
rx (Resistance)	Base spreading resistance
cmu (Capacitance)	Feedback capacitance
cpi (Capacitance)	Base capacitance
gpi(Resistance)	Base conductance
rpi(Resistance)	Base resistance
dic_dvbe(Conductance)	
dic_dvbc (Conductance)	
dib_dvbe (Conductance)	
dib_dvbc (Conductance)	
rbi (Resistance)	Resistance in bb branch
rci (Resistance)	Resistance in cc branch

Parameter	Description
rbp (Resistance)	Resistance in bbp branch
cjc (Capacitance)	Capacitance in bc branch
cje (Capacitance)	Capacitance in be branch
cbex (Capacitance)	Capacitance in bex branch
cbcx (Capacitance)	Capacitance in bcx branch
cbep (Capacitance)	Capacitance in bep branch
cbcp (Capacitance)	Capacitance in bcp branch
betaac	AC current amplification
betadc	DC current amplification
temp (Temperature)	Simulation temperature
ft (Frequency)	Cutoff frequency approximation
pwr (Power)	Dissipated power

.

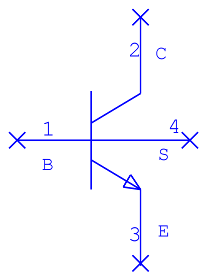
References

[1] C. McAndrew et al, VBIC95 - The Vertical Bipolar Inter-Company Model, IEEE Journal of Solid State Circuits, October, 1995.

[2] C. McAndrew et al (Compact Modeling Council), VBIC95 Model Definition: Release 1.1.5, July, 1996.

(Obsolete) VBIC95 NPN BJT Model: VBIC95N

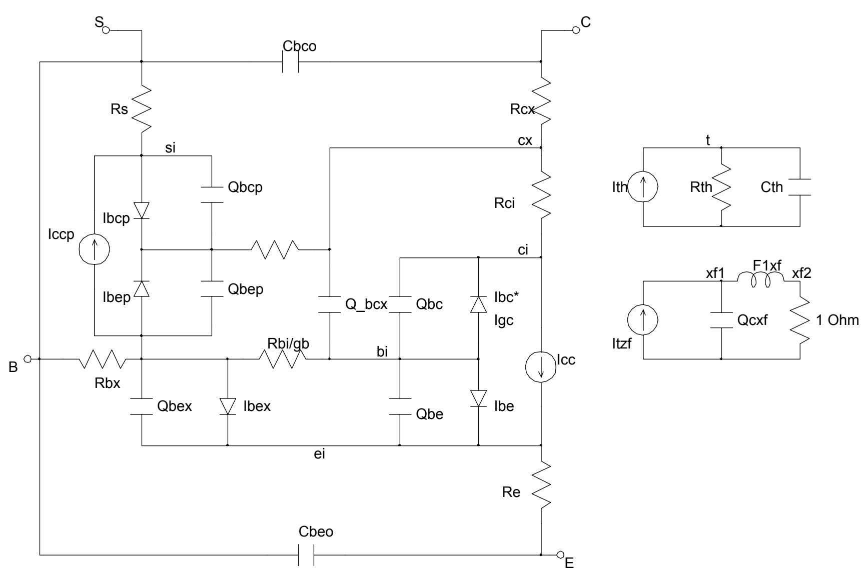
Symbol



Summary

This element is OBSOLETE and is replaced by the VBIC BJT Model (VBIC) element. The Vertical Bipolar Inter-Company (VBIC) model was developed to be an industry standard replacement for the SPICE Gummel-Poon (SGP) general-purpose BJT model. The VBIC model offers improved modeling of the early effect, quasi-saturation, substrate and oxide parasitics, avalanche multiplication and temperature behavior including self-heating. The improved modeling of these effects can be selectively invoked depending on the model parameters chosen. Since the VBIC model was developed to utilize the extraction techniques of the SGP model, it is possible that VBIC can degenerate to be similar to the SGP model. The Microwave Office implementation of the VBIC is largely based on the Saber VBIC95 Model Definition, Ver. 1.1.5, which has been in the public domain since June 1996.

Equivalent Circuit



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	
*NPN	Model Type (Information only)		1
*PNP	Model Type (Information only)		0
*TNOM	Extraction temperature	Celsius	25
*TAMB	Baseplate temperature	Celsius	25
*RCX	Extrinsic Collector Resistance	ohm	0.01
*RCI	Intrinsic Collector Resistance (must be>0)	ohm	0.01
*VO	Epi drift saturation voltage	Volts	0
*GAMM	Epi doping parameter		0
*HRCF	High current RC factor		1.0
*RBX	Extrinsic base resistance	ohm	0.01
*RBI	Intrinsic base resistance (must be>0)	ohm	0.01
*RE	Emitter resistance	ohm	0.01
*RS	Substrate resistance	ohm	0.01
*RBP	Parasitic base resistance (must be>0)	ohm	0.01
*IS	Transport saturation current	A	1e-16
*NF	Forward emission coefficient		1.0
*NR	Reverse emission coefficient		1.0
*FC	Forward Bias Junction Threshold Capacitance (0.05<FC<0...)		0.9
*CBEO	Base-Emitter Small Signal Capacitance	F	0.0
*CJE	Base-Emitter zero bias junction capacitance	F	0.0
*PE	Base-Emitter grading coefficient	Volts	0.75
*ME	Base-Emitter junction exponent		0.33
*AJE	Base-Emitter capacitance smoothing factor		-0.5
*CBCO	Extrinsic Base-Collector overlap capacitance	F	0.0
*CJC	Base-Collector zero-bias capacitance	F	0.0
*QCO	Collector charge at zero-bias		0.0
*CJEP	Extrinsic Base-Emitter zero-bias capacitance	F	0.0
*PC	Base-Collector grading coefficient	Volts	0.75
*MC	Base-Collector junction exponent		0.33
*AJC	Base-Collector capacitance smoothing factor		-0.5
*CJCP	Extensive Base-Collector zero bias capacitance	F	0.0
*PS	Collector-Substrate grading coefficient	Volts	0.75
*MS	Collector-Substrate junction exponent		0.33
*AJS	Collector-Substrate capacitance smoothing factor		-0.5

Name	Description	Unit Type	Default
*IBEI	Ideal Base-Emitter saturation current	A	1e-18
*WBE	Portion of IBEI from Vbei, (1-Wbe) is from Vbex		1.0
*NEI	Ideal Base-Emitter emission coefficient		1.0
*IBEN	Non-Ideal Base-Emitter saturation current	A	0.0
*NEN	Non-Ideal Base-Emitter emission coefficient		2.0
*IBCI	Ideal Base-Emitter saturation current	A	1.0e-16
*NCI	Ideal Base-Emitter emission coefficient		1.0
*IBCN	Non-Ideal Base-Collector saturation current	A	0.0
*NCN	Non-Ideal Base-Collector emission coefficient		2.0
*ISP	Parasitic transport saturation current	A	0.0
*WSP	Portion of ICCP from Vbep, 1-Wsp from Vbci		1.0
*NFP	Parasitic fwd emission coefficient		1.0
*IBEIP	Ideal parasitic Base-Emitter saturation current	A	0.0
*IBENP	Non-Ideal parasitic Base-Emitter saturation current	A	0.0
*IBCIP	Ideal parasitic Base-Collector saturation current	A	0.0
*NCIP	Ideal parasitic Base-Collector emission coefficient		1.0
*IBCNP	Non-Ideal parasitic Base-Collector saturation current	A	0.0
*AVC1	Base-Collector weak avalanche parameter 1		0.0
*AVC2	Base-Collector weak avalanche parameter 2		0.0
*NCNP	Non-Ideal parasitic Base-Collector emission coefficient		2.0
*VEF	Fwd Early voltage	Volt	1e9
*VER	Rev Early voltage	Volt	1e9
*IKF	Fwd knee current	A	1e09
*IKR	Rev knee current	A	1e09
*IKP	Parasitic knee current	A	1e09
*TF	Forward transit time (must be>0)	sec	0.0
*QTF	Variation of TF with base-width modulation		0.0
*XTF	Coeff. of TF bias dependence		0.0
*VTF	Coeff. of TF dependence on Vbc	Volt	0.0
*ITF	Coeff. of TF dependence in Ic	A	0.0
*TR	Ideal reverse transit time (must be>0)	sec	0.0
*TD	Forward excess-phase delay time (must be>0)	sec	0.0
*KFN	Flicker noise coefficient		0.0
*AFN	Flicker noise exponent		1.0
*BFN	Flicker noise frequency exponent		1.0
*NFLAG	Noise Analysis Flag; 0=no noise analysis		1.0
*XRE	Temperature exponent of emit resistance		0.0

Name	Description	Unit Type	Default
*XRB	Temperature exponent of base resistance		0.0
*XRC	Temperature exponent of collector resistance		0.0
*XRS	Temperature exponent of substrate resistance		0.0
*XVO	Temperature exponent of VO		0.0
*EA	Activation energy for Is (eV)	eV	1.12
*EAIE	Activation energy for Ibei (eV)	eV	1.12
*EAIC	Activation energy for IbcI/Ibeip (eV)	eV	1.12
*EAIS	Activation energy for Ibcip (eV)	eV	1.12
*EANE	Activation energy for Iben (eV)	eV	1.12
*EANC	Activation energy for Ibcn/Ibenp (eV)	eV	1.12
*EANS	Activation energy for Ibcnp (eV)	eV	1.12
*XIS	Temperature exponent of Is		3.0
*XII	Temperature exponent of IbeI/Ibci/Ibeip/Ibcip		3.0
*XIN	Temperature exponent of Iben/Ibcn/Ibenp/Ibcnp		3.0
*TNF	Temperature coefficient of nf		0.0
*TAVC	Temperature coefficient of AVC2		0.0
*RTH	Thermal resistance	ohm	0.0
*CTH	Thermal capacitance	F	1.0
*IMAX	Explosion current (not implemented)	A	1.0
AFAC	Area scale factor		1.0

** indicates a secondary parameter*

Operating Points

The following character strings have been used to identify the NL branches: be, bc, ce, bep, bcp, cep, bb, bbp, bex and cc. Here be, bc and ce correspond to the intrinsic base-emitter, base-collector and collector-emitter branches. bep, bcp and cep correspond to the base-emitter, base-collector and collector-emitter branches of the parasitic transistor associated with the substrate. bb and bbp denote the branches associated with the nonlinear base resistors of the main and parasitic transistors, respectively. Finally, bex and cc correspond to the external base-emitter and the non-linear collector resistor branches, respectively. These names are used to identify branch related operating point information, i.e., branch voltages, currents, etc.

Parameter	Description
igc (A)	Breakdown current.
pwr (W)	Power dissipation.
go (S)	Intrinsic small-signal output conductance.
gm (S)	Intrinsic small-signal transconductance.
gpi (S)	Intrinsic small-signal input conductance.
gmu (S)	Intrinsic small-signal Collector-Base conductance.

Parameter	Description
rbi (ohm)	Intrinsic base resistance.
rci (ohm)	Intrinsic collector resistance.
rbp (ohm)	Parasitic transistor base resistance.
cje (F)	Intrinsic be capacitance.
cjc (F)	Intrinsic bc capacitance.
cbex (F)	bex junction capacitance.
cbcx (F)	bcx junction capacitance.
cbep (F)	bep junction capacitance.
cbcp (F)	bcp junction capacitance.
betaac (A/A)	Small-signal common-emitter current gain.
ft** (Hz)	Unity small-signal current-gain frequency.
betadc (A/A)	Ratio of DC collector current to DC base current
dic_dvbe (S)	Intrinsic dIc/dVbe.
dic_dvbc (S)	Intrinsic dIc_dVbc.
dib_dvbe (S)	Intrinsic dIb_dVbe.
dib_dvbc (S)	Intrinsic dIb_dVbc.
temp (C)	Device temperature.
cpi(F)	Common-emitter input capacitance.
cmu (F)	Common-base output capacitance.
rpi (ohm)	Intrinsic small-signal input resistance.
ro(ohm)	Intrinsic small-signal output resistance.
rx(ohm)	Parasitic base resistance.

.

Implementation Details

The complete set of equations is too complex to be listed here. The user should consult the references for specific information. For such details particularly attention should be given to the works of Colin McAndrew et. al. in [3] [11–356] and [4] [11–356].

SCALING

The Microwave Office implementation includes an area scale factor, AFAC, which is not in the original definition of the model. All currents and charges, and linear capacitors are scaled in proportion to AFAC, and all liner resistors are scaled inversely with AFAC. The thermal resistance, RTH, and thermal capacitance, CTH, are scaled according to these rules as well.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

Recommendations for Use

VBIC is an advanced model. It should be used only where the need for accuracy justifies its complexity, and where parameter libraries are available. The Gummel-Poon model, GBJT, is recommended for ordinary applications.

References

[1] G. M. Kull et al, A Unified Circuit Model for Bipolar transistors including quasi-saturation, IEEE Trans. Elec. Dev., Vol.32, No.2, June, 1985.

[2] M. Bayer et al, A Precise model for the DC and transient characteristics of BJTs, IEEE PESC Proc., June, 1994.

[3] C. McAndrew et al, VBIC95 - The Vertical Bipolar Inter-Company Model, IEEE Journal of Solid State Circuits, October, 1995.

[4] C. McAndrew et al, VBIC95: An Improved Vertical IC Bipolar Transistor Model, IEEE BCTM Proc., September, 1995.

[5] R. Kraus, H.J.Mattausch, Status and trends of power semiconductor device models for circuit simulation, IEEE Trans. Pwr. Elec., Vol. 13, No. 3, May, 1998.

[6] B. Fatemizadeh et al, Modeling of power semiconductor devices, problems, limitations, and future trends, Proc. IEEE PELS Workshop on Computers in Power Electronics, August 1996.

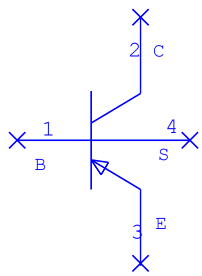
[7] C. McAndrew et al (Compact Modeling Council), VBIC95 Model Definition: Release 1.1.5, July, 1996.

[8] VBIC Web Page, http://www.fht-esslingen.de/institute/iafgp/neu/VBIC/ (University of Esslingen, Esslingen, Germany).

[9] X. Cao, J. McMacken et al, Parameter Extraction and Optimization for New Industry Standard VBIC Model, 2nd International Conference on Advance Semiconductor Devices and Microsystems (ASDAM), Slovakia, October 5-7, 1998.

(Obsolete) VBIC95 PNP BJT Model: VBIC95P

Symbol



Summary

This element is OBSOLETE and is replaced by the VBIC BJT Model (VBIC) element.

Equivalent Circuit

Same as VBIC95N, except that all diode and source currents are reversed.

Parameters

Parameters are identical to those of VBIC95N

Operating Points

Operating points are identical to those of VBIC95N.

Implementation Details

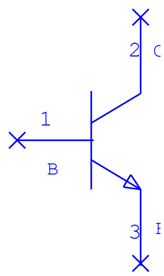
Except for the PNP structure, this model is equivalent to VBIC95N, the NPN implementation of the Vertical Bipolar Inter-Company (VBIC) model.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Simple Volterra BJT Model: VBJT1

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	VB1
*IC	Collector current	Current	10 mA
*A1	1st Volterra coef for alpha		0.99
*A2	2nd Volterra coef for alpha		0
*A3	3rd Volterra coef for alpha		0
*TD	Excess phase time delay	Time	0 ns
*TT	Base transit time	Time	0 ns
*RB1	Base resistance 1	Resistance	0.01 ohm
*RB2	Base resistance 2	Resistance	0.01 ohm
*RE	Emitter resistance	Resistance	0.01 ohm
*RC	Collector resistance	Resistance	0.01 ohm
*LB	Base inductance	Inductance	0 nH
*LE	Emitter Inductance	Inductance	0 nH
*LC	Collector inductance	Inductance	0 nH
*CBE0	PN Junction 0V BE capacitance	Capacitance	0 pF
*VJE	BE Built-in voltage	Voltage	1 V
*IS	Reverse saturation current	Current	10^{-11} mA
*N	Ideality factor		1
*M	Grading coefficient		0.5
*CBC	Base-collector capacitance	Capacitance	0 pF
*CCE	Collector-emitter capacitance	Capacitance	0 pF
*T	Temperature	Temperature	27 DegC
AFAC	Gate width scaling factor (total; not per finger)		1

** indicates a secondary parameter*

Implementation Details

The controlled current source is nonlinear to allow for the transistor's nonlinear small-signal current gain. The bias point is specified by the collector bias current, IC. This quantity determines the Taylor-series representation for the BE diode and nonlinear capacitance.

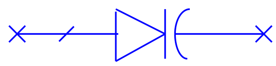
VBJT1 uses the VVCN model to implement the nonlinear I/V behavior. The Alpha parameters are the polynomial coefficients to the I/V equation. See the VVCN model documentation for more information.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Volterra PN Junction Capacitance: VCJCN

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	VC1
*CJ0	Zero-voltage junction capacitance	Capacitance	1 pF
*VJ	Built-In voltage	Voltage	1 V
*VDD	Junction DC bias voltage	Voltage	0 V
*M	Denominator exponent		0.5
AFAC	Junction area scaling factor		1

** indicates a secondary parameter*

Implementation Details

VCJCN is a nonlinear PN junction depletion capacitance described by a Taylor series expansion in the vicinity of its DC bias voltage, VDD. Its charge-voltage characteristic is give by

$$q = (q_1 v + q_2 v^2 + q_3 v^3) \cdot \mathrm{AFAC}$$

where q_n are the coefficients of a Taylor-series expansion of the junction charge function

$$Q(V) = \frac{-\varphi C_{j0}}{1-M}\left(1-\frac{V}{\varphi}\right)^{1-M}$$

The current in the capacitor is simply

$$i = \frac{\mathrm{dq}}{\mathrm{dt}}$$

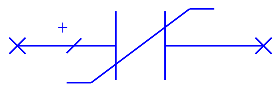
where i is the current in the controlled source and v is the control voltage. Since the q_n are evaluated at a particular DC bias voltage, DC bias is implicit in the model. Thus, DC bias must not be included in the circuit.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

Volterra Nonlinear Capacitance: VCNL

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	CN1
*Q1	1st volterra charge coef		1e-12
*Q2	2nd volterra charge coef		0
*Q3	3rd volterra charge coef		0
AFAC	Charge scaling factor		1

** indicates a secondary parameter*

Implementation Details

$$q = (q_1 v + q_2 v^2 + q_3 v^3) \cdot \mathrm{AFAC}$$

$$i = \frac{\mathrm{dq}}{\mathrm{dt}}$$

where i is the current in the controlled source and v is the control voltage. The q_n coefficients are the Qn parameters of the model.

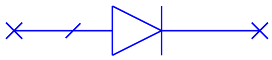
Since the q_n are evaluated at a particular DC bias voltage, DC bias is implicit in the model. Thus, DC bias must not be included in the circuit.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

Volterra Diode Model: VDIODE1

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Diode ID	Text	VD1
*IS	Reverse saturation current	Current	1e-11 mA
*RS	Series resistance	Resistance	0.001 ohm
*N	Ideality factor		1
*TT	Storage time	Time	0 ns
*CJ0	Zero-voltage junction capacitance	Capacitance	0 pF
*VJ	Built-in voltage	Voltage	1 V
*VDD	Junction bias voltage	Voltage	0 V
*M	Grading coefficient		0.5
*FC	Depletion capacitance linearization parameter		0.5
*BV	Breakdown voltage	Voltage	1e+06 V
*IBV	(Not implemented)	Current	0 mA
*T	Temperature	Temperature	27 DegC
AFAC	Junction area scaling factor		1

** indicates a secondary parameter*

Implementation Details

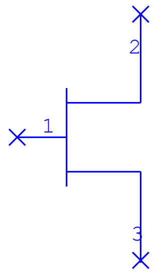
VDIODE1 is a nonlinear incremental model of the Schottky diode. It is equivalent to SDIODE, but is intended primarily for Volterra analysis. The DC junction bias voltage, VDD, is specified by the user, and the program calculates the Taylor series coefficients for both the resistive and capacitive parts of the junction. See the SDIODE element for equations.

Layout

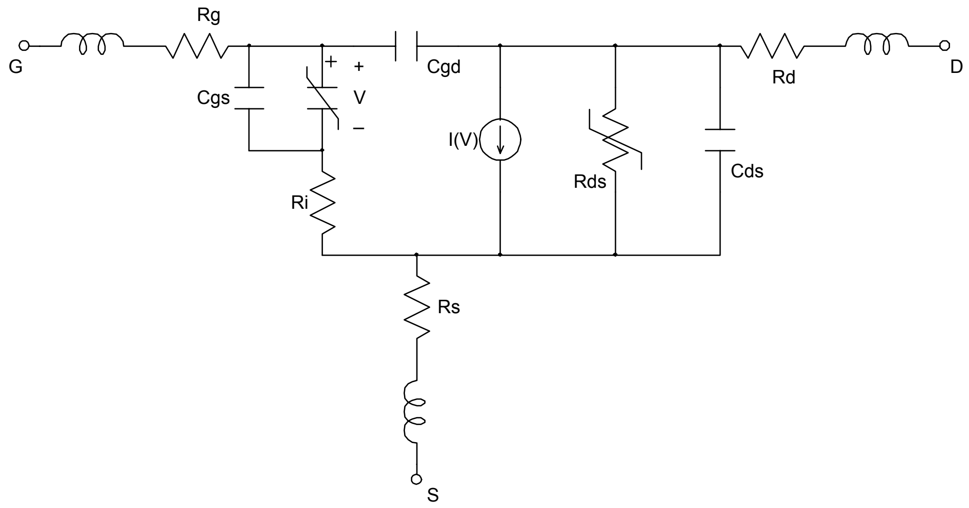
This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Simple Volterra FET Model: VFET1

Symbol



Equivalent Circuit



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	VF1
*GS1	1st volterra coef for controlled source		0.1
*GS2	2nd volterra coef for controlled source		0
*GS3	3nd volterra coef for controlled source		0
*GD1	1st volterra coef for Gds		0.001
*GD2	2nd volterra coef for Gds		0
GD3	3rd volterra coef for Gds		0
*TD	Time delay	Time	0 ns
*RG	Gate resistance	Resistance	0.001 ohm
*RS	Source resistance	Resistance	0.001 ohm

Name	Description	Unit Type	Default
*RD	Drain resistance	Resistance	0.001 ohm
*RI	Intrinsic resistance	Resistance	0.001 Ohm
*LG	Gate inductance	Inductance	0 nH
*LS	Source inductance	Inductance	0 nH
*LD	Drain inductance	Inductance	0 nH
*CGS0	PN junction 0V gate capacitance	Capacitance	0 pF
*PHI	Gate built-in voltage	Voltage	1 V
*VGG	Gate bias (for Cgs only)	Voltage	0 V
*CGD	Gate-drain capacitance	Capacitance	0 pF
*CGS	Gate-source fixed capacitance	Capacitance	0 pF
*CDS	Drain-source capacitance	Capacitance	0 pF
AFAC	Gate width scaling factor (total; not per finger)		1
NFING	Scale factor for number of gate fingers		1

** indicates a secondary parameter*

Implementation Details

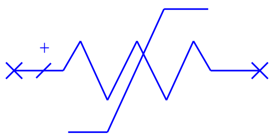
The FET's gate-to-source capacitor is modeled by a VCJCN element. The nonlinear controlled current source is modeled by a VVCN element, and the nonlinear drain-to-source resistance by a VGNL . See these elements for further information.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Volterra Nonlinear Conductance: VGNL

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	GN1
*G1	1st volterra coef		0.001
*G2	2nd volterra coef		0
*G3	3rd volterra coef		0
AFAC	Current scaling factor		1

** indicates a secondary parameter*

Implementation Details

VGNL is a nonlinear resistor whose I/V characteristic is described by a Taylor series in the vicinity of its DC bias voltage. As such, it is ideal for use with Volterra analysis or with small-signal IM calculations by harmonic balance. The I/V characteristic is

$$i(v) = \frac{\mathrm{dI}}{\mathrm{dV}}\Big|_{V=V_0} v + \frac{d^2 I}{\mathrm{dV}^2}\Big|_{V=V_0} v^2 + \frac{1}{6}\frac{d^3 I}{\mathrm{dV}^3}\Big|_{V=V_0} v^3$$

which can be expressed as

$$i(v) = g_1 v + g_2 v^2 + g_3 v^3$$

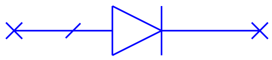
The g_n coefficients are the Gn parameters of the model. Since the g_n are evaluated at a particular DC bias voltage, DC bias is implicit in the model. Thus, DC bias must not be included in the circuit.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

Volterra Diode Junction Model: VJCN

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Diode ID	Text	VJ1
*IJ	Diode junction DC current	Current	5 mA
*N	Ideality factor		1
*T	Temperature	Temperature	27 DegC
AFAC	Junction area scaling factor		1

** indicates a secondary parameter*

Implementation Details

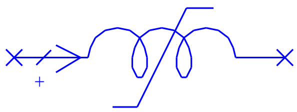
VJCN is a nonlinear incremental model of the Schottky diode. It is intended primarily for Volterra analysis. The user enters conventional Schottky diode parameters, and the DC current, and the Volterra (Taylor series) coefficients are computed within the model.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Volterra Nonlinear Inductance: VLNL

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	LN1
*L1	1st volterra flux coef		1e-06
*L2	2nd volterra flux coef		0
*L3	3rd volterra flux coef		0
AFAC	Flux scaling factor		1

** indicates a secondary parameter*

Implementation Details

The model is implemented as a flux-current nonlinearity

$$\varphi(t) = (L_1 i + L_2 i^2 + L_3 i^3) \cdot \mathrm{AFAC}$$

where the Ln are Taylor-series coefficients of an expansion of the Φ(I) characteristic in the vicinity of the bias point. The small-signal voltage across the inductor, v, is

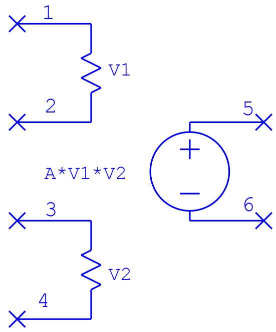
$$v = \frac{d\varphi(t)}{\mathrm{dt}}$$

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

Voltage Multiplier: VOLTMULT

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	VM1
A	Voltage gain		1
RO	Output impedance	Resistance	0.001 ohm

Implementation Details

Implements an ideal voltage multiplier. The output voltage is given by:

$$V_o = A \cdot V1 \cdot V2$$

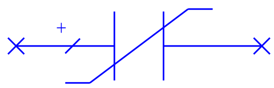
The input impedance is $10^{12}\Omega$.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Simple Varactor Model: VRCTR

Symbol



Summary

VRCTR models a lossless varactor. Internally, this element is identical to the nonlinear capacitor model (NLCAP), but the nonlinear characteristics are extracted automatically from tabulated data.

Parameters

Name	Description	Unit Type	Default
ID	Device ID	Text	C1
CData	Capacitance data	Capacitance	{1}
VData	Voltage data	Voltage	{-1} V
PolyOrd	Polynomial order		5
Tolerance	Maximum fitting error		2

Parameter Details

The C/V data is entered in the CData and VData fields. It can be entered manually, for example:

CData = {30,15,10,5,4,3}

VData = {0,1,2,3,4,5}

or by referencing file-based data.

Suppose that measured C/V data is stored in a file called cv.txt. The data is column-ordered, with C in the first column and V in the second column.

Import the file by right-clicking on **Data Files** in the Project Browser and choosing **Import Data File**. In the Open dialog box, in **Files of Type**, select **Text Data Files** and then browse for cv.txt. Now, place the following equation on the schematic:

cv_data = DataFile("cv.txt");

cv_data is now a matrix with the C/V data in its columns.

Type the following equations in the CData and VData parameter fields to access the needed columns of cv_data:

For CData: Col(cv_data, 1)

For VData: Col(cv_data, 2)

PolyOrd is the order of the polynomial used for numerical fitting of the capacitance data. Higher polynomial orders may result in convergence problems, so it is best to keep it as low as possible. It is rarely necessary to specify orders higher than 5.

Once a numerical fit to C/V data has been performed, the model will compute the maximum relative error between the measured data and the polynomial prediction. If that number, measured in %, is larger than Tolerance, the model will report a warning.

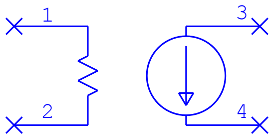
Simulation of the model's C/V characteristic and hence the quality of the polynomial fit, is highly recommended. Please see the *Examples\Circuit Design Types\Modeling\varactorplot.emp* example project.

Layout

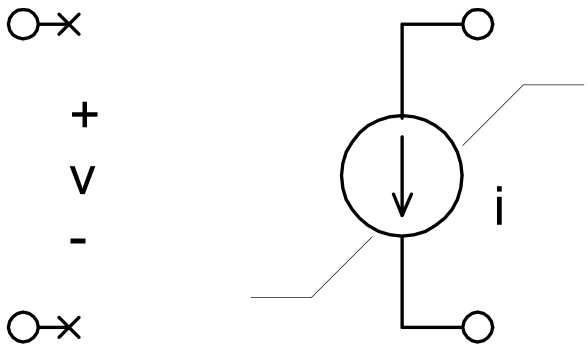
This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

Volterra Nonlinear Controlled Current Source: VVCN

Symbol



Equivalent Circuit



Parameters

Name	Description	Unit Type	Default
ID	ID	Text	VN1
*G1	1st volterra coef		0.1
*G2	2nd volterra coef		0
*G3	3rd volterra coef		0
AFAC	Current scaling factor		1

** indicates a secondary parameter*

Implementation Details

VVCN is a nonlinear voltage-controlled current source whose I/V characteristic is described by a Taylor series in the vicinity of its DC bias voltage. As such, it is ideal for use with Volterra analysis or with small-signal IM calculations by harmonic balance. The I/V characteristic is

$$i(v) = \frac{\mathrm{dI}}{\mathrm{dV}}\Big|_{V=V_0} v + \frac{d^2 I}{\mathrm{dV}^2}\Big|_{V=V_0} v^2 + \frac{1}{6}\frac{d^3 I}{\mathrm{dV}^3}\Big|_{V=V_0} v^3$$

which can be expressed as

$$i(v) = g_1 v + g_2 v^2 + g_3 v^3$$

where i is the current in the controlled source and v is the control voltage. The g_n coefficients are the Gn parameters of the model.

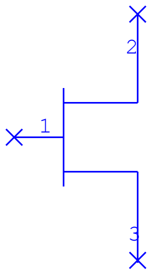
Since the g_n are evaluated at a particular DC bias voltage, DC bias is implicit in the model. Thus, DC bias must not be included in the circuit.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Yhland MESFET Model: YHLAND

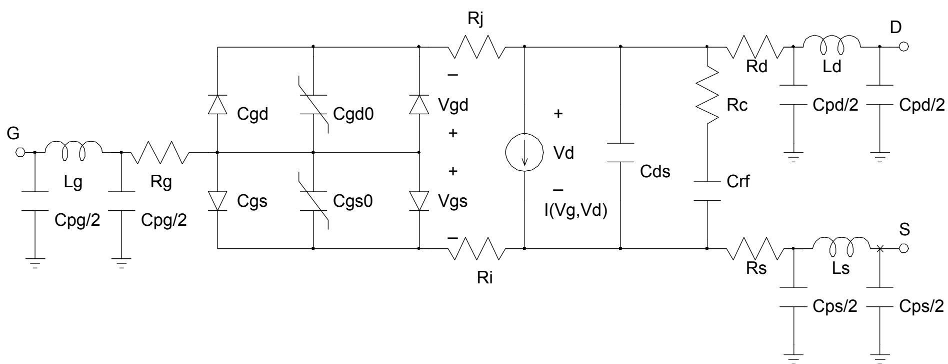
Symbol



Summary

The Yhland model is designed specifically for resistive FET devices, such as FET switches, attenuators, and mixers. It is strongly recommended over conventional "active" models for such circuits.

Equivalent Circuit



Parameters

Name	Description	Unit Type	Default
ID	Device ID	Text	YF1
*A	Model param a; see Yland paper		0.04
*B	Model param b; see Yland paper		1.9
*C	Model param c; see Yland paper		0.22
*D	Model param d; see Yland paper		6
*G	Model param g; see Yland paper		0.092
*PHI	Model param phi; see Yland paper	Angle	1.9022 deg
*IS	Gate-conduction diode current parameter	Current	10^{11} mA
*N	Gate-conduction diode ideality factor		1
*VBO	Gate-channel breakdown voltage (pos.)	Voltage	1e6 V

Name	Description	Unit Type	Default
*VBI	Gate-source capacitance built-in voltage	Voltage	1 V
*RJ	G-D resistance	Resistance	0.001 ohm
*RI	G-S resistance	Resistance	0.001 ohm
*RD	Drain resistance	Resistance	0.001 ohm
*RG	Gate resistance	Resistance	0.001 ohm
*RS	Source resistance	Resistance	0.001 ohm
*CGS0	Gate-source capacitance at 0V	Capacitance	0 pF
*CGD0	Gate-drain capacitance at 0V	Capacitance	0 pF
*FC	Gate-capacitance linearization parameter		0.5
*CDS	Drain-source capacitance	Capacitance	0 pF
*CGS	Fixed gate-source capacitance	Capacitance	0 pF
*CGD	Fixed gate-drain capacitance	Capacitance	0 pF
*CPD	Drain pad capacitance (not scaled)	Capacitance	0 pF
*CPS	Source pad capacitance (not scaled)	Capacitance	0 pF
*CPG	Gate pad capacitance (not scaled)	Capacitance	0 pF
*LG	Gate inductance	Inductance	0 nH
*LS	Source inductance	Inductance	0 nH
*LD	Drain inductance	Inductance	0 nH
*TNOM	Temperature	Temperature	0 nH
AFAC	Gate-width scale factor		27 deg C
NFING	Number of gate fingers scale factor		1

** indicates a secondary parameter*

Implementation Details

The Yhland FET model is a new model intended primarily for use in resistive FET circuits, especially FET resistive mixers and attenuators. The model was developed by Klas Yhland, a Ph. D. graduate of Chalmers University in Goteborg, Sweden (see References.)

To be consistent with the Chalmers implementation, this model includes an L-C pi model of the pad parasitics.

Drain Current

The drain current is given by

$$I_d = g(f_1(U_{gd}^+, U_{gs}^+) \cdot f_2(V_s - V_{gd}) - f_1(U_{gs}^-, U_{gd}^-) \cdot f_2(V_{gd} - V_{gs}))$$

where

$$f_1(x, y) = (1 + \alpha x)(1 - \tanh(e^{-b(y+c)}))$$

$$f_2(z) = 1 - \tanh(e^{-dz})$$

U_{gd}+ and U_{gs}+ define a counterclockwise-rotated coordinate system, relative to the V_{gd}/V_{gs} axis:

$$\begin{bmatrix} U_{gd}^{+} \\ U_{gs}^{+} \end{bmatrix} = \begin{bmatrix} \cos(\varphi) & -\sin(\varphi) \\ \sin(\varphi) & \cos(\varphi) \end{bmatrix} \begin{bmatrix} V_{gd} \\ V_{gs} \end{bmatrix}$$

where

$$\varphi$$

is the angle of rotation. Similarly, U_{gd}- and U_{gs}-; define a clockwise-rotated coordinate system, relative to the V_{gd}/V_{gs} axis:

$$\begin{bmatrix} U_{gd}^{-} \\ U_{gs}^{-} \end{bmatrix} = \begin{bmatrix} \cos(\varphi) & -\sin(\varphi) \\ \sin(\varphi) & \cos(\varphi) \end{bmatrix} \begin{bmatrix} V_{gd} \\ V_{gs} \end{bmatrix}$$

Please see the references for explanations of the use of this rotated coordinate system.

Capacitances

The original model, as described in the references, does not include nonlinear capacitances. We have implemented simple, PN-junction capacitances for gate-source and gate-drain capacitance. The parameters of these elements (CGSO/CGDO, FC, and VBI) are essentially the same as the corresponding ones in the SDIODE model.

Gate Conduction

Gate conduction is modeled by PNIV elements. The parameters (IS, N, VBO) are the same as corresponding parameters in SDIODE.

Pad Capacitance and Lead Inductance

These quantities are modeled by pi networks at each terminal. Because the source terminal in resistive FET circuits frequently is not grounded, a pi network is included at the source as well as the gate and drain.

SCALING

Resistances except gate resistance are scaled as 1/AFAC.

Gate resistance is scaled as $AFAC/NFING^2$.

Capacitances and currents are scaled in proportion to AFAC.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

References

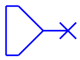
[1] K. Yhland, Resistive FET Mixers, Ph. D. Diss., Chalmers University of Technology, Goteborg, Sweden, 1999

[2] K. Yhland, N. Rorsman, M. Garcia, and H. Merkel, "A Symmetrical HFET/MESFET Model Suitable for Intermodulation analysis of Amplifiers and Resistive FET Mixers," IEEE Trans. Microwave Theory Tech., vol. 48, p. 15 (Jan., 2000).

Ports

Circuit Port: PORT

Symbol



Parameters

Name	Description	Unit Type	Default
P	Port number		0
Z	Termination impedance	Resistance	50 ohm
PIN_ID	Name identifier for pin	Text	

Implementation Details

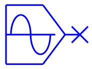
This is a passive circuit termination that applies a load specified by Z. The port number P is set automatically. If PIN_ID is not empty, text is displayed on the pin when it is instantiated as a subcircuit.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Port with 1-Tone HB Source: PORT1

Symbol



Summary

This is an active circuit termination that applies a load resistance specified by Z while supplying a power signal of magnitude Pwr and angle Ang swept across the frequency range specified for the schematic containing the port. The port number P is set automatically. If PIN_ID is not empty, text is displayed on the pin when it is instantiated as a subcircuit.

Parameters

Name	Description	Unit Type	Default
P	Port number		0
Z	Termination impedance	Resistance	50 ohm
Pwr	Power in magnitude	DB Power[1]	0 dBm
*Ang	Relative angle of the excitation	Angle[1]	0 Deg
PIN_ID	Name identifier for pin	Text	

[1] Pwr and Ang may be vectors.

Parameter Details

Pwr and Ang. Typically, Pwr and Ang represent the available power and angle of the excitation at the fundamental harmonic. To supply power at multiple harmonics, one specifies the power and angle parameters as vectors. For example,

Pwr = {0,10,-10,-20}

Ang = {0,0,45,90}

applies power (in DB power units) equal to 0 at DC, 10 at the fundamental, -10 at the second and -20 at the third harmonic, with excitation angles as shown.

Unless the number of Ang vector elements equals the number of Pwr vector elements, all angles are assumed equal to zero. If the vectors have only one element, the excitation is assumed to be at the fundamental.

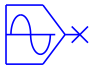
NOTE: If a sinusoidal nonlinear source like this one has the parameter Ang=0 and is ideally terminated, then a nonlinear measurement (e.g. Pcomp or Vcomp) made at the fundamental output of that source will have an angle of -90deg. This discrepancy is consistent with the definition of the sine wave sources (as in SPICE), and the Fourier based harmonic component measurements. To avoid confusion, always measure or calculate the gain, and plot its angle.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Port with 1-Tone Volterra Source: PORT1V

Symbol



Parameters

Name	Description	Unit Type	Default
P	Port number		0
Z	Termination impedance	Resistance	50 ohm
Pwr	Power in magnitude	DB Power	0 dBm
*Ang	Relative angle of the excitation	Angle	0 Deg

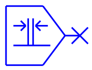
** indicates a secondary parameter*

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

Port with 2-Tone HB Source: PORT2

Symbol



Parameters

Name	Description	Unit Type	Default
P	Port number		0
Z	Termination impedance	Resistance	50 ohm
Fdelt	Delta frequency (f2=f1+Fdelt)	Frequency	0.1 GHz
Pwr1	Magnitude of power of tone one	DB Power	0 dBm
Pwr2	Magnitude of power of tone two	DB Power	0 dBm
*Ang1	Relative angle of tone one	Angle	0 Deg
*Ang2	Relative angle of tone two	Angle	0 Deg
PIN_ID	Name identifier for pin	Text	

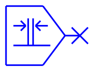
NOTE: If a sinusoidal nonlinear source like this one has the parameter Ang=0 and is ideally terminated, then a nonlinear measurement (e.g. Pcomp or Vcomp) made at the fundamental output of that source will have an angle of -90deg. This discrepancy is consistent with the definition of the sine wave sources (as in SPICE), and the Fourier based harmonic component measurements. To avoid confusion, always measure or calculate the gain, and plot its angle.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

Port with 2-Tone Volterra Source: PORT2V

Symbol



Parameters

Name	Description	Unit Type	Default
P	Port number		0
Z	Termination impedance	Resistance	50 ohm
Fdelt	Delta frequency (f2=f1+Fdelt)	Frequency	0.1 GHz
Pwr1	Magnitude of power of tone one	DB Power	0 dBm
Pwr2	Magnitude of power of tone two	DB Power	0 dBm
*Ang1	Relative angle of tone one	Angle	0 Deg
*Ang2	Relative angle of tone two	Angle	0 Deg

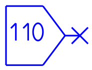
** indicates a secondary parameter*

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Arbitrary, User-specified Bit Sequence: PORT_ARBS

Symbol



Summary

PORT_ARBS is a source in the form of a user-specified bit sequence.

Parameters

Name	Description	Unit Type	Default
P	Port number		0
Z	Termination impedance	Resistance	50 ohm
RATE	Symbol rate	Frequency	1 GHz
SEQ	Symbol sequence		{1,0,1,0}
SAMP	Samples per symbol		8
HI	High signal level	Voltage	1
LO	Low signal level	Voltage	0
TR	Rise time	Time	0
TF	Fall time	Time	0
TYPE	Signaling format		NRZ[1]
WINDOW	Window type		DEFAULT[2]
*TONE	Tone number		1
PIN_ID	Name identifier for pin	Text	

** indicates a secondary parameter*

[1] Allowed formats are return to zero (RZ) and non-return to zero (NRZ).

[2] Window type options are none (NONE), Lanczos (DEFAULT, ref [1]), Triangular (TRIANG), Hanning (HANN), Hamming (HAMM), and Blackman (BLACK).

Parameter Details

SEQ. Specifies the symbol sequence. You can type the sequence or load it from a text file. To load a sequence from a file, type `vfile("name_of_file")` in the SEQ field. The file format is:

1

0

1

0

etc.

In the absence of a full path specification, place the file in the *\Data* directory under the Microwave Office installation directory.

Symbols in multi-level signal formats are encoded as integers. In a four-level scheme, for example, symbols are represented by 0, 1, 2 and 3.

The data files with sequences can also be imported into Microwave Office to save the data with the project.

To use bit data from a file stored in Microwave Office:

1. Import the data file as a "Text Data File" type.
2. Assign a variable to the data file using the following syntax:

   ```
   variable1 = DataFile("data_file_name")
   ```

   where `variable1` is the variable name of your choice and `"data_file_name"` is the name of the imported data file (the name must be enclosed in quotes). This function saves the data in a matrix form.
3. Assign a variable to be the first column of the data in the first variable assigned in the following syntax:

   ```
   variable2 = col(variable1,1)
   ```

   where `variable2` is the variable name of your choice and now stores the bit sequence assigned to the SEQ parameter.

Implementation Details

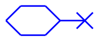
With the exception of the SEQ parameter, PORT_ARBS is very similar to PORT_PRBS, which applies a random rather than user-specified sequence. See PORT_PRBS for information on the remaining parameters.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

Export Name Port Properties: PORT_NAME

Symbol



Summary

PORT_NAME is used to create schematic connectivity by name instead of by wire. See "Element Connection by Name" for details.

Parameters

Name	Description	Unit Type	Default
P	Port number		0
Name	Port Name		0
PNO	Port name order value		1

Parameter Details

PNO. Specify the preferred order of the port. This value is read when all ports in a circuit (for example, mixed PORT and PORT_NAME elements) are sorted to determine port numbers/ordering. You can also edit this value to ensure GDS cell connections.

Implementation Details

PORT_NAME implements a named port. When used in a schematic, subcircuit instances of the schematic have named pins (if named symbol pins are present) or named parameters that drive connectivity.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

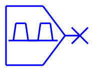
1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See "Cell Options Dialog Box: Layout Tab " for Cell Options dialog box **Layout** tab details.

See "The Layout Process File (LPF)" for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

Port with Pulse Train Signal: PORT_PLS

Symbol



Parameters

Name	Description	Unit Type	Default
P	Port number		0
Z	Termination impedance	Resistance	50 ohm
AMP	Open circuit signal magnitude	Voltage	0 V
TW	Pulse width	Time	0 ns
TR	Rise time	Time	0 ns
TF	Fall time	Time	0 ns
TD	Time delay	Time	0 ns
WINDOW	Window type		DEFAULT
DCVal	DC Value (Used for DC analysis)	Voltage	0
PIN_ID	Name identifier for pin	Text	

Implementation Details

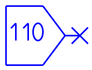
This is an active circuit termination that applies a load resistance specified by Z while supplying a pulsed power swept across the frequency range specified for the schematic containing the port. The port number P is set automatically. If PIN_ID is not empty, text is displayed on the pin when it is instantiated as a subcircuit.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Pseudo-random Bit Sequence: PORT_PRBS

Symbol



Summary

PORT_PRBS is a source in the form of a pseudo-random bit sequence.

Parameters

Name	Description	Unit Type	Default
P	Port number		0
Z	Termination impedance	Resistance	50 ohm
RATE	Symbol rate	Frequency	1 GHz
NSYMB	Number of symbols		16
SAMP	Samples per symbol		8
BITW	Number of bits in symbol		1
HI	High signal level	Voltage	1
LO	Low signal level	Voltage	0
TR	Rise time	Time	0
TF	Fall time	Time	0
TYPE	Signaling format		NRZ[1]
WINDOW	Window type		DEFAULT[2]
SEED	Random number generator seed		-1
*TONE	Tone number		1
PIN_ID	Name identifier for pin	Text	

** indicates a secondary parameter*

[1] Allowed formats are return to zero (RZ) and non-return to zero (NRZ).

[2] Window type options are none (NONE), Lanczos (DEFAULT, ref [1]), Triangular (TRIANG), Hanning (HANN), Hamming (HAMM), and Blackman (BLACK).

Parameter Details

BITW. This parameter may be used to apply multilevel PRSB signals. The number of levels is equal to 2^{BITW}. Levels are equally spaced between HI and LO.

WINDOW. The PRBS sequence must be represented in the frequency domain for the purposes of harmonic balance analysis. To control overshoot (Gibbs phenomenon) the sequence is windowed prior to simulation. Note that windows attenuate high-frequency components and therefore induce a finite rise and fall time. With the exception of the triangular window, the windows are listed in the order of smaller overshoot/longer rise time.

TR and **TF.** Rise and fall times are strongly influenced by the window type and the number of samples used to represent the sequence (the product of SAMP and NSYMB). If precise rise and fall times are critical to the application, use WINDOW=DEFAULT and increase SAMP as necessary.

NSYMB. There are no known limitations on the number of symbols in the sequence. However, the number of frequencies in the Fourier representation of a PRBS sequence equals one half of the product of NSYMB and SAMP. For this reason, longer sequences result in longer simulations.

NSYMB and **SAMP.** The product of these two parameters must be a power of 2 or an error is produced.

SEED. If less than zero, the current tick count of the system clock is used for seed.

Implementation Details

Harmonic balance options and project frequencies (or nonlinear frequencies) are ignored for the purposes of simulation with PORT_PRBS.

The fundamental frequency is determined by RATE/NSYMB.

The number of frequencies simulated is determined by NSYBM*SAMP/2 +1. (The +1 is for the DC component.)

The maximum frequency in the spectrum is determined by RATE *SAMP/2.

The number of frequencies is determined by NSYMB and SAMP. Likewise, the fundamental frequency is determined by RATE and NSYMB and is in no relation to the project and nonlinear frequencies.

When BITW is 1, the binary sequence generated is such that there are NSYMB 1 bits and NSYMB 0 bits. A Wichmann-Hill random number generator is used to determine which positions in the 2*NSYMB bits are to be set to 1, with the numbers generated by the random number generator corresponding to the bit positions. Random numbers are generated until NSYMB unique positions out of the 2*NSYMB available positions have been selected. The bits at these positions are set to 1.

When BITW is greater than 1, a Wichmann-Hill random number generator is used to generate 2*NSYMB numbers in a range of 2^{BITW} discrete values. The generated values are then used to determine the level to be output for each symbol.

NOTE: You can use Veye and Ieye measurements to display eye diagrams in conjunction with PORT_PRBS simulations.

If PIN_ID is not empty, text is displayed on the pin when it is instantiated as a subcircuit.

Layout

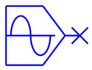
This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

References

[1] R.W. Hamming, Numerical methods for scientists and engineers, pp 534.

Port with Power Sweep/1-Tone HB Source: PORT_PS1

Symbol



Parameters

Name	Description	Unit Type	Default
P	Port number		0
Z	Termination impedance	Resistance	50 ohm
PStart	Swept power magnitude start	DB Power	0 dBm
PStop	Swept power magnitude stop	DB Power	0 dBm
PStep	Swept power magnitude step	DB	0 dB
*Ang	Relative angle of the excitation	Angle	0 Deg
PIN_ID	Name identifier for pin	Text	

Implementation Details

This is an active circuit termination that applies a load resistance specified by Z while supplying a power signal swept from *PStart* to *PStop* and also swept across the frequency range specified for the schematic containing the port. The port number P is set automatically. If PIN_ID is not empty, text is displayed on the pin when it is instantiated as a subcircuit.

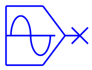
NOTE: If a sinusoidal nonlinear source like this one has the parameter Ang=0 and is ideally terminated, then a nonlinear measurement (e.g. Pcomp or Vcomp) made at the fundamental output of that source will have an angle of -90deg. This discrepancy is consistent with the definition of the sine wave sources (as in SPICE), and the Fourier based harmonic component measurements. To avoid confusion, always measure or calculate the gain, and plot its angle.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

Port with Power Sweep/1-Tone Volterra Source: PORT_PS1V

Symbol



Parameters

Name	Description	Unit Type	Default
P	Port number		0
Z	Termination impedance	Resistance	50 ohm
PStart	Swept power magnitude start	DB Power	0 dBm
PStop	Swept power magnitude stop	DB Power	0 dBm
PStep	Swept power magnitude step	DB	0 dB
*;Ang	Relative angle of the excitation	Angle	0 Deg

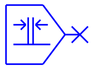
** indicates a secondary parameter*

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Port with Power Sweep/2-Tone HB Source: PORT_PS2

Symbol



Parameters

Name	Description	Unit Type	Default
P	Port number		0
Z	Termination impedance	Resistance	50 ohm
Fdelt	Delta frequency (f2=f1+Fdelt)	Frequency	0.1 GHz
PStart	Swept power magnitude start	DB Power	0 dBm
PStop	Swept power magnitude stop	DB Power	0 dBm
PStep	Swept power magnitude step	DB	0 dB
PIN_ID	Name identifier for pin	Text	

Implementation Details

This is an active circuit termination that applies a load resistance specified by Z while supplying a power signal of two tones, separated by frequency *Fdelt*, swept from *PStart* to *PStop* and also swept across the frequency range specified for the schematic containing the port. The power in the PORT_PS2 element is applied to both tones simultaneously at the level specified by Pstart. The port number *P* is set automatically. If *PIN_ID* is not empty, text is displayed on the pin when it is instantiated as a subcircuit.

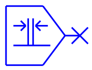
NOTE: If a sinusoidal nonlinear source like this one has the parameter Ang=0 and is ideally terminated, then a nonlinear measurement (e.g. Pcomp or Vcomp) made at the fundamental output of that source will have an angle of -90deg. This discrepancy is consistent with the definition of the sine wave sources (as in SPICE), and the Fourier based harmonic component measurements. To avoid confusion, always measure or calculate the gain, and plot its angle.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Port with Power Sweep/2-Tone Volterra Source: PORT_PS2V

Symbol



Parameters

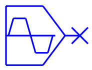
Name	Description	Unit Type	Default
P	Port number		0
Z	Termination impedance	Resistance	50 ohm
Fdelt	Delta frequency (f2=f1+Fdelt)	Frequency	0.1 GHz
PStart	Swept power magnitude start	DB Power	0 dBm
PStop	Swept power magnitude stop	DB Power	0 dBm
PStep	Swept power magnitude step	DB	0 dB

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

Port with Piecewise Linear Signal: PORT_PWL

Symbol



Summary

PORT_PWL consists of a Thevenin circuit with a piecewise linear (PWL) voltage source and an impedance Z0 in series with it.

The behavior of the PWL voltage source is similar to that of the PWL source in SPICE. The open circuit voltage is specified by the pairs of the time-voltage values (TIME[i], VOLTAGE[i]). The values of open circuit voltage between these time instances are obtained by linear interpolation.

The fundamental frequency is equal to that specified for the schematic containing the port.

Parameters

Name	Description	Unit Type	Default
P	Port number	Text	0
Z	Termination Impedance	Resistance	50 ohm
TIME	Vector of time values	Time	User must override the default
VOLTAGE	Vector of open-circuit voltage values	Voltage	User must override the default
WINDOW	Window type		DEFAULT[1]
DCVal	DC Value (Used for DC analysis)	Voltage	0
PIN_ID	Name identifier for pin	Text	

[1] Window type options are none (NONE), Lancoz (DEFAULT, ref. [1], Triangular (TRIANG), Hanning (HANN), Hamming (HAMM), and Blackman (BLACK)

Parameter Details

The dimension of TIME and VOLTAGE vectors must be identical. This dimension should be greater than 3, as at least 4 points are required to specify a meaningful PWL waveform. An error message is issued if one of these conditions is not satisfied, and the model is rejected. The impedance Z should be real to enable SPICE translation. The values of time should be sorted in ascending order (TIME[i+1]>TIME[i]). Discontinuities in the voltage value are not allowed (do not specify two different voltage values for the same time instance).

Implementation Details

Note that the signal needs to be periodic with the period equal to that of the fundamental frequency, otherwise results may be unexpected. The accuracy of the simulation results depends on the number of harmonics of the fundamental frequency. This number of harmonics is determined from the options for the schematic containing this port.

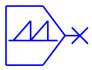
The port number P is set automatically. If PIN_ID is not empty, text is displayed on the pin when it is instantiated as a subcircuit.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Port with Saw Tooth Signal: PORT_SAW

Symbol



Parameters

Name	Description	Unit Type	Default
P	Port number		0
Z	Termination impedance	Resistance	50 ohm
AMP	Open circuit signal magnitude	Voltage	0 V
TF	Fall time	Time	0 ns
TD	Time delay	Time	0 ns
WINDOW	Window type		DEFAULT
DCVal	DC Value (Used for DC analysis)	Voltage	0
PIN_ID	Name identifier for pin	Text	

Implementation Details

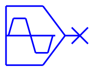
This is an active circuit termination that applies a load resistance specified by Z while supplying a sawtooth-wave power swept across the frequency range specified for the schematic containing the port. The port number P is set automatically. If PIN_ID is not empty, text is displayed on the pin when it is instantiated as a subcircuit.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Port with Square Wave Signal: PORT_SQR

Symbol



Parameters

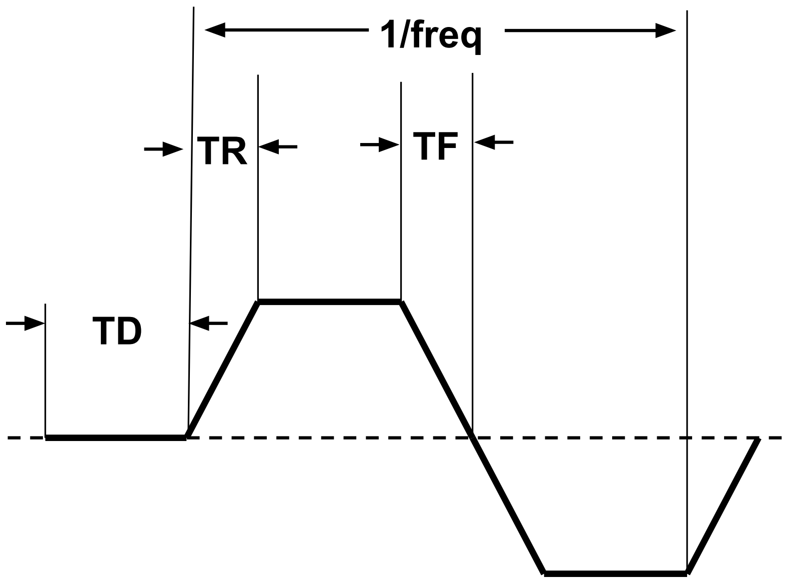
Name	Description	Unit Type	Default
P	Port number		0
Z	Termination impedance	Resistance	50 ohm
AMP	Open circuit signal magnitude	Voltage	0 V
TR	Rise time	Time	0 ns
TF	Fall time	Time	0 ns
TD	Time delay	Time	0 ns
WINDOW	Window type		DEFAULT
DCVal	DC Value (Used for DC analysis)	Voltage	0
PIN_ID	Name identifier for pin	Text	

Implementation Details

This is an active circuit termination that applies a load resistance specified by Z while supplying a square-wave power swept across the frequency range specified for the schematic containing the port. The port number P is set automatically. If PIN_ID is not empty, text is displayed on the pin when it is instantiated as a subcircuit.

Waveform

The waveform below shows how the delay, rise time, and fall time are defined for the square wave.

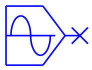


Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Dynamic Source Port: PORT_SRC

Symbol



Summary

The PORT_SRC element is a versatile port that encapsulates the functionality of many source ports into a single source port. A variety of signal types may be generated with this port, include sinusoid, piecewise linear, pulse, saw, square and triangle signals. In addition, you can specify what to use for frequency, how to calculate bandwidth/number of harmonics, the tone number and how to sweep power (for the sinusoid case).

This port combines the functionality of the following ports: PORT1, PORTF, PORTFN, PORTFNS, PORT_PS1, PORT_PLS, PORT_PWL, PORT_SAW, PORT_SQR and PORT_TRI.

Parameters

Name	Description	Unit Type	Default
P	Port number		1
Z	Termination impedance	ohm	50
Signal	Signal type		Sinusoid
SpecType	Specify type		Use doc freqs
SpecBW	Specify bandwidth		Use doc # harms
Sweep	Power sweep type (only used when Signal=Sinusoid)		
Tone	Tone number for the signal		1
*Hint	Hint		Auto
*NSamp	Oversample factor		1
PIN_ID	Name identifier for pin	Text	

** indicates a secondary parameter*

Parameter Details

Signal. Specifies the signal type as "Sinusoid", "PWL", "Pulse", "Saw", "Square" or "Triangle".

SpecType. Specifies whether to use the document frequencies or specify a frequency or time period. Choices are "Use doc freqs", "Specify freq", "Specify period", "Trans only (Freq)", or "Trans only (Period)". The Trans only settings can only be used with transient simulators.

SpecBW. Specifies whether to use the number of harmonics in the document settings or to specify the number of harmonics or minimum time resolution. Choices are "Use doc # harms", "Specify # harms" and "Min time res".

Sweep. Specifies whether power is swept. Choices are "None", "Vector", "Linear", and "Linear (# pts)".

Tone. Tone number for the signal.

Hint. If the type of signal is known (large signal or small signal), the solvers can take advantage of this hint to help solve the problem in the most efficient way. Choices are "Auto", "Large signal" and "Small signal".

NSamp. Oversample factor. This sets the oversampling factor at the specified tone. Specify a value of zero to indicate that this parameter does not override the value specified for the **Tone X Over Sample** in the project or document harmonic balance settings. (Choose **Options > Default Circuit Options** and click the **AWR Sim** tab.) This only applies when the Tone parameter is set to 1, 2, or 3 for the element. An error is issued if this parameter is set to zero for a source with Tone equal to 4 or more.

Dynamic Parameters

Name	Description	Unit Type	Default
Pwr	Available power	Power	{0} dBm
Ang	Relative angle of excitation	Angle	{0} Deg
Freq	Frequency for tone N	Frequency	1 GHz
Period	Period	Time	1 ns
NHarm	Maximum number of harmonics for this tone		5
MinRes	Minimum time resolution	Time	1 ns
PStart	Swept power magnitude start	Power	0 dBm
PStop	Swept power magnitude stop	Power	0 dBm
PStep	Swept power magnitude step	Power	0 dB
NumPts	Number of swept voltage points		2
DCVal	DC value (used for DC analysis)	Voltage	0 V
WINDOW	Window type		
TIME	Time vector	Time	{0,0.5,1} ns
VOLTAGE	Voltage vector	Voltage	{1,1,1} V
HI	High voltage level	Voltage	0.5 V
LO	Low voltage level	Voltage	-0.5 V
TD	Time delay	Time	0 ns
TW	Pulse width	Time	0 ns
TR	Rise time	Time	0 ns
TF	Fall time	Time	0 ns

Dynamic parameters are parameters whose existence depend on the settings of the static parameters. The following is a list all of possible parameters that may display for the PORT_SRC depending on the settings of the static parameters Signal, SpecType, SpecBW and Sweep.

Dynamic Parameter Details

Pwr. Displays when Signal is "Sinusoid" and Sweep is "None" or "Vector". If Sweep is "None", the values in the vector correspond to the power at each harmonic (the first value represents power at the fundamental, the second value represents power at the 2nd harmonic, and so on) similar to the PORT1 element (see PORT1 for more details). If Sweep is "Vector", each value represents a value in the power sweep.

Ang. Displays when Signal is "Sinusoid". Represents the relative angle of excitation. If Sweep is "None", the values in the vector correspond to the angle of each harmonic (first value represents the angle of the fundamental, the second value represents the angle of the 2nd harmonic, and so on) similar to the PORT1 element (see PORT1 for more details). If Sweep is "Vector", each value represents the angles of each value in the power sweep.

Freq. Displays when SpecType is "Specify freq". Allows you to specify a single source frequency directly on the port instead of having to use the document or project frequencies.

Period. Displays when SpecType is "Specify period". Allows the user to specify the time period of the source signal directly on the port instead of having to use the document or project frequencies.

NHarm. Displays when SpecBW is "Specify # harms". Allows you to specify the number of harmonics to use directly on the port instead of having to use the document or project setting. Specify a value of zero to indicate that this parameter does not override the value specified for the **Tone X Harmonics** in the project or document harmonic balance settings. (Choose **Options > Default Circuit Options** and click the **AWR Sim** tab.) This only applies when the Tone parameter is set to 1, 2, or 3 for the element. An error is issued if this parameter is set to zero for a source with Tone equal to 4 or more.

MinRes. Displays when SpecBW is "Min time res". Allows you to specify the time resolution directly on the port instead of having to use the document or project **Tone X Harmonics** setting.

PStart, PStop. Displays when *Signal=Sinusoid* and *Sweep=Linear or Linear* (# pts). Allows the user to specify the start and stop power values in a power sweep.

PStep. Displays when Signal is "Sinusoid" and Sweep is "Linear". Allows you to specify the power step value of a power sweep.

NumPts. Displays when Signal is "Sinusoid" and Sweep is "Linear (# pts)". Allows you to specify the number of points in a power sweep.

DCVal. Displays when Signal is set to anything except "Sinusoid". Allows you to specify a DC voltage value that can be used for DC analysis.

WINDOW. Displays when Signal is set to anything except "Sinusoid". Allows you to specify a data window type that will be applied to the source signal. Choices include "NONE", "DEFAULT", "TRIANG", "HANN", "HAMM", and "BLACK".

TIME and **VOLTAGE.** Displays when Signal is "PWL". See PORT_PWL for details on these parameters.

HI and **LO.** Displays when Signal is "Pulse", "Saw", "Square", or "Triangle". Represents the high and low voltage values of the source signal.

TD. Displays when Signal is "Pulse", "Saw", "Square", or "Triangle". Represents the time delay of the signal.

TW. Displays when Signal is "Pulse". Represents the pulse width in time units.

TR. Displays when Signal is "Pulse" or "Square". Represents the rise time.

TF. Displays when Signal is "Pulse", "Saw", or "Square". Represents the fall time.

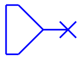
NOTE: If a sinusoidal nonlinear source like this one has the parameter Ang=0 and is ideally terminated, then a nonlinear measurement (for example, Pcomp or Vcomp) made at the fundamental output of that source will have an angle of -90deg. This discrepancy is consistent with the definition of the sine wave sources (as in SPICE), and the Fourier-based harmonic component measurements. To avoid confusion, always measure or calculate the gain, and plot its angle.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Network Terminated Circuit Port: PORT_TN

Symbol



Parameters

Name	Description	Unit Type	Default
P	Port number		0
NET	Network used for termination impedance	Text	name
NP	The port number to be used as the termination		1
PIN_ID	Name identifier for pin	Text	

Implementation Details

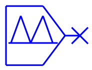
This is a passive circuit termination that applies a load specified as the port NP of the schematic named in NET. The port number P is set automatically. If PIN_ID is not empty, text is displayed on the pin when it is instantiated as a subcircuit.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Port with Triangle Wave Signal: PORT_TRI

Symbol



Parameters

Name	Description	Unit Type	Default
P	Port number		0
Z	Termination impedance	Resistance	50 ohm
AMP	Open circuit signal magnitude	Voltage	0 V
TD	Time delay	Time	0 ns
WINDOW	Window type		DEFAULT
DCVal	DC Value (Used for DC analysis)	Voltage	0
PIN_ID	Name identifier for pin	Text	

Implementation Details

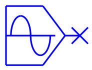
This is an active circuit termination that applies a load resistance specified by Z while supplying a triangle-wave power swept across the frequency range specified for the schematic containing the port. The port number P is set automatically. If PIN_ID is not empty, text is displayed on the pin when it is instantiated as a subcircuit.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Port with Defined Frequency/1-Tone HB Source (2nd Tone): PORTF

Symbol



Parameters

Name	Description	Unit Type	Default
P	Port number		0
Z	Termination impedance	Resistance	50 ohm
Freq	Frequency for tone 2	Frequency	1 GHz
Pwr	Power in magnitude	DB Power	0 dBm
*Ang	Relative angle of the excitation	Angle	0 Deg
PIN_ID	Name identifier for pin	Text	

Implementation Details

This is an active circuit termination that applies a load resistance specified by Z while supplying a power signal of magnitude Pwr and angle Ang and fixed frequency Freq. The port number P is set automatically. If PIN_ID is not empty, text is displayed on the pin when it is instantiated as a subcircuit.

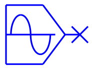
NOTE: If a sinusoidal nonlinear source like this one has the parameter Ang=0 and is ideally terminated, then a nonlinear measurement (e.g. Pcomp or Vcomp) made at the fundamental output of that source will have an angle of -90deg. This discrepancy is consistent with the definition of the sine wave sources (as in SPICE), and the Fourier based harmonic component measurements. To avoid confusion, always measure or calculate the gain, and plot its angle.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

Port with Defined Frequency at Tone N: PORTFN

Symbol



Parameters

Name	Description	Unit Type	Default
P	Port number		0
Z	Termination impedance	Resistance	50 ohm
Freq	Frequency for tone N	Frequency	1 GHz
Pwr	Available power	DB Power	0 dBm
Ang	Relative angle of the excitation	Angle	0 Deg
Tone	Tone number for the signal		2
*NHarm	Maximum number of harmonics for this tone		0
*NSamp	Sampling rate for this tone		1
PIN_ID	Name identifier for pin	Text	

Parameter Details

NHarm & NSamp. Specify a value of zero to indicate that these parameters do not override the values specified for the **Tone X Harmonics** and **Tone X Over sample** in the project or document harmonic balance settings. (Choose **Options > Default Circuit Options** and click the **AWR Sim** tab.) This only applies when the Tone parameter is set to 1, 2, or 3 for the element. An error is issued if these parameters are set to zero for a source with Tone equal to 4 or more.

Implementation Details

PORTFN is a source of available power *Pwr*, at a relative angle *Ang*, and with a port impedance *Z*. The tone number and the fundamental frequency of that tone may be specified by means of Tone and Freq parameters, respectively.

NOTE: If a sinusoidal nonlinear source like this one has the parameter Ang=0 and is ideally terminated, then a nonlinear measurement (e.g. Pcomp or Vcomp) made at the fundamental output of that source will have an angle of -90deg. This discrepancy is consistent with the definition of the sine wave sources (as in SPICE), and the Fourier based harmonic component measurements. To avoid confusion, always measure or calculate the gain, and plot its angle.

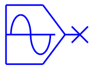
The port number P is set automatically. If PIN_ID is not empty, text is displayed on the pin when it is instantiated as a subcircuit.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Port with Defined Frequency at Tone N (Swept Power): PORTFNS

Symbol



Parameters

Name	Description	Unit Type	Default
P	Port number		0
Z	Termination impedance	Resistance	50 ohm
Freq	Frequency for tone N	Frequency	1 GHz
PStart	Swept power magnitude start	DB Power	0 dBm
PStop	Swept power magnitude stop	DB Power	0 dBm
PStep	Swept power magnitude step	DB	0 dB
Tone	Tone number for the signal		2
*NHarm	Maximum number of harmonics for this tone		0
*NSamp	Sampling rate for this tone		1
PIN_ID	Name identifier for pin	Text	

** indicates a secondary parameter*

Parameter Details

NHarm & NSamp. Specify a value of zero to indicate that these parameters do not override the values specified for the **Tone X Harmonics** and **Tone X Over sample** in the project or document harmonic balance settings. (Choose **Options > Default Circuit Options** and click the **AWR Sim** tab.) This only applies when the Tone parameter is set to 1, 2, or 3 for the element. An error is issued if these parameters are set to zero for a source with Tone equal to 4 or more.

Implementation Details

PORTFNS is similar to PORTFN, but more general in that a power sweep may be specified. The available source power is swept in increments of *PStep*, starting at *PStart* and ending at *PStop*.

NOTE: If a sinusoidal nonlinear source like this one has the parameter Ang=0 and is ideally terminated, then a nonlinear measurement (e.g. Pcomp or Vcomp) made at the fundamental output of that source will have an angle of -90deg. This discrepancy is consistent with the definition of the sine wave sources (as in SPICE), and the Fourier based harmonic component measurements. To avoid confusion, always measure or calculate the gain, and plot its angle.

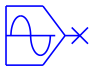
The port number P is set automatically. If PIN_ID is not empty, text is displayed on the pin when it is instantiated as a subcircuit.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Port with Defined Frequency/1-Tone Volterra Source (2nd Tone): PORTFV

Symbol



Parameters

Name	Description	Unit Type	Default
P	Port number		0
Z	Termination impedance	Resistance	50 ohm
Freq	Frequency for tone 2	Frequency	1 GHz
Pwr	Power in magnitude	DB Power	0 dBm
*Ang	Relative angle of the excitation	Angle	0 Deg

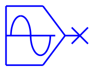
** indicates a secondary parameter*

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

Port with Modulated Signal (Data File-based): PORTIQ_ENV

Symbol



Summary

PORTIQ_ENV is a modulated carrier source for Circuit Envelope analysis. The modulation is specified by a data file under the Project Browser **Data Files** node.

Parameters

Name	Description	Unit Type	Default
P	Port number		1
Z	Termination impedance	Resistance	50 ohm
Pwr	Available power	DB Power	
NET	Modulated signal data file name	String	
COL	Data file column to use		1
TCOL	Data file column for time (optional)		
Tone	Tone number for the signal		1
Freq	Tone frequency	Frequency	1 GHz
PIN_ID	Name identifier for pin	Text	

Parameter Details

COL. The base column from the data file to use. The left-most column is column 1. The values in this column are combined with the values in the following column if they can form complex value pairs, depending on the heading of the data file. See SRC_C for an example of the data file format.

TCOL. Optional column of time values. If empty, an attempt is made to determine the modulating signal time step from the data file. The time step can be declared in the data file by using the TSTEP or SMPFRQ tags.

Freq. The frequency of the tone. Can be specified if tone > 1.

Implementation Details

PORTIQ_ENV is a modulated carrier source for Circuit Envelope analysis. The RF carrier is centered at the specified tone frequency or the project frequency if tone is 1. The available signal power is given by Pwr and the port impedance is given by Z.

The modulation signal is specified in the data file referred to by the NET parameter. The file must be imported into the NI AWRDE as a text data file type. The file specifies the in-phase and quadrature components of the modulating complex signal as a function of time. See SRC_C for an example of the data file format. The modulating signal is scaled such that the total available power is equal to Pwr.

You can use the VSS FILE_SNK block to generate the files used by this source. If you do use this block, on the Element Options dialog box **Setup** tab, set **Destination** to **Write data file to project tree**, set **File format** to “Text Data File”, and make sure to specify enough **Duration** to properly capture the signal.

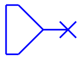
The port number P is set automatically. If PIN_ID is not empty, text is displayed on the pin when it is instantiated as a subcircuit.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

Gamma Terminated Circuit Port (50 Ohm Ref): PORTG

Symbol



Parameters

Name	Description	Unit Type	Default
P	Port number		0
GM	Magnitude of the termination gamma		0
GA	Angle of the termination gamma	Angle	0 Deg
PIN_ID	Name identifier for pin	Text	

Implementation Details

The port element PORTG allows the termination impedance of the port to be specified as a magnitude and angle of reflection coefficient. The default value of GM is .99 rather than 1 because it is impossible to compute the response of the circuit with perfect magnitude.

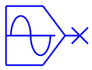
The port number P is set automatically. If PIN_ID is not empty, text is displayed on the pin when it is instantiated as a subcircuit.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Port with Modulated Signal: PORTMOD

Symbol



Summary

PORTMOD is a modulated carrier source. The frequency components of the modulated signal are specified by a data file on your computer. PORTMOD_F is a more advanced source that NI AWR recommends over the PORTMOD source.

Parameters

Name	Description	Unit Type	Default
P	Port number		0
Z	Termination impedance	Resistance	50 ohm
Pwr	Available power	DB Power	0 dBm
SIG	Modulated signal file name	Text	QPSK
FRes	Frequency resolution	Frequency	0.1 GHz
WINDOW	Window type		DEFAULT
PIN_ID	Name identifier for pin	Text	

Implementation Details

PORTMOD is a modulated carrier source. The carrier is centered at the project frequency (or nonlinear frequency); the available signal power is given by Pwr; the port impedance is given by Z; and the frequency resolution (resolution bandwidth) of the modulation signal (the complex envelope) is given by Fres.

The modulation signal is specified in the SIG text file, which should be placed in the */signals* subdirectory of the AWR program directory. The file specifies the frequency components of the complex envelope as follows:

- The magnitude of the frequency components is expressed in dBm and is placed in the first column.
- The phase of the frequency components is expressed in degrees and is placed in the second column.
- The components are specified in the order of increasing frequency.
- If the envelope spectrum is non-symmetric, the file should include the header DOUBLE_SIDED. If the envelope spectrum is symmetric, you can choose to specify only the right-hand side of the spectrum by including the header CONJ_SYM.
- The file name extension must be *.sig*.

The frequency components, as specified in the signal file, are scaled so that the total power is equal to Pwr. You do not, therefore, need to be concerned with the scaling of the signal components.

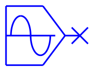
The port number P is set automatically. If PIN_ID is not empty, text is displayed on the pin when it is instantiated as a subcircuit.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Port with Modulated Signal (Data File-based): PORTMOD_ENV

Symbol



Summary

PORTMOD_ENV is a modulated carrier source for Circuit Envelope analysis. The frequency components of the modulated signal are specified by a data file under the **Data Files** node in the Project Browser.

Parameters

Name	Description	Unit Type	Default
P	Port number		1
Z	Termination impedance	Resistance	50 ohm
Pwr	Available power	DB Power	0 dBm
NET	Modulated signal data file name	String	
FRes	Frequency resolution	Frequency	
WINDOW	Window type		DEFAULT
Tone	Tone number for the signal		1
Freq	Tone frequency	Frequency	1 GHz
PIN_ID	Name identifier for pin	Text	

Parameter Details

FRes. If empty, an attempt is made to determine the frequency resolution from the data file. The frequency resolution can be declared in the data file by using the FREQ tag. For example: FREQ=12500

WINDOW. Window that can be applied to the data. Options are NONE, DEFAULT, TRIANG, HANN, HAMM and BLACK.

Freq. The frequency of the tone. Can be specified if tone > 1.

Implementation Details

PORTMOD_ENV is a modulated carrier source for Circuit Envelope analysis. The RF carrier is centered at the specified tone frequency or the project frequency if tone is 1; the available signal power is given by Pwr; the port impedance is given by Z; and the frequency resolution (resolution bandwidth) of the modulation signal (the complex envelope) is given by FRes.

The modulation signal is specified in the data file referred to by the NET parameter. The file must be imported into the NI AWRDE as a text data file type. The file specifies the frequency components of the complex envelope as follows:

- The magnitude of the frequency component is expressed in dBm and is placed in the first column
- The phase of the frequency component is expressed in degrees and is placed in the second column

- If the envelope spectrum is non-symmetric, the file should include the tag SPECTRUM=DOUBLE_SIDED. If the envelope spectrum is symmetric, you can choose to specify only the right-hand side of the spectrum by including the tag SPECTRUM=CONJ_SYM.

The frequency components, as specified in the data file, are scaled so that the total available power is equal to Pwr. See PORTMOD_F for an example of the data file format.

You can use the VSS FILE_SNK block to generate the files used by this source. If you do use this block, on the Element Options dialog box **Setup** tab, set **Destination** to **Write data file to project tree**, set **File format** to “Microwave Office Signal File”, and make sure to specify enough **Duration** to properly capture the signal.

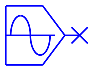
The port number P is set automatically. If PIN_ID is not empty, text is displayed on the pin when it is instantiated as a subcircuit.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

Port with Modulated Signal (Data File-Based): PORTMOD_F

Symbol



Summary

PORTMOD_F is a modulated carrier source. The frequency components of the modulated signal are specified by a data file under the **Data Files** node in the Project Browser.

Parameters

Name	Description	Unit Type	Default
P	Port number		0
Z	Termination impedance	Resistance	50 ohm
Pwr	Available power	DB Power	0 dBm
NET	Modulated signal file name	String	"M"
FRes	Frequency resolution	Frequency	
WINDOW	Window type		DEFAULT
PIN_ID	Name identifier for pin	Text	

Parameter Details

FRes. If empty, an attempt is made to determine frequency resolution from the data file. The frequency resolution can be declared in the data file by using the FREQ tag. An example is:

FREQ=12500

WINDOW. Window that can be applied to the data. Choices are NONE, DEFAULT, TRIANG, HANN, HAMM and BLACK.

Implementation Details

PORTMOD_F is a modulated carrier source. The carrier is centered at the project frequency (or nonlinear frequency); the available signal power is given by Pwr; the port impedance is given by Z; and the frequency resolution (resolution bandwidth) of the modulation signal (the complex envelope) is given by Fres.

When using the PORTMOD_F source, you are performing a two-tone simulation. Tone 1 is the center frequency of the modulation and Tone 2 defines the modulation signal tones. You must make sure to set up enough tone-two harmonics to properly capture all the modulation tones.

The modulation signal is specified in the data file referred to by the NET parameter. Currently, you must type the name of the NET parameter with quotes for it to be recognized. The file must be imported into the NI AWRDE as a text data file type. The file specifies the frequency components of the complex envelope as follows:

- The magnitude of the frequency components is expressed in dBm and is placed in the first column.
- The phase of the frequency components is expressed in degrees and is placed in the second column.

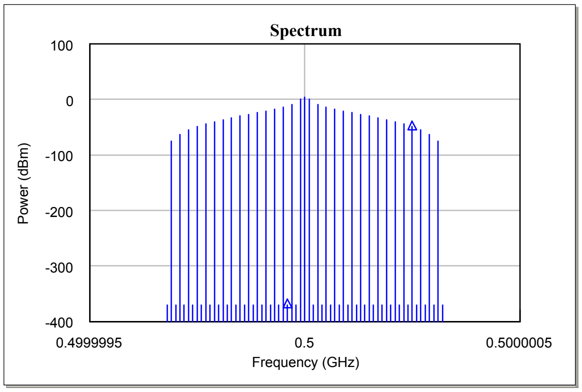
- The components are specified in the order of increasing frequency.
- If the envelope spectrum is non-symmetric, the file should include the tag SPECTRUM=DOUBLE_SIDED. If the envelope spectrum is symmetric, you can choose to specify only the right-hand side of the spectrum by including the tag SPECTRUM=CONJ_SYM.

The frequency components, as specified in the signal file, are scaled so that the total power is equal to Pwr. You do not, therefore, need to be concerned with the scaling of the signal components.

The following is a portion of a modulated signal data file for a square wave. When used with PORTMOD_F this generates a pulsed RF source with 50% duty cycle, where the project frequency defines the RF frequency and FRES defines the square wave period:

```
SPECTRUM=CONJ_SYM
-6.0205999132796 0
-9.9395089618129 92.8125
-6.14e3 0
-19.454005916507 98.4375
-6.14e3 0
-23.835016465249 104.0625
-6.14e3 0
-26.673358097512 109.6875
-6.14e3 0
-28.74344117002 115.3125
-6.14e3 0
-30.344597920947 120.9375
-6.14e3 0
-31.624141340467 126.5625
-6.14e3 0
-32.665282370505 132.1875
-6.14e3 0
-33.519390719192 137.8125
-6.14e3 0
-34.220154921542 143.4375
-6.14e3 0
-34.790597408595 149.0625
-6.14e3 0
-35.246865213854 154.6875
-6.14e3 0
-35.600412484374 160.3125
-6.14e3 0
-35.859314018019 165.9375
-6.14e3 0
-36.029075371558 171.5625
-6.14e3 0
-36.113130638856 177.1875
-6.14e3 0
```

When this file is used with FRES set to 10 Hz and 32 harmonics set up for tone 2 without limiting any higher order tones, the input power spectrum of this source displays as follows:



You can use the VSS FILE_SNK block to generate the files used by this source. If you do use this block, the important parameters to set are **Write data file to project tree**, **File** set to "Microwave Office Signal File", and make sure to have enough **Duration** to properly capture the signal.

Examples of using the PORTMOD_F source are found in the "Modulation_Signals.emp" example project file.

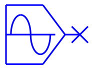
The port number P is set automatically. If PIN_ID is not empty, text is displayed on the pin when it is instantiated as a subcircuit.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

Port with NPR Signal: PORTNPR

Symbol



Parameters

Name	Description	Unit Type	Default
P	Port number		1
Z	Termination impedance	Resistance	50 ohm
Pwr	Available power	dB Power	0 dBm
BW	Signal Bandwidth	Frequency	$1*10^8$
NotchPercent	Notch size as percent BW	Percent	10%
NotchFreqs	Number of frequencies in the notch	Integer	10
PIN_ID	Name identifier for pin	Text	

Implementation Details

PORTNPR is defined in the frequency domain as follows: It is a two-tone source modulated source. The carrier frequency (tone 1) is the document frequency, or the current frequency of the frequency sweep. The tone 2 frequencies are distributed uniformly in the specified bandwidth BW. Its signal consists of N harmonics with identical amplitude but random phase selected by a generator of pseudorandom numbers with uniform distribution from -π to π. N is determined as N=NothFreqs/(NothPercent/100). Amplitude is determined such that the total available power of the port is equal to Pwr. In the center of the specified bandwidth there is a notch such that harmonics corresponding to the notch have zero amplitude.

Consider the PORTNPR with default values of its parameters loaded with the matching load shown in the following figure:

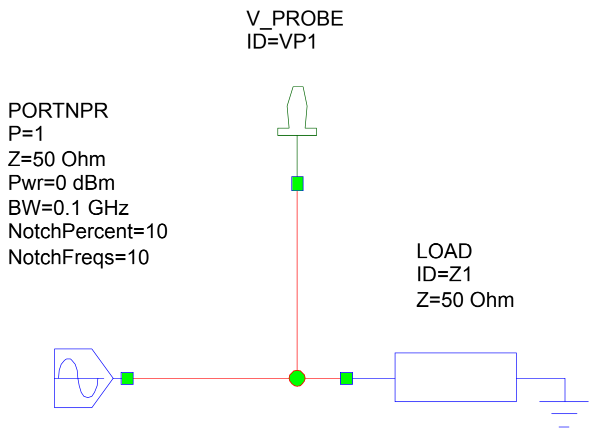


Figure 1. PORTNPR with default parameters loaded with a matching load. The document frequency is 1 GHz.

The spectrum of this PORTNPR measured at the voltage probe is

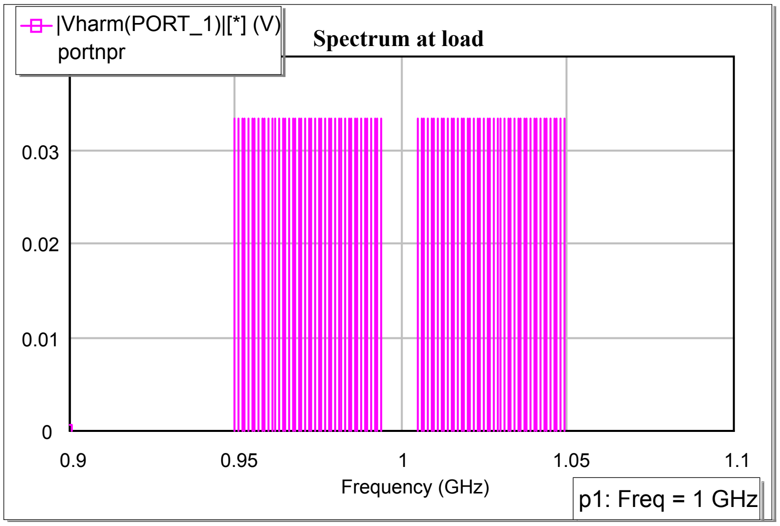


Figure 2. Amplitude voltage spectrum of PORTNPR in Fig.1 seen at the load

The port number P is set automatically. If PIN_ID is not empty, text is displayed on the pin when it is instantiated as a subcircuit.

Notes

Increasing the total number of harmonics, N, yields more repeatable simulation results at the expense of simulation speed. Note that it is necessary to set Harmonic Balance options with relatively few tone 1 harmonics (e.g., 3 to 5) and a large number of tone 2 harmonics (on the order of several hundred). The objective of the simulation is usually to detect 3rd order intermods. In this case the expression for the number of tone 2 harmonics is Nk/2, where k is the order of intermods (k=3 for 3rd order intermods in this example).

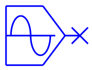
In the example in figure 1, N= 100 and in order to see the 3rd order intermods, you need 150 tone 2 harmonics. It is also important to set maximum harmonics equal to tone 2 harmonics, again because of speed. The applied harmonic setting should be done at the schematic level in the schematic's **Options...** settings. Available from right clicking the schematic and selecting **Options...** and navigating to the APLAC Sim tab and locating the **Harmonic Balance Options** section. By using the individual schematic settings the large amount of Tone 2 harmonics will not be applied to any other schematic simulations, which if set global would dramatically increase simulation times.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

Port with Fourier-specified Signal: PORTSIG

Symbol



Summary

PORTSIG is a signal source. The frequency components of a single-tone signal are specified by a data file on your computer. You can use this source if you want to control the power levels of each harmonic of a source with arbitrary values. For example, you can generate a square wave (however, NI AWR has square wave sources that make this much easier than using this source).

Parameters

Name	Description	Unit Type	Default
P	Port number		1
Z	Termination impedance	Resistance	50 ohm
Pwr	Available power	DB Power	0 dBm
SIG	Signal file name	Text	SQR_WAVE
WINDOW	Window type		DEFAULT
PIN_ID	Name identifier for pin	Text	

Implementation Details

PORTSIG supplies a signal whose frequency components are given in the data file specified by the SIG parameter. The available signal power is Pwr and the port impedance is Z. The port number P is set automatically. If PIN_ID is not empty, text is displayed on the pin when it is instantiated as a subcircuit.

See PORTMOD for details on the signal file format. The signal file formats are identical with the exception of an optional token used in conjunction with eye diagram displays. If the signal file contains the spectrum of a symbol sequence, and if you want to view eye diagrams of the circuit waveforms, you must add the number of symbols to the signal file by adding the statement:

NSYMB = *number of symbols*

on the line following the CONJ_SYM/DOUBLE_SIDED specification.

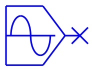
NOTE: When performing eye diagram simulations, NI AWR recommends you first use the PORT_PRBS or PORT_ARBS sources.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Port with Fourier-specified Signal (Data File-based): PORTSIG_F

Symbol



Summary

PORTSIG_F is a signal source. The frequency components of a single-tone signal are specified by a data file under the **Data Files** node in the Project Browser. You can use this source if you want to control the power levels of each harmonic of a source with arbitrary values. For example, you can generate a square wave (however, NI AWR has square wave sources that make this much easier than using this source).

Parameters

Name	Description	Unit Type	Default
P	Port number		1
Z	Termination impedance	Resistance	50 ohm
Pwr	Available power	DB Power	0 dBm
NET	Signal data file name	String	"M"
WINDOW	Window type		DEFAULT
PIN_ID	Name identifier for pin	Text	

Implementation Details

PORTSIG_F supplies a signal whose frequency components are given in the data file specified by the NET parameter. The available signal power is Pwr and the port impedance is Z. The port number P is set automatically. If PIN_ID is not empty, text is displayed on the pin when it is instantiated as a subcircuit.

When using the PORTSIG_F source, you are setting up a Tone 1 source. You must be sure to have enough tone 1 harmonics to properly capture all the components specified in the file.

See PORTMOD_F for details on the signal file format. The signal file formats are identical with the exception of an optional token used in conjunction with eye diagram displays. If the signal file contains the spectrum of a symbol sequence, and if you want to view eye diagrams of the circuit waveforms, you must add the number of symbols to the data file by adding the statement:

NSYMB=[*number of symbols*]

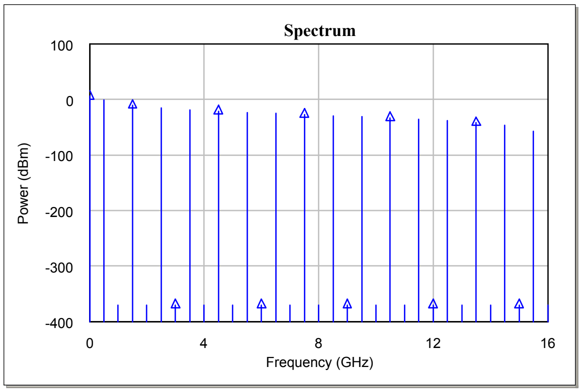
NOTE: When performing eye diagram simulations, NI AWR recommends you first use the PORT_PRBS or PORT_ARBS sources.

The following is a portion of a signal data file for a square wave:

```
SPECTRUM=CONJ_SYM
-6.0205999132796 0
-9.9395089618129 92.8125
-6.14e3 0
-19.454005916507 98.4375
```

```
-6.14e3 0
-23.835016465249 104.0625
-6.14e3 0
-26.673358097512 109.6875
-6.14e3 0
-28.74344117002 115.3125
-6.14e3 0
-30.344597920947 120.9375
-6.14e3 0
-31.624141340467 126.5625
-6.14e3 0
-32.665282370505 132.1875
-6.14e3 0
-33.519390719192 137.8125
-6.14e3 0
-34.220154921542 143.4375
-6.14e3 0
-34.790597408595 149.0625
-6.14e3 0
-35.246865213854 154.6875
-6.14e3 0
-35.600412484374 160.3125
-6.14e3 0
-35.859314018019 165.9375
-6.14e3 0
-36.029075371558 171.5625
-6.14e3 0
-36.113130638856 177.1875
-6.14e3 0
```

When using this file with 32 harmonics set up for tone 1 without limiting any higher order tones, the input power spectrum of this source displays as:



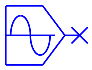
You can use Visual System Simulator™ (VSS) to generate the files this source uses with the FILE_SNK block. If you do use this block, select the **Write data file to project tree** parameter, set **File** to "Microwave Office Signal File", and make sure to have enough **Duration** to properly capture the signal.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Port with Fourier-specified Signal (Tone N): PORTSIGF

Symbol



Parameters

Name	Description	Unit Type	Default
P	Port number		1
Z	Termination impedance	Resistance	50 ohm
Pwr	Available power	DB Power	0 dBm
SIG	Signal file name	Text	SQR_WAVE
Freq	Frequency for tone N	Frequency	1 GHz
Tone	Tone number for the signal		2
WINDOW	Window type		Default
PIN_ID	Name identifier for pin	Text	

Implementation Details

This element is similar to PORTSIG, but more general in that you can specify a fundamental tone and the frequency of the fundamental tone by means of the Tone and Freq parameters.

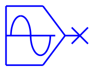
When using the PORTSIGF source, you are defining which tone sets use this signal file. You must be sure to have enough harmonics specified for that tone to properly capture all the components specified in the file.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Port with Fourier-Specified Signal (Tone N, Data File-Based): PORTSIGF_F

Symbol



Summary

PORTSIGF_F is a signal source similar to PORTSIG_F but more general in that a fundamental tone and the frequency of the fundamental tone may be specified. Please refer to the help for PORTSIG_F for more details.

Parameters

Name	Description	Unit Type	Default
P	Port number		1
Z	Termination impedance	Resistance	50 ohm
Pwr	Available power	DB Power	0 dBm
NET	Signal data file name	String	"M"
Freq	Frequency for tone N	Frequency	1 GHz
Tone	Tone number for the signal		2
WINDOW	Window type		Default
PIN_ID	Name identifier for pin	Text	

Implementation Details

When using the PORTSIGF_F source, you are defining which tone sets will use this signal file. You must be sure to have enough harmonics specified for that tone to properly capture all the components specified in the file. The port number P is set automatically. If PIN_ID is not empty, text is displayed on the pin when it is instantiated as a subcircuit.

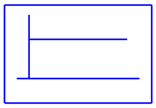
Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Simulation Control

Constant Output Power Control Block: CONSTPOUT

Symbol



Summary

CONSTPOUT allows you to set up a schematic such that the measurements are calculated normally, but the output power is constant for each sweep point. CONSTPOUT changes the input power (DC or RF) in order to achieve the specified output power. You need to specify the power measurement component and frequency indices. This element only works with the APLAC Harmonic Balance simulator.

Parameters

Name	Description	Unit Type	Default
ID	Element ID		PO1
IN	ID of the source element		PORT_P1
OUT	ID of the power measurement element		PORT_P2
POUT	Target output level	dBm	-30
MEAS	Target power type		Function of Harmonics
SOURCE_TYPE	Source harmonic component that is allowed to change		HB
ERR	Maximum allowed error in output power		0.01
*MAX_ITER	Maximum number of iterations		100
*MIN	Lower limit for the source value (relative to the normal value)		0
*MAX	Upper limit for the source value (relative to the normal value)		10000
RESET	Reset source nominal value at each sweep point		No

** indicates a secondary parameter*

Parameter Details

IN: Specifies the ID of the input (source) element(s), the amplitude of which is modified. It is limited to sources and port elements. You can specify multiple sources by using a comma separated list (for example, {Port_P1, Port_P2}).

OUT: Specifies the ID of the output measurement element. The output measurement element can be a Port or P_METER3 element.

MEAS: Specifies the output power measurement function.

- Function of Harmonics: Measures the power at the harmonic(s) specified in the FUNC parameter.
  - In APLAC syntax, "f1" means tone 1, "f2" means tone 2, and "f3" means tone 3, etc. Additionally, "2f1" means the second harmonic of tone 1, and "3f1" means the third harmonic of tone 1, etc.

- The "P()" function means power at the specified tone. For instance, "P(f1)" means the power at tone 1 and P("f1")+P("f2") means the sum of power at tone 1 and tone 2.
- Adding quotes around the tone specification is optional (i.e. P("f1") vs. P(f1)).

- All Harmonics in Band: Measures the total power in a user-specified bandwidth in the LOWERFREQ and UPPERFREQ parameters.

MAXITER: Specifies the maximum number of optimization iterations for each sweep point that is used to achieve the constant output power.

MIN: Specifies the minimum scale factor applied to the source. The default value is 0; the same as -800 dBm, 0 W or 0 V based on the source.

MAX: Specifies the maximum scale factor applied to the source. The default value is 10 which is 10x the nominal voltage or 10x the nominal power based on the source.

NOTE: The Pout parameter can be swept, allowing you to measure the circuit characteristics at several constant output power levels simultaneously.

Dynamic Parameters

Dynamic parameters are parameters whose existence depends on the settings of the static parameters. The following is a list of all possible CONSTPOUT parameters, depending on the settings of MEAS.

Name	Description	Unit Type	Default
LOWERFREQ	Lower limit for frequency	Frequency	0 GHz
UPPERFREQ	Upper limit for frequency	Frequency	1000 GHz
FUNC	Target output power function, e. g. P("f1")+P("f2")		P("f1")

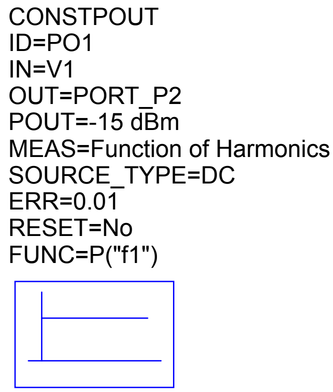
Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Examples

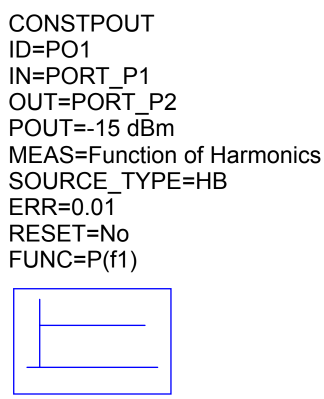
SINGLE-TONE ANALYSIS, VARY SINGLE DC SOURCE

The following figure depicts a CONSTPOUT block set up to vary a DC source in a single-tone Harmonic Balance analysis. SOURCE_TYPE is set to DC, therefore CONSTPOUT only varies the DC component of the input power (the V1 source) to achieve the specified output power.



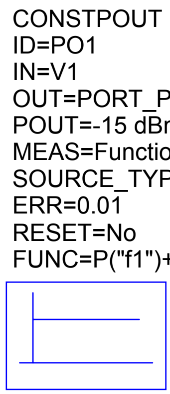
SINGLE-TONE ANALYSIS, VARY SINGLE AC SOURCE

The following figure depicts CONSTPOUT set up to vary an AC source in a single-tone Harmonic Balance analysis. SOURCE_TYPE is set to HB, therefore CONSTPOUT only varies the AC component of the input power (the PORT_P1 source) to achieve the specified output power.



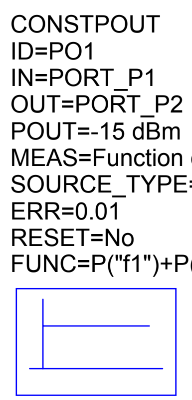
TWO-TONE ANALYSIS, VARY SINGLE DC SOURCE

The following figure depicts CONSTPOUT set up to vary a DC source in a two-tone Harmonic Balance analysis. SOURCE_TYPE is set to DC, therefore CONSTPOUT only varies the DC component of the input power to achieve the specified output power. There are two fundamental tones in the circuit. Their total power is measured as shown in the following figure.



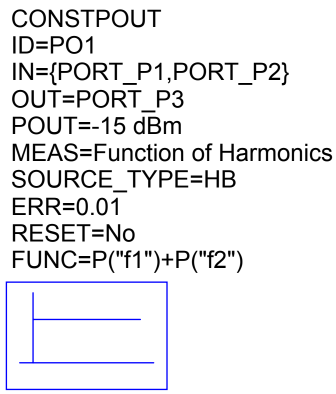
TWO-TONE ANALYSIS, VARY SINGLE AC SOURCE

The following figure depicts CONSTPOUT set up to vary an AC source in a two-tone Harmonic Balance analysis. SOURCE_TYPE is set to HB, therefore CONSTPOUT only varies the AC component of the input power to achieve the specified output power. There are two fundamental tones in the circuit. The total power of these tones is measured as shown in the following figure.



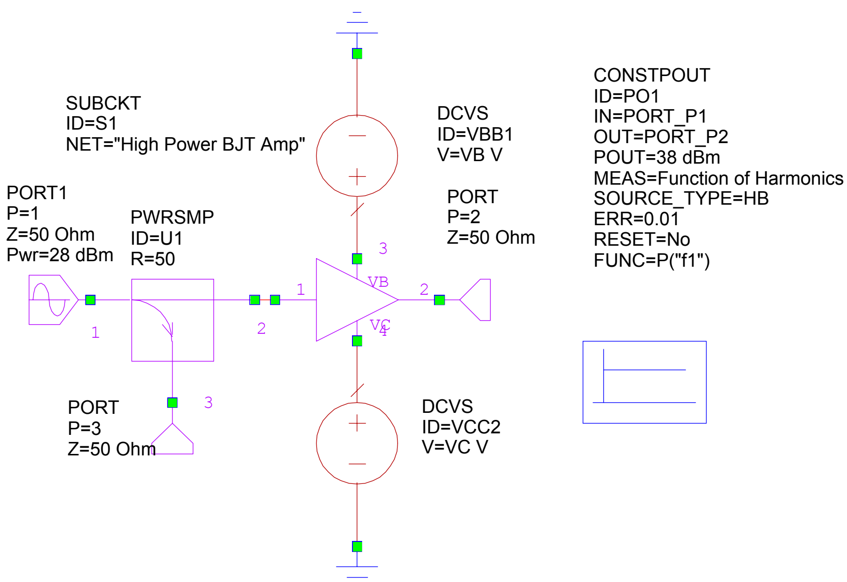
TWO-TONE ANALYSIS, VARY MULTIPLE AC SOURCES

The following figure depicts CONSTPOUT set up to vary multiple input AC sources in a two-tone Harmonic Balance analysis. SOURCE_TYPE is set to HB, therefore CONSTPOUT only varies the AC component of the two input power sources (Port_P1 and PORT_P2) to achieve the specified output power.

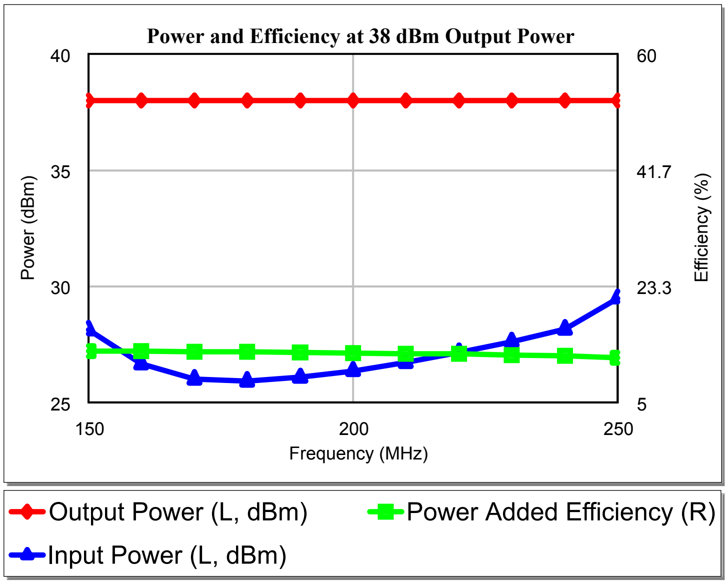


Examples

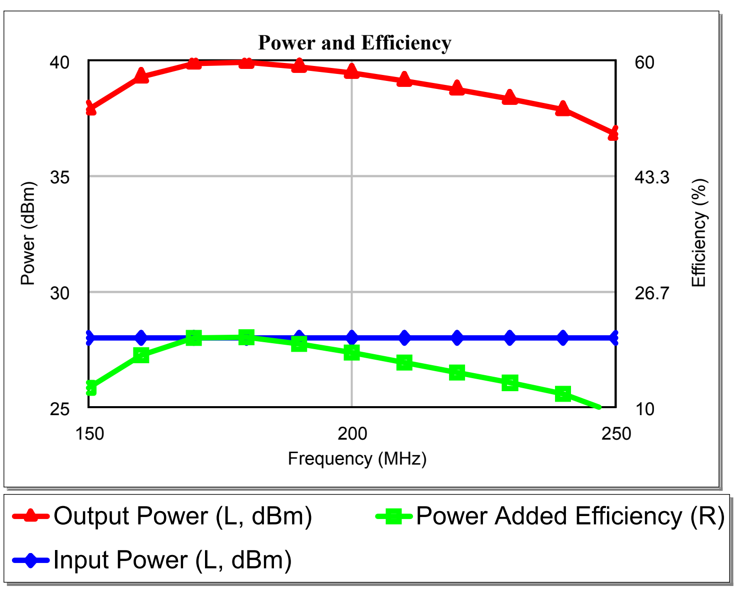
The following figure shows how to use the CONSTPOUT element in a schematic. In this example, the CONSTPOUT element is set up for 38 dBm constant output power. The PWRSMP element is connected in the schematic to measure the incident input power.



The following figure shows the power and efficiency for the previous schematic. Because the CONSTPOUT element is present in the schematic, the output power is held constant at 38 dBm. The input power and PAE are calculated at this output power.

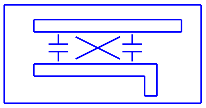


If the same schematic is simulated without the CONSTPOUT element, it results in the following figure. In this case, the output power varies over the range of frequency and the input power is constant.



Extraction Control Block: EXTRACT

Symbol



Summary

The EXTRACT block is a simulation control that allows a group of associated schematic elements to be electrically modeled via a physical simulation (for example, EM simulation or parasitic extraction) of the layout of these components. Upon simulating, the layout cells of all of the associated components are ported to an EM document and simulated. After this simulation is complete the electrical results are automatically merged back into the schematic and simulation of the entire schematic is performed. See *"EM: Creating EM Structures with Extraction"* for a detailed explanation of the EM-extraction process.

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	EX1
EM_Doc	Name for EM structure	String	"EM_Extract_Doc"
Name	Extraction Group Name	String	"EM_Extract"
Simulator	EM Simulator	N/A	{Choose}
*X_Dimension	X dimension of the enclosure for the EM_Doc	Length	N/A[a]
*Y_Dimension	Y dimension of the enclosure for the EM_Doc	Length	N/A[a]
X_Cell_Size	Enclosure X Cell Size	Length	1 um
Y_Cell_Size	Enclosure Y Cell Size	Length	1 um
Port Type	Port Type	N/A	Default[a]
STACKUP	STACKUP name	N/A	
*Create_Enclosure	Create enclosure and stack-up	N/A	Yes
*Create_Shapes	Create shapes for all layout objects	N/A	Yes
Extension	Extension distance	um	100 um[a]
Override_options	Set EM document options each time the EM document is generated	N/A	Yes
Hierarchy	When On, this extraction group generates an individual EM document and is not part of a higher level document.	N/A	Off
*Extract_LineTypes	When No (off), flattens cells or shapes drawn with line types.	N/A	No
*Union_Shapes	When Yes (on), extracted shapes are unioned into the EM document.	N/A	No[b]
*Extract_PinShapes	When Yes (on), extracts shapes of area pins into the EM document.	N/A	No[b]
*Explicit_Ground	Specify how an edge port is connected to ground.	N/A	None[c]

Name	Description	Unit Type	Default
SweepVar_Names	Specify the variable names of any SWPVAR blocks associated with this extraction.		

[a]This parameter is not available when ACE or AXIEM is the selected simulator.
[b]This parameter is not available when ACE is the selected simulator.
[c]This parameter is only available for AXIEM.

** indicates a secondary parameter*

Parameter Details

EM_Doc: The name of the EM document that performs the extraction. If there is no EM document with this name a document is automatically created.

Name: The name of the extraction group. This name allows schematic elements to be associated with this extraction. Association with the extraction group is specified by clicking the **Model Options** tab on the Element Options dialog box of the element you want to associate with the extraction group. On this tab, select the **Enable** check box and specify the **Group name** enclosed in quotes. If there are multiple groups defined, you can enter multiple names separated by commas. For example, if groups a, b, and c are used for various models or shapes in the design, you can type **{"a","b","c"}** (including the curly bracket and quotes) to include all of these groups in one EM structure.

Simulator: The type of EM simulator to use for the extraction. The following options are available:

- **Default** - A dialog box allows you to select the EM simulator type from a list of all the simulators integrated via the EM socket.
- **EMSight** - NI AWR's native EM simulator (EMSight) performs the simulation.
- **OEA** - OEA's Net-An parasitic network extractor performs the simulation.
- **ACE** - Automated Circuit Extraction. NI AWR's transmission line extractor performs the simulation.
- **AXIEM** - NI AWR's native EM simulator (AXIEM) performs the simulation

X_Dimension: Supplies an alternate way of specifying the X dimension of the enclosure for the specified EM document. The Extension parameter is the preferred method for setting the enclosure size.

Y_Dimension: Supplies an alternate way of specifying the Y dimension of the enclosure for the specified EM document. The Extension parameter is the preferred method for setting the enclosure size.

X_Cell_Size: Determines the X dimension of a single cell for the specified EM document.

Y_Cell_Size: Determines the Y dimension of a single cell for the specified EM document.

PortType: Determines the types of ports used in the extraction. Two options are available:

- **Default** - De-embedded edge ports are used where possible; via ports are used in other instances.
- **Via Ports only** - Only via ports are used in the simulation.

STACKUP: Specifies the name of the STACKUP model to use in establishing the dielectric stackup to be applied to the specified EM structure. Select a name from the drop-down list or type a new name.

Create_Enclosure: Two options are available:

- Yes - The enclosure and stackup are created or modified every time an extraction is performed.

- No - The enclosure and stackup are created if the specified EM document does not exist. If the EM document does exist, the enclosure and stackup are not modified. This option allows you to modify these items manually. This setting is not commonly used since simulator specific settings can be set on the EXTRACT block.

Create_Shapes: Two options are available:

- **Yes** - All layout shapes are ported to the specified EM document every time simulation results are required.
- **No** - Layout shapes are not ported to the specified EM document if the specified EM document already exists. If the EM document does not exist, the layout shapes are not ported, but a blank EM document is created. This option allows you to use the present extraction defined in the EM document even if small changes in layout occur. This significantly speeds up the simulation of the overall schematic.

Extension: Preferred method of specifying the EM document's enclosure dimensions for simulators that require an enclosure size (specifically EMSight). Specifies the distance to extend the enclosure beyond edges of the extraction group. In this mode a rectangular bounding box of the extraction grouping is determined. This box is then extended outward in all dimensions (+X, -X, +Y, -Y) by the specified extension distance.

Override_options: Determines whether the EXTRACT block options overwrite those set directly on the EM structure. EXTRACT blocks in older designs do not overwrite by default, to preserve the previous behavior and results.

Hierarchy: Two options are available:

- Off - When set to Off for any EXTRACT block at a lower level than the level being simulated, any shapes in that block are only extracted using a top level EXTRACT block.
- On - When set to On for any EXTRACT block at a lower level than the level being simulated, any shapes in that block are extracted in the lower level document and the top EXTRACT block includes any shapes not controlled by a lower level block.

Extract_LineTypes: Two options are available:

- No - Any shapes using a line type in the schematic layout have those individual shapes available for editing in the EM document.
- Yes - Any shapes using a line type in the schematic layout retain the line type setting (so all shapes move as one object).

Union_Shapes: Two options are available:

- No - No Boolean operations are performed on the shapes before movement to the EM document.
- Yes - Shapes follow the positive, negative, normal layer rules for unioning shapes together. See “Negative Layers ” for more information.

Extract_PinShapes: Two options are available:

- No - No area pins for layout cells other than ports have areas used in the EM document if an item connecting to that pin is extracted.
- Yes - All area pins for layout cells have areas used in the EM document if an item connecting to that pin is extracted.

Explicit_Ground: Three options are available:

- None - AXIEM does not change the ground properties of each edge port.
- Connect to lower - Each port sets its ground setting to connect to lower.

- Connect to upper - Each port sets its ground setting to connect to upper.
- Connect to both - Each port sets its ground setting to connect to both.

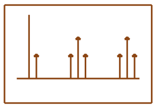
SweepVar_Names: The names of any swept variables used on elements that are being extracted. Without this setting, extraction uses the default value of the swept variable when running the EM simulations. The swept variable must be enclosed in quotes. For example, if the swept variable is named "l", you add the parameter as "l". If there are multiple swept variables, you can enter multiple variable names separated by commas. For example, if variables a, b, and c are used for sweeps in the design, you can type **{"a","b","c"}** (including the curly brackets and quotes) to include all of those sweeps in the extraction. See “Extraction and Swept Variables” for more information.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

Large-Signal Small-Signal Analysis Control Block: LSSS

Symbol



Summary

LSSS instructs the APLAC simulator to run N-tone HB analysis so that one of the HB tones is considered small. The (N-1)-tone HB analysis defines the time varying operating point for the linear analysis.

Parameters

Name	Description	Unit Type	Default
ID	LSSS control name		LSSS1
SSTone	Tone-index of small signal excitation		2

** indicates a secondary parameter*

Parameter Details

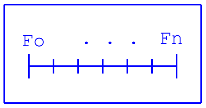
SSTone. Defines which of the large signal tones is treated as small signal. The original N-tone HB analysis is run internally as (N-1) tone HB analysis + small signal analysis. Special elements that refer to the symbolic analysis frequencies, such as PHD-models, are treated so that their symbolic frequencies refer to the HB analysis nonlinear tones. If the schematic has 3 tones [f1,f2,f3] and SSTone=2, the 2-tone HB analysis is done with tone frequencies [f1,f3]. These two tones then define the large signal tones, for example, for a PHD-model with 2-tone data.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Defines Frequency Set to Support Multirate Harmonic Balance (APLAC Only): MRHB

Symbol



Summary

Multi-Rate Harmonic Balance (MRHB) is a way of simplifying harmonic balance simulations, typically where there is frequency conversion stages in the simulation. Each part of a circuit can be told which frequencies to simulate (and conversely which frequencies to ignore). The MRHB control block is a part of setting up different models to simulate different frequencies.

See "Multi-rate (Nonlinear) Harmonic Balance Analysis" for details on how to use multi-rate harmonic balance.

Parameters

Name	Description	Unit Type	Default
ID	Element ID		HB1
NTONES	Number of independent tones	Integer	1[a]
TRUNC	Truncation method		Diamond[b]

[a]New parameters display for each integer from 1 to N where N is the number entered.
[b]Different parameters display based on the truncation method chosen.

** indicates a secondary parameter*

Dynamic Parameters

Name	Description	Unit Type	Default
ID	Element ID		HB1
MAXHARMDIA	Max harmonic for diamond truncation	Integer	8
TONESPEC<num>	Tone specification #<num>		"f1"
MAXHARM<num>	Maximum number of harmonics for tone #<num>	Integer	8

Parameter Details

NTONES. Set to the number of tones for this block. New parameters for specifying the TONESPEC and truncation methods.

Truncation method. Set to "Diamond", "Box" or "Box and Diamond". When set to "Diamond", the MAXHARMDIA parameter is available for each tone specified. When set to "Box", the MAXHARM<num> parameter is available for each tone specified. When set to "Box and Diamond", both of these parameters are available for each tone specified.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Script Block for APLAC Simulations: SCRIPT

Symbol

Summary

The Script block is used to define additional simulation parameters for the APLAC simulation engines. See "Using Scripted APLAC" for details on using the script block.

Parameters

Name	Description	Unit Type	Default
ID	Element ID		SC1
POSITION	Position of the script in the netlist		After Analysis
ORDER	Netlisting order		1
SIM	Allow script to spawn a simulation		Yes

** indicates a secondary parameter*

Parameter Details

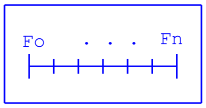
POSITION Determines where in the APLAC netlist the scripted content is added. The available values are "Before Elements", "After Simulation", and "Before Simulation".

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Swept Frequency Control: SWPFRQ

Symbol



Summary

The SWPFRQ simulation control is used in schematics to indicate a frequency sweep that can be used during a swept frequency analysis in addition to, or as an alternate to, the document and project frequencies. SWPFRQ sorts and removes duplicates from its frequency list before passing it to the simulator (to facilitate graph drawing).

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	FSWP1
Values	Vector of frequency values (in Hz)	Vector	{0}

Parameter Details

Values. Vector of frequency values (in Hertz) to use during a swept frequency analysis. Please see the "Swept Parameter Analysis" chapter or the discussion on vector notation in “Swept Parameter Analysis ” for information on the variety of methods to define a vector.

Layout

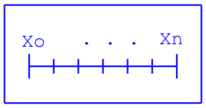
This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

Recommendations for Use

See for detailed information regarding swept parameter analysis.

Swept Variable Control: SWPVAR

Symbol



Summary

The SWPVAR simulation control is used in schematics to indicate the variable and the values to use during a swept parameter analysis. The values must be real.

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	SWP1
VarName	Name of variable to sweep	String	""
Values	Vector of values	Vector (real)	{0}
*Unit Type	Unit type (used for plotting only, and when sweep values are in base units)	Enumeration (pull-down)	

** indicates a secondary parameter*

Parameter Details

VarName. Name of the variable to sweep. The variable must be defined in the schematic.

Values. Vector of values used for sweeping the variable. See the discussion on vector notation in "Swept Parameter Analysis " for information on the variety of methods to define a vector.

The values are assumed to be in base units.

Unit Type. The units are applied to the swept variable for plotting purposes only. Units should only be used when the swept variable is specified in base units.

Layout

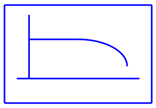
This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Recommendations for Use

See "Swept Parameter Analysis " for detailed information regarding swept parameter analysis, and "Built-in Functions " for a table of swept functions.

X dB Compression Point Control Block: XDB

Symbol



Summary

XDB allows you to set up a schematic for measuring, or making other measurements at an X dB compression point, where X is user-specified. This element only functions with the APLAC Harmonic Balance simulator.

Parameters

Name	Description	Unit Type	Default
ID	Element ID		PO1
IN	ID of the source element		PORT_P1
OUT	ID of the power measurement element		PORT_P2
XDB	Amount of dB into compression	dB	1
GAIN_TYPE	Method used to compute gain		Linear
FUNC_IN	Source input power function, e.g. P("f1")+P("f2")		P("f1")
FUNC_OUT	Target output power function, e.g. P("f1")+P("f2")		P("f1")
ERR	Maximum allowed error in output power		0.01
*MAX_ITER	Maximum number of iterations		100
*MIN	Lower limit for the source value (relative to the normal value)		0
*MAX	Upper limit for the source value (relative to the normal value)		10000
RESET	Reset source nominal value at each sweep point		No
*OPT_METHOD	Optimization method		Random
GAIN_BACKOFF	Gain backoff (in dB) for small signal gain calculation	dB	-60

** indicates a secondary parameter*

Parameter Details

IN: Specifies the ID of the input (source) element(s), the amplitude of which is modified. It is limited to sources and port elements. You can specify multiple sources by using a comma separated list (for example, {Port_P1, Port_P2}).

OUT: Specifies the ID of the output measurement element. The output measurement element can be a Port or P_METER3 element.

XDB: Specifies the amount of dB into compression. For instance, if you set it to 3, it sets up the schematic for measuring a 3 dB compression point.

GAIN_TYPE: Specifies the type of gain.

- **Linear**: Computes the compression point from the linear gain.

- **MaxGain**: Computes the compression point from the maximum (for example, the gain expansion region).

MIN: Specifies the minimum scale factor applied to the source. The default value is 0; the same as -800 dBm, 0 W or 0 V based on the source.

MAX: Specifies the maximum scale factor applied to the source. The default value is 10 which is 10x the nominal voltage or 10x the nominal power based on the source.

OPT_METHOD: Sets up the optimization method for APLAC to find the compression point. This setting is unrelated to the optimization methods discussed in "Optimization Methods".

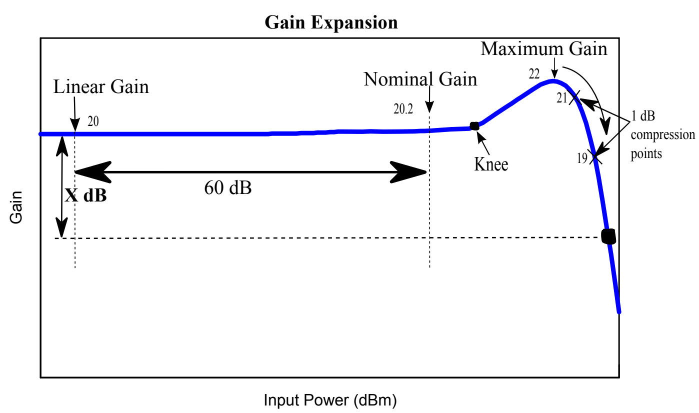
GAIN_BACKOFF: Sets the power level of the source relative to the nominal value for computing the linear gain of the circuit.

Implementation Details

The following figure demonstrates how the XDB element functions. The nominal gain is the gain at the input power level specified on the schematic and should be below the knee of the gain expansion region as shown in the following figure.

The XDB element finds the linear region by reducing the nominal input power by 60 dB, then it finds the maximum gain by raising the input power from the nominal input power. If there is no gain expansion, the nominal gain is used for the maximum gain. The desired compression point is the point X dB below the linear gain if "Linear" is the GAIN_TYPE, and X dB below the maximum gain if "MaxGain" is the GAIN_TYPE. The XDB element always finds the compression point in the compression area (it only looks at input power levels above the input power that produces maximum gain.)

For example, if the nominal gain is 20.2 dB, linear gain is 20 dB, and maximum gain is 22 dB, the 1 dB compression point can have two different answers depending on what you specify. If you select "Linear", the 1 dB compression point is at 20 - 1 = 19 dB, but if you select "MaxGain", the 1 dB compression point is at 22 - 1 = 21 dB. All of these points are shown in the following figure.

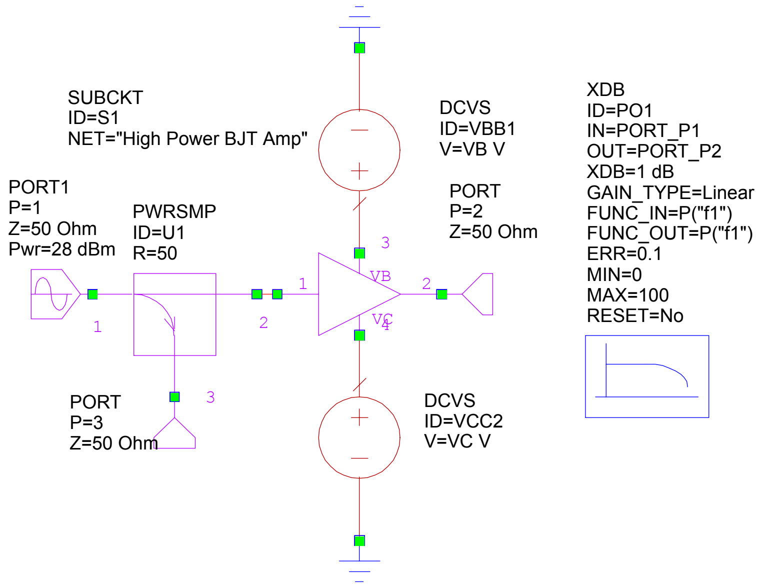


Layout

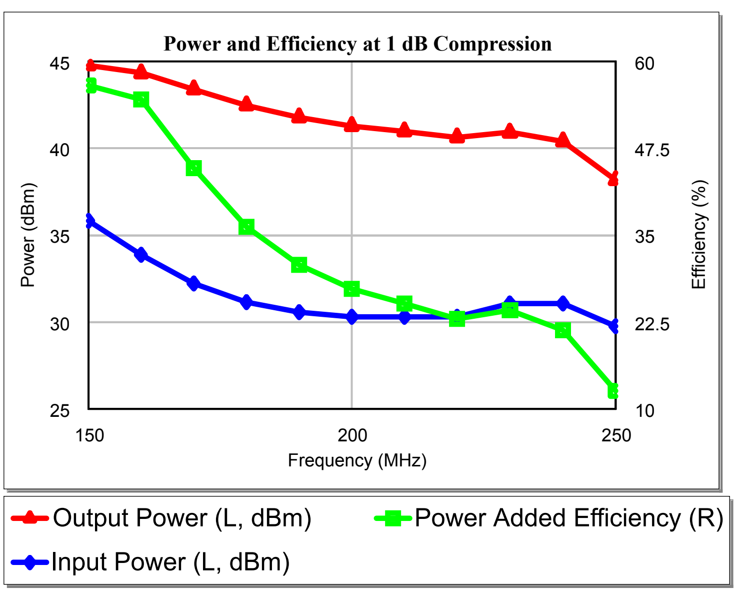
This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Examples

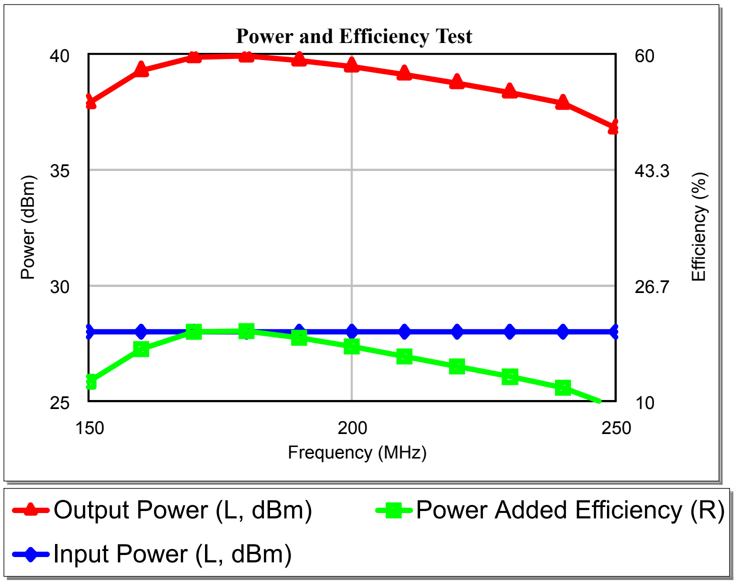
The following figure shows an example of how to use the XDB element in a schematic. In this example, the XDB element is set up for 1 dB compression from the linear gain. The PWRSMP element is connected in the schematic to measure the incident input power.



The following figure shows the power and efficiency for the previous schematic. Because the XDB element is present in the schematic, the power added efficiency, input power, and output power are all calculated at the 1 dB gain compression point. This is why the input power changes vs. frequency.



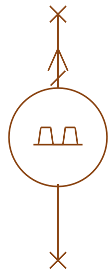
If the same schematic is simulated without the XDB element, it results in the following figure. In this case, the input power is constant, similarly, the output power and PAE are computed at the user-specified input power.



Sources

Dynamic AC Current Source: AC_I

Symbol



Summary

AC_I is a versatile AC current source that encapsulates the functionality of many AC current sources into a single AC current source. A variety of signal types may be generated with this source, including sinusoid, piecewise linear, pulse, saw, square, and triangle signals. In addition,you can specify what to use for frequency, how to calculate bandwidth/number of harmonics, the tone number, and how to sweep voltage.

This source combines the functionality of the following current sources: ACCS, ACCS2, ACCSN, ACCSNS, ACCSS, I_PLS, I_PWL, I_SAW, I_SQR, and I_TRI.

Parameters

Name	Description	Unit Type	Default
ID	Current source ID		I1
Signal	Signal type		Sinusoid
SpecType	Specify type		Use doc freqs
SpecBW	Specify bandwidth		Use doc # harms
Sweep	Current sweep type (only used when Signal=Sinusoid)		
Tone	Tone number for the signal		1
DCVal	DC value (used for DC analysis)	Current	0 A
*Hint	Hint		Auto
*NSamp	Over sample factor		1

** indicates a secondary parameter*

Parameter Details

Signal. Specifies the signal type as "Sinusoid", "PWL", "Pulse", "Saw", "Square" or "Triangle".

SpecType. Specifies whether to use the document frequencies or specify a frequency or time period. Choices are "Use doc freqs", "Specify freq", "Specify period", "Trans only (Freq)", or "Trans only (Period)". The Trans only settings can only be used with transient simulators.

SpecBW. Specifies whether to use the number of harmonics in the document settings or to specify the number of harmonics or minimum time resolution. Choices are "Use doc # harms", "Specify # harms" and "Min time res".

Sweep. Specifies whether power is swept. Choices are "None", "Vector", "Linear", "Linear (# pts)", "Log (Dec)" and "Log (Oct)".

Tone. Tone number for the signal.

DCVal. DC current value used for DC analysis.

Hint. If the type of signal is known (large signal or small signal), the solvers can take advantage of this hint to help solve the problem in the most efficient way. Choices are "Auto", "Large signal" and "Small signal".

NSamp. Oversample factor. This sets the oversampling factor at the specified tone. Specify a value of zero to indicate that this parameter does not override the value specified for the **Tone X Over Sample** in the project or document harmonic balance settings. (Choose **Options > Default Circuit Options** and click the **AWR Sim** tab.) This only applies when the Tone parameter is set to 1, 2, or 3 for the element. An error is issued if this parameter is set to zero for a source with Tone equal to 4 or more.

Dynamic Parameters

Name	Description	Unit Type	Default
Mag	AC current magnitude	Current	1000 mA
Ang	AC current angle	Angle	0 Deg
Freq	Frequency for tone N	Frequency	1 GHz
Period	Period	Time	1 ns
NHarm	Maximum number of harmonics for this tone		5
MinRes	Minimum time resolution	Time	1 ns
IStart	AC current start	Current	0 mA
IStop	AC current stop	Current	5000 mA
IStep	AC current step	Current	1000 mA
NumPts	Number of swept current points		2
Offset	Waveform offset (does not affect DC)	Current	0 mA
WINDOW	Window type		
TIME	Time vector	Time	{0,0.5,1} ns
CURRENT	Current vector	Current	{1000,1000,1000} mA
HI	High current level	Current	500 mA
LO	Low current level	Current	-500 mA
TD	Time delay	Time	0 ns
TW	Pulse width	Time	0 ns
TR	Rise time	Time	0 ns
TF	Fall time	Time	0 ns

Dynamic parameters are parameters whose existence depends on the settings of the static parameters. The following is a list all of possible parameters that may display for the AC_I depending on the settings of the static parameters Signal, SpecType, SpecBW and Sweep.

Dynamic Parameter Details

Mag. Displays when Signal is "Sinusoid" and Sweep is "None" or "Vector". If Sweep is "None", only a single current value is specified. If Sweep is "Vector", each value represents a value in the current sweep.

Ang. Displays when Signal is "Sinusoid". Represents the relative angle of excitation. If Sweep is "None", only a single angle value is specified. If Sweep is "Vector", each value represents the angles of each value in the current sweep.

Freq. Displays when SpecType is "Specify freq". Allows you to specify a single source frequency directly on the port instead of having to use the document or project frequencies.

Period. Displays when SpecType is "Specify period". Allows you to specify the time period of the source signal directly on the port instead of having to use the document or project frequencies.

NHarm. Displays when SpecBW is "Specify # harms". Allows you to specify the number of harmonics to use directly on the port instead of having to use the document or project setting. Specify a value of zero to indicate that this parameter does not override the value specified for the **Tone X Harmonics** in the project or document harmonic balance settings. (Choose **Options > Default Circuit Options** and click the **AWR Sim** tab.) This only applies when the Tone parameter is set to 1, 2, or 3 for the element. An error is issued if this parameter is set to zero for a source with Tone equal to 4 or more.

MinRes. Displays when SpecBW is "Min time res". Allows you to specify the time resolution directly on the port instead of having to use the document or project **Tone X Harmonics** setting.

IStart, IStop. Displays when Signal is "Sinusoid" and Sweep is "Linear", "Linear (# pts)", "Log (Dec)" or "Log (Oct)". Allows you to specify the start and stop current values in a current sweep.

IStep. Displays when Signal is "Sinusoid" and Sweep is "Linear". Allows you to specify the current step value of a current sweep.

NumPts. Displays when Signal is "Sinusoid" and Sweep is "Linear (# pts)", "Log (Dec)", or "Log (Oct)". Allows you to specify the number of points in a current sweep. When Sweep is "Log (Dec)" and "Log (Oct)", this represents the number of points per decade and octave, respectively.

Offset. Displays when Signal is "Sinusoid". Allows you to specify a DC current offset for the sinusoid that DOES NOT affect DC analysis.

WINDOW. Displays when Signal is set to anything except "Sinusoid". Allows you to specify a data window type that is applied to the source signal. Choices include "NONE", "DEFAULT", "TRIANG", "HANN", "HAMM", and "BLACK".

TIME and **CURRENT.** Displays when Signal is "PWL". See I_PWL for details on these parameters.

HI and **LO.** Displays when Signal is "Pulse", "Saw", "Square", or "Triangle". Represents the high and low current values of the signal.

TD. Displays when Signal is "Pulse", "Saw", "Square", or "Triangle". Represents the time delay of the signal.

TW. Displays when Signal is "Pulse". Represents the pulse width in time units.

TR. Displays when Signal is "Pulse" or "Square". Represents the rise time.

TF. Displays when Signal is "Pulse", "Saw", or "Square". Represents the fall time.

NOTES: AWR simulators add some series resistance to this element during simulation. This may affect results if injecting current into a large resistance. To change the value of the resistance, choose **Options > Default Circuit Options** to display the Circuit Options dialog box, then click the **AWR Sim** tab and under **Convergence Aids** specify the **Series source resistance**.

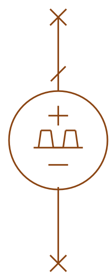
If a sinusoidal nonlinear source like this one has the parameter Ang=0 and is ideally terminated, then a nonlinear measurement (for example, Pcomp or Vcomp) made at the fundamental output of that source has an angle of -90deg. This discrepancy is consistent with the definition of the sine wave sources (as in SPICE), and the Fourier-based harmonic component measurements. To avoid confusion, always measure or calculate the gain, and plot its angle.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Dynamic AC Voltage Source: AC_V

Symbol



Summary

AC_V is a versatile AC voltage source that encapsulates the functionality of many AC voltage sources into a single AC voltage source. A variety of signal types may be generated with this source, including sinusoid, piecewise linear, pulse, saw, square and triangle signals. In addition, you can specify what to use for frequency, how to calculate bandwidth/number of harmonics, the tone number, and how to sweep voltage.

This source combines the functionality of the following voltage sources: ACVS, ACVS2, ACVSN, ACVSNS, ACVSS, V_PLS, V_PWL, V_SAW, V_SQR and V_TRI.

Parameters

Name	Description	Unit Type	Default
ID	Voltage source ID	Text	V1
Signal	Signal type		Sinusoid
SpecType	Specify type		Use doc freqs
SpecBW	Specify bandwidth		Use doc # harms
Sweep	Voltage sweep type (only used when Signal=Sinusoid)		
Tone	Tone number for the signal		1
DCVal	DC value (used for DC analysis)	Voltage	0 V
*Hint	Hint		Auto
*NSamp	Over sample factor		1

** indicates a secondary parameter*

Parameter Details

Signal. Specifies the signal type as "Sinusoid", "PWL", "Pulse", "Saw", "Square" or "Triangle".

SpecType. Specifies whether to use the document frequencies or specify a frequency or time period. Choices are "Use doc freqs", "Specify freq", "Specify period", "Trans only (Freq)", or "Trans only (Period)". The Trans only settings can only be used with transient simulators.

SpecBW. Specifies whether to use the number of harmonics in the document settings or to specify the number of harmonics or minimum time resolution. Choices are "Use doc # harms", "Specify # harms" and "Min time res".

Sweep. Specifies whether power is swept. Choices are "None", "Vector", "Linear", "Linear (# pts)", "Log (Dec)" and "Log (Oct)".

Tone. Tone number for the signal.

DCVal. DC voltage value used for DC analysis.

Hint. If the type of signal is known (large signal or small signal), the solvers can take advantage of this hint to help solve the problem in the most efficient way. Choices are "Auto", "Large signal" and "Small signal".

NSamp. Oversample factor. This sets the oversampling factor at the specified tone. Specify a value of zero to indicate that this parameter does not override the value specified for the **Tone X Over Sample** in the project or document harmonic balance settings. (Choose **Options > Default Circuit Options** and click the **AWR Sim** tab.) This only applies when the Tone parameter is set to 1, 2, or 3 for the element. An error is issued if this parameter is set to zero for a source with Tone equal to 4 or more.

Dynamic Parameters

Name	Description	Unit Type	Default
Mag	AC voltage magnitude	Voltage	1 V
Ang	AC voltage angle	Angle	0 Deg
Freq	Frequency for tone N	Frequency	1 GHz
Period	Period	Time	1 ns
NHarm	Maximum number of harmonics for this tone		5
MinRes	Minimum time resolution	Time	1 ns
VStart	AC voltage start	Voltage	0 V
VStop	AC voltage stop	Voltage	5 V
VStep	AC voltage step	Voltage	1 V
NumPts	Number of swept voltage points		2
Offset	Waveform offset (does not affect DC)	Voltage	0 V
WINDOW	Window type		
TIME	Time vector	Time	{0,0.5,1} ns
VOLTAGE	Voltage vector	Voltage	{1,1,1} V
HI	High voltage level	Voltage	0.5 V
LO	Low voltage level	Voltage	-0.5 V
TD	Time delay	Time	0 ns
TW	Pulse width	Time	0 ns
TR	Rise time	Time	0 ns
TF	Fall time	Time	0 ns

Dynamic parameters are parameters whose existence depends on the settings of the static parameters. The following is a list all of possible parameters that may display for the AC_V depending on the settings of the static parameters Signal, SpecType, SpecBW and Sweep.

Dynamic Parameter Details

Mag. Displays when Signal is "Sinusoid" and Sweep is "None" or "Vector". If Sweep is "None", only a single current value is specified. If Sweep is "Vector", each value represents a value in the current sweep.

Ang. Displays when Signal is "Sinusoid". Represents the relative angle of excitation. If Sweep is "None", only a single angle value is specified. If Sweep is "Vector", each value represents the angles of each value in the current sweep.

Freq. Displays when SpecType is "Specify freq". Allows you to specify a single source frequency directly on the port instead of having to use the document or project frequencies.

Period. Displays when SpecType is "Specify period". Allows you to specify the time period of the source signal directly on the port instead of having to use the document or project frequencies.

NHarm. Displays when SpecBW is "Specify # harms". Allows you to specify the number of harmonics to use directly on the port instead of having to use the document or project setting. Specify a value of zero to indicate that this parameter does not override the value specified for the **Tone X Harmonics** in the project or document harmonic balance settings. (Choose **Options > Default Circuit Options** and click the **AWR Sim** tab.) This only applies when the Tone parameter is set to 1, 2, or 3 for the element. An error is issued if this parameter is set to zero for a source with Tone equal to 4 or more.

MinRes. Displays when SpecBW is "Min time res". Allows you to specify the time resolution directly on the port instead of having to use the document or project **Tone X Harmonics** setting.

IStart, IStop. Displays when Signal is "Sinusoid" and Sweep is "Linear", "Linear (# pts)", "Log (Dec)" or "Log (Oct)". Allows you to specify the start and stop current values in a current sweep.

IStep. Displays when Signal is "Sinusoid" and Sweep is "Linear". Allows you to specify the current step value of a current sweep.

NumPts. Displays when Signal is "Sinusoid" and Sweep is "Linear (# pts)", "Log (Dec)", or "Log (Oct)". Allows you to specify the number of points in a current sweep. When Sweep is "Log (Dec)" and "Log (Oct)", this represents the number of points per decade and octave, respectively.

Offset. Displays when Signal is "Sinusoid". Allows you to specify a DC current offset for the sinusoid that DOES NOT affect DC analysis.

WINDOW. Displays when Signal is set to anything except "Sinusoid". Allows you to specify a data window type that is applied to the source signal. Choices include **NONE**, **DEFAULT**, **TRIANG**, **HANN**, **HAMM**, and **BLACK**.

TIME and **CURRENT.** Displays when Signal is "PWL". See I_PWL for details on these parameters.

HI and **LO.** Displays when Signal is "Pulse", "Saw", "Square", or "Triangle". Represents the high and low current values of the signal.

TD. Displays when Signal is "Pulse", "Saw", "Square", or "Triangle". Represents the time delay of the signal.

TW. Displays when Signal is "Pulse". Represents the pulse width in time units.

TR. Displays when Signal is "Pulse" or "Square". Represents the rise time.

TF. Displays when Signal is "Pulse", "Saw", or "Square". Represents the fall time.

NOTE: If a sinusoidal nonlinear source like this one has the parameter Ang=0 and is ideally terminated, then a nonlinear measurement (for example, Pcomp or Vcomp) made at the fundamental output of that source will have an angle of

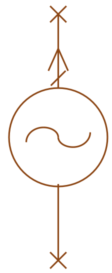
-90deg. This discrepancy is consistent with the definition of the sine wave sources (as in SPICE), and the Fourier-based harmonic component measurements. To avoid confusion, always measure or calculate the gain, and plot its angle.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

AC Current Source: ACCS

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Current source ID	Text	V1
Mag	AC current magnitude	Current	1 mA
Ang	AC current angle	Angle	0 Deg
Offset	Waveform offset (does not affect DC)	Current	0
DCVal	DC Value (Used for DC analysis)	Current	0

Implementation Details

Produces an alternating current with a frequency defined by the project frequency set up for the schematic containing this element.

NOTE: AWR simulators add some series resistance to this element during simulation. This may affect results if injecting current into a large resistance. To change the value of the resistance, choose **Options > Default Circuit Options** to display the Circuit Options dialog box, then click the **AWR Sim** tab and under **Convergence Aids** specify the **Series source resistance**.

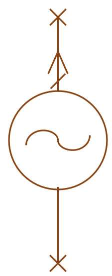
NOTE: If a sinusoidal nonlinear source like this one has the parameter Ang=0 and is ideally terminated, then a nonlinear measurement (for example, Pcomp or Vcomp) made at the fundamental output of that source will have an angle of -90deg. This discrepancy is consistent with the definition of the sine wave sources (as in SPICE), and the Fourier based harmonic component measurements. To avoid confusion, always measure or calculate the gain, and plot its angle.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

AC Current Source (Tone 2): ACCS2

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Current source ID	Text	V1
Mag	AC current magnitude	Current	10 mA
Ang	AC current angle	Angle	0 Deg
F	AC frequency	Frequency	1 GHz
Offset	Waveform offset (does not affect DC)	Current	0
DCVal	DC Value (Used for DC analysis)	Current	0

Implementation Details

Produces a tone two alternating current with the frequency defined by F. The presence of one of these sources in the schematic automatically causes a two-tone analysis to be performed.

NOTE: AWR simulators add some series resistance to this element during simulation. This may affect results if injecting current into a large resistance. To change the value of the resistance, choose **Options > Default Circuit Options** to display the Circuit Options dialog box, then click the **AWR Sim** tab and under **Convergence Aids** specify the **Series source resistance**.

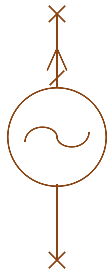
NOTE: If a sinusoidal nonlinear source like this one has the parameter Ang=0 and is ideally terminated, then a nonlinear measurement (e.g. Pcomp or Vcomp) made at the fundamental output of that source will have an angle of -90deg. This discrepancy is consistent with the definition of the sine wave sources (as in SPICE), and the Fourier based harmonic component measurements. To avoid confusion, always measure or calculate the gain, and plot its angle.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

AC Current Source (tone N): ACCSN

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Current source ID	Text	V1
Mag	AC current magnitude	Current	10 mA
Ang	AC current angle	Angle	0 Deg
F	AC frequency	Frequency	1 GHz
Tone	Tone number for the signal		2
Offset	Waveform offset (does not affect DC)	Current	0
DCVal	DC Value (Used for DC analysis)	Current	0
*NHarm	Maximum number of harmonics for this tone		0
*NSamp	Sampling rate for this tone		2

Parameter Details

NHarm & NSamp. Specify a value of zero to indicate that these parameters do not override the values specified for the **Tone X Harmonics** and **Tone X Over sample** in the project or document harmonic balance settings. (Choose **Options > Default Circuit Options** and click the **AWR Sim** tab.) This only applies when the Tone parameter is set to 1, 2, or 3 for the element. An error is issued if these parameters are set to zero for a source with Tone equal to 4 or more.

Implementation Details

This is an ideal current source similar to ACCS, but is more general in that it allows you to independently specify the tone number and the fundamental frequency of that tone.

NOTES: AWR simulators add some series resistance to this element during simulation. This may affect results if injecting current into a large resistance. To change the value of the resistance, choose **Options > Default Circuit Options** to display the Circuit Options dialog box, then click the **AWR Sim** tab and under **Convergence Aids** specify the **Series source resistance**.

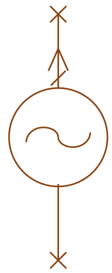
If a sinusoidal nonlinear source like this one has the parameter Ang=0 and is ideally terminated, then a nonlinear measurement (for example, Pcomp or Vcomp) made at the fundamental output of that source will have an angle of -90deg. This discrepancy is consistent with the definition of the sine wave sources (as in SPICE), and the Fourier based harmonic component measurements. To avoid confusion, always measure or calculate the gain, and plot its angle.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Swept AC Current Source (tone N): ACCSNS

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Current source ID	Text	V1
IStart	AC current magnitude start	Current	0 mA
IStop	AC current magnitude stop	Current	50 mA
IStep	AC current magnitude step	Current	1 mA
Ang	AC current angle	Angle	0 Deg
F	AC frequency	Frequency	1 GHz
Tone	Tone number for the signal		2
Offset	Waveform offset (does not affect DC)	Current	0
DCVal	DC Value (Used for DC analysis)	Current	0
*NHarm	Maximum number of harmonics for this tone		0
*NSamp	Sampling rate for this tone		2

** indicates a secondary parameter*

Parameter Details

NHarm & NSamp. Specify a value of zero to indicate that these parameters do not override the values specified for the **Tone X Harmonics** and **Tone X Over sample** in the project or document harmonic balance settings. (Choose **Options > Default Circuit Options** and click the **AWR Sim** tab.) This only applies when the Tone parameter is set to 1, 2, or 3 for the element. An error is issued if these parameters are set to zero for a source with Tone equal to 4 or more.

Implementation Details

This element is identical to ACCSN except that it allows you to specify a current magnitude sweep.

NOTES: AWR simulators add some series resistance to this element during simulation. This may affect results if injecting current into a large resistance. To change the value of the resistance, choose **Options > Default Circuit Options** to display the Circuit Options dialog box, then click the **AWR Sim** tab and under **Convergence Aids** specify the **Series source resistance**.

If a sinusoidal nonlinear source like this one has the parameter Ang=0 and is ideally terminated, then a nonlinear measurement (for example, Pcomp or Vcomp) made at the fundamental output of that source will have an angle of

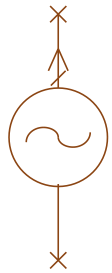
-90deg. This discrepancy is consistent with the definition of the sine wave sources (as in SPICE), and the Fourier based harmonic component measurements. To avoid confusion, always measure or calculate the gain, and plot its angle.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Swept AC Current Source: ACCSS

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Current source ID	Text	V1
IStart	AC current magnitude start	Current	0 mA
IStop	AC current magnitude stop	Current	50 mA
IStep	AC current magnitude step	Current	1 mA
Ang	AC current angle	Angle	0 Deg
Offset	Waveform offset (does not affect DC)	Current	0
DCVal	DC value (Used for DC analysis)	Current	0

Implementation Details

This element is identical to ACCS except that it allows you to specify a current magnitude sweep.

NOTE: AWR simulators add some series resistance to this element during simulation. This may affect results if injecting current into a large resistance. To change the value of the resistance, choose **Options > Default Circuit Options** to display the Circuit Options dialog box, then click the **AWR Sim** tab and under **Convergence Aids** specify the **Series source resistance**.

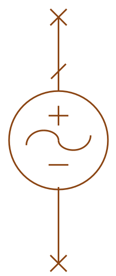
NOTE: If a sinusoidal nonlinear source like this one has the parameter Ang=0 and is ideally terminated, then a nonlinear measurement (e.g. Pcomp or Vcomp) made at the fundamental output of that source will have an angle of -90deg. This discrepancy is consistent with the definition of the sine wave sources (as in SPICE), and the Fourier based harmonic component measurements. To avoid confusion, always measure or calculate the gain, and plot its angle.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

AC Voltage Source: ACVS

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Voltage source ID	Text	V1
Mag	AC voltage magnitude	Voltage	1 V
Ang	AC voltage angle	Angle	0 Deg
Offset	Waveform offset (does not affect DC)	Voltage	0
DCVal	DC value (Used for DC analysis)	Voltage	0

Implementation Details

Produces an alternating voltage with a frequency defined by the project frequency set up for the schematic containing this element.

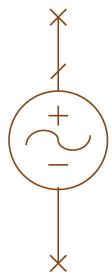
NOTE: If a sinusoidal nonlinear source like this one has the parameter Ang=0 and is ideally terminated, then a nonlinear measurement (e.g. Pcomp or Vcomp) made at the fundamental output of that source will have an angle of -90deg. This discrepancy is consistent with the definition of the sine wave sources (as in SPICE), and the Fourier based harmonic component measurements. To avoid confusion, always measure or calculate the gain, and plot its angle.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

AC Voltage Source (Tone 2): ACVS2

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Voltage source ID	Text	V1
Mag	AC voltage magnitude	Voltage	1 V
Ang	AC voltage angle	Angle	0 Deg
F	AC frequency	Frequency	1 GHz
Offset	Waveform offset (does not affect DC)	Voltage	0
DCVal	DC value (Used for DC analysis)	Voltage	0

Implementation Details

Produces a tone two alternating voltage with the frequency defined by F. The presence of one of these sources in the schematic automatically causes a two-tone analysis to be performed.

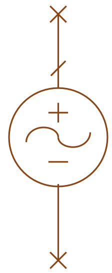
NOTE: If a sinusoidal nonlinear source like this one has the parameter Ang=0 and is ideally terminated, then a nonlinear measurement (for example, Pcomp or Vcomp) made at the fundamental output of that source will have an angle of -90deg. This discrepancy is consistent with the definition of the sine wave sources (as in SPICE), and the Fourier based harmonic component measurements. To avoid confusion, always measure or calculate the gain, and plot its angle.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

AC Voltage Source (tone N): ACVSN

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Voltage source ID	Text	V1
Mag	AC voltage magnitude	Voltage	1 V
Ang	AC voltage angle	Angle	0 Deg
F	AC frequency of the signal	Frequency	1 GHz
Tone	Tone number for the signal		2
Offset	Waveform offset (does not affect DC)	Voltage	0
DCVal	DC value (Used for DC analysis)	Voltage	0
*NHarm	Maximum number of harmonics for this tone		0
*NSamp	Sampling rate for this tone		2

** indicates a secondary parameter*

Parameter Details

NHarm & NSamp. Specify a value of zero to indicate that these parameters do not override the values specified for the **Tone X Harmonics** and **Tone X Over sample** in the project or document harmonic balance settings. (Choose **Options > Default Circuit Options** and click the **AWR Sim** tab.) This only applies when the Tone parameter is set to 1, 2, or 3 for the element. An error is issued if these parameters are set to zero for a source with Tone equal to 4 or more.

Implementation Details

This element is similar to ACVS, but is more general in that it allows you to independently specify the tone number and the fundamental frequency of that tone.

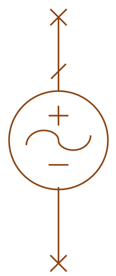
NOTE: If a sinusoidal nonlinear source like this one has the parameter Ang=0 and is ideally terminated, then a nonlinear measurement (for example, Pcomp or Vcomp) made at the fundamental output of that source will have an angle of -90deg. This discrepancy is consistent with the definition of the sine wave sources (as in SPICE), and the Fourier based harmonic component measurements. To avoid confusion, always measure or calculate the gain, and plot its angle.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Swept AC Voltage Source (Tone N): ACVSNS

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Voltage source ID	Text	V1
VStart	AC voltage magnitude start	Voltage	0 V
VStop	AC voltage magnitude stop	Voltage	5 V
VStep	AC voltage magnitude step	Voltage	1 V
Ang	AC voltage angle	Angle	0 Deg
F	AC frequency of the signal	Frequency	1 GHz
Tone	Tone number for the signal		2
Offset	Waveform offset (does not affect DC)	Voltage	0
DCVal	DC value (Used for DC analysis)	Voltage	0
*NHarm	Maximum number of harmonics for this tone		0
*NSamp	Sampling rate for this tone		2

** indicates a secondary parameter*

Parameter Details

NHarm & NSamp. Specify a value of zero to indicate that these parameters do not override the values specified for the **Tone X Harmonics** and **Tone X Over sample** in the project or document harmonic balance settings. (Choose **Options > Default Circuit Options** and click the **AWR Sim** tab.) This only applies when the Tone parameter is set to 1, 2, or 3 for the element. An error is issued if these parameters are set to zero for a source with Tone equal to 4 or more.

Implementation Details

This element is identical to ACVSN, except that it allows you to specify a voltage magnitude sweep.

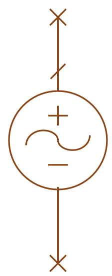
NOTE: If a sinusoidal nonlinear source like this one has the parameter Ang=0 and is ideally terminated, then a nonlinear measurement (for example, Pcomp or Vcomp) made at the fundamental output of that source will have an angle of -90deg. This discrepancy is consistent with the definition of the sine wave sources (as in SPICE), and the Fourier based harmonic component measurements. To avoid confusion, always measure or calculate the gain, and plot its angle.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Swept AC Voltage Source: ACVSS

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Voltage source ID	Text	V1
VStart	DC voltage start	Voltage	0 V
VStop	DC voltage stop	Voltage	5 V
VStep	DC voltage step	Voltage	1 V
Ang	AC voltage angle	Angle	0 Deg
Offset	Waveform offset (does not affect DC)	Voltage	0
DCVal	DC value (Used for DC analysis)	Voltage	0

Implementation Details

This element is identical to ACVS, except that it allow you to specify a voltage magnitude sweep.

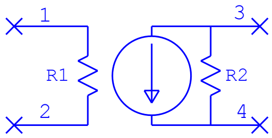
NOTE: If a sinusoidal nonlinear source like this one has the parameter Ang=0 and is ideally terminated, then a nonlinear measurement (e.g. Pcomp or Vcomp) made at the fundamental output of that source will have an angle of -90deg. This discrepancy is consistent with the definition of the sine wave sources (as in SPICE), and the Fourier based harmonic component measurements. To avoid confusion, always measure or calculate the gain, and plot its angle.

Layout

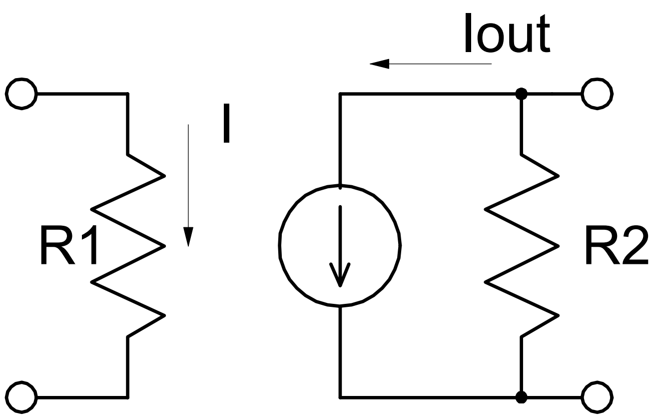
This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Current-Controlled Current Source (Closed Form): CCCS

Symbol



Topology



Parameters

Name	Description	Unit Type	Default
ID	Name	Text	U1
M	DC current gain		1
A	Phase offset	Angle	0 Deg
R1	Input resistance	Resistance	1 ohm
R2	Output resistance	Resistance	0 ohm
F	Break frequency	Frequency	0 GHz
T	Time delay	Time	0 ns

Implementation Details

Implements an ideal current-controlled current source with input resistance R1 across nodes 1 and 2 and output resistance R2 across nodes 3 and 4. The output current is given by

$$I_{\text{out}} = M \cdot I\frac{e^{-j(\omega T + A)}}{1 + j\frac{f}{F}}$$

NOTES:

R2 resistance value of 0 means infinite impedance.

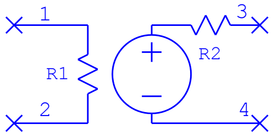
If F = 0, then gain has no frequency dependence.

Layout

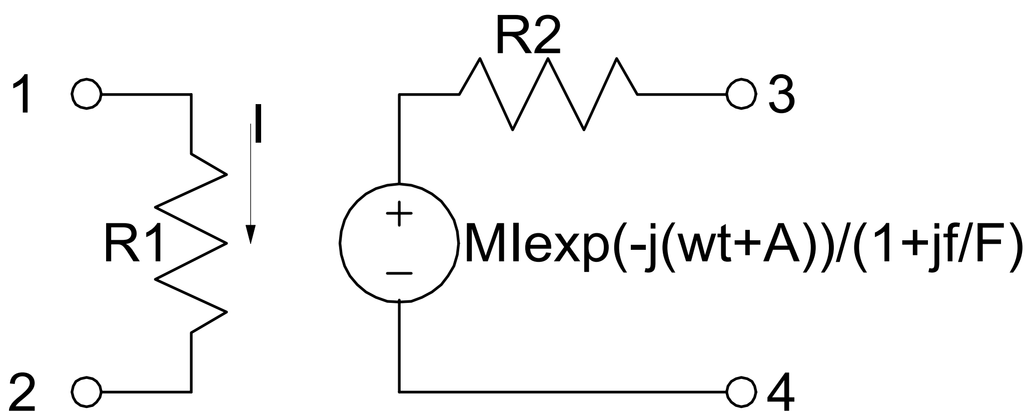
This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Current-Controlled Voltage Source (Closed Form): CCVS

Symbol



Topology



Parameters

Name	Description	Unit Type	Default
ID	Name	Text	U1
M	DC transresistance	Resistance	1 ohm
A	Phase offset	Angle	0 Deg
R1	Input resistance	Resistance	1 ohm
R2	Output resistance	Resistance	1 ohm
F	Break frequency	Frequency	0 GHz
T	Time delay	Time	0 ns

Implementation Details

Implements an ideal current-controlled voltage source with input resistance R1 across nodes 1 and 2 and output resistance R2 across nodes 3 and 4. The output voltage is given by

$$V_{\text{out}} = R_2 \cdot I \frac{e^{-j(\omega T + A)}}{1 + j\frac{f}{F}}$$

NOTES:

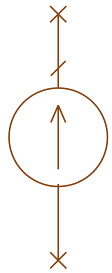
If F = 0, then gain has no frequency dependence.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Dynamic DC Current Source: DC_I

Symbol



Summary

DC_I combines the functionality of the DCCS and DCCSS into one current source.

Parameters

Name	Description	Unit Type	Default
ID	Current source ID	Text	I1
Sweep	Current sweep type		

Parameter Details

Sweep. Specifies whether DC current is swept. Choices are **None**, **Vector**, **Linear**, **Linear (# pts)**, **Log (Dec)** and **Log (Oct)**.

Dynamic Parameters

Name	Description	Unit Type	Default
I	DC current	Current	1000 mA
IStart	DC current start	Current	0 mA
IStop	DC current stop	Current	5000 mA
IStep	DC current step	Current	1000 mA
NumPts	Number of swept current points		2

Dynamic parameters are parameters whose existence depends on the settings of the static parameters. The following is a list of all parameters that can appear for the DC_I depending on the setting of the static Sweep parameter.

Dynamic Parameter Details

I. Appears when **Sweep** is **None** or **Vector**. If **Sweep** is **None**, I represents a single DC current value. If **Sweep** is **Vector**, V represents a vector of swept current values.

IStart, IStop. Appears when **Signal** is **Sinusoid** and **Sweep** is **Linear**, **Linear (# pts)**, **Log (Dec)** or **Log (Oct)**. Allows you to specify the start and stop current values in a current sweep.

IStep. Appears when **Sweep** is **Linear**. Allows you to specify the current step value of a current sweep.

NumPts. Appears when **Sweep** is **Linear (# pts)**, **Log (Dec)** or **Log (Oct)**. Allows you to specify the number of points in a current sweep. For **Sweep** is **Log (Dec)** or **Log (Oct)**, this represents the number of points per decade and octave, respectively.

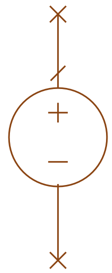
NOTE: AWR simulators add some series resistance to this element during simulation. This may affect results if injecting current into a large resistance. To change the value of the resistance, choose **Options > Default Circuit Options** to display the Circuit Options dialog box, then click the **AWR Sim** tab and under **Convergence Aids** specify the **Series source resistance**.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Dynamic DC Voltage Source: DC_V

Symbol



Summary

DC_V combines the functionality of the DCVS and DCVSS into one voltage source.

Parameters

Name	Description	Unit Type	Default
ID	Voltage source ID	Text	V1
Sweep	Voltage sweep type		

Parameter Details

Sweep. Specifies whether DC current is swept. Choices are **None**, **Vector**, **Linear**, **Linear (# pts)**, **Log (Dec)** and **Log (Oct)**.

Dynamic Parameters

Name	Description	Unit Type	Default
V	DC voltage	Voltage	1 V
VStart	DC voltage start	Voltage	0 V
VStop	DC voltage stop	Voltage	5 V
VStep	DC voltage step	Voltage	1 V
NumPts	Number of swept voltage points		2

Dynamic parameters are parameters whose existence depends on the settings of the static parameters. The following is a list of parameters that can appear for the DC_V depending on the setting of the static Sweep parameter.

Dynamic Parameter Details

V. Appears when **Sweep** is **None** or **Vector**. If **Sweep** is **None**, V represents a single DC voltage value. If **Sweep** is **Vector**, V represents a vector of swept voltage values.

VStart, VStop. Appears when **Signal** is **Sinusoid** and **Sweep** is **Linear**, **Linear (# pts)**, **Log (Dec)** or **Log (Oct)**. Allows you to specify the start and stop voltage values in a voltage sweep.

VStep. Appears when **Sweep** is **Linear**. Allows you to specify the voltage step value of a voltage sweep.

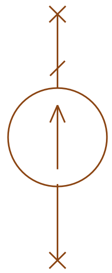
NumPts. Appears when **Sweep** is **Linear (# pts)**, **Log (Dec)** or **Log (Oct)**. Allows you to specify the number of points in a voltage sweep. For **Sweep** is **Log (Dec)** and **Log (Oct)**, this represents the number of points per decade and octave, respectively.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

DC Current Source: DCCS

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Current source ID	Text	I1
I	DC current	Current	10 mA

Implementation Details

Produces a DC current.

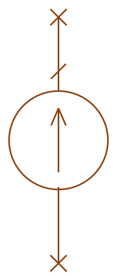
NOTE: AWR simulators add some series resistance to this element during simulation. This may affect results if injecting current into a large resistance. To change the value of the resistance, choose **Options > Default Circuit Options** to display the Circuit Options dialog box, then click the **AWR Sim** tab and under **Convergence Aids** specify the **Series source resistance**.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Swept DC Current Source: DCCSS

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Current source ID	Text	I1
IStart	DC current start	Current	0 mA
IStop	DC current stop	Current	5 mA
IStep	DC current step	Current	1 mA

Implementation Details

This element is identical to DCCS except that a current sweep may be specified.

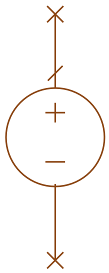
NOTE: AWR simulators add some series resistance to this element during simulation. This may affect results if injecting current into a large resistance. To change the value of the resistance, choose **Options > Default Circuit Options** to display the Circuit Options dialog box, then click the **AWR Sim** tab and under **Convergence Aids** specify the **Series source resistance**.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

DC Voltage Source: DCVS

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Voltage source ID	Text	V1
V	DC voltage	Voltage	1 V

Implementation Details

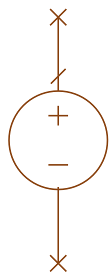
Produces a DC voltage.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

Swept DC voltage source: DCVSS

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Voltage source ID	Text	V1
VStart	DC voltage start	Voltage	0 V
VStop	DC voltage stop	Voltage	5 V
VStep	DC voltage step	Voltage	1 V

Implementation Details

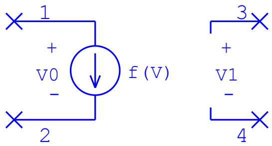
This element is identical to DCVS except that a voltage sweep may be specified.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Dynamic Nonlinear Voltage-Controlled Current/Voltage Source: DYN_VCCS

Symbol



Summary

DYN_VCCS is a nonlinear voltage-controlled current or voltage source (VCVS) whose number of nodes and parameters are dynamic depending on the parameter EQN.

Parameters

Name	Description	Unit Type	Default
ID	Device ID		VC1
TNOM	Parameter extraction temperature	DegC	27
TEMP	Device temperature	DegC	27
MULT	Number of devices in parallel		1
TC1	Linear temperature coefficient		0
TC2	Quadratic temperature coefficient		0
TYPE	Current or Voltage source mode operation		current source or voltage source
EQN	Current equation		A0+A1*V0+A2*V1

Parameter Details

EQN. This is the current equation. The number of nodes is determined by the voltage variables V0, V1, etc. that are present in the equation. The number of coefficients is based on any other variables that are present in the equation.

For example, the equation A0 + A1*V0 + B0*V1 creates a four-node source due to the voltage variables V0 and V1. Three dynamic parameters are created: A0, A1, and B0.

You can use the following math functions in expressions for this parameter:

`sin`, `cos`, `tan`, `asin`, `acos`, `atan`, `sinh`, `cosh`, `tanh`, `abs`, `min`, `max`, `exp`, `log`, `log10`, `arcsin`, `arccos`, `arctan`, `pow`, `sqrt`, `sqr`

Functions `min`, `max`, and `pow` take two arguments; all others take one. `sqr` and `sqrt` are both square-root functions.

TYPE. The current or voltage source mode operation. Choose **current source** or **voltage source** from the drop-down list to specify the mode.

Dynamic Parameters

Name	Description	Unit Type	Default
A0	Coefficient		0
A1	Coefficient		0
A2	Coefficient		0

Dynamic Parameter Details

The parameters A0, A1, and A2 are the default dynamic parameters. Any variables in the equation string of the EQN parameter are considered dynamic parameters with the exception of the variables Vx where x is any integer that is greater than or equal to zero.

Implementation Details

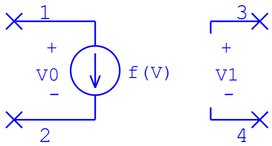
The current equation is defined by the EQN parameter.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Dynamic Nonlinear Voltage-Controlled Charge Source: DYN_VCQS

Symbol



Summary

DYN_VCQS is a nonlinear voltage-controlled charge source whose number of nodes and parameters are dynamic depending on the parameter EQN.

Parameters

Name	Description	Unit Type	Default
ID	Device ID	Text	VC1
TNOM	Parameter extraction temperature	DegC	27
TEMP	Device temperature	DegC	27
MULT	Number of devices in parallel		1
TC1	Linear temperature coefficient		0
TC2	Quadratic temperature coefficient		0
EQN	Charge equation		A0+A1*V0+A2*V1

Parameter Details

EQN. This is the charge equation. The number of nodes is determined by the voltage variables V0, V1, etc. that are present in the equation. The number of coefficients is based on any other variables that are present in the equation.

For example, the equation A0 + A1*V0 + B0*V1 creates a four-node source due to the voltage variables V0 and V1. Three dynamic parameters are created: A0, A1, and B0.

You can use the following math functions in expressions for this parameter:

`sin`, `cos`, `tan`, `asin`, `acos`, `atan`, `sinh`, `cosh`, `tanh`, `abs`, `min`, `max`, `exp`, `log`, `log10`, `arcsin`, `arccos`, `arctan`, `pow`, `sqrt`, `sqr`

Functions `min`, `max`, and `pow` take two arguments; all others take one. `sqr` and `sqrt` are both square-root functions.

Dynamic Parameters

Name	Description	Unit Type	Default
A0	Coefficient		0
A1	Coefficient		0
A2	Coefficient		0

Dynamic Parameter Details

The parameters A0, A1, and A2 are the default dynamic parameters. Any variables in the equation string of the EQN parameter are considered dynamic parameters with the exception of the variables Vx where x is any integer that is greater than or equal to zero.

Implementation Details

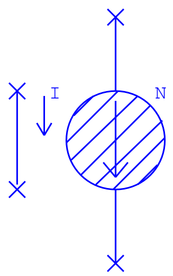
The charge equation is defined by the EQN parameter.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Burst Noise Current: I_BURST

Symbol



Summary

I_BURST models burst noise current. This is a nonwhite noise source.

Parameters

Name	Description	Unit Type	Default
ID	Element ID		NS1
KB	Burst noise coefficient		1e-14
AB	Burst noise exponent		1.0
FB	Burst noise cutoff frequency	Frequency	1.0

Implementation Details

The burst noise current flows through the N(I) branch and is dependent on the current in the I branch. The spectral density of burst noise current, S(f), is

$$S(f) = \mathrm{KB}\frac{I_{\mathrm{dc}}^{\mathrm{AB}}}{1+\left(\frac{f}{\mathrm{FB}}\right)^2} \qquad A^2/\mathrm{Hz}$$

,

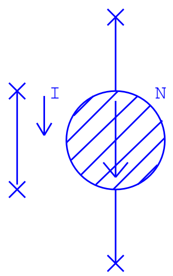
where I_{dc} represents the DC component of the current in the I branch.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

Flicker Noise Current: I_FLICK

Symbol



Summary

I_FLICK models flicker noise current. This is a nonwhite noise source.

Parameters

Name	Description	Unit Type	Default
ID	Element ID		NS1
KF	Flicker noise coefficient		1e-14
AF	Flicker noise exponent		1.0
FFE	Flicker noise freq. exponent		1.0

Implementation Details

The flicker noise current flows through the N(I) branch and is dependent on the current in the I branch. The spectral density of the noise current, S(f), is

$$S(f) = \mathrm{KF}\frac{I_{\mathrm{dc}}^{\mathrm{AF}}}{f^{\mathrm{FFE}}} \qquad A^2 / \mathrm{Hz}$$

,

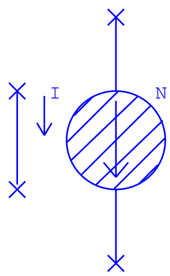
where I_{dc} represents the DC component of the current in the I branch.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Bias-dependent Noise Current: I_GEN

Symbol



Summary

I_GEN models a general bias-dependent noise current. With a proper choice of parameters, it can represent thermal, shot, flicker, or burst noise.

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	NS1
P1	P1 parameter		0
P2	P2 parameter		0
P3	P3 parameter		1.0
P4	P4 parameter		0
P5	P5 parameter		0
COMPAT	Model compatibility selector		AWR

Parameter Details

COMPAT. Allows toggling between two different behaviors: the behavior corresponding to the NI AWR implementation of the model (COMPAT=AWR) and a new one, the ADS compatibility mode of operation (COMPAT=ADS).

Implementation Details

The spectral density of noise current through the N(I) branch, S(f), is

$$S(f) = \mathrm{P1}\frac{I_{\mathrm{dc}}^{P2}}{P3 + P4 f^{P5}} \qquad A^2/\mathrm{Hz}$$

,

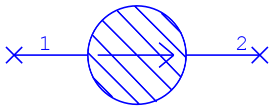
where I_{dc} represents the DC component of the current in the I branch.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Table Specified Frequency-Dependent Spectral Density of Noise Current: I_N_TAB

Symbol



Summary

I_N_TAB models a table-specified frequency-dependent noise current density.

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	NSTAB
FREQUENCY	Frequency Table	Frequency units	{0, 1e6, 1e9} Hz
Fc	Center frequency	Frequency units	1e9 Hz
SP_DENSITY	Normalized spectral density	Sm/Hz(Conductance units)	{0.02, 0.02, 0.02}
METHOD	Linear or Spline Interpolation in $x=\log_{10}f$		Linear

Implementation Details

I_N_TAB specifies the spectral density S(f) of the noise current (in A^2/Hz) that is further normalized by dividing by 4kT0 where k is the Boltzmann constant, and T0=290 K is the noise reference temperature. After this normalization, S(f) is expressed in Sm/Hz units. For example, the default value of 0.02 corresponds to the spectral density of the thermal noise current of a 50 ohm resistor at the reference temperature of 290 K.

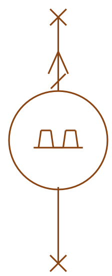
The frequency dependence is specified by a table (frequency, value). Note that FREQUENCY specified in that table is the offset frequency of the noise sideband, and the center frequency of the sideband is Fc. Linear or Spline interpolation is used with the interpolation key $x=\log_{10}|f\text{-}Fc|$ where Fc is the center frequency. If $f<f_{min}$, $S=S(f_{min})$. Interpolation is performed with logarithms of <II*>/(4kT0) so that the plot of spectral density S(f) looks linear. If $f> f_{max}$, $S(f)=S(f_{max})$ where fmin is the smallest frequency value in the table, and fmax is the largest frequency value in the table. You should avoid f=0 since the logarithmic axis is used. The handling of f=0 is to use f=1e-6 Hz, which is not desirable. Note that if the table does not have enough points for the function to be smooth, using the Spline method may result in spurious oscillations, and Linear interpolation (the default) is a better choice.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Pulse Train Current Source: I_PLS

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Current source ID	Text	V1
AMP	Signal magnitude	Current	0 mA
TW	Pulse width	Time	0 ns
TR	Rise time	Time	0 ns
TF	Fall time	Time	0 ns
TD	Time delay	Time	0 ns
WINDOW	Window type		DEFAULT[a]
Offset	Waveform offset (does not affect DC)	Current	0
DCVal	DC value (used for DC analysis)	Current	0

[a]Window type options are none (NONE), Lanczos (DEFAULT, ref [1]), Triangular (TRIANG), Hanning (HANN), Hamming (HAMM), Blackman (BLACK). Please see I_PLS for additional details on the WINDOW parameter.

Parameter Details

WINDOW. To control overshoot (Gibbs phenomenon) the signal spectrum is windowed prior to simulation. Windows attenuate high-frequency components inducing finite rise and fall times. With the exception of the triangular window, the windows are listed in the order of smaller overshoot/longer rise time.

NOTE: AWR simulators add some series resistance to this element during simulation. This may affect results if injecting current into a large resistance. To change the value of the resistance, choose **Options > Default Circuit Options** to display the Circuit Options dialog box, then click the **AWR Sim** tab and under **Convergence Aids** specify the **Series source resistance**.

Layout

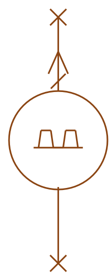
This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

References

[1] R.W. Hamming, Numerical methods for scientists and engineers, pp 534.

Pulse Train Current Source (Tone 2): I_PLS2

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Current source ID	Text	V1
AMP	Signal magnitude	Current	0 mA
F	Signal frequency	Frequency	0 GHz
TW	Pulse width	Time	0 ns
TR	Rise time	Time	0 ns
TF	Fall time	Time	0 ns
TD	Time delay	Time	0 ns
WINDOW	Window type		DEFAULT[a]
Offset	Waveform offset (does not affect DC)	Current	0
DCVal	DC value (used for DC analysis)	Current	0

[a]Window type options are none (NONE), Lanczos (DEFAULT, ref [1]), Triangular (TRIANG), Hanning (HANN), Hamming (HAMM), Blackman (BLACK). Please see I_PLS for additional details on the WINDOW parameter.

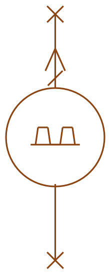
NOTE: AWR simulators add some series resistance to this element during simulation. This may affect results if injecting current into a large resistance. To change the value of the resistance, choose **Options > Default Circuit Options** to display the Circuit Options dialog box, then click the **AWR Sim** tab and under **Convergence Aids** specify the **Series source resistance**.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Piecewise Linear Current Source: I_PWL

Symbol



Summary

The behavior of I_PWL is similar to that of the PWL source in SPICE. The current is specified by the pairs of time-current values (TIME[i], CURRENT[i]). The values of voltage between these time instances are obtained by linear interpolation. The fundamental frequency is equal to that specified for the schematic containing the source.

Parameters

Name	Description	Unit Type	Default
ID	Voltage source ID	Text	V1
TIME	Vector of time values	Time	User has to override the default
CURRENT	Vector of current values	Current	User has to override the default
WINDOW	Window Type		DEFAULT[a]
DCVal	DC value (used for DC analysis)	Current	0

[a]Window type options are none (NONE), Lanczos (DEFAULT, ref [1]), Triangular (TRIANG), Hanning (HANN), Hamming (HAMM), Blackman (BLACK). Please see I_PLS for additional details on the WINDOW parameter.

Parameter Details

The dimension of TIME and CURRENT vectors must be identical. This dimension should be greater than 3, as at least 4 points are required to specify a meaningful PWL waveform. An error message is issued if one of these conditions is not satisfied, and the model is rejected. The values of time should be sorted in ascending order (TIME[i+1]>TIME[i]). Discontinuities in the current value are not allowed (do not specify two different current values for the same time instance).

Implementation Details

Note that the signal needs to be periodic with the period equal to that of the fundamental frequency, otherwise results may be unexpected. The accuracy of the simulation results depends on the number of harmonics of the fundamental frequency. This number of harmonics is determined from the options for the schematic containing this port.

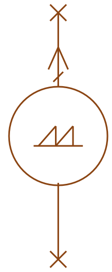
NOTE: AWR simulators add some series resistance to this element during simulation. This may affect results if injecting current into a large resistance. To change the value of the resistance, choose **Options > Default Circuit Options** to display the Circuit Options dialog box, then click the **AWR Sim** tab and under **Convergence Aids** specify the **Series source resistance**.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Saw Wave Current Source: I_SAW

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Current source ID	Text	V1
AMP	Signal magnitude	Current	0 mA
TF	Fall time	Time	0 ns
TD	Time delay	Time	0 ns
WINDOW	Window type		DEFAULT[a]
Offset	Waveform offset (does not affect DC)	Current	0
DCVal	DC value (used for DC analysis)	Current	0

[a]Window type options are none (NONE), Lanczos (DEFAULT, ref [1]), Triangular (TRIANG), Hanning (HANN), Hamming (HAMM), Blackman (BLACK). Please see I_PLS for additional details on the WINDOW parameter.

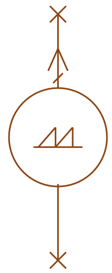
NOTE: AWR simulators add some series resistance to this element during simulation. This may affect results if injecting current into a large resistance. To change the value of the resistance, choose **Options > Default Circuit Options** to display the Circuit Options dialog box, then click the **AWR Sim** tab and under **Convergence Aids** specify the **Series source resistance**.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Saw Wave Current Source (Tone 2): I_SAW2

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Current source ID	Text	V1
AMP	Signal magnitude	Current	0 mA
F	Signal frequency	Frequency	0 GHz
TF	Fall time	Time	0 ns
TD	Time delay	Time	0 ns
WINDOW	Window type		DEFAULT[a]
Offset	Waveform offset (does not affect DC)	Current	0
DCVal	DC value (used for DC analysis)	Current	0

[a]Window type options are none (NONE), Lanczos (DEFAULT, ref [1]), Triangular (TRIANG), Hanning (HANN), Hamming (HAMM), Blackman (BLACK). Please see I_PLS for additional details on the WINDOW parameter.

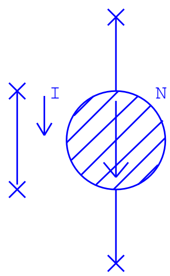
NOTE: AWR simulators add some series resistance to this element during simulation. This may affect results if injecting current into a large resistance. To change the value of the resistance, choose **Options > Default Circuit Options** to display the Circuit Options dialog box, then click the **AWR Sim** tab and under **Convergence Aids** specify the **Series source resistance**.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Shot Noise Current: I_SHOT

Symbol



Summary

I_SHOT models shot noise current.

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	NS1

Implementation Details

The shot noise current flows through the N(I) branch and is dependent on the current in the I branch. The spectral density of shot noise current, S(f), is

$$S(f) = 2qI_{dc} \qquad A^2 / \mathrm{Hz}$$

,

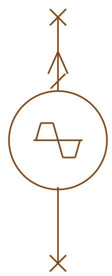
where I_{dc} represents the DC component of the current in the I branch and q is the charge of an electron.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Square Wave Current Source: I_SQR

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Current source ID	Text	V1
AMP	Signal magnitude	Current	0 mA
TR	Rise time	Time	0 ns
TF	Fall time	Time	0 ns
TD	Time delay	Time	0 ns
WINDOW	Window type		DEFAULT[a]
Offset	Waveform offset (does not affect DC)	Current	0
DCVal	DC value (used for DC analysis)	Current	0

[a]Window type options are none (NONE), Lanczos (DEFAULT, ref [1]), Triangular (TRIANG), Hanning (HANN), Hamming (HAMM), Blackman (BLACK). Please see I_PLS for additional details on the WINDOW parameter.

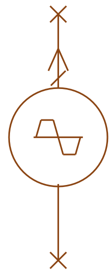
NOTE: AWR simulators add some series resistance to this element during simulation. This may affect results if injecting current into a large resistance. To change the value of the resistance, choose **Options > Default Circuit Options** to display the Circuit Options dialog box, then click the **AWR Sim** tab and under **Convergence Aids** specify the **Series source resistance**.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Square Wave Current Source (Tone 2): I_SQR2

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Current source ID	Text	V1
AMP	Signal magnitude	Current	0 mA
F	Signal frequency	Frequency	0 GHz
TR	Rise time	Time	0 ns
TF	Fall time	Time	0 ns
TD	Time delay	Time	0 ns
WINDOW	Window type		DEFAULT[a]
Offset	Waveform offset (does not affect DC)	Current	0
DCVal	DC value (used for DC analysis)	Current	0

[a]Window type options are none (NONE), Lanczos (DEFAULT, ref [1]), Triangular (TRIANG), Hanning (HANN), Hamming (HAMM), Blackman (BLACK). Please see I_PLS for additional details on the WINDOW parameter.

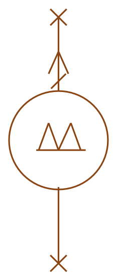
NOTE: AWR simulators add some series resistance to this element during simulation. This may affect results if injecting current into a large resistance. To change the value of the resistance, choose **Options > Default Circuit Options** to display the Circuit Options dialog box, then click the **AWR Sim** tab and under **Convergence Aids** specify the **Series source resistance**.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Triangle Wave Current Source: I_TRI

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Current source ID	Text	V1
AMP	Signal magnitude	Current	0 mA
TD	Time delay	Time	0 ns
WINDOW	Window type		DEFAULT[a]
Offset	Waveform offset (does not affect DC)	Current	0
DCVal	DC value (used for DC analysis)	Current	0

[a]Window type options are none (NONE), Lanczos (DEFAULT, ref [1]), Triangular (TRIANG), Hanning (HANN), Hamming (HAMM), Blackman (BLACK). Please see I_PLS for additional details on the WINDOW parameter.

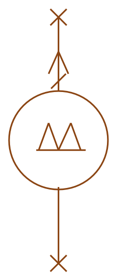
NOTE: AWR simulators add some series resistance to this element during simulation. This may affect results if injecting current into a large resistance. To change the value of the resistance, choose **Options > Default Circuit Options** to display the Circuit Options dialog box, then click the **AWR Sim** tab and under **Convergence Aids** specify the **Series source resistance**.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Triangle Wave Current Source (Tone 2): I_TRI2

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Current source ID	Text	V1
AMP	Signal magnitude	Current	0 mA
F	Signal frequency	Frequency	0 GHz
TD	Time delay	Time	0 ns
WINDOW	Window type		DEFAULT[a]
Offset	Waveform offset (does not affect DC)	Current	0
DCVal	DC value (used for DC analysis)	Current	0

[a]Window type options are none (NONE), Lanczos (DEFAULT, ref [1]), Triangular (TRIANG), Hanning (HANN), Hamming (HAMM), Blackman (BLACK). Please see I_PLS for additional details on the WINDOW parameter.

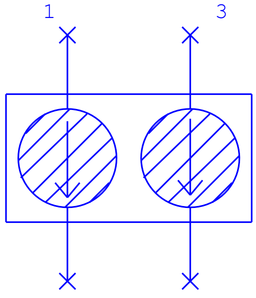
NOTE: AWR simulators add some series resistance to this element during simulation. This may affect results if injecting current into a large resistance. To change the value of the resistance, choose **Options > Default Circuit Options** to display the Circuit Options dialog box, then click the **AWR Sim** tab and under **Convergence Aids** specify the **Series source resistance**.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Two Correlated Noise Current Sources: INCOR2

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	IN1
I1SQ	Noise source current from 1 to 2 in (pA^2)/Hz		320.3
I2SQ	Noise source current from 3 to 4 in (pA^2)/Hz		320.3
I12R	Real part of the correlation coefficient between sources		0
I12I	Imaginary part of the correlation coefficient between sources		0

Implementation Details

I1SQ, I2SQ = Mean-square noise current spectral density in $(pA)^2/Hz$

I1SQ or I2SQ = 320.31312, with I12R = I12I = 0, is T=290K in 50Ω.

The correlation coefficient, I12, is defined as

$$I12 = \frac{i_1 i_2^*}{\sqrt{\overline{i_1^2 i_2^2}}}$$

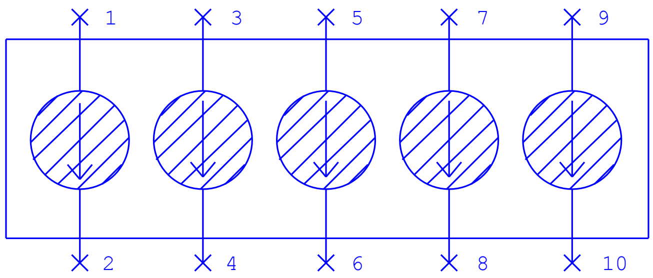
I12 is limited by the software to I12 < 1.0.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

Five Correlated Noise Current Sources: INCOR5

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	IN1
I1SQ	Noise source current from 1 to 2 in (pA^2)/Hz		320.3
I2SQ	Noise source current from 3 to 4 in (pA^2)/Hz		320.3
I3SQ	Noise source current from 5 to 6 in (pA^2)/Hz		320.3
I4SQ	Noise source current from 7 to 8 in (pA^2)/Hz		320.3
I5SQ	Noise source current from 9 to 10 in (pA^2)/Hz		320.3
I12R	Real part of correlation between sources 1 and 2		0
I12I	Imaginary part of correlation between sources 1 and 2		0
I13R	Real part of correlation between sources 1 and 3		0
I13I	Imaginary part of correlation between sources 1 and 3		0
I14R	Real part of correlation between sources 1 and 4		0
I14I	Imaginary part of correlation between sources 1 and 4		0
I15R	Real part of correlation between sources 1 and 5		0
I15I	Imaginary part of correlation between sources 1 and 5		0
I23R	Real part of correlation between sources 2 and 3		0
I23I	Imaginary part of correlation between sources 2 and 3		0
I24R	Real part of correlation between sources 2 and 4		0
I24I	Imaginary part of correlation between sources 2 and 4		0
I25R	Real part of correlation between sources 2 and 5		0
I25I	Imaginary part of correlation between sources 2 and 5		0
I34R	Real part of correlation between sources 3 and 4		0
I34I	Imaginary part of correlation between sources 3 and 4		0
I35R	Real part of correlation between sources 3 and 5		0
I35I	Imaginary part of correlation between sources 3 and 5		0
I45R	Real part of correlation between sources 4 and 5		0

Name	Description	Unit Type	Default
I45I	Imaginary part of correlation between sources 4 and 5		0

Implementation Details

I1SQ, I2SQ, I3SQ, I4SQ, I5SQ = Mean-square noise current spectral density in (pA)2/Hz

I1SQ, I2SQ, I3SQ, I4SQ or I5SQ = 320.31312, with ImnR = ImnI = 0, is T=290K in 50Ω.

The correlation coefficient, Imn, is defined as

$$\mathrm{Imn} = \frac{\overline{i_m i_n^*}}{\sqrt{\overline{i_m^2}\,\overline{i_n^2}}}$$

In the above equations, 1 <= m <= 4, 2 <= n <= 5, and m < n.

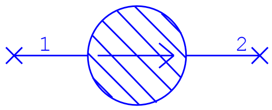
The value of Inm is limited by the software to |Inm| < 1.0 for all m,n.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Current Noise Source: INOISE

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	IN1
I	Noise current in pA/sqrt(Hz)		17.9

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

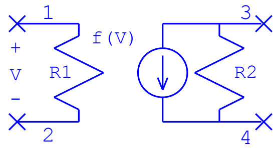
Implementation Details

INOISE implements a white noise source having the stated noise spectral density.

For I=17.897294, T=290K in 50Ω.

Nonlinear Voltage-Controlled Current Source: NLVCCS

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Device ID	Text	VC1
*R1	Input resistance	Resistance	1e+06 ohm
*R2	output resistance	Resistance	1e+06 ohm
*SC	Scale factor		1
*VTH	Threshold voltage	Voltage	-1e+08 V
*TAU	Delay	Time	0 ns
*A0	Polynomial coefficient		0
*A1	Polynomial coefficient		0.01
*A2	Polynomial coefficient		0
*A3	Polynomial coefficient		0
*A4	Polynomial coefficient		0
*A5	Polynomial coefficient		0
*A6	Polynomial coefficient		0
*A7	Polynomial coefficient		0
*A8	Polynomial coefficient		0
*A9	Polynomial coefficient		0
*A10	Polynomial coefficient		0
*A11	Polynomial coefficient		0
*A12	Polynomial coefficient		0
*A13	Polynomial coefficient		0
*A14	Polynomial coefficient		0
*A15	Polynomial coefficient		0

** indicates a secondary parameter*

Implementation Details

For $V \leq VTH$

$f(V)=0.0$

For V>VTH

$$f(V) = \mathrm{SC} \cdot (A0 + A1 \cdot V + A2 \cdot V^2 + \ldots + A15 \cdot V^{15})$$

where

$$V \rightarrow V(t - \tau)$$

NOTES:

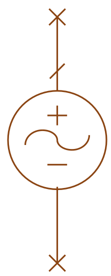
1. SC is a scale factor that scales the parameters as if it were a FET.
2. R1 and R2 are scaled inversely with SC.
3. TAU is a constant time delay.
4. The model does not ensure that dI/dV is continuous at VTH. The user should select polynomial coefficients to guarantee this.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

Harmonic Oscillator at the Specified Frequency (Tone N) with Phase Noise: OSC_W_PH_NOISE

Symbol



Summary

OSC_W_PH_NOISE models a harmonic oscillator at the specified frequency Fc, with the available power Pwr, with the phase noise specified by a table of {FREQUENCY, Phase Noise}. The noise sideband is symmetric around Fc.

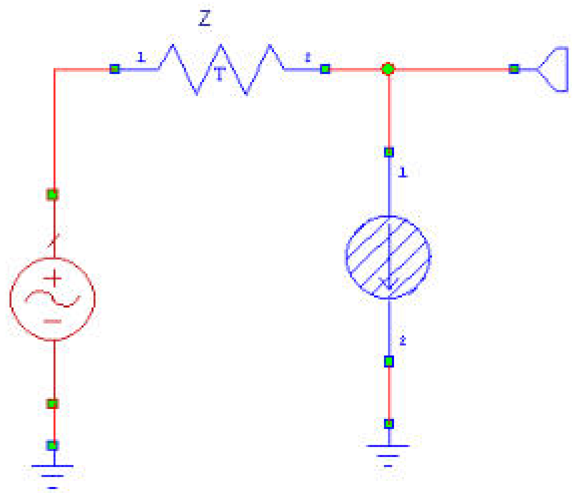
Parameters

Name	Description	Unit Type	Default
ID	Element ID		V1
Z	Termination Impedance	Resistance	50 ohm
Fc	Oscillation Frequency	Frequency	$1x10^9$ Hz
Pwr	Available Power	dB power	10 dBm
Ang	Relative angle of the excitation	Angle	0
Tone	Tone Number for the signal		1
FREQUENCY	Frequency Table	Frequency	{1e4, 1e5, 1e6, 1e7, 1e8} Hz
Phase Noise L(f)	Phase Noise in dB/Hz	dB/Hz	{-75, -100, -120, -141, -145}
Temp	Temperature for thermal noise calculation	Temperature	290 K
METHOD	Linear or Spline Interpolation in $x=\log_{10}f$		Linear
NHarm	Maximum number of harmonics for this tone. Value of zero causes this parameter to be ignored.		0
NSamp	Oversampling factor		1

Parameter Details

NHarm & NSamp. Specify a value of zero to indicate that these parameters do not override the values specified for the **Tone X Harmonics** and **Tone X Over sample** in the project or document harmonic balance settings. (Choose **Options > Default Circuit Options** and click the **AWR Sim** tab.) This only applies when the Tone parameter is set to 1, 2, or 3 for the element. An error is issued if these parameters are set to zero for a source with Tone equal to 4 or more.

Implementation Details



OSC_W_PH_NOISE consists of a sinusoidal voltage source with the series resistance Z, and the table-specified noise source I_N_TAB connected as shown in the previous figure. The values of phase noise L'(f) (in dB) are converted into the normalized spectral density S(f) according to the following expression:

$$S(f) = \frac{10^{0.1(L'+P)}}{4kT_0 Z}$$

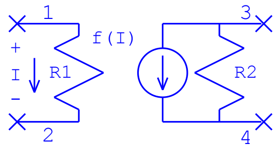
See I_N_TAB for further details on the interpolation of S(f).

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

SPICE Nonlinear Current-Controlled Current Source: SCCCS1

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Device ID	Text	VC1
*R1	Input resistance	Resistance	1 ohm
*R2	output resistance	Resistance	1e+06 ohm
*SC	Scale factor		1
*A0	Polynomial coefficient		0
*A1	Polynomial coefficient		0.01
*A2	Polynomial coefficient		0.01
*A3	Polynomial coefficient		0
*A4	Polynomial coefficient		0
*A5	Polynomial coefficient		0
*A6	Polynomial coefficient		0
*A7	Polynomial coefficient		0
*A8	Polynomial coefficient		0
*A9	Polynomial coefficient		0
*A10	Polynomial coefficient		0
*A11	Polynomial coefficient		0
*A12	Polynomial coefficient		0
*A13	Polynomial coefficient		0
*A14	Polynomial coefficient		0
*A15	Polynomial coefficient		0

** indicates a secondary parameter*

Implementation Details

The polynomials in SPICE controlled sources operate on the instantaneous, time domain representations of their input(s), not on the spectra. Non-zero nonlinear coefficients cause frequency translations. For example, a non-zero square coefficient converts a sinusoidal input into an output with frequency components at DC and at twice the input frequency.

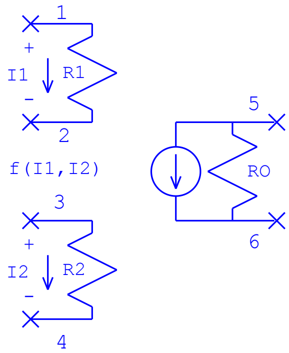
$$f(I) = \mathrm{SC} \cdot (A0 + A1 \cdot I + A2 \cdot I^2 + A3 \cdot I^3 + ...)$$

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

SPICE Current-Controlled Current Source poly=2: SCCCS2

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Device ID	Text	VC1
*R1	Input resistance 1	Resistance	1 ohm
*R2	Input resistance 2	Resistance	1 ohm
*SC	Scale factor		1
*A0	Polynomial coefficient		0
*A1	Polynomial coefficient		0.01
*A2	Polynomial coefficient		0
*A3	Polynomial coefficient		0
*A4	Polynomial coefficient		0
*A5	Polynomial coefficient		0
*A6	Polynomial coefficient		0
*A7	Polynomial coefficient		0
*A8	Polynomial coefficient		0
*A9	Polynomial coefficient		0
*A10	Polynomial coefficient		0
*A11	Polynomial coefficient		0
*A12	Polynomial coefficient		0
*A13	Polynomial coefficient		0
*A14	Polynomial coefficient		0
*A15	Polynomial coefficient		0
*A16	Polynomial coefficient		0
*A17	Polynomial coefficient		0
*A18	Polynomial coefficient		0

Name	Description	Unit Type	Default
*A19	Polynomial coefficient		0
*A20	Polynomial coefficient		0
RO	Output resistance	Resistance	1e+06 ohm

** indicates a secondary parameter*

Implementation Details

The polynomials in SPICE controlled sources operate on the instantaneous, time domain representations of their input(s), not on the spectra. Non-zero nonlinear coefficients cause frequency translations. For example, a non-zero square coefficient converts a sinusoidal input into an output with frequency components at DC and at twice the input frequency.

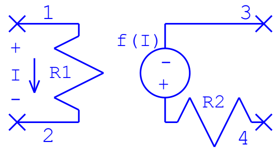
$$f(I_1, I_2) = \mathrm{SC} \cdot (A0 + A1 \cdot I_1 + A2 \cdot I_2 + A3 \cdot I_1^2 + A4 \cdot I_1 I_2 + A5 \cdot I_2^2 + A6 \cdot I_1^3 + A7 \cdot I_1^2 I_2 + \ldots)$$

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

SPICE Current-Controlled Voltage Source poly=1: SCCVS1

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Device ID	Text	VC1
*R1	Input resistance	Resistance	1 ohm
*R2	output resistance	Resistance	1e+06 ohm
*SC	Scale factor		1
*A0	Polynomial coefficient		0
*A1	Polynomial coefficient		0.01
*A2	Polynomial coefficient		0
*A3	Polynomial coefficient		0
*A4	Polynomial coefficient		0
*A5	Polynomial coefficient		0
*A6	Polynomial coefficient		0
*A7	Polynomial coefficient		0
*A8	Polynomial coefficient		0
*A9	Polynomial coefficient		0
*A10	Polynomial coefficient		0
*A11	Polynomial coefficient		0
*A12	Polynomial coefficient		0
*A13	Polynomial coefficient		0
*A14	Polynomial coefficient		0
*A15	Polynomial coefficient		0

** indicates a secondary parameter*

Implementation Details

The polynomials in SPICE controlled sources operate on the instantaneous, time domain representations of their input(s), not on the spectra. Non-zero nonlinear coefficients cause frequency translations. For example, a non-zero square coefficient converts a sinusoidal input into an output with frequency components at DC and at twice the input frequency.

$$f(I) = \mathrm{SC} \cdot (A0 + A1 \cdot I + A2 \cdot I^2 + A3 \cdot I^3 + \ldots)$$

NOTES:

1. SC is a scale factor that scales the parameters as if it were a FET.

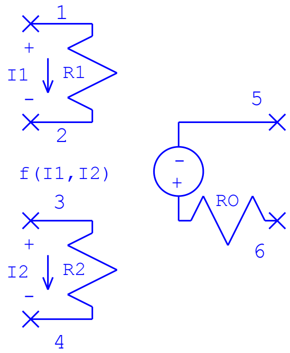
2. R1 and R2 are scaled inversely with SC.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

SPICE Current-Controlled Voltage Source poly=2: SCCVS2

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Device ID	Text	VC1
*R1	Input resistance R1	Resistance	1 ohm
*R2	Input resistance R2	Resistance	1 ohm
*SC	Scale factor		1
*A0	Polynomial coefficient		0
*A1	Polynomial coefficient		0.01
*A2	Polynomial coefficient		0
*A3	Polynomial coefficient		0
*A4	Polynomial coefficient		0
*A5	Polynomial coefficient		0
*A6	Polynomial coefficient		0
*A7	Polynomial coefficient		0
*A8	Polynomial coefficient		0
*A9	Polynomial coefficient		0
*A10	Polynomial coefficient		0
*A11	Polynomial coefficient		0
*A12	Polynomial coefficient		0
*A13	Polynomial coefficient		0
*A14	Polynomial coefficient		0
*A15	Polynomial coefficient		0
*A16	Polynomial coefficient		0
*A17	Polynomial coefficient		0
*A18	Polynomial coefficient		0

Name	Description	Unit Type	Default
*A19	Polynomial coefficient		0
*A20	Polynomial coefficient		0
RO	Output resistance	Resistance	1e+06 ohm

** indicates a secondary parameter*

Implementation Details

The polynomials in SPICE controlled sources operate on the instantaneous, time domain representations of their input(s), not on the spectra. Non-zero nonlinear coefficients cause frequency translations. For example, a non-zero square coefficient converts a sinusoidal input into an output with frequency components at DC and at twice the input frequency.

$$f(I) = \mathrm{SC} \cdot (A0 + A1 \cdot I + A2 \cdot I^2 + A3 \cdot I^3 + \ldots)$$

NOTES:

1. SC is a scale factor that scales the parameters as if it were a FET

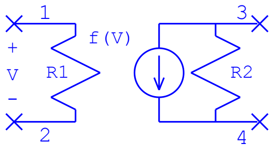
2. R1, R2, and RO are scaled inversely with SC.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

SPICE Voltage-Controlled Current Source: SVCCS1

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Device ID	Text	VC1
*R1	Input resistance	Resistance	1e+06 ohm
*R2	output resistance	Resistance	1e+06 ohm
*SC	Scale factor		1
*A0	Polynomial coefficient		0
*A1	Polynomial coefficient		0.01
*A2	Polynomial coefficient		0
*A3	Polynomial coefficient		0
*A4	Polynomial coefficient		0
*A5	Polynomial coefficient		0
*A6	Polynomial coefficient		0
*A7	Polynomial coefficient		0
*A8	Polynomial coefficient		0
*A9	Polynomial coefficient		0
*A10	Polynomial coefficient		0
*A11	Polynomial coefficient		0
*A12	Polynomial coefficient		0
*A13	Polynomial coefficient		0
*A14	Polynomial coefficient		0
*A15	Polynomial coefficient		0

** indicates a secondary parameter*

Implementation Details

The polynomials in SPICE controlled sources operate on the instantaneous, time domain representations of their input(s), not on the spectra. Non-zero nonlinear coefficients cause frequency translations. For example, a non-zero square coefficient converts a sinusoidal input into an output with frequency components at DC and at twice the input frequency.

$$f(V) = \mathrm{SC} \cdot (A0 + A1 \cdot V + A2 \cdot V^2 + A3 \cdot V^3 + \ldots)$$

NOTES:

1. SC is a scale factor that scales the parameters as if it were a FET.

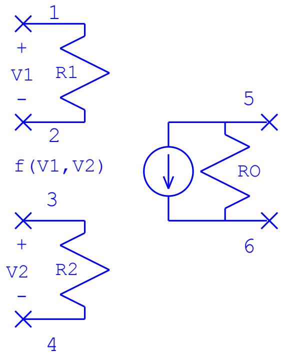
2. R1 and R2 are scaled inversely with SC.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

SPICE Voltage-Controlled Current Source poly=2: SVCCS2

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Device ID	Text	VC1
*R1	Input resistance 1	Resistance	1e+06 ohm
*R2	Input resistance 2	Resistance	1e+06 ohm
*SC	Scale factor		1
*A0	Polynomial coefficient		0
*A1	Polynomial coefficient		0.01
*A2	Polynomial coefficient		0
*A3	Polynomial coefficient		0
*A4	Polynomial coefficient		0
*A5	Polynomial coefficient		0
*A6	Polynomial coefficient		0
*A7	Polynomial coefficient		0
*A8	Polynomial coefficient		0
*A9	Polynomial coefficient		0
*A10	Polynomial coefficient		0
*A11	Polynomial coefficient		0
*A12	Polynomial coefficient		0
*A13	Polynomial coefficient		0
*A14	Polynomial coefficient		0
*A15	Polynomial coefficient		0
*A16	Polynomial coefficient		0
*A17	Polynomial coefficient		0
*A18	Polynomial coefficient		0

Name	Description	Unit Type	Default
*A19	Polynomial coefficient		0
*A20	Polynomial coefficient		0
RO	Output resistance	Resistance	1e+06 ohm

** indicates a secondary parameter*

Implementation Details

The polynomials in SPICE controlled sources operate on the instantaneous, time domain representations of their input(s), not on the spectra. Non-zero nonlinear coefficients cause frequency translations. For example, a non-zero square coefficient converts a sinusoidal input into an output with frequency components at DC and at twice the input frequency.

$$f(V_1, V_2) = \mathrm{SC} \cdot (A0 + A1 \cdot V_1 + A2 \cdot V_2 + A3 \cdot V_1^2 + A4 \cdot V_1 V_2 + A5 \cdot V_2^2 + A6 \cdot V_1^3 + A7 \cdot V_1^2 V_2 + \ldots$$

NOTES:

1. SC is a scale factor that scales the parameters as if it were a FET.

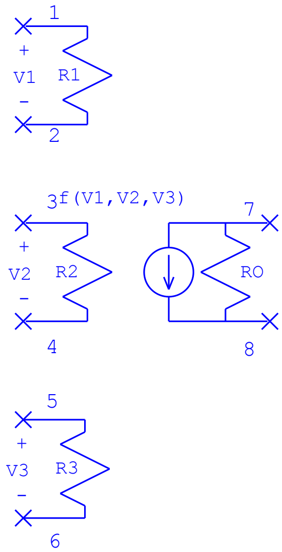
2. R1 and R2 are scaled inversely with SC.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

SPICE Voltage-Controlled Current Source poly=3: SVCCS3

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Device ID	Text	VC1
*R1	Input resistance 1	Resistance	1e+06 ohm
*R2	Input resistance 2	Resistance	1e+06 ohm
*SC	Scale factor		1
*A0	Polynomial coefficient		0
*A1	Polynomial coefficient		0.01
*A2	Polynomial coefficient		0
*A3	Polynomial coefficient		0
*A4	Polynomial coefficient		0
*A5	Polynomial coefficient		0
*A6	Polynomial coefficient		0
*A7	Polynomial coefficient		0
*A8	Polynomial coefficient		0
*A9	Polynomial coefficient		0
*A10	Polynomial coefficient		0
*A11	Polynomial coefficient		0
*A12	Polynomial coefficient		0
*A13	Polynomial coefficient		0
*A14	Polynomial coefficient		0

Name	Description	Unit Type	Default
*A15	Polynomial coefficient		0
*A16	Polynomial coefficient		0
*A17	Polynomial coefficient		0
*A18	Polynomial coefficient		0
*A19	Polynomial coefficient		0
*R3	Input resistance 3	Resistance	1e+06 ohm
RO	Output resistance	Resistance	1e+06 ohm

** indicates a secondary parameter*

Implementation Details

The polynomials in SPICE controlled sources operate on the instantaneous, time domain representations of their input(s), not on the spectra. Non-zero nonlinear coefficients cause frequency translations. For example, a non-zero square coefficient converts a sinusoidal input into an output with frequency components at DC and at twice the input frequency.

$$f(V_1, V_2) = \mathrm{SC} \cdot (A0 + A1 \cdot V_1 + A2V_2 + A3 \cdot V_3 + A4 \cdot V_1^2 + A5 \cdot V_1V_2 + A6 \cdot V_1V_3 + A7 \cdot V_2^2 + A8 \cdot V_2V_3 + A$$

NOTES:

1. SC is a scale factor that scales the parameters as if it were a FET.

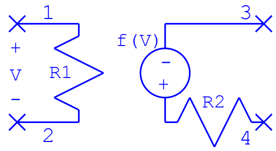
2. R1 and R2 are scaled inversely with SC.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

SPICE Nonlinear Voltage-Controlled Voltage Source: SVCVS1

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Device ID	Text	VC1
*R1	Input resistance	Resistance	1e+06 ohm
*R2	output resistance	Resistance	1 ohm
*SC	Scale factor		1
*A0	Polynomial coefficient		0
*A1	Polynomial coefficient		0.01
*A2	Polynomial coefficient		0
*A3	Polynomial coefficient		0
*A4	Polynomial coefficient		0
*A5	Polynomial coefficient		0
*A6	Polynomial coefficient		0
*A7	Polynomial coefficient		0
*A8	Polynomial coefficient		0
*A9	Polynomial coefficient		0
*A10	Polynomial coefficient		0
*A11	Polynomial coefficient		0
*A12	Polynomial coefficient		0
*A13	Polynomial coefficient		0
*A14	Polynomial coefficient		0
*A15	Polynomial coefficient		0

** indicates a secondary parameter*

Implementation Details

The polynomials in SPICE controlled sources operate on the instantaneous, time domain representations of their input(s), not on the spectra. Non-zero nonlinear coefficients cause frequency translations. For example, a non-zero square coefficient converts a sinusoidal input into an output with frequency components at DC and at twice the input frequency.

$$f(V) = \mathrm{SC} \cdot (A0 + A1 \cdot V + A2 \cdot V^2 + A3 \cdot V^3 + \ldots)$$

NOTES:

1. SC is a scale factor that scales the parameters as if it were a FET.

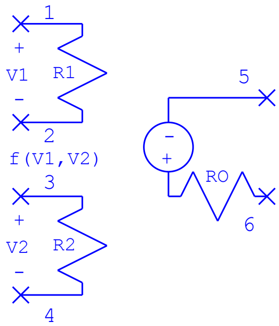
2. R1 and R2 are scaled inversely with SC.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

SPICE Voltage-Controlled Voltage Source poly=2: SVCVS2

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Device ID	Text	VC1
*R1	Input resistance 1	Resistance	1e+06 ohm
*R2	Input resistance 2	Resistance	1e+06 ohm
*SC	Scale factor		1
*A0	Polynomial coefficient		0
*A1	Polynomial coefficient		0.01
*A2	Polynomial coefficient		0
*A3	Polynomial coefficient		0
*A4	Polynomial coefficient		0
*A5	Polynomial coefficient		0
*A6	Polynomial coefficient		0
*A7	Polynomial coefficient		0
*A8	Polynomial coefficient		0
*A9	Polynomial coefficient		0
*A10	Polynomial coefficient		0
*A11	Polynomial coefficient		0
*A12	Polynomial coefficient		0
*A13	Polynomial coefficient		0
*A14	Polynomial coefficient		0
*A15	Polynomial coefficient		0
*A16	Polynomial coefficient		0
*A17	Polynomial coefficient		0
*A18	Polynomial coefficient		0
*A19	Polynomial coefficient		0

Name	Description	Unit Type	Default
*A20	Polynomial coefficient		0
RO	Output resistance	Resistance	1

** indicates a secondary parameter*

Implementation Details

The polynomials in SPICE controlled sources operate on the instantaneous, time domain representations of their input(s), not on the spectra. Non-zero nonlinear coefficients cause frequency translations. For example, a non-zero square coefficient converts a sinusoidal input into an output with frequency components at DC and at twice the input frequency.

$$f(V_1, V_2) = \mathrm{SC} \cdot (A0 + A1 \cdot V_1 + A2 \cdot V_2 + A3 \cdot V_1^2 + A4 \cdot V_1 V_2 + A5 \cdot V_2^2 + A6 \cdot V_1^3 + A7 \cdot V_1^2 V_2 + \ldots$$

NOTES:

1. SC is a scale factor that scales the parameters as if it were a FET.

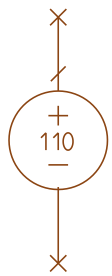
2. R1 and R2 are multiplied by 1/SC.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

Arbitrary, User-specified Bit Sequence: V_ARBS

Symbol



Summary

V_ARBS is a voltage source in the form of a user-specified bit sequence.

Parameters

Name	Description	Unit Type	Default
ID	Voltage source ID	Text	V1
RATE	Symbol rate	Frequency	1 GHz
SEQ	Symbol sequence		{1,0,1,0}
SAMP	Samples per symbol		8
HI	High signal level	Voltage	1
LO	Low signal level	Voltage	0
*TR	Rise time	Time	0
*TF	Fall time	Time	0
TYPE	Signalling format		NRZ[1]
WINDOW	Window type		DEFAULT[2]

** indicates a secondary parameter*

[1] Allowed formats are return to zero (RZ) and non-return to zero (NRZ).

[2] Window type options are none (NONE), Lanczos (DEFAULT, ref [1]), Triangular (TRIANG), Hanning (HANN), Hamming (HAMM), and Blackman (BLACK).

Parameter Details

SEQ. Specifies the symbol sequence. The sequence may be typed in or loaded from a text file. To load a sequence from a file, type **vfile("name_of_file")** in the SEQ field. The file format is:

1

0

1

0

etc

In the absence of a full path specification, place the file in the *\Data* directory under the directory in which MWO is installed.

Symbols in multi-level signal formats are encoded as integers. In a four-level scheme, for example, symbols are represented by 0, 1, 2 and 3.

To save the data with the project you can also import into MWO the data files with sequences.

To use bit data from a file stored in MWO:

1. Import the data file as a "Text Data File" type.
2. Assign a variable to the data file using the following syntax:

   ```
   variable1 = DataFile("data_file_name")
   ```

   where `variable1` is the variable name of your choice and `"data_file_name"` is the name of the imported data file (the name must be enclosed in quotes). This function saves the data in a matrix form.
3. Assign a variable to be the first column of the data in the first variable assigned in the following syntax:

   ```
   variable2 = col(variable1,1)
   ```

   where `variable2` is the variable name of your choice and now stores the bit sequence assigned to the SEQ parameter.

Implementation Details

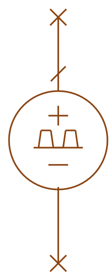
With the exception of the SEQ parameter, V_ARBS is very similar to V_PRBS, which applies a random rather than user-specified sequence. See V_PRBS for information on the remaining parameters.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Jitter Voltage Source with Arbitrary, User-Specified Bit Sequence: V_JITTER

Symbol



Summary

V_JITTER is a voltage source (two terminal device) that can only be used in transient simulations. It is essentially derived from the PORT_ARBS (but made to be an ideal two-terminal voltage source) with the most general specification of the deterministic jitter, and random jitter with Gaussian distribution added to it. The jitter type is selectable using the JITTER_TYPE parameter (so you can get deterministic jitter, random jitter, or both).

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	V1
Rate	Frequency (project units)	GHz	1
SEQ	Bit vector of length N		(1,0,1,0)
SAMP	Integer number		128
LOW	Voltage value	V	0
HIGH	Voltage value	V	1
SignalingType	RZ or NRZ		NR2
TR	Time in project units		
TF	Time in project units		
JITTER_TYPE	(Deterministic, Random, Both)		Deterministic
RE_DELTA	Vector of time values of length N	ns	(0,0,0,0)
FE_DELTA	Vector of time values of length N	ns	(0,0,0,0)
MEAN_R	Time value in project units	ns	0
SIGMA_R	Time value in project units	ns	0
MEAN_F	Time value in project units	ns	0
SIGMA_F	Time value in project units	ns	0
CONST_WIDTH	(Yes, NO)		yes
SEED	Random number generator seed		-1

** indicates a secondary parameter*

Parameter Details

Rate. Symbol Rate

SEQ. Bit sequence

SAMP. Samples per symbol: Used only for harmonic balance simulation. For this type of source, a SAMP on the order of approximately 128 is expected. Thus HB simulations are likely to succeed only for linear circuits (Fourier analysis), and will diverge for strongly nonlinear circuits.

LOW. Low voltage value.

HIGH. High voltage value.

Signaling Type. Waveform type.

TR. Rise time.

TF. Fall time.

JITTER_TYPE. Specifies the type of jitter to be added.

RE_DELTA. Increment of the rising edge start time.

FE_DELTA. Increment of the rising edge start time.

MEAN_R. Mean value of delta for the rising edge.

SIGMA_R. Gaussian distribution width for the rising edge.

MEAN_F. Mean value of delta for the falling edge.

SIGMA_F. Gaussian distribution width of the falling edge.

CONST_WIDTH. If Yes, maintain constant pulse width. This means that exactly the same deterministic and random numbers are used to modify the start of the falling edge as were used to modify the start of the rising edge. The values specified for the rising edge win over those for the falling edge.

SEED. If SEED >=1, the value used to seed the random number generator with Gaussian distribution. If SEED<0, the last tick of the CPU clock is used to seed the random number generator. In this case, random numbers are truly random (not repeatable).

Implementation Details

The V_ JITTER source is closely related to PORT_ARBS with the following differences:

(1) It is a two-terminal voltage source rather than one-terminal port.

(2) The instances of time when the rising and falling edges start is affected by user- specified vectors RE_DELTA (for the rising edge) and FE_DELTA (for the falling edge). Let t_{Rk} (k=1, ..., N) be the time instance when the leading edge of the bit k starts in PORT_ARBS, and t_{Fk} be the time instance when the falling edge begins in PORT_ARBS. In Jitter_D source, the rising edge of bit k begins at t_{Rk}+RE_DELTA[k]+ R[k], and the falling edge begins at t_{Fk}+FE_DELTA[k]+ F[k]. R[k] and F[k] are pseudo-random numbers corresponding to the realizations of Gaussian processes with mean and sigma specified above. If JITTER_TYPE=Random, RE_DELTA and FE_DELTA are ignored. Similarly, if

JITTER_TYPE=Deterministic, no random process is generated. Thus the vectors RE_DELTA and FE_DELTA provide the most general description of deterministic jitter, and MEAN_ and SIGMA_ specify the Gaussian random processes for the start time of the rising and falling edge.

Layout

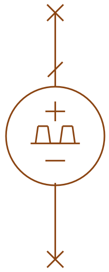
This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Usage

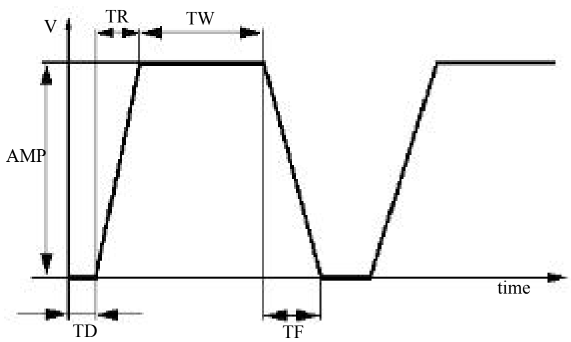
To use deterministic jitter, you must generate a data file with 3 columns (bit value, rising edge delay, and falling edge delay) and import it into Microwave Office. This allows for specifying the desired deterministic functions that could be calculated in a tool other than Microwave Office (like MathCad and Matlab and exported into a file).

Pulse Train Voltage Source: V_PLS

Symbol



Topology



Parameters

Name	Description	Unit Type	Default
ID	Voltage source ID	Text	V1
AMP	Open circuit signal magnitude	Voltage	1 V
TW	Pulse width	Time	0 ns
TR	Rise time	Time	0 ns
TF	Fall time	Time	0 ns
TD	Time delay	Time	0 ns
WINDOW	Window type		DEFAULT[a]
Offset	Waveform offset (does not affect DC)	Voltage	0
DCVal	DC value (used for DC analysis)	Voltage	0

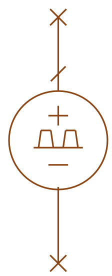
[a]Window type options are none (NONE), Lanczos (DEFAULT, ref [1]), Triangular (TRIANG), Hanning (HANN), Hamming (HAMM), Blackman (BLACK). Please see I_PLS for additional details on the WINDOW parameter.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

Pulse Train Voltage Source (Tone 2): V_PLS2

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Voltage source ID	Text	V1
AMP	Open circuit signal magnitude	Voltage	1 V
F	Signal frequency	Frequency	0 GHz
TW	Pulse width	Time	0 ns
TR	Rise time	Time	0 ns
TF	Fall time	Time	0 ns
TD	Time delay	Time	0 ns
WINDOW	Window type		DEFAULT[a]
Offset	Waveform offset (does not affect DC)	Voltage	0
DCVal	DC value (used for DC analysis)	Voltage	0

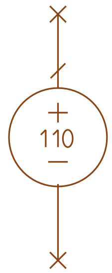
[a]Window type options are none (NONE), Lanczos (DEFAULT, ref [1]), Triangular (TRIANG), Hanning (HANN), Hamming (HAMM), Blackman (BLACK). Please see I_PLS for additional details on the WINDOW parameter.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Pseudo-random Bit Sequence: V_PRBS

Symbol



Summary

V_PRBS is a voltage source in the form of a pseudo-random bit sequence.

Parameters

Name	Description	Unit Type	Default
ID	Voltage source ID	Text	V1
RATE	Symbol rate	Frequency	1 GHz
NSYMB	Number of symbols		16
SAMP	Samples per symbol		8
BITW	Number of bits in symbol		1
HI	High signal level	Voltage	1
LO	Low signal level	Voltage	0
*TR	Rise time	Time	0
*TF	Fall time	Time	0
TYPE	Signalling format		NRZ[1]
WINDOW	Window type		DEFAULT[2]
SEED	Random number generator seed		-1

** indicates a secondary parameter*

[1] Allowed formats are return to zero (RZ) and non-return to zero (NRZ).

[2] Window type options are none (NONE), Lanczos (DEFAULT, ref [1]), Triangular (TRIANG), Hanning (HANN), Hamming (HAMM), and Blackman (BLACK).

Parameter Details

BITW. Can be used to apply multilevel PRSB signals. The number of levels is equal to 2^{BITW}. Levels are equally spaced between HI and LO.

WINDOW. The PRBS sequence must be represented in the frequency domain for the purposes of harmonic balance analysis. To control overshoot (Gibbs phenomenon) the sequence is windowed prior to simulation. Note that windows attenuate high-frequency components and therefore induce a finite rise and fall time. With the exception of the triangular window, the windows are listed in the order of smaller overshoot/longer rise time.

TR and **TF.** Rise and fall times are strongly influenced by the window type and the number of samples used to represent the sequence (the product of SAMP and NSYMB). If precise rise and fall times are critical to the application, use WINDOW=DEFAULT and increase SAMP as necessary.

NSYMB. There are no a priori limitations on the number of symbols in the sequence. However, the number of frequencies in the Fourier representation of a PRBS sequence equals one half of the product of NSYMB and SAMP, so longer sequences result in longer simulations.

NSYMB and **SAMP.** The product of these two parameters must be a power of 2 or an error is produced.

SEED. If less than zero, the current tick count of the system clock is used for seed.

Implementation Details

Harmonic balance options and project frequencies (or nonlinear frequencies) are ignored for the purposes of simulation with V_PRBS.

The fundamental frequency is determined by RATE/NSYMB.

The number of frequencies simulated is determined by NSYBM*SAMP/2 +1. (The +1 is for the DC component.)

The maximum frequency in the spectrum is determined by RATE *SAMP/2.

The number of frequencies is determined by NSYMB and SAMP. Likewise, the fundamental frequency is determined by RATE and NSYMB and is in no relation to the project and nonlinear frequencies.

NOTE:

You can use Veye and Ieye measurements to display eye diagrams in conjunction with V_PRBS simulations.

Layout

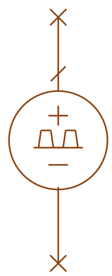
This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

References

[1] R.W. Hamming, Numerical methods for scientists and engineers, pp 534.

Piecewise Linear Voltage Source: V_PWL

Symbol



Summary

The behavior of V_PWL is similar to that of the PWL source in SPICE. The voltage is specified by the pairs of time-voltage values (TIME[i], VOLTAGE[i]). The values of voltage between these time instances are obtained by linear interpolation. The fundamental frequency is equal to that specified for the schematic containing the port.

Parameters

Name	Description	Unit Type	Default
ID	Voltage source ID	Text	V1
TIME	Vector of time values	Time	User has to override the default
Voltage	Vector of voltage values	Voltage	User has to override the default
WINDOW	Window type		DEFAULT[a]
DCVal	DC value (used for DC analysis)	Voltage	0

[a]Window type options are none (NONE), Lanczos (DEFAULT, ref [1]), Triangular (TRIANG), Hanning (HANN), Hamming (HAMM), Blackman (BLACK). Please see I_PLS for additional details on the WINDOW parameter.

Parameter Details

The dimension of TIME and SIG vectors must be identical. This dimension should be greater than 3, as at least 4 points are required to specify a meaningful PWL waveform. An error message is issued if one of these conditions is not satisfied, and the model is rejected. The values of time should be sorted in ascending order (TIME[i+1]>TIME[i]. Discontinuities in the voltage value are not allowed (do not specify two different voltage values for the same time instance).

Implementation Details

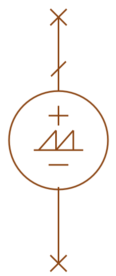
Note that the signal needs to be periodic with the period equal to that of the fundamental frequency, otherwise results may be unexpected. The accuracy of the simulation results depends on the number of the harmonics of the fundamental frequency. This number of harmonics is determined from the options for the schematic containing this source.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Saw Wave Voltage Source: V_SAW

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Voltage source ID	Text	V1
AMP	Open circuit signal magnitude	Voltage	1 V
TF	Fall time	Time	0 ns
TD	Time delay	Time	0 ns
WINDOW	Window type		DEFAULT[a]
Offset	Waveform offset (does not affect DC)	Voltage	0
DCVal	DC value (used for DC analysis)	Voltage	0

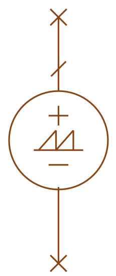
[a]Window type options are none (NONE), Lanczos (DEFAULT, ref [1]), Triangular (TRIANG), Hanning (HANN), Hamming (HAMM), Blackman (BLACK). See I_PLS for additional details on the WINDOW parameter.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Saw Wave Voltage Source (Tone 2): V_SAW2

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Voltage source ID	Text	V1
AMP	Open circuit signal magnitude	Voltage	1 V
F	Signal frequency	Frequency	0 GHz
TF	Fall time	Time	0 ns
TD	Time delay	Time	0 ns
WINDOW	Window type		DEFAULT[a]
Offset	Waveform offset (does not affect DC)	Voltage	0
DCVal	DC value (used for DC analysis)	Voltage	0

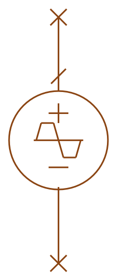
[a]Window type options are none (NONE), Lanczos (DEFAULT, ref [1]), Triangular (TRIANG), Hanning (HANN), Hamming (HAMM), Blackman (BLACK). See I_PLS for additional details on the WINDOW parameter.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Square Wave Voltage Source: V_SQR

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Voltage source ID	Text	V1
AMP	Open circuit signal magnitude	Voltage	1 V
TR	Rise time	Time	0 ns
TF	Fall time	Time	0 ns
TD	Time delay	Time	0 ns
WINDOW	Window type		DEFAULT[a]
Offset	Waveform offset (does not affect DC)	Voltage	0
DCVal	DC value (used for DC analysis)	Voltage	0

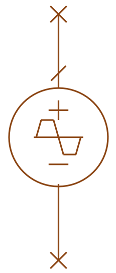
[a]Window type options are none (NONE), Lanczos (DEFAULT, ref [1]), Triangular (TRIANG), Hanning (HANN), Hamming (HAMM), Blackman (BLACK). See I_PLS for additional details on the WINDOW parameter.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Square Wave Voltage Source (Tone 2): V_SQR2

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Voltage source ID	Text	V1
AMP	Open circuit signal magnitude	Voltage	1 V
F	Signal frequency	Frequency	0 GHz
TR	Rise time	Time	0 ns
TF	Fall time	Time	0 ns
TD	Time delay	Time	0 ns
WINDOW	Window type		DEFAULT[a]
Offset	Waveform offset (does not affect DC)	Voltage	0
DCVal	DC value (used for DC analysis)	Voltage	0

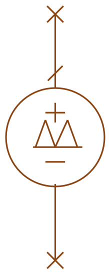
[a]Window type options are none (NONE), Lanczos (DEFAULT, ref [1]), Triangular (TRIANG), Hanning (HANN), Hamming (HAMM), Blackman (BLACK). Please see I_PLS for additional details on the WINDOW parameter.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Triangle Wave Voltage Source: V_TRI

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Voltage source ID	Text	V1
AMP	Open circuit signal magnitude	Voltage	1 V
TD	Time delay	Time	0 ns
WINDOW	Window type		DEFAULT[a]
Offset	Waveform offset (does not affect DC)	Voltage	0
DCVal	DC value (used for DC analysis)	Voltage	0

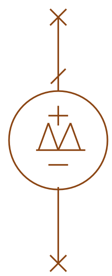
[a]Window type options are none (NONE), Lanczos (DEFAULT, ref [1]), Triangular (TRIANG), Hanning (HANN), Hamming (HAMM), Blackman (BLACK). Please see I_PLS for additional details on the WINDOW parameter.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Triangle Wave Voltage Source (Tone 2): V_TRI2

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Voltage source ID	Text	V1
AMP	Open circuit signal magnitude	Voltage	1 V
F	Signal frequency	Frequency	0 GHz
TD	Time delay	Time	0 ns
WINDOW	Window type		DEFAULT[a]
Offset	Waveform offset (does not affect DC)	Voltage	0
DCVal	DC value (used for DC analysis)	Voltage	0

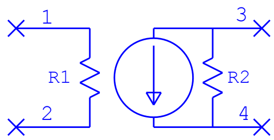
[a]Window type options are none (NONE), Lanczos (DEFAULT, ref [1]), Triangular (TRIANG), Hanning (HANN), Hamming (HAMM), Blackman (BLACK). Please see I_PLS for additional details on the WINDOW parameter.

Layout

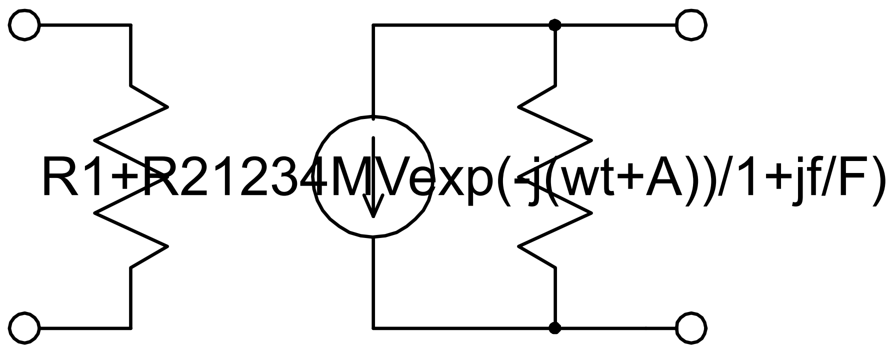
This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

Voltage-Controlled Current Source (Closed Form): VCCS

Symbol



Topology



Parameters

Name	Description	Unit Type	Default
ID	Name	Text	U1
M	DC transconductance	Conductance	1 S
A	Phase offset	Angle	0 Deg
R1	Input resistance	Resistance	0 ohm
R2	Output resistance	Resistance	0 ohm
F	Break frequency	Frequency	0 GHz
T	Time delay	Time	0 ns

Implementation Details

Implements an ideal voltage-controlled current source with output current given by

$$I_{\text{out}} = M \cdot V \frac{e^{-j(\omega T + A)}}{1 + j\frac{f}{F}}$$

NOTES:

R1 and R2 resistance value of 0 means infinite impedance.

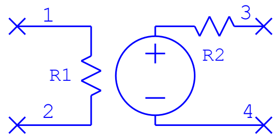
If F = 0, then gain has no frequency dependence.

Layout

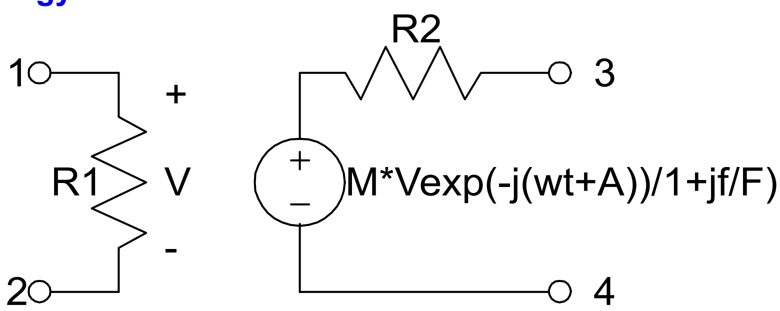
This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Voltage-Controlled Voltage Source: VCVS

Symbol



Topology



Parameters

Name	Description	Unit Type	Default
ID	Name	Text	U1
M	DC voltage gain		1
A	Phase offset	Angle	0 Deg
R1	Input resistance	Resistance	0 ohm
R2	Output resistance	Resistance	1 ohm
F	Break frequency	Frequency	0 GHz
T	Time delay	Time	0 ns

Implementation Details

Implements an ideal voltage-controlled voltage source with output voltage given by

$$V_{\text{out}} = M \cdot V \frac{e^{-j(\omega T + A)}}{1 + j\frac{f}{F}}$$

NOTES:

R1 resistance value of 0 means infinite impedance.

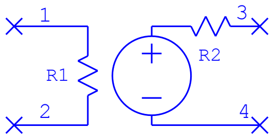
If F = 0, then gain has no frequency dependence.

Layout

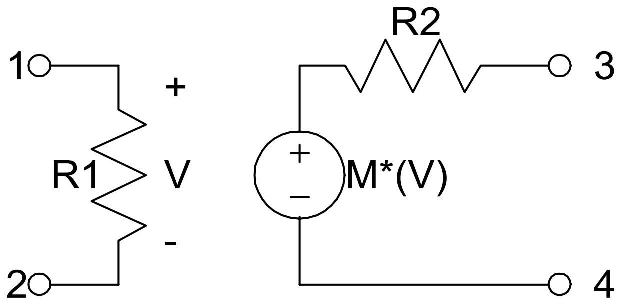
This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Voltage-Controlled Voltage Source: VCVS2

Symbol



Topology



Parameters

Name	Description	Unit Type	Default
ID	Name	Text	U1
M	DC voltage gain		1
R1	Input resistance	Resistance	0 ohm
R2	Output resistance	Resistance	1 ohm

Implementation Details

Implements an ideal voltage-controlled voltage source with output voltage equal to the input voltage times the voltage gain.

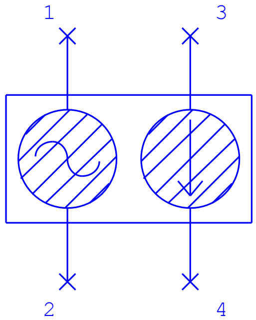
NOTES: An R1 resistance value of 0 means infinite impedance.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Correlated Voltage and Current Noise Sources: VINCOR

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	VIN1
V1SQ	Noise source voltage from 1 to 2 in (nV^2)/Hz		0.8008
I2SQ	Noise source current from 3 to 4 in (pA^2)/Hz		320.3
CorrR	Real part of the correlation coefficient between sources		0
CorrI	Imaginary part of the correlation coefficient between sources		0

Implementation Details

V1SQ = Mean-square noise-voltage spectral density in $(\mathrm{nV})^2/\mathrm{Hz}$

I2SQ = Mean-square noise-current spectral density in $(\mathrm{pA})^2/\mathrm{Hz}$

The correlation coefficient, Corr, is defined as

$$\mathrm{Corr} = \frac{v_1 i_2^*}{\sqrt{\overline{v_1^2 i_2^2}}}$$

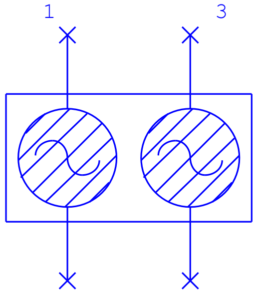
Corr is limited by the software to |Corr| < 1.0.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

Two Correlated Voltage Noise Sources: VNCOR2

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	VIN1
V1SQ	Noise source voltage from 1 to 2 in (nV^2)/Hz		0.8008
V2SQ	Noise source current from 3 to 4 in (nV^2)/Hz		0.8008
V12R	Real part of the correlation coefficient between source		0
V12I	Imaginary part of the correlation coefficient between sources		0

Implementation Details

V1SQ, V2SQ = Mean-square noise voltage spectral density in $(\text{nV})^2/\text{Hz}$

V1SQ or V2SQ = 0.8007832, with V12R = V12I = 0, is T=290K in 50Ω.

The correlation coefficient, V12, is defined as

$$V12 = \frac{v_1 v_2^*}{\sqrt{\overline{v_1^2}\,\overline{v_2^2}}}$$

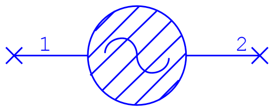
V12 is limited by the software to |V12| < 1.0.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

Voltage Noise Source: VNOISE

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	VN1
V	Noise voltage in nV/sqrt(Hz)		0.8949

Implementation Details

VNOISE implements a white noise source having the stated noise spectral density.

For V=0.8949, T=290K in 50Ω.

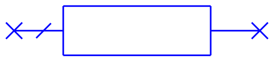
Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Stripline

Stripline (EM Quasi-Static): S1LIN

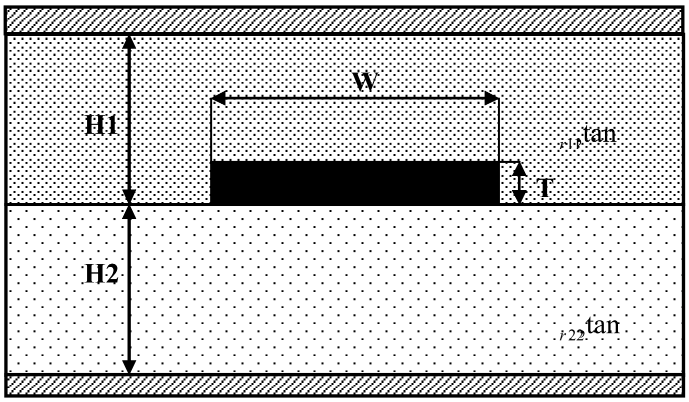
Symbol



Summary

S1LIN models a section of single stripline. The model allows thickness and properties of dielectrics at both sides of a strip to be different.

Topology



Parameters

Name	Description	Unit Type	Default
ID	Name	Text	TL1
W	Conductor width	Length	W[a]
L	Conductor length	Length	L[a]
Acc	Accuracy parameter		1
SSUBL	Substrate definition	Text	3[b]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory.

[b]If only one SSUBL is present in the schematic, this substrate is automatically used. If multiple SSUBL substrate definitions are present, you must specify which to use.

Parameter Details

SSUBL. Two-layer substrate parameters (H1, H2, Er1, Tand1, Er2, Tand2) are listed in the SSUBL model description. The substrate is comprised of the layered substrate itself, metallic cover, and backing ground plane.

Acc. The accuracy parameter. The default value for Acc is 1. If Acc is less than 1 or greater than 10 it is set automatically to 2.

Parameter Restrictions and Recommendations

1. Accuracy parameter A is limited to $1 \le Acc \le 10$. A larger value of Acc increases the density of mesh used in computations. The accuracy of model parameters may gain slightly from increasing Acc at the expense of a noticeable growth in computation time. Generally, a good trade-off between accuracy and computation time is to set Acc to 1.
2. This model does not impose restrictions on the conductor thickness T: thickness may be zero, positive, or negative. However, the model checks if $|T|<0.95H$, where H=H1 if T>0 or H=H2 if T<0 (see Topology). A negative thickness means that the conductor is recessed into the bottom dielectric layer.

Implementation Details

Model implementation is based on EM Quasi-Static technique described in [1]. It accounts for losses in metal and in substrate dielectric. Dispersion is partly included.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See “Cell Options Dialog Box: Layout Tab ” for Cell Options dialog box **Layout** tab details.

See “The Layout Process File (LPF)” for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

Recommendations for Use

NOTE: The implementation of EM Quasi-Static models relies heavily on the involved numerical algorithms. This may lead to a noticeable increase in simulation time for schematics that employ many such models.

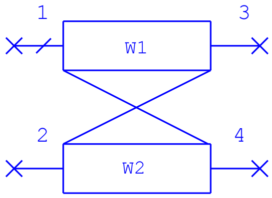
If the thickness of any layer is too small compared to the thickness of another layer, simulation time may also noticeably grow.

References

[1] M.B. Bazdar, A.R. Djordjevic, R.F. Harrington, and T.K. Sarkar, "Evaluation of quasi-static matrix parameters for multiconductor transmission lines using Galerkin's method," IEEE Trans. Microwave Theory Tech., vol. MTT-42, July 1994, pp. 1223-1228.

2 Edge Coupled Striplines (EM Quasi-Static): S2CLIN

Symbol



Parameters

Name	Description	Unit Type	Default
ID	Name	Text	TL1
W1	Conductor 1 width-nodes 1&&3	Length	W[a]
W2	Conductor 2 width-nodes 2&&4	Length	W[a]
S	Gap width - conductors 1&&2	Length	W[a]
L	Conductor length	Length	L[a]
Acc	Accuracy parameter	none	1
SSUBL	Substrate definition	Text	[b]

[a]If only one SSUBL is present in the schematic, this substrate is automatically used. If multiple SSUBL substrate definitions are present, you must specify which to use.

[b]If only one SSUBL is present in the schematic, this substrate is automatically used. If multiple SSUBL substrate definitions are present, you must specify which to use.

Implementation Details

This circuit component models a section of two edge coupled striplines. The model allows the thickness and characteristics of dielectrics at both sides of striplines to be different.

The parameters W1, W2 (strip widths), S (gap between strips) and L (line length) are dimensions entered in the default length units. The parameter SSUBL specifies the two-layered stripline substrate element, which defines additional cross sectional parameters of the transmission line. If blank, a default is used.

The parameter Acc is the accuracy parameter (1<=Acc<=10). The default value for Acc is 1. If Acc is less than 1 or greater than 10 it is sets automatically to 2. Larger value of Acc increase density of mesh used in computations. Accuracy of model parameters may gain slightly from increasing Acc at the expense of noticeable growth of computation time. As a rule of thumb, good trade-off between accuracy and computation time is to set Acc to 1.

This component doesn't impose restrictions on the conductor thickness (thickness may be zero, positive, or negative). Negative thickness means that the conductor is recessed into the underlying dielectric layer.

NOTE: The component accounts for losses in metal and in substrate dielectric. Dispersion is not included.

The implementation of advanced models heavily relies on the involved numerical algorithms. This may lead to the noticeable increase of computation time for some schematics that employ many models of this kind.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See "Cell Options Dialog Box: Layout Tab " for Cell Options dialog box **Layout** tab details.

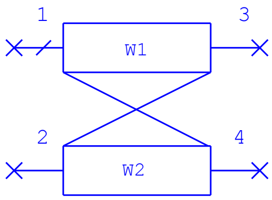
See "The Layout Process File (LPF)" for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

References

[1] M.B. Bazdar, A.R. Djordjevic, R.F. Harrington, and T.K. Sarkar, "Evaluation of quasi-static matrix parameters for multiconductor transmission lines using Galerkin's method," IEEE Trans. Microwave Theory Tech., vol. MTT-42, July 1994, pp. 1223-1228

2 Broadside Coupled Striplines (EM Quasi-Static): SBCPL

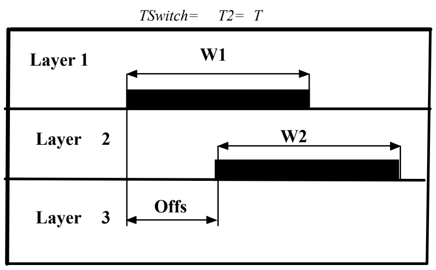
Symbol

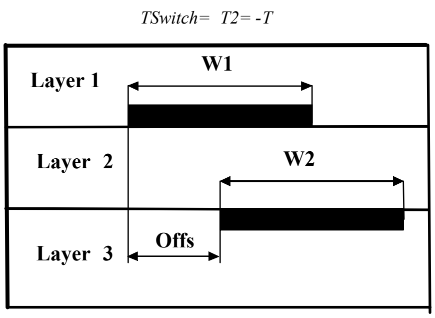


Summary

SBCPL models a section of two broadside coupled striplines arranged at the two opposite boundaries of the internal layer of the three-layer shielded substrate. The bottom line may be displaced laterally relative to the left edge of the top line.

Topology





Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	TL1
W1	Top Conductor width	Length	W[a]
W2	Bottom Conductor width	Length	W[a]
L	Conductor length	Length	W[a]
Offs	Horizontal offset	Length	L[a]
Acc	Accuracy parameter		1
TSwitch	Bottom Conductor Thickness T2= T or T2= -T		T2=T
SSUBT	Substrate Definition	Text	SSUBT1[b]

Name	Description	Unit Type	Default
SNAME1	Structure name from lpf file for top conductor	Text	TOP_BCLIN
SNAME2	Structure name from lpf file for bottom conductor	Text	BOT_BCLIN

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See the Microwave Office Layout Guide for details.

[b]Modify only if the schematic contains multiple substrates. See “Using Elements With Model Blocks” for details

Parameter Details

SSUBT. Three-layer substrate. Parameters are listed in the SSUBT model description.

Acc. The accuracy parameter. The default value is 1. If Acc is less than 1 or greater than 10 it automatically defaults to 2.

TSwitch. Defines how the bottom conductor protrudes: upward or downward. The direction is defined by a combination of the sign of parameter T and the value of TSwitch. For TSwitch="T2=T" T>0 makes both conductors extend their thickness upward; TSwitch="T2=T" and T<0 makes them extend downward. Setting TSwitch="T2=-T" makes the bottom conductor extend the opposite direction relative to the top conductor (if T>0 it extends downward, if T<0 it extends upward.) The default is T>0 and TSwitch="T2=T". See the "Topology" section where two possible layouts are displayed

Offs. A relative horizontal offset of the bottom line left edge from the left edge of the top line. Offs may be positive (when the bottom line is displaced to the right), zero (when both lines are aligned) or negative (when the bottom line is displaced to the left)

Parameter Restrictions and Recommendations

1. Acc is limited to 1≤Acc≤10. A larger value of Acc increases the density of mesh used in computations. The accuracy of model parameters may improve slightly when increasing Acc, but the computation time increases. Generally, a good trade-off between accuracy and computation time is to set Acc to 1.
2. SBCPL does not impose restrictions on conductor thickness (thickness may be zero, positive, or negative). A negative thickness indicates that the conductor is recessed into the substrate.
3. The Offs parameter has an upper limit. If Offs/Wmax exceeds 200, the model issues a warning of decreasing accuracy.
4. Parameter H1 has an upper limit. If H1/(H2+H3) exceeds 50, the model issues a warning of decreasing accuracy.
5. SNAME1 and SNAME2 are for layout only and have no effect on the electrical performance.

Implementation Details

Model implementation is based on EM Quasi-Static technique, described in [1]. It accounts for losses in metal and in substrate dielectric. Dispersion is partly included.

Layout

When you specify SNAME1 and SNAME2, the structures with corresponding names are identified in a $STRUCT_TYPE_BEGIN section of the LPF file. If a structure with the corresponding name is not found, the name of the missing structure is drawn on the error layer. The structure named SNAME1 must contain a text line starting with the line type name used for the top conductor. A structure named SNAME2 must contain a text line starting with the line type name used for the bottom conductor. In a structure named SNAME1=TOP_TRACE and SNAME2= BOT_TRACE with line type names "Top Copper" and "Bot Copper", you need to add the following code to the LPF file (user defined line types may contain any number of layers; structures must contain only one text line):

```
$LINE_TYPE_BEGIN "Top Copper"
  ! -> Layer   offset   minWidth   flags
  "Cu_1"  0  5e-005  0
$LINE_TYPE_END
$LINE_TYPE_BEGIN "Bot Copper"
  ! -> Layer   offset   minWidth   flags
  "Cu_2"  0  5e-005  0
$LINE_TYPE_END
$STRUCT_TYPE_BEGIN "TOP_TRACE"
   "Top Copper" 0 0 0
$STRUCT_TYPE_END
$STRUCT_TYPE_BEGIN "BOT_TRACE"
   "Bot Copper" 0 0 0
$STRUCT_TYPE_END
```

Note that the text inside the structures contains line type names in quotation marks followed by three blank separated zeros.

Recommendations for Use

NOTE: The implementation of EM Quasi-Static models relies heavily on the involved numerical algorithms. This may lead to a noticeable increase in simulation time for schematics that employ many such models.

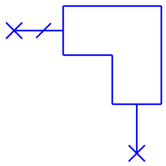
If the thickness of any layer is too small in comparison to the thickness of another layer, simulation time may also noticeably increase.

References

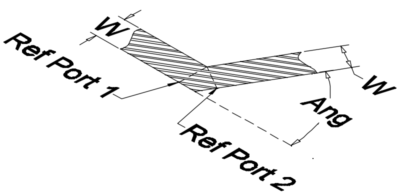
[1] M.B. Bazdar, A.R. Djordjevic, R.F. Harrington, and T.K. Sarkar, "Evaluation of quasi-static matrix parameters for multiconductor transmission lines using Galerkin's method," IEEE Trans. Microwave Theory Tech., vol. MTT-42, July 1994, pp. 1223-1228.

Stripline Bend Unmitered, Arbitrary Angle (Closed Form): SBEND

Symbol



Topology



Parameters

SBEND$ is an iCell. See "Intelligent Cells (iCells)" for an explanation.

Name	Description	Unit Type	Default
ID	Element ID	Text	SL1
W	Conductor width	Length	W[a]
ANG	Angle of bend	Angle	90 Deg
SSUB	Substrate definition	Text	SSUB#[b]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory.

[b]If only one SSUB is present in the schematic, this substrate is automatically used. If multiple SSUB substrate definitions are present, the user must specify.

Implementation Details

This circuit component models an un-mitered sharp bend in the Stripline center conductor. This model includes the reactive components of the bend analyzed as a zero thickness bend of the effective width of the line. The model does not include any losses associated with this parasitic reactance. The parameter W1 (Strip Width) is a dimension entered in the default length units. The parameter Ang (Bend Angle) is a dimension entered in the default angular units. The parameter SSUB specifies the Stripline Substrate element, which defines additional cross sectional parameters of the transmission line. This model also warns of the possibility of higher order propagating modes if they exist.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.

4. Click **OK** to use the new line type in the layout.

See "Cell Options Dialog Box: Layout Tab " for Cell Options dialog box **Layout** tab details.

See "The Layout Process File (LPF)" for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

Restrictions

0.25<W/B ≤ 1.75 (B = Ground Plane Spacing)

$15^{o} \leq ANG \leq 120^{o}$ Recommended

T/B≤0.1 Recommended

$\varepsilon_r \leq 16.0$ Recommended

$1 \leq \varepsilon_r$ Required

Recommendations for Use

Discontinuity models function most accurately when attached to lines that match their corresponding edges. Directly connecting discontinuity models to one another reduces their accuracy.

References

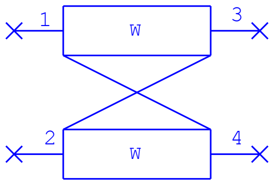
[1] A.A. Oliner, _Equivalent Circuits for Discontinuities in Balanced Strip Transmission Line" IRE Transactions on Microwave Theory and Techniques, March 1955 p.134-143

[2] H.M. Altschuler and A.A. Oliner, "Discontinuities in the Center Conductor of Symmetric Strip Transmission Line", IRE Transactions on Microwave Theory and Techniques, May 1960 p.328-339

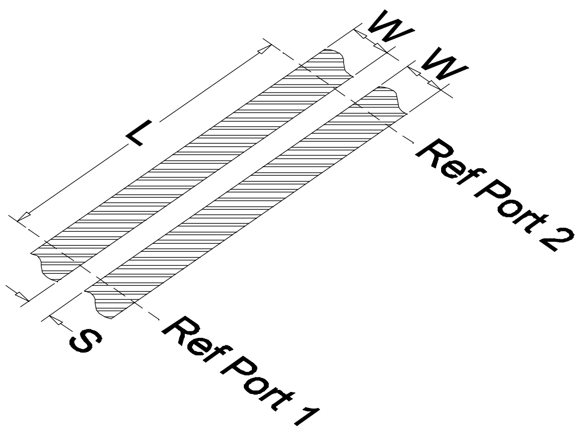
[3] KC. Gupta "Computer-Aided Design of Microwave Circuits", Artech House 1981 p.186-187

Symmetric Edge Coupled Striplines (Closed Form): SCLIN

Symbol



Topology



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	SL1
W	Conductor width	Length	W[a]
S	Gap width	Length	W[a]
L	Conductor length	Length	L[a]
SSUB	Substrate definition	Text	SSUB[b]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory.

[b]If only one SSUB is present in the schematic, this substrate is automatically used. If multiple SSUB substrate definitions are present, you must specify which to use.

Implementation Details

This circuit component models a section of coupled strip transmission lines residing in the same plane. These strips are assumed to reside on the centerline between the two ground planes and be of the same width. The model accounts for both conductive and dielectric loss and corrects for a finite strip thickness.

The parameters W1 (strip widths), S (gap between strips) and L (coupled line length) are dimensions entered in the default length units. The parameter SSUB specifies the stripline substrate element, which defines additional cross sectional parameters of the transmission line.

Important: This model has two implementations: basic implementation and modified impementation. Basic implementation provides frequency-dependent behavior that matches references [1-3]; unfortunately, formulations [1-3] are not very good in low frequency and DC region. Modified implementation improves low frequency and DC behavior but demonstrates frequency behavior somehow different from those of basic implementation at frequencies below 1 GHz. For example, in this frequency region model may underestimate total loss if dielectric loss substantially exceeds conductor loss. SCLIN is not intended for modeling substrates with exorbitant loss tangents (say, above 0.1): Loss of this magnitude should be modeled by EM Quasi-Static models.

To switch between implementation, right-click the schematic that contains the SCLIN model(s), choose **Options**, then on the **AWR Sim** tab, specify the **Model compatibility version**.

To set all SCLIN models in this schematic to basic implementation, select **Version 5.5**.

To set all SCLIN models in this schematic to modified implementation, select **Version 6.0** or **Version 6.5**.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See “Cell Options Dialog Box: Layout Tab ” for Cell Options dialog box **Layout** tab details.

See “The Layout Process File (LPF)” for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

Recommendations for Use

Modified implementation of SCLIN better works in time-domain than basic implementation; however, in critical time-domain designs EM Quasi-Static models like S2CLIN are preferable.

Restrictions

$W/B \geq 0.35$ (Recommended for $T \neq 0$)(B=Ground Plane Spacing)

$T/B \leq 0.1$ Recommended B=Center Strip Thinkness

$T/S \leq 2$ Recommended

NOTE: SCLIN implies that parameter Rho specified on substrate definition SSUB represents bulk resistance of conductor metal normalized to **copper.** This normalization is specific to SCLIN (and SLIN as well); other transmission line models use Rho normalized to **gold.** So, if one obtained value of Rho normalized to gold ($\rho_{\text{norm to gold}}$) than Rho normalized to copper ($\rho_{\text{norm to copper}}$) can be evaluated as $\rho_{\text{norm to copper}}=1.7353\rho_{\text{norm to gold}}$

Reference

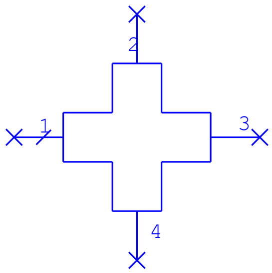
[1] S.B. Cohn "Problems in Strip Transmission Lines" IRE Trans MTT March, 1955 p.119-126

[2] S.B. Cohn "Shielded Coupled-Strip Transmission Line" IRE Trans MTT October, 1955 p.29-38

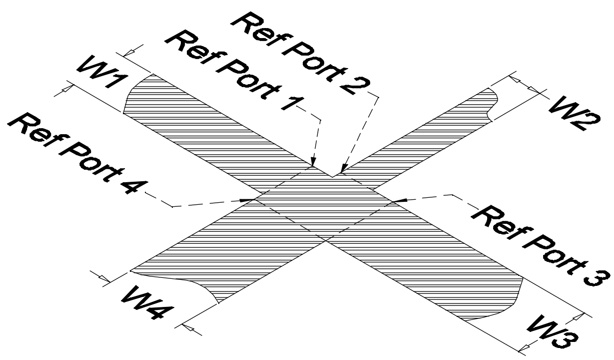
[3] S.B. Cohn "Thickness Corrections for Capacitive Obstacles and Strip Conductors" IRE Trans MTT November, 1960 p.638-644

Stripline Cross - Junction (Closed Form): SCROS

Symbol



Topology



Parameters

SCROS$ is an iCell. See "Intelligent Cells (iCells)" for an explanation.

Name	Description	Unit Type	Default
ID	Element ID	Text	SL1
W1	Conductor width @ port 1	Length	W[a]
W2	Conductor width @ port 2	Length	W[a]
W3	Conductor width @ port 3	Length	W[a]
W4	Conductor width @ port 4	Length	W[a]
SSUB	Substrate definition	Text	[b]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory.

[b]If only one SSUB is present in the schematic, this substrate is automatically used. If multiple SSUB substrate definitions are present, the user must specify.

Implementation Details

This circuit component models a four way right angle junction of Stripline Transmission Lines. This model includes the reactive components of the cross junction analyzed as a zero thickness junction of the effective widths of each section. The model assumes that W1 is equal to W3 and W2 is equal to W4. If this assumption is not true, then the model assumes $W1=W3 = (W1*W3)^{1/2}$ and $W2=W4 = (W2*W4)^{1/2}$. This model does not include any losses associated with this parasitic reactance of the junction. The parameters W1, W2, W3 and W4 (Strip Widths) are dimensions entered in the default length units. The parameter SSUB specifies the Stripline Substrate element, which defines additional cross sectional parameters of the transmission line. This model also warns of the possibility of higher order propagating modes if they exist.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See "Cell Options Dialog Box: Layout Tab " for Cell Options dialog box **Layout** tab details.

See "The Layout Process File (LPF)" for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

Restrictions

$0.2 \leq (Zo1 \cdot (Zo3)/(Zo2 \cdot Zo4))^{1/2} \leq 5$ Recommended

$Zo > 94.25/(\varepsilon_r)^{1/2}$ (for $T \neq 0$) with Zo= Characteristic Impedance of Trans Line.

$T/B \leq 0.1$ Recommended

Recommendations for Use

Discontinuity models function most accurately when attached to lines that match their corresponding edges. Directly connecting discontinuity models to one another reduces their accuracy.

References

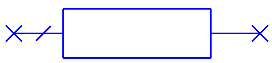
[1] A.A. Oliner, _Equivalent Circuits for Discontinuities in Balanced Strip Transmission Line_ IRE Transactions on Microwave Theory and Techniques, March 1955 p.134-143

[2] H.M. Altschuler and A.A. Oliner, "Discontinuities in the Center Conductor of Symmetric Strip Transmission Line", IRE Transactions on Microwave Theory and Techniques, May 1960 p.328-339

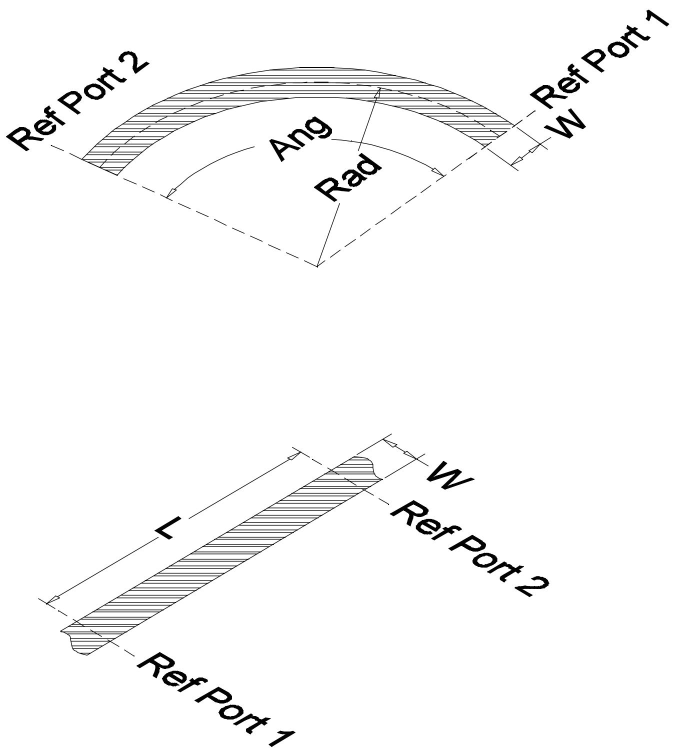
[3] This model was developed by NI AWR via a modal excitation method, which allows the cross to be analyzed as a series of Stripline T-Junctions.

Meander Stripline with Radiused Corners (Closed Form): SCTRACE

Symbol



Topology



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	TL1
W	Conductor width	Length	W[a]
L	Calculated center line length	Length	L[a]
R	Radius of curvature of centerline	Length	Rad[a]
SSUB	Substrate definition	Text	[b]
*DB	Bend position array	Length	Hidden, Changed in Layout
*RB	Bend angle array (degrees)		Hidden, Changed in Layout

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory.

[b]If only one SSUB is present in the schematic, this substrate is automatically used. If multiple SSUB substrate definitions are present, the user must specify.

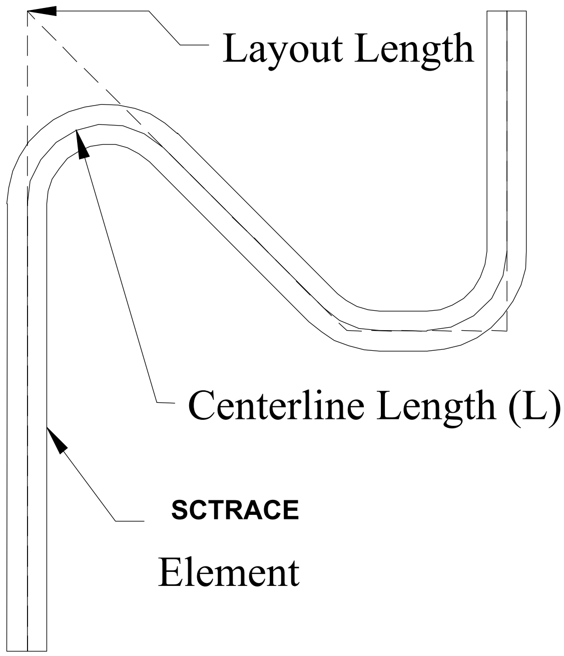
* *indicates a secondary parameter*

Implementation Details

The SCTRACE element is a stripline line that can be routed by graphically editing the element in the layout view. The model of the element is constructed by cascading stripline curves (SCURVE), and stripline lines (SLIN). Reference the SCURVE and SLIN help for details about the models.

In general, the graphical editing of the SCTRACE element is done using mouse commands. Double-clicking on the element will activate the blue grab diamonds that are used to manipulate the bends and lines. Graphical editing of the SCTRACE element is exactly the same as MTRACE. Reference the MTRACE help for specifics in this area.

The model parameter, L, is altered when graphically altering the trace, such as adding or changing bends. In practice, the layout length (defined below) remains constant during these operations, while the center line length (parameter L) changes due to the shortening effect caused by the introducing or modifying curves.



Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

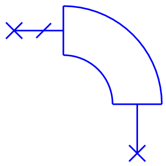
1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See "Cell Options Dialog Box: Layout Tab " for Cell Options dialog box **Layout** tab details.

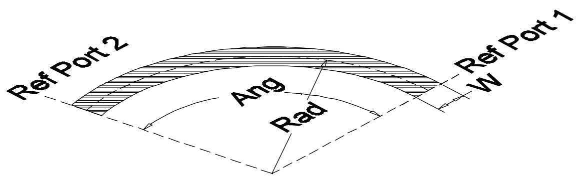
See "The Layout Process File (LPF)" for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

Stripline Radius Corner, Arbitrary Angle (Closed Form): SCURVE

Symbol



Topology



Parameters

SCURVE$ is an iCell. See "Intelligent Cells (iCells)" for an explanation.

Name	Description	Unit Type	Default
ID	Element ID	Text	SL1
W	Conductor width	Length	W[a]
ANG	Angle of bend	Angle	90 Deg
R	Radius of curvature to centerline	Length	Rad[a]
SSUB	Substrate definition	Text	3[b]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory.

[b]If only one SSUB is present in the schematic, this substrate is automatically used. If multiple SSUB substrate definitions are present, you must specify.

Implementation Details

This circuit component models a curved section Stripline Transmission Line including the end effect. The model assumes a TEM (Transverse Electric Magnetic) mode of propagation and incorporates the effects of dielectric and conductive losses. The model determines an effective length and impedance of linear Stripline to incorporate this model. The parameters W (Strip Width) and Rad (Centerline Strip Radius) are lengths entered in the default length units. The parameter ANG (Angle of Curvature) is expressed in the default angular units. The parameter SSUB specifies the Stripline Substrate element, which defines additional cross sectional parameters of the transmission line. This model also warns of the possibility of higher order propagating modes if they exist.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.

2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See "Cell Options Dialog Box: Layout Tab " for Cell Options dialog box **Layout** tab details.

See "The Layout Process File (LPF)" for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

Restrictions

Rad>(W+B/2)/2

$0^o \leq ANG \leq 360^o$ Recommended

T/B≤0.1 Recommended

Rho≤100 Recommended

Recommendations for Use

Discontinuity models function most accurately when attached to lines that match their corresponding edges. Directly connecting discontinuity models to one another reduces their accuracy.

Reference

[1] H.A. Wheeler, "Transmission-Line Properties of a Strip Line Between Parallel Planes", IEEE Trans on Microwave Theory and Techniques, Vol. MTT-26, No. 11, November 1978 p.866-876

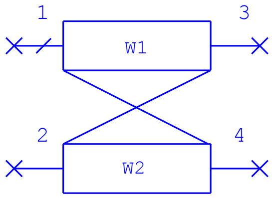
[2] S.B. Cohn, "Characteristic Impedance of the Shielded-Strip Transmission Line", IRE Transactions MTT, July 1954 p. 52-55

[3] H.A. Wheeler, "Formulas for the Skin Effect", Proceedings of the IRE September, 1942 p.412-424

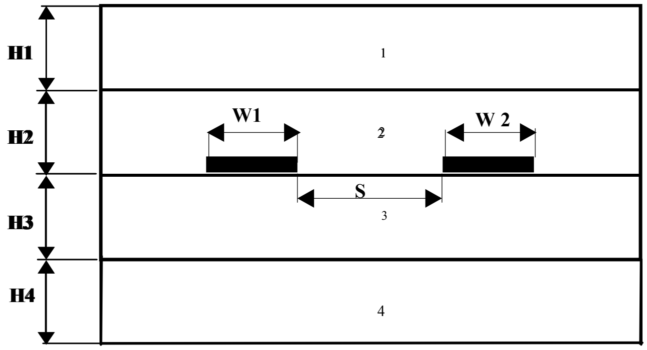
[4] K.C. Gupta "Computer-Aided Design of Microwave Circuits", Artech House1981 p.60

2 Embedded Edge Coupled Striplines (EM Quasi-Static): SEM2LIN

Symbol



Topology



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	TL1
W1	Conductor 1 width - nodes 1&&3	Length	W[a]
W2	Conductor 2 width - nodes 2&&4	Length	W[a]
S	Gap width conductors 1&&2	Length	W[a]
L	Conductor length	Length	L[a]
Acc	Accuracy parameter		1
SSUB4	Substrate definition	Text	[b]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory.

[b]If only one SSUB4 is present in the schematic, this substrate is automatically used. If multiple SSUB4 substrate definitions are present, the user must specify.

Implementation Details

This circuit component models a section of two coupled striplines embedded into a four layered substrate.

The parameters W1, W2 (strips widths), S (spacing between lines) and L (line length) are dimensions entered in the default length units.

The parameter Acc is the accuracy parameter (1<=Acc<=10). The default value for Acc is 1. If Acc is less than 1 or greater than 10 it is sets automatically to 2. Larger value of Acc increase density of mesh used in computations. Accuracy of model parameters may gain slightly from increasing Acc at the expense of noticeable growth of computation time. As a rule of thumb, good trade-off between accuracy and computation time is to set Acc to 1.

The parameter SSUB4 specifies the four-layer stripline substrate element, which defines additional cross sectional parameters of the transmission line. If blank, a default is used. Conductors are implied to reside at the boundary between layer 2 and layer 3.

This component doesn't impose restrictions on the conductor thickness (thickness may be zero, positive, or negative). Negative thickness means that the conductor is recessed into the layer 3.

SEM2LIN may be used to implement various kinds of coupled striplines: buried, coated, inverted, and suspended simply by selecting relevant dielectric constants and thicknesses.

The component accounts for losses in metal and in substrate dielectric. Dispersion is not included.

NOTE: When implementing suspended, coated etc. lines don't make, if possible, thickness of any layer too small in comparison with other layers: the computation time may noticeably grow.

NOTE: The implementation of EM Quasi-Static models heavily relies on the involved numerical algorithms. This may lead to the noticeable increase of computation time for some schematics that employ many models of this kind.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See “Cell Options Dialog Box: Layout Tab ” for Cell Options dialog box **Layout** tab details.

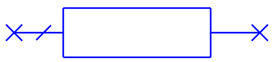
See “The Layout Process File (LPF)” for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

References

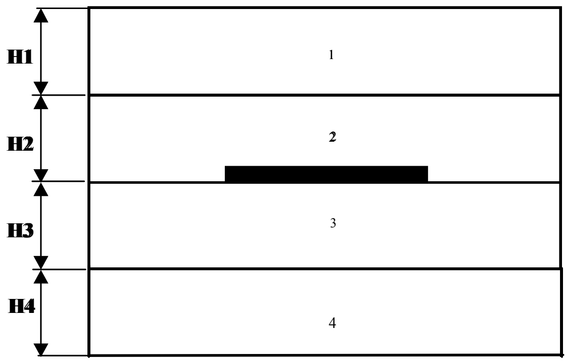
[1] M.B. Bazdar, A.R. Djordjevic, R.F. Harrington, and T.K. Sarkar, "Evaluation of quasi-static matrix parameters for multiconductor transmission lines using Galerkin's method," IEEE Trans. Microwave Theory Tech., vol. MTT-42, July 1994, pp. 1223-1228.

Embedded Stripline: SEMLIN

Symbol



Topology



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	TL1
W	Conductor width	Length	W[a]
L	Conductor length	Length	L[a]
Acc	Accuracy parameter		1
SSUB4	Substrate definition	Text	[b]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory.

[b]If only one SSUB4 is present in the schematic, this substrate is automatically used. If multiple SSUB4 substrate definitions are present, the user must specify.

Implementation Details

This circuit component models a section of single stripline embedded into four layered substrate. Metallic cover is present on top of layer 1.

The parameters W (strip widths) and L (line length) are dimensions entered in the default length units.

The parameter Acc is the accuracy parameter (1<=Acc<=10). The default value for Acc is 1. If Acc is less than 1 or greater than 10 it is sets automatically to 2. Larger value of Acc increase density of mesh used in computations. Accuracy of model parameters may gain slightly from increasing Acc at the expense of noticeable growth of computation time. As a rule of thumb, good trade-off between accuracy and computation time is to set Acc to 1.

The parameter SSUB4 specifies the four-layer stripline substrate element, which defines additional cross sectional parameters of the transmission line. If blank, a default is used. Conductor is implied to reside on the boundary between layer 2 and layer 3.

This component doesn't impose restrictions on the conductor thickness (thickness may be zero, positive, or negative). Negative thickness means that the conductor is recessed into the layer 3.

SEMLIN may be used to implement offset (off-center) and suspended striplines simply by selecting relevant dielectric constants and thicknesses. It also can be used for implementation of covered microstrip line.

The component accounts for losses in metal and in substrate dielectric. Dispersion is not included.

NOTE: Making the thickness of any layer too small in comparison with other layers can increase the calculation time.

WARNING: The implementation of EM Quasi-Static models relies heavily on the involved numerical algorithms. This may lead to the noticeable increase of computation time for some schematics that employ many models of this kind.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See “Cell Options Dialog Box: Layout Tab ” for Cell Options dialog box **Layout** tab details.

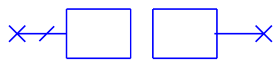
See “The Layout Process File (LPF)” for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

References

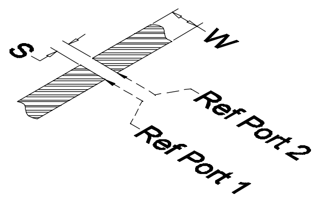
[1] M.B. Bazdar, A.R. Djordjevic, R.F. Harrington, and T.K. Sarkar, "Evaluation of quasi-static matrix parameters for multiconductor transmission lines using Galerkin's method," IEEE Trans. Microwave Theory Tech., vol. MTT-42, July 1994, pp. 1223-1228.

Stripline Gap in Conductor (Closed Form): SGAP

Symbol



Topology



Parameters

SGAP$ is an iCell. See “Intelligent Cells (iCells)” for more information.

Name	Description	Unit Type	Default
ID	Element ID	Text	SL1
W	Conductor width	Length	W[a]
S	Gap width	Length	W[a]
SSUB	Substrate definition	Text	SSUB[b]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory.

[b] If only one SSUB is present in the schematic, this substrate is automatically used. If multiple SSUB substrate definitions are present, you must specify which to use.

Implementation Details

The series gap results in a dominantly series capacitance of a relatively small value. This model does not include the losses associated with the dielectric or conductor and is based on static solutions of a zero thickness strip.

W (Stripline Width) and S (Gap Width) are entered in the default length units.

SSUB references a Stripline substrate, which defines cross sectional parameters.

Layout

This element does not have a visual layout cell. The layout cell creates a gap between the two microstrip or stripline elements that connect to each side. The size of the gap is determined by the S parameter. Since there is no layer to draw, linetypes do not apply. You can select the layout cell to snap the microstrip or stripline elements to it.

Restrictions

W/B≥0.2 Recommended

S/T≥5 Recommeneded

Recommendations for Use

Discontinuity models function most accurately when attached to lines that match their corresponding edges. Directly connecting discontinuity models to one another reduces their accuracy.

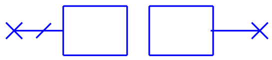
Reference

[1] A.A. Oliner, _Equivalent Circuits for Discontinuities in Balanced Strip Transmission Line_ IRE Transactions on Microwave Theory and Techniques, March 1955 p.134-143

[2] H.M. Altschuler and A.A. Oliner, "Discontinuities in the Center Conductor of Symmetric Strip Transmission Line", IRE Transactions on Microwave Theory and Techniques, May 1960 p.328-339

SGap (EM Base): SGAPX

Symbol

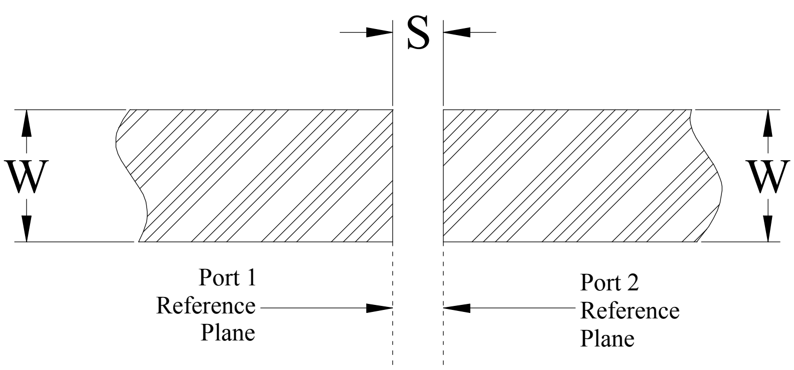


Summary

The SGAPX element models the electrical characteristics of a gap in the signal conductor for a stripline transmission line system. The model is based upon multiple full wave electromagnetic simulations performed on the present substrate configuration. As the results of these EM simulations are stored, these lengthy simulations only have to be performed once.

SGAPX$ is the corresponding intelligent cell (iCell). An iCell model is identical to its non-iCell equivalent with the following exception: Certain dimension-related parameters are not explicitly specified by the user; rather, they are automatically and dynamically determined by the dimensions of the attached elements. See “Intelligent Cells (iCells)” for a detailed discussion of how to use iCells, their advantages, and their limitations.

Topology



Parameters

Name	Description	Unit Type	Default
ID	Name	Text	TL1
W	Conductor width	Length	W[a]
S	Gap width	Length	W[a]
SSUB2	Substrate definition	Text	[b]
*AutoFill	AutoFill database if not equal to 0		0

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See the Microwave Office Layout Guide for details.

[b] Modify only if schematic contains multiple substrates. See “Using Elements With Model Blocks” for details.

** indicates a secondary parameter*

Parameter Restrictions and Recommendations

1. 0.025≤S/B≤0.5 Recommended
2. 0.1≤W/B≤2.0 Recommend
3. 0.1≤H1/H2 and 0.1≤H2/H1
4. B=H1+H2 with H1 and H2 defined in SSUB2

Implementation Details

SGAPX is implemented as an EM-based model, or "X-model". An X-model is based on stored full-wave EM simulations and is the most accurate of models available. See “ EM-based Models (X-models) ” for a detailed discussion of X-models, their advantages, and their limitations.

Layout

This element does not have a visual layout cell. The layout cell creates a gap between the two microstrip or stripline elements that connect to each side. The size of the gap is determined by the S parameter. Since there is no layer to draw, linetypes do not apply. You can select the layout cell to snap the microstrip or stripline elements to it.

Recommendations for Use

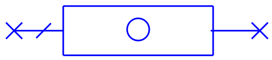
Discontinuity models function most accurately when attached to lines that match their corresponding edges. Directly connecting discontinuity models to one another reduces their accuracy.

References

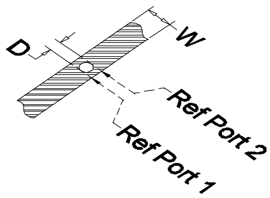
This model was developed under research performed at NI AWR Corporation. The full set of details of the implementation are considered proprietary in nature

Stripline Circular Hole in Center Conductor (Closed Form): SHOLE

Symbol



Topology



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	SL1
W	Conductor width	Length	W[a]
D	Hole diameter	Length	T[a]
SSUB	Substrate definition	Text	SSUB[b]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory.

[b]If only one SSUB is present in the schematic, this substrate is automatically used. If multiple SSUB substrate definitions are present, you must specify which to use.

Implementation Details

This model does not include the losses associated with the dielectric or conductor and is based on static solutions of a zero thickness strip.

W (Stripline Width) and D (Aperture Diameter) are entered in the default length units.

SSUB references a Stripline substrate, which defines cross sectional parameters.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See "Cell Options Dialog Box: Layout Tab " for Cell Options dialog box **Layout** tab details.

See “The Layout Process File (LPF)” for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

Restrictions

D/W≤0.8

Recommendations for Use

Discontinuity models function most accurately when attached to lines that match their corresponding edges. Directly connecting discontinuity models to one another reduces their accuracy.

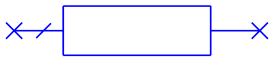
Reference

[1] A.A. Oliner, _Equivalent Circuits for Discontinuities in Balanced Strip Transmission Line_ IRE Transactions on Microwave Theory and Techniques, March 1955 p.134-143

[2] H.M. Altschuler and A.A. Oliner, "Discontinuities in the Center Conductor of Symmetric Strip Transmission Line", IRE Transactions on Microwave Theory and Techniques, May 1960 p.328-339

Inverted Stripline (EM Quasi-Static): SI1LIN

Symbol



Summary

SI1LIN models section of single stripline with a conductor strip placed on the lower surface of a suspended substrate (suspended substrate is a single layer substrate elevated over the infinite grounded plane).

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	TL1
W	Top Conductor width	Length	W[a]
L	Conductor length	Length	L[a]
Acc	Accuracy parameter		1
SPSUB	Substrate Definition	Text	SPSUB1[b]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See the Microwave Office Layout Guide for details.

[b] Modify only if schematic contains multiple substrates. See “Using Elements With Model Blocks” for details.

Parameter Details

SPSUB. Suspended substrate parameters are listed in SPSUB model description.

Acc. The parameter Acc is the accuracy parameter. The default value for Acc is 1. If Acc is less than 1 or greater than 10 it is set automatically to 2.

Parameter Restrictions and Recommendations

1. Accuracy parameter Acc is limited to $1 \le Acc \le 10$. Larger value of Acc increases density of mesh used in computations. Accuracy of model parameters may gain slightly from increasing Acc at the expense of noticeable growth of computation time. As a rule of thumb, good trade-off between accuracy and computation time is to set Acc to 1.
2. This component does not impose restrictions on the conductor thickness (thickness may be zero, positive, or negative). Negative thickness means that the conductor is recessed into the substrate.

Implementation Details

Model implementation is based on EM Quasi-Static technique described in [1]. It accounts for losses in metal and in substrate dielectric. Dispersion is partly included

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.

2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See "Cell Options Dialog Box: Layout Tab " for Cell Options dialog box **Layout** tab details.

See "The Layout Process File (LPF)" for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

Recommendations for Use

NOTE: The implementation of EM Quasi-Static models relies heavily on the involved numerical algorithms. This may lead to a noticeable increase in simulation time for schematics that employ many such models.

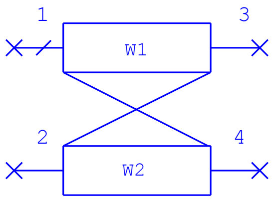
If thickness of any layer is too small in comparison with the thickness of another layer, simulation time may also noticeably grow.

References

[1] M.B. Bazdar, A.R. Djordjevic, R.F. Harrington, and T.K. Sarkar, "Evaluation of quasi-static matrix parameters for multiconductor transmission lines using Galerkin's method," IEEE Trans. Microwave Theory Tech., vol. MTT-42, July 1994, pp. 1223-1228.

2 Inverted Coupled Striplines (EM Quasi-Static): SI2CLIN

Symbol



Summary

SI2CLIN models a section of two-edge coupled asymmetric stripline with a conductor strip placed on the lower surface of a suspended substrate (suspended substrate is a single layer substrate elevated over the infinite grounded plane).

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	TL1
W1	Width of conductor 1	Length	W[a]
W2	Width of conductor 2	Length	W[a]
S	Spacing between conductors	Length	W[a]
L	Conductor length	Length	L[a]
Acc	Accuracy parameter		1
SPSUB	Substrate Definition	Text	SPSUB1[b]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See the Microwave Office Layout Guide for details.

[b]Modify only if schematic contains multiple substrates. See "Using Elements With Model Blocks" for details.

Parameter Details

SPSUB. Suspended substrate parameters are listed in the SPSUB model description.

Acc. The Acc parameter is the accuracy parameter. The default value for Acc is 1. If Acc is less than 1 or greater than 10 it is set automatically to 2.

Parameter Restrictions and Recommendations

1. Acc is limited to $1 \le Acc \le 10$. A larger value of Acc increases the density of mesh used in computations. The accuracy of model parameters may gain slightly from increasing Acc at the expense of noticeable growth of computation time. Generally, a good trade-off between accuracy and computation time is to set Acc to 1.
2. This component does not impose restrictions on the conductor thickness (thickness may be zero, positive, or negative). Negative thickness indicates that the conductor is recessed into the substrate.

Implementation Details

Model implementation is based on the EM Quasi-Static technique described in [1]. It accounts for losses in metal and in substrate dielectric. Dispersion is partly included

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See “Cell Options Dialog Box: Layout Tab ” for Cell Options dialog box **Layout** tab details.

See “The Layout Process File (LPF)” for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

Recommendations for Use

NOTE: The implementation of EM Quasi-Static models relies heavily on the involved numerical algorithms. This may lead to a noticeable increase in simulation time for schematics that employ many such models.

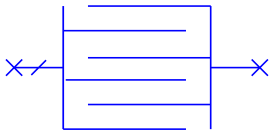
If the thickness of any layer is too small in comparison with the thickness of another layer, simulation time may also noticeably grow.

References

[1] M.B. Bazdar, A.R. Djordjevic, R.F. Harrington, and T.K. Sarkar, "Evaluation of quasi-static matrix parameters for multiconductor transmission lines using Galerkin's method," IEEE Trans. Microwave Theory Tech., vol. MTT-42, July 1994, pp. 1223-1228.

Stripline Interdigital Capacitor: SICAP

Symbol

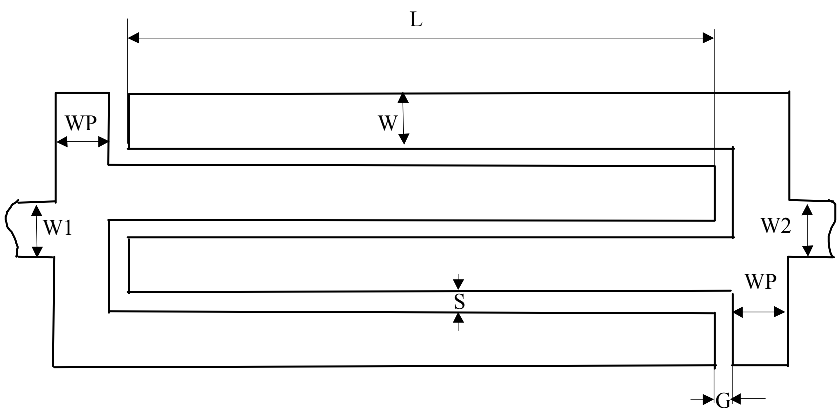


Summary

SICAP models a stripline interdigital capacitor that has an emphasized transverse metallic strip connecting fingers at both input ports. SICAP takes advantage of the EM Quasi-Static coupled lines approach to considering interaction between all fingers.

SICAP$ is the corresponding intelligent cell (iCell). An iCell model is identical to its non-iCell equivalent with the following exception: Certain dimension-related parameters are not explicitly user-specified; rather, they are automatically and dynamically determined by the dimensions of the attached elements. See "Intelligent Cells (iCells)" for a detailed discussion of how to use iCells, their advantages, and their limitations.

Topology



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	MI1
W	Finger width	Length	W[a]
S	Spacing between adjacent fingers	Length	S[a]
G	End gap width	Length	S[a]
L	Length of the overlap region of the fingers	Length	L[a]
N	Number of fingers		4
WP	Width of the fingers transverse interconnect	Length	W[a]
W1	Width of the feeding line at port 1	Length	W[a]
W2	Width of the feeding line at port 2	Length	W[a]

Name	Description	Unit Type	Default
SSUB	Substrate definition	Text	SSUB1[b]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See the Microwave Office Layout Guide for details.

[b]Modify only if schematic contains multiple substrates. See "Using Elements With Model Blocks" for details.

Parameter Details

W1. This parameter is secondary for the SICAP$ iCell model.

W2. This parameter is secondary for the SICAP$ iCell model.

Parameter Restrictions and Recommendations

1. The number of fingers N must be 4≤N≤16. You should use the SICAP1 model instead for all capacitors with N=2 and N=3.
2. Conductor thickness is set via substrate parameters. SICAP1 does not impose restrictions on thickness except for the requirement to be non-negative.

Implementation Details

The EM Quasi-Static technique allows you to model a stripline interdigital capacitor with a wide range of conductor thicknesses.

SICAP1 accounts for the effect of phase shift along the stripline that connects the fingers. It also includes the effect of the presence of width steps at ports.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See "Cell Options Dialog Box: Layout Tab " for Cell Options dialog box **Layout** tab details.

See "The Layout Process File (LPF)" for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

Recommendations for Use

You should use this model for stripline interdigital capacitors that have a prominent metallic strip of width WP connecting fingers at both ports. If this is not the case, and one or both widths of feeding lines exceed 50% of the capacitor width, you should use the SICAP1 model. SICAP1 is geared toward interdigital capacitors that are incorporated into stripline having a width equal to that of the capacitor.

This model accounts for losses in metal and in substrate dielectric. Dispersion is partly included.

To decrease the calculation time for schematics that contain several stripline interdigital capacitors, cache is implemented for this model. This means that during the first evaluation of a schematic, the most time-consuming intermediate parameters for each capacitor instance are stored in memory cache. Each interdigital capacitor model checks this cache looking for its duplicate. Duplicate capacitors copy the appropriate parameters from the memory cache, saving substantially on their recalculation.

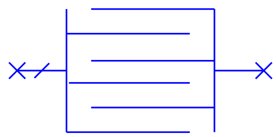
NOTE: The implementation of EM Quasi-Static models relies heavily on the involved numerical algorithms. This may lead to a noticeable increase in simulation time for schematics that employ many such models.

References

[1] B. Bazdar, A.R. Djordjevic, R.F. Harrington, and T.K. Sarkar, "Evaluation of quasi-static matrix parameters for multiconductor transmission lines using Galerkin's method," IEEE Trans. Microwave Theory Tech., vol. MTT-42, July 1994, pp. 1223-1228.

Stripline Interdigital Capacitor (No Steps at Ports): SICAP1

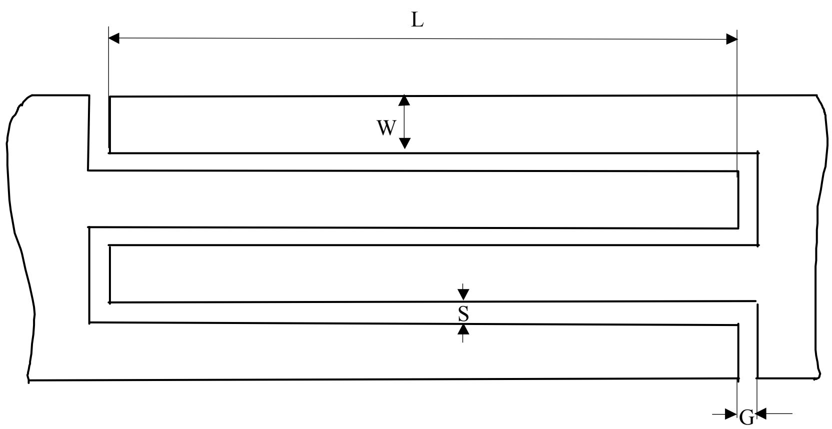
Symbol



Summary

SICAP1 models a stripline interdigital capacitor that connects to striplines without width steps. Model takes advantage of EM Quasi-Static coupled lines approach to consider interaction between all fingers.

Topology



Parameter

Name	Description	Unit Type	Default
ID	Element ID	Text	MI1
W	Finger width	Length	W[a]
S	Spacing between adjacent fingers	Length	S[a]
G	End gap width	Length	S[a]
L	Length of the overlap region of the fingers	Length	L[a]
N	Number of fingers		4
W1	Width of the feeding line at port 1	Length	W[a]
W2	Width of the feeding line at port 2	Length	W[a]
SSUB	Substrate definition	Text	SSUB1[b]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See Microwave Office Layout Guide for details.

[b]Modify only if schematic contains multiple substrates. See the "Using Elements With Model Blocks" for details

Parameter Details

W1, W2. Parameters W1, W2 are read-only (output) parameters, represent the capacitor total width at ports 1 and 2 and are evaluated in the model as N(W+S)-S.

Parameter Restrictions and Recommendations

1. The number of fingers N must be $2 \leq N \leq 16$.
2. Conductor thickness is set via substrate parameters. Model doesn't impose restrictions on thickness except requirement to be non-negative.

Implementation Details

EM Quasi Static technique allows user to model stripline interdigital capacitor with wide range of conductor thicknesses.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See “Cell Options Dialog Box: Layout Tab ” for Cell Options dialog box **Layout** tab details.

See “The Layout Process File (LPF)” for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

Recommendations for Use

The model should be used for stripline interdigital capacitors that connect to feeding stripline without width step. If this is not the case and the emphasized metallic strip that connects end of fingers at input ports is present at both sides (or at one side) it is recommended that model SICAP (or SICAP2) is used instead SICAP1. SICAP is geared toward interdigital capacitor that connects to stripline having widths that are small comparing to capacitor width (less than 50% of capacitor width); SICAP2 has width step only at one port.

The model accounts for losses in metal and in substrate dielectric. Dispersion is partly included.

To decrease the calculation time for schematics that contain several stripline interdigital capacitors cache is implemented for this model. It means that during the first evaluation of schematic the most time consuming intermediate parameters for each capacitor instance are being stored in memory cache. Each interdigital capacitor model checks this cache looking for its duplicate. Duplicate capacitors copy the appropriate parameters from memory cache saving substantially on their recalculation.

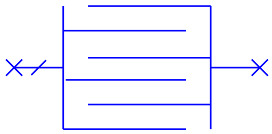
NOTE: The implementation of EM Quasi-Static models relies heavily on the involved numerical algorithms. This may lead to a noticeable increase in simulation time for schematics that employ many such models.

References

[1] B. Bazdar, A.R. Djordjevic, R.F. Harrington, and T.K. Sarkar, "Evaluation of quasi-static matrix parameters for multiconductor transmission lines using Galerkin's method," IEEE Trans. Microwave Theory Tech., vol. MTT-42, July 1994, pp. 1223-1228.

Stripline Interdigital Capacitor (Step at Port 1, No Step at Port 2): SICAP2

Symbol

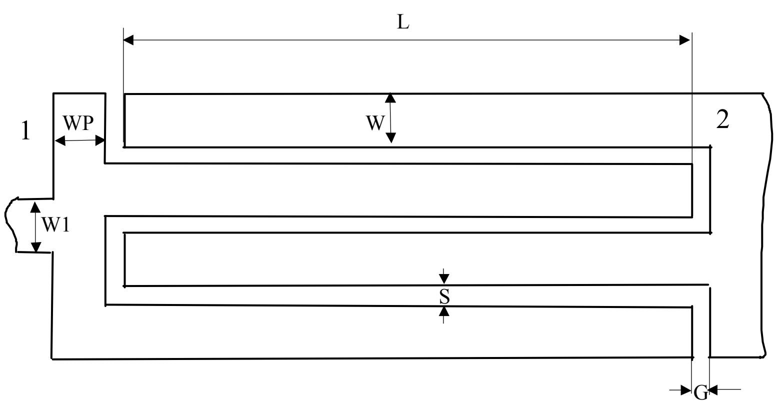


Summary

SICAP2 models a stripline interdigital capacitor that connects to striplines with width step at port 1 and without width step at port 2. The capacitor should have an emphasized metallic strip connecting the ends of fingers at port 1. SICAP2 takes advantage of the EM Quasi-Static coupled lines approach to considering interaction between all fingers.

SICAP2$ is the corresponding intelligent cell (iCell). An iCell model is identical to its non-iCell equivalent with the following exception: Certain dimension-related parameters are not explicitly user-specified; rather, they are automatically and dynamically determined by the dimensions of the attached elements. See "Intelligent Cells (iCells)" for a detailed discussion of how to use iCells, their advantages, and their limitations.

Topology



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	MI1
W	Finger width	Length	W[a]
S	Spacing between adjacent fingers	Length	S[a]
G	End gap width	Length	S[a]
L	Length of the overlap region of the fingers	Length	L[a]
N	Number of fingers		4
WP	Width of the fingers transverse interconnect	Length	W[a]
W1	Width of the feeding line at port 1	Length	W[a]
W2	Width of the feeding line at port 2	Length	W[a]

Name	Description	Unit Type	Default
SSUB	Substrate definition	Text	SSUB1[b]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See Microwave Office Layout Guide for details.

[b] Modify only if the schematic contains multiple substrates. See the "Using Elements With Model Blocks" for details.

Parameter Details

W1. This parameter is secondary for the SICAP2$ iCell model.

W2. A read-only (output) parameter that represents the capacitor total width at ports 2 and is evaluated in the model as N(W+S)-S.

Parameter Restrictions and Recommendations

1. The number of fingers N must be 4≤N≤16.
2. Conductor thickness is set via substrate parameters. SICAP2 does not impose restrictions on thickness except for the requirement to be non-negative.

Implementation Details

The EM Quasi-Static technique allows you to model stripline interdigital capacitors with a wide range of conductor thicknesses.

SICAP2 accounts for the effect of phase shift along the stripline that connects fingers. It also includes the effect of the presence of the width step at one port.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See "Cell Options Dialog Box: Layout Tab " for Cell Options dialog box **Layout** tab details.

See "The Layout Process File (LPF)" for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

Recommendations for Use

This model should be used for stripline interdigital capacitors that have a prominent metallic strip of width WP connecting fingers at port 1. The capacitor and stripline have the same width at port 2. This model is convenient for modeling capacitors grounded at port 2. If W1 exceeds 50% of capacitor width you should use the SICAP1 model. SICAP1 is geared toward interdigital capacitors that are incorporated into stripline having a width equal to that of the capacitor. You should also use SICAP1 for all capacitors with N=2 and N=3.

This model accounts for losses in metal and in substrate dielectric. Dispersion is partly included.

To decrease the calculation time for schematics that contain several stripline interdigital capacitors, cache is implemented for this model. This means that during the first evaluation of a schematic the most time-consuming intermediate parameters for each capacitor instance are stored in a memory cache. Each interdigital capacitor model checks this cache looking for its duplicate. Duplicate capacitors copy the appropriate parameters from the memory cache, saving substantially on their recalculation.

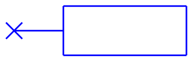
NOTE: The implementation of EM Quasi-Static models relies heavily on the involved numerical algorithms. This may lead to a noticeable increase in simulation time for schematics that employ many such models.

References

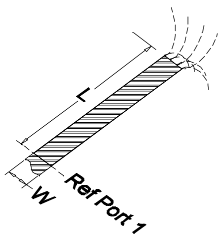
[1] B. Bazdar, A.R. Djordjevic, R.F. Harrington, and T.K. Sarkar, "Evaluation of quasi-static matrix parameters for multiconductor transmission lines using Galerkin's method," IEEE Trans. Microwave Theory Tech., vol. MTT-42, July 1994, pp. 1223-1228.

Open Stripline With End Effect (Closed Form): SLEF

Symbol



Topology



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	SL1
W	Conductor width	Length	W[a]
L	Conductor length	Length	L[a]
SSUB	Substrate definition	Text	[b]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory.

[b]If only one SSUB is present in the schematic, this substrate is automatically used. If multiple SSUB substrate definitions are present, the user must specify.

Implementation Details

This circuit component models an open circuited length of Stripline Transmission Line including the end effect. The model assumes a TEM (Transverse Electric Magnetic) mode of propagation and incorporates the effects of dielectric and conductive losses. The parameters W (Strip Width) and L (Strip Length) are lengths entered in the default length units. The parameter SSUB specifies the Stripline Substrate element, which defines additional cross sectional parameters of the transmission line. This model also warns of the possibility of higher order propagating modes if they exist.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See "Cell Options Dialog Box: Layout Tab " for Cell Options dialog box **Layout** tab details.

See "The Layout Process File (LPF)" for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

Restrictions

Zo>94.25/$(\varepsilon_r)^{1/2}$ (for T≠0) with Zo= Characteristic Impedance of Trans Line.

T/B≤0.1 Recommended

Rho≤100 Recommended

Reference

[1] H.A. Wheeler, "Transmission-Line Properties of a Strip Line Between Parallel Planes", IEEE Trans on Microwave Theory and Techniques, Vol. MTT-26, No. 11, November 1978 p.866-876

[2] S.B. Cohn, "Characteristic Impedance of the Shielded-Strip Transmission Line", IRE Transactions MTT, July 1954 p. 52-55

[3] H.A. Wheeler, "Formulas for the Skin Effect", Proceedings of the IRE September, 1942 p.412-424

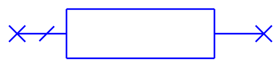
[4] K.C. Gupta "Computer-Aided Design of Microwave Circuits", Artech House1981 p.60

[5] A.A. Oliner, "Equivalent Circuits for Discontinuities in Balanced Strip Transmission Line" IRE Transactions on Microwave Theory and Techniques, March 1955 p.134-143

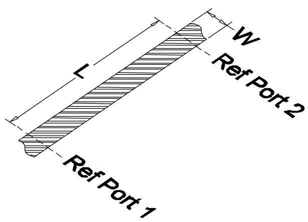
[6] H.M. Altschuler and A.A. Oliner, "Discontinuities in the Center Conductor of Symmetric Strip Transmission Line", IRE Transactions on Microwave Theory and Techniques, May 1960 p.328-339

Single Conductor Stripline (Closed Form): SLIN

Symbol



Topology



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	SL1
W	Conductor width	Length	W[a]
L	Conductor length	Length	L[a]
SSUB	Substrate definition	Text	3[b]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory.

[b]If only one SSUB is present in the schematic, this substrate is automatically used. If multiple SSUB substrate definitions are present, the user must specify.

Implementation Details

This circuit component models a length of Stripline Transmission Line. The model assumes a TEM (Transverse Electric Magnetic) mode of propagation and incorporates the effects of dielectric and conductive losses. The parameters W (Strip Width) and L (Strip Length) are lengths entered in the default length units. The parameter SSUB specifies the Stripline Substrate element, which defines additional cross sectional parameters of the transmission line. This model also warns of the possibility of higher order propagating modes if they exist.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See “Cell Options Dialog Box: Layout Tab ” for Cell Options dialog box **Layout** tab details.

See “The Layout Process File (LPF)” for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

Restrictions

$Zo>94.25/(\varepsilon_r)^{1/2}$ (for $T\neq0$) with Zo= Characteristic Impedance of Trans Line.

$T/B\leq0.1$ Recommended

$Rho\leq100$ Recommended

NOTE: SCLIN implies that parameter Rho specified on substrate definition SSUB represents bulk resistance of conductor metal normalized to **copper**. This normalization is specific to SCLIN (and SLIN as well); other transmission line models use Rho normalized to **gold**. So, if one obtained value of Rho normalized to gold ($\rho_{norm\ to\ gold}$) than Rho normalized to copper ($\rho_{norm\ to\ copper}$) can be evaluated as $\rho_{norm\ to\ copper}=1.4353\rho_{norm\ to\ gold}$

References

[1] H.A. Wheeler, "Transmission-Line Properties of a Strip Line Between Parallel Planes", IEEE Trans on Microwave Theory and Techniques, Vol. MTT-26, No. 11, November 1978 p.866-876

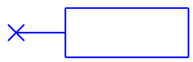
[2] S.B. Cohn, "Characteristic Impedance of the Shielded-Strip Transmission Line", IRE Transactions MTT, July 1954 p. 52-55

[3] H.A. Wheeler, "Formulas for the Skin Effect", Proceedings of the IRE September, 1942 p.412-424

[4] K.C. Gupta "Computer-Aided Design of Microwave Circuits", Artech House1981 p.60

(Obsolete) Open Ended Stripline without End Effect (Closed Form): SLOC

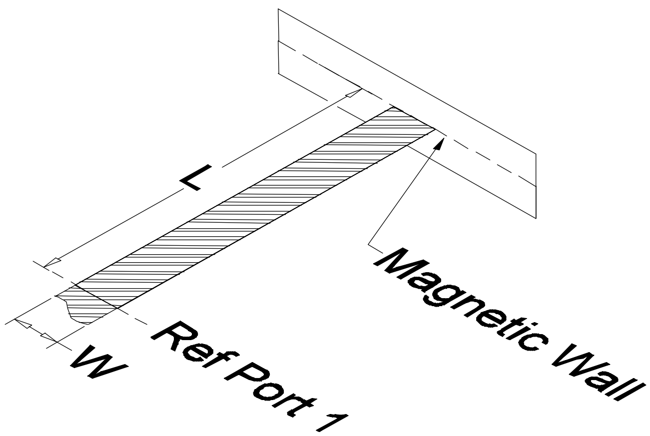
Symbol



Summary

This element is OBSOLETE and is replaced by the Open Stripline With End Effect (Closed Form) (SLEF) element.

Topology



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	SL1
W	Conductor width	Length	W[a]
L	Conductor length	Length	L[a]
SSUB	Substrate definition	Text	SSUB[b]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory.

[b]If only one SSUB is present in the schematic, this substrate is automatically used. If multiple SSUB substrate definitions are present, you must specify which to use.

Implementation Details

This circuit component models an open circuited length of Stripline Transmission Line. The model assumes a TEM (Transverse Electric Magnetic) mode of propagation and incorporates the effects of dielectric and conductive losses. Importantly, this model does not include the loss or capacitance of the end effect of the transmission line; rather, it models the transmission line terminated in a magnetic wall. The parameters W (Strip Width) and L (Strip Length) are lengths entered in the default length units. The parameter SSUB specifies the Stripline Substrate element, which defines additional cross sectional parameters of the transmission line. This model also warns of the possibility of higher order propagating modes if they exist.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See "Cell Options Dialog Box: Layout Tab " for Cell Options dialog box **Layout** tab details.

See "The Layout Process File (LPF)" for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

Restrictions

$Zo>94.25/(\varepsilon_r)^{1/2}$ (for $T\neq0$) with Zo= Characteristic Impedance of Trans Line.

$T/B\leq0.1$ Recommended

$Rho\leq 100$ Recommended

Reference

[1] H.A. Wheeler, "Transmission-Line Properties of a Strip Line Between Parallel Planes", IEEE Trans on Microwave Theory and Techniques, Vol. MTT-26, No. 11, November 1978 p.866-876

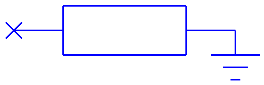
[2] S.B. Cohn, "Characteristic Impedance of the Shielded-Strip Transmission Line", IRE Transactions MTT, July 1954 p. 52-55

[3] H.A. Wheeler, "Formulas for the Skin Effect", Proceedings of the IRE September, 1942 p.412-424

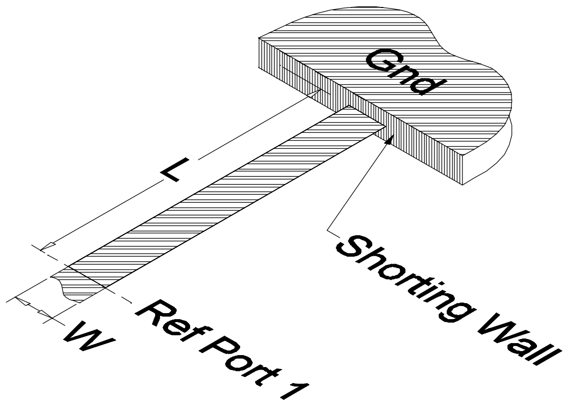
[4] K.C. Gupta "Computer-Aided Design of Microwave Circuits", Artech House1981 p.60

Shorted Circuited Stripline Without End Effect (Closed Form): SLSC

Symbol



Topology



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	SL1
W	Conductor width	Length	W[a]
L	Conductor length	Length	L[a]
SSUB	Substrate definition	Text	[b]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory.

[b]If only one SSUB is present in the schematic, this substrate is automatically used. If multiple SSUB substrate definitions are present, the user must specify.

Implementation Details

This circuit component models a shorted length of Stripline Transmission Line. The model assumes a TEM (Transverse Electric Magnetic) mode of propagation and incorporates the effects of dielectric and conductive losses. Importantly, this model does not include the loss or inductance of a grounding strip, rather, it models the transmission line terminated in a perfectly conducting wall. The parameters W (Strip Width) and L (Strip Length) are lengths entered in the default length units. The parameter SSUB specifies the Stripline Substrate element, which defines additional cross sectional parameters of the transmission line. This model also warns of the possibility of higher order propagating modes if they exist.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See "Cell Options Dialog Box: Layout Tab " for Cell Options dialog box **Layout** tab details.

See "The Layout Process File (LPF)" for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

Restrictions

Zo>94.25/$(\varepsilon_r)^{1/2}$ (for T≠0) with Zo= Characteristic Impedance of Trans Line.

T/B≤0.1 Recommended

Rho≤100 Recommended

References

[1] H.A. Wheeler, "Transmission-Line Properties of a Strip Line Between Parallel Planes", IEEE Trans on Microwave Theory and Techniques, Vol. MTT-26, No. 11, November 1978 p.866-876

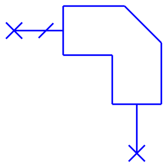
[2] S.B. Cohn, "Characteristic Impedance of the Shielded-Strip Transmission Line", IRE Transactions MTT, July 1954 p. 52-55

[3] H.A. Wheeler, "Formulas for the Skin Effect", Proceedings of the IRE September, 1942 p.412-424

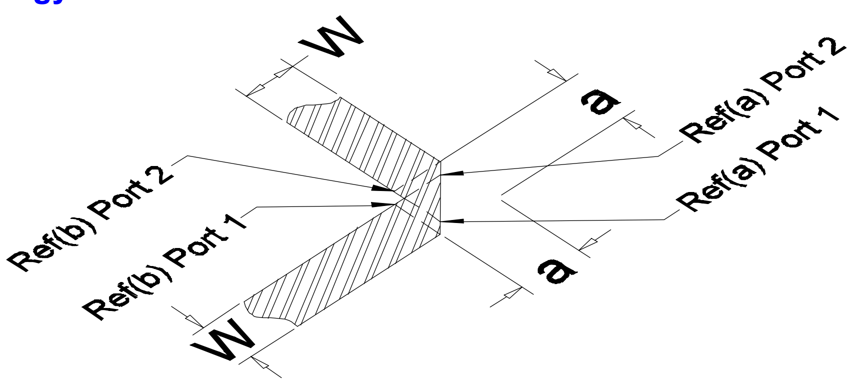
[4] K.C. Gupta "Computer-Aided Design of Microwave Circuits", Artech House1981 p.60

Stripline Bend Optimally Mitered, 90 deg (Closed Form): SMITER

Symbol



Topology



Parameters

SMITER$ is an iCell. See "Intelligent Cells (iCells)" for an explanation of iCells.

Name	Description	Unit Type	Default
ID	Element ID	Text	SL1
W	Conductor width	Length	W[a]
SSUB	Substrate definition	Text	SSUB[b]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory.

[b]If only one SSUB is present in the schematic, this substrate is automatically used. If multiple SSUB substrate definitions are present, you must specify which to use.

Implementation Details

This circuit component models an optimally mitered 90° bend in the Stripline center conductor. This circuit component is modeled by a section of straight section of transmission line of characteristic impedance equal to that of a strip of width W. The model includes the losses associated with the conductor and substrate. The formula below should be used to determine the value of the dimension "a" of the chamfer. Note, the reference planes for the two port changes depending upon the value of W/a. The parameter W1 (Strip Width) is a dimension entered in the default length units. The parameter SSUB specifies the Stripline Substrate element, which defines additional cross sectional parameters of the transmission line.

Use Reference Plane (a) if $a/W \leq 1.0$

Use Reference Plane (b) if $a/W \geq 1.0$

If $W/B \leq 0.2$ ($B \equiv$ Ground Plane Spacing)

$$\frac{a}{W} = 1.267 - 0.35 \cdot (\frac{W}{B} - 0.2)$$

if 0.2≤W/B≤1.6

$$\frac{a}{W} = 1.012 + [1.6 - \frac{W}{B}] \cdot (0.08 + [1.6 - \frac{W}{B}] \cdot [0.013 + 0.043 \cdot [1.6 - \frac{W}{B}]])$$

if if 1.6≤W/B≤14.25

$$\frac{a}{W} = 0.884 + 0.08 \cdot [3.2 - \frac{W}{B}]$$

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See “Cell Options Dialog Box: Layout Tab ” for Cell Options dialog box **Layout** tab details.

See “The Layout Process File (LPF)” for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

Restrictions

W/B≤14.25

T/B≤0.1 recommended

Recommendations for Use

Discontinuity models function most accurately when attached to lines that match their corresponding edges. Directly connecting discontinuity models to one another reduces their accuracy.

References

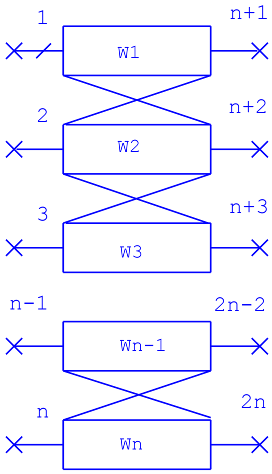
[1] G. Matthaei, L. Young, E.M.T. Jones "Microwave Filters, Impedance-Matching

[2] Networks and Coupling Structures", Artech House, Inc. 1980 pp 203,206.

[3] Harlan Howe, Jr. "Stripline Circuit Design", Artech House, Inc. 1982

Multiple Edge Coupled Striplines (EM Quasi-Static): SNCLIN

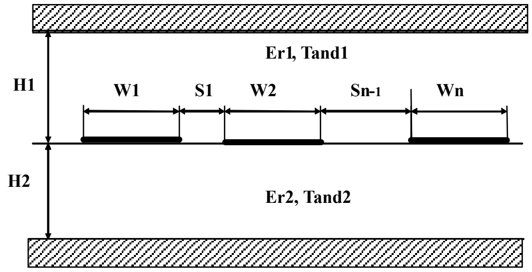
Symbol



Summary

SNCLIN models a section of several (1 to 50) edge coupled/single microstrip lines arranged on a single-layer substrate. A backing ground plane is always present. SNCLIN implements the same modeling techniques as the S2CLIN,...S16CLIN models. In addition, SNCLIN is a dynamic or scalable model; it accepts a number of lines as input parameters so the model and its schematic symbol expands/shrinks as the number of lines increases/decreases. This model uses disk cache to speed up simulation.

Topology



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	TL1
N	Number of conductors		2
L	Conductor length	Length	L[a]
Acc	Accuracy parameter		1
SSUBL	Substrate Definition	Text	SSUBL1[a]
*SaveToFile	"Save to txt file"=Yes/No		"No"

Name	Description	Unit Type	Default
*FileName	Name of text file with computed model parameters	String	Same as model name
Wi, i=1..nn-number of lines	Width of conductor No i	Length	W[b]
Si, i=1..n-1n-number of lines	Spacing between conductors No i and No i+1	Length	W[b]

[a]Modify only if the schematic contains multiple substrates. See "Using Elements With Model Blocks" for details.

[b]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See Microwave Office Layout Guide for details.

** indicates a secondary parameter*

Parameter Details

See SXCLIN for a detailed description of SNCLIN parameters. SNCLIN differs from SXCLIN in parameter order only: For SXCLIN models, the L, Acc, and MSUB parameters follow the widths (W*i*), and spacings (S*i*); for SNCLIN, the L, Acc, MSUB, Save to File, and FileName parameters precede the W*i and* S*i* parameters.

SaveToFile. This parameter is hidden by default and set to No. You can toggle the parameter to Yes or No. If set to Yes, the model creates a text file (named *SNCLIN.txt*) at the current project location. This text file contains a table of values of per-unit-length RLGC line parameters at each project frequency. Each row contains RLGC values computed at the frequency specified in the first column (frequency in GHz, R in ohms/m, L in H/m, G in S/m, C in F/m).

The structure of this text file is essentially the same for n>2. All models output RLGC matrices to columns that immediately follow the frequency column. Entries of each matrix are placed column-wise; the first column is first: R11, R21, R31.. Rn1; and then the second column: R12, R22, R32,..Rn2 and so on. The total number of columns in the file is 4*n*n+1, where n is the number of lines.

If N=2, SNCLIN implies the existence of two modes, namely, C and P (see [2]) and places additional columns in the text file. Note that if a system of coupled lines is fully symmetrical (as might be the case with edge-coupled microstrip lines) C-mode corresponds to even mode and P-mode corresponds to odd mode.

When N=2, SNCLIN outputs complex characteristic impedances Re(Zc1), Im(Zc1), Re(Zp1), Im(Zp1), Re(Zc2), Im(Zc2), Re(Zp2), and Im(Zp2); complex effective dielectric constants Re(EeffC), Im(EeffC), Re(EeffP), and Im(EeffP) (see traditional effective dielectric constants in columns 38, 39); and losses for C and P modes LossC (dB/m), LossP(dB/m) to columns 2-15. Entries of R, L, G, and C matrices are distributed among columns 16-31 in corresponding order (see above). Columns 32-37 contain Re(Rc), Im(Rc), Re(Rp), Im(Rp), BetaC, and BetaP; where Rc is the ratio of C-mode voltage in the second line to C-mode voltage in the first line; Rp is the same ratio in P-mode (see details in [2], section 4.3.1) and BetaC and BetaP are propagation constants of C- and P-modes in Rad/m. Note that columns 38 and 39 contain traditional effective dielectric constants ErC_Eff, ErP_Eff (they do not account for losses). The total number of columns in the file is 39 (at N=2).

If N=1, SNCLIN creates a file that contains complex characteristic impedance Re(Zo) and Im(Zo), complex effective dielectric constant Re(Eeff) and Im (Eff) (see traditional effective dielectric constant in column 12), and Loss (dB/m) in columns 2-6. Columns 7-10 contain R, L, G, and C. Column 11 contains propagation constant Beta in Rad/m. Note that column 12 contains traditional effective dielectric constant Er_Eff that does not account for loss. The total number of columns in the file is 12 (at N=1).

The created text file might be linked or imported to a project as a data file and you can view the frequency behavior of any above mentioned parameter using the proper data measurement. Note that the first column (frequency) is always in

GHz so these measurements might be incompatible with other Microwave Office measurements placed on the same graph; you may prefer to place these data measurements on a separate dedicated graph.

FileName. By default this parameter is hidden and is set to model *SNCLIN.txt*. You can change the file name for each model instance to an arbitrary name with a length not exceeding 64 symbols.

Parameter Restrictions and Recommendations

1. The number of conductors N cannot exceed 50.
2. For more information about restrictions and recommendations common to SNCLIN and SXCLIN, see SXCLIN.

Implementation Details

Model implementation is based on the EM Quasi-Static technique described in [1] [15–56]. It computes matrices of per-unit-length RLGC parameters and uses them to evaluate circuit parameters of coupled lines. This model saves frequency-independent RLGC matrices to a disk cache. Before calculation of RLGC matrix models, it checks to see if the disk cache contains data that has been saved earlier with the same set of input parameters. If a match is found, the model reads RLGC matrices from the disk cache and saves time on their calculation. If no match is found, this model calculates RLGC matrices and places a new record into the disk cache. All subsequent runs of any project containing this model with the identical set of input parameters uses the disk cache for increased speed.

This model accounts for losses in metal and in substrate dielectric. Dispersion is partly included (in frequency dependence of output parameters due to the presence of frequency-dependent losses).

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See “Cell Options Dialog Box: Layout Tab ” for Cell Options dialog box **Layout** tab details.

See “The Layout Process File (LPF)” for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

Recommendations for Use

See SXCLIN for details.

NOTE: The implementation of EM Quasi-Static models relies heavily on the involved numerical algorithms. This may lead to a noticeable increase in simulation time for schematics that employ many such models.

If a layer thickness is too small compared to the thickness of another layer, simulation time may also noticeably increase.

Caution regarding units of data in saved text files: If a project that reads saved text files uses frequency, resistance, inductance or conductance units different from GHz, ohm, henry or siemens, you may need to scale input values manually.

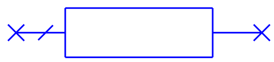
References

[1] M.B. Bazdar, A.R. Djordjevic, R.F. Harrington, and T.K. Sarkar, "Evaluation of quasi-static matrix parameters for multiconductor transmission lines using Galerkin's method," IEEE Trans. Microwave Theory Tech., vol. MTT-42, July 1994, pp. 1223-1228.

[2] R. Mongia, I. Bahl, and P. Bhartia, RF and Microwave Coupled-Line Circuits, Artech House, Norwood, MA, 1999.

Suspended Substrate Stripline (EM Quasi-Static): SS1LIN

Symbol



Summary

SS1LIN models section of single stripline with a conductor strip placed on the upper surface of a suspended substrate (suspended substrate is a single layer substrate elevated over the infinite grounded plane).

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	TL1
W	Top Conductor width	Length	W[a]
L	Conductor length	Length	L[a]
Acc	Accuracy parameter		1
SPSUB	Substrate Definition	Text	SPSUB1[b]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See the Microwave Office Layout Guide for details.

[b] Modify only if schematic contains multiple substrates. See "Using Elements With Model Blocks" for details.

Parameter Details

SPSUB. Suspended substrate parameters are listed in SPSUB model description.

Acc. The parameter Acc is the accuracy parameter. The default value for Acc is 1. If Acc is less than 1 or greater than 10 it is set automatically to 2.

Parameter Restrictions and Recommendations

1. Accuracy parameter A is limited to $1 \le Acc \le 10$. Larger value of Acc increases density of mesh used in computations. Accuracy of model parameters may gain slightly from increasing Acc at the expense of noticeable growth of computation time. As a rule of thumb, good trade-off between accuracy and computation time is to set Acc to 1.
2. This component does not impose restrictions on the conductor thickness (thickness may be zero, positive, or negative). Negative thickness means that the conductor is recessed into the substrate.

Implementation Details

Model implementation is based on the EM Quasi-Static technique described in [1]. It accounts for losses in metal and in substrate dielectric. Dispersion is partly included.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.

2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See "Cell Options Dialog Box: Layout Tab " for Cell Options dialog box **Layout** tab details.

See "The Layout Process File (LPF)" for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

Recommendations for Use

NOTE: The implementation of EM Quasi-Static models relies heavily on the involved numerical algorithms. This may lead to a noticeable increase in simulation time for schematics that employ many such models.

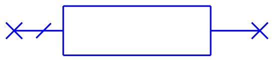
Making the thickness of any layer too small in comparison with other layers can increase the calculation time

References

[1] M.B. Bazdar, A.R. Djordjevic, R.F. Harrington, and T.K. Sarkar, "Evaluation of quasi-static matrix parameters for multiconductor transmission lines using Galerkin's method," IEEE Trans. Microwave Theory Tech., vol. MTT-42, July 1994, pp. 1223-1228.

Unbalanced Single Stripline on Suspended Substrate (EM Based): SS1LINX

Symbol



Summary

SS1LINX models a section of single stripline with a conductor strip placed on the upper surface of a suspended substrate (suspended substrate is a single layer substrate sandwiched between two infinite grounded planes and separated from them with air layers).

This model is constructed as an X-model (table-based interpolation) using the results of a EM 2-D quasi-static cross-sectional analysis based on Method of Moments. For a more detailed discussion of the X-models see " EM-based Models (X-models) " . EM 2-D quasi-static analysis is the same method as that used in the SS1LIN, however, SS1LINX gains large computational speed increases due to the table-based interpolation.

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	TL1
W	Conductor width	Length	W[a]
L	Line length	Length	L[a]
Acc	Accuracy parameter		1
SPSUB	Substrate definition	Text	See[ab]
*AutoFill	Autofill database if not equal to 0		0

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory.

[b] If only one SPSUB is present in the schematic, this substrate is automatically used. If multiple SPSUB substrate definitions are present, you must specify which to use.

** indicates a secondary parameter*

W. Conductor width is an independent parameter.

Acc. The input parameter Acc is an accuracy parameter influencing the EM 2-D quasi-static cross-sectional analysis, and can range from 1 to 10. This value is used to automatically fill the model tables if the necessary table is missing and the Autofill parameter is set to 1. Higher values of Acc may improve accuracy but also may slow the filling process. SS1LINX **does not use Acc in a normal, table-based interpolation mode.**

SPSUB. Suspended substrate parameters are listed in the SPSUB model documentation.

Autofill. A hidden input which allows you to specify that the entire interpolation table should be filled automatically at the current values. To instigate this filling process, this parameter should be set equal to 1. During normal operation, this parameter should be set equal to zero. You can access the hidden parameter by double- clicking on the schematic element.

Parameter Restrictions and Recommendations

1. SS1LINX implies that the ratio W/H2 (H2 is substrate thickness) lies within a predefined range of $0.05 \le W/H2 \le 18.5$. Outside of this range, a model extrapolates output parameters and issues a warning.

2. Cover height above the substrate H1, substrate thickness H2, substrate elevation above ground H3, nominal dielectric constant ErNom, dielectric tangent Tand, relative conductor bulk resistance Rho, and accuracy Acc are fixed parameters. Microwave Office provides pre-generated tables for several typical values of SS1LINX fixed parameters. Changes to any fixed parameter start the automatic filling process (if AutoFill is set to 1), the length of which depends on the Acc value.
3. You can change any fixed parameter to create corresponding tables.
4. The dielectric constant Er of the substrate SPSUB is a statistical parameter. It means that models account for the relative deviation of Er from ErNom within 20%; larger deviation demands a new fill of model tables.

Implementation Details

This model was developed under research performed at NI AWR Corporation. The full set of details of the implementation are considered proprietary in nature.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See “Cell Options Dialog Box: Layout Tab ” for Cell Options dialog box **Layout** tab details.

See “The Layout Process File (LPF)” for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

Recommendations for Use

To create new tables for a substrate dielectric constant different from those supplied with Microwave Office, you need to set ErNom = Er = needed-value-of-dielectric-constant, set AutoFill = 1 and simulate.

In exchange for a speed increase, small errors resulting from the interpolation should be expected and the range of the input parameters is restricted.

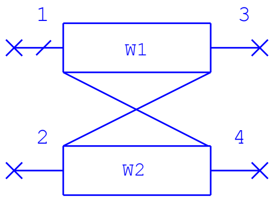
If you wish to implement values outside of a specified range of W/H2 (see the *Parameter Restrictions and Recommendations* section), you can use the SS1LIN element.

References

[1] M.B. Bazdar, A.R. Djordjevic, R.F. Harrington, and T.K. Sarkar, "Evaluation of quasi-static matrix parameters for multiconductor transmission lines using Galerkin's method," IEEE Trans. Microwave Theory Tech., vol. MTT-42, July 1994, pp. 1223-1228

2 Suspended Substrate Edge Coupled Striplines (EM Quasi-Static): SS2CLIN

Symbol



Summary

SS2CLIN models a section of two-edge coupled asymmetric stripline on a suspended substrate, that is, on a single layer substrate elevated over the infinite grounded plane.

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	TL1
W1	Conductor 1 Width- Nodes 1&&3	Length	W[a]
W2	Conductor 2 Width- Nodes 2&&4	Length	W[a]
S	Gap width conductors 1&&2	Length	W[a]
L	Conductor length	Length	L[a]
Acc	Accuracy parameter		1
SPSUB	Substrate Definition	Text	SPSUB1[b]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See the Microwave Office Layout Guide for details.

[b]Modify only if schematic contains multiple substrates. See "Using Elements With Model Blocks" for details.

Parameter Details

SPSUB. Suspended substrate parameters are listed in the SPSUB model description.

Acc. The Acc parameter is the accuracy parameter. The default value for Acc is 1. If Acc is less than 1 or greater than 10 it is set automatically to 2.

Parameter Restrictions and Recommendations

1. Acc is limited to 1≤Acc≤10. A larger value of Acc increases the density of mesh used in computations. The accuracy of model parameters may gain slightly from increasing Acc at the expense of noticeable growth of computation time. Generally, a good trade-off between accuracy and computation time is to set Acc to 1.
2. This component does not impose restrictions on the conductor thickness (thickness may be zero, positive, or negative). Negative thickness indicates that the conductor is recessed into the substrate.

Implementation Details

Model implementation is based on the EM Quasi-Static technique described in [1]. It accounts for losses in metal and in substrate dielectric. Dispersion is partly included.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See "Cell Options Dialog Box: Layout Tab " for Cell Options dialog box **Layout** tab details.

See "The Layout Process File (LPF)" for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

Recommendations for Use

NOTE: The implementation of EM Quasi-Static models relies heavily on the involved numerical algorithms. This may lead to a noticeable increase in simulation time for schematics that employ many such models.

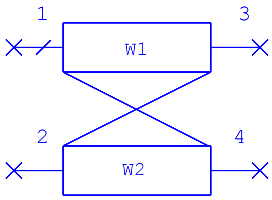
If the thickness of any layer is too small in comparison with the thickness of another layer, simulation time may also noticeably grow.

References

[1] M.B. Bazdar, A.R. Djordjevic, R.F. Harrington, and T.K. Sarkar, "Evaluation of quasi-static matrix parameters for multiconductor transmission lines using Galerkin's method," IEEE Trans. Microwave Theory Tech., vol. MTT-42, July 1994, pp. 1223-1228.

2 Suspended Substrate Edge Coupled Symmetric Striplines (EM Quasi-Static Based): SS2CLINX

Symbol



Summary

SS2CLINX models a section of two **symmetric** edge coupled striplines with conductor strips placed on the upper surface of a suspended substrate (suspended substrate is a single layer substrate sandwiched between two infinite grounded planes and separated from them with air layers).

This model is constructed as an X-model (table-based interpolation) using the results of an EM 2-D quasi-static cross-sectional analysis based on Method of Moments. For a more detailed discussion of the X-models see “ EM-based Models (X-models) ” . EM 2-D quasi-static analysis is the same method as that used in the SS2CLIN model, however, SS2CLINX gains large computational speed increases due to the table-based interpolation.

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	TL1
W	Conductor width	Length	W[a]
S	Spacing between conductors	Length	W[a]
L	Line length	Length	L[a]
Acc	Accuracy parameter		1
SPSUB	Substrate definition	Text	See[b]
*AutoFill	Autofill database if not equal to 0		0

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory.

[b]If only one SPSUB is present in the schematic, this substrate is automatically used. If multiple SPSUB substrate definitions are present, you must specify which to use.

** indicates a secondary parameter*

Parameter Details

W. Conductor width is an independent parameter.

S. Spacing between conductors is an independent parameter.

Acc. The input parameter Acc is an accuracy parameter influencing the EM 2-D quasi-static cross-sectional analysis, and can range from 1 to 10. This value is used to automatically fill the model tables if the necessary table is missing and the Autofill parameter is set to 1. Higher values of Acc may improve accuracy but also may slow the filling process. SS2CLINX **does not use Acc in normal, table-based interpolation mode.**

SPSUB. Suspended substrate parameters are listed in the SPSUB model documentation.

Autofill. The AutoFill parameter is a hidden input which allows you to specify that the entire interpolation table should be filled automatically at the current values. To instigate this filling process, this parameter should be set equal to 1. During normal operation, this parameter should be set equal to zero. You can access the hidden parameter by double-clicking on the schematic element.

Parameter Restrictions and Recommendations

1. SS2CLINX implies that the ratio W/H2 lies within a predefined range of $0.05 \leq W/H2 \leq 18.0$ and the ratio S/H2 lies within a predefined range of $0.025 \leq S/H2 \leq 70$ (H2 is substrate thickness) Outside of these ranges, this model extrapolates output parameters and issues a warning.
2. Cover height above substrate H1, substrate thickness H2, substrate elevation above ground H3, nominal dielectric constant ErNom, dielectric tangent Tand, relative conductor bulk resistance Rho, and accuracy Acc are fixed parameters. Microwave Office provides pre-generated tables for several typical values of SS2CLINX fixed parameters. Changes to any fixed parameter may start the automatic filling process (if Autofill is set to 1), the length of which depends on the value of Acc. You can change any fixed parameter to create corresponding tables.
3. The dielectric constant Er of the substrate SPSUB is a statistical parameter. It means that models account for the relative deviation of Er from ErNom within 20%; larger deviation demands a new fill of model tables.

Implementation Details

This model was developed under research performed at NI AWR Corporation. The full set of details of the implementation are considered proprietary in nature.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See "Cell Options Dialog Box: Layout Tab " for Cell Options dialog box **Layout** tab details.

See "The Layout Process File (LPF)" for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

Recommendations for Use

To create new tables for the substrate dielectric constant different from those supplied with Microwave Office, you must set ErNom = Er = needed-value-of-dielectric-constant, set AutoFill = 1 and simulate.

In exchange for speed increase, small errors resulting from the interpolation should be expected and the range of the input parameters is restricted.

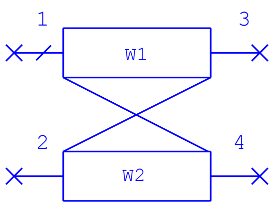
To implement values outside of the specified ranges of W/H2, S/H2 (see "Parameter Restrictions and Recommendations"), you can use the SS2CLIN element.

References

[1] M.B. Bazdar, A.R. Djordjevic, R.F. Harrington, and T.K. Sarkar, "Evaluation of quasi-static matrix parameters for multiconductor transmission lines using Galerkin's method," IEEE Trans. Microwave Theory Tech., vol. MTT-42, July 1994, pp. 1223-1228

2 Suspended Substrate Broadside Coupled Striplines (EM Quasi-Static Based): SSBCPLX

Symbol

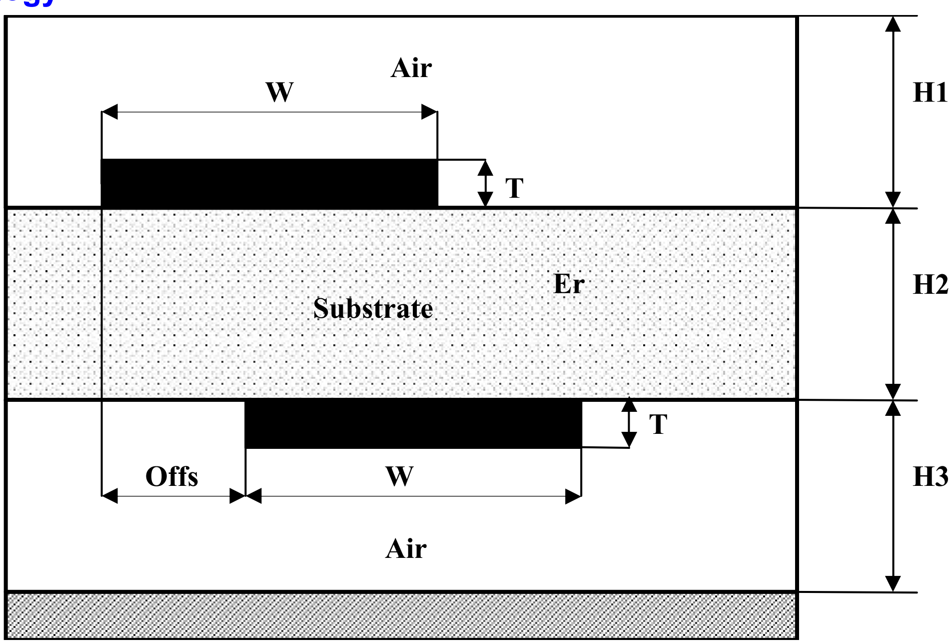


Summary

SSBCPLX models a section of two **equal width and thickness** broadside coupled striplines with conductor strips placed on the top and bottom surfaces of a suspended substrate (suspended substrate is a single layer substrate sandwiched between two infinite grounded planes and separated from them with air layers). Lateral offset is allowed for the bottom line.

SSBCPLX is constructed as an X-model (table-based interpolation) using the results of an EM 2-D quasi-static cross-sectional analysis based on Method of Moments. For a detailed discussion of the X-models see " EM-based Models (X-models) ". EM 2-D quasi-static analysis is the same method as that used in SBCPL, however, SSBCPLX gains large computational speed increases due to the table-based interpolation.

Topology



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	TL1

Name	Description	Unit Type	Default
W	Conductors width	Length	W[a]
Offset	Offset of bottom conductors	Length	W[a]
L	Line length	Length	L[a]
Acc	Accuracy parameter		1
SPSUB	Substrate definition	Text	See[ab]
*AutoFill	Autofill database if not equal to 0		0
SNAME1	Structure name from lpf file for top conductor	Text	TOP_BCLIN
SNAME2	Structure name from lpf file for bottom conductor	Text	BOT_BCLIN

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory.

[b]If only one SPSUB is present in the schematic, this substrate is automatically used. If multiple SPSUB substrate definitions are present, you must specify which to use.

** indicates a secondary parameter*

Parameter Details

W. Conductor width is an independent parameter.

Offs. The offset of the bottom conductor is an independent parameter.

Acc. An accuracy parameter influencing the EM 2-D quasi-static cross-sectional analysis; it can range from 1 to 10. This value is used to automatically fill the model tables if the necessary table is missing and the Autofill parameter is set to 1. Higher values of Acc may improve accuracy but also may slow the filling process. SSBCPLX **does not use Acc in normal, table-based interpolation mode.**

SPSUB. Suspended substrate parameters are listed in the SPSUB model documentation.

Autofill. A hidden input which allows you to specify that the entire interpolation table should be filled automatically at the current values. To initiate this filling process, this parameter should be set to 1. During normal operation, this parameter should be set to zero. You can access the hidden parameter by double-clicking the schematic element.

Parameter Restrictions and Recommendations

1. SSBCPLX implies that the ratio W/H2 lies within a **predefined** range of $0.05 \le W/H2 \le 18.0$ and the ratio Offs/H2 lies within a **predefined** range of $0.025 \le Offs/H2 \le 70$ (H2 is substrate thickness). Outside of these ranges, this model extrapolates output parameters and issues a warning.
2. Cover height above substrate H1, substrate thickness H2, substrate elevation above ground H3, nominal dielectric constant ErNom, dielectric tangent Tand, relative conductor bulk resistance Rho, and accuracy Acc are fixed parameters. MWO provides pre-generated tables for several typical values of SSBCPLX fixed parameters. Changes to any fixed parameter may start the automatic filling process (if Autofill is set to 1), the length of which depends on the Acc value. You can change any fixed parameter to create the corresponding tables.
3. The dielectric constant Er of the substrate SPSUB is a statistical parameter. This means that models account for the relative deviation of Er from ErNom within 20%; larger deviation demands a new fill of the model tables.
4. Conductor thickness is specified by substrate SPSUB. Note that the bottom of the conductor juts out downward (see the "Topology" section).
5. SNAME1 and SNAME2 are for layout only and have no effect on the electrical performance.

Implementation Details

This model was developed under research performed at NI AWR Corp. The full details of the implementation are considered proprietary in nature.

Layout

When you specify SNAME1 and SNAME2, the structures with corresponding names are identified in a $STRUCT_TYPE_BEGIN section of the LPF file. If a structure with the corresponding name is not found, the name of the missing structure is drawn on the error layer. The structure named SNAME1 must contain a text line starting with the line type name used for the top conductor. A structure named SNAME2 must contain a text line starting with the line type name used for the bottom conductor. In a structure named SNAME1=TOP_TRACE and SNAME2= BOT_TRACE with line type names "Top Copper" and "Bot Copper", you need to add the following code to the LPF file (user defined line types may contain any number of layers; structures must contain only one text line):

```
$LINE_TYPE_BEGIN "Top Copper"
  ! -> Layer   offset   minWidth   flags
  "Cu_1"  0  5e-005  0
$LINE_TYPE_END
$LINE_TYPE_BEGIN "Bot Copper"
  ! -> Layer   offset   minWidth   flags
  "Cu_2"  0  5e-005  0
$LINE_TYPE_END
$STRUCT_TYPE_BEGIN "TOP_TRACE"
   "Top Copper" 0 0 0
$STRUCT_TYPE_END
$STRUCT_TYPE_BEGIN "BOT_TRACE"
   "Bot Copper" 0 0 0
$STRUCT_TYPE_END
```

Note that the text inside structures contains line type names in quotation marks followed by three blank separated zeros.

Recommendations for Use

To create new tables for the substrate dielectric constant different from those supplied with MWO, you must set ErNom = Er = needed-value-of-dielectric-constant, set AutoFill = 1 and simulate.

In exchange for a speed increase, you should expect small errors resulting from the interpolation, and the range of the input parameters is restricted.

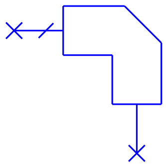
To implement values outside of the specified ranges of W/H2, Offs/H2 (see the "Parameter Restrictions and Recommendations" section), you can use the SBCPL element.

References

[1] M.B. Bazdar, A.R. Djordjevic, R.F. Harrington, and T.K. Sarkar, "Evaluation of quasi-static matrix parameters for multiconductor transmission lines using Galerkin's method," IEEE Trans. Microwave Theory Tech., vol. MTT-42, July 1994, pp. 1223-1228

Suspended Substrate Stripline Bend, Mitered 90deg (EM Based): SSBND90X

Symbol



Summary

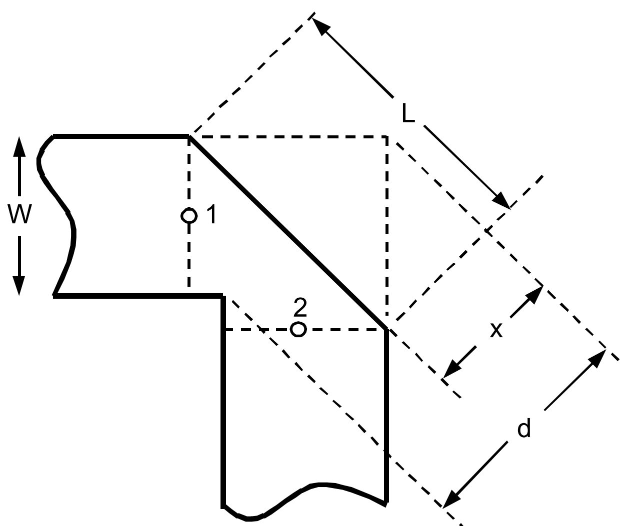
SSBND90X models a mitered bend of stripline placed on the top surface of a suspended substrate (suspended substrate is a single layer substrate sandwiched between two infinite grounded planes and separated from them with air layers).

This model is constructed as an X-model (table-based interpolation) using the results of a EM analysis based on Method of Moments. For a more detailed discussion of the X-models see " EM-based Models (X-models) " .

This model does not include effects of dielectric/conductor/radiation losses; it also implies that conductor thickness is zero.

SSBND90X$ is the corresponding intelligent cell (iCell). An iCell model is identical to its non-iCell equivalent with the following exception: Certain dimension-related parameters are not explicitly user-specified; rather, they are automatically and dynamically determined by the dimensions of the attached elements. See "Intelligent Cells (iCells)" for a detailed discussion of how to use iCells, their advantages, and their limitations.

Topology



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	SS1
W	Conductor Width	Length	W[a]

Name	Description	Unit Type	Default
M	Miter Fraction (0-0.9)		0
SPSUB	Substrate definition	Text	See[bc]
*AutoFill	AutoFill dataBase if not equal to 0		0

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory.

[b] If only one SPSUB is present in the schematic, this substrate is automatically used. If multiple SPSUB substrate definitions are present, you must specify which to use.

[c]Modify only if the schematic contains multiple substrates.

** indicates a secondary parameter*

Parameter Details

W. Conductor width is an independent parameter.

M. Parameter M (miter) is an independent parameter. M is defined as (see the "Topology" section)

$$M = \frac{x}{d}$$

. Other relations for right-mitered bend are:

$$\mathrm{M} = \frac{x}{d}$$

$$d = W\sqrt{2}$$

;

$$L = 2x$$

;

$$d - x = \sqrt{2} \cdot W \cdot (1 - M)$$

SPSUB. Suspended substrate parameters are listed in the SPSUB model documentation.

Autofill. The AutoFill parameter is a hidden input which allows you to specify that the entire interpolation table should be filled automatically at the current values. To instigate this filling process, this parameter should be set equal to 1. During normal operation, this parameter should be set equal to zero. You can access the hidden parameter by double-clicking on the schematic element.

Parameter Restrictions and Recommendations

1. SSBND90X implies that the ratio W/H2 lies within a predefined range of $0.25 \le W/H2 \le 16$ and M lies within a predefined range of $0 \le M \le 0.9125$ (H2 is substrate thickness). Outside of these ranges, this model extrapolates output parameters and issues a warning.
2. This bend is modeled as a two-port discontinuity. Reference planes corresponding to ports are located as shown in the *Topology* section.
3. Cover height above substrate H1, substrate thickness H2, substrate elevation above ground H3, nominal dielectric constant ErNom, dielectric tangent Tand, and relative conductor bulk resistance Rho are fixed parameters. Microwave

Office provides pre-generated (pre-filled) tables for several typical values of SSBND90X fixed parameters. Changes to any fixed parameter may start the automatic filling process if Autofill is set to 1. However, changes to Tand and Rho do not affect the results and do not instigate table filling.

4. You can change any fixed parameter to create corresponding tables.
5. The dielectric constant Er of the substrate SPSUB is a statistical parameter. It means that models account for the relative deviation of Er from ErNom within 20%; larger deviation demands a new fill of model tables.
6. The frequency limits of this model are dynamic with respect to the dimensions of the bend as discontinuity, to the line width. This dynamic frequency limit is displayed via warning messages for the relative size, and dielectric in use. Note that this recommended frequency limit changes as a function of the largest width in the discontinuity for a given substrate definition. The frequency limit warns you that at least one of the transmission lines constructing the discontinuity is approaching the frequency limit where multiple modes can propagate. See “Upper Frequency Limitations” for more information.

Implementation Details

This model was developed under research performed at NI AWR Corporation. The full set of details of the implementation are considered proprietary in nature.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See “Cell Options Dialog Box: Layout Tab ” for Cell Options dialog box **Layout** tab details.

See “The Layout Process File (LPF)” for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

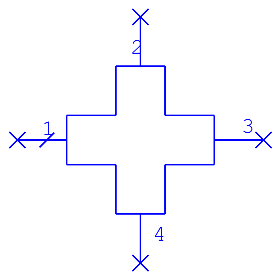
Recommendations for Use

To create new tables for the substrate dielectric constant different from those supplied with Microwave Office, you must set ErNom = Er = needed-value-of-dielectric-constant, set AutoFill = 1 and simulate. Please allow several hours (the actual time depends on your computer capabilities) for generating tables.

In exchange for a speed increase, small errors resulting from the interpolation should be expected and the range of the input parameters is restricted.

Suspended Substrate Stripline Cross - Junction Bend (EM Based): SSCROSSX

Symbol



Summary

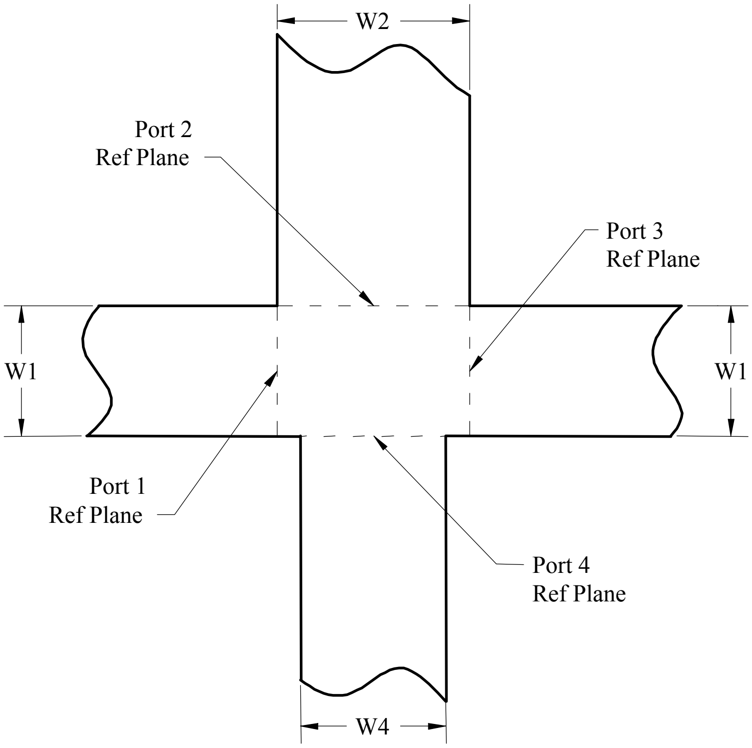
SSCROSSX models the cross junction of striplines lines placed on the top surface of a suspended substrate (suspended substrate is a single layer substrate sandwiched between two infinite grounded planes and separated from them with air layers).

This model is constructed as an X-model (table-based interpolation) using the results of EM analysis based on Method of Moments. For a more detailed discussion of the X-models see the " EM-based Models (X-models) ".

This model does not include effects of dielectric/conductor/radiation losses; it also implies that conductor thickness is zero.

SSCROSSX$ is the corresponding intelligent cell (iCell). An iCell model is identical to its non-iCell equivalent with the following exception: Certain dimension-related parameters are not explicitly user-specified; rather, they are automatically and dynamically determined by the dimensions of the attached elements. See "Intelligent Cells (iCells)" for a detailed discussion of how to use iCells, their advantages, and their limitations.

Topology



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	SX1
W1	Conductor Width @ Port 1	Length	W[a]
W2	Conductor Width @ Port 2	Length	W[a]
*W3	Conductor Width @ Port 3 (model always sets W3=W1; layout cell accepts arbitrary W3)	Length	W[a]
W4	Conductor Width @ Port 4	Length	W[a]
SPSUB	Substrate definition	Text	See[bc]
*AutoFill	AutoFill dataBase if not equal to 0		0

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory.

[b] If only one SPSUB is present in the schematic, this substrate is automatically used. If multiple SPSUB substrate definitions are present, you must specify which to use.

[c]Modify only if the schematic contains multiple substrates.

** indicates a secondary parameter*

Parameter Details

W1, W2, W4. Conductor widths are independent parameters.

W3. SSCROSSX does not use this parameter, it always sets W3=W1. However, layout cells read this parameter so it affects the layout.

SPSUB. Suspended substrate parameters are listed in SPSUB model documentation.

Autofill. The AutoFill parameter is a hidden input which allows you to specify that the entire interpolation table should be filled automatically at the current values. To instigate this filling process, this parameter should be set equal to 1. During normal operation, this parameter should be set equal to zero. You can access the hidden parameter by double-clicking on the schematic element.

Parameter Restrictions and Recommendations

1. SSCROSSX implies that the ratio Wwidest/H2 lies within a **predefined** range of $0.25 \le W_{widest}/H2 \le 16$, where Wwidest is the maximum value of W1, W2, and W4 (H2 is substrate thickness); it also implies that the ratio Wothers/Wwidest lies within a **predefined** range of $0.2 \le W_{others}/W_{widest} \le 1$ where Wothers are all values of W1, W2, W4, excluding Wwidest. (Note that W3=W1). Outside of these ranges, this model extrapolates output parameters and issues a warning.
2. Cross is modeled as a four-port discontinuity. Reference planes corresponding to ports are located as shown in the "Topology" section.
3. Cover height above substrate H1, substrate thickness H2, substrate elevation above ground H3, nominal dielectric constant ErNom, dielectric tangent Tand, and relative conductor bulk resistance Rho are fixed parameters. Microwave Office provides pre-generated (pre-filled) tables for several typical values of SSCROSSX fixed parameters. Changes to any fixed parameter may start the automatic filling process if Autofill is set to 1. However, changes to Tand and Rho do not affect results and do not instigate the tables filling.
4. You can change any fixed parameter to create corresponding tables.
5. The dielectric constant Er of the substrate SPSUB is a statistical parameter. It means that models account for the relative deviation of Er from ErNom within 20%; larger deviation demands a new fill of model tables.
6. The frequency limits of this model are dynamic with respect to the dimensions of the bend as discontinuity, to the line width. This dynamic frequency limit is displayed via warning messages for the relative size, and dielectric in use. Note that this recommended frequency limit changes as a function of the largest width in the discontinuity for a given substrate definition. The frequency limit warns you that at least one of the transmission lines constructing the discontinuity is approaching the frequency limit where multiple modes can propagate. See "Upper Frequency Limitations" for more information.

Implementation Details

This model was developed under research performed at NI AWR Corporation. The full set of details of the implementation are considered proprietary in nature.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See "Cell Options Dialog Box: Layout Tab " for Cell Options dialog box **Layout** tab details.

See “The Layout Process File (LPF)” for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

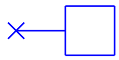
Recommendations for Use

To create new tables for the substrate dielectric constant different from those supplied with Microwave Office, you must set ErNom = Er = needed-value-of-dielectric-constant, set AutoFill = 1, and simulate. Please allow several hours (the actual time depends on your computer capabilities) for generating tables.

In exchange for speed increase, small errors resulting from the interpolation should be expected and the range of the input parameters is restricted.

Suspended Substrate Stripline Open End Effect (EM Based): SSOPENX

Symbol



Summary

SSOPENX models an effect of the open end of stripline placed on the top surface of a suspended substrate (suspended substrate is a single layer substrate sandwiched between two infinite grounded planes and separated from them with air layers). This model should be used in conjunction with any suitable stripline suspended line model.

This model is constructed as an X-model (table-based interpolation) using the results of EM analysis based on Method of Moments. For a more detailed discussion of the X-models see " EM-based Models (X-models) ".

This model does not include effects of dielectric/conductor/radiation losses; it also implies that conductor thickness is zero.

SSOPENX$ is the corresponding intelligent cell (iCell). An iCell model is identical to its non-iCell equivalent with the following exception: Certain dimension-related parameters are not explicitly user-specified; rather, they are automatically and dynamically determined by the dimensions of the attached elements. See "Intelligent Cells (iCells)" for a detailed discussion of how to use iCells, their advantages, and their limitations.

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	SO1
W	Conductor width	Length	W[a]
SPSUB	Substrate definition	Text	See[bc]
*AutoFill	AutoFill dataBase if not equal to 0		0

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory.

[b] If only one SPSUB is present in the schematic, this substrate is automatically used. If multiple SPSUB substrate definitions are present, you must specify which to use.

[c]Modify only if the schematic contains multiple substrates.

** indicates a secondary parameter*

Parameter Details

W. Conductor width is an independent parameter.

SPSUB. Suspended substrate parameters are listed in the SPSUB model documentation.

Autofill. The AutoFill parameter is a hidden input which allows you to specify that the entire interpolation table should be filled automatically at the current values. To instigate this filling process, this parameter should be set equal to 1. During normal operation, this parameter should be set equal to zero. You can access the hidden parameter by double-clicking on the schematic element.

Parameter Restrictions and Recommendations

1. SSOPENX implies that the ratio W/H2 lies within a **predefined** range of 0.05≤W/H2≤16, where H2 is substrate thickness. Outside of these ranges, this model extrapolates output parameters and issues a warning.
2. Open end is modeled as a one-port discontinuity. The reference plane corresponding to the port is located at the edge of the open end of the line you attach to this port; SSOPENX adds a small stretch of suspended stripline.
3. Cover height above substrate H1, substrate thickness H2, substrate elevation above ground H3, nominal dielectric constant ErNom, dielectric tangent Tand, and relative conductor bulk resistance Rho are fixed parameters. Microwave Office provides pre-generated (pre-filled) tables for several typical values of SSOPENX fixed parameters. Changes to any fixed parameter may start the automatic filling process if Autofill is set to 1. However, changes to Tand and Rho do not affect results and do not instigate the tables filling.
4. You can change any fixed parameter to create corresponding tables.
5. The dielectric constant Er of the substrate SPSUB is a statistical parameter. It means that models account for the relative deviation of Er from ErNom within 20%; larger deviation demands a new fill of model tables.
6. The frequency limits of this model are dynamic with respect to the dimensions of the bend as discontinuity, to the line width. This dynamic frequency limit is displayed via warning messages for the relative size, and dielectric in use. Note that this recommended frequency limit changes as a function of the largest width in the discontinuity for a given substrate definition. The frequency limit warns you that at least one of the transmission lines constructing the discontinuity is approaching the frequency limit where multiple modes can propagate. See "Upper Frequency Limitations" for more information.

Implementation Details

This model was developed under research performed at NI AWR Corporation. The full set of details of the implementation are considered proprietary in nature.

Layout

This element does not have an assigned layout cell nor does it use line types to determine its layout. The element models an open end effect.

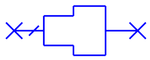
Recommendations for Use

To create new tables for the substrate dielectric constant different from those supplied with MWO, you must set ErNom = Er = needed-value-of-dielectric-constant, set AutoFill = 1 and simulate. Allow several hours (the actual time depends on your computer capabilities) for generating tables.

In exchange for speed increase, small errors resulting from the interpolation should be expected and the range of the input parameters is restricted.

Suspended Substrate Stripline Step In Width with Offset (EM Based): SSSTEPX

Symbol



Summary

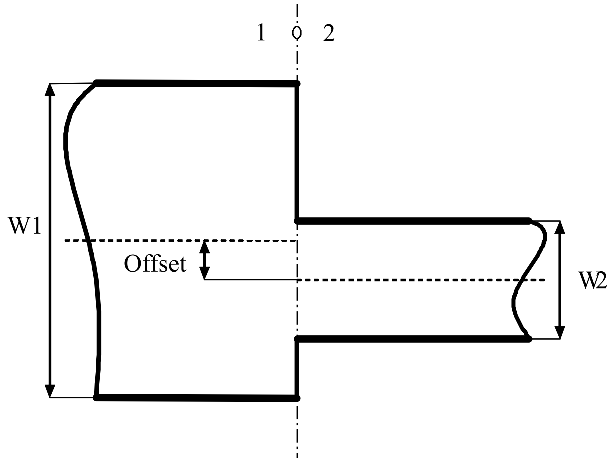
SSSTEPX models a step in width at the asymmetric junction of two striplines placed on the top surface of a suspended substrate (suspended substrate is a single layer substrate sandwiched between the infinite grounded planes and separated from them with air layers). This model allows the relative offset of lines.

This model is constructed as an X-model (table-based interpolation) using the results of a EM analysis based on Method of Moments. For a more detailed discussion of the X-models see the " EM-based Models (X-models) ".

This model does not include the effects of dielectric/conductor/radiation losses; it also implies that conductor thickness is zero.

SSSTEPX$ is the corresponding intelligent cell (iCell). An iCell model is identical to its non-iCell equivalent with the following exception: Certain dimension-related parameters are not explicitly user-specified; rather, they are automatically and dynamically determined by the dimensions of the attached elements. See "Intelligent Cells (iCells)" for a detailed discussion of how to use iCells, their advantages, and their limitations.

Topology



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	SS1
W1	Conductor Width @ Node 1	Length	W[a]
W2	Conductor Width @ Node 2	Length	W[a]
Offset	Centerline Offset Dimension	Length	W[a]
SPSUB	Substrate definition	Text	See[bc]

Name	Description	Unit Type	Default
*AutoFill	AutoFill dataBase if not equal to 0		0

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory.

[b]If only one SPSUB is present in the schematic, this substrate is automatically used. If multiple SPSUB substrate definitions are present, you must specify which to use.

[c]Modify only if the schematic contains multiple substrates.

** indicates a secondary parameter*

Parameter Details

W1, W2. Conductor widths are independent parameters.

Offset. Offset is an independent parameter.

SPSUB. Suspended substrate parameters are listed in the SPSUB model documentation.

Autofill. The AutoFill parameter is a hidden input which allows you to specify that the entire interpolation table should be filled automatically at the current values. To instigate this filling process, this parameter should be set equal to 1. During normal operation, this parameter should be set equal to zero. You can access the hidden parameter by double-clicking on the schematic element.

Parameter Restrictions and Recommendations

1. SSSTEPX implies that the ratio Wmax/H2 lies within a **predefined** range of $0.1 \le W_{max}/H2 \le 16$, the ratio Wmin/Wmax lies within a **predefined** range of $0 \le W_{min}/W_{max} \le 1$, and Offset/Wmax lies within a predefined range of $0 \le Offset/W_{max} \le 0.5$, where H2 is substrate thickness, Wmax is the maximum value of W1, W2; Wmin is the minimum value of W1, W2. Outside of these ranges, this model extrapolates output parameters and issues a warning.
2. Step is modeled as a two-port discontinuity. Reference planes corresponding to ports are located as shown in the "Topology" section.
3. Cover height above substrate H1, substrate thickness H2, substrate elevation above ground H3, nominal dielectric constant ErNom, dielectric tangent Tand, and relative conductor bulk resistance Rho are fixed parameters. Microwave Office provides pre-generated (pre-filled) tables for several typical values of SSSTEPX fixed parameters. Changes to any fixed parameter may start the automatic filling process if Autofill is set to 1. However, changes to Tand and Rho do not affect results and do not instigate the tables filling.
4. You can change any fixed parameter to create corresponding tables.
5. The dielectric constant Er of the substrate SPSUB is a statistical parameter. It means that models account for relative deviation of Er from ErNom within 20%; larger deviation demands a new fill of model tables.
6. The frequency limits of this model are dynamic with respect to the dimensions of the bend as discontinuity, to the line width. This dynamic frequency limit is displayed via warning messages for the relative size, and dielectric in use. Note that this recommended frequency limit changes as a function of the largest width in the discontinuity for a given substrate definition. The frequency limit warns you that at least one of the transmission lines constructing the discontinuity is approaching the frequency limit where multiple modes can propagate. See “Upper Frequency Limitations” for more information.

Implementation Details

This model was developed under research performed at NI AWR Corporation. The full set of details of the implementation are considered proprietary in nature.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See "Cell Options Dialog Box: Layout Tab " for Cell Options dialog box **Layout** tab details.

See "The Layout Process File (LPF)" for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

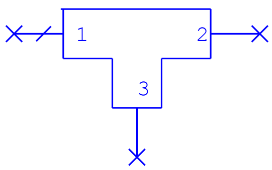
Recommendations for Use

To create new tables for the substrate dielectric constant different from those supplied with Microwave Office, you must set ErNom = Er = needed-value-of-dielectric-constant, set AutoFill = 1 and simulate. Please allow several hours (the actual time depends on your computer capabilities) for generating tables.

In exchange for speed increase, small errors resulting from the interpolation should be expected and the range of the input parameters is restricted.

Suspended Substrate Stripline Tee (EM Based): SSTEEX

Symbol



Summary

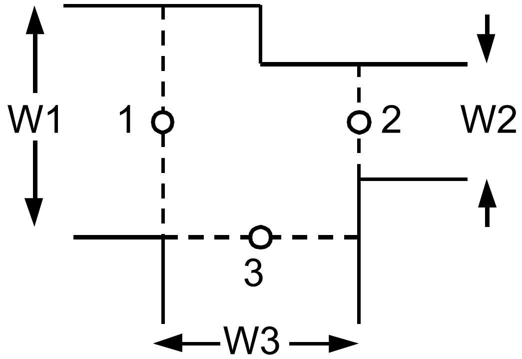
SSTEEX models a T-junction of three striplines placed on the top surface of a suspended substrate (suspended substrate is a single layer substrate sandwiched between two infinite grounded planes and separated from them with air layers).

This model is constructed as an X-model (table-based interpolation) using the results of EM analysis based on Method of Moments. For a more detailed discussion of the X-models see “ EM-based Models (X-models) ”.

This model does not include effects of dielectric/conductor/radiation losses; it also implies that conductor thickness is zero.

SSTEEX$ is the corresponding intelligent cell (iCell). An iCell model is identical to its non-iCell equivalent with the following exception: Certain dimension-related parameters are not explicitly user-specified; rather, they are automatically and dynamically determined by the dimensions of the attached elements. See “Intelligent Cells (iCells)” for a detailed discussion of how to use iCells, their advantages, and their limitations.

Topology



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	ST1
W1	Conductor Width @ Node 1	Length	W[a]
W2	Conductor Width @ Node 2	Length	W[a]
W3	Conductor Width @ Node 3	Length	W[a]
SPSUB	Substrate definition	Text	See [bc]

Name	Description	Unit Type	Default
*AutoFill	AutoFill database if not equal to 0		0

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory.

[b] If only one SPSUB is present in the schematic, this substrate is automatically used. If multiple SPSUB substrate definitions are present, you must specify which to use.

[c]Modify only if the schematic contains multiple substrates.

** indicates a secondary parameter*

Parameter Details

W1, W2, W3. Conductor widths are independent parameters.

SPSUB. Suspended substrate parameters are listed in the SPSUB model documentation.

Autofill. The AutoFill parameter is a hidden input which allows you to specify that the entire interpolation table should be filled automatically at the current values. To instigate this filling process, this parameter should be set equal to 1. During normal operation, this parameter should be set equal to zero. You can access the hidden parameter by double-clicking on the schematic element.

Parameter Restrictions and Recommendations

1. SSTEEX implies that the ratio Wwidest/H2 lies within a **predefined** range of $0.25 \le W_{widest}/H2 \le 16$ where Wwidest is the maximum value of W1, W2, and W3 (H2 is substrate thickness); it also implies that the ratio Wothers/Wwidest lies within a **predefined** range of $0.2 \le W_{others}/W_{widest} \le 1$ where Wothers are all values of W1, W2, W3 excluding Wwidest. Outside of these ranges, this model extrapolates output parameters and issues a warning.
2. Tee is modeled as a three-port discontinuity. Reference planes corresponding to ports are located as shown in the "Topology" section.
3. Cover height above substrate H1, substrate thickness H2, substrate elevation above ground H3, nominal dielectric constant ErNom, dielectric tangent Tand, and relative conductor bulk resistance Rho are fixed parameters. Microwave Office provides pre-generated (pre-filled) tables for a several typical values of SSTEEX fixed parameters. Changes to any fixed parameter may start the automatic filling process if Autofill is set to 1. However, changes to Tand and Rho do not affect results and do not instigate the tables filling.
4. You can change any fixed parameter to create corresponding tables.
5. The dielectric constant Er of the substrate SPSUB is a statistical parameter. It means that models accounts for relative deviation of Er from ErNom within 20%; larger deviation demands a new fill of model tables.
6. The frequency limits of this model are dynamic with respect to the dimensions of the bend as discontinuity, to the line width. This dynamic frequency limit is displayed via warning messages for the relative size, and dielectric in use. Note that this recommended frequency limit changes as a function of the largest width in the discontinuity for a given substrate definition. The frequency limit warns you that at least one of the transmission lines constructing the discontinuity is approaching the frequency limit where multiple modes can propagate. See “Upper Frequency Limitations” for more information.

Implementation Details

This model was developed under research performed at NI AWR Corporation. The full set of details of the implementation are considered proprietary in nature.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See “Cell Options Dialog Box: Layout Tab ” for Cell Options dialog box **Layout** tab details.

See “The Layout Process File (LPF)” for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

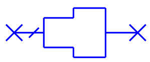
Recommendations for Use

To create new tables for the substrate dielectric constant different from those supplied with Microwave Office, you must set ErNom = Er = needed-value-of-dielectric-constant, set AutoFill = 1 and simulate. You should allow several hours (the actual time depends on your computer capabilities) for generating tables.

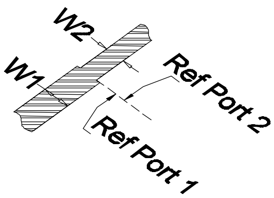
In exchange for speed increase, small errors resulting from the interpolation should be expected and the range of the input parameters is restricted.

Stripline Step in Width (Closed Form): SSTEP

Symbol



Topology



Parameters

SSTEP$ is an iCell. See "Intelligent Cells (iCells)" for an explanation.

Name	Description	Unit Type	Default
ID	Element ID	Text	SL1
W1	Conductor width @ node 1	Length	W[a]
W2	Conductor width @ node 2	Length	W[a]
SSUB	Substrate definition	Text	2[b]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory.

[b]If only one SSUB is present in the schematic, this substrate is automatically used. If multiple SSUB substrate definitions are present, the user must specify.

Implementation Details

This circuit component models a step in the width of the Stripline center conductor. This model includes the reactive components of the step junction analyzed as a zero thickness step of the effective widths of each section. The model does not include any losses associated with this parasitic reactance. The parameters W1 and W2 (Strip Widths) are dimensions entered in the default length units. The parameter SSUB specifies the Stripline Substrate element, which defines additional cross sectional parameters of the transmission line. This model also warns of the possibility of higher order propagating modes if they exist.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See "Cell Options Dialog Box: Layout Tab " for Cell Options dialog box **Layout** tab details.

See “The Layout Process File (LPF)” for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

Restrictions

$0.1 \leq (W2)/(W1) \leq 10$

$Zo > 94.25/(\varepsilon_r)^{1/2}$ for $T \neq 0$ with Zo≡Chracteristic Impedance of Trans Line

$T/B \leq 0.1$ recommended

Recommendations for Use

Discontinuity models function most accurately when attached to lines that match their corresponding edges. Directly connecting discontinuity models to one another reduces their accuracy.

Reference

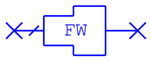
[1] K.C. Gupta "Computer-Aided Design of Microwave Circuits", Artech House1981 p.60

[2] A.A. Oliner, "Equivalent Circuits for Discontinuities in Balanced Strip Transmission Line" IRE Transactions on Microwave Theory and Techniques, March 1955 p.134-143

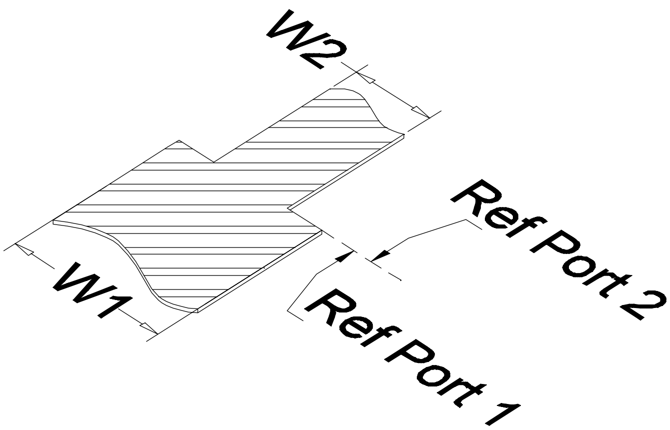
[3] H.M. Altschuler and A.A. Oliner, "Discontinuities in the Center Conductor of Symmetric Strip Transmission Line", IRE Transactions on Microwave Theory and Techniques, May 1960 p.328-339

Stripline Step in Width (Mode Matching): SSTEP2

Symbol



Topology



Parameters

SSTEP2$ is an iCell. See "Intelligent Cells (iCells)" for an explanation.

Name	Description	Unit Type	Default
ID	Element ID	Text	SL1
W1	Conductor width @ node 1	Length	W[a]
W2	Conductor width @ node 2	Length	W[a]
Modes	# Of higher order modes		4
SSUB	Substrate definition	Text	[b]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory.

[b]If only one SSUB is present in the schematic, this substrate is automatically used. If multiple SSUB substrate definitions are present, the user must specify.

Implementation Details

The model assumes a TEM mode of incident and reflected propagating waves and also calculates the stored energy in higher order evanescent waves. The model is based on the Full Wave Solution of an equivalent parallel plate wave guide impedance step and incorporates the frequency-dependent nature of the parasitic inductance and capacitance and further incorporates the effects of dispersion. The model does not incorporate the resistive effects due to radiation or dielectric and conductive losses. The parameters W1 and W2 (Strip Widths) are lengths entered in the default length units. The parameter SSUB specifies the Stripline Substrate element, which defines additional cross sectional parameters of the transmission line. The Modes parameter (Number of Higher Order Even Modes) indicates the number of modes to sum to determine the amount of stored energy in the junction. As the number of modes increase, the accuracy and calculation time increases. Further, as the aspect ratio (W1/W2) deviates away from unity (1) the number of modes required for a given accuracy increases.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See "Cell Options Dialog Box: Layout Tab " for Cell Options dialog box **Layout** tab details.

See "The Layout Process File (LPF)" for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

Restrictions

$Z_o>94.25/(\varepsilon_r)^{1/2}$ (for $T\neq0$)	$\varepsilon_r\leq16.0$ Recommended
$T/B\leq0.1$ Recommended	$1\leq\varepsilon_r$ Required
$1\leq$Modes≤30 Recommended	$0.05\leq W_1/W_{r2}\leq20$ Recommended

Recommendations for Use

Discontinuity models function most accurately when attached to lines that match their corresponding edges. Directly connecting discontinuity models to one another reduces their accuracy.

Reference

[1] Menzel, W. and Wolff, I. "A Method of Calculating the Frequency Dependent Properties of Microstrip Discontinuities" IEEE Trans. On MTT Vol.25 pp.,107-112 February 1977

[2] Chu, T.S., Itoh, T. and Shih, Y.C. "Comparative Study of Mode-Matching Formulations for Microstrip Discontinuity Problems" IEEE Trans. on MTT Vol-33 No.10 Oct. 1985

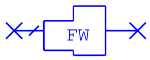
[3] Chu, T.S. and Itoh, T. "Analysis of Microstrip Step Discontinuity by the Modified Residue Calculus Technique" IEEE Trans. on MTT Vol-33 No.10 Oct 1985

[4] Gupta, K.C. "Microstrip Lines and Slotlines" Artech 1996 pp.190

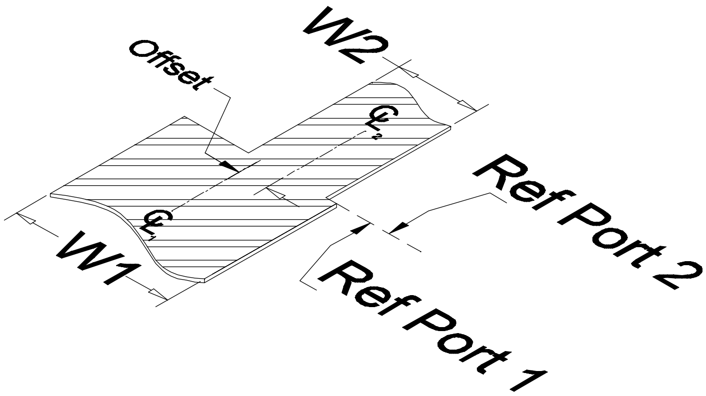
[5] Itoh, Tatsuo, "Numerical Techniques for Microwave and Millimeter-Wave Passive Structures" John Wiley & Sons Inc. 1989 p. 466-472

Stripline Offset Step in Width (Mode Matching): SSTEPO

Symbol



Topology



Parameters

SSTEPO$ is an iCell. See "Intelligent Cells (iCells)" for an explanation.

Name	Description	Unit Type	Default
ID	Element ID	Text	SL1
W1	Conductor width @ node 1	Length	W[a]
W2	Conductor width @ node 2	Length	W[a]
Offset	Centerline offset dimension	Length	W[a]
Modes	# Of higher order modes		8
SSUB	Substrate definition	Text	[b]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory.

[b]If only one SSUB is present in the schematic, this substrate is automatically used. If multiple SSUB substrate definitions are present, the user must specify.

Implementation Details

The model assumes a TEM mode of incident and reflected propagating waves and also calculates the stored energy in higher order evanescent waves. The model is based on the Full Wave Solution of an equivalent parallel plate wave guide impedance step and incorporates the effects of dispersion and the frequency-dependent nature of the parasitic inductance and capacitance. The model does not incorporate the resistive effects due to radiation or dielectric and conductive losses. The parameters W1, W2 (Strip Widths) and Offset are lengths entered in the default length units. The parameter SSUB specifies the Stripline Substrate element, which defines additional cross sectional parameters of the transmission line. The Modes parameter (Number of Higher Order Odd and Even order Modes) indicates the number of modes to sum to determine the amount of stored energy in the junction. The amount of energy stored in the higher modes is increased due to the ability of the asymmetric step to excite odd ordered modes. As the number of modes increases, the accuracy and calculation time increases. Further, as the aspect ratio (W1/W2) deviates away from unity (1) the number of modes required for a given accuracy increases.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See "Cell Options Dialog Box: Layout Tab " for Cell Options dialog box **Layout** tab details.

See "The Layout Process File (LPF)" for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

Restrictions

$Z_o>94.25/(\varepsilon_r)^{1/2}$ (for $T\neq0$)	$\varepsilon_r\leq16.0$ Recommended
$T/B\leq0.1$ Recommended	$1\leq\varepsilon_r$ Required
$1\leq$Modes≤30 Recommended	$0.05\leq W_1/W_{r2}\leq20$ Recommended

Recommendations for Use

Discontinuity models function most accurately when attached to lines that match their corresponding edges. Directly connecting discontinuity models to one another reduces their accuracy.

Reference

[1] Menzel, W. and Wolff, I. "A Method of Calculating the Frequency Dependent Properties of Microstrip Discontinuities" IEEE Trans. On MTT Vol.25 pp.,107-112 February 1977

[2] Chu, T.S., Itoh, T. and Shih, Y.C. "Comparative Study of Mode-Matching Formulations for Microstrip Discontinuity Problems" IEEE Trans. on MTT Vol-33 No.10 Oct. 1985

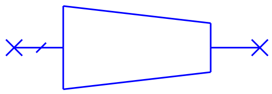
[3] Chu, T.S. and Itoh, T. "Analysis of Microstrip Step Discontinuity by the Modified Residue Calculus Technique" IEEE Trans. on MTT Vol-33 No.10 Oct. 1985

[4] Gupta, K.C. "Microstrip Lines and Slotlines" Artech 1996 pp.190

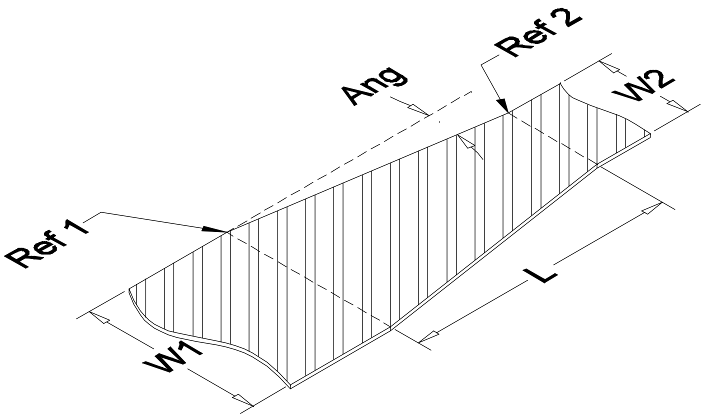
[5] Itoh, Tatsuo, "Numerical Techniques for Microwave and Millimeter-Wave Passive Structures" John Wiley & Sons Inc. 1989 p. 466-472

Tapered Stripline (Closed Form): STAPER

Symbol



Topology



Parameters

STAPER$ is an iCell. See “Intelligent Cells (iCells)” for more information.

Name	Description	Unit Type	Default
ID	Element ID	Text	SL1
W1	Conductor width @ node 1	Length	W[a]
W2	Conductor width @ node 2	Length	W[a]
L	Length of taper	Length	L[a]
SSUB	Substrate definition	Text	SSUB[b]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory.

[b]If only one SSUB is present in the schematic, this substrate is automatically used. If multiple SSUB substrate definitions are present, you must specify.

Implementation Details

This is constructed out of a cascaded series of constant width Stripline transmission lines. The number of sections used is frequency-dependent and is constant as a function of the Length divided by the guided wavelength. The model assumes a TEM mode of propagation and incorporates the effects of dielectric and conductive losses. The parameters W1, W2 (Strip Widths) and L (Strip Length) are lengths entered in the default length units. The parameter SSUB specifies the Stripline Substrate element, which defines additional cross sectional parameters of the transmission line.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See "Cell Options Dialog Box: Layout Tab " for Cell Options dialog box **Layout** tab details.

See "The Layout Process File (LPF)" for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

Restrictions

$Z_o > 94.25/(\varepsilon_r)^{1/2}$ (for $T \neq 0$)	$1 \leq \varepsilon_r$ Required
$T/B \leq 0.1$ Recommended	$Tand \geq 0$ Required
$Rho \leq 100$ Recommended	$Ang \leq 45$ degs or $L \geq \|W1-W2\|/2$

$0.05 \leq W/H \leq 20$ Recommended

$T/W \leq 0.5$ Recommended

$T/H \leq 0.5$ Recommended

$\varepsilon_r \leq 16.0$ Recommended

$1 \leq \varepsilon_r$ Required

$Tand \geq$ Required

$0 \leq Rho \leq 1000$ Required

$Rho \leq 100$ Required

$T/B \leq 0.1$ Required

Recommendations for Use

Discontinuity models function most accurately when attached to lines that match their corresponding edges. Directly connecting discontinuity models to one another reduces their accuracy.

References

[1] NI AWR Corporation Developed Model. The references for the straight section Stripline model are shown below.

[2] H.A. Wheeler, "Transmission-Line Properties of a Strip Line Between Parallel Planes", IEEE Trans on Microwave Theory and Techniques, Vol. MTT-26, No. 11, November 1978 p.866-876

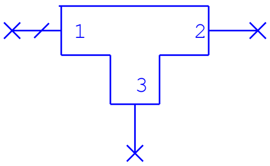
[3] S.B. Cohn, "Characteristic Impedance of the Shielded-Strip Transmission Line", IRE Transactions MTT, July 1954 p. 52-55

[4] H.A. Wheeler, "Formulas for the Skin Effect", Proceedings of the IRE September, 1942 p.412-424

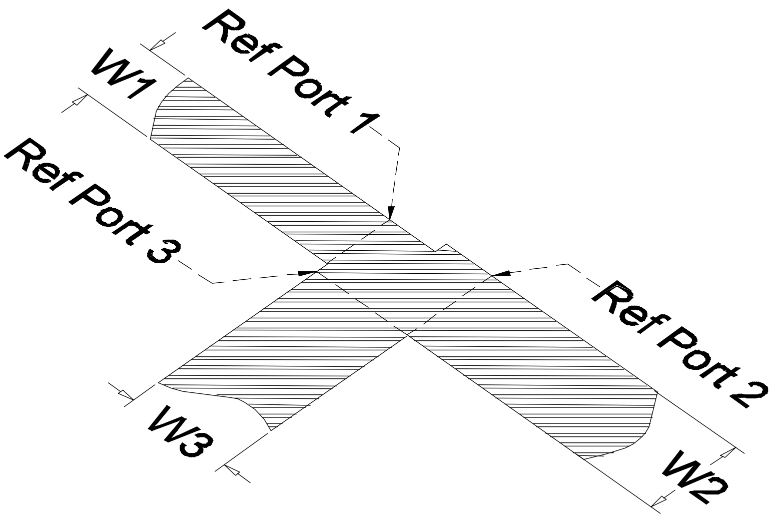
[5] K.C. Gupta "Computer-Aided Design of Microwave Circuits", Artech House1981 p.60

Stripline Tee - Junction (Closed Form): STEE

Symbol



Topology



Parameters

STEE$ is an iCell. See "Intelligent Cells (iCells)" for more information.

Name	Description	Unit Type	Default
ID	Element ID	Text	SL1
W1	Conductor width @ port 1	Length	W[a]
W2	Conductor width @ port 2	Length	W[a]
W3	Conductor width @ port 3	Length	W[a]
SSUB	Substrate definition	Text	[b]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory.

[b]If only one SSUB is present in the schematic, this substrate is automatically used. If multiple SSUB substrate definitions are present, the user must specify.

Implementation Details

This circuit component models a three-way right-angle junction of Stripline Transmission Lines. This model includes the reactive components of the tee junction analyzed as a zero thickness junction of the effective widths of each section. The model assumes that W1 is equal to W2. If this assumption is not true, then the model assumes $W1=W2 = (W1*W2)^{1/2}$. This model does not include any losses associated with this parasitic reactance of the junction. The parameters W1, W2 and W3 (Strip Widths) are dimensions entered in the default length units. The parameter SSUB specifies the Stripline

Substrate element, which defines additional cross-sectional parameters of the transmission line. This model also warns of the possibility of higher-order propagating modes if they exist.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See "Cell Options Dialog Box: Layout Tab " for Cell Options dialog box **Layout** tab details.

See "The Layout Process File (LPF)" for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

Restrictions

Let Z_{01} be characteristic impedances of attached transmission lines (note that the model assumes $W_1 = W_2$ so condition $Z_{01}=Z_{02}$ always holds.)

Z_{01} and Z_{03} must be within the following limits:

$$0.1 \le \frac{Z_{01}}{Z_{03}} \le 4$$

NI AWR recommends that Z_{01} and Z_{03} satisfy the following inequality:

$$0.2 \le \frac{Z_{01}}{Z_{03}} \le 2$$

Recommendations for Use

Discontinuity models function most accurately when attached to lines that match their corresponding edges. Directly connecting discontinuity models to one another reduces their accuracy.

Reference

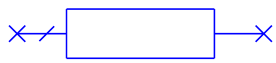
[1] A.A. Oliner, _Equivalent Circuits for Discontinuities in Balanced Strip Transmission Line_ IRE Transactions on Microwave Theory and Techniques, March 1955 p.134-143

[2] H.M. Altschuler and A.A. Oliner, "Discontinuities in the Center Conductor of Symmetric Strip Transmission Line", IRE Transactions on Microwave Theory and Techniques, May 1960 p.328-339

[3] KC. Gupta "Computer-Aided Design of Microwave Circuits", Artech House 1981 p.188-191

Stripline Meander Line (Closed Form): STRACE

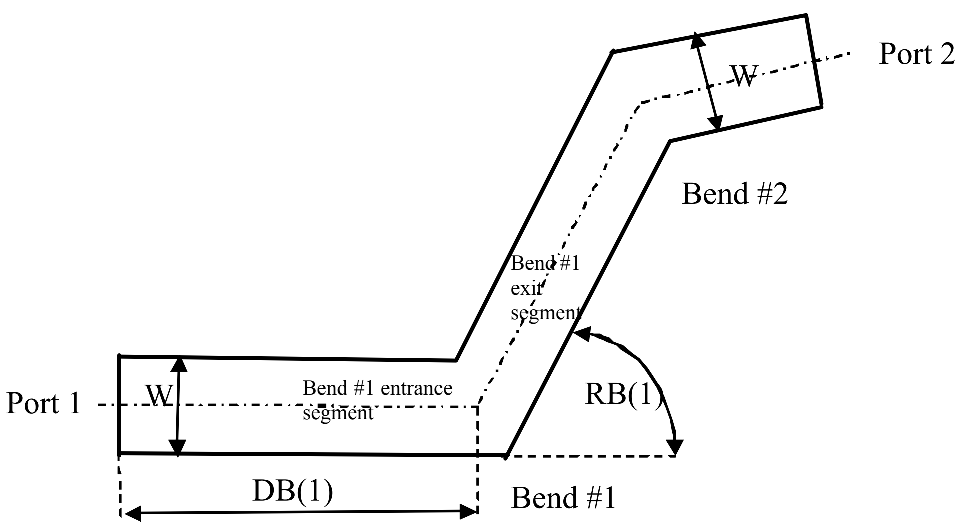
Symbol



Summary

The STRACE element is a stripline that you can route by graphically editing the element in a layout view. The model of the element is constructed by cascading striplines and non-mitered stripline bends.

Topology



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	TL#
W	Conductor width	Length	W[a]
L	Center line length	Angle	L[a]
SSUB	Substrate definition	Text	SSUB[b]
*DB	Bend position array	Length	{0}[b] Vector values are determined dynamically in the layout view of the schematic.
*RB	Bend angle position array (degrees)	Angle	{0}[b]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See Microwave Office Layout Guide for details.

[b]Modify only if the schematic contains multiple substrates. See the "Using Elements With Model Blocks" for details.

** indicates a secondary parameter*

Parameter Details

L. Total length of the line measured along the central line of the trace (see the "Topology" section).

RB and DB. RB contains the central line lengths of trace segments between bends along the trace central line, starting from the entrance segment of bend #1. Note that the length of the last segment is equal to the difference between total line length L and the sum of all DB entries. DB contains the angle positions of the exit segment (see the "Topology" section) for each bend relative to the horizontal axis. These parameters are for internal use by the model. *Manually editing the RB and DB parameters is not recommended*; you should only change the STRACE element by graphically editing the layout cell.

Implementation Details

STRACE is a stripline that you can route by graphically editing the element in the Layout View. The model of the element is constructed by cascading stripline bends (SBENDs) and striplines (SLINs). Note that the bends are non-mitered (miter M=0) arbitrary angle bends.

See the SBEND and SLIN Help for details about these models.

In general, you can graphically edit the STRACE element using mouse commands. Double-clicking the element activates blue diamonds you can use to manipulate the bends and lines.

See the MTRACE2 Help for a detailed description of graphical routing.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

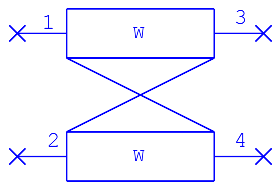
1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See “Cell Options Dialog Box: Layout Tab ” for Cell Options dialog box **Layout** tab details.

See “The Layout Process File (LPF)” for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

Differential Stripline (Closed Form): STRACED

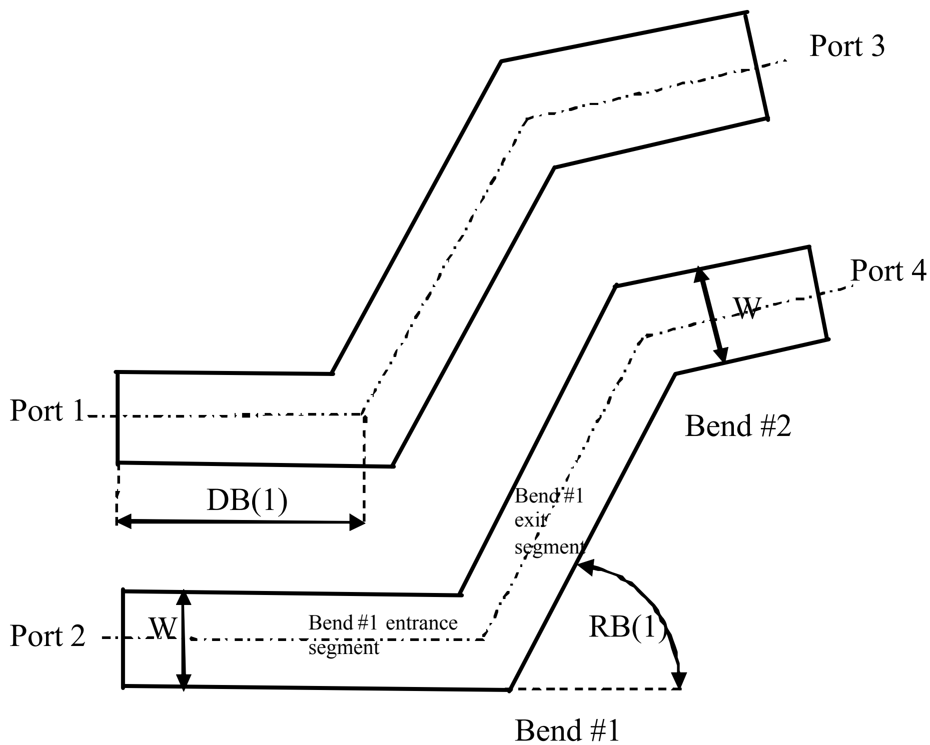
Symbol



Summary

STRACED is a set of cascaded segments of edge-coupled striplines and non-mitered stripline bends arranged at angles relative to each other. This line can be routed by graphically editing the element in the Layout View and may be beneficial for design of differential transmission lines.

Topology



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	TL#
W	Conductor width	Length	W[a]

Name	Description	Unit Type	Default
L	Center line length	Angle	L[a]
SSUB	Substrate definition	Text	SSUB[b]
*DB	Bend position array	Length	{0}[b] Vector values are determined dynamically in the layout view of the schematic.
*RB	Bend angle position array (degrees)	Angle	{0}[b]
S	Spacing between lines edges	Length	L[a]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See Microwave Office Layout Guide for details.

[b]Modify only if the schematic contains multiple substrates. See the "Using Elements With Model Blocks" for details.

** indicates a secondary parameter*

Parameter Details

L. Total length of the line measured along the central line of the trace (see the "Topology" section).

RB and DB. RB contains central line lengths of trace segments between bends along trace #1 central line starting from the entrance segment of bend #1. Note that the length of the last segment is equal to the difference between total line length L and the sum of all DB entries. DB contains the angle positions of the exit segment (see the "Topology" section) for each bend relative to the horizontal axis. These parameters are for internal use by the model. *Manually editing the RB and DB parameters is not recommended*; you should only change the STRACED element by graphically editing the layout cell.

Implementation Details

STRACED is a stripline that you can route by graphically editing the element in the Layout View. The model of the element is constructed by cascading stripline bends (SBENDs) and striplines (SLINs). Note that bends are non-mitered (miter M=0) arbitrary angle bends.

See the SBEND and SLIN Help for details about these models.

In general, you can graphically edit the STRACED element using mouse commands. Double-clicking the element activates blue diamonds you can use to manipulate the bends and lines.

See the MTRACE2 Help for a detailed description of graphical routing.

Layout

This element uses line types to determine its layout. By default, the layout uses the first line type defined in your Layout Process File (LPF). You can change the element to use any of the line types configured in your process:

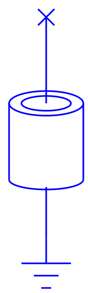
1. Select the item in the layout.
2. Right-click and choose **Shape Properties** to display the Cell Options dialog box.
3. Click the **Layout** tab and select a **Line Type**.
4. Click **OK** to use the new line type in the layout.

See "Cell Options Dialog Box: Layout Tab " for Cell Options dialog box **Layout** tab details.

See "The Layout Process File (LPF)" for more information on editing Layout Process Files (LPFs) and to learn about adding or editing line types.

Cylindrical One Port Via with Stripline Pad (Closed Form): SVIA1P

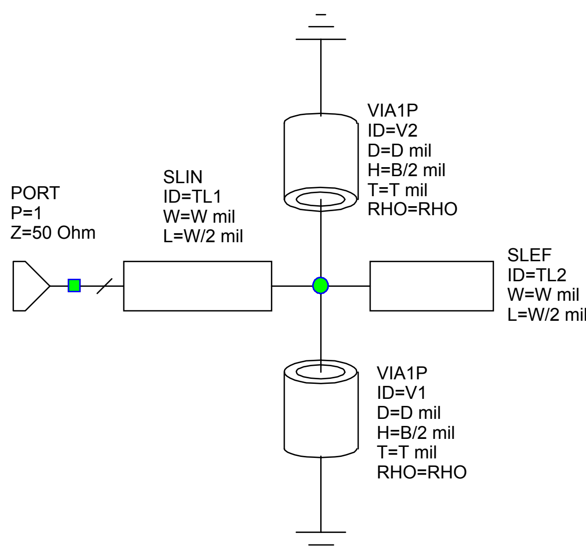
Symbol



Summary

SVIA1P models a stripline short at the end of a transmission line. It is an aggregate model comprised of a transmission line and a transmission line open-end effect, with two vias to ground at the junction of the two transmission line elements (one via up to the upper ground and one via down to the lower ground). The result is a square pad with a via in the middle.

Equivalent Circuit



Parameters

Name	Description	Unit Type	Default
ID	Element ID		V1
D	Hole diameter	um	40
H	Substrate thickness	Length	LPF default
T	Metal thickness	Length	LPF default

Name	Description	Unit Type	Default
W	Pad width	Length	LPF default
RHO	Metal bulk resistivity normalized to gold		1
*SNAME	Structure name from LPF file, empty string uses VIA, DEFINE, BEGIN from LPF file.		
SSUB	Substrate definition		

** indicates a secondary parameter*

Parameter Restrictions and Recommendations

- Parameter W should match the width of the transmission line connected to SVIA1P. Otherwise, you can use any of the SSTEP family of junctions to model the change in line width.
- Parameter H should be set to the substrate's B/2, since each via only goes through half the ground-ground spacing.

Implementation Details

SVIA1P is implemented as a series combination of SLIN and SLEF, with two VIA1P's at the transmission line junction.

Layout

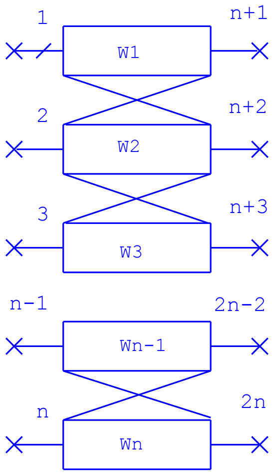
VIA models draw according to the SNAME parameter setting on the via, as configured in the LPF. If this parameter is left blank the via definition in the LPF is used to define the via layers. See “Via Definitions” for details. This typically works well if there is only one via type in the design. If you need to have different via layers, the best approach is to create structures in the LPF for each type, then set the SNAME parameter to match the structure name in the LPF. See “Structure Type Definitions” for more information.

Recommendations for Use

When using this model, keep in mind the physical nature of the square pad surrounding the via. You will effectively add a length to the targeted transmission line equal to W/2, which is the distance the signal sees in the SVIA1P before encountering the via. The element connected to SVIA1P should, therefore, be shortened by W/2 if electrical length is critical.

Multiple Edge Coupled Striplines (EM Quasi-Static) (X=3 to 16): SXCLIN

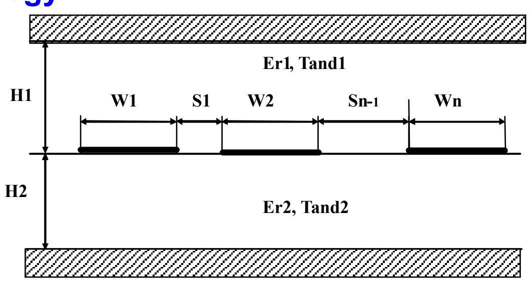
Symbol



Summary

A set of circuit components models sections of multiple edge coupled striplines in a two-layer dielectric substrate. Line conductors are sandwiched between top and bottom dielectric layers. This model allows arbitrary unequal heights of dielectric layers. The model name includes the number of coupled lines (>=3): S3CLIN, S4CLIN etc up to S16CLIN. A backing metallic ground plane and metallic cover are always present.

Topology



Parameters

Name	Description	Unit Type	Default
ID	Name	Text	TL1
W1	Width of microstrip line	Length	W[a]
W2	Width of microstrip line	Length	W[a]
Wn, n=3..XX - number of lines	Width of microstrip line	Length	W[a]
S1	Spacing between lines	Length	W[a]
Sn, n=2..XX - number of lines	Spacing between lines	Length	W[a]

Name	Description	Unit Type	Default
L	Physical length	Length	L[a]
Acc	Accuracy parameter		1
SSUBL	Substrate name	Text	[b]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory.

[b]If only one SSUBL is present in the schematic, this substrate is automatically used. If multiple SSUBL substrate definitions are present, you must specify which to use.

Implementation Details

Model implementation is based on the EM Quasi-Static technique described in [1]. It accounts for losses in metal and in substrate dielectric. Dispersion is partly included.

These circuit components model sections of multiple edge coupled striplines. The models allow the thickness and characteristics of dielectrics at both sides of striplines to be different. S3CLIN models three edge-coupled striplines, S4CLIN - four, S5CLIN - five, S6CLIN - six, and so on up to S16CLIN. Two edge-coupled striplines are modeled by S2CLIN; documentation for this model is provided separately.

The parameters W1..WX (strip widths), S1.. SX (gaps between strips), and L (line length) are dimensions entered in the default length units. Here X stands for the number of coupled lines that is different for each model.

The Acc parameter is the accuracy parameter (1<=Acc<=10). The default value for Acc is 1. If Acc is less than 1 or greater than 10 it is set automatically to 2. Larger values of Acc increase the density of mesh used in computations. The accuracy of model parameters may gain slightly from increasing Acc at the expense of a noticeable growth of computation time. Generally, a good trade-off between accuracy and computation time is to set Acc to 1.

The SSUBL parameter specifies the two-layered stripline substrate element which defines additional cross-sectional parameters of the transmission line. If blank, a default is used.

These components do not impose restrictions on the conductor thickness (the thickness may be zero, positive, or negative). Negative thickness means that the conductor is recessed into the substrate.

The components account for losses in metal and in substrate dielectric. Dispersion is not included.

NOTE: The implementation of EM Quasi-Static models relies heavily on the involved numerical algorithms. This may lead to the noticeable increase of computation time for some schematics that employ many models of this kind.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

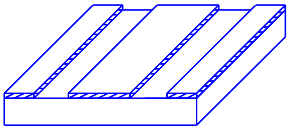
References

[1] M.B. Bazdar, A.R. Djordjevic, R.F. Harrington, and T.K. Sarkar, "Evaluation of quasi-static matrix parameters for multiconductor transmission lines using Galerkin's method," IEEE Trans. Microwave Theory Tech., vol. MTT-42, July 1994, pp. 1223-1228.

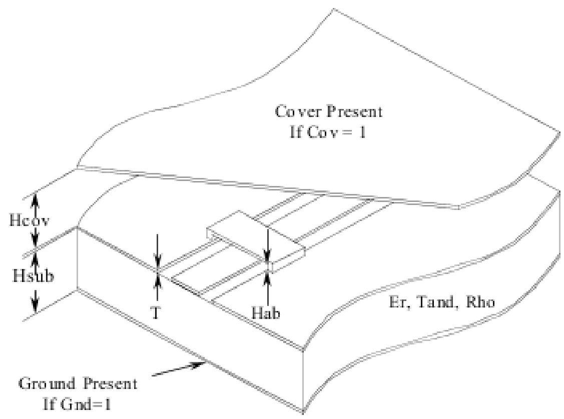
Substrates

Coplanar Waveguide Substrate Definition: CPW_SUB

Symbol



Topology



Parameters

Name	Description	Unit Type	Default
Er	Relative dielectric constant		Er[a]
H	Substrate thickness	Length	L[a]
T	Conductor thickness	Length	T[a]
Rho	Metal bulk resistivity normalized to gold		1
Tand	Loss tangent of dielectric		Tand[a]
Hcover	Cover height above substrate	Length	L[a]
Hab	Air bridge height	Length	L[a]
Cover	No cover if 0, else metallic cover at Hcover		L[a]
Gnd	No ground plane if 0, else metallic ground plane backing		L[a]
Er_Nom	Nominal dielectric constant		Er[a]
H_Nom	Nominal substrate thickness	Length	L[a]
Hcov_Nom	Nominal cover height	Length	L[a]

Name	Description	Unit Type	Default
Hab_Nom	Nominal air bridge height	Length	L[a]
T_Nom	Nominal conductor thickness	Length	T
Name	Substrate name	Text	CPW_SUB1[b]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See the Microwave Office Layout Guide for details.

[b] Modify only if schematic contains multiple substrates. See "Using Elements With Model Blocks" for details.

Parameter Details

Er. Relative (to permittivity of free space $\omega_0 = 8.85e^{-12}$ F/M^2) dielectric constant of substrate material.

Rho. Rho is the bulk resistivity of conductor metal normalized to gold (that is, to Rho_Gold=2.44 x $10^{-8}\Omega$*m) So actual bulk resistivity of conductor metal is Rho_Gold*Rho Ω*m.

Hcover. Optional metallic cover elevation above the substrate. Valid only if parameter Cover is nonzero; not used otherwise.

Hab. Hab represents height of wire or strip air bridge that may be used to suppress an undesirable slotline (odd) mode.

Cover, Gnd. Cover and Gnd are unitless indicators; setting them to nonzero value (for example, 1) makes models that use this substrate to account for presence of metallic cover (element Cover) and/or ground plane (Gnd). Zero values of these indicators inform the model that corresponding item is not present.

Er_Nom. Er_Nom represents the nominal dielectric constant of the substrate relative to free space permittivity and is used only by the EM based models where all EM data is collected at Er_Nom and a variational approach is used to estimate the performance for small variations in Er about Er_Nom. See "How to Properly Set Up Substrate Parameters for X-models" for more details.

Hsub_Nom, Hcov_Nom, Hab_Nom, and T_Nom. These parameters are nominal values of corresponding parameters Er, Hsub, Hcover, Hab, and T; they are used only by the EM based models. See "How to Properly Set Up Substrate Parameters for X-models" for more details

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Recommendations for Use

CPW_SUB must be either present on the schematics that contain elements using CPW_SUB or it may be placed in the Global Definitions window. In the latter case, models using a global definition of CPW_SUB must refer to it explicitly.

Elements used with CPW_SUB

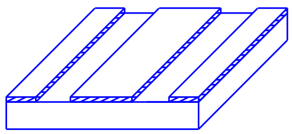
LINES	OTHERS
CPW1LINE	CPWABRGX
CPWALINE	CPWEG
CPWLINE	CPWGAP
CPWLINX	CPWOC

LINES	**OTHERS**
CPWTAPER	CPWSC

BEND	**JUNCTIONS**	**COUPLED LINES**
CPWBENDX	CPWTEEX	CPW2LINA
		CPW2LINE
		CPW3LINA

(Obsolete) Coplanar Waveguide Substrate Definition: CPWSUB

Symbol



Summary

This element is OBSOLETE and is replaced by the Coplanar Waveguide Substrate Definition (CPW_SUB) element.

Parameters

Name	Description	Unit Type	Default
Er	Relative dielectric constant		0
H	Substrate thickness	Length	0 um
T	Conductor thickness	Length	0 um
Rho	Metal bulk resistivity normalized to gold		0
Tand	Loss tangent of dielectric		0
HC	Cover height above substrate	Length	0 um
Name	Substrate name	Text	SUB1[a]

[a]Modify only if schematic contains multiple substrates. See "Using Elements With Model Blocks" for details.

Parameter Details

Er. The permittivity of the substrate material relative to freespace $\varepsilon o = 8.854$ x 10-12 C2/N^2meter2.

Tand. The dielectric loss tangent of the substrate material: (Tand = εr"/εr' with εr 1/2 εr'- jεr")

Rho. Rho is the volume resistivity of the conductors relative to gold: 2.44 x 10 $^{-8}\Omega$ * m)

Physical volume resistivity = 2.44 x 10^{-8} x Rho Ω * m.

H, HC, and T are cross-sectional dimensional variables given in default length units.

In a schematic that contains coplanar waveguide elements, at least one instance of this element must exist.

Name. Provides a unique identifier for each instance of CPWSUB. The coplanar waveguide elements have a CPWSUB parameter that indicates which CPWSUB element to use.

Elements used with CPWSUB

OBSOLETE
CPWLIN
CPW1LIN
CPW2LIN

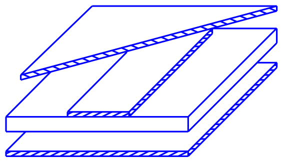
OBSOLETE
CPW3LIN

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

General Multilayer Substrate Definition: GMSUB

Symbol

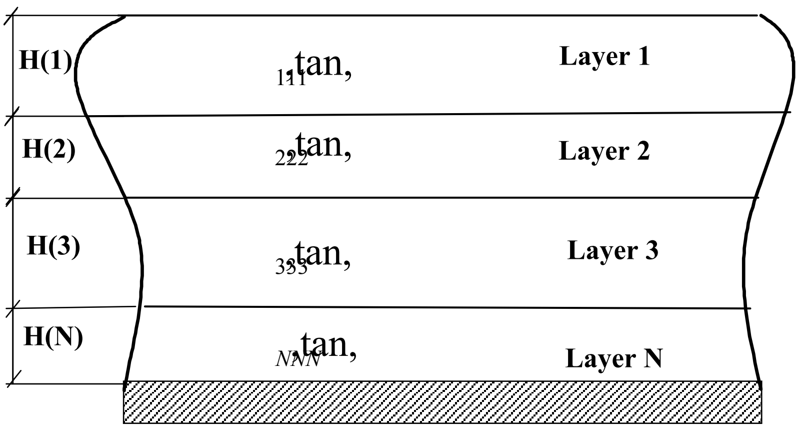


Summary

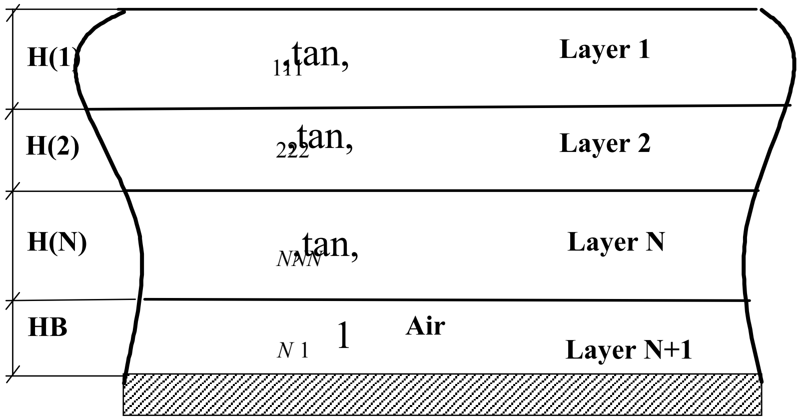
GMSUB provides a stratified inhomogeneous substrate (dielectric stack) definition used either for stripline (optional metallic cover present) or microstrip (no cover) GMnLIN (n=1..10), GM1LIN, GMCLIN, and GFMCLIN schematic elements. The substrate may be elevated over the infinite backing ground plate, thus providing suspended substrate. All layers may have an arbitrary height and may be made of various materials. Each layer may carry any number of conductors. All conductor on the same layer have the same thickness; conductor thickness may vary between layers. The number of conductors is specified in the respective model. In addition to metallic cover, GMSUB may define perfectly conducting side walls thus confining the whole structure into metallic box.

Topology

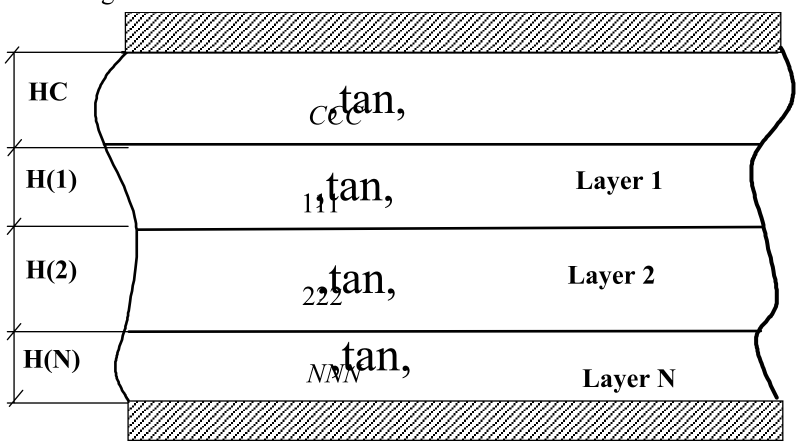
Open grounded substrate (no cover)



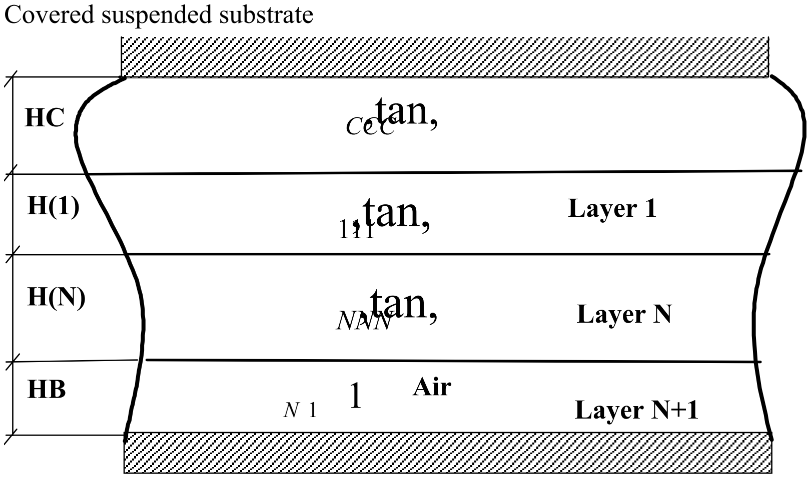
Open suspended substrate (no cover)



Covered grounded substrate



Covered suspended substrate



Parameters

Name	Description	Unit Type	Default
N	Number of substrate dielectric layers		1

Name	Description	Unit Type	Default
Er	Substrate relative dielectric constants (vector)		Er[a]
Tand	Substrate loss tangents (vector)		0
H	Heights of substrate layers (vector)	Length	H[a]
ErC	Relative dielectric constants of layer adjacent to cover		1
TandC	Loss tangent of layer adjacent to cover		0
HC	Cover height over the substrate	Length	H[a]
HB	Substrate elevation over the backing ground plane	Length	H[a]
T	Conductors thicknesses (vector)	Length	T[a]
Rho	Metal bulk resistivity relative to gold		1
Cover	Switch "Metallic Cover" / "No Cover"/Metallic Box		"No Cover"
Gnd	Switch "Suspended Substrate" / "Grounded Substrate"		"Grounded Substrate"
SW	Separation of vertical metallic walls from extreme left/right conductors (optional)	Length	L[a]
Name	Substrate name	Text	GMSUB1[b]
SWRight[c]	Separation of RIGHT vertical metallic wall from extreme right conductor (optional)	Length	0
SigmaC[c]	Bulk conductivity (S/m) of layer adjacent to cover		0
Sigma[c]	Substrate bulk conductivity (S/m) (vector)		0
RhoV[c]	Metal bulk resistivity relative to gold of conductor metals in layers (vector)		1
Undercut[c]	Conductor etching undercut (vector)		0
MuP[c]	Real part of substrate relative magnetic permeability (vector)		1
MuPP[c]	Imaginary part of substrate relative magnetic permeability (vector)		0
Rgh[d]	Surface roughness (RMS surface profile height)	Length	0

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See Microwave Office Layout Guide for details.

[b] Modify only if schematic contains multiple substrates. See “Using Elements With Model Blocks” for details.

[c]For use with element GFMCLIN only

[d]For use with elements GMCLIN, GM1LIN, GMnCLIN only

Parameter Details

N. Number of dielectric layers that may carry conductors on their top surface. Layers are enumerated from top to bottom. Note that if switch parameter Cover is set to "Metallic Cover" the topmost layer adjacent to cover is not included in count; likewise, if switch parameter Gnd is set to "Suspended Substrate" the bottom air layer is not included in count.

Er. Relative dielectric constants of substrate layers are specified as values of vector Er components. You can assign values to vector components via array of constants (or equation variables) on the corresponding parameter field (for example, {3.8,4.1,12,12.5}). The same assignment might be implemented in equation. Models GMnCLIN, GMCLIN check if the length of vector Er is equal to the specified number of layers N.

Tand. Substrate loss tangents of substrate layers are specified as values of vector Tand components. You can assign values to vector components via array of constants (or equation variables) on the corresponding parameter field (for example, {0.001,1e-6,0}). The same assignment might be implemented in equation. Models GMnCLIN, GMCLIN check if the length of vector Tand is equal to the specified number of layers N.

H. Heights of substrate layers are specified as values of vector H components. You can assign values to vector components via array of constants (or equation variables) on the corresponding parameter field (for example, {10,20,20,10}). The same assignment might be implemented in equation. Models GMnCLIN, GMCLIN check if the length of vector H is equal to the specified number of layers N.

ErC. If switch Cover is set to "Metallic Cover" or "Metallic Box" this parameter specifies relative dielectric constant of a layer that adjoins the cover. If switch Cover is set to "No Cover" parameter ErC is not used.

TandC. If switch Cover is set to "Metallic Cover" or "Metallic Box" this parameter specifies loss tangent of a layer that adjoins the cover. If switch Cover is set to "No Cover" parameter TandC is not used.

HC. Specification of height over the substrate HC is mandatory if switch Cover is set to "Metallic Cover" or "Metallic Box" . If switch Cover is set to "No Cover" parameter HC is not used.

HB. Specification of substrate elevation over the backing ground plane is mandatory if switch Gnd is set to "Suspended Substrate". If switch Gnd is set to "Grounded Substrate" parameter HB is not used.

T. Thicknesses of conductors for each layer are specified as values of vector T components. All conductors on the same layer have identical thickness. Thicknesses may vary between layers. Each dielectric layer must have corresponding entry in vector T even if this specific layer does not carry conductors. For example, if model GM1LIN (single line model) is used and number of layers is five, vector T must contain five entries no matter that only one entry corresponds to the layer that actually carries single conductor. All entries may be set to the same value. You can assign values to vector components via array of constants (or equation variables) on the corresponding parameter field (for example, {10,20,20,10}). The same assignment might be implemented in equation. Models GMnCLIN, GMCLIN check if the length of vector T is equal to the number of dielectric layers that may carry conductors (that is, N if switch Gnd is set to "Grounded Substrate" and N+1 if Gnd is set to "Suspended Substrate").

Rho. All conductors have the same metal bulk resistivity relative to gold Rho (relevant only to GMnCLIN, GMCLIN models). GFMCLIN does not use this parameter.

Cover. This switch can be set to one of three selections: "Metallic Cover", "No Cover", "Metallic Box." "Metallic Cover" provides a covered structure typical for striplines; HC, ErC, TandC are used. "No Cover" provides an open structure typical for microstrips; HC, ErC, TandC are not used. Selections "Metallic Cover" as well as "Metallic Box" imply that layer immediately under the cover is excluded from layer count; material properties of this layer are defined via parameters ErC and TandC. Selection "Metallic Box" installs metallic cover and adds side metallic walls thus confining the whole structure into rectangular metallic enclosure. Note that for models GMnCLIN and GMCLIN distance from the leftmost conductor to the left wall is always equal to the distance from the right-most conductor to the right wall; this distance is set by parameter SW. (see below section Examples).

Gnd. This switch can be set to one of two selections: "Suspended Substrate" or "Grounded Substrate." "Suspended Substrate" provides an elevated structure typical for suspended substrates. Selection "Metallic Cover" implies that air layer adjacent to the ground plane is excluded from substrate layer count; material properties of this layer are defined inside the model. Nevertheless, this air layer potentially may carry conductors (obviously, conductors, if any, are affixed to the substrate but they stick out downwards and are recessed into the air layer) so suspended substrate uses HB and demands vector T to be of length N+1. "Grounded Substrate" places a backing ground plane back to back with substrate; in this case vector T has length N and HB is not used.

SW. Distance from the left-most/right-most conductor to the left/right wall. GMnCLIN/GMCLIN use SW as (equal) distances to the left and right wall because GMnCLIN/GMCLIN imply that the modeled structure sits symmetrically between the side walls. See the GMnCLIN documentation "Parameter Restrictions and Recommendations" section for additional explanations and examples. The SW parameter is used only if the Cover parameter is set to "Metallic Box."

NOTE: GFMCLIN uses SW as distance to the left wall only. Actual interpretation of the SW parameter depends on the setting of the GFMCLIN element IsGndStraps parameter. See GFMCLIN documentation for details.

Additional parameters used by the GFMCLIN model only. See GFMCLIN for details.

SWRight. Same as SW, but defines distance to the right wall only. This parameter is used only if the Cover parameter is set to "Metallic Box." The actual interpretation of this parameter depends on the GFMCLIN IsGndStraps parameter setting. See GFMCLIN for details.

SigmaC. Bulk conductivity of layer adjacent to cover. If switch Cover="No Metallic Cover" this parameter is irrelevant.

Sigma. This vector parameter must contain exactly N entries. Each entry provides bulk conductivity of respective layer counting from top layer.

RhoV. This vector parameter must contain exactly N entries if switch Gnd="Grounded Substrate" and N+1 (exactly) entries if Gnd="Suspended Substrate". Each entry provides bulk conductivity of respective conductor metal layer counting from top.

Undercut. GFMCLIN optionally uses this vector parameter to apply its values as etch undercuts to conductors on each layer. Etch undercut specifies non-rectangular (trapezoidal) distortion of conductor cross-section due to manufacturing errors. Undercut may take positive, zero, and negative values. All conductors that belong to the same layer have an identical undercut equal to the corresponding entry of vector Undercut. Vector parameter Undercut must contain exactly N entries if the GMSUB Gnd parameter is "Grounded Substrate", and exactly N+1 entries if Gnd is "Suspended Substrate".

MuP, MuPP. GFMCLIN optionally (if parameter IsMagSub=Magnetic substrate) uses these vector parameters to apply its values as real and imaginary parts of relative magnetic permeability of dielectric layers. Vector parameters MuP and MuPP must contain exactly N entries each.

Rgh. Surface roughness (RMS surface profile height). Models GMCLIN, GM1LIN, and GM2CLIN..GM10CLIN use Rgh to evaluate effect of metal surface roughness.

Implementation Details

Figures in Topology section demonstrate various implementations of GMSUB. Note that suspended substrate (parameter Gnd is set to "Suspended substrate") implies that layer N+1(air) is placed automatically under the layer N; grounded substrate (parameter Gnd is set to "Grounded substrate") implies that ground plane immediately backs layer N.

Covered substrate (parameter Cover is set to "Metallic Cover") implies that additional unnumbered layer (carrying cover) is automatically placed above layer 1. Material parameters of this layer must be set via parameters ErC and TandC.

Evaluation of the surface roughness impact is based on the improved Hammerstad formula.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

Recommendations for Use

GMSUB must either be present on the schematics that contain elements using GMSUB or be placed on the Global Definitions window. In the latter case, models using a global definition of GMSUB must refer to it explicitly.

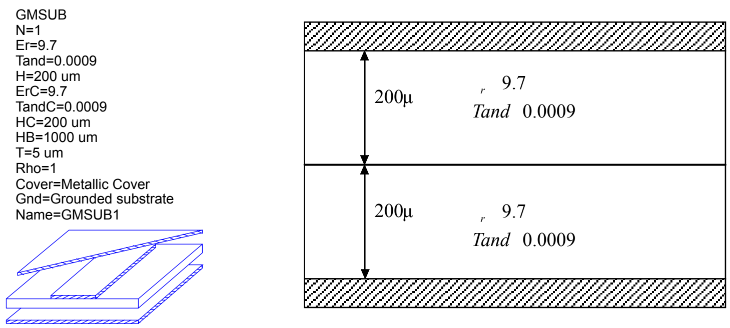
The following examples demonstrate how to configure GMSUB parameters for modeling commonly used single layer (N=1) substrates.

See GMnCLIN documentation for example of configuration of GMSUB enclosed in metallic box.

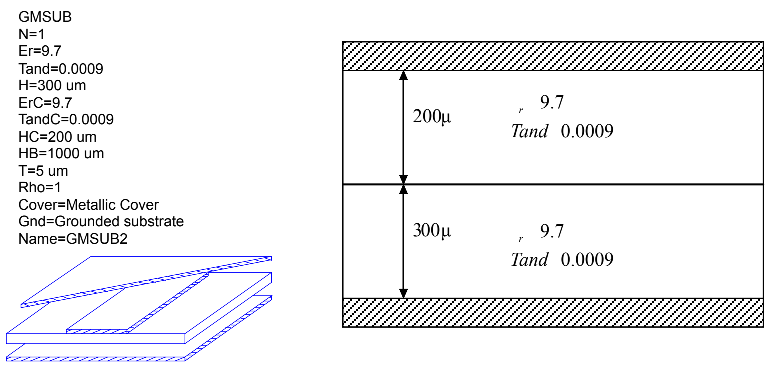
NOTE: Large ratios for SW/Hstack and HC/Hstack (Hstack stands for total height of dielectric stack plus optional HC and HB) should be avoided with models GFMnCLIN and GMCLIN due to a possible reduction in accuracy. It is not recommended to set these ratios above 30-40.

Examples

Centered Stripline Substrate

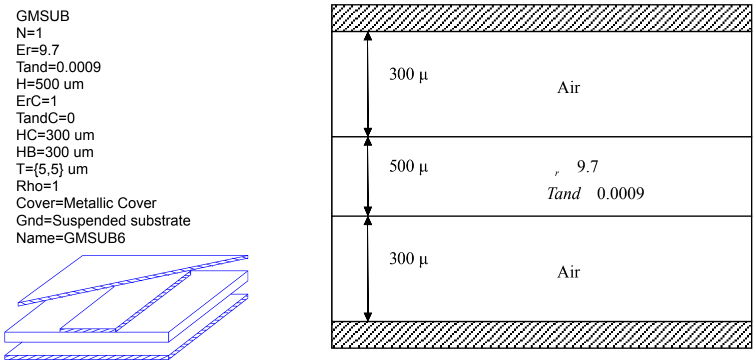


Off-center Stripline Substrate



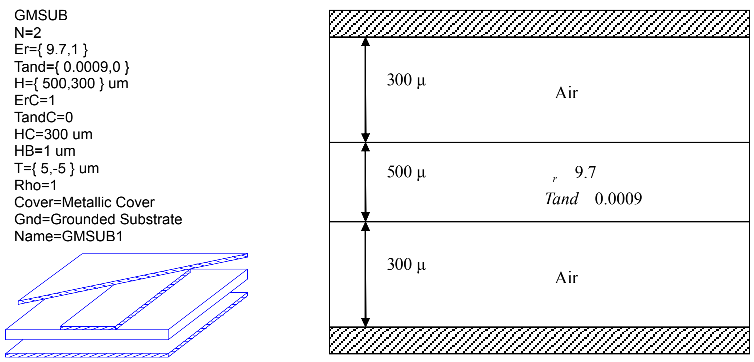
Suspended (Inverted) Stripline Substrate

Substrate allows to arrange conductors at both sides of layer with Er=9.7. Air layers are not included into layer count due to usage of options Cover=Metallic Cover and Gnd=Suspended Substrate. Only middle layer with Er=9.7 is included in layer count. Note that vector T has two entries because the air layer under the substrate may "carry" conductors that stick out from the substrate bottom, but are recessed into the air layer. The negative thickness of these conductors is imposed (sign of second entry of vector T gets effectively overridden) by models GM1LIN, GMnLIN, and GMCLIN.



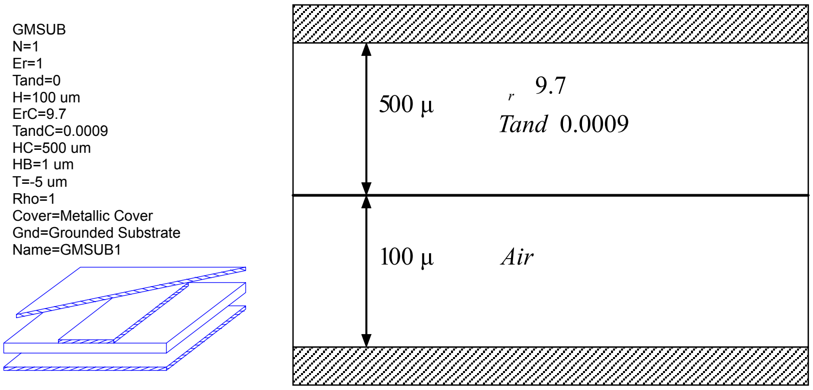
Alternative implementation of Suspended (Inverted) Stripline Substrate

This example demonstrates how we can alternatively (and less effectively) describe same substrate as above not using option Gnd = Suspended Substrate. Substrate allows to arrange conductors at both sides of layer with Er=9.7. Let us include bottom air layer as well as middle layer into layer count. Note that vector T has two entries because the air layer under the substrate may "carry" conductors that stick out from the substrate bottom but are recessed into the air layer. Thus, we must set second entry of vector T explicitly to negative value; must do this explicitly because in case Gnd = Grounded Substrate models do not change sign of T. Models ignore parameter HB at specified setting of Gnd so HB is set to arbitrary value. We must also set explicitly each of two entries of vectors Er, Tand, and H. Evidently, option Gnd=Suspended Substrate simplifies description of this substrate.

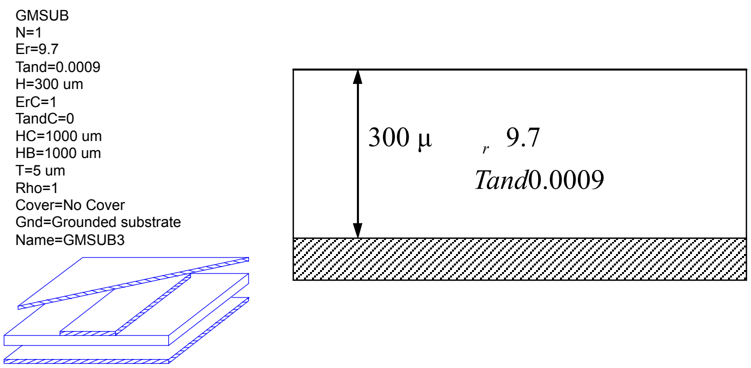


TWO-LAYER INVERTED AND SUSPENDED STRIPLINE SUBSTRATE

Covered substrate allows to arrange conductors only at bottom boundary of layer with Er=9.7 (inverted arrangement). We cannot use Gnd= Suspended Substrate and must assume that air layer is included in layer count. Conductors must recess into air layer so T is set explicitly to negative value. Models ignore parameter HB at specified setting of Gnd so HB is set to arbitrary value. All parameters of layer with Er=9.7 are entered as parameters of layer adjacent to cover (ErC, TandC, and HC).

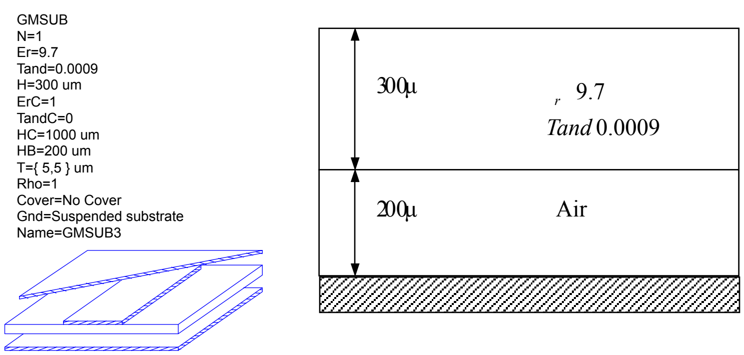


Microstrip Substrate



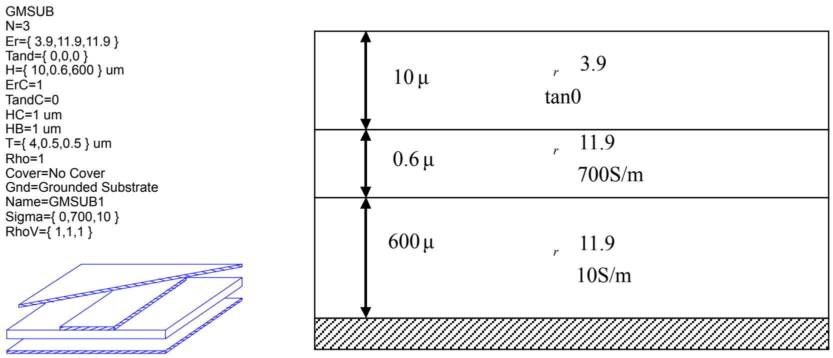
Suspended (Inverted) Microstrip Substrate

Note that vector T has two entries because the air layer under the substrate may carry conductors that stick out from the substrate bottom, but are recessed into the air layer. The negative thickness of these conductors is provided by GM1LIN, GMnLIN, and GMCLIN models.

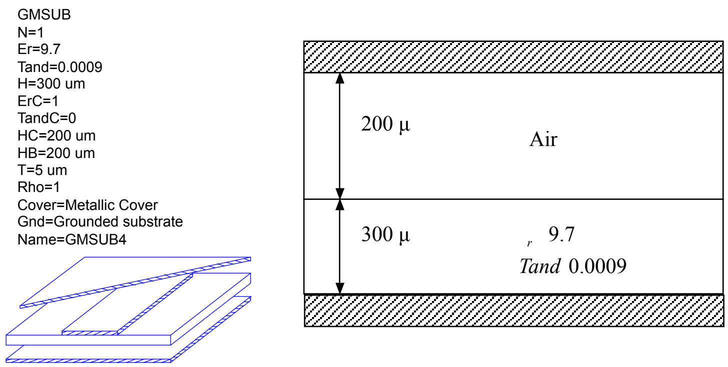


CONDUCTING SUBSTRATE (FOR USE WITH GFMCLIN)

This three-layer substrate features two conducting layers. Conductivity of these layers are specified in vector Sigma. Note that length of vector Sigma must be equal to number of layers (this requirement refers equally to vectors RhoV and T).



Covered Microstrip Substrate

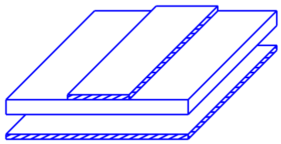


Elements Used with GMSUB

LINES
GFMCLIN
GMnCLIN
GM1LIN
GMCLIN
MXOVER2

Suspended Microstrip Substrate Definition: MPSUB

Symbol



Summary

MPSUB provides a suspended substrate definition used for microstrip schematic elements. Suspended substrate is elevated over the infinite ground plate; no cover is present over the substrate. Substrate is backed by an infinite ground plane and separated from it by an air layer.

Parameters

Name	Description	Unit Type	Default
Er	Substrate relative dielectric constant		Er[a]
Tand	Substrate loss tangent		0
H1	Substrate height	Length	H[a]
H2	Substrate elevation over the ground	Length	H[a]
T	Conductor thickness	Length	T[a]
Rho	Metal bulk resistivity relative to gold		1
ErNom	Nominal relative dielectric constant		Er[a]
Name	Substrate name	Text	MPSUB1[b]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See the Microwave Office Layout Guide for details.

[b]Modify only if a schematic contains multiple substrates. See “Using Elements With Model Blocks” for details.

Implementation Details

Note that you can also add this substrate to the Global Definitions window and then reference the material from any schematic.

ErNom represents the nominal dielectric constant of the substrate relative to free space and is used only by the EM Based models where all EM data is collected at Er_Nom and a variational approach is used to estimate the performance for small variations in Er about Er_Nom. See “How to Properly Set Up Substrate Parameters for X-models” for more details.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

Recommendations for Use

MPSUB must be present on the schematics that contain elements using MPSUB.

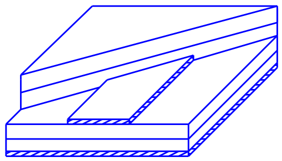
Elements Used with MPSUB

BENDS	COUPLED LINES	JUNCTIONS
MSBND90X	MS2CLINX	MSCROSSX
	MSBCPLX	MSSTEPX
	MI2CLIN	MSTEEX
	MS2CLIN	
	MSBCPL	

LINES	OTHER
MS1LIN	MSOPENX
MI1LIN	
MS1LINX	

RFIC Three-Layer Substrate Definition: MRFSUB

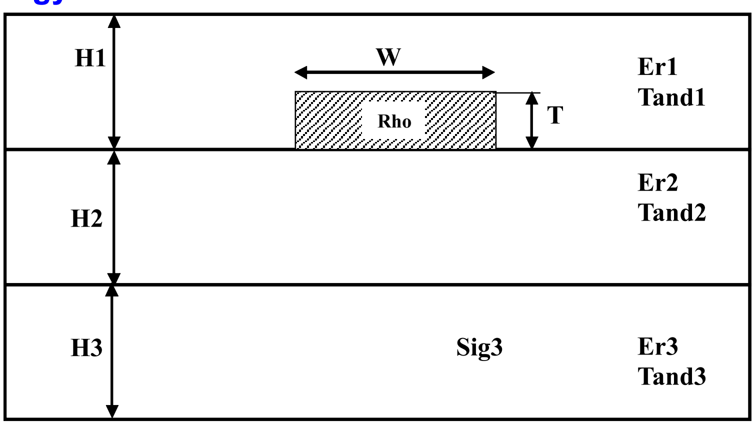
Symbol



Summary

MRFSUB provides a stratified substrate definition suitable for typical RFIC applications. MRFSUB is used by several RFIC schematic elements like FM1LIN, FM2CLIN, FMOCIND, FMRIND, FMCIND and FMLINX. Substrates specify parameters of three layers: two upper insulating layers (like passivation and oxide) and a third (optionally conducting) layer (like silicon or GaAs).

Topology



Parameters

Name	Description	Unit Type	Default
Er1	Relative dielectric constant of top layer		Er[a]
Er2	Relative dielectric constant of middle layer		Er[a]
Er3	Relative dielectric constant of bottom layer		Er[a]
Tand1	Loss tangents of top layer		0
Tand2	Loss tangents of middle layer		0
Tand3	Loss tangents of bottom layer		0
Sig3	Bulk conductance of bottom layer	Conductance in Siemens/m	0
H1	Heights of top layer	Length	H[a]
H2	Heights of middle layer	Length	H[a]
H3	Heights of bottom layer	Length	H[a]
T	Conductor(s) thickness	Length	T[a]

Name	Description	Unit Type	Default
Rho	Metal bulk resistivity relative to gold		1
Name	Substrate name	Text	MRFSUB1[b]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See the Microwave Office Layout Guide for details.

[b]Modify only if schematic contains multiple substrates. See "Using Elements With Model Blocks" for details.

Parameter Restrictions and Recommendations

T, H1. A model that uses MRFSUB usually places the conductor on top of layer 2 and checks if H1>T. Models that use MRFSUB supply a conductor width W.

Layout

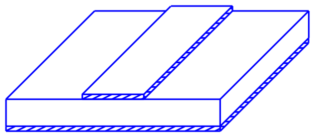
This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Recommendations for Use

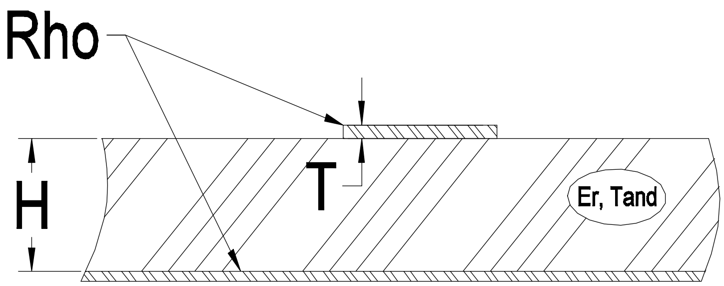
MRFSUB must either be present on the schematics that contain elements using MRFSUB, or be placed in the Global Definitions window. In the latter case, models using a global definition of MRFSUB must refer to it explicitly.

Microstrip Substrate Definition: MSUB

Symbol



Topology



Parameters

Name	Description	Unit Type	Default
Er	Relative dielectric constant		Er[a]
H	Substrate thickness	Length	H[a]
T	Conductor thickness	Length	T[a]
Rho	Metal bulk resistivity normalized to gold		Rho[a]
Tand	Loss tangent of dielectric		0
ErNom[b]	Nominal relative dielectric constant		Er[a]
Name	Substrate name	Text	SUB

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory.

[b]See "X-model Tables Shipped with the NI AWRDE" for a list of available X-model tables.

Implementation Details

Er is the permittivity of the substrate material relative to the permittivity of free space $\varepsilon_0 = 8.85e^{-12}$ F/M^2

ErNom represents the nominal dielectric constant of the substrate relative to free space and is used only by X-Models where all EM data is collected at Er_Nom and a variational approach is used to estimate the performance for small variations in Er about Er_Nom. See "How to Properly Set Up Substrate Parameters for X-models" for details.

Tand is the dielectric loss tangent of the substrate material: Tand = $\varepsilon_r''/\varepsilon_r'$ where $\varepsilon_r = \varepsilon_r' - j\varepsilon_r''$

Rho is the bulk resistivity of conductor metal normalized to gold (that is, to 2.44 x $10^{-8}\Omega$*m)

So actual metal bulk resistivity = 2.44 x $10^{-8}\Omega$*Rho*m.

H and T are cross-sectional dimensional variables given in default length units.

In a schematic that contains microstrip elements, at least one instance of this element must exist. The Name parameter provides a unique identifier for each instance of MSUB. The microstrip elements have an MSUB parameter that indicates which MSUB element to use.

Note that you can also add this substrate to the Global Definitions window and then reference the material from any schematic.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

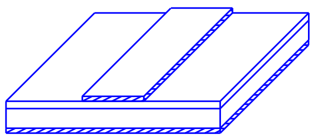
Elements Used with MSUB

BENDS	COMPONENTS	COUPLED LINES	INTERCON-NECTS	JUNCTIONS
MBENDA	TFRM	M2CLIN	WIRE1	MCROSS
MBEND90X	TFR	MACLIN	WIRE2	MCROSSX
MCURVE	TFCM	MCFIL		MSTEP
MBENDRWX	TFC2	MCLIN		MSTEPX
	MRINDSBR	MLANGE		MTEE
	MRINDNBR	MXCLIN		MTEEX
	MCINDN			
	MCINDS			
	MICAP			
	MICAP1			
	MICAP2			
	MICAP3			
	TFCMP			

LINES	OTHER	OBSOLETE	PWRDIVIDER
MCTRACE	MDRSTUB	MBEND	MBLCOUP
MLEF	MGAP2	MBEND2	MLANGE1
MLEFX	MOPEN	MBEND3	MRRCOUP
MLIN	MOPENX	MBENDR	WILKE1
MLSC	MSRSTUB2	MGAP	WILKE2
MTAPER	MRSTUB2	MLOC	WILKN1
MTRACE	ABRIDGE	MSTEP2	WILKN2
MTRACE2	MDRSTUB2	MSTEPO	
	MGAPX	MRSTUB	
		MSRSTUB	

Two-Layer Microstrip Substrate Definition: MSUB2

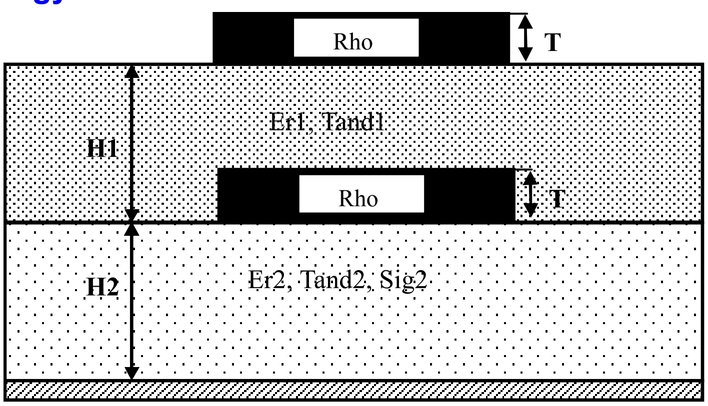
Symbol



Summary

MSUB2 provides two-layer substrate definition used for microstrip schematic elements. Substrate is backed by infinite ground plane; no cover is present over the substrate. Lower layer may have conductive loss.

Topology



Parameters

Name	Description	Unit Type	Default
Er1	Relative dielectric constant of upper layer		Er1
Er2	Relative dielectric constant of lower layer		Er[a]
Tand1	Loss layer tangent of upper layer		0
Tand2	Loss layer tangent of lower layer		0
Sig2	Bulk conductivity of lower layer in S/m		0
H1	Height of upper layer	Length	H[a]
H2	Height of lower layer	Length	H[a]
T	Conductor thickness	Length	T[a]
Rho	Metal bulk resistivity relative to gold		1
Name	Substrate name	text	MSUB1[b]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See the Microwave Office Layout Guide for details.

[b]Modify only if schematic contains multiple substrates. See "Using Elements With Model Blocks" for details.

Parameter Details

T. Line models based on MSUB2 use the same thickness T for all line conductors. EM quasi-static models accept negative T: in this case, model implies that conductors is recessed into underlaying layer.

Rho. Line models based on MSUB2 use the same metal bulk resistivity relative (normalized) to resistivity of gold for all line conductors.

Parameter Restrictions and Recommendations

Sig2. EM quasi-static models based on MSUB2 imply that bulk conductivity of lower layer Sig2 must be less or equal to 3 S/m due to approximation used for evaluation of substrate conductance loss. Unrestricted substrate conductance is provided by Finite Element Method (FEM) based models FM1LIN, FM2CLIN, FMRIND, FMOCIND (use substrate FMSUB).

Recommendations for Use

MSUB2 must either be present on the schematics that contain elements using MSUB2 or be placed in the Global Definitions window. In the latter case, models using a global definition of MSUB2 must refer to it explicitly.

Layout

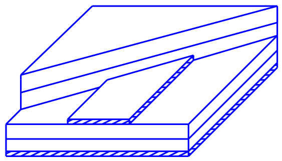
This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Elements Used with MSUB2

OTHER	LINES	COUPLED LINES
MXOVER	MM1LIN	MM2CLIN
	SIGCMOSX	MMBCPL

Four Layer Microstrip Substrate Definition: MSUB4

Symbol



Topology

Parameters

Name	Description	Unit Type	Default
Er1	First (top) layer relative dielectric constant		Er[a]
Er2	Second layer relative dielectric constant		Er[a]
Er3	Third layer relative dielectric constant		Er[a]
Er4	Fourth layer relative dielectric constant		Er[a]
Tand1	Loss tangent of the first (top) layer		0
Tand2	Loss tangent of the second layer		0
Tand3	Loss tangent of the third layer		0
Tand4	Loss tangent of fourth (bottom) layer		0
H1	First (top) layer thickness	Length	H[a]
H2	Second layer thickness	Length	H[a]
H3	Third layer thickness	Length	H[a]
H4	Fourth (bottom) layer thickness	Length	H[a]
T	Conductor thickness	Length	T[a]
Rho	Metal bulk resistivity relative to gold		Rho[a]
Name	Substrate name	Text	MSUB4

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory.

Implementation Details

Er1, Er2, Er3, Er4 are the permittivities of the substrate materials relative to free space $\varepsilon_0 = 8.85e^{-12}$ F/M^2. Tand1, Tand2, Tand3, Tand4 are the dielectric loss tangent of the substrate material: Tand = $\varepsilon_r''/\varepsilon_r'$ where $\varepsilon_r = \varepsilon_r'$ -jε_r''

Rho is the bulk resistivity of conductor metal normalized to gold (that is, to 2.44 x $10^{-8}\Omega$*m)

So actual metal bulk resistivity = 2.44 x $10^{-8}\Omega$*Rho*m.

H1, H2, H3, H4, and T are the cross-sectional dimensional variables entered in default length units.

In a schematic that contains microstrip elements, at least one instance of this element must exist. The Name parameter provides a unique identifier for each instance of MSUB4. The microstrip elements have an MSUB4 parameter that indicates which MSUB4 element to use.

This substrate is used in MEMLIN (Embedded Microstrip Line) and MEM2LIN (2 Embedded Coupled Microstrip Lines) models.

Note that you can also add this substrate to the Global Definitions window and then reference the material from any schematic.

Layout

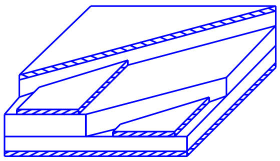
This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Elements Used with MSUB4

COUPLED LINES	LINES
MEM2LIN	MEMLIN

(Obsolete) Multi-Layer Substrate Definition: NSUB

Symbol



Summary

There is no replacement for this OBSOLETE element. NSUB provides a multi-layer substrate definition that can be used with multi-layer models such as the MTRACEM or intelligent net (iNet). The substrate is capable of defining any type of planar substrate stack-up in which each layer is homogeneous (i.e. dielectric bricks are not supported). Each substrate layer may have an arbitrary thickness, dielectric constant, loss tangent, and conductivity that are different from other layers. An infinite number of ground planes may be specified. In addition to substrate parameters, the NSUB also defines certain parameters of the line types used for a particular process.

NOTE: The parameter settings of the NSUB must correspond to a particular process utility for models to be simulated electrically and rendered in the layout correctly.

Parameters

Name	Description	Unit Type	Default
HSub	Dielectric boundary heights relative to the bottom of the stack (vector)	Length	{H[a]}
Er	Substrate relative dielectric constants (vector)		{Er[a]}
Tand	Substrate loss tangents (vector)		{Tand[a]}
Sigma	Substrate conductivities (vector)		{0}
HGnd	Ground plane heights relative to the bottom of the stack (vector)	Length	{H[a]}
HLine	Line type heights relative to the bottom of the stack (vector)	Length	{H[a]}
W	Line type conductor widths (vector). Only used if the widths cannot be obtained from the multi-layer electrical model.	Length	{W[a]}
T	Line type conductor thicknesses (vector)	Length	{T[a]}
Rho	Line type conductor bulk resistivities relative to gold (vector)		{Rho[a]}
ProcUtil	Process utility name.	String	""
Name	Substrate name	Text	NSUB1

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See the Microwave Office Layout Guide for details.

Parameter Details

HSub. Dielectric boundary heights relative to the bottom of the stack. Layers are enumerated from bottom to top. The number of elements in this vector parameter represents the number of dielectric layers in the stack-up.

Er. Relative dielectric constants. Note that Er may be different for each layer. The number of elements in this vector must be equal to the number of elements in the HSub parameter.

Tand. Substrate loss tangents. Note that Tand may be different for each layer. The number of elements in this vector must be equal to the number of elements in the HSub parameter.

Sigma. Substrate conductivities. Note that Sigma may be different for each layer. The number of elements in this vector must be equal to the number of elements in the HSub parameter.

HGnd. Ground plane heights relative to the bottom of the stack. Ground plane layers are enumerated from bottom to top. An infinite number of ground planes may be specified. If a ground plane at the bottom of the stack is desired, the first element of the vector must be a zero. It is assumed that the ground plane covers the entire layer (i.e. partial ground planes are not allowed).

HLine. Line type heights relative to the bottom of the stack. The purpose of this vector parameter is to define the height of each line type defined in the LPF file. The number of elements in this vector must be less than or equal to the number of line types defined in the LPF file. Also, each element of the vector corresponds to the line type in that position in the LPF file. For example, the first element of the vector corresponds to the first line type defined in the LPF, the second element to the second line type, etc.

W. Line type widths. The purpose of this vector parameter is to define the default width of each line type defined in the LPF file. NOTE: This parameter is only used if the line width cannot be obtained from the multi-layer electrical model. The number of elements in this vector must be equal to the number of elements in the HLine parameter.

T. Line type conductor thicknesses. The purpose of this vector parameter is to define the conductor thickness of each line type defined in the LPF file. The number of elements in this vector must be equal to the number of elements in the HLine parameter.

Rho. Line type conductor bulk resistivities relative to gold. The purpose of this vector parameter is to define the conductor bulk resistivities of each line type defined in the LPF file. The number of elements in this vector must be equal to the number of elements in the HLine parameter.

ProcUtil. Process utility name. This parameter specifies the name of the process utility associated with the current NSUB settings. The process utility defines which electrical models to use for each layer in the stack-up and how to draw the discontinuity model layout cell. Note: If the current settings of the NSUB do not correspond to the specified process utility, the multi-layer electrical models will not simulate and the layout will look incorrect. If a process utility name is not specified, AWR_NLAYER will be used. This utility defines a two-layer stack-up in which the bottom layer is considered stripline and the top layer is considered microstrip.

Implementation Details

This substrate may be placed locally within a schematic or in the Global Definitions window. If placed in the Global Definitions window, it may be accessed from any schematic.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Recommendations for Use

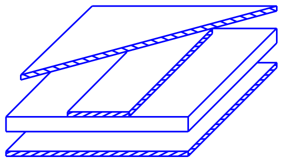
An NSUB must be defined before any multi-layer model can be modeled electrically or rendered in a layout.

Elements Used with NSUB

Lines		Nonlinear > FET
MTRACEM	ELENET (iNet)	BSIM3V322
SIG1LNX	EXTRACT	BSIM3V322P
		MOSN1A
		MOSN1_4A
		MOSN3A
		MOSN3_4A
		MOSP1A
		MOSP1_4A
		MOSP3A
		MOSP3_4A
		MOSN2
		MOSN2_4
		MOSP2
		MOSP2_4

Homogeneous Stratified Substrate Definition: PCSUB

Symbol



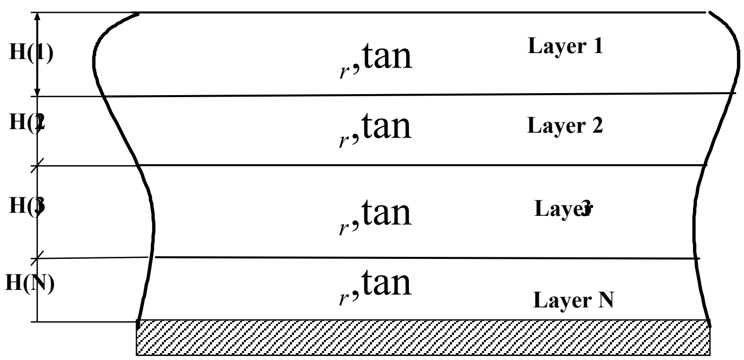
Summary

PCSUB provides stratified homogeneous substrate definition used either for stripline (optional cover present) or microstrip (no cover) PCnCLIN, PC1LIN schematic elements. Substrate may be elevated over the infinite backing ground plate, thus providing suspended substrate. All layers may have arbitrary height and are made of the same material. Each layer may carry any number of conductors. All conductor on the same layer have the same thickness; thickness may vary between layers. The number of conductors is specified in the PCnCLIN model. In addition to metallic cover, PCSUB may define side perfectly conducting walls thus confining the whole structure into metallic box.

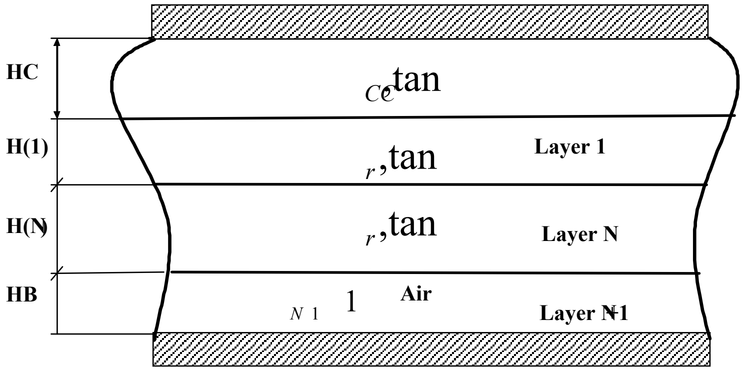
PCSUB models PCB (Printed Circuit Board) or LTCC/HTCC (Low/High Temperature Cofired Ceramic) substrate.

Topology

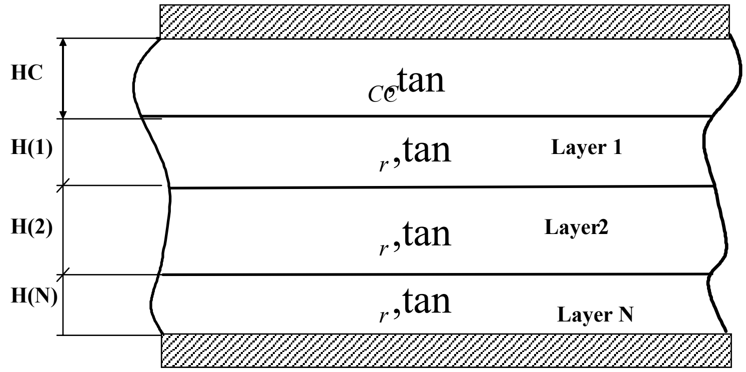
Open grounded substrate (no cover)



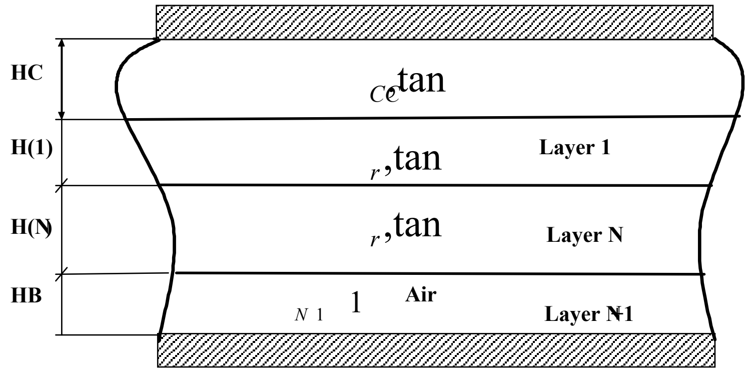
Open suspended substrate (no cover)



Covered grounded substrate



Covered suspended substrate



Parameters

Name	Description	Unit Type	Default
N	Number of substrate dielectric layers		1
Er	Substrate relative dielectric constant		Er[a]
Tand	Substrate loss tangent		0
H	Heights of substrate layers (vector)	Length	H[b]
ErC	Relative dielectric constants of layer adjacent to cover		1
TandC	Loss tangent of layer adjacent to cover		0
HC	Cover height over the substrate	Length	H[b]
HB	Substrate elevation over the backing ground plane	Length	H[b]
T	Conductors thicknesses (vector)	Length	T[b]
Rho	Metal bulk resistivity relative to gold		1
Cover	Switch "Metallic Cover" / "No Cover"/Metallic Box		"No Cover"
Gnd	Switch "Suspended Substrate" / "Grounded Substrate"		"Grounded Substrate"
SW	Separation of vertical metallic walls from extreme left/right conductors (optional)	Length	L[b]

Name	Description	Unit Type	Default
Name	Substrate name	Text	PCSUB1[b]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See "Nonlinear Oscillator Analysis" for details.

[b]Modify only if schematic contains multiple substrates. See "Using Elements With Model Blocks" for details.

Parameter Details

N. Number of dielectric layers that may carry conductors on their top surface. Layers are enumerated from top to bottom. Note that if switch parameter Cover is set to "Metallic Cover" the topmost layer adjacent to cover is not included in count; likewise, if switch parameter Gnd is set to "Suspended Substrate" the bottom air layer is not included in count.

Er. Relative dielectric constant is the same for all N layers.

Tand. Substrate loss tangent is the same for all N layers.

H. Heights of substrate layers are specified as values of vector H components. User can assign values to vector components via array of constants (or equation variables) on the corresponding parameter field (for example, {10,20,20,10}). The same assignment might be implemented in equation. Model PCnCLIN checks if the length of vector H is equal to the specified number of layers N.

ErC. If switch Cover is set to "Metallic Cover" this parameter specifies relative dielectric constant of a layer that adjoins the cover. If switch Cover is set to "No Cover" parameter ErC is not used.

TandC. If switch Cover is set to "Metallic Cover" this parameter specifies loss tangent of a layer that adjoins the cover. If switch Cover is set to "No Cover" parameter TandC is not used.

HC. Specification of height over the substrate HC is mandatory if switch Cover is set to "Metallic Cover". If switch Cover is set to "No Cover" parameter HC is not used.

HB. Specification of substrate elevation over the backing ground plane is mandatory if switch Gnd is set to "Suspended Substrate".If switch Gnd is set to "Grounded Substrate" parameter HB is not used.

T. Thicknesses of conductors for each layer are specified as values of vector T components. All conductors on the same layer have similar thickness. This thickness may vary between layers. Each dielectric layer must have corresponding entry in vector T even if this specific layer does not carry conductors. For example, if model PC1LIN (single line model) is used and number of layers is five, vector T must contain five entries no matter that only one entry corresponds to the layer that carries single conductor. All entries may be set to the same value. User can assign values to vector components via array of constants (or equation variables) on the corresponding parameter field (for example, {1,2.5,2,4}). The same assignment might be implemented in equation. Model PCnCLIN checks if the length of vector T is equal to the number of dielectric layers that may carry conductors (that is, N if switch Gnd is set to "Grounded Substrate" and N+1 if Gnd is set to "Suspended Substrate").

Rho. All conductors have the same metal bulk resistivity relative to gold Rho.

Cover. This switch can be set to one of three selections: "Metallic Cover", "No Cover", "Metallic Box." "Metallic Cover" provides a covered structure typical for striplines; HC, ErC, TandC are used. "No Cover" provides an open structure typical for microstrips; HC, ErC, TandC are not used. Selection "Metallic Cover" implies that layer immediately under the cover is excluded from layer count; material properties of this layer are defined via parameters ErC and TandC. Selection "Metallic Box" installs metallic cover (see above) and adds side metallic walls thus confining the whole structure into rectangular metallic box. Note that distance from the leftmost conductor to the left wall is always equal to the distance from the rightmost conductor to the right wall; this distance is set by parameter SW.

Gnd. This switch can be set to one of two selections: "Suspended Substrate" or "Grounded Substrate." "Suspended Substrate" provides an elevated structure typical for suspended substrates. Selection "Metallic Cover" implies that air layer adjacent to the ground plane is excluded from substrate layer count; material properties of this layer are defined inside the model. Nevertheless, this air layer potentially may carry conductors (obviously, conductors, if any, are affixed to the substrate but they stick out downwards and are recessed into the air layer) so suspended substrate uses HB and demands vector T to be of length N+1. "Grounded Substrate" places a backing ground plane back to back with substrate; in this case vector T has length N and HB is not used.

SW. Distance from the leftmost/rightmost conductor to the left/right wall. The parameter is used only if switch Cover is set to "Metallic Box." Model implies that structure sits symmetrically between side walls.

Implementation Details

Figures in Topology section demonstrate various implementations of PCSUB. Note that suspended substrate (parameter Gnd is set to "Suspended substrate") implies that layer N+1(air) is placed automatically under the layer N; grounded substrate (parameter Gnd is set to "Grounded substrate") implies that ground plane immediately backs layer N.

Covered substrate (parameter Cover is set to "Metallic Cover") implies that additional unnumbered layer (carrying cover) is automatically placed over layer 1. Material parameters of this layer must be set via parameters ErC and TandC.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Recommendations for Use

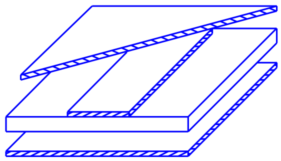
PCSUB must be either present on the schematics that contain elements using PCSUB or be placed in the Global Definitions window. In the latter case, models using a global definition of PCSUB must refer to it explicitly.

Elements Used with PCSUB

LINES
PCnCLIN
PC1LIN

Suspended Stripline Substrates Definition: SPSUB

Symbol



Summary

SPSUB provides a suspended substrate definition used for stripline schematic elements. Suspended substrate is elevated over the infinite ground plate; metallic cover is located over the substrate. Substrate is backed by an infinite ground plane and separated from it by an air layer. All layers may have arbitrary height.

Parameters

Name	Description	Unit Type	Default
Er	Substrate relative dielectric constant		Er[a]
Tand	Substrate loss tangent		0
H1	Cover height over the substrate	Length	H[b]
H2	Substrate height	Length	H[b]
H3	Substrate elevation over the ground	Length	H[b]
T	Conductor thickness	Length	T[b]
Rho	Metal bulk resistivity relative to gold		1
ErNom	Nominal relative dielectric constant		Er[a]
Name	Substrate name	Text	SPUB1[b]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See the Microwave Office Layout Guide for details.

[b]Modify only if the schematic contains multiple substrates. See the “Using Elements With Model Blocks” for details.

Implementation Details

Note that you can also add this substrate to the Global Definitions window and then reference the material from any schematic.

ErNom represents the nominal dielectric constant of the substrate relative to free space and is used only by the EM-Based models where all EM data is collected at Er_Nom and a variational approach is used to estimate the performance for small variations in Er about Er_Nom. See “How to Properly Set Up Substrate Parameters for X-models” for details.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

Recommendations for Use

SPSUB must be present on the schematics that contain elements using SPSUB.

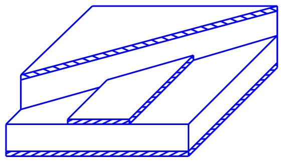
Elements Used with SPSUB

BENDS	COUPLED LINES	JUNCTIONS
SSBND90X	SS2CLINX	SSCROSSX
	SSBCPLX	SSSTEPX
	SS2CLIN	SSTEEX
	SI2CLIN	

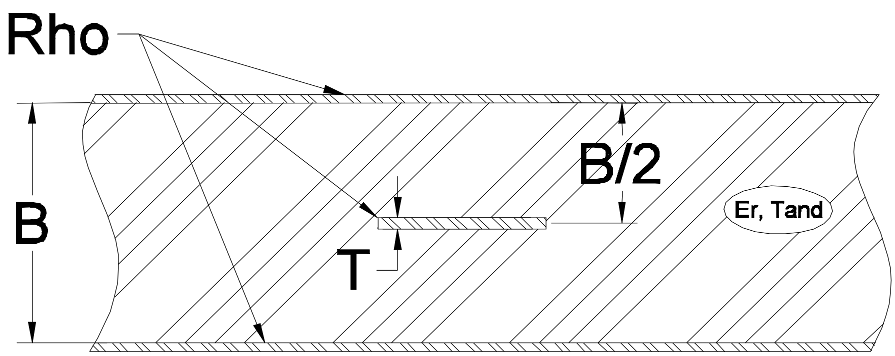
LINES	OTHER
SS1LIN	SSOPENX
SS1LINX	
SI1LIN	

Balanced Stripline Substrate Definition: SSUB

Symbol



Topology



Parameters

Name	Description	Unit Type	Default
Er	Relative dielectric constant		Er[a]
B	Substrate thickness	Length	H[a]
T	Conductor thickness	Length	T[a]
Rho	Metal bulk resistivity normalized to copper		Rho[a]
Tand	Loss tangent of dielectric		0
Name	Substrate name	Text	SUB

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory.

Implementation Details

Er is the permittivity of the substrate material relative to the permittivity of free space $\varepsilon_0 = 8.85e^{-12}$ F/M^2

Tand is the dielectric loss tangent of the substrate material: Tand = $\varepsilon_r''/\varepsilon_r'$ where $\varepsilon_r = \varepsilon_r' - j\varepsilon_r''$

B and T are cross sectional dimensional variables given in default length units.

In a schematic that contains stripline elements, at least one instance of this element must exist. The Name parameter provides a unique identifier for each instance of SSUB. The stripline elements have a SSUB parameter that indicates which SSUB element to use.

Note that you can also add this substrate to the Global Definitions window and then reference the material from any schematic.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

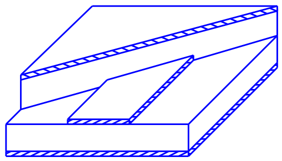
Elements used with SSUB

BENDS	COMPONENTS	JUNCTIONS	LINES
SBEND	SICAP	SCROS	SLEF
SCURVE	SICAP1	SSTEP	SLIN
SMITER	SICAP2	SSTEPO	STAPER
		SSTEP2	SLSC
		STEE	SCTRACE

OTHER	OBSOLETE	COUPLED LINES
SGAP	SLOC	SCLIN
SHOLE		

Unbalanced Stripline Substrate Definition: SSUB2

Symbol



Summary

SSUB2 provides an unbalanced substrate definition used for stripline schematic elements. The substrate is backed by an infinite ground plane and a conducting cover is present over the substrate. An unbalanced substrate is comprised of two stacked substrates made of the same material but having different height. A strip conductor is sandwiched between these substrates.

Parameters

Name	Description	Unit Type	Default
Er	Relative dielectric constant		Er[a]
H1	Substrate No1 thickness	Length	H[a]
H2	Substrate No2 thickness	Length	H[a]
T	Conductor thickness	Length	T[a]
Rho	Metal bulk resistivity normalized to copper		1
Tand	Loss tangent of dielectric		0
ErNom	Nominal relative dielectric constant		Er[a]
Name	Substrate name		SSUB1[b]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See the Microwave Office Layout Guide for details.

[b]Modify only if the schematic contains multiple substrates. See "Using Elements With Model Blocks" for details.

Parameter Details

Er. Relative dielectric constant of the homogeneous dielectric that makes all substrate.

H1. Thickness of the upper substrate (No 1)

H2. Thickness of the lower substrate (No 2)

ErNom. ErNom represents the nominal dielectric constant of the substrate relative to free space and is used only by the EM-Based models where all EM data is collected at Er_Nom and a variational approach is used to estimate the performance for small variations in Er about Er_Nom. See "How to Properly Set Up Substrate Parameters for X-models" for more details.

Implementation Details

Note that you can also add this substrate to the Global Definitions window and then reference the material from any schematic.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Recommendations for Use

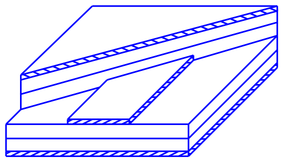
SSUB2 must be present on the schematics that contain elements using SSUB2.

Elements Used with SSUB2

OTHERS
SGAPX

Four Layer Stripline Substrate Definition: SSUB4

Symbol



Topology

Parameters

Name	Description	Unit Type	Default
Er1	First (top) layer relative dielectric constant		Er[a]
Er2	Second layer relative dielectric constant		Er[a]
Er3	Third layer relative dielectric constant		Er[a]
Er4	Fourth (bottom) layer relative dielectric constant		Er[a]
Tand1	Loss tangent of the first (top) layer		0
Tand2	Loss tangent of the second layer		0
Tand3	Loss tangent of the third layer		0
Tand4	Loss tangent of the fourth (bottom) layer		0
H1	First (top) layer thickness	Length	H[a]
H2	Second layer thickness	Length	H[a]
H3	Third layer thickness	Length	H[a]
H4	Fourth (bottom) layer thickness	Length	H[a]
T	Conductor thickness	Length	T[a]
Rho	Metal bulk resistivity normalized to gold		Rho[a]
Name	Substrate name	Text	SUB

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory.

Implementation Details

Er1, Er2, Er3, Er4 are the permittivities of the substrate materials relative to free space $\varepsilon_0 = 8.85e^{-12}$ F/M^2

Tand1, Tand2, Tand3, Tand4 are are the dielectric loss tangent of the substrate material: Tand = $\varepsilon_r''/\varepsilon_r'$ where $\varepsilon_r = \varepsilon_r' - j\varepsilon_r''$

H1, H2, H3, H4, and T are the cross-sectional dimensional variables entered in default length units.

In a schematic that contains stripline elements, at least one instance of this element must exist. The Name parameter provides a unique identifier for each instance of SSUB4. The microstrip elements have a SSUB4 parameter that indicates which SSUB4 element to use.

This substrate is used in SEMLIN (Embedded Stripline) and SEM2LIN (2 Embedded Coupled Striplines) models.

Note that you can also add this substrate to the Global Definitions window and then reference the material from any schematic.

Layout

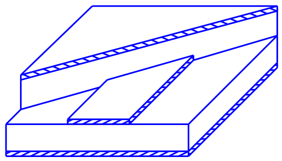
This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

Elements Used with SSUB4

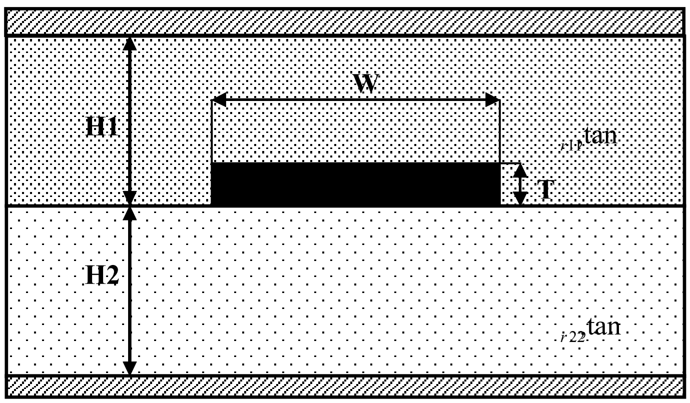
COUPLED LINES	LINES
SEM2LIN	SEMLIN

Two-Layer Stripline Substrate Definition: SSUBL

Symbol



Topology



Parameters

Name	Description	Unit Type	Default
Er1	Top layer relative dielectric constant		Er[a]
Er2	Bottom layer relative dielectric constant		Er[a]
Tand1	Loss tangent of the top layer		0
Tand2	Loss tangent of the bottom layer		0
H1	Top layer thickness	Length	H[a]
H2	Bottom layer thickness	Length	H[a]
T	Conductor thickness	Length	T[a]
Rho	Metal bulk resistivity relative to gold		0[a]
Name	Substrate name	Text	SUB

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory.

Implementation Details

Er1 and Er2 are the permittivities of the substrate materials relative to free space $\varepsilon_0 = 8.85e^{-12}$ F/M^2

Tand1 and Tand2 are the dielectric loss tangent of the substrate material: Tand = $\varepsilon_r''/\varepsilon_r'$ where $\varepsilon_r = \varepsilon_r' - j\varepsilon_r''$

H1, H2, and T are the cross-sectional dimensional variables entered in default length units. Conductor thickness T may be zero, positive, or negative. A negative thickness means that the conductor is recessed into the bottom dielectric layer

In a schematic that contains stripline elements, at least one instance of this element must exist. The Name parameter provides a unique identifier for each instance of SSUBL. The stripline elements have a SSUBL parameter that indicates which SSUBL element to use.

This substrate is used in models S1LIN, SxCLIN (x=2..16), and SNCLIN.

Note that you can also add this substrate to the Global Definitions window and then reference the material from any schematic.

Layout

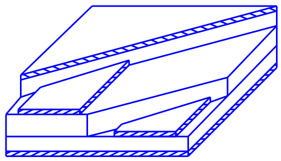
This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Elements Used with SSUBL

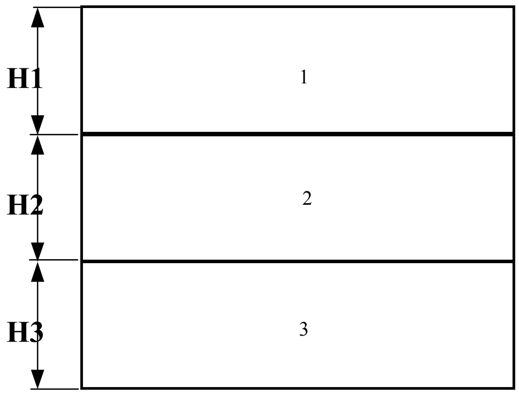
COUPLED LINES	LINES
SxCLIN (x=3..16)	S1LIN
S2CLIN	
SNCLIN	

Three Layered Stripline Substrate Definition: SSUBT

Symbol



Topology



Parameters

Name	Description	Unit Type	Default
Er1	Top layer relative dielectric constant		Er[a]
Er2	Middle layer relative dielectric constant		Er[a]
Er3	Bottom layer relative dielectric constant		Er[a]
Tand1	Loss tangent of the top layer		0
Tand2	Loss tangent of the middle layer		0
Tand3	Loss tangent of the bottom layer		0
H1	Top layer thickness	Length	H[a]
H2	Middle layer thickness	Length	H[a]
H3	Bottom layer thickness	Length	H[a]
T	Conductor thickness	Length	T[a]
Rho	Metal bulk resistivity relative to gold		Rho[a]
Name	Substrate name	Text	SUB

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory.

Implementation Details

Er1, Er2, and Er3 are the permittivities of the substrate materials relative to free space $\varepsilon_0 = 8.85e^{-12}$ F/M^2

Tand1, Tand2, and Tand3 are the dielectric loss tangent of the substrate material: Tand = $\varepsilon_r''/\varepsilon_r'$ where $\varepsilon_r = \varepsilon_r' - j\varepsilon_r''$

H1, H2, H3, and T are the cross-sectional dimensional variables entered in default length units.

In a schematic that contains stripline elements, at least one instance of this element must exist. The Name parameter provides a unique identifier for each instance of SSUBT. The stripline elements have a SSUBT parameter that indicates which SSUBT element to use.

This substrate is used in SBCPL (2 Broadside Coupled Striplines) model.

Note that you can also add this substrate to the Global Definitions window and then reference the material from any schematic.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

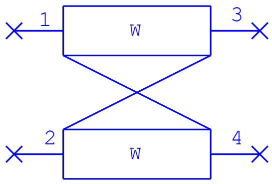
Element Used with SSUBT

COUPLED LINES
SBCPL

Transmission Lines

Phase Spec: Coupled Lines (Closed Form): CLIN

Symbol



Summary

CLIN simulates a pair of symmetric coupled lossless transmission line that supports pure TEM mode of propagation. Model implies that line length is specified as electrical length (phase lag of mode propagating along the line) at user specified frequency. User must supply values of odd and even characteristic impedances.

Parameters

Name	Description	Unit Type	Default
ID	Name	Text	TL1
ZE	Even mode characteristic impedance	Resistance	50 ohm
ZO	Odd mode characteristic impedance	Resistance	100 ohm
EL	Electrical length (phase length) at F0	Angle	90 Deg
F0	Frequency reference for electrical length	Frequency	10 GHz

Parameter Details

EL and **F0.** These parameters determine the frequency dependence of the electrical length of the line, described as $\beta L = EL \cdot freq/Fo \cdot \pi/180$ where *freq* is the evaluation frequency. EL specifies phase lag of propagating TEM mode, that is it accounts for effective dielectric constant of line. Note that even and odd phase velocities are implied equal in this model.

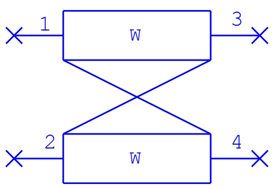
ZE and **ZO**. Even and odd characteristic impedances are used to obtain entries of line Y-matrix.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Physical Spec: Coupled Lines, Grounded Shield, Improved Accuracy (Closed Form): CLINP

Symbol



Summary

CLINP simulates a pair of symmetric coupled lossy transmission lines that supports non-TEM modes of propagation (odd and even). Each mode is specified by its effective dielectric constant and per-unit-length attenuation. By default, CLINP provides partial account for circuit loss. You can toggle the secondary parameter Total_Loss to obtain full access to insertion loss, return loss, and dielectric loss, as well as to loss scaling at a user-specified frequency.

Parameters

Name	Description	Unit Type	Default
ID	Name	Text	TL1
ZE	Even mode characteristic impedance	Resistance	50 ohm
ZO	Odd mode characteristic impedance	Resistance	100 ohm
L	Length of the lines	Length	0 um
KE	Even mode effective dielectric constant		1
KO	Odd mode effective dielectric constant		1
AE	Even mode attenuation (dB/meter)		0
AO	Odd mode attenuation (dB/meter)		0
*LossModel	Partial/Full account for conductor and dielectric loss		Partial
*F0	Frequency for scaling loss	Frequency	0
*TanD	Dielectric loss tangent		0
*Mur	Relative dielectric permeability		1
*TanM	Dielectric magnetic loss tangent		0
*Sigma	Dielectric bulk conductivity (S/m)		0
*IsEpsFreqDep	Dielectric constant is frequency dependent/independent		Frequency independent dielectric constant
*Fmin	Low roll-off frequency of Tand frequency dependence	Frequency	1kHz
*Fmax	High roll-off frequency of Tand frequency dependence	Frequency	1THz
*Fspec	Er and Tand are specified at this frequency	Frequency	1GHz

** indicates a secondary parameter*

Parameter Details

ZE, ZO, KE, KO. Even and odd characteristic parameters used to obtain entries of line Y-matrix.

AE, AO, F0. Frequency for scaling (imparting the frequency-dependence to the attenuation constant AE and AO). If F0 is not equal to zero (0.0), then

$$\alpha = \text{Loss}\sqrt{\text{freq} / \text{F0}}$$

(dB/m) where *freq* is the evaluation frequency, and Loss is attenuations AE and/or AO. If F0 is equal to zero, the attenuation is constant versus frequency and is equal to parameter AE and/or AO(dB/m). Note that loss scaling is available only if the control parameter Total_Loss=Full.

LossModel is a secondary control parameter that defines how the model accounts for loss and dielectric parameters. If LossModel=Partial (default value) than the model accounts for insertion loss (parameter Loss) only. If LossModel=Full then the model accounts for insertion loss, return loss, dielectric loss, and for frequency dependent model of dielectric parameters (for example, for microstrips, stripline, or other).

Note: Secondary parameter LossModel is always visible at model Properties, tab Parameters if Show Secondary is selected. If LossModel=Partial, other secondary parameters are also visible at tab Parameters but they are not used by the model and do not appear at the schematic until LossModel=Full is selected.

IsEpsFreqDep is a (hidden, secondary) control parameter that defines if dielectric constant and dielectric loss tangent exhibit the predefined frequency dependence, suggested in [1]. This dependence is controlled by the three frequency parameters Fmin, Fmax, and Fspec, and imparts causal behavior to the model.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Recommendations for Use

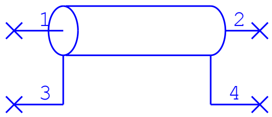
CLINP is a behavioral model that allows you to enter six electric parameters of your choice (unlike models such as MCLIN where you enter primary geometrical/material parameters). The set of parameters may not be feasible or even totally non-physical, and this may affect transient simulation. In the event of a transient simulation failure, NI AWR recommends checking error/warning messages in the HSPICE output file *AwrHSpice.lis* and modifying parameter values accordingly.

References

[1] A.R. Djordjevic et al., "Wideband Frequency-Domain Characterization of FR-4 and Time-Domain Causality", IEEE Trans. of Electromagn. Compat., Vol. 43, No. 4, Nov. 2001

Physical Specification, Floating Shield (Closed Form): COAX

Symbol



Summary

COAX simulates a lossy coaxial transmission line with isolated shield terminals. Losses are determined via physical dimensions and material parameters of metal and dielectric filling.

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	CX1
Di	Inner diameter	Length	2.9027 mm
Do	Outer diameter	Length	10.287 mm
L	Length	Length	0 um
ER	Relative dielectric constant		1
TAND	Dielectric loss tangent		0
RHO	Metal bulk resistivity normalized to copper		1

Parameter Details

Di,Do. This parameters define inner and outer diameters of internal cross-section of coaxial line. Defaults values are set to dimensions of RG-213 50 ohm coax.

Implementation Details

Implementation is based on modeling the coax as TEM line and determination of primary per-unit-length RLGC parameters. These RLGC parameters are used to obtain complex characteristic parameters of dominant propagating TEM mode (characteristic impedance and propagation constant). Model does not account for possible propagation of higher order modes.

See descriptions of model TLIN4 for details of implementation of models with floating ground.

Recommendations for Use

Extreme care should be used with this element as it is meant to work in concert with additional elements which relate the voltages at both ends of the transmission line to the global ground.

NOTE: Because the model definition does not include interactions with the global ground, unusual and unexpected results can occur if other components are not used to relate the voltage on both sides of the transmission line to global ground.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

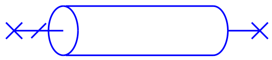
References

[1] I. Bahl, P. Bhartia, Microwave Solid State Circuit Design, John Wiley & Sons, New York, NY, 1988.

[2] M. Gunston, Microwave Transmission-Line Impedance Data, Noble, Atlanta, 1997

Electrical Specification, Grounded Shield (Closed Form): COAX2

Symbol



Summary

COAX2 simulates a lossless transmission line with grounded shield. Model implies that line length is specified as electrical length (phase lag of mode propagating along the line) at user specified frequency.

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	CX1
EL	Electrical length	Angle	90 Deg
Fo	Frequency at which length is specified	Frequency	0.1 GHz
Z	Characteristic impedance		50

Parameter Details

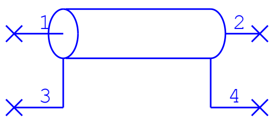
EL and **Fo.** These parameters determine the frequency dependence of the electrical length of the line, described as $\beta L = EL \cdot frq/Fo \cdot \pi/180$ where *freq* is the evaluation frequency.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Electrical Specification, Floating Shield (Closed Form): COAX4

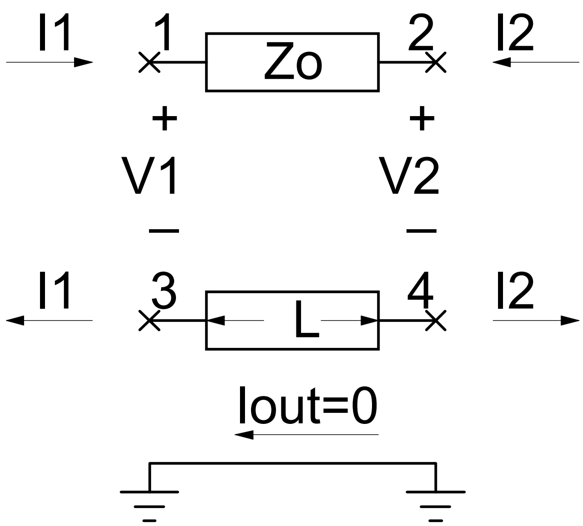
Symbol



Summary

COAX4 simulates a coaxial transmission line with an isolated outer conductor. The characteristics of the transmission line are specified as the characteristic impedance and the electrical length at a given frequency. Extreme care should be used with this element as it is meant to work in concert with additional elements which relate the voltages at both ends of the transmission line to the global ground. Unusual and unexpected behavior can result if these additional elements are omitted.

Equivalent Circuit



Note that any interaction with the global ground on either side of the transmission line is ignored. All current on one conductor is equal and opposite to the current on the other at any given distance along the line. Importantly, this condition is enforced regardless of the length of the transmission line or the operating frequency. The exception to this behavior is at DC (Frequency = 0 Hz) where current on the two conductors can be unequal thus allowing both conductors to be used for biasing active devices. This change in behavior at DC causes a discontinuity of the model parameters at DC. This discontinuity is expected, and additional circuit components relating the voltage at each end of the transmission line to ground should be added, allowing flexibility in implementing the desired transition to the RF-to-DC performance.

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	CX1

Name	Description	Unit Type	Default
EL	Electrical length	Angle	90 Deg
Fo	Frequency at which length is specified	Frequency	0.1 GHz
Z	Characteristic impedance		50

Parameter Details

EL and **Fo.** These parameters determine the frequency dependence of the electrical length of the line, described as βL = EL· frq/Fo · π/180 where *freq* is the evaluation frequency.

Implementation Details

The following is a Y-matrix for a grounded transmission line system:

$$[Y_{(2\times2)}] = \begin{bmatrix} Y_{11(2\times2)} & Y_{12(2\times2)} \\ Y_{21(2\times2)} & Y_{22(2\times2)} \end{bmatrix} = \begin{bmatrix} \dfrac{1}{Z\tanh(j\beta L)} & \dfrac{-1}{Z\sinh(j\beta L)} \\ \dfrac{-1}{Z\sinh(j\beta L)} & \dfrac{1}{Z\tanh(j\beta L)} \end{bmatrix}$$

Applying the equivalent circuit above, the Y-matrix of the floating transmission line system can be shown to be the following:

$$[[Y_{(4\times4)}] = \begin{bmatrix} Y_{11(2\times2)} & Y_{12(2\times2)} & -Y_{11(2\times2)} & -Y_{12(2\times2)} \\ Y_{21(2\times2)} & Y_{22(2\times2)} & -Y_{21(2\times2)} & -Y_{22(2\times2)} \\ -Y_{11(2\times2)} & -Y_{12(2\times2)} & Y_{11(2\times2)} & Y_{12(2\times2)} \\ -Y_{21(2\times2)} & -Y_{22(2\times2)} & Y_{21(2\times2)} & Y_{22(2\times2)} \end{bmatrix}$$

At DC, the Y-matrix changes to a model of two wires above a ground plane:

$$[Y_{(4\times4)}] = \begin{bmatrix} 1/R & -1/R & 0 & 0 \\ -1/R & 1/R & 0 & 0 \\ 0 & 0 & 1/R & -1/R \\ 0 & 0 & -1/R & 1/R \end{bmatrix}$$

where R is a real resistance approaching zero.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

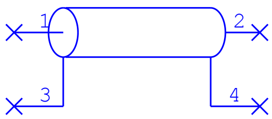
Recommendations for Use

This model, along with additional components can be used to model transmission line baluns and transmission line transformers.

NOTE: Because the model definition does not include interactions with the ground, unusual and unexpected results can occur if other components are not used to relate the voltage on both sides of the transmission line to ground.

Physical Specification: Floating Shield, Improved Causality (Closed Form): COAXC

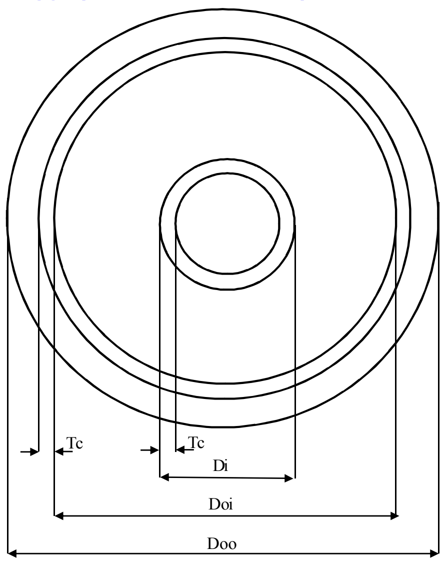
Symbol



Summary

COAXC simulates a lossy, optionally coated coaxial transmission line with isolated shield terminals. The physical model fully accounts for the complex skin effect in the inner conductor and outer (shield) conductor as well as in the metal coating. COAXC is able to model causal frequency-dependence of material properties of the filling dielectric. COAXC is recommended for modeling in both time and frequency domains.

Topology (Cross-section)



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	CX1
Di	Diameter of inner conductor (with coating)	Length	2.9027 mm
Doi	Inner diameter of outer conductor (with coating)	Length	10.287 mm
Doo	Outer diameter of outer conductor	Length	10.387 mm
Tc	Thickness of coating metal	Length	0 mm
L	Line length	Length	10 mm
Er	Relative dielectric constant of filling dielectric		1
Tand	Dielectric loss tangent of filling dielectric		0

Name	Description	Unit Type	Default
Rho	Coaxial metal bulk resistivity normalized to gold		1
RhoC	Coating metal bulk resistivity normalized to gold		1
Mur	Relative permeability of filling dielectric		1
*IsEpsFreqDep	Switch: Dielectric constant is frequency dependent/independent		Frequency independent dielectric constant
*Fmin	Low roll-off frequency of Tand frequency dependence	Frequency	1 KHz
*Fmax	High roll-off frequency of Tand frequency dependence	Frequency	1 THz
*Fspec	Er and Tand are specified at this frequency	Frequency	1 GHz

** indicates a secondary parameter*

Parameter Details

Di, Doi, Doo. These parameters define inner and outer diameters of internal cross-section of coaxial line. Default values are set to dimensions of RG-213 50 ohm coax. Note that coating thickness Tc is included in values Di and Doi (see the "Topology" section).

IsEpsFreqDep. This parameter assumes two options: "Frequency independent dielectric constant" and "Frequency dependent dielectric constant". The first option implies that Er and Tand do not change with frequency. The second option applies frequency dependence on Er and Tand based on the causal model suggested in [4] [17–12], which employs parameters Fdef, Fmin, and Fmax.

Fmin, Fspec, Fmax. These parameters are in effect only if IsEpsFreqDep = "Frequency dependent dielectric constant". Those are frequencies defining behavior of the causal model suggested in [4] [17–12]. Fspec is the frequency at which COAXC parameters Er and Tand are specified; Fmin and Fmax are frequencies at which frequency dependences of Er and Tand roll off. Mandatory relation between these frequencies is: Fmin<<Fspec<<Fmax.

Implementation Details

Implementation is based on modeling the coax as TEM line and the determination of primary per-unit-length RLGC parameters ([1] [17–12]). R and L parameters fully account for the skin effect ([2] [17–12], [3] [17–12]). Causal frequency-dependence of dielectric constant and loss tangent is implemented after [4] [17–12]. These RLGC parameters are used to obtain complex characteristic parameters of dominant propagating TEM mode (characteristic impedance and propagation constant). This model does not account for the possible propagation of higher-order modes.

See the descriptions of the TLIN4 model for details of implementation of models with floating ground.

Recommendations for Use

You should exercise extreme care with this element as it is meant to work in concert with additional elements which relate the voltages at both ends of the transmission line to the global ground.

NOTE: Because the model definition does not include interactions with the global ground, unusual and unexpected results can occur if other components are not used to relate the voltage on both sides of the transmission line to global ground.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

References

[1] M.Gunston, Microwave Transmission Line Impedance Data: Noble Publishing Corporation, Tucker,GA, 1997, (reprint), Section 2.3

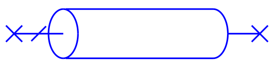
[2] S.Ramo and J.R.Whinnery, Fields and Waves in Modern Radio, 1st ed., General Electric Advanced Engineering Program, New York, NY; London, England: J.Wiley and Sons, Inc.; Chapman and Hall, 1944"

[3] S. A. Schelkunoff, "The Electromagnetic Theory of Coaxial Transmission Lines and Cylindrical Shields", Bell System Technical Journal, Vol. 13, No. 4, pp. 532–579, October 1934

[4] A.R. Djordjevic et al., "Wideband Frequency-Domain Characterization of FR-4 and Time-Domain Causality", IEEE Trans. of Electromagn. Compat., Vol. 43, No. 4, Nov. 2001

Physical Length Spec, Grounded Shield (Closed Form): COAXI2

Symbol



Summary

COAXI2 simulates a lossy transmission line with a grounded shield. This model implies that the total loss is proportional to the square root of the evaluation frequency and demands that you input a specified loss at a specified frequency. Characteristic impedance is represented by user-specified input.

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	CX1
Z	Characteristic impedance		50
L	Length	Length	0
K	Dielectric constant		2.3
A	Loss in dB/meter		0.0833
F	Frequency loss is specified at	Frequency	0.1 GHz

Parameter Details

A and **F**. These parameters determine the frequency-dependence of the attenuation constant. If F is not equal to zero (0.0), then

$$\alpha = A\sqrt{\mathrm{freq}/F}$$

(dB/m) where *freq* is the evaluation frequency. If F is equal to zero, the attenuation is zero at all frequencies. Note that attenuation is extrapolated across frequency sweep, so do not set F to an extremely low value.

Implementation Details

Dielectric constant K and computed (see the previous details of parameters A and F) attenuation are used to obtain a complex propagation constant. Entries of Y-matrix of a grounded transmission line system are determined via the characteristic impedance and computed complex propagation constant that you specify.

The following is a Y-matrix for a grounded transmission line system:

$$[Y_{(2\times2)}] = \begin{bmatrix} Y_{11(2\times2)} & Y_{12(2\times2)} \\ Y_{21(2\times2)} & Y_{22(2\times2)} \end{bmatrix} = \begin{bmatrix} \dfrac{1}{Z\tanh((\alpha + j\beta)L)} & \dfrac{-1}{Z\sinh((\alpha + j\beta)L)} \\ \dfrac{-1}{Z\sinh((\alpha + j\beta)L)} & \dfrac{1}{Z\tanh((\alpha + j\beta)L)} \end{bmatrix}$$

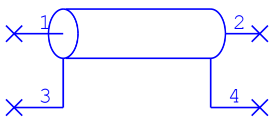
where α+jβ represents the complex propagation constant as derived from the input parameters, K, A and F. Z represents the characteristic impedance of the line and L is the length of the line.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Physical Length Spec, Floating Shield (Closed Form): COAXI4

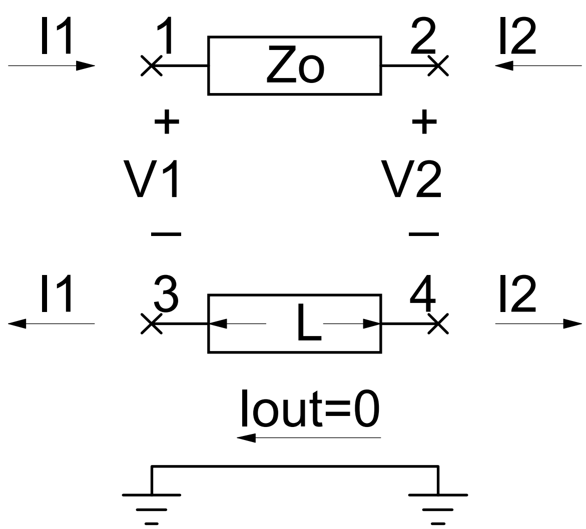
Symbol



Summary

COAXI4 simulates a lossy transmission line with a floating shield. This model implies that the total loss is proportional to the square root of the evaluation frequency and demands that you input a specified loss at a specified frequency. Characteristic impedance is represented by user specified input.

Equivalent Circuit



Note that any interaction with the ground on either side of the transmission line is ignored. All current on one conductor is equal and opposite to the current on the other at any given distance along the line. Importantly, this condition is enforced regardless of the length of the transmission line or the operating frequency. The exception to this behavior is at DC (Frequency = 0 Hz) where the current on the two conductors can be unequal to allow both conductors to be used for biasing active devices. This change in behavior at DC causes a discontinuity of the model parameters at DC. This discontinuity is expected, and additional circuit components relating the voltage at each end of the transmission line to ground should be added, allowing flexibility in implementing the desired transition to the RF-to-DC performance.

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	CX1
Z	Characteristic impedance		50
L	Length	Length	0 um

Name	Description	Unit Type	Default
K	Dielectric constant		2.3
A	Loss in dB/meter		0.0833
F	Frequency loss is specified at	Frequency	0.1 GHz

Parameter Details

A and **F.** These parameters determine the frequency-dependence of the attenuation constant. If F is not equal to zero (0.0), then

$$\alpha = A\sqrt{\mathrm{freq}/F}$$

(dB/m) where *freq* is the evaluation frequency. If F is equal to zero, the attenuation is zero at all frequencies. Note that attenuation is extrapolated across frequency sweep so do not set F to an extremely low value.

Implementation Details

Dielectric constant K and computed (see the previous details of parameters A and F) attenuation are used to obtain a complex propagation constant. Entries of Y-matrix of a grounded transmission line system are determined via the characteristic impedance and computed complex propagation constant that you specify.

The following is a Y-matrix for a grounded transmission line system:

$$[Y_{(2\times2)}] = \begin{bmatrix} Y_{11(2\times2)} & Y_{12(2\times2)} \\ Y_{21(2\times2)} & Y_{22(2\times2)} \end{bmatrix} = \begin{bmatrix} \dfrac{1}{Z\tanh((\alpha+j\beta)L)} & \dfrac{-1}{Z\sinh((\alpha+j\beta)L)} \\ \dfrac{-1}{Z\sinh((\alpha+j\beta)L)} & \dfrac{1}{Z\tanh((\alpha+j\beta)L)} \end{bmatrix}$$

where $\alpha+j\beta$ represents the complex propagation constant as derived from the input parameters, K, A and F. Z represents the characteristic impedance of the line and L is the length of the line.

Applying the equivalent circuit shown above, the Y-matrix of the floating transmission line system can be shown as:

$$[[Y_{(4\times4)}] = \begin{bmatrix} Y_{11(2\times2)} & Y_{12(2\times2)} & -Y_{11(2\times2)} & -Y_{12(2\times2)} \\ Y_{21(2\times2)} & Y_{22(2\times2)} & -Y_{21(2\times2)} & -Y_{22(2\times2)} \\ -Y_{11(2\times2)} & -Y_{12(2\times2)} & Y_{11(2\times2)} & Y_{12(2\times2)} \\ -Y_{21(2\times2)} & -Y_{22(2\times2)} & Y_{21(2\times2)} & Y_{22(2\times2)} \end{bmatrix}$$

At DC, the Y-Matrix changes to a model of two wires above a ground plane:

$$[Y_{(4\times4)}] = \begin{bmatrix} 1/R & -1/R & 0 & 0 \\ -1/R & 1/R & 0 & 0 \\ 0 & 0 & 1/R & -1/R \\ 0 & 0 & -1/R & 1/R \end{bmatrix}$$

where *R* is a real resistance approaching zero.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

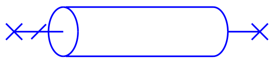
Recommendations for Use

You can use this model along with additional components to model transmission line baluns and transmission line transformers.

NOTE: Because the model definition does not include interactions with the ground, unusual and unexpected results can occur if other components are not used to relate the voltage on both sides of the transmission line to ground.

Physical Spec, Grounded Shield (Closed Form): COAXP2

Symbol



Summary

COAXP2 simulates a lossy coaxial transmission line with a grounded shield. This model implies that the total loss is proportional to the square root of the evaluation frequency and demands that you input a specified loss at a specified frequency. Characteristic impedance is specified by dimensions of coaxial cross-section.

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	CX1
Di	Inner diameter	Length	2903 um
Do	Outer diameter	Length	1.029e+04 um
L	Length	Length	0 um
K	Dielectric constant		2.3
A	Loss in dB/meter		0.0833
F	Frequency at which loss is specified	Frequency	0.1 GHz

Parameter Details

A and **F**. These parameters determine the frequency-dependence of the attenuation constant. If F is not equal to zero (0.0), then

$$\alpha = A\sqrt{\mathrm{freq} / F}$$

(dB/m) where *freq* is the evaluation frequency. If F is equal to zero, the attenuation is zero at all frequencies. Note that attenuation is extrapolated across frequency sweep, so do not set F to an extremely low value.

Di, Do. Inner and outer diameters and dielectric constant K are used to evaluate characteristic impedance Zo as Zo = 138/sqrt(K)*log(Do/Di).

Implementation Details

Dielectric constant K and computed (see the previous details of parameters A and F) attenuation are used to obtain a complex propagation constant. Entries of Y-matrix of a grounded transmission line are determined via computed characteristic impedances (see the details of parameters Di, Do) and computed complex propagation constant.

The following is a Y-matrix for a grounded transmission line system:

$$[Y_{(2\times2)}] = \begin{bmatrix} Y_{11(2\times2)} & Y_{12(2\times2)} \\ Y_{21(2\times2)} & Y_{22(2\times2)} \end{bmatrix} = \begin{bmatrix} \dfrac{1}{Z\tanh((\alpha+j\beta)L)} & \dfrac{-1}{Z\sinh((\alpha+j\beta)L)} \\ \dfrac{-1}{Z\sinh((\alpha+j\beta)L)} & \dfrac{1}{Z\tanh((\alpha+j\beta)L)} \end{bmatrix}$$

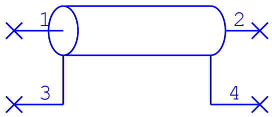
where $\alpha + j\beta$ represents the complex propagation constant as derived from the input parameters, K, A and F. Z represents the characteristic impedance of the line and L is the length of the line.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Physical Spec, Floating Shield (Closed Form): COAXP4

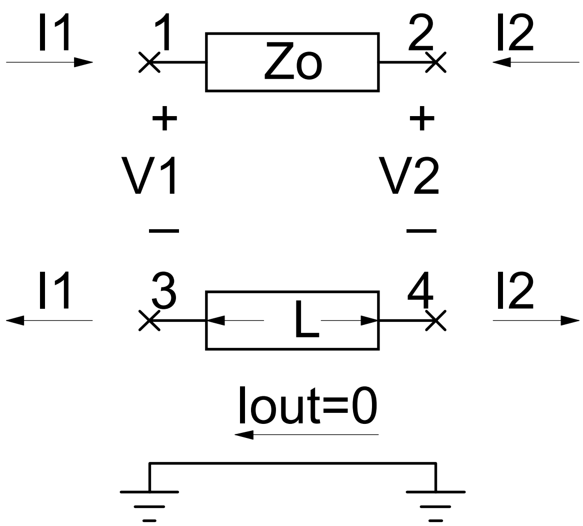
Symbol



Summary

COAXP4 simulates a coaxial transmission line with an isolated outer conductor. The characteristics of the transmission line are specified as inner and outer conductor diameters and a length. The complex propagation constant is specified via a relative dielectric constant and loss term. Extreme care should be used with this element as it is meant to work in concert with additional elements which relate the voltages at both ends of the transmission line to the global ground. Unusual and unexpected behavior can result if these additional elements are omitted.

Equivalent Circuit



Note that any interaction with the ground on either side of the transmission line is ignored. All current on one conductor is equal and opposite to the current on the other at any given distance along the line. Importantly, this condition is enforced regardless of the length of the transmission line or the operating frequency. The exception to this behavior is at DC (Frequency = 0 Hz) where current on the two conductors can be unequal to allow both conductors to be used for biasing active devices. This change in behavior at DC causes a discontinuity of the model parameters at DC. This discontinuity is expected, and additional circuit components relating the voltage at each end of the transmission line to ground should be added, allowing flexibility in implementing the desired transition to the RF-to-DC performance.

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	CX1

Name	Description	Unit Type	Default
Di	Inner diameter	Length	2.903 mm.
Do	Outer diameter	Length	10.297 mm.
L	Length	Length	L[a]
K	Dielectric constant		2.3
A	Loss in dB/meter		0.0833
F	Frequency at which loss is specified	Frequency	0.1 GHz

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See the Microwave Office Layout Guide for details.

Parameter Details

Di and **Do**. These parameters define inner and outer diameters of the internal cross-section of coaxial line. Default values are set to dimensions of RG-213 50 ohm coax.

K. Dielectric constant K and dimensions Di, Do determine the characteristic impedance of the transmission line via the following equation:

$$Z = \frac{138}{\sqrt{K}} \ln\frac{\text{Do}}{\text{Di}}$$

A and **F**. These parameters determine the frequency-dependence of the attenuation constant. If F is not equal to zero (0.0), then

$$\alpha = A\sqrt{\text{freq}/F}$$

(dB/m) where *freq* is the evaluation frequency. If F is equal to zero, the attenuation is zero at all frequencies. Note that attenuation is extrapolated across frequency sweep, so do not set F to an extremely low value.

Implementation Details

The following is a Y-matrix for a grounded transmission line system:

$$[Y_{(2\times2)}] = \begin{bmatrix} Y_{11(2\times2)} & Y_{12(2\times2)} \\ Y_{21(2\times2)} & Y_{22(2\times2)} \end{bmatrix} = \begin{bmatrix} \frac{1}{Z\tanh((\alpha+j\beta)L)} & \frac{-1}{Z\sinh((\alpha+j\beta)L)} \\ \frac{-1}{Z\sinh((\alpha+j\beta)L)} & \frac{1}{Z\tanh((\alpha+j\beta)L)} \end{bmatrix}$$

Applying the equivalent circuit shown above, the Y-matrix of the floating transmission line system can be shown as:

$$[[Y_{(4\times4)}] = \begin{bmatrix} Y_{11(2\times2)} & Y_{12(2\times2)} & -Y_{11(2\times2)} & -Y_{12(2\times2)} \\ Y_{21(2\times2)} & Y_{22(2\times2)} & -Y_{21(2\times2)} & -Y_{22(2\times2)} \\ -Y_{11(2\times2)} & -Y_{12(2\times2)} & Y_{11(2\times2)} & Y_{12(2\times2)} \\ -Y_{21(2\times2)} & -Y_{22(2\times2)} & Y_{21(2\times2)} & Y_{22(2\times2)} \end{bmatrix}$$

At DC, the Y-Matrix changes to a model of two wires above a ground plane:

$$[Y_{(4\times4)}] = \begin{bmatrix} 1/R & -1/R & 0 & 0 \\ -1/R & 1/R & 0 & 0 \\ 0 & 0 & 1/R & -1/R \\ 0 & 0 & -1/R & 1/R \end{bmatrix}$$

where R is a real resistance approaching zero.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

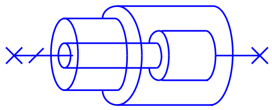
Recommendations for Use

You can use this model along with additional components to model transmission line baluns and transmission line transformers.

NOTE: Because the model definition does not include interactions with the ground, unusual and unexpected results can occur if other components are not used to relate the voltage on both sides of the transmission line to ground.

Step in Coaxial TEM Waveguide: CSTEPIO

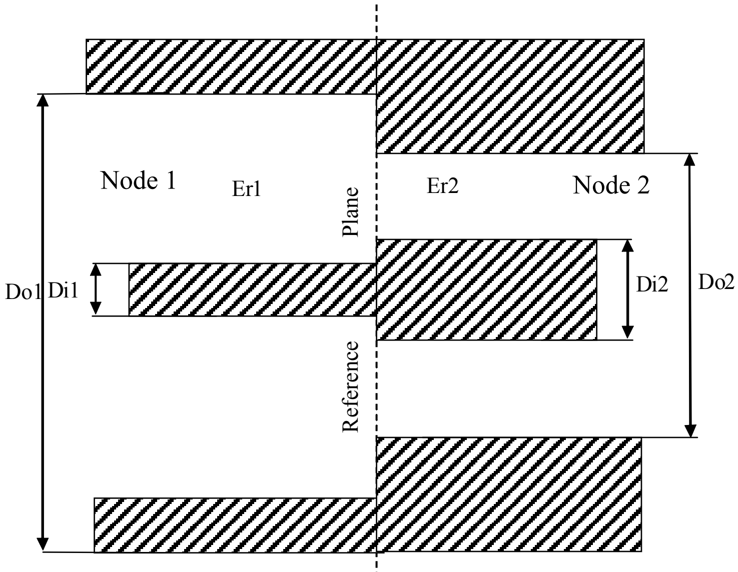
Symbol



Summary

CSTEPIO is a closed form model of a double-step in inner and outer conductors. Both steps share the same cross-section and have a common reference plane. CSTEPIO allows a step in the dielectric constant at the same cross section. Step discontinuity is represented as shunt frequency-independent capacitance.

Topology



Parameters

Name	Description	Unit Type	Default
Di1	Diameter of inner conductor @ Node 1	Length	2.9027 mm
Do1	Diameter of outer conductor @ Node 1	Length	10.287 mm
Di2	Diameter of inner conductor @ Node 2	Length	2.9027 mm
Do2	Diameter of outer conductor @ Node 2	Length	10.287 mm
Er1	Relative dielectric constant filling the coaxial waveguide @ Node 1		1
Er2	Relative dielectric constant filling the coaxial waveguide @ Node 2		1

Parameter Details

Er1, Er2. Dielectric constants of media filling the coaxial waveguide at both sides of the reference plane (see "Topology").

Implementation Details

CSTEPIO is implemented as a closed form approximation of total step capacitance based on [1]. Note that [1] uses the results of a full solution ([2]) to obtain this approximation. The absolute approximation error in capacitance (if $\varepsilon_{r1}=\varepsilon_{r2}$)can be estimated (according to [1]) as $0.18(D_i+D_o)$pF where D_i and D_o stand for averaged inner and outer diameters (expressed in meters). If $\varepsilon_{r1}\neq\varepsilon_{r2}$, then the error may increase because accounting for the inhomogeneous dielectric does not come from the approximation of the full solution, but is rather based on phenomenologically justified assumptions.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Recommendations for Use

CSTEPIO allows any combination of Di1, Do1, Di2, Do2, Er1, Er2. However, the best results are achieved if the following inequalities hold:

$0.01<\alpha<0.7$

$1.5<\tau<6$

For outer step α is $(D_{omin}-D_i)/(D_{omax}-D_i)$. Here D_{omin},D_{omax} stands for the smaller and larger of the outer diameters and D_i stands for each of the inner diameters. α should be tested for both D_i. The τ variable should be evaluated and tested as D_{omax}/D_i for both D_i.

For the inner step, α is $(D_o-D_{imax})/(D_{omax}-D_{imin})$. Here D_{imin}, D_{imax} stands for the smaller and larger of the inner diameters and D_o stands for each of the outer diameters. α should be tested for both D_o. The τ variable should be evaluated and tested as D_o/D_{imin} for both D_o.

This model implies that step capacitance is frequency-independent. According to [1], step capacitance generally grows with frequency and deviates from static value at about 10% if the evaluation frequency exceeds 0.4 Fc, where Fc is the lower cutoff frequency of mode TM_{01} for coaxial waveguides at nodes 1 and 2.

This lower cutoff frequency may be evaluated as the lower value

$$c/((D_o-D_i)\sqrt{\varepsilon})$$

(where c in the nominator is the speed of light, ε stands for dielectric constant, and D_o,D_i are outer and inner diameters) applied in turn to coaxial waveguides at node 1 and node 2.

NOTE: This model is developed to work in a frequency range where only dominant TEM mode propagates.

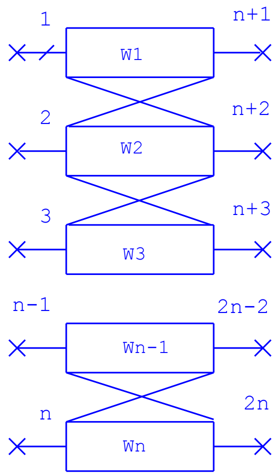
References

[1] Somlo, P.I, "The Computation of Coaxial Line Step Capacitances", IEEE Trans on MTT, vol. 15, No. 1, January 1967, pp. 48-53.

[2] Whinnery J.R, Jamieson H.W, Robins T.E, "Coaxial-lines discontinuities", Proc. of the IRE, vol. 32, November 1944, pp.695-709

Multiple Rectangular Coupled Coaxial Lines (Dynamic Model) (EM Quasi-Static): RCCOAX

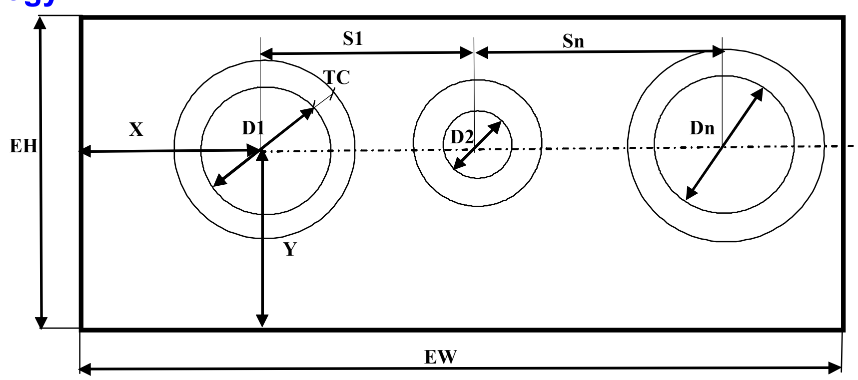
Symbol



Summary

RCCOAX models a section of several (up to 50) coupled dielectric-coated conductors of circular cross-section confined inside a metallic rectangular enclosure filled with lossy dielectric (single conductor also models). Conductors are situated in a row with arbitrary distances between conductors axes. The conductors row is parallel to one pair of enclosure walls and can be arbitrarily positioned within the enclosure. This model can be used for design of machined combline and interdigital filters.

Topology



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	TL1
N	Number of conductors (1..50)		2
L	Conductor length	Length	L[a]
TC	Thickness of coating	Length	L[a]

Name	Description	Unit Type	Default
X	X-coordinate of the leftmost conductor center	Length	L[a]
Y	Y-coordinate of the leftmost conductor center	Length	L[a]
EW	Enclosure width	Length	L[a]
EH	Enclosure height	Length	L[a]
ErC	Dielectric constant of coating material		1
ErF	Dielectric constant of filling material		1
TandC	Loss tangent of coating material		1
TandF	Loss tangent of filling material		1
Rho	Bulk resistance of conductor metal relative to gold		1
Acc	Accuracy parameter (1..10)		1
*SaveToFile	Switch "Save/Do not save data to text file"=Yes/No		"No"
*FileName	Name of text file with computed model parameters	String	Same as model name
Di, i=1..N	Diameter of conductor #i	Length	L[a]
Si, i=1..N-1	Spacing between conductors #i and #i+1 (not available if N=1)	Length	L[a]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See the Microwave Office Layout Guide for details.

** indicates a secondary parameter*

Parameter Details

TC. Conductors may be coated with a lossy dielectric of thickness TC. TC is the same for all conductors.

Rho. Uses bulk resistivity of conductor metal normalized to bulk resistivity of gold ($2.44 \cdot 10^{-8}$ Ω·m). This model implies that the enclosure is made of perfect electrical conductor.

Acc. The accuracy parameter. The default value for Acc is 1. If Acc is less than 1 or greater than 10 it is set automatically to 2.

SaveToFile. This toggled parameter is hidden by default and set to "No". If set to "Yes", the model creates a text file (named "RCCOAX.TXT" by default) at the current project location. This text file contains a table of values of RLGC line parameters at each project frequency. Each row contains RLGC values computed at the frequency specified in the first column (frequency in GHz, R in ohms/m, L in H/m, G in S/m, C in F/m).

The structure of this text file is essentially the same for RCCOAX models at N>2 (see below for special cases N=1 and N=2). This model outputs RLGC matrices to columns that immediately follow the frequency column. Entries of each matrix are placed column-wise, that is first column goes first: R11, R21, R31.. RN1; then the second column: R12, R22, R32,..RN2 etc. The total number of columns in the file is $4N^2+1$, where N is the number of conductors.

- Case N=1.

  If N=1 this model adds complex characteristic impedance Re(Zo), Im(Zo); complex effective dielectric constant Re(Eeff); Im (Eff) (please find traditional effective dielectric constant at column 12); and Loss (dB/m) at columns 2-6. Columns 7-10 contain R, L, G, and C. Column 11 contains the propagation constant Beta in Rad/m. Note that column 12 contains the traditional effective dielectric constant Er_Eff that does not account for loss. The total number of columns in the text file is 12.

- Case N=2

  If N=2 this model implies the existence of two modes, namely, C and P (see [2]) and places additional columns to the output text file. Note that if a system of coupled conductors is fully symmetrical, C-mode corresponds to even mode and P-mode corresponds to odd mode. For N=2 this model adds complex characteristic impedances Re(Zc1), Im(Zc1), Re(Zp1), Im(Zp1), Re(Zc2), Im(Zc2), Re(Zp2), Im(Zp2); complex effective dielectric constants Re(EeffC), Im(EeffC), Re(EeffP), Im(EeffP) (please find traditional effective dielectric constants at columns 38, 39); and losses for C and P modes LossC (dB/m), LossP(dB/m) at columns 2-15. Entries of R, L, G, and C matrices are distributed among columns 16-31 in corresponding order. Columns 32-37 contain Re(Rc), Im(Rc), Re(Rp), Im(Rp), BetaC, BetaP where Rc is the ratio of C - mode voltage in the second line to C - mode voltage in the first line; Rp is the same ratio in the P-mode case (see details in [2], section 4.3.1); BetaC and BetaP are propagation constants of C- and P-modes in Rad/m. Note that columns 38, and 39 contain traditional effective dielectric constants ErC_Eff, ErP_Eff (they do not account for losses). The total number of columns in the text file is 39. The created text file may be linked or imported to a project as a data file, and the frequency behavior of any of these parameters may be viewed using the proper data measurement. Note that the first column (frequency) is always in GHz so these measurements might be incompatible with other measurements placed on the same graph; thus, you might prefer to place the aforementioned data measurements on a separate dedicated graph.

FileName. By default this parameter is hidden and is set to model "*RCCOAX.TXT*". You can change the file name for each model instance to an arbitrary name with a length not exceeding 64 symbols.

Parameter Restrictions and Recommendations

1. The number of conductors N cannot exceed 50.
2. Accuracy parameter A is limited to $1 \le Acc \le 10$. A larger value of Acc increases the density of mesh used in computations. The accuracy of model parameters may gain slightly from increasing Acc at the expense of a noticeable growth of computation time. A good trade-off between accuracy and computation time is to set Acc to 1.
3. This model checks the positions of conductors inside the enclosure and issues error messages if any conductors touch other conductors or enclosure walls.

Implementation Details

Model implementation is based on the EM Quasi-Static technique described in [1] [17–29]. It accounts for losses in metal and in dielectrics. Dispersion is partly included.

Recommendations for Use

NOTE: The implementation of EM Quasi-Static models relies heavily on the involved numerical algorithms. This may lead to a noticeable increase in simulation time for schematics that employ many such models.

Caution regarding units of data in saved text files: If a project that reads the saved text files uses frequency, resistance, inductance or conductance units different from GHz, Ohm, Henry or Siemens, you may need to scale input values manually.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

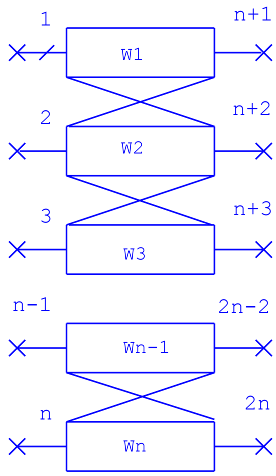
References

[1] M.B. Bazdar, A.R. Djordjevic, R.F. Harrington, and T.K. Sarkar, "Evaluation of quasi-static matrix parameters for multiconductor transmission lines using Galerkin's method," IEEE Trans. Microwave Theory Tech., vol. MTT-42, July 1994, pp. 1223-1228.

[2] R. Mongia, I. Bahl, and P. Bhartia, RF and Microwave Coupled-Line Circuits, Artech House, Norwood, MA, 1999.

Multiple Rectangular Slot Coupled Coaxial Lines (Dynamic Model) (EM Quasi-Static): RCCOAX2

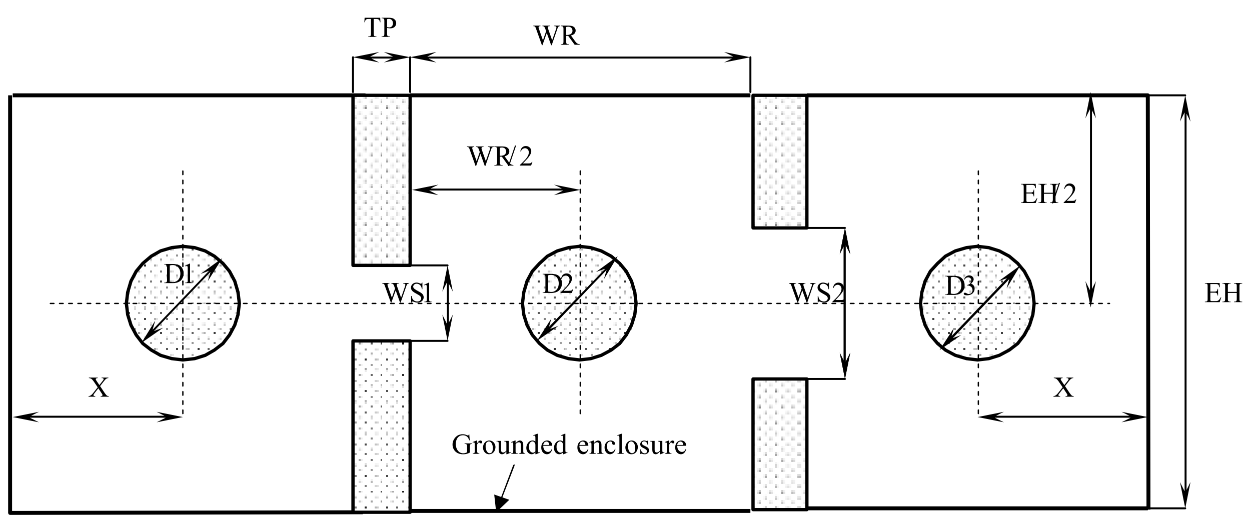
Symbol



Summary

RCCOAX2 models a section of several (2 through 50) coupled conductors of circular cross-section confined inside a metallic rectangular enclosure filled with lossy dielectric. Each conductor is located inside a resonator made up by metallic slotted partitions. You can use slot width to control coupling between conductors. This model can be used for design of machined combline and interdigital filters.

Topology



Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	TL1
N	Number of conductors (1..50)		2
L	Conductor length	Length	L[a]
X	Distance from the enclosure side wall to the nearest conductor center	Length	L[a]
WR	Resonator width"	Length	L[a]
TP	Thickness of partition between resonators"	Length	L[a]
EH	Enclosure height	Length	L[a]
ErF	Dielectric constant of filling material		1
TandF	Loss tangent of filling material		1
Rho	Bulk resistivity of conductor metal relative to gold		1
Acc	Accuracy parameter (1..10)		1
*SaveToFile	Switch "Save/Do not save data to text file"=Yes/No		"No"
*FileName	Name of text file with computed model parameters	String	Same as model name
Di, i=1..N	Diameter of conductor #i	Length	L[a]
WSi, i=1..N-1	Width of slot between conductors/resonators #i and #i+1	Length	L[a]

[a]User-modifiable default. Modify by editing under $DEFAULT_VALUES in the *default.lpf* file in the root installation directory. See the Microwave Office Layout Guide for details.

** indicates a secondary parameter*

Parameter Details

Rho. Uses bulk resistivity of conductor metal normalized to bulk resistivity of gold ($2.44 \cdot 10^{-8}$ Ω·m). This model implies that the enclosure is made of perfect electrical conductor.

Acc. The accuracy parameter. The default value for Acc is 1. If Acc is less than 1 or greater than 10 it is set automatically to 2.

SaveToFile. This toggled parameter is hidden by default and set to "No". If set to "Yes", the model creates a text file (named "*RCCOAX2.TXT*" by default) at the current project location. This text file contains a table of values of RLGC line parameters at each project frequency. Each row contains RLGC values computed at the frequency specified in the first column (frequency in GHz, R in ohms/m, L in H/m, G in S/m, C in F/m).

The structure of this text file is essentially the same for RCCOAX2 models at N>2 (see below for special case N=2). This model outputs RLGC matrices to columns that immediately follow the frequency column. Entries of each matrix are placed column-wise, that is first column goes first: R11, R21, R31.. RN1; then the second column: R12, R22, R32,..RN2 etc. The total number of columns in the file is $4N^2+1$, where N is the number of conductors.

- Case N=2

  If N=2 this model implies the existence of two modes, namely, C and P (see [2]) and places additional columns to the output text file. Note that if a system of coupled conductors is fully symmetrical, C-mode corresponds to even mode and P-mode corresponds to odd mode. For N=2 this model adds complex characteristic impedances Re(Zc1), Im(Zc1), Re(Zp1), Im(Zp1), Re(Zc2), Im(Zc2), Re(Zp2), Im(Zp2); complex effective dielectric constants Re(EeffC), Im(EeffC), Re(EeffP), Im(EeffP) (please find traditional effective dielectric constants at columns 38, 39); and losses for C and P modes LossC (dB/m), LossP(dB/m) at columns 2-15. Entries of R, L, G, and C matrices are distributed among columns 16-31 in corresponding order. Columns 32-37 contain Re(Rc), Im(Rc), Re(Rp), Im(Rp), BetaC, BetaP where Rc is the ratio of C - mode voltage in the second line to C - mode voltage in the first line; Rp is the same ratio in the P-mode case (see details in [2], section 4.3.1); BetaC and BetaP are propagation constants of C- and P-modes in Rad/m. Note that columns 38, and 39 contain traditional effective dielectric constants ErC_Eff, ErP_Eff (they do not account for losses). The total number of columns in the text file is 39. The created text file may be linked or imported to a project as a data file, and the frequency behavior of any of these parameters may be viewed using the proper data measurement. Note that the first column (frequency) is always in GHz so these measurements might be incompatible with other measurements placed on the same graph; thus, you might prefer to place the aforementioned data measurements on a separate dedicated graph.

FileName. By default this parameter is hidden and is set to model "*RCCOAX2.TXT*". You can change the file name for each model instance to an arbitrary name with a length not exceeding 64 symbols.

Parameter Restrictions and Recommendations

1. The number of conductors N must be greater than or equal to 2. Also, N cannot exceed 50.
2. Accuracy parameter A is limited to 1≤Acc≤10 . A larger value of Acc increases the density of mesh used in computations. The accuracy of model parameters may gain slightly from increasing Acc at the expense of a noticeable growth of computation time. A good trade-off between accuracy and computation time is to set Acc to 1.
3. This model checks the positions of conductors inside the enclosure and issues error messages if any conductors touch other conductors or enclosure walls.

Implementation Details

Model implementation is based on the EM Quasi-Static technique described in [1] [17–32]. It accounts for losses in metal and in dielectrics. Dispersion is partly included.

Recommendations for Use

NOTE: The implementation of EM Quasi-Static models relies heavily on the involved numerical algorithms. This may lead to a noticeable increase in simulation time for schematics that employ many such models.

Caution regarding units of data in saved text files: If a project that reads the saved text files uses frequency, resistance, inductance or conductance units different from GHz, Ohm, Henry or Siemens, you may need to scale input values manually.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

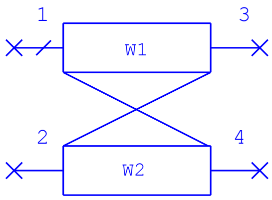
References

[1] M.B. Bazdar, A.R. Djordjevic, R.F. Harrington, and T.K. Sarkar, "Evaluation of quasi-static matrix parameters for multiconductor transmission lines using Galerkin's method," IEEE Trans. Microwave Theory Tech., vol. MTT-42, July 1994, pp. 1223-1228.

[2] R. Mongia, I. Bahl, and P. Bhartia, RF and Microwave Coupled-Line Circuits, Artech House, Norwood, MA, 1999.

Two RLGC Asymmetric Coupled Transmitting Lines (Closed Form): RLGC_CTx

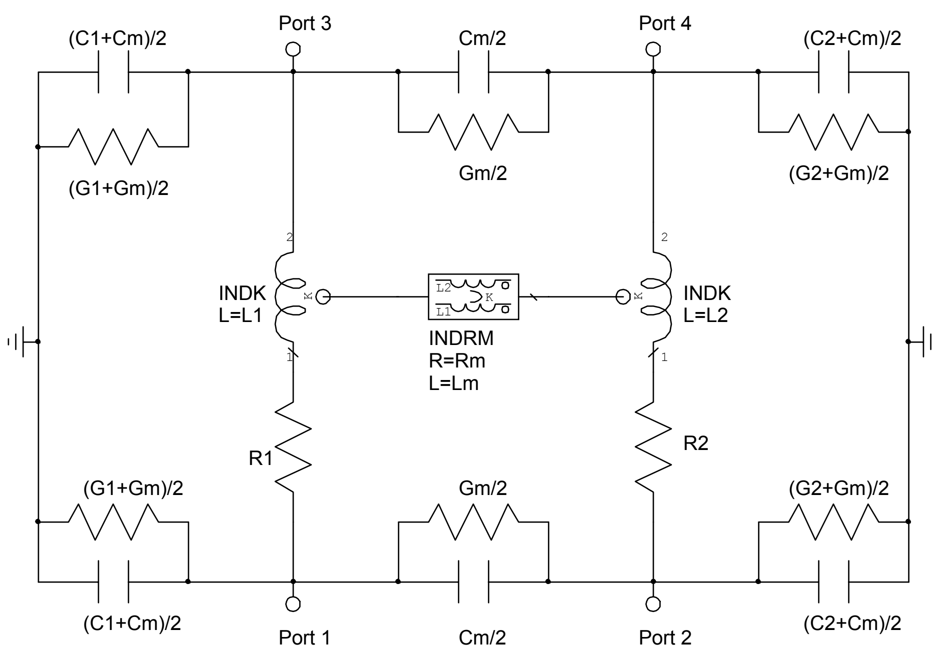
Symbol



Summary

RLGC_CTx provides computation of circuit parameters for pairs of general asymmetric coupled transmission lines whose behavior in a frequency domain is governed by user-supplied tables of primary per-unit-length line parameters R, L, G, and C. Model assumes that G and C parameters are obtained from some EM solver and are defined for ground referenced potentials.

Equivalent Circuit



Parameters

Name	Description	Unit Type	Default
ID	Name	Text	TL1
Len	Length of Line	Length	L[a]
F	Vector of Frequencies at which L, C, R, G specified	Frequency	1 GHz

Name	Description	Unit Type	Default
R1	Vector of Series R1 per-unit-length	Resistance/meter	100 ohms
R2	Vector of Series R2 per-unit-length	Resistance/meter	100 ohms
Rm	Vector of mutual Rm per-unit-length	Resistance/mete	0
L1	Vector of Series L1 per-unit-length	Inductance/meter	Li[a]
L2	Vector of Series L2 per-unit-length	Inductance/meter	Li[a]
Lm	Vector of mutual Lm per-unit-length	Inductance/meter	Li[a] nH
G1	Vector of Shunt G1 per-unit-length	Conductance/meter	0
G2	Vector of Shunt G2 per-unit-length	Conductance/meter	0
Gm	Vector of mutual Gm per-unit-length	Conductance/meter	0
C1	Vector of Shunt C1 per-unit-length	Capacitance/meter	C[a] pF
C2	Vector of Shunt C2 per-unit-length	Capacitance/meter	C[a] pF
Cm	Vector of mutual Cm per-unit-length	Capacitance/meter	0
*ChkPassv	Switch "Do not Check passivity/Check passivity"		No

[a] User-modifiable default. Modify by editing the *DEFAULT.LPF* file in the root installation directory. See “Default Values” for details.

** indicates a secondary parameter*

Parameter Details

F. Vector of frequencies at which R, L, G, and C parameters are specified. Frequencies must be sequential and specified in ascending order.

R1, R2. Vectors of series resistances (see Equivalent Circuit) per-unit-length specified in resistance project units per meter. Each vector entry must be specified at the corresponding frequency entry from frequency vector **F.**

Rm. Vectors of mutual resistances (see Equivalent Circuit) per-unit-length specified in resistance project units per meter. Each vector entry must be specified at the corresponding frequency entry from frequency vector **F.**

L1, L2. Vectors of series inductances (see Equivalent Circuit) per-unit-length specified in inductance project units per meter. Each vector entry must be specified at the corresponding frequency entry from frequency vector **F.**

Lm. Vectors of mutual inductances (see Equivalent Circuit) per-unit-length specified in inductance project units per meter. Each vector entry must be specified at the corresponding frequency entry from frequency vector **F.**

G1, G2. Vector of shunt "electromagnetic" (see explanation below) conductances (see Equivalent Circuit) per-unit-length specified in conductance project units per meter. Each vector entry must be specified at the corresponding frequency entry from frequency vector **F.**

Gm. Vectors of mutual "electromagnetic" (see explanation below) conductances (see Equivalent Circuit) per-unit-length specified in conductance project units per meter. Each vector entry must be specified at the corresponding frequency entry from frequency vector **F.**

C1, C2. Vector of shunt "electromagnetic" (see explanation below) capacitances (see Equivalent Circuit) per-unit-length specified in capacitance project units per meter. Each vector entry must be specified at the corresponding frequency entry from frequency vector **F.**

Cm. Vectors of mutual "electromagnetic" (see explanation below) capacitances (see Equivalent Circuit) per-unit-length specified in capacitance project units per meter. Each vector entry must be specified at the corresponding frequency entry from frequency vector **F.**

ChkPassv. If set to Yes, tests RLGC_CTx for passivity and displays a warning if there is a passivity violation. The default setting is No (Do not check passivity).

Explanation of "electromagnetic" parameters C and G

Model implies that per-unit-length parameters C_1,C_2,$C_m$and G_1,G_2,G_m are "electromagnetic" parameters, that is, parameters defined as contributions to total charges at each conductor of potentials **between conductor and common ground.** In this case, Cm is always negative (Cm<0) and Gm is negative (Gm<0) in most cases. Example of these parameters can be seen in the text file optionally created by model GM2CLIN (two asymmetric coupled lines).

Sometimes, different set of so called "circuit" RLGC parameters might be available: That is, G and C parameters might be defined as contributions (to total charges at each conductor)of **relative** potentials **between conductors** . These capacitances and admittances comprise the equivalent circuit shown at the Figure Equivalent Circuit (see above).

Let us assume that we obtained somehow a set of circuit parameters. We need to obtain electromagnetic parameters to feed to RLGC_CTx. Denote circuit parameters as

$$\tilde{C}_1,\ \tilde{C}_2,\ \tilde{C}_m$$

$$\tilde{G}_1,\ \tilde{G}_2,\ \tilde{G}_m$$

Denote desirable electromagnetic parameters as C_1,C_2,$C_m$and G_1,G_2,G_m. Electromagnetic and circuit parameters are related by the following simple formulas:

$$C_1 = \tilde{C}_1 + \tilde{C}_m$$

$$C_2 = \tilde{C}_2 + \tilde{C}_m$$

$$C_m = -\tilde{C}_m$$

$$G_1 = \tilde{G}_1 + \tilde{G}_m$$

$$G_2 = \tilde{G}_2 + \tilde{G}_m$$

$$G_m = -\tilde{G}_m$$

Parameter Restrictions and Recommendations

1. Lengths of vector parameters R, L, G, and C must be exactly equal to the length of frequency vector F.
2. Model checks if all entries in vectors R1, R2, L1, L2, C1, and C2 are positive. Vectors Rm, Lm, Gm, and Cm are not checked for sign because as a rule of thumb Cm<0, Lm>0, Rm>=0, and Gm<=0. It may also happen that Rm<0. Thus it is user responsibility to provide sanity check of each entry of mutual parameters.

3. If project operational frequency is out of range of frequencies in F then R, L, G, and C parameters are extrapolated as constant values equal to the first/last entries of corresponding vectors. No warning is issued.
4. Vector can be specified in three ways: First, it can be entered as a right side value of model parameter, e.g. R={100,102,110,113,120}; Second, vector can be specified elsewhere in equation; Third way is specification of vector in a column or row of a text file. Third way provides a convenient and flexible method of specification of all RLGC parameters in a single location. Just create, for example, file RLGC.txt containing space separated columns of R1, R2 etc. First column must represent frequency in project units (note that changing of project default frequency units will demand manual scaling of frequencies in this file). Import or link this file to your project and give it a name, say, RLGC_15. Now you can specify, say, parameter R1 as R1 = Col(datafile("RLGC_15"),1). It means that values of vector R1 will be copied to the model from the column 1 of file RLGC.txt imported under name RLGC_15. If you prefer to deploy your data row-wise use R1 = Row(datafile("RLGC_15"),1).
5. If your project uses text file input to feed data to this model be aware what frequency, resistance, inductance or conductance units this file implies. There is a chance that default units of your project differ from those in your data file. If this happens, you must scale input values multiplying call of function Col or Row by scaling coefficient. For example, if your project uses capacitances in picofarads and data file contains data in Farads you may get capacitance data from the fifth column of data file RLGC_15 like this: C1= 1e+12*Col(datafile("RLGC_15"),5).
6. Equivalent circuit (see above) is based on Microwave Office elements INDK and INDRM. This circuit provides very accurate representation of electrical behavior of short asymmetric coupled lines. Note that each RLGC parameter used in this equivalent circuit for purpose of modeling must be multiplied by line length. Remember that RLGC values must be in ohm/m, mH/m, pF/m etc and line length must be in meters.

Implementation Details

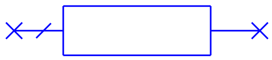
Model implementation is based on linear interpolation of RLGC parameters at each project evaluation frequency. Interpolation uses user-supplied via parameters look-out tables. If project operational frequency is out of range of frequencies in F then R, L, G, and C parameters are extrapolated as constant values equal to the first/last entries of corresponding vectors.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

RLGC Transmitting Line (Closed Form): RLGC_Tx

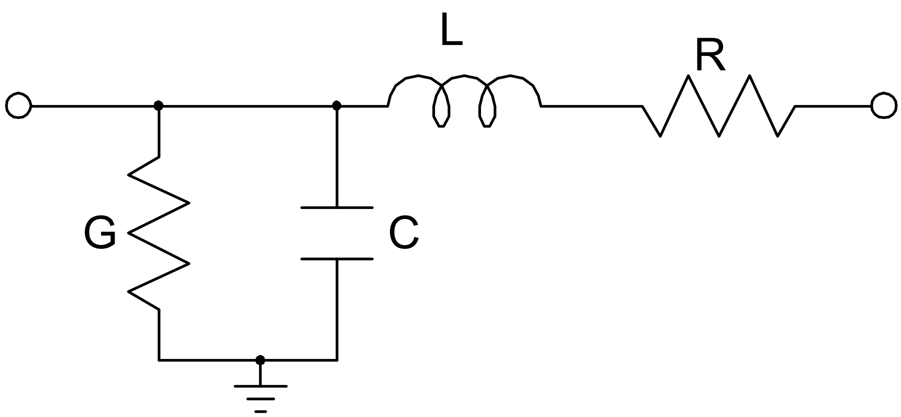
Symbol



Summary

RLGC_Tx provides computation of circuit parameters for general transmission lines whose behavior in the frequency domain is governed by a user-supplied table of primary per-unit-length line parameters R, L, G, and C.

Equivalent Circuit



Parameters

Name	Description	Unit Type	Default
ID	Name	Text	TL1
Len	Length of Line	Length	L[a]
F	Vector of Frequencies at which L, C, R, G specified	Frequency	1 GHz
R	Vector of Series R per-unit-length	Resistance/meter	100
L	Vector of Series L per-unit-length	Inductance/meter	Li[a]
G	Vector of Shunt G per-unit-length	Conductance/meter	0
C	Vector of Shunt C per-unit-length	Capacitance/meter	C[a]

[a] User-modifiable default. Modify by editing the *DEFAULT.LPF* file in the root installation directory. See the “Default Values” for details.

Parameter Details

F. Vector of frequencies at which R, L, G, and C parameters are specified. Frequencies must be sequential and specified in ascending order.

R. Vector of series resistance (see Equivalent Circuit) per-unit-length specified in resistance project units per meter. Each vector entry must be specified at the corresponding frequency entry from frequency vector >F.

L. Vector of series inductance (see Equivalent Circuit) per-unit-length specified in inductance project units per meter. Each vector entry must be specified at the corresponding frequency entry from frequency vector F.

G. Vector of shunt conductance (see Equivalent Circuit) per-unit-length specified in conductance project units per meter. Each vector entry must be specified at the corresponding frequency entry from frequency vector F.

C. Vector of shunt capacitance (see Equivalent Circuit) per-unit-length specified in capacitance project units per meter. Each vector entry must be specified at the corresponding frequency entry from frequency vector F.

Parameter Restrictions and Recommendations

1. Size of vector parameters R, L, G, and C must be exactly equal to the size of frequency vector F.
2. If project evaluation frequency is out of range of frequencies in F then R, L, G, and C parameters are extrapolated as constant values equal to the first/last entries of corresponding vectors. No warning is issued.
3. Vector can be specified in three ways: First, it can be entered as a right side value of model parameter, e.g. R={100,102,110,113,120}; Second, vector can be specified elsewhere in equation; Third way is specification of vector in a column or row of a text file; Third way provides a convenient and flexible method of specification of all RLGC parameters in the single location. Just create for example file RLGC.txt containing space separated columns of R1, R2 etc. First column must represent frequency in project units (note that changing of project default frequency units will demand manual scaling of frequencies in this file). Import or link this file to your project and give it a name, say, RLGC_15. Now you can specify, say, parameter R as R = Col(datafile("RLGC_15"),2). This means that values of vector R will be copied to the model from the column 2 of file RLGC.txt imported under name RLGC_15. If you prefer to deploy your data row-wise use R = Row(datafile("RLGC_15"),2).
4. If your project uses text file input to feed data to this model be aware what frequency, resistance, inductance or conductance units this file implies. There is a chance that default units of your project differ from those in your data file. If this happens, you must scale input values multiplying call of function Col or Row by scaling coefficient. For example, if your project uses capacitances in picofarads and data file contains data in Farads you may get capacitance data from the column 5 of data file RLGC_15 like this: C1= 1e+12*Col(datafile("RLGC_15"),5).

Implementation Details

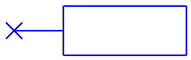
Model implementation is based on linear interpolation of RLGC parameters at each project evaluation frequency. Interpolation uses lookup tables user-supplied via parameters. If project evaluation frequency is out of range of frequencies in F then R, L, G, and C parameters are extrapolated as constant values equal to the first/last entries of corresponding vectors.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Open Circuited RLGC Transmitting Line (Closed Form): RLGC_TxO

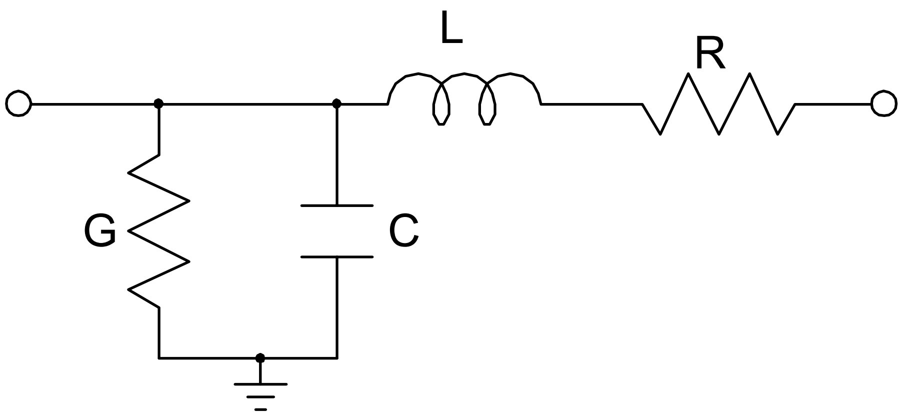
Symbol



Summary

RLGC_TxO provides computation of circuit parameters for general open-circuited transmission lines whose behavior in a frequency domain is governed by a user-supplied table of primary per-unit-length line parameters R, L, G, and C.

Equivalent Circuit



Parameters

Name	Description	Unit Type	Default
ID	Name	Text	TL1
Len	Length of Line	Length	L[a]
F	Vector of Frequencies at which L, C, R, G specified	Frequency	1 GHz
R	Vector of Series R per-unit-length	Resistance/meter	100
L	Vector of Series L per-unit-length	Inductance/meter	Li[a]
G	Vector of Shunt G per-unit-length	Conductance/meter	0
C	Vector of Shunt C per-unit-length	Capacitance/meter	C[a]

[a] User-modifiable default. Modify by editing the *DEFAULT.LPF* file in the root installation directory. See "Default Values" for details.

Parameter Details

F. Vector of frequencies at which R, L, G, and C parameters are specified. Frequencies must be sequential and specified in ascending order.

R. Vector of series resistance (see Equivalent Circuit) per-unit-length specified in resistance project units per meter. Each vector entry must be specified at the corresponding frequency entry from frequency vector **F.**

L. Vector of series inductance (see Equivalent Circuit) per-unit-length specified in inductance project units per meter. Each vector entry must be specified at the corresponding frequency entry from frequency vector F.

G. Vector of shunt conductance (see Equivalent Circuit) per-unit-length specified in conductance project units per meter. Each vector entry must be specified at the corresponding frequency entry from frequency vector **F.**

C. Vector of shunt capacitance (see Equivalent Circuit) per-unit-length specified in capacitance project units per meter. Each vector entry must be specified at the corresponding frequency entry from frequency vector **F.**

Parameter Restrictions and Recommendations

1. Lengths of vector parameters R, L, G, and C must be exactly equal to the length of frequency vector F.
2. If project operational frequency is out of range of frequencies in F then R, L, G, and C parameters are extrapolated as constant values equal to the first/last entries of corresponding vectors. No warning is issued.
3. Vector can be specified in three ways: First, it can be entered as a right side value of model parameter, e.g. R={100,102,110,113,120}; Second, vector can be specified elsewhere in equation; Third way is specification of vector in a column or row of a text file. Third way provides a convenient and flexible method of specification of all RLGC parameters in the single location. Just create, for example, file RLGC.txt containing space separated columns of R1, R2 etc. First column must represent frequency in project units (note that changing of project default frequency units will demand manual scaling of frequencies in this file). Import or link this file to your project and give it a name, say, RLGC_15. Now you can specify, say, parameter R as R = Col(datafile("RLGC_15"),1). It means that values of vector R will be copied to the model from the column 1 of file RLGC.txt imported under name RLGC_15. If you prefer to deploy your data row-wise use R = Row(datafile("RLGC_15"),1).
4. If your project uses text file input to feed data to this model be aware what frequency, resistance, inductance or conductance units this file implies. There is a chance that default units of your project differ from those in your data file. If this happens, you must scale input values multiplying call of function Col or Row by scaling coefficient. For example, if your project uses capacitances in picofarads and data file contains data in Farads you may get capacitance data from the fifth column of data file RLGC_15 like this: C1= 1e+12*Col(datafile("RLGC_15"),5).

Implementation Details

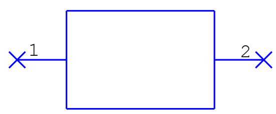
Model implementation is based on linear interpolation of RLGC parameters at each project evaluation frequency. Interpolation uses user-supplied via parameters look-out tables. If project operational frequency is out of range of frequencies in F then R, L, G, and C parameters are extrapolated as constant values equal to the first/last entries of corresponding vectors.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

Open Circuited RLGC Transmitting Line with Floating Ground (Closed Form): RLGC_TxO2

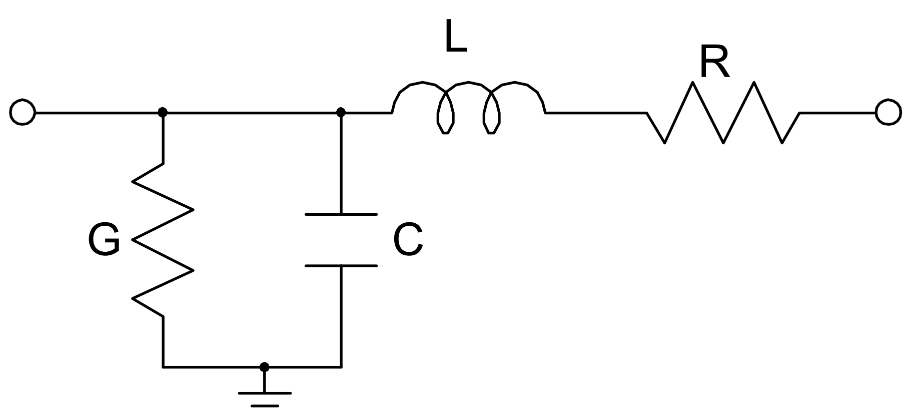
Symbol



Summary

RLGC_TxO2 simulates a general open circuited transmission line with isolated ground terminals. This model provides computation of circuit parameters for general transmission line whose behavior in frequency domain is governed by user supplied table of primary per-unit-length line parameters R, L, G, and C.Extreme care should be used with this element as it is meant to work in concert with additional elements which relate the voltages at the end of the transmission line to the global ground. Unusual and unexpected behavior can result if these additional elements are omitted. See the RLGC_Tx4 model documentation for implementation details.

Equivalent Circuit of Single Line



Parameters

Name	Description	Unit Type	Default
ID	Name	Text	TL1
Len	Length of Line	Length	L[a]
F	Vector of Frequencies at which L, C, R, G specified	Frequency	1 GHz
R	Vector of Series R per-unit-length	Resistance/meter	100
L	Vector of Series L per-unit-length	Inductance/meter	Li[a]
G	Vector of Shunt G per-unit-length	Conductance/meter	0
C	Vector of Shunt C per-unit-length	Capacitance/meter	C[a]

[a] User-modifiable default. Modify by editing the *DEFAULT.LPF* file in the root installation directory. See "Default Values" for details.

Parameter Details

F. Vector of frequencies at which R, L, G, and C parameters are specified. Frequencies must be sequential and specified in ascending order.

R. Vector of series resistance (see Equivalent Circuit) per-unit-length specified in resistance project units per meter. Each vector entry must be specified at the corresponding frequency entry from frequency vector **F.**

L. Vector of series inductance (see Equivalent Circuit) per-unit-length specified in inductance project units per meter. Each vector entry must be specified at the corresponding frequency entry from frequency vector F.

G. Vector of shunt conductance (see Equivalent Circuit) per-unit-length specified in conductance project units per meter. Each vector entry must be specified at the corresponding frequency entry from frequency vector **F.**

C. Vector of shunt capacitance (see Equivalent Circuit) per-unit-length specified in capacitance project units per meter. Each vector entry must be specified at the corresponding frequency entry from frequency vector **F.**

Parameter Restrictions and Recommendations

1. Lengths of vector parameters R, L, G, and C must be exactly equal to the length of frequency vector F.
2. If project operational frequency is out of range of frequencies in F then R, L, G, and C parameters are extrapolated as constant values equal to the first/last entries of corresponding vectors. No warning is issued.
3. Vector can be specified in three ways: First, it can be entered as a right side value of model parameter, e.g. R={100,102,110,113,120}; Second, vector can be specified elsewhere in equation; Third way is specification of vector in a column or row of a text file. Third way provides a convenient and flexible method of specification of all RLGC parameters in the single location. Just create, for example, file RLGC.txt containing space separated columns of R1, R2 etc. First column must represent frequency in project units (note that changing of project default frequency units will demand manual scaling of frequencies in this file). Import or link this file to your project and give it a name, say, RLGC_15. Now you can specify, say, parameter R as R = Col(datafile("RLGC_15"),1). It means that values of vector R will be copied to the model from the column 1 of file RLGC.txt imported under name RLGC_15. If you prefer to deploy your data row-wise use R = Row(datafile("RLGC_15"),1).
4. If your project uses text file input to feed data to this model be aware what frequency, resistance, inductance or conductance units this file implies. There is a chance that default units of your project differ from those in your data file. If this happens, you must scale input values multiplying call of function Col or Row by scaling coefficient. For example, if your project uses capacitances in picofarads and data file contains data in Farads you may get capacitance data from the fifth column of data file RLGC_15 like this: C1= 1e+12*Col(datafile("RLGC_15"),5).

Implementation Details

Model implementation is based on linear interpolation of RLGC parameters at each project evaluation frequency. Interpolation uses user-supplied via parameters look-out tables. If project operational frequency is out of range of frequencies in F then R, L, G, and C parameters are extrapolated as constant values equal to the first/last entries of corresponding vectors.

The following is a Y-matrix for a grounded transmission line system:

$$[Y_{(2\times2)}] = \begin{bmatrix} Y_{11(2\times2)} & Y_{12(2\times2)} \\ Y_{21(2\times2)} & Y_{22(2\times2)} \end{bmatrix} = \begin{bmatrix} \dfrac{1}{Z\tanh((\alpha+j\beta)L)} & \dfrac{-1}{Z\sinh((\alpha+j\beta)L)} \\ \dfrac{-1}{Z\sinh((\alpha+j\beta)L)} & \dfrac{1}{Z\tanh((\alpha+j\beta)L)} \end{bmatrix}$$

where α+jβ represents the complex propagation constant, Z is the characteristic impedance of the line and L is the length of the line as derived from the input parameters.

Applying the equivalent circuit shown above, the Y-matrix of the floating transmission line system can be shown to be the following:

$$[[Y_{(4\times4)}] = \begin{bmatrix} Y_{11(2\times2)} & Y_{12(2\times2)} & -Y_{11(2\times2)} & -Y_{12(2\times2)} \\ Y_{21(2\times2)} & Y_{22(2\times2)} & -Y_{21(2\times2)} & -Y_{22(2\times2)} \\ -Y_{11(2\times2)} & -Y_{12(2\times2)} & Y_{11(2\times2)} & Y_{12(2\times2)} \\ -Y_{21(2\times2)} & -Y_{22(2\times2)} & Y_{21(2\times2)} & Y_{22(2\times2)} \end{bmatrix}$$

At DC, the Y-Matrix changes to a model of two wires above a ground plane:

$$[Y_{(4\times4)}] = \begin{bmatrix} 1/R & -1/R & 0 & 0 \\ -1/R & 1/R & 0 & 0 \\ 0 & 0 & 1/R & -1/R \\ 0 & 0 & -1/R & 1/R \end{bmatrix}$$

where R is a real resistance approaching zero.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

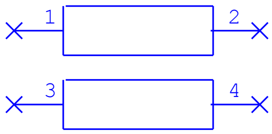
Recommendations for Use

This model, along with additional components can be used to model transmission line baluns and transmission line transformers in which one of the conductors is shielded from ground, like a coaxial line.

NOTE: Because the model definition does not include interactions with the ground, unusual and unexpected results can occur if other components are not used to relate the voltage on both sides of the transmission line to ground.

RLGC Transmitting Line with Floating Ground (Closed Form): RLGC_Tx4

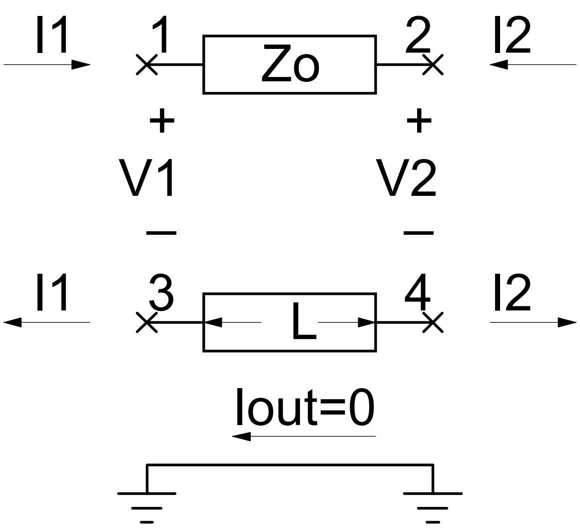
Symbol



Summary

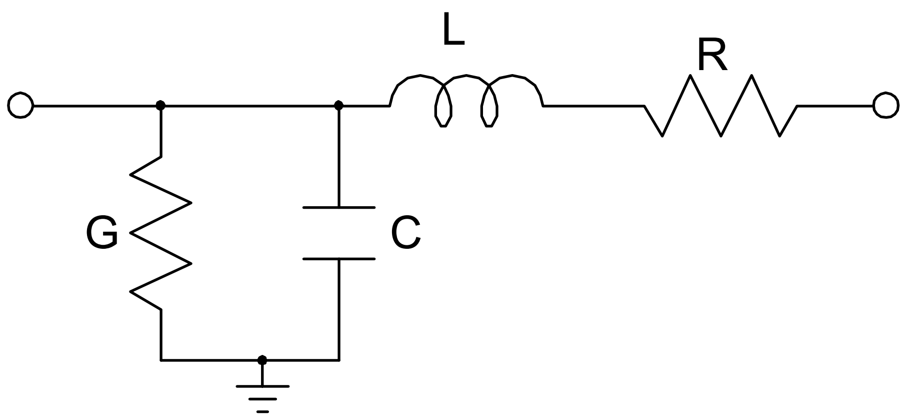
RLGC_Tx4 simulates a general transmission line with isolated ground terminals. This model provides computation of circuit parameters for general transmission lines whose behavior in frequency domain is governed by a user-supplied table of primary per-unit-length line parameters R, L, G, and C. Extreme care should be used with this element as it is meant to work in concert with additional elements which relate the voltages at both ends of the transmission line to the global ground. Unusual and unexpected behavior can result if these additional elements are omitted.

Equivalent Circuit (Lines and Currents)



Note that any interaction with the ground on either side of the transmission line is ignored. All current on one conductor is equal and opposite to the current on the other at any given distance along the line. Importantly, this condition is enforced regardless of the length of the transmission line or the operating frequency. The exception to this behavior is at DC (Frequency = 0 Hz) where current on the two conductors can be unequal to allow both conductors to be used for biasing active devices. This change in behavior at DC causes a discontinuity of the model parameters at DC. This discontinuity is expected, and additional circuit components relating the voltage at each end of the transmission line to ground should be added, allowing flexibility in implementing the desired transition to the RF-to-DC performance.

Equivalent Circuit of Single Line



Parameters

Name	Description	Unit Type	Default
ID	Name	Text	TL1
Len	Length of Line	Length	L[a]
F	Vector of Frequencies at which L, C, R, G specified	Frequency	1 GHz
R	Vector of Series R per-unit-length	Resistance/meter	100
L	Vector of Series L per-unit-length	Inductance/meter	Li[a]
G	Vector of Shunt G per-unit-length	Conductance/meter	0
C	Vector of Shunt C per-unit-length	Capacitance/meter	C[a]

[a]User-modifiable default. Modify by editing the *DEFAULT.LPF* file in the root installation directory. See "Default Values" for details.

Parameter Details

F. Vector of frequencies at which R, L, G, and C parameters are specified. Frequencies must be sequential and specified in ascending order.

R. Vector of series resistance (see Equivalent Circuit) per-unit-length specified in resistance project units per meter. Each vector entry must be specified at the corresponding frequency entry from frequency vector **F.**

L. Vector of series inductance (see Equivalent Circuit) per-unit-length specified in inductance project units per meter. Each vector entry must be specified at the corresponding frequency entry from frequency vector F.

G. Vector of shunt conductance (see Equivalent Circuit) per-unit-length specified in conductance project units per meter. Each vector entry must be specified at the corresponding frequency entry from frequency vector **F.**

C. Vector of shunt capacitance (see Equivalent Circuit) per-unit-length specified in capacitance project units per meter. Each vector entry must be specified at the corresponding frequency entry from frequency vector **F.**

Parameter Restrictions and Recommendations

1. Lengths of vector parameters R, L, G, and C must be exactly equal to the length of frequency vector F.
2. If project operational frequency is out of range of frequencies in F then R, L, G, and C parameters are extrapolated as constant values equal to the first/last entries of corresponding vectors. No warning is issued.
3. Vector can be specified in three ways: First, it can be entered as a right side value of model parameter, e.g. R={100,102,110,113,120}; Second, vector can be specified elsewhere in equation; Third way is specification of vector in a column or row of a text file. Third way provides a convenient and flexible method of specification of all RLGC parameters in the single location. Just create, for example, file RLGC.txt containing space separated columns of R1, R2 etc. First column must represent frequency in project units (note that changing of project default frequency units will demand manual scaling of frequencies in this file). Import or link this file to your project and give it a name, say, RLGC_15. Now you can specify, say, parameter R as R = Col(datafile("RLGC_15"),1). It means that values of vector R will be copied to the model from the column 1 of file RLGC.txt imported under name RLGC_15. If you prefer to deploy your data row-wise use R = Row(datafile("RLGC_15"),1).
4. If your project uses text file input to feed data to this model be aware what frequency, resistance, inductance or conductance units this file implies. There is a chance that default units of your project differ from those in your data file. If this happens, you must scale input values multiplying call of function Col or Row by scaling coefficient. For example, if your project uses capacitances in picofarads and data file contains data in Farads you may get capacitance data from the fifth column of data file RLGC_15 like this: C1= 1e+12*Col(datafile("RLGC_15"),5).

Implementation Details

Model implementation is based on linear interpolation of RLGC parameters at each project evaluation frequency. Interpolation uses user-supplied via parameters look-out tables. If project operational frequency is out of range of frequencies in F then R, L, G, and C parameters are extrapolated as constant values equal to the first/last entries of corresponding vectors.

The following is a Y-matrix for a grounded transmission line system:

$$[Y_{(2\times2)}] = \begin{bmatrix} Y_{11(2\times2)} & Y_{12(2\times2)} \\ Y_{21(2\times2)} & Y_{22(2\times2)} \end{bmatrix} = \begin{bmatrix} \dfrac{1}{Z\tanh((\alpha + j\beta)L)} & \dfrac{-1}{Z\sinh((\alpha + j\beta)L)} \\ \dfrac{-1}{Z\sinh((\alpha + j\beta)L)} & \dfrac{1}{Z\tanh((\alpha + j\beta)L)} \end{bmatrix}$$

where α+jβ represents the complex propagation constant, *Z* is the characteristic impedance of the line and *L* is the length of the line as derived from the input parameters.

Applying the equivalent circuit shown above, the Y-matrix of the floating transmission line system can be shown to be the following:

$$[[Y_{(4\times4)}] = \begin{bmatrix} Y_{11(2\times2)} & Y_{12(2\times2)} & -Y_{11(2\times2)} & -Y_{12(2\times2)} \\ Y_{21(2\times2)} & Y_{22(2\times2)} & -Y_{21(2\times2)} & -Y_{22(2\times2)} \\ -Y_{11(2\times2)} & -Y_{12(2\times2)} & Y_{11(2\times2)} & Y_{12(2\times2)} \\ -Y_{21(2\times2)} & -Y_{22(2\times2)} & Y_{21(2\times2)} & Y_{22(2\times2)} \end{bmatrix}$$

At DC, the Y-Matrix changes to a model of two wires above a ground plane:

$$[Y_{(4\times4)}] = \begin{bmatrix} 1/R & -1/R & 0 & 0 \\ -1/R & 1/R & 0 & 0 \\ 0 & 0 & 1/R & -1/R \\ 0 & 0 & -1/R & 1/R \end{bmatrix}$$

where R is a real resistance approaching zero.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

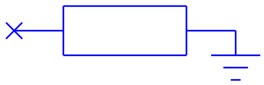
Recommendations for Use

This model, along with additional components can be used to model transmission line baluns and transmission line transformers in which one of the conductors is shielded from ground, like a coaxial line.

NOTE: Because the model definition does not include interactions with the ground, unusual and unexpected results can occur if other components are not used to relate the voltage on both sides of the transmission line to ground.

Short Circuited RLGC Transmitting Line (Closed Form): RLGC_TxS

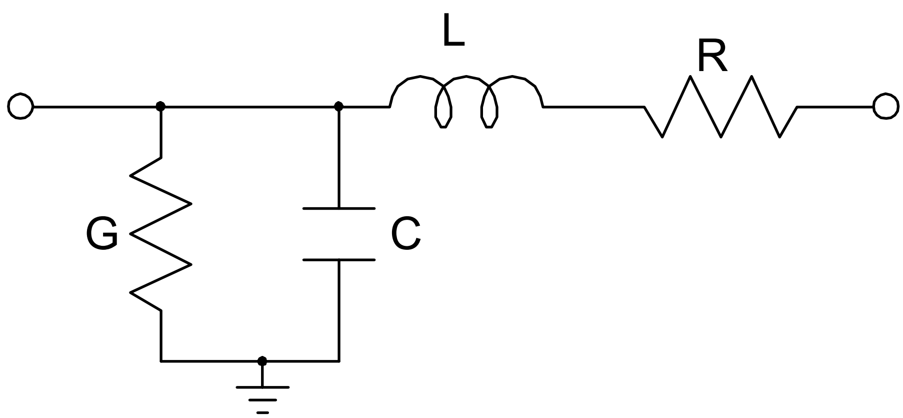
Symbol



Summary

RLGC_TxS provides computation of circuit parameters for general short-circuited transmission lines whose behavior in a frequency domain is governed by a user-supplied table of primary per-unit-length line parameters R, L, G, and C.

Equivalent Circuit



Parameters

Name	Description	Unit Type	Default
ID	Name	Text	TL1
Len	Length of Line	Length	L[a]
F	Vector of Frequencies at which L, C, R, G specified	Frequency	1GHz
R	Vector of Series R per-unit-length	Resistance/meter	100
L	Vector of Series L per-unit-length	Inductance/meter	Li[a]
G	Vector of Shunt G per-unit-length	Conductance/meter	0
C	Vector of Shunt C per-unit-length	Capacitance/meter	C[a]

[a] User-modifiable default. Modify by editing the *DEFAULT.LPF* file in the root installation directory. See "Default Values" for details.

Parameter Details

F. Vector of frequencies at which R, L, G, and C parameters are specified. Frequencies must be sequential and specified in ascending order.

R. Vector of series resistance (see Equivalent Circuit) per-unit-length specified in resistance project units per meter. Each vector entry must be specified at the corresponding frequency entry from frequency vector **F.**

L. Vector of series inductance (see Equivalent Circuit) per-unit-length specified in inductance project units per meter. Each vector entry must be specified at the corresponding frequency entry from frequency vector F.

G. Vector of shunt conductance (see Equivalent Circuit) per-unit-length specified in conductance project units per meter. Each vector entry must be specified at the corresponding frequency entry from frequency vector **F.**

C. Vector of shunt capacitance (see Equivalent Circuit) per-unit-length specified in capacitance project units per meter. Each vector entry must be specified at the corresponding frequency entry from frequency vector **F.**

Parameter Restrictions and Recommendations

1. Lengths of vector parameters R, L, G, and C must be exactly equal to the length of frequency vector F.
2. If project operational frequency is out of range of frequencies in F then R, L, G, and C parameters are extrapolated as constant values equal to the first/last entries of corresponding vectors. No warning is issued.
3. Vector can be specified in three ways: First, it can be entered as a right side value of model parameter, e.g. R={100,102,110,113,120}; Second, vector can be specified elsewhere in equation; Third way is specification of vector in a column or row of a text file. Third way provides a convenient and flexible method of specification of all RLGC parameters in the single location. Just create, for example, file RLGC.txt containing space separated columns of R1, R2 etc. First column must represent frequency in project units (note that changing of project default frequency units will demand manual scaling of frequencies in this file). Import or link this file to your project and give it a name, say, RLGC_15. Now you can specify, say, parameter R as R = Col(datafile("RLGC_15"),1). It means that values of vector R will be copied to the model from the column 1 of file RLGC.txt imported under name RLGC_15. If you prefer to deploy your data row-wise use R = Row(datafile("RLGC_15"),1).

Implementation Details

Model implementation is based on linear interpolation of RLGC parameters at each project evaluation frequency. Interpolation uses user-supplied via parameters look-out tables. If project operational frequency is out of range of frequencies in F then R, L, G, and C parameters are extrapolated as constant values equal to the first/last entries of corresponding vectors.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

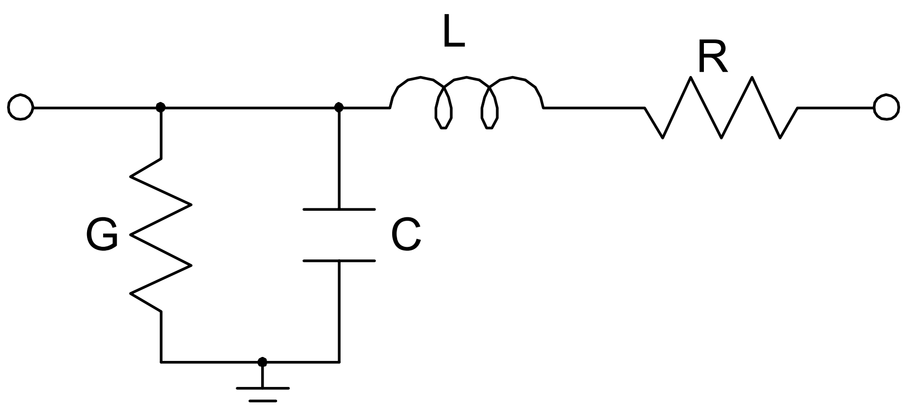
Short Circuited RLGC Transmitting Line with Floating Ground (Closed Form): RLGC_TxS2

Symbol

Summary

RLGC_TxO2 simulates a general short-circuited transmission line with isolated ground terminals. This model provides computation of circuit parameters for general transmission lines whose behavior in a frequency domain is governed by a user- supplied table of primary per-unit-length line parameters R, L, G, and C. Extreme care should be used with this element as it is meant to work in concert with additional elements which relate the voltages at the end of the transmission line to the global ground. Unusual and unexpected behavior can result if these additional elements are omitted. See the RLGC_Tx4 model documentation for implementation details.

Equivalent Circuit of Single Line



Parameters

Name	Description	Unit Type	Default
ID	Name	Text	TL1
Len	Length of Line	Length	L[a]
F	Vector of Frequencies at which L, C, R, G specified	Frequency	1 GHz
R	Vector of Series R per-unit-length	Resistance/meter	100
L	Vector of Series L per-unit-length	Inductance/meter	Li[a]
G	Vector of Shunt G per-unit-length	Conductance/meter	0
C	Vector of Shunt C per-unit-length	Capacitance/meter	C[a]

[a] User-modifiable default. Modify by editing the *DEFAULT.LPF* file in the root installation directory. See “Default Values” for details.

Parameter Details

F. Vector of frequencies at which R, L, G, and C parameters are specified. Frequencies must be sequential and specified in ascending order.

R. Vector of series resistance (see Equivalent Circuit) per-unit-length specified in resistance project units per meter. Each vector entry must be specified at the corresponding frequency entry from frequency vector **F.**

L. Vector of series inductance (see Equivalent Circuit) per-unit-length specified in inductance project units per meter. Each vector entry must be specified at the corresponding frequency entry from frequency vector F.

G. Vector of shunt conductance (see Equivalent Circuit) per-unit-length specified in conductance project units per meter. Each vector entry must be specified at the corresponding frequency entry from frequency vector **F.**

C. Vector of shunt capacitance (see Equivalent Circuit) per-unit-length specified in capacitance project units per meter. Each vector entry must be specified at the corresponding frequency entry from frequency vector **F.**

Parameter Restrictions and Recommendations

1. Lengths of vector parameters R, L, G, and C must be exactly equal to the length of frequency vector F.
2. If project operational frequency is out of range of frequencies in F then R, L, G, and C parameters are extrapolated as constant values equal to the first/last entries of corresponding vectors. No warning is issued.
3. Vector can be specified in three ways: First, it can be entered as a right side value of model parameter, e.g. R={100,102,110,113,120}; Second, vector can be specified elsewhere in equation; Third way is specification of vector in a column or row of a text file. Third way provides a convenient and flexible method of specification of all RLGC parameters in the single location. Just create, for example, file RLGC.txt containing space separated columns of R1, R2 etc. First column must represent frequency in project units (note that changing of project default frequency units will demand manual scaling of frequencies in this file). Import or link this file to your project and give it a name, say, RLGC_15. Now you can specify, say, parameter R as R = Col(datafile("RLGC_15"),1). It means that values of vector R will be copied to the model from the column 1 of file RLGC.txt imported under name RLGC_15. If you prefer to deploy your data row-wise use R = Row(datafile("RLGC_15"),1).
4. If your project uses text file input to feed data to this model be aware what frequency, resistance, inductance or conductance units this file implies. There is a chance that default units of your project differ from those in your data file. If this happens, you must scale input values multiplying call of function Col or Row by scaling coefficient. For example, if your project uses capacitances in picofarads and data file contains data in Farads you may get capacitance data from the fifth column of data file RLGC_15 like this: C1= 1e+12*Col(datafile("RLGC_15"),5).

Implementation Details

Model implementation is based on linear interpolation of RLGC parameters at each project evaluation frequency. Interpolation uses user-supplied via parameters look-out tables. If project operational frequency is out of range of frequencies in F then R, L, G, and C parameters are extrapolated as constant values equal to the first/last entries of corresponding vectors.

The following is a Y-matrix for a grounded transmission line system:

$$[Y_{(2\times2)}] = \begin{bmatrix} Y_{11(2\times2)} & Y_{12(2\times2)} \\ Y_{21(2\times2)} & Y_{22(2\times2)} \end{bmatrix} = \begin{bmatrix} \dfrac{1}{Z\tanh((\alpha + j\beta)L)} & \dfrac{-1}{Z\sinh((\alpha + j\beta)L)} \\ \dfrac{-1}{Z\sinh((\alpha + j\beta)L)} & \dfrac{1}{Z\tanh((\alpha + j\beta)L)} \end{bmatrix}$$

where α+jβ represents the complex propagation constant, *Z* is the characteristic impedance of the line and *L* is the length of the line as derived from the input parameters.

Applying the equivalent circuit shown above, the Y-matrix of the floating transmission line system can be shown to be the following:

$$[[Y_{(4\times4)}] = \begin{bmatrix} Y_{11(2\times2)} & Y_{12(2\times2)} & -Y_{11(2\times2)} & -Y_{12(2\times2)} \\ Y_{21(2\times2)} & Y_{22(2\times2)} & -Y_{21(2\times2)} & -Y_{22(2\times2)} \\ -Y_{11(2\times2)} & -Y_{12(2\times2)} & Y_{11(2\times2)} & Y_{12(2\times2)} \\ -Y_{21(2\times2)} & -Y_{22(2\times2)} & Y_{21(2\times2)} & Y_{22(2\times2)} \end{bmatrix}$$

At DC, the Y-Matrix changes to a model of two wires above a ground plane:

$$[Y_{(4\times4)}] = \begin{bmatrix} 1/R & -1/R & 0 & 0 \\ -1/R & 1/R & 0 & 0 \\ 0 & 0 & 1/R & -1/R \\ 0 & 0 & -1/R & 1/R \end{bmatrix}$$

where *R* is a real resistance approaching zero.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

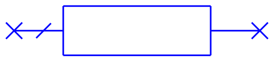
Recommendations for Use

This model, along with additional components can be used to model transmission line baluns and transmission line transformers in which one of the conductors is shielded from ground, like a coaxial line.

NOTE: Because the model definition does not include interactions with the ground, unusual and unexpected results can occur if other components are not used to relate the voltage on both sides of the transmission line to ground.

Phase Spec: Grounded Shield (Closed Form): TLIN

Symbol



Summary

TLIN simulates an ideal, lossless transmission line. Model implies that line length is specified as electrical length (phase lag of mode propagating along the line) at user specified frequency. User must supply value of characteristic impedance.

Parameters

Name	Description	Unit Type	Default
ID	Name	Text	TL1
Z0	Transmission line impedance	Resistance	50 ohm
EL	Electrical length (phase length) at F0	Angle	90 Deg
F0	Frequency used to specify EL	Frequency	10 GHz

Parameter Details

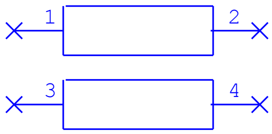
EL and **F0.** These parameters determine the frequency dependence of the electrical length of the line, described as $\beta L=EL \cdot freq/Fo \cdot \pi/180$ where *freq* is the evaluation frequency.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Phase Spec: Floating Shield (Closed Form): TLIN4

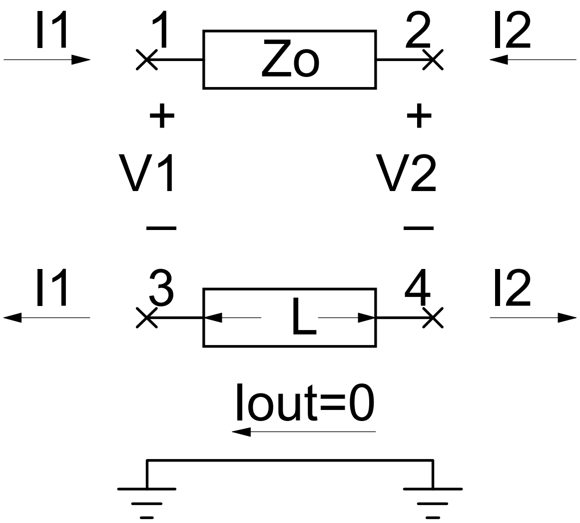
Symbol



Summary

TLIN4 simulates a transmission line with isolated ground terminals. The characteristics of the transmission line are specified as the characteristic impedance and the electrical length at a given frequency. Extreme care should be used with this element as it is meant to work in concert with additional elements which relate the voltages at both ends of the transmission line to the global ground. Unusual and unexpected behavior can result if these additional elements are omitted.

Equivalent Circuit



Note that any interaction with the ground on either side of the transmission line is ignored. All current on one conductor is equal and opposite to the current on the other at any given distance along the line. Importantly, this condition is enforced regardless of the length of the transmission line or the operating frequency. The exception to this behavior is at DC (Frequency = 0 Hz) where current on the two conductors can be unequal to allow both conductors to be used for biasing active devices. This change in behavior at DC causes a discontinuity of the model parameters at DC. This discontinuity is expected, and additional circuit components relating the voltage at each end of the transmission line to ground should be added, allowing flexibility in implementing the desired transition to the RF-to-DC performance.

Parameters

Name	Description	Unit Type	Default
ID	Name	Text	TL1

Name	Description	Unit Type	Default
Z0	Transmission line impedance	Resistance	50 ohm
EL	Electrical length (phase length) at F0	Angle	90 Deg
F0	Frequency used to specify EL	Frequency	10 GHz

Parameter Details

EL and **F0**. These parameters determine the frequency dependence of the electrical length of the line, described as $\beta L = EL \cdot freq/Fo \cdot \pi/180$ where *freq* is the evaluation frequency.

Implementation Details

The following is a Y-matrix for a grounded transmission line system:

$$[Y_{(2\times2)}] = \begin{bmatrix} Y_{11(2\times2)} & Y_{12(2\times2)} \\ Y_{21(2\times2)} & Y_{22(2\times2)} \end{bmatrix} = \begin{bmatrix} \frac{1}{Z0\tanh(j\beta L)} & \frac{-1}{Z0\sinh(j\beta L)} \\ \frac{-1}{Z0\sinh(j\beta L)} & \frac{1}{Z0\tanh(j\beta L)} \end{bmatrix}$$

where β represents the propagation constant, *Z* is the characteristic impedance of the line and *L* is the length of the line as derived from the input parameters.

Applying the equivalent circuit shown above, the Y-matrix of the floating transmission line system can be shown to be the following:

$$[[Y_{(4\times4)}] = \begin{bmatrix} Y_{11(2\times2)} & Y_{12(2\times2)} & -Y_{11(2\times2)} & -Y_{12(2\times2)} \\ Y_{21(2\times2)} & Y_{22(2\times2)} & -Y_{21(2\times2)} & -Y_{22(2\times2)} \\ -Y_{11(2\times2)} & -Y_{12(2\times2)} & Y_{11(2\times2)} & Y_{12(2\times2)} \\ -Y_{21(2\times2)} & -Y_{22(2\times2)} & Y_{21(2\times2)} & Y_{22(2\times2)} \end{bmatrix}$$

At DC, the Y-Matrix changes to a model of two wires above a ground plane:

$$[Y_{(4\times4)}] = \begin{bmatrix} 1/R & -1/R & 0 & 0 \\ -1/R & 1/R & 0 & 0 \\ 0 & 0 & 1/R & -1/R \\ 0 & 0 & -1/R & 1/R \end{bmatrix}$$

where *R* is a real resistance approaching zero.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

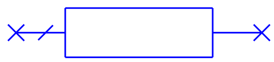
Recommendations for Use

This model, along with additional components can be used to model transmission line baluns and transmission line transformers in which one of the conductors is shielded from ground, like in a coaxial line.

NOTE: Because the model definition does not include interactions with the ground, unusual and unexpected results can occur if other components are not used to relate the voltage on both sides of the transmission line to ground.

Physical Spec: Grounded Shield, Improved Accuracy (Closed Form): TLINP

Symbol



Summary

TLINP simulates an ideal transmission line with loss. Mode propagating in line is specified by its effective dielectric constant and per-unit-length attenuation at a user-specified frequency. TLINP scales loss with evaluation frequency. By default, this model provides a partial account for circuit loss (insertion loss only). You can toggle the secondary parameter Total_Loss to obtain access to insertion loss, return loss, and dielectric loss (for example, for microstrips, stripline or other.)

Parameters

Name	Description	Unit Type	Default
ID	Name	Text	TL1
Z0	Transmission line impedance	Resistance	50 ohm
L	Transmission line length	Length	0 um
Eeff	Relative effective dielectric constant		1
Loss	Loss (dB/m)		0 dB/m
F0	Frequency for scaling loss	Frequency	0 GHz
*LossModel	Partial/Full account for conductor and dielectric loss		Partial
*TanD	Dielectric loss tangent		0
*Mur	Relative dielectric permeability		1
*TanM	Dielectric magnetic loss tangent		0
*Sigma	Dielectric bulk conductivity (S/m)		0
*IsEpsFreqDep	Dielectric constant is frequency dependent/independent		Frequency independent dielectric constant
*Fmin	Low roll-off frequency of Tand frequency dependence	Frequency	1kHz
*Fmax	High roll-off frequency of Tand frequency dependence	Frequency	1THz
*Fspec	Er and Tand are specified at this frequency	Frequency	1GHz

** indicates a secondary parameter*

Parameter Details

Loss and **F0.** Determine the frequency-dependence of the attenuation constant. If F0 is not equal to zero (0.0), then

$$\alpha = \mathrm{Loss}\sqrt{\mathrm{freq} / \mathrm{F0}}$$

(dB/m) where *freq* is the evaluation frequency. If F0 is equal to zero, the attenuation is constant versus frequency and is equal to Loss(dB/m).

LossModel is a secondary control parameter that defines how the model accounts for loss and dielectric parameters. If LossModel=Partial (default value) than the model accounts for insertion loss (parameter Loss) only. If LossModel=Full then the model accounts for insertion loss, return loss, dielectric loss, and for frequency dependent model of dielectric parameters (for example, for microstrips, stripline, or other).

Note: Secondary parameter LossModel is always visible at model Properties, tab Parameters if Show Secondary is selected. If LossModel=Partial, other secondary parameters are also visible at tab Parameters but they are not used by the model and do not appear at the schematic until LossModel=Full is selected.

IsEpsFreqDep is a (hidden, secondary) control parameter that defines if dielectric constant and dielectric loss tangent exhibit predefined frequency dependence, suggested in [1]. This dependence is controlled by the three frequency parameters Fmin, Fmax, and Fspec, and imparts causal behavior to the model.

Layout

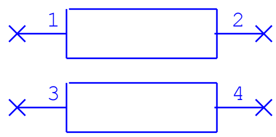
This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

References

[1] A.R. Djordjevic et al., "Wideband Frequency-Domain Characterization of FR-4 and Time-Domain Causality", IEEE Trans. of Electromagn. Compat., Vol. 43, No. 4, Nov. 2001

Physical Spec: Floating Shield, Improved Accuracy (Closed Form): TLINP4

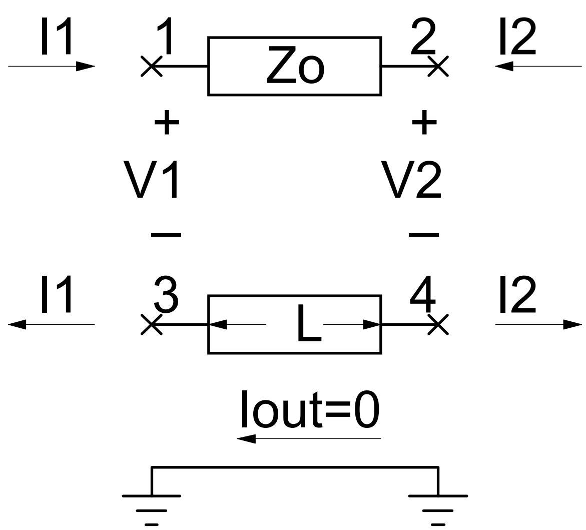
Symbol



Summary

TLINP4 simulates a transmission line with isolated ground terminals. The characteristics of the transmission line are specified as physical length and electrical specification of the characteristic impedance and propagation constant. This model scales loss with a user-specified evaluation frequency. By default, this model provides a partial account for circuit loss (insertion loss only). You can toggle the secondary parameter LossModel to obtain access to insertion loss, return loss, and dielectric loss (for example; for microstrips or stripline). Use extreme care with this element as it is meant to work in concert with additional elements which relate the voltages at both ends of the transmission line to the global ground. Unusual and unexpected behavior can result if these additional elements are omitted.

Equivalent Circuit



Note that any interaction with the ground on either side of the transmission line is ignored. All current on one conductor is equal and opposite to the current on the other at any given distance along the line. Importantly, this condition is enforced regardless of the length of the transmission line or the operating frequency. The exception to this behavior is at DC (Frequency = 0 Hz) where current on the two conductors can be unequal to allow both conductors to be used for biasing active devices. This change in behavior at DC causes a discontinuity of the model parameters at DC. This discontinuity is expected, and you should add additional circuit components relating the voltage at each end of the transmission line to ground, allowing flexibility in implementing the desired transition to the RF-to-DC performance.

Parameters

Name	Description	Unit Type	Default
ID	Name	Text	TL1
Z	Transmission line impedance	Resistance	50 ohm
L	Transmission line length	Length	0 um
K	Relative dielectric constant		1
A	Loss (dB/m)		0
F	Frequency for scaling losses	Frequency	0
*LossModel	Partial/Full account for conductor and dielectric loss		Partial
*TanD	Dielectric loss tangent		0
*Mur	Relative dielectric permeability		1
*TanM	Dielectric magnetic loss tangent		0
*Sigma	Dielectric bulk conductivity (S/m)		0
*EpsFreqDep	Dielectric constant is frequency dependent (No/Yes)		No
*Fmin	Low roll-off frequency of Tand frequency dependence	Frequency	1kHz
*Fmax	High roll-off frequency of Tand frequency dependence	Frequency	1THz
*Fspec	Er and Tand are specified at this frequency	Frequency	1GHz

** indicates a hidden secondary parameter*

Parameter Details

Note that parameters marked by asterisk (*) are hidden/invisible while the control parameter LossModel=Partial (default value). Setting LossModel=Full displays these secondary parameters in the model's Element Options dialog box (except parameters Fmin, Fmax, and Fspec which are visible only when parameter EpsFreqDep=Yes).

A and **F**. These parameters determine the frequency dependence of the attenuation constant. If F is not equal to zero (0.0), then

$$\alpha = A\sqrt{\mathrm{freq}/F}$$

(dB/m) where *freq* is the evaluation frequency. If F is equal to zero, the attenuation is constant versus frequency and is equal to A (dB/m).

LossModel is a secondary control parameter that defines how the model accounts for loss and dielectric parameters. If LossModel=Partial (default value) than the model accounts for insertion loss (parameter Loss) only. If LossModel=Full then the model accounts for insertion loss, return loss, dielectric loss, and for frequency dependent model of dielectric parameters (for example, for microstrips, stripline, or other).

Note: Secondary parameter LossModel is always visible at model Properties, tab Parameters if Show Secondary is selected. <u>If LossModel=Partial, other secondary parameters are also visible at tab Parameters but they are not used by the model and do not appear at the schematic until LossModel=Full is selected</u>.

EpsFreqDep is a (hidden, secondary) control parameter that defines if dielectric constant and dielectric loss tangent exhibit predefined frequency dependence, suggested in [1]. This dependence is controlled by the three frequency parameters

Fmin, Fmax, and Fspec, and imparts causal behavior to the model. Fmin, Fmax, and Fspec are hidden (invisible) if EpsFreqDep=No.

Implementation Details

The following is a Y-matrix for a grounded transmission line system:

$$[Y_{(2\times2)}] = \begin{bmatrix} Y_{11(2\times2)} & Y_{12(2\times2)} \\ Y_{21(2\times2)} & Y_{22(2\times2)} \end{bmatrix} = \begin{bmatrix} \dfrac{1}{Z\tanh((\alpha+j\beta)L)} & \dfrac{-1}{Z\sinh((\alpha+j\beta)L)} \\ \dfrac{-1}{Z\sinh((\alpha+j\beta)L)} & \dfrac{1}{Z\tanh((\alpha+j\beta)L)} \end{bmatrix}$$

where α+jβ represents the complex propagation constant, *Z* is the characteristic impedance of the line and *L* is the length of the line as derived from the input parameters.

Applying the equivalent circuit shown above, the Y-matrix of the floating transmission line system can be shown as follows:

$$[[Y_{(4\times4)}] = \begin{bmatrix} Y_{11(2\times2)} & Y_{12(2\times2)} & -Y_{11(2\times2)} & -Y_{12(2\times2)} \\ Y_{21(2\times2)} & Y_{22(2\times2)} & -Y_{21(2\times2)} & -Y_{22(2\times2)} \\ -Y_{11(2\times2)} & -Y_{12(2\times2)} & Y_{11(2\times2)} & Y_{12(2\times2)} \\ -Y_{21(2\times2)} & -Y_{22(2\times2)} & Y_{21(2\times2)} & Y_{22(2\times2)} \end{bmatrix}$$

At DC, the Y-matrix changes to a model of two wires above a ground plane:

$$[Y_{(4\times4)}] = \begin{bmatrix} 1/R & -1/R & 0 & 0 \\ -1/R & 1/R & 0 & 0 \\ 0 & 0 & 1/R & -1/R \\ 0 & 0 & -1/R & 1/R \end{bmatrix}$$

where *R* is a real resistance approaching zero.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Recommendations for Use

This model, along with additional components can be used to model transmission line baluns and transmission line transformers in which one of the conductors is shielded from ground, like a coaxial line.

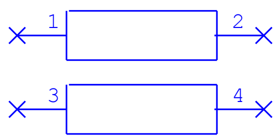
NOTE: Because the model definition does not include interactions with the ground, unusual and unexpected results can occur if other components are not used to relate the voltage on both sides of the transmission line to ground.

References

[1] A.R. Djordjevic et al., "Wideband Frequency-Domain Characterization of FR-4 and Time-Domain Causality", IEEE Trans. of Electromagn. Compat., Vol. 43, No. 4, Nov. 2001

WaveLength Spec, Floating Shield, Improved Accuracy (Closed Form): TLINP4NL

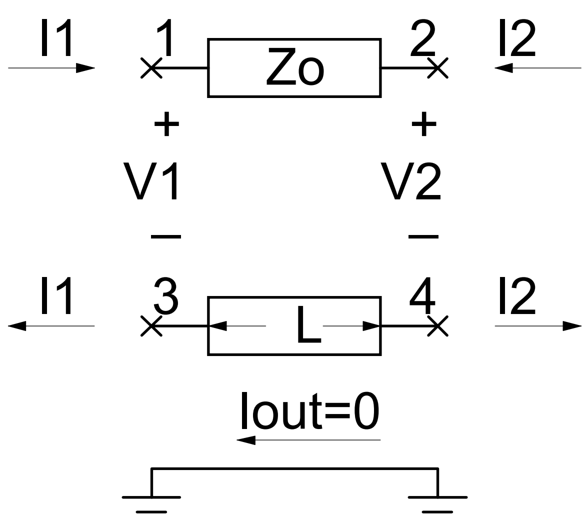
Symbol



Summary

TLINP4NL simulates a transmission line with isolated (floating) ground terminals. The characteristics of the transmission line are specified as the characteristic impedance and the electrical length at a given frequency. By default, this model provides a partial account for circuit loss (insertion loss only). You can toggle the secondary parameter LossModel to obtain access to insertion loss, return loss, and dielectric loss (for example; for microstrips or stripline). Use extreme care with this element as it is meant to work in concert with additional elements which relate the voltages at both ends of the transmission line to the global ground. Unusual and unexpected behavior can result if these additional elements are omitted.

Equivalent Circuit



Note that any interaction with the ground on either side of the transmission line is ignored. All current on one conductor is equal and opposite to the current on the other at any given distance along the line. Importantly, this condition is enforced regardless of the length of the transmission line or the operating frequency. The exception to this behavior is at DC (Frequency = 0 Hz) where current on the two conductors can be unequal to allow both conductors to be used for biasing active devices. This change in behavior at DC causes a discontinuity of the model parameters at DC. This discontinuity is expected, and additional circuit components relating the voltage at each end of the transmission line to ground should be added, allowing flexibility in implementing the desired transition to the RF-to-DC performance.

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	TL1
Z	Impedance		50
NL	Number of wavelengths at F_NL		0.25
F_NL	Frequency at NL wavelengths	Frequency	1 GHz
K	Dielectric constant		1
A	Loss (dB/m)		0
F	Frequency for scaling loss		0
*LossModel	Partial/Full account for conductor and dielectric loss		Partial
*TanD	Dielectric loss tangent		0
*Mur	Relative dielectric permeability		1
*TanM	Dielectric magnetic loss tangent		0
*Sigma	Dielectric bulk conductivity (S/m)		0
*IsEpsFreqDep	Dielectric constant is frequency dependent (No/Yes)		No
*Fmin	Low roll-off frequency of Tand frequency dependence	Frequency	1kHz
*Fmax	High roll-off frequency of Tand frequency dependence	Frequency	1THz
*Fspec	Er and Tand are specified at this frequency	Frequency	1GHz

** indicates a hidden secondary parameter*

Parameter Details

Note that parameters marked by asterisk (*) are hidden/invisible while the control parameter LossModel=Partial (default value). Setting LossModel=Full displays these secondary parameters in the model's Element Options dialog box (except parameters Fmin, Fmax, and Fspec which are visible only when parameter EpsFreqDep=Yes).

NL and **F_NL.** Determine the frequency dependence of the electrical length of the line, described as βL=2πNL(freq/F_NL) where *freq* is the evaluation frequency. The length of the line is given as L=(c·NL)/(F_NL√k) where *c* is the speed of light in a vacuum.

A and **F**. Determine the frequency dependence of the attenuation constant. If F is not equal to zero (0.0), then

$$\alpha = A\sqrt{\mathrm{freq}/F}$$

(dB/m) where *freq* is the evaluation frequency. If F is equal to zero, the attenuation is constant versus frequency and is equal to A (dB/m).

LossModel is a secondary control parameter that defines how the model accounts for loss and dielectric parameters. If LossModel=Partial (default value) than the model accounts for insertion loss (parameter Loss) only. If LossModel=Full then the model accounts for insertion loss, return loss, dielectric loss, and for frequency dependent model of dielectric parameters (for example, for microstrips, stripline, or other).

Note: Secondary parameter LossModel is always visible at model Properties, tab Parameters if Show Secondary is selected. If LossModel=Partial, other secondary parameters are also visible at tab Parameters but they are not used by the model and do not appear at the schematic until LossModel=Full is selected.

IsEpsFreqDep is a (hidden, secondary) control parameter that defines if dielectric constant and dielectric loss tangent exhibit predefined frequency dependence, suggested in [1]. This dependence is controlled by the three frequency parameters **Fmin, Fmax, and Fspec**, and imparts causal behavior to the model. Fmin, Fmax, and Fspec are hidden (invisible) if EpsFreqDep=No.

Implementation Details

The following is a Y-matrix for a grounded transmission line system:

$$L = (c \cdot \mathrm{NL}) / (F\_\mathrm{NL}\sqrt{K})$$

where α+jβ represents the complex propagation constant, *Z* is the characteristic impedance of the line and *L* is the length of the line as derived from the input parameters.

Applying the equivalent circuit shown above, the Y-matrix of the floating transmission line system can be shown as follows:

$$\alpha = A\sqrt{(\mathrm{freq}) / F}$$

At DC, the Y-matrix changes to a model of two wires above a ground plane:

$$[Y_{(2\times2)}] = \begin{bmatrix} Y_{11(2\times2)} & Y_{12(2\times2)} \\ Y_{21(2\times2)} & Y_{22(2\times2)} \end{bmatrix} = \begin{bmatrix} \dfrac{1}{Z\tanh((\alpha+j\beta)L)} & \dfrac{-1}{Z\sinh((\alpha+j\beta)L)} \\ \dfrac{-1}{Z\sinh((\alpha+j\beta)L)} & \dfrac{1}{Z\tanh((\alpha+j\beta)L)} \end{bmatrix}$$

where *R* is a real resistance approaching zero.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

Recommendations for Use

This model, along with additional components can be used to model transmission line baluns and transmission line transformers in which one of the conductors is shielded from ground, like a coaxial line.

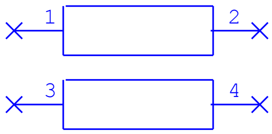
NOTE: Because the model definition does not include interactions with the ground, unusual and unexpected results can occur if other components are not used to relate the voltage on both sides of the transmission line to ground.

References

[1] A.R. Djordjevic et al., "Wideband Frequency-Domain Characterization of FR-4 and Time-Domain Causality", IEEE Trans. of Electromagn. Compat., Vol. 43, No. 4, Nov. 2001

Time Spec, Floating Shield, Improved Accuracy (Closed Form): TLINP4T

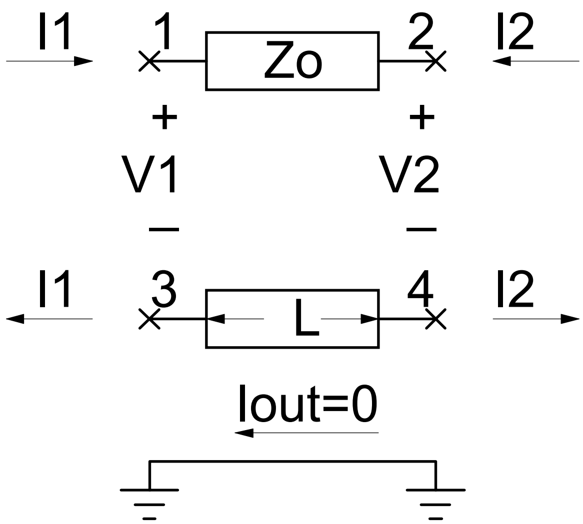
Symbol



Summary

TLINP4T simulates a transmission line with isolated (floating) ground terminals. The characteristics of the transmission line are specified as the characteristic impedance, the electrical length as a time delay, and attenuation at a particular frequency. By default, this model provides a partial account for circuit loss (insertion loss only). You can toggle the secondary parameter LossModel to obtain access to insertion loss, return loss, and dielectric loss (for example; for microstrips or stripline). Use extreme care with this element as it is meant to work in concert with additional elements which relate the voltages at both ends of the transmission line to the global ground. Unusual and unexpected behavior can result if these additional elements are omitted.

Equivalent Circuit



Note that any interaction with the ground on either side of the transmission line is ignored. All current on one conductor is equal and opposite to the current on the other at any given distance along the line. Importantly, this condition is enforced regardless of the length of the transmission line or the operating frequency. The exception to this behavior is at DC (Frequency = 0 Hz) where current on the two conductors can be unequal to allow both conductors to be used for biasing active devices. This change in behavior at DC causes a discontinuity of the model parameters at DC. This discontinuity is expected, and additional circuit components relating the voltage at each end of the transmission line to ground should be added, allowing flexibility in implementing the desired transition to the RF-to-DC performance.

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	TL1
Z	Impedance		50
TD	Time delay in secs	Time	1 ns
K	Dielectric constant		1
A	Loss in (db/m)		0
F	Frequency for scaling losses	Frequency	0
*LossModel	Partial/Full account for conductor and dielectric loss		Partial
*TanD	Dielectric loss tangent		0
*Mur	Relative dielectric permeability		1
*TanM	Dielectric magnetic loss tangent		0
*Sigma	Dielectric bulk conductivity (S/m)		0
*EpsFreqDep	Dielectric constant is frequency dependent (No/Yes)		No
*Fmin	Low roll-off frequency of Tand frequency dependence	Frequency	1kHz
*Fmax	High roll-off frequency of Tand frequency dependence	Frequency	1THz
*Fspec	Er and Tand are specified at this frequency	Frequency	1GHz

** indicates a hidden secondary parameter*

Parameter Details

Note that parameters marked by asterisk (*) are hidden/invisible while the control parameter LossModel=Partial (default value). Setting LossModel=Full displays these secondary parameters in the model's Element Options dialog box (except parameters Fmin, Fmax, and Fspec which are visible only when parameter EpsFreqDep=Yes).

TD. Determines the frequency-dependent electrical length of the line, described as $\beta = 2\pi \cdot TD \cdot freq$ where *freq* is the evaluation frequency. The length of the line is given as $L = c \cdot TD/\sqrt{K}$ where *c* is the speed of light in a vacuum.

A and **F**. Determine the frequency dependence of the attenuation constant. If F is not equal to zero (0.0), then

$$\alpha = A\sqrt{\mathrm{freq}/F}$$

(dB/m) where *freq* is the evaluation frequency. If F is equal to zero, the attenuation is constant versus frequency and is equal to A (dB/m).

LossModel is a secondary control parameter that defines how the model accounts for loss and dielectric parameters. If LossModel=Partial (default value) than the model accounts for insertion loss (parameter Loss) only. If LossModel=Full then the model accounts for insertion loss, return loss, dielectric loss, and for frequency dependent model of dielectric parameters (for example, for microstrips, stripline, or other).

Note: Secondary parameter LossModel is always visible at model Properties, tab Parameters if Show Secondary is selected. If LossModel=Partial, other secondary parameters are also visible at tab Parameters but they are not used by the model and do not appear at the schematic until LossModel=Full is selected.

EpsFreqDep is a (hidden, secondary) control parameter that defines if dielectric constant and dielectric loss tangent exhibit predefined frequency dependence, suggested in [1]. This dependence is controlled by the three frequency parameters **Fmin, Fmax, and Fspec**, and imparts causal behavior to the model. Fmin, Fmax, and Fspec are hidden (invisible) if EpsFreqDep=No.

Implementation Details

The following is a Y-matrix for a grounded transmission line system:

$$[Y_{(2\times2)}] = \begin{bmatrix} Y_{11(2\times2)} & Y_{12(2\times2)} \\ Y_{21(2\times2)} & Y_{22(2\times2)} \end{bmatrix} = \begin{bmatrix} \dfrac{1}{Z\tanh((\alpha+j\beta)L)} & \dfrac{-1}{Z\sinh((\alpha+j\beta)L)} \\ \dfrac{-1}{Z\sinh((\alpha+j\beta)L)} & \dfrac{1}{Z\tanh((\alpha+j\beta)L)} \end{bmatrix}$$

where $\alpha+j\beta$ represents the complex propagation constant, Z is the characteristic impedance of the line and L is the length of the line as derived from the input parameters.

Applying the equivalent circuit shown above, the Y-matrix of the floating transmission line system can be shown as follows:

$$[[Y_{(4\times4)}] = \begin{bmatrix} Y_{11(2\times2)} & Y_{12(2\times2)} & -Y_{11(2\times2)} & -Y_{12(2\times2)} \\ Y_{21(2\times2)} & Y_{22(2\times2)} & -Y_{21(2\times2)} & -Y_{22(2\times2)} \\ -Y_{11(2\times2)} & -Y_{12(2\times2)} & Y_{11(2\times2)} & Y_{12(2\times2)} \\ -Y_{21(2\times2)} & -Y_{22(2\times2)} & Y_{21(2\times2)} & Y_{22(2\times2)} \end{bmatrix}$$

At DC, the Y-matrix changes to a model of two wires above a ground plane:

$$[Y_{(4\times4)}] = \begin{bmatrix} 1/R & -1/R & 0 & 0 \\ -1/R & 1/R & 0 & 0 \\ 0 & 0 & 1/R & -1/R \\ 0 & 0 & -1/R & 1/R \end{bmatrix}$$

where R is a real resistance approaching zero.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Recommendations for Use

This model, along with additional components can be used to model transmission line baluns and transmission line transformers in which one of the conductors is shielded from ground, like a coaxial line.

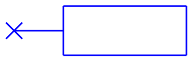
NOTE: Because the model definition does not include interactions with the ground, unusual and unexpected results can occur if other components are not used to relate the voltage on both sides of the transmission line to ground.

References

[1] A.R. Djordjevic et al., "Wideband Frequency-Domain Characterization of FR-4 and Time-Domain Causality", IEEE Trans. of Electromagn. Compat., Vol. 43, No. 4, Nov. 2001

Phase Spec: Grounded Shield, Open (Closed Form): TLOC

Symbol



Summary

TLOC simulates an ideal, lossless open-circuited transmission line. Model implies that line length is specified as electrical length (phase lag of mode propagating along the line) at user specified frequency. User must supply value of characteristic impedance.

Parameters

Name	Description	Unit Type	Default
ID	Name	Text	TL1
Z0	Transmission line impedance	Resistance	50 ohm
EL	Electrical length (phase length) at F0	Angle	90 Deg
F0	Frequency used to specify EL	Frequency	10 GHz

Parameter Details

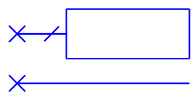
EL and **F0.** These parameters determine the frequency dependence of the electrical length of the line, described as $\beta L = EL \cdot freq/Fo \cdot \pi/180$ where *freq* is the evaluation frequency.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Phase Spec: Floating Shield, Open (Closed Form): TLOC2

Symbol



Summary

TLOC2 simulates a two terminal ideal open-circuited transmission line with isolated (floating) ground terminal. The characteristics of the transmission line are specified as the characteristic impedance and the electrical length at a given frequency. Extreme care should be used with this element as it is meant to work in concert with additional elements which relate the voltages at ports of the transmission line to the global ground. Unusual and unexpected behavior can result if these additional elements are omitted.

Parameters

Name	Description	Unit Type	Default
ID	Name	Text	TL1
Z0	Transmission line impedance	Resistance	50 ohm
EL	Electrical length (phase length) at F0	Angle	90 Deg
F0	Frequency used to specify EL	Frequency	10 GHz

Equivalent Circuit

See equivalent circuit of TLIN4.

Parameters

Name	Description	Unit Type	Default
ID	Name	Text	TL1
Z0	Transmission line impedance	Resistance	50 ohm
EL	Electrical length (phase length) at F0	Angle	90 Deg
F0	Frequency used to specify EL	Frequency	10 GHz

Parameter Details

EL and **F0**. These parameters determine the frequency dependence of the electrical length of the line, described as $\beta L = EL \cdot freq/Fo \cdot \pi/180$ where *freq* is the evaluation frequency.

Implementation Details

See the TLIN4 "Implementation Details" section.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

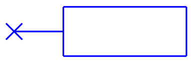
Recommendations for Use

This model, along with additional components can be used to model transmission line baluns and transmission line transformers in which one of the conductors is shielded from ground, like in a coaxial line.

NOTE: Because the model definition does not include interactions with the ground, unusual and unexpected results can occur if other components are not used to relate the voltage on both sides of the transmission line to ground.

Physical Spec: Grounded Shield, Open, Improved Accuracy (Closed Form): TLOCP

Symbol



Summary

TLOCP simulates an ideal open-circuited transmission line with loss. Mode propagating in line is specified by its effective dielectric constant and per-unit-length attenuation at a user-specified frequency. This model scales loss with evaluation frequency. You can toggle the secondary parameter LossModel to obtain access to insertion loss, return loss, and dielectric loss (for example; for microstrips or stripline).

Parameters

Name	Description	Unit Type	Default
ID	Name	Text	TL1
Z0	Transmission line impedance	Resistance	50 ohm
L	Transmission line length	Length	0 um
Eeff	Relative effective dielectric constant		1
Loss	Loss (dB/m)		0 GHz
F0	Frequency for scaling loss		
*LossModel	Partial/Full account for conductor and dielectric loss		Partial
*TanD	Dielectric loss tangent		0
*Mur	Relative dielectric permeability		1
*TanM	Dielectric magnetic loss tangent		0
*Sigma	Dielectric bulk conductivity (S/m)		0
*EpsFreqDep	Dielectric constant is frequency dependent(No/Yes)		No
*Fmin	Low roll-off frequency of Tand frequency dependence	Frequency	1kHz
*Fmax	High roll-off frequency of Tand frequency dependence	Frequency	1THz
*Fspec	Er and Tand are specified at this frequency	Frequency	1GHz

** indicates a hidden secondary parameter*

Parameter Details

Note that parameters marked by asterisk (*) are hidden/invisible while the control parameter LossModel=Partial (default value). Setting LossModel=Full displays these secondary parameters in the model's Element Options dialog box (except parameters Fmin, Fmax, and Fspec which are visible only when parameter EpsFreqDep=Yes).

Loss and **F0.** Determine the frequency dependence of the attenuation constant. If F0 is not equal to zero (0.0), then

$$\alpha = \mathrm{Loss}\sqrt{\mathrm{freq} / F0}$$

(dB/m) where *freq* is the evaluation frequency. If F0 is equal to zero, the attenuation is constant versus frequency and is equal to Loss(dB/m).

LossModel is a secondary control parameter that defines how the model accounts for loss and dielectric parameters. If LossModel=Partial (default value) than the model accounts for insertion loss (parameter Loss) only. If LossModel=Full then the model accounts for insertion loss, return loss, dielectric loss, and for frequency dependent model of dielectric parameters (for example, for microstrips, stripline, or other).

Note: Secondary parameter LossModel is always visible at model Properties, tab Parameters if Show Secondary is selected. If LossModel=Partial, other secondary parameters are also visible at tab Parameters but they are not used by the model and do not appear at the schematic until LossModel=Full is selected.

EpsFreqDep is a (hidden, secondary) control parameter that defines if dielectric constant and dielectric loss tangent exhibit predefined frequency dependence, suggested in [1]. This dependence is controlled by the three frequency parameters Fmin, Fmax, and Fspec, and imparts causal behavior to the model. Fmin, Fmax, and Fspec are hidden (invisible) if EpsFreqDep=No.

Layout

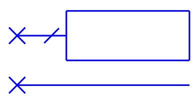
This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

References

[1] A.R. Djordjevic et al., "Wideband Frequency-Domain Characterization of FR-4 and Time-Domain Causality", IEEE Trans. of Electromagn. Compat., Vol. 43, No. 4, Nov. 2001

Physical Spec: Floating Shield, Open, Improved Accuracy (Closed Form): TLOCP2

Symbol



Summary

TLOCP2 simulates an open-circuited transmission line with isolated (floating) ground terminals. The characteristics of the transmission line are specified as physical length and electrical specification of the characteristic impedance and propagation constant. By default, this model provides partial account for circuit loss (insertion loss only). You can toggle the secondary parameter LossModel to obtain access to insertion loss, return loss, and dielectric loss (for example; for microstrips or stripline). Use extreme care with this element as it is meant to work in concert with additional elements which relate the voltages at ports of the transmission line to the global ground. Unusual and unexpected behavior can result if these additional elements are omitted.

Parameters

Name	Description	Unit Type	Default
ID	Name	Text	TL1
Z0	Transmission line impedance	Resistance	50 ohm
L	Transmission line length	Length	0 um
Eeff	Relative effective dielectric constant		0
Loss	Loss (db/meter)		0
F0	Frequency for scaling losses	Frequency	0 GHz
*LossModel	Partial/Full account for conductor and dielectric loss		Partial
*TanD	Dielectric loss tangent		0
*Mur	Relative dielectric permeability		1
*TanM	Dielectric magnetic loss tangent		0
*Sigma	Dielectric bulk conductivity (S/m)		0
*EpsFreqDep	Dielectric constant is frequency dependent(No/Yes)		No
*Fmin	Low roll-off frequency of Tand frequency dependence	Frequency	1kHz
*Fmax	High roll-off frequency of Tand frequency dependence	Frequency	1THz
*Fspec	Er and Tand are specified at this frequency	Frequency	1GHz

** indicates a secondary parameter*

Equivalent Circuit

See the TLINP4 equivalent circuit.

Parameter Details

Note that parameters marked by asterisk (*) are hidden/invisible while the control parameter LossModel=Partial (default value). Setting LossModel=Full displays these secondary parameters in the model's Element Options dialog box (except parameters Fmin, Fmax, and Fspec which are visible only when parameter EpsFreqDep=Yes).

Loss and **F0**. Determine the frequency dependence of the attenuation constant. If F0 is not equal to zero (0.0), then

$$\alpha = \mathrm{Loss}\sqrt{\mathrm{freq} / F0}$$

(dB/m) where *freq* is the evaluation frequency. If F0 is equal to zero, the attenuation is constant versus frequency and is equal to Loss(dB/m).

LossModel is a secondary control parameter that defines how the model accounts for loss and dielectric parameters. If LossModel=Partial (default value) than the model accounts for insertion loss (parameter Loss) only. If LossModel=Full then the model accounts for insertion loss, return loss, dielectric loss, and for frequency dependent model of dielectric parameters (for example, for microstrips, stripline, or other).

Note: Secondary parameter LossModel is always visible at model Properties, tab Parameters if Show Secondary is selected. If LossModel=Partial, other secondary parameters are also visible at tab Parameters but they are not used by the model and do not appear at the schematic until LossModel=Full is selected.

EpsFreqDep is a (hidden, secondary) control parameter that defines if dielectric constant and dielectric loss tangent exhibit predefined frequency dependence, suggested in [1]. This dependence is controlled by the three frequency parameters Fmin, Fmax, and Fspec, and imparts causal behavior to the model. Fmin, Fmax, and Fspec are hidden (invisible) if EpsFreqDep=No.

Implementation Details

See the TLINP4 "Implementation Details" section.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

Recommendations for Use

This model, along with additional components can be used to model transmission line baluns and transmission line transformers in which one of the conductors is shielded from ground, like a coaxial line.

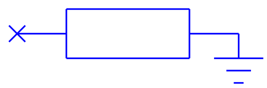
NOTE: Because the model definition does not include interactions with the ground, unusual and unexpected results can occur if other components are not used to relate the voltage on both sides of the transmission line to ground.

References

[1] A.R. Djordjevic et al., "Wideband Frequency-Domain Characterization of FR-4 and Time-Domain Causality", IEEE Trans. of Electromagn. Compat., Vol. 43, No. 4, Nov. 2001

Phase Spec: Grounded Shield, Shorted (Closed Form): TLSC

Symbol



Summary

TLSC simulates an ideal, lossless short-circuited transmission line. Model implies that line length is specified as electrical length (phase lag of mode propagating along the line) at user specified frequency. User must supply value of characteristic impedance.

Parameters

Name	Description	Unit Type	Default
ID	Name	Text	TL1
Z0	Transmission line impedance	Resistance	50 ohm
EL	Electrical length (phase length) at F0	Angle	90 Deg
F0	Frequency used to specify EL	Frequency	10 GHz

Parameter Details

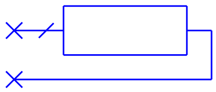
EL and **F0.** These parameters determine the frequency dependence of the electrical length of the line, described as $\beta L = EL \cdot freq/Fo \cdot \pi/180$ where *freq* is the evaluation frequency.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Phase Spec: Floating Shield, Shorted (Closed Form): TLSC2

Symbol



Summary

TLSC2 simulates a two-terminal ideal short-circuited transmission line with an isolated (floating) ground terminal. The characteristics of the transmission line are specified as the characteristic impedance and the electrical length at a given frequency. Extreme care should be used with this element as it is meant to work in concert with additional elements which relate the voltages at ports of the transmission line to the global ground. Unusual and unexpected behavior can result if these additional elements are omitted.

Parameters

Name	Description	Unit Type	Default
ID	Name	Text	TL1
Z0	Transmission line impedance	Resistance	50 ohm
EL	Electrical length (phase length) at F0	Angle	90 Deg
F0	Frequency used to specify EL	Frequency	10 GHz

Equivalent Circuit

See the "Equivalent Circuit" section of TLIN4.

Parameter Details

EL and **F0**. These parameters determine the frequency-dependence of the electrical length of the line, described as $\beta L = EL \cdot freq/Fo \cdot \pi/180$ where *freq* is the evaluation frequency.

Implementation Details

See the "Implementation Details" section of TLIN4.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

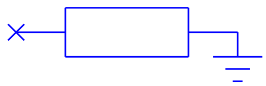
Recommendations for Use

You can use this model along with additional components to model transmission line baluns and transmission line transformers in which one of the conductors is shielded from ground, like in a coaxial line.

NOTE: Because the model definition does not include interactions with the ground, unusual and unexpected results can occur if other components are not used to relate the voltage on both sides of the transmission line to ground.

Physical Spec: Grounded Shield, Shorted, Improved Accuracy (Closed Form): TLSCP

Symbol



Summary

TLSCP simulates an ideal short-circuited transmission line with loss. Mode propagating in line is specified by its effective dielectric constant and per-unit-length attenuation at a user-specified frequency. This model scales loss with evaluation frequency. By default, TLSCP provides partial account for circuit loss (insertion loss only). You can toggle the secondary parameter LossModel to obtain access to insertion loss, return loss, and dielectric loss (for example; for microstrips or stripline).

Parameters

Name	Description	Unit Type	Default
ID	Name	Text	TL1
Z0	Transmission line impedance	Resistance	50 ohm
L	Transmission line length	Length	0 um
Eeff	Relative effective dielectric constant		1
Loss	Loss (dB/m)		0 dB/m
F0	Frequency for scaling loss	Frequency	0 GHz
*LossModel	Partial/Full account for conductor and dielectric loss		Partial
*TanD	Dielectric loss tangent		0
*Mur	Relative dielectric permeability		1
*TanM	Dielectric magnetic loss tangent		0
*Sigma	Dielectric bulk conductivity (S/m)		0
*EpsFreqDep	Dielectric constant is frequency dependent (No/Yes)		No
*Fmin	Low roll-off frequency of Tand frequency dependence	Frequency	1kHz
*Fmax	High roll-off frequency of Tand frequency dependence	Frequency	1THz
*Fspec	Er and Tand are specified at this frequency	Frequency	1GHz

** indicates a hidden secondary parameter*

Parameter Details

Note that parameters marked by asterisk (*) are hidden/invisible while the control parameter LossModel=Partial (default value). Setting LossModel=Full displays these secondary parameters in the model's Element Options dialog box (except parameters Fmin, Fmax, and Fspec which are visible only when parameter EpsFreqDep=Yes).

Loss and F0. Determine the frequency dependence of the attenuation constant. If F0 is not equal to zero (0.0), then

$$\alpha = \text{Loss}\sqrt{\text{freq} / F0}$$

(dB/m) where *freq* is the evaluation frequency. If F0 is equal to zero, the attenuation is constant versus frequency and is equal to Loss(dB/m).

LossModel is a secondary control parameter that defines how the model accounts for loss and dielectric parameters. If LossModel=Partial (default value) than the model accounts for insertion loss (parameter Loss) only. If LossModel=Full then the model accounts for insertion loss, return loss, dielectric loss, and for frequency dependent model of dielectric parameters (for example, for microstrips, stripline, or other).

Note: Secondary parameter LossModel is always visible at model Properties, tab Parameters if Show Secondary is selected. If LossModel=Partial, other secondary parameters are also visible at tab Parameters but they are not used by the model and do not appear at the schematic until LossModel=Full is selected.

EpsFreqDep is a (hidden, secondary) control parameter that defines if dielectric constant and dielectric loss tangent exhibit predefined frequency dependence, suggested in [1]. This dependence is controlled by the three frequency parameters Fmin, Fmax, and Fspec, and imparts causal behavior to the model. Fmin, Fmax, and Fspec are hidden (invisible) if EpsFreqDep=No.

Layout

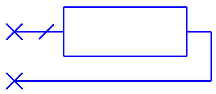
This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

References

[1] A.R. Djordjevic et al., "Wideband Frequency-Domain Characterization of FR-4 and Time-Domain Causality", IEEE Trans. of Electromagn. Compat., Vol. 43, No. 4, Nov. 2001

Physical Spec: Floating Shield, Shorted, Improved Accuracy (Closed Form): TLSCP2

Symbol



Summary

TLSCP2 simulates a transmission short-circuited line with isolated (floating) ground terminals. The characteristics of the transmission line are specified as physical length and electrical specification of the characteristic impedance and propagation constant. By default, this model provides partial account for circuit loss (insertion loss only). You can toggle the secondary parameter LossModel to obtain access to insertion loss, return loss, and dielectric loss (for example; for microstrips or stripline). Use extreme care with this element as it is meant to work in concert with additional elements which relate the voltages at ports of the transmission line to the global ground. Unusual and unexpected behavior can result if these additional elements are omitted.

Parameters

Name	Description	Unit Type	Default
ID	Name	Text	TL1
Z0	Transmission line impedance	Resistance	50 ohm
L	Transmission line length	Length	0 um
Eeff	Relative effective dielectric constant		0
Loss	Loss (db/meter)		0
F0	Frequency for scaling losses	Frequency	0 GHz
*LossModel	Partial/Full account for conductor and dielectric loss		Partial
*TanD	Dielectric loss tangent		0
*Mur	Relative dielectric permeability		1
*TanM	Dielectric magnetic loss tangent		0
*Sigma	Dielectric bulk conductivity (S/m)		0
*EpsFreqDep	Dielectric constant is frequency dependent (No/Yes)		No
*Fmin	Low roll-off frequency of Tand frequency dependence	Frequency	1kHz
*Fmax	High roll-off frequency of Tand frequency dependence	Frequency	1THz
*Fspec	Er and Tand are specified at this frequency	Frequency	1GHz

** indicates a hidden secondary parameter*

Equivalent Circuit

See the TLINP4 equivalent circuit.

Parameter Details

Note that parameters marked by asterisk (*) are hidden/invisible while the control parameter LossModel=Partial (default value). Setting LossModel=Full displays these secondary parameters in the model's Element Options dialog box (except parameters Fmin, Fmax, and Fspec which are visible only when parameter EpsFreqDep=Yes).

Loss and **F0**. Determine the frequency dependence of the attenuation constant. If F0 is not equal to zero (0.0), then

$$\alpha = \mathrm{Loss}\sqrt{\mathrm{freq} / F0}$$

(dB/m) where *freq* is the evaluation frequency. If F0 is equal to zero, the attenuation is constant versus frequency and is equal to Loss(dB/m).

LossModel is a secondary control parameter that defines how the model accounts for loss and dielectric parameters. If LossModel=Partial (default value) than the model accounts for insertion loss (parameter Loss) only. If LossModel=Full then the model accounts for insertion loss, return loss, dielectric loss, and for frequency dependent model of dielectric parameters (for example, for microstrips, stripline, or other).

Note: Secondary parameter LossModel is always visible at model Properties, tab Parameters if Show Secondary is selected. If LossModel=Partial, other secondary parameters are also visible at tab Parameters but they are not used by the model and do not appear at the schematic until LossModel=Full is selected.

EpsFreqDep is a (hidden, secondary) control parameter that defines if dielectric constant and dielectric loss tangent exhibit predefined frequency dependence, suggested in [1]. This dependence is controlled by the three frequency parameters Fmin, Fmax, and Fspec, and imparts causal behavior to the model. Fmin, Fmax, and Fspec are hidden (invisible) if EpsFreqDep=No.

Implementation Details

See the TLINP4 "Implementation Details" section.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

Recommendations for Use

This model, along with additional components can be used to model transmission line baluns and transmission line transformers in which one of the conductors is shielded from ground, like a coaxial line.

NOTE: Because the model definition does not include interactions with the ground, unusual and unexpected results can occur if other components are not used to relate the voltage on both sides of the transmission line to ground.

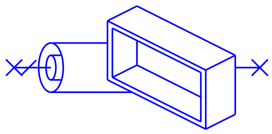
References

[1] A.R. Djordjevic et al., "Wideband Frequency-Domain Characterization of FR-4 and Time-Domain Causality", IEEE Trans. of Electromagn. Compat., Vol. 43, No. 4, Nov. 2001

Waveguide

Ideal Coaxial-Rectangular Waveguide Transition (TE10): COAXRWG_TE10

Symbol



Summary

COAXRWG_TE10 is a behavioral model that represents an ideal matched transition between transmission lines featuring different characteristic impedances of coaxial and TE10 rectangular waveguide.

Parameters

Name	Description	Unit Type	Default
Wa	Rectangular waveguide width	Length	22.86 mm
Wb	Rectangular waveguide height	Length	10.16 mm
Er	Relative dielectric constant filling the waveguide		1
Tand	Loss tangent of dielectric filling the waveguide		0
*Mur	Relative permeability filling the waveguide		1
*Tanm	Magnetic loss tangent filling the waveguide		0
*Sigma	Dielectric conductivity filling the waveguide	Siemens/m	0
ZCoax	Coaxial characteristic impedance	Resistance	50 ohm
*ZCalc	Switch - selector of TE10 characteristic impedance definition ("Power-Voltage"/"Voltage-Current"/"Normalized")		Power-Voltage

** indicates a secondary parameter*

Parameter Details

Er, Tand, Mur, Tanm, Sigma. The material properties of the media filling the waveguide.

Zcalc. Allows you to select a definition of characteristic impedance of the TE10 mode propagating in a rectangular waveguide with dimensions Wa*Wb. Options include "Power-Voltage" ,"Voltage-Current", and "Normalized." The default option is "Power-Voltage." This model uses the value of characteristic impedance to denormalize the computed normalized y-matrix of modeled discontinuity. Note that this selection must match the selection of the same parameter in theRWGIRIS_TE10, RWG_TEmn, and RWGT_TEmn elements used around the same schematic.

The characteristic impedance definitions used [1] are:

$$Z_{\text{power-voltage}} = Z\frac{2W_b}{W_a}$$

$$Z_{\text{voltage-current}} = Z\pi\frac{W_b}{2W_a}$$

$$Z = \frac{\eta}{\sqrt{1 - (\frac{f_c}{f})^2}}$$

f_c is the cutoff frequency for TEmn and f is the operational frequency; and η is the wave impedance of open space filled with waveguide dielectric.

$Z_{Normalized}=1$

Implementation Details

COAXRWG_TE10 is implemented as an impedance transformer between two physical transmission lines. The transmission line at port 1 has the frequency-independent user-supplied characteristic impedance of a coaxial line; and the transmission line at port 2 has the frequency-dependent characteristic impedance of a rectangular waveguide with a propagating mode TE10.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Recommendations for Use

COAXRWG_TE10 provides an ideal match at port 1 if port 2 is terminated to an infinite waveguide. To provide this termination, use RWGT_TEmn (m=1, n=0) model. Alternatively, use PORT_TN port model that refer to a subcircuit containing a single RWGT_TEmn.

Caution: This model is developed to work in a frequency range when only dominant TE10 propagates. It provides reasonable results out of this range, but in this case user discretion is advised.

Normalized characteristic impedance implies that waveguide mode is propagating. This means: Never set ZCalc to "Normalized" if your operational frequency gets into below-cutoff region.

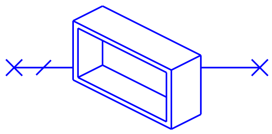
Keep in mind that the results depend on the selected definition of waveguide characteristic impedance.

References

[1] K. C. Gupta, Ramesh Garg, Rakesh Chadha, Computer Aided Design of Microwave Circuits, Artech House, Mass., 1981.

Rectangular Waveguide (TEmn): RWG_TEmn

Symbol



Summary

RWG_TEmn models a transmission line equivalent to a transverse electric waveguide mode of the order mn (TEmn) existing in a section of rectangular waveguide. The mode may be either propagating or evanescent (non-propagating); waveguide metal has a final conductivity and is filled with lossy dielectric.

Parameters

Name	Description	Unit Type	Default
Wa	Width of rectangular waveguide	Length	22860 um
Wb	Height of rectangular waveguide t	Length	10160 um
L	Length of rectangular waveguide l	Length	10000 um
M	Mode order (along Wa)		1
N	Mode order (along Wb		0
Er	Relative dielectric constant filling the waveguide		1
Rho	Metal bulk resistivity of waveguide metal normalized to copper		1
Tand	Loss tangent of dielectric filling the waveguide		0
*Mur	Relative permeability of dielectric filling the waveguide		1
*Tanm	Magnetic loss tangent of dielectric filling the waveguide		0
*Sigma	Dielectric conductivity of dielectric filling the waveguide	Siemens/m	0
*ZCalc	Switch - selector of TE10 characteristic impedance definition ("Power-Voltage"/"Voltage-Current"/"Normalized")		Power-Voltage

** indicates a secondary parameter*

Parameter Details

M, N. Parameters M and N make a pair of indices that define the TEmn mode selected for modeling. M represents the number of field variation along the Wa dimension; N provides the same relative to Wb.

Rho. The bulk resistivity of the waveguide metal. Note that this parameter is dimensionless because it represents the resistivity normalized to that of copper (i.e. to 1.7E-8 ohm/m.)

Er, Tand, Mur, Tanm, Sigma. The material properties of media filling the waveguide.

Zcalc. Allows you to select a definition of characteristic impedance of the TE10 mode propagating in a rectangular waveguide with dimensions Wa*Wb. Options include "Power-Voltage" ,"Voltage-Current", and "Normalized." The default option is "Power-Voltage." This model uses the value of characteristic impedance to denormalize the computed normalized y-matrix of modeled discontinuity. Note that this selection must match the selection of the same parameter in theRWGIRIS_TE10 and RWGT_TEmn elements used around the same schematic.

The following characteristic impedance definitions are used [1]:

$$Z_{\text{power-voltage}} = Z\frac{2W_b}{W_a}$$

$$Z_{\text{voltage-current}} = Z\pi\frac{W_b}{2W_a}$$

$$Z = \frac{\eta}{\sqrt{1-(\frac{f_c}{f})^2}}$$

Here, f_c is the cutoff frequency for TE10 and f is the operational frequency; η is the wave impedance of the open space filled with the waveguide dielectric.

$Z_{Normalized}=1$

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See “Assigning Artwork Cells to Layout of Schematic Elements” for details.

Recommendations for Use

NOTE: Results depend on the selected definition of waveguide characteristic impedance.

Normalized characteristic impedance implies that waveguide mode is propagating. **This means: Never set ZCalc to "Normalized" if your operational frequency gets into below-cutoff region.**

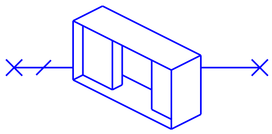
Ensure that the bulk resistivity of waveguide metal is normalized to correct the value of **copper** resistivity (**not gold**).

References

[1] K. C. Gupta, Ramesh Garg, Rakesh Chadha, Computer Aided Design of Microwave Circuits, Artech House, Mass., 1981.

Symmetrical Inductive Iris (TE10): RWGIRIS_TE10

Symbol



Summary

RWGIRIS_TE10 models the behavior of a thick, perfectly conducting, air-filled symmetrical, inductive diaphragm (metallic iris with internal edges parallel to vector E of a TE10 waveguide mode) represented as a discontinuity in a transmission line equivalent to the TE10 mode propagating in a rectangular waveguide. This model assumes that rectangular waveguides on both sides of the diaphragm have the same size and are filled with the same dielectric.

Parameters

Name	Description	Unit Type	Default
Wa	Width of rectangular waveguide	Length	22.86 mm
Wb	Height of rectangular waveguide t	Length	10.16 mm
IW	Width of iris aperture (window)	Length	5 mm
IT	Thickness of iris wall	Length	1.5 mm
M	Number of higher order modes accounted for discontinuity modeling		10
Er	Relative dielectric constant of dielectric filling the waveguide		1
Tand	Loss tangent of dielectric filling the waveguide		0
*Mur	Relative permeability of dielectric filling the waveguide		1
*Tanm	Magnetic loss tangent of dielectric filling the waveguide		0
*Sigma	Bulk conductivity of dielectric filling the waveguide	Siemens/m	0
*ZCalc	Switch - selector of TE10 characteristic impedance definition ("Power-Voltage"/"Voltage-Current"/"Normalized")		Power-Voltage

** indicates a secondary parameter*

Parameter Details

M. Sets the number of higher modes that the model uses for series expansions representing fields on the diaphragm window.

Er, Tand, Mur, Tanm, Sigma. The material properties of the media filling the waveguide. These parameters are not used in the evaluation of discontinuity properties; they are used only for calculation of the denormalizing characteristic impedance of a rectangular waveguide.

Zcalc. Allows you to select a definition of the characteristic impedance of the TE10 mode propagating in a rectangular waveguide with Wa x Wb dimensions. Options include "Power-Voltage" ,"Voltage-Current", and "Normalized." This model uses the value of characteristic impedance to denormalize the computed normalized y-matrix of the modeled discontinuity. This selection must match the selection of the same parameter in the RWG_TEmn and RWGT_TEmn elements used in the same schematic.

The characteristic impedance definitions [1] are:

$$Z_{\text{power-voltage}} = Z\frac{2W_b}{W_a}$$

$$Z_{\text{voltage-current}} = Z\pi\frac{W_b}{2W_a}$$

$$Z = \frac{\eta}{\sqrt{1-\left(\frac{f_c}{f}\right)^2}}$$

Here, f_c is the cutoff frequency for TE10 and f is the operational frequency; η is the wave impedance of the open space filled with the waveguide dielectric.

$Z_{\text{Normalized}}=1$

Parameter Restrictions and Recommendations

1. The values of Wa and Wb must provide propagation of the single mode TE10 within the evaluation frequency range.
2. The IT diaphragm thickness must be greater than zero.
3. The IW width must be greater or less than Wa.
4. The number of discontinuity modes M should be around 10. You can experiment and determine a value of M large enough (for example, M=M1) to provide stable results for all the values M>M1.

Implementation Details

This model implies that a diaphragm is placed between the two segments of identical rectangular waveguides that support the dominant mode TE10. A mode-matching method [1] is applied to evaluate the scattering properties of two-width steps separated by a short waveguide of length equal to the diaphragm thickness.

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Recommendations for Use

To obtain the best results in simulated application based on RWGIRIS_TE10 (filters etc.), you should use the matching termination provided by the RWGT_TEmn model. For this purpose, terminate the application with the PORT_TN port model. PORT_TN should refer to a subcircuit that contains a RWGT_TEmn model (m=1, n=0).

NOTE: The implementation of this model relies on the involved numerical algorithm. Large values of the M parameter (above 10-15) may result in an extended execution time.

This model is developed to work in a frequency range when only dominant TE10 propagates. It provides reasonable results out of this range, but in this case user discretion is advised.

Normalized characteristic impedance implies that waveguide mode is propagating. **This means: Never set ZCalc to "Normalized" if your operational frequency gets into below-cutoff region.**

Note that the results depend on the selected definition of a waveguide characteristic impedance.

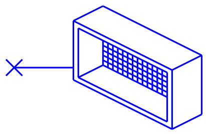
References

[1] K. C. Gupta, Ramesh Garg, Rakesh Chadha, Computer Aided Design of Microwave Circuits, Artech House, Mass., 1981.

[2] Tatsuo Itoh (Editor), Numerical Techniques for Millimeter-Wave Passive Structures, Chapter 9, John Wiley & Sons, New York, 1989.

Rectangular Waveguide Termination (TEmn): RWGT_TEmn

Symbol



Summary

RWG_TEmn models an input of a transmission line of infinite length. This line represents an equivalent of a transverse electric waveguide mode of the order mn (TEmn) existing in a rectangular waveguide. The mode may be either propagating or evanescent (non-propagating); waveguide metal has a final conductivity and is filled with lossy dielectric.

Parameters

Name	Description	Unit Type	Default
Wa	Width of rectangular waveguide	Length	22860 um
Wb	Height of rectangular waveguide t	Length	10160 um
M	Mode order (along Wa)		1
N	Mode order (along Wb		0
Er	Relative dielectric constant of dielectric filling the waveguide		1
Rho	Metal bulk resistivity of waveguide metal normalized to copper		1
Tand	Loss tangent of dielectric filling the waveguide		0
*Mur	Relative permeability of dielectric filling the waveguide		1
*Tanm	Magnetic loss tangent of dielectric filling the waveguide		0
*Sigma	Conductivity of dielectric filling the waveguide	Siemens/m	0
*ZCalc	Switch - selector of TE10 characteristic impedance definition ("Power-Voltage"/"Voltage-Current"/"Normalized")		Power-Voltage

** indicates a secondary parameter*

Parameter Details

M, N. Parameters M and N make a pair of indices that define the TEmn mode selected for modeling. M represents the number of field variation along the dimension Wa; N provides the same relative to Wb.

Rho. The bulk resistivity of the waveguide metal. Note that this parameter is dimensionless because it represents the resistivity normalized to that of copper (i.e. to 1.7E-8 ohm/m.)

Er, Tand, Mur, Tanm, Sigma. The material properties of media filling the waveguide.

Zcalc. Allows you to select a definition of characteristic impedance of the TE10 mode propagating in a rectangular waveguide with dimensions Wa*Wb. Options include "Power-Voltage" ,"Voltage-Current", and "Normalized." This model uses the value of characteristic impedance to denormalize the computed normalized y-matrix of modeled discontinuity. Note that this selection must match the selection of the same parameter in the RWGIRIS_TE10 and RWG_TEmn elements used around the same schematic.

The following characteristic impedance definitions are used [1]:

$$Z_{\text{power-voltage}} = Z\frac{2W_b}{W_a}$$

$$Z_{\text{voltage-current}} = Z\pi\frac{W_b}{2W_a}$$

$$Z = \frac{\eta}{\sqrt{1-(\frac{f_c}{f})^2}}$$

Here, f_c is the cutoff frequency for TE10 and f is the operational frequency; η is the wave impedance of the open space filled with the waveguide dielectric.

$Z_{Normalized}=1$

Layout

This element does not have an assigned layout cell. You can assign artwork cells to any element. See "Assigning Artwork Cells to Layout of Schematic Elements" for details.

Recommendations for Use

This model may be used as a subcircuit referenced in the PORT_TN port. This usage provides a port with a frequency-depended termination.

NOTE: Results depend on the selected definition of waveguide characteristic impedance.

Normalized characteristic impedance implies that waveguide mode is propagating. **Never set ZCalc to "Normalized" if your operational frequency gets into below-cutoff region.**

Ensure that the bulk resistivity of waveguide metal is normalized to the correct value of **copper** resistivity (**not gold**).

References

[1] K. C. Gupta, Ramesh Garg, Rakesh Chadha, Computer Aided Design of Microwave Circuits, Artech House, Mass., 1981.

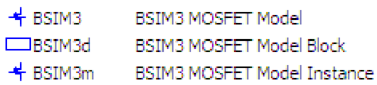
Appendix A. Supplemental Model Information

The following information regarding microstrip iCells, linear models for transmission line systems, and EM-based discontinuity models supplements the information provided in this catalog.

Models and Model Blocks

For most models in the NI AWRDE, you define the model parameters directly for each model. One element can also get model parameters from another element. A good example is transmission line models where you define the geometry of the model and then point to a substrate block. In this manner, the substrate parameters are only defined once, not per block.

You can use the same approach for some nonlinear models. In the model library, for some nonlinear models, there are three different choices for the same model as shown in the following figure for the BSIM3 model.



The element name by itself is the model with all the model parameters defined directly on the element.

The element name ending with a "d" is the model block element. This block is used only to define the element parameters. This element has no nodes since it is not wired into the schematic.

The element name ending with a "m" is the model instance element. This block is used to define parameters specific to the instance and has a parameter for the name of the model block instance.

Microstrip iCells

Microstrip iCells, which are identified by an element name in the form M****$, have the following attributes:

Symbol

The symbol is the same as the corresponding non-iCell element.

Topology

The topology is the same as the corresponding non-iCell element.

Parameters

iCells are parameter-less elements that automatically and dynamically size the width at each port to the width (w) parameter of the element attached to it. For example, if an MSTEP$ iCell has an MLIN attached to port 1 with a width of w=10 mils and an MLIN attached to port 2 with a width of w=15 mils, the iCell automatically sizes the width of port 1 of the MSTEP$ to 10mils and the width of port 2 to 15mils. As a default, iCells automatically size to elements that have only one w parameter.

Linear Models for Transmission Line Systems

This section provides information about the various types of linear models currently available in the Element Browser to use in modeling distributed transmission line systems. It addresses basic methods and limitations of each general model type, providing you with the information required to choose between two models of different types.

In general, linear models of a given distributed transmission line system fit into the following categories:

- Discontinuity Models: These models are meant to predict electrical effects of the connections between transmission line sections. These models include bends, dimensional changes and the union of up to four transmission lines at a single junction.
- Transmission Line Models: These models are meant to model the lengths of lines between connections. These models include single lines and coupled lines.

These model types are described in the next two sections. Each section addresses the general types of models available and the limitations and advantages of models developed in that manner. Finally, a suggestion for the preferred approach to the design process is addressed in regard to the type of model used in various locations throughout the design process.

Discontinuity Models

Quasi-Static or Empirical Models from Measured Data

The base set of linear models for discontinuities available in Microwave Office/Analog Office (MWO) are based on closed-form approximations of the electrical characteristics based on a quasi-static solution of an approximate problem, or upon fitting an equation to a limited amount of measured data. These models were developed throughout the years (late 1940's to present) by esteemed members of our profession and are typically published in technical literature. From a computer-aided design point of view, this closed-form implementation of these models is well suited to the vast number of evaluations required when optimizing or tuning over frequency. However, the accuracy of the approximation of electrical parameters varies from model to model and typically it also varies over frequency and the input parameter ranges.

- Models Based Upon Measured Data: The accuracy of models based on equations fit to a set of measured data can be far ranging and highly dependent upon the skill of the originator of the model. In each model of this type, NI AWR has evaluated all of the models available in the public domain literature and has selected those which give the most accurate results. Considering the complexity of the problem, the quality of the models constructed is amazing. Consider for example, a lossless two-port device, such as a microstrip symmetric step in width, that has been parameterized by a limited set of five inputs (ε_r, h, freq, w1 and w2). For a lossless two-port, the electrical characteristics of the device can be uniquely described by deriving an equation for three parameters as a function of these five inputs. Now the selection of the model parameters is part of the art-in-model development: they might be S-parameters, inductances, capacitances or any other similar parameter. Once determined, you must then measure these three parameters as a function of the five input parameters. Thus you have three output parameters, which are mapped to five input parameters at discrete points (you have sampled the three output parameters in five spaces). You have to design an equation for each of these output parameters, which minimizes the error throughout this five-dimensional space. The amazing part about the development of these models is the time frame for their development (40's and on) -- a majority of this was done with a slotted line and slide rule. Having shown the daunting task associated with development of such models, you should look at their limitations. Models based on equations fit to a limited set of measured data tend to have good accuracy over the range of the parameter over which they are fit. However, these models are typically extended to approximate the performance over a wider range of input parameter ranges than the measured data set. For example, it is typical to have such a model developed on one dielectric constant (typically alumina) and then have the model extended to other dielectric constants via a scaling or other transformation of the original equations. Once we depart from the original data set on which these models are based, the associated error in the model increases. Finally, a model based on measured results tends to duplicate the original measurement error. As you can imagine, the quality and range of measurements has improved over the years. The implementation of Automatic Network Analyzers (ANA) and multi-term calibration methods has allowed the accuracy and frequency of measurement to extend to levels unheard of during the time of the development of some of these models. Thus, as you would expect, the accuracy and frequency range of the original data set on which the models were based results in degraded accuracy. This is especially true as you extend the model to frequencies beyond the original data set.

- Models Based on Closed-Form Quasi-Static Solutions: Models based on closed-form quasi-static evaluations of discontinuities have excellent performance at low frequencies. However, as the dimensions of the modeled component become large when compared to a wavelength, the accuracy of the model degrades. This results from the modeling of the stored energy of the discontinuity as equivalent capacitances or inductances where the actual reactance results from stored energy in higher-order evanescent (below cut-off) modes of the guiding structure. As in metallic waveguides, the frequency-dependence of these higher order modes do not have the same dependence as that of the modeled reactive components which cause the model to degrade over frequency. Further, the closed-form solution of Maxwell's equations typically require that the guiding structure and resulting fields conform to a standard coordinate system or a coordinate system which can be mapped to one of these systems. There is an extremely limited number of useful discontinuities in which this occurs, and most lie in the coaxial or metallic waveguide class of transmission lines. To further extend this technique to include some of the standard transmission line systems such as microstrip and stripline, an approximation of the transmission line system is used which fits into one of these coordinate systems. For example, multiple microstrip and stripline discontinuities are based on a parallel plate waveguide model, which consists of a rectangular waveguide with magnetic walls on the two sides and conductive walls on the top and the bottom. A closed-form solution of the quasi-static fields can be determined for this discontinuity and thus a lumped element equivalent circuit can be determined. However, in reaching this model, you have assumed that the approximate transmission line system has all of the traits of the original system. This difference in the actual and assumed transmission line systems results in simulation errors within the model. In some instances, the model developed using a quasi-static solution of an approximate model is augmented with measured data in order to minimize the associated error. While this is an improved model, accuracy will still suffer as frequency increases and as you extend the model out of the range of measurements taken.

Mode-Matching Models

A mode-matching model is very similar to a closed-form quasi-static model as discussed previously. The only difference in this method is that a full-wave solution of the model is developed using the higher order evanescent (below cut-off) modes. The term "mode-matching" originates from the fact that the fields at the discontinuity interface are expanded as a summation of the modes in each of the waveguide systems. The values of these modes are then determined such that the E and H fields at the interface are continuous (match). Due to the high number of calculations required in the mode-matching technique, you will discover that these models are not as well suited for tuning and optimization as the closed-form expressions shown in the previous section.

This form of model has excellent simulation performance for transmission line systems, which can be expressed or mapped to a regular coordinate system where expressions for all modes of the structure can be determined. However, this limits you to the use of this technique for coaxial and metallic waveguides. As with the closed-form quasi-static models, an approximate parallel plate waveguide structure can be used to simulate microstrip and stripline discontinuities and a full-wave closed-form solution can be determined. An error in the simulation results from using the approximate parallel plate waveguide to model the fields of a microstrip or stripline discontinuity.

MWO currently implements four mode-matching models, which model a step and an offset step in the conductor width of a microstrip and stripline. NI AWR determined that the computational expense required in these models does not result in drastic reductions in the modeling error when compared to a full EM simulation of the discontinuity and the quasi-static model. However, the microstrip and stripline offset step models available have no counterpart in a quasi-static solution and prove useful when you encounter this situation.

EM-based Models (X-models)

The X-models are a group of discontinuity models that use the results of full-wave electromagnetic solutions of the parameterized discontinuity in order to estimate the electrical performance of the discontinuity. These models are a result of ongoing internal research and development at NI AWR and are intended to give you the most accurate discontinuity models at a computational speed adequate for tuning, optimization and yield analysis. See "X-models" for details on using X-models.

Although full-wave electromagnetic solutions are extremely time-consuming, the X-models only perform these simulations at predetermined sets of input parameters and the results are stored in a database. This database is then used to interpolate the electrical response from the stored data. Further, these models allow you to automatically generate the full database for a given discontinuity on a given substrate without a user presence. This allows you to generate the entire table for all discontinuities on a given substrate overnight or on the weekend. Once the database is filled, interpolation of the resulting electrical characteristics proceeds rather quickly.

If an X-model is available for the discontinuity you are using, this is the most accurate of all models available. As this internal research and development project at NI AWR continues, you can expect to find more EM-based models available in MWO.

Suggestions in the Use of Discontinuity Models

With a general knowledge of the advantages and limitations of the discontinuity models offered in MWO, the following procedure for selecting the appropriate model is suggested:

1. In the initial conceptual stage of the design, the use of the closed-form discontinuities or no discontinuity model at all is suggested. In this stage, rapid evaluation and optimization of the circuit performance can be obtained, and several topologies investigated.
2. After a topology decision and an initial estimate of the input parameters is obtained, the discontinuity models can be added to increase the accuracy of the simulation. In this stage, the X-models should be used if available. A great time savings and reduction of data entry errors can be obtained by use of the intelligent discontinuity models which sense needed information from the surrounding parts.
3. The problem should then be re-optimized, correcting widths and lengths of the transmission line sections to negate the effects of the discontinues. If warnings occur for the discontinuity models, you should address them by changes in the circuit to eliminate the warning, or the discontinuity should be simulated using the EM simulator.

Discontinuity models function most accurately when attached to lines that match their corresponding edges. Directly connecting discontinuity models to one another reduces their accuracy.

The following table summarizes the discontinuity models sorted by system and model type:

Transmission Line System	Closed-Form Transmission Line Models	EM-based Models and Mode Matching Models
General	GND_STRAP, VIA, WIRE, RIBBON	
Interconnects	TVIA	
Lumped Element	TFC	
Microstrip	MBENDA, MCROSS, MCURVE, MLEF, MLSC, MSTEP, MTAPER, MTEE, TFC2, TFR	MBEND90X, MCROSSX, MLEFX, MOPENX, MRINDSBR, MSTEPX, MTEEX

Transmission Line System	Closed-Form Transmission Line Models	EM-based Models and Mode Matching Models
Stripline	SBEND, SCROSS, SCURVE, SGAP, SHOLE, SLEF, SLSC, SMITER, SSTEP, STAPER, STEE	SSTEPO, SSTEP2

Transmission Line Models

Closed-Form Transmission Line Models

Closed-form equations for the equivalent parameters of a given transmission line have been developed throughout history in the same manner as the discontinuity models previously mentioned. When compared to discontinuities, a greater variety of transmission lines either fit into a regular coordinate system or can be mapped into one of these systems. This is due to reduction of the problem to a 2-dimensional one, since the transmission line is assumed to be infinitely long with no changes in the transverse dimensions of the structure. Closed-form and quasi-static solutions of the characteristic parameters of a transmission line system have been determined for microstrip, stripline and co-planar waveguide among others. Typically, these systems make approximations about the problem being solved in order to reach these closed-form solutions. Such approximations may come in the form of one of the following: perfect electrical conductors, conductor thickness equal to zero, or transverse electro-magnetic (TEM) wave. Beginning with these base forms of the equations, corrections based on variational methods, measurements results, or full-wave electromagnetic simulation are used to estimate the effects of losses, metal thickness and dispersion. Again, the accuracy of these corrections varies considerably from one application to the next and must be viewed on an individual basis.

Although transmission line models are more easily obtained than discontinuity models for the same system, not all desired transmission line models can be obtained in this manner. Typically, these include coupled line models, which do not have a plane of symmetry. Often, these models are obtained by fitting equations to samples of measured data or data obtained via numerical electromagnetic simulations. The range of the sampled data and the skill of author at fitting the equations once again limits the accuracy of such models.

2-Dimensional Method of Moments Based Models (Advanced Numerical Models)

LINPAR is a commercially available stand-alone software package from Artech House Publishers. The primary function of this software is to determine the electrical characteristics of transmission line models such as the characteristic impedance, propagation and attenuation constants for single lines. Further, this same analysis can be used to determine the characteristics of coupled- or multiple-coupled lines. This analysis is performed via a quasi-static electromagnetic simulation of the cross-section of the transmission line section. The method of analysis used is the Method of Moments (MoM) implementing the Galerkin Method of basis and testing functions. MWO has integrated this simulation engine into a variety of transmission line topologies available to you.

The major benefit of these quasi-static MoM transmission models is the ability to model many transmission line systems which do not have adequate closed-form solutions. Examples of the types of transmission line systems that can be modeled using this method include: multiple coupled line of all types, co-planar waveguide (CPW), coupled CPW, suspended stripline, covered microstrip, and inverted microstrip. Further, this method also allows you to model the effects of parameters sometimes neglected or approximated in the closed-form solutions. An example of this is the ability of this method to model thick conductors.

The limitations of transmission line models incorporating this analysis are primarily due to two factors. First, calculation of the electrical parameters of the model requires an electromagnetic simulation, which can be computationally expensive. You should expect the simulation speed to decrease when using one of these models. Secondly, the performed electromagnetic simulation is only quasi-static, so you should expect the results to degrade when the cross-sectional dimensions of the transmission line approach a fraction of a wavelength. In transmission lines, this deviation from the quasi-static or DC characteristics is termed "dispersion". Importantly, you should realize that transmission line models constructed via a closed-form solution can model dispersion.

When choosing between a transmission line model developed using a closed-form solution and a quasi-static MoM, you must determine if the effects of dispersion will significantly influence the results. Importantly, dispersion only occurs for mixed dielectric transmission line structures such as microstrip. Single dielectric transmission lines such as coax and stripline only have dispersion as second order effects due to the frequency-dependent materials of which it is constructed.

Suggestions in the Use of Distributed Transmission Line Models

With a general knowledge of the advantages and limitations of the transmission line models offered in MWO, the following procedure for selecting the appropriate model is suggested.

1. Before entering the initial design stage, compare the difference in performance of all available models over the design frequency range and at a frequency where dispersion can be ignored. This allows you to evaluate the electrical effects of dispersion and of the additional parameters available with the numerical models.
2. Once a familiarity with the model limitations is determined for your range of input parameters, an engineering judgment can be made on which model you should use for the initial design phase.
3. Finally, further accuracy can be obtained from this initial investigation by adding correction factors to accommodate the effects of dispersion or other input values. For example, you might notice that the addition of conductor thickness changes the even and odd mode impedance of a pair of coupled lines by ~2ohms. Thus, an effective change in the dispersion model parameters can then be obtained to realize this same change.

The following table summarizes distributed transmission line models sorted by system.

Transmission Line System	Closed-Form Transmission Line Models	2D MoM-Based Models (Advanced Numerical Models)
Coplanar Waveguide		CPW1LIN, CPW2LIN, CPW3LIN
Microstrip	MLIN, MACLIN, MCLIN, MCFIL	M2CLIN, M3CLIN, M4CLIN, M5CLIN, M6CLIN, M7CLIN, M8CLIN,MEMLI
Stripline	SLIN, SCLIN	S1LIN, S2CLIN, S3CLIN, SEMLIN

X-models

EM-based models leverage the tight integration of the EM simulator (EMSight) and linear simulator in MWO to interactively construct discontinuity models based on full-wave electromagnetic simulations of the parameterized structure. The model automatically constructs and simulates the discontinuity using EMSight. A database is constructed from the EM simulations and then interpolated for the specific data points. When all points are filled in the database and saved in a file, the simulation is extremely fast with EM-based accuracy. The models require the EMSight option in MWO to be functional. An addition advantage of X-models is they can predict when the model's accuracy will start to decay, something that closed-form models cannot do. See "Upper Frequency Limitations" for more details.

Using X-models

When using X-models, you should understand three issues.

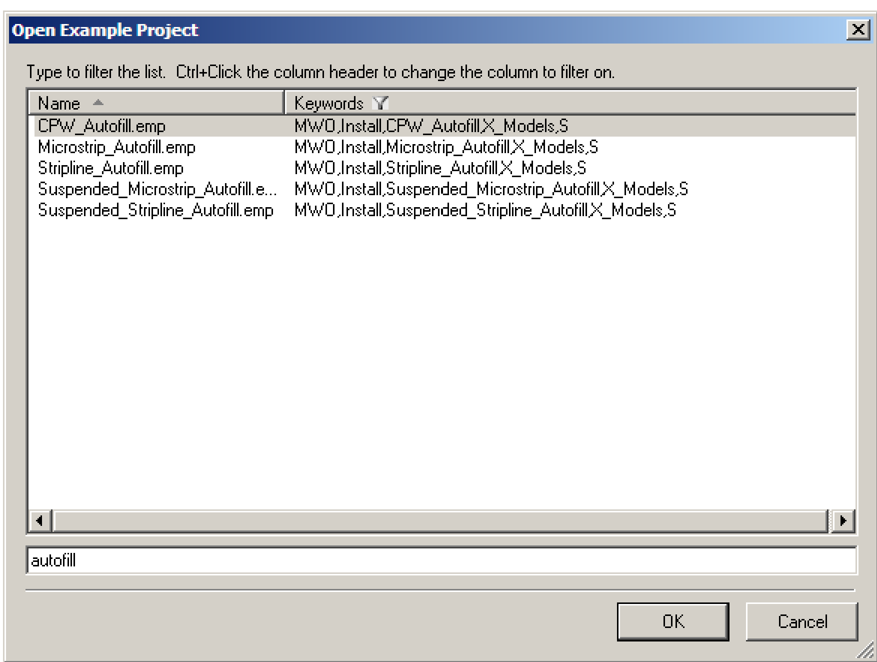
1. Which models are X-models
2. How to properly set up substrate parameters for X-models
3. How to fill a new database

Which Models are X-models

X-models have model names that end in either X or X$. Currently they fall into one of five substrate types: Microstrip, Stripline, CPW, Suspended Microstrip, or Suspended Stripline.

The simplest way to view all of the X-models for a given substrate type is to open the standard example shipped with the software to help generate new X-models (example specifics are discussed in a further section):

1. Choose **Help > Open Example**.
2. In the Open Example Project dialog box that opens, type "autofill" to filter for only the X-model autofill examples.

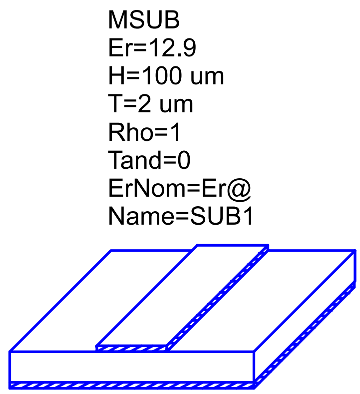


3. Open the specific type you are interested in and view the only schematic in those projects. This schematic contains all of the X-models for that given type.

How to Properly Set Up Substrate Parameters for X-models

There are many X-model database files shipped with the NI AWRDE installation, see “X-model Tables Shipped with the NI AWRDE” for the exact list. If your substrate values match any of these, there will be no issues using X-models in your first simulation. However, if your substrate parameters do not match a shipping substrate, simulation cannot occur and a "The X-model Database has not been filled" error message displays. There are two options to correct this error.

The simplest way to correct this error is to understand the relationship between the X-model fixed parameters and statistical parameters. For details on these parameter definitions, see “Interpolation Database Concept”. For microstrip, the fixed parameter is ErNom and its statistical parameter is Er. By default, the ErNom parameter has the same value as the Er value, as shown in the following example by using the syntax "Er@".



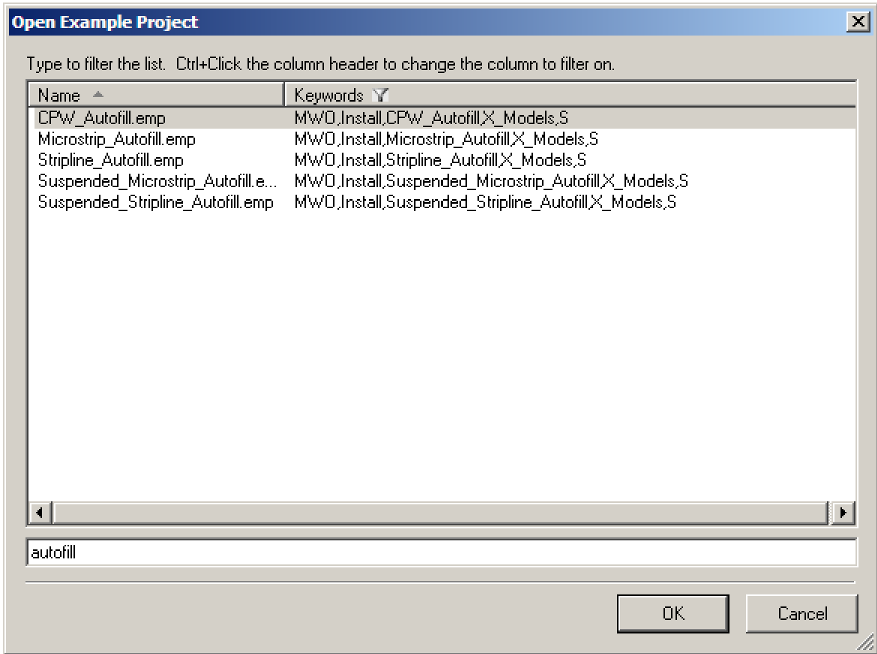
This syntax indicates "use the value of the Er parameter". However, you can use any value for the ErNom parameter. The fixed parameter (ErNom for microstrip) chooses the proper X-model database to use and the Er parameter is used in the simulation. These values can be 10% different and still be accurate. You can also set your fixed parameter to one of those shipped with the software as long as the Er you are using is less than 10% different than this value. For example, if you want to simulate with Er=10 and see that there is a 10.2 and 9.8 available in the shipping substrates, you can set Er=10 and ErNom = 10.2 and the X-models will work correctly.

The second way to correct this problem is to fill new X-model database tables. Since this is a lengthy process (but only has to be done once per substrate values), NI AWR recommends using the first approach before filling a new database. The following section discusses how to fill a new database.

How to Fill a New Database

When you decide you want to fill a new X-model database, NI AWR strongly suggests using the autofill examples shipped with the NI AWR software to do so. These examples come with design notes to help you know exactly what to do, and scripts to help you give the X-model files meaningful names after they are filled. These examples automatically fill the entire database for a particular set of fixed parameters. In this way, all of the required EM simulations can be completed and stored without user interaction. Typically, you should "autofill" all of the discontinuity models for one substrate at one time. To find these examples:

1. Choose **Help > Open Example**.
2. In the Open Example Project dialog box, type "autofill" to filter for only the X-model autofill examples.



3. Open the specific type you are interested in filling and read the design notes.

After the autofill example is done, the new X-model database files are ready to use with the project. See "X-model Filenames and Locations" for details on the X-model names and file locations.

The software decides to fill a new X-model database based on the substrate settings and the Autofill parameter on each X-model. If the proper X-model database files do not exist for a model and its substrate values, and Autofill is set to 1, then the files for that model are created. A common mistake is to use one X-model in a design, (for example, a MBEND90X) when the database has not been filled. Based on the error, you change the Autofill parameter to 1 and wait for a long time while that specific table fills. If you then decide to use a different X-model such as the MTEEX, for example, then that database needs to fill. It is much better to use the autofill examples and run them overnight so you know all tables for a given substrate are filled and ready.

Using X-models with Tuning/Yield/Optimization

Because there are fixed and statistical parameters for the substrates that X-models use, you can vary the statistical parameters using Tuning, Yield, and Optimization. The models issue a warning when the statistical value varies 10% from the fixed value. The default configuration for the fixed parameters using the "@Er" type syntax does not work with Tuning, Yield, and Optimization. You must change these values (ending in Nom) to the fixed value number. If you had to change the fixed parameters to find the right X-model database files, then you don't need to do anything different. If you did not have to do this, simply set the Nom parameters to be identical to the non-Nom parameter names (Er and ErNom, for example) and then vary the non-Nom parameter. If you don't make this change one of two things will happen. If the Autofill on the individual X-models is set to 1 (not the default) then a new database file must be filled at each iteration. If Autofill is set to 0, simulation errors about database files not being filled display.

X-model Tables Shipped with the NI AWRDE

The following sections list the X-models and Er values shipped with the NI AWRDE.

Microstrip

The following substrates are filled for the MSTEPX, MTEEX, MBENDRWX, MGAPX, MCROSSX, MBEND90X, MLEFX and MOPENX models:

```
Er = 2.2
Er = 2.33
Er = 2.45
Er = 2.5
Er = 2.94
Er = 3
Er = 3.2
Er = 3.25
Er = 3.38
Er = 3.4
Er = 3.48
Er = 3.66
Er = 4.5
Er = 4.99
Er = 6
Er = 6.15
Er = 9.2
Er = 9.7
Er = 9.8
Er = 10.2
Er = 10.8
Er = 11.2
Er = 12.4
Er = 12.9
```

Stripline

The SGAPX model has the same values as the microstrip models.

Coplanar Waveguide

The following substrates are filled for the CPWABRGX, CPWBENDX, and CPWTEEX models:

```
Er = 3.38, H=813um, Hab=35.6um, No Cover, No Gnd
Er = 3.38, H=1000um, Hab=50um, NoCover, No Gnd
Er = 9.8, H=508um, Hab=25.4um, HCover=508um, Gnd
Er = 12.4, H=280um, Hab=1.5um, No Cover, Gnd
Er = 12.9, H=100um, Hab=3um, No Cover, No Gnd
```

The following substrates are filled for the CPWLINX model:

```
Er = 3.38, H=813um, Hab=35.6um, No Cover, No Gnd, T=35.6um, Rho=1, Tand=0, Acc_10
Er = 3.38, H=1000um, Hab=50um, No Cover, No Gnd, T=0um, Rho=1, Tand=0, Acc_10
Er = 9.8, H=508um, Hab=25.4um, HCover=508um, Gnd, T=0um, Rho=0, Tand=0, Acc_10
```

```
Er = 12.4, H=280um, Hab=1.5um, No Cover, Gnd, T=0um, Rho=1, Tand=0, Acc_10
Er = 12.9, H=100um, Hab=3um, No Cover, No Gnd, T=2um, Rho=1, Tand=0.001, Acc_10
```

Suspended Microstrip

The following substrates are filled for the MSBND90X, MSCROSSX, MSOPENX, MSSTEPX, and MSTEEX, models:

```
Er = 3.38, Tand=0, H1=500um, H2=1000um
```

The following substrates are filled for the MS1LINX, MS2CLINX and MSBCPLX models:

```
Er = 3.38, Tand=0, H1=500um, H2=1000um, T=25um, Rho=1, Acc=1
```

Suspended Stripline

The following substrates are filled for the SSBND90X, SSCROSSX, SSOPENX, SSSTEPX, and SSTEEX models:

```
Er = 3.38, Tand=0, H1=1000um, H2=500um, H3=1000um
```

The following substrates are filled for the SS1LINX, SS2CLINX and SSBCPLX models:

```
Er = 3.38, Tand=0, H1=1000um, H2=500um, H3=1000um, T=25um, Rho=1, Acc=1
```

X-model Specifics

The following sections discuss X-model specifications.

Upper Frequency Limitations

The parasitic reactance associated with discontinuities is the circuit realization of stored energy in evanescent higher-order modes of the structure. As the frequency approaches the cutoff frequency of the first higher-order mode of the structure, the stored energy increases dramatically, making this structure unsuitable for typical circuit design. The upper frequency limit of this model is set at 80% of an estimate of the cutoff frequency of the first higher-order mode encountered in the discontinuity. For example, a microstrip line with a substrate thickness which is small compared to wavelength encounters a first higher mode when the effective width of the widest line of the structure is one-half of a guided wavelength. An approximation of this condition can be obtained with the following equation:

$$Fc=Z_o \cdot c/(2H \cdot 377) \tag{A.1}$$

where Z_o=Characteristic Impedance of TxLine with W/H = 4.0 and c = speed of light in the chosen units. The upper frequency limit is defined by the 80% of this calculated cutoff frequency:

$$Fmax=0.80 \cdot 20 \cdot c/(2H \cdot 377) \tag{A.2}$$

For previous frequencies, this maximum frequency, an extrapolation based upon results within the limits of the parameterized model, is used. A warning is issued indicating that results produced are not based on surrounding EM simulations. Typically, this warning is not seen during linear circuit design, but rather during harmonic balance-based nonlinear simulations where the number of harmonics used causes this limit to be violated.

X-model Filenames and Locations

In the program installation directory there is a folder named *EM_Models* that contains several defined X-models that NI AWR provides. These models are stored as binary (non-ASCII) files that have an *.EMX* extension, and are name coded to their function. The naming convention is <substrate>_<parameters>_<model>, where <substrate> is the type of substrate, <parameters> are the parameters used in filling the model, and <model> is the model name.

The location of newly user-filled X-models is the *EMModelUser* directory. To locate this directory choose **Help > Show Files/Directories**. You can set up NI AWRDE configurations so the *EMModelUser* directory is a different folder (such as a network path). This is useful if a group of designers wants to share a common set of X-models. See the *Installation Guide* for details on how to redirect folder names.

Although most substrate dielectric values are covered by the X-model files included with the NI AWRDE, there may be times when you need to create a new X-model file. When you do so, the file is assigned a name based on the discontinuity type and an incrementing 3-digit number to ensure that it is unique. For example, if a Microstrip 90-Degree Angle Bend model named "MS90B000.EMX" exists; if you create another model it is named "MS90B001.EMX". This naming method is useful for avoiding naming conflicts, however you should rename new X-model files as follows.

After creating a new X-model file, NI AWR recommends that you rename it to prevent future confusion about what the file may represent, especially when the files are shared within an organization, transferred between computers, or if you must save the models when upgrading the software. NI AWR recommends creating unique file names to distinguish them from other X-model files. For example, you can append your company name to the beginning of the file name and use the NI AWR convention for defining the discontinuity and substrate.

Note that the *.EMX* extension and directory location of the file are the only significant items in loading the files. The rest of the information is stored in the binary file and is used to load the correct file. Therefore, you can rename the file once created to apply user significance. For example, you could rename the *MS90B9_80.emx* file (the EM data for a 90-degree bend on alumina) to *AluminaBend.emx*, and the program still functions correctly.

The autofill examples that ship with the NI AWRDE contain scripts to help rename the files generated by the autofill process. See the design notes for these examples for information on running the scripts.

The following table lists all current AWR X-models and the name given to them automatically by the software when a new model is created.

Discontinuity Name	AutoName
CPWABRGX	CPABG000
CPWBENDX	CP90B000
CPWLINX	CPTX000
CPWTEEX	CPTA000/CPTB000
MBEND90X	MS90B000
MBENDRWX	MS90BW000
MCROSSX	MSXA000/MSXB000
MGAPX	MSGAP000
MLEFX	MSO000
MOPENX	MSO000
MSTEPX	MSS000
MTEEX	MSTA000/MSTB000

Discontinuity Name	AutoName
MS1LINX	MS1LIN000
MS2CLINX	MS2CLIN000
MSBCPLX	MSBC000
MSBND90X	MSSBND000
MSCROSSX	MSSXA000/MSSXB000
MSOPENX	MSSO000
MSSTEPX	MSSSTP000
MSTEEX	MSSTA000/MSSTB000
SIG1LNX	SIG1LNX000
SGAPX	SLGAP000
SS1LINX	SS1LIN000
SS2CLINX	SS2CLIN000
SSBCPLX	SSBC000
SSBND90X	SHBND000
SSCROSSX	SHXA000/SHXB000
SSOPENX	SHSO000
SSSTEPX	SHSTP000
SSTEEX	SHTEA000/SHTEB000

Interpolation Database Concept

The model operates via the interpolation of EM simulations of surrounding evaluation points of the model parameters. Importantly, the EM simulation at the surrounding points are stored to disk, eliminating the need to perform repetitive EM simulations at these points. The EM simulations are performed on an N-dimensional grid of the parameters. To limit the size of the database required for storage in RAM, the model parameters are divided into four groups consisting of the following:

Parameters	Description
Independent	Parameters that typically vary during the design process on one particular substrate.
Scalable	Parameters that relate to the independent parameters via a scaling of the structure to a different size.
Fixed	Parameters that do not vary in the design process.
Statistical	Parameters that do not vary in the design process but vary in manufacturing and require small changes for statistical and sensitivity analysis.

Independent Parameters

The database is constructed by varying the independent parameters and keeping the other three parameter types constant. A finite range and number of values for each independent variable is selected and EM simulations are required at each of these evaluation points. All EM simulation points the model requires are checked to ensure they have been previously simulated. Any required simulations that are missing from the database are automatically completed and stored. After the EM simulations are complete, an interpolation of the data are performed to predict the performance of the discontinuity at the specified evaluation point.

From the previous discussion, the data storage requirements suffer the problem of dimensionality. For every additional independent parameter added, the number of EM simulation points increases by a factor equal to the number of points simulated in that dimension. For example, if there are three independent parameters: W1, W2, and W3, with eight values for each parameter, and 10 frequency values, the database would require 8*8*8*10 = 5120 EM simulations to be completely filled. As a result, the number of independent variables is kept to a minimum.

Scalable Parameters

Scalable parameters can be changed without needing to create a new database. This type of parameter scales the EM structure so that the solution for the model can be related to a scaled frequency solution. As a result, when a data table has been created with the scaled solution already contained, the model does not have to create those particular solutions. For the MTEEX model, simulations at one particular height of a substrate can lead to electrical parameters of a second height by scaling the frequency by the ratio of the two H dimensions. Scalable parameters do not add a dimension to the database and can be evaluated for any substrate height. Therefore, when observing the EM simulations, the simulation frequencies may be quite different than the project simulation frequency range due to frequency scaling.

Fixed Parameters

Fixed parameters are fixed for the given database. These parameters do not vary for a typical design so variation of these parameters is not evaluated in one given database. Importantly, you should not attempt to tune on a fixed parameter as this results in the generation of a new database for each value of the fixed parameter evaluated.

Statistical Parameters

Statistical parameters are not varied during the design process but do vary due to manufacturing tolerances. To perform statistical analysis, these parameters require a small variance to quantify the final design or to perform design centering. During acquisition of the database this type of parameter is considered fixed. A first-order model of the dependence upon this parameter is used to vary the solution. This method is valid for small variations from a nominal value. For microstrip models, the only statistical parameter is the relative dielectric constant of the substrate. Small variations in Er <10% recommended and <20% required is more than adequate to perform statistical evaluation. Another possible use involves the frequency- or temperature-dependent nature of the dielectric constant. This model supports these types of variations in Er up to the previously mentioned range.

Index

Symbols

A

B

C

D

H

I

J

K

L

M

N

O

P

T

V

W

X

Y

Z