Pulsonix Spice Simulator

Evaluation Guide

Copyright Notice

Pulsonix, a division of WestDev Ltd.

Printed in the UK.   Issue date: 25/03/2014 iss 5

Pulsonix
20 Miller Court
Severn Drive
Tewkesbury
Tewkesbury Business Park
Glos, GL20 8DN
United Kingdom

Phone   +44 (0)1684 296 551
Fax   +44 (0)1684 296 515
Email   info@pulsonix.com

Contents

Welcome To Pulsonix Spice

Getting Started

This is a brief guided tour of the evaluation version of the Pulsonix Spice A/D mixed mode simulator. The evaluation version is a fully functional version of the program but is limited in its available design capacity. You will find the capacity to try some simple circuits of your own but nothing too elaborate.

The evaluation guide is intended to take you on a short journey through the main features of Pulsonix Spice. It is not intended as a training course but is provided to give you a 'taste' of some of the powerful features in the product. We recommend that you print out this guide before starting to run the software. You will then be able to compare the printed instructions with the information on the screen without having to keep switching tasks all the time.

As this evaluation version does not allow you to save your work as you proceed, we have provided some example files. These can be loaded at any point and the simulation run.

Pulsonix Spice requires Pulsonix Schematics in order to generate a netlist, which is then simulated. This guide will start with the Schematic design and move quickly onto the simulator. If you wish to know more about Pulsonix, please read the demo guide for the Pulsonix product, which is also supplied on the CD.

There are no time limits on this evaluation and it can be removed at any time using the Windows **Add / Remove Programs** mechanism in the **Control Panel**.

Hardware Requirements

As with Pulsonix, Pulsonix Spice doesn't require special or 'high powered' hardware to run on. A Pentium 2.0Ghz processor machine (or faster) is suitable provided it has enough spare disk space to load Pulsonix and Pulsonix Spice (about 190Mb is required), and enough RAM to ensure that the program isn't being paged in and out of memory, so at least 1-2Gb minimum for the evaluation is fine. For the fully functional Pulsonix product you would probably need more than 2Gb of RAM to get the best performance out of the application. The evaluation runs on Windows XP to Windows 8 operating systems but cannot run on older Windows, i.e. Windows 2000 or earlier. A CD-ROM drive and mouse are also required to load and operate the demo.

Software Requirements

You can also run the demonstration of Pulsonix Spice with your existing 'purchased' copy of Pulsonix. The Pulsonix Spice product will still run in demo mode with its demo limits but it does mean you can save different circuits. You should temporarily swap you license for the demo.lic supplied on the product CD. This can be done using the License Manager.

Simulation for the Novice

When measuring a real circuit, you would probably connect up a power source - bench power supply perhaps - maybe also some signal source, then switch it on and take measurements using - probably - an oscilloscope. You can also make adjustments to the circuit and see the effects straight away.

For simulation, you have a choice of analysis modes and not all of them have an exact "real life" equivalent. The analysis mode that most closely resembles the method of bench testing a circuit described above is "Transient Analysis". This performs an analysis over a specified (by you) time period and saves all the results - i.e. all the voltages and currents in the circuit - to your hard disk. The difference between real life testing and simulation is that the simulation is not running all the time. If you want to see the effects of changing a component value, you have to change it then re-run the simulation.

In order to solve the circuit, the simulator has to calculate the voltage at every node at every time point. Disk space is cheap and plentiful so Pulsonix Spice saves all these values as well as the device currents. Not all simulators do this, some require you to state in advance what you want saved.

After the run is complete, you can then randomly probe the circuit to look at any voltage, current or device power over the analysis time period. You can also place fixed probes on the circuit before running the analysis, which will cause the waveform at that point of the circuit to be automatically be displayed while the simulation is running or optionally after its completion.

Some of the other analysis modes are: AC analysis which performs a frequency sweep, DC sweep which ramps a voltage or current source and noise analysis which calculates total noise at a specified point and which components are responsible for that noise.

What Should You Expect From A Simulator ?

Simulation should be seen as a complement to the traditional bread-boarding phase of circuit design, rather than a complete replacement. With simulation, you can gain an insight into the workings of your circuit that are difficult or impossible to do in "real life". Here are some examples of things you can do with a simulator that can't be done on the bench:

- Make "non-invasive" measurements. For example, current is always difficult to measure without adding components to the circuit. Even magnetic current probes can alter a circuit's behaviour. Measuring current with a simulator, however, is straightforward and doesn't affect its characteristics.

- Remove parasitic elements. For example, you can remove the capacitance that cause "Miller Effect" in an amplifier to see if that is the cause of a misbehaving circuit.

- Operate components outside their safe operating area. Sometimes it is useful to see what will happen when you run - say - a power supply at beyond its ratings. In real life you risk damaging some components in the circuit, but a simulator does not suffer such limitations.

It is often not viable to simulate a complete circuit either because the simulation would take too long or because models are not available for all the components you are using. But, it is nearly always possible to simulate a part of your circuit and learn a great deal about the design by doing so.

You can also use a simulator to try out ideas at a system level. Pulsonix Spice has a number of components that can be defined by an equation either to describe a non-linear characteristic or to implement an arbitrary frequency response. You can use these blocks to implement functionality without having to think about the underlying implementation.

What's In The Product ?

Pulsonix Spice has three main elements:

1. The Schematic capture stage. This allows you to graphically enter the circuit using Parts, which have Spice parameters already attached to them. You can also apply stimulus Symbols, Probes and functional symbols to complete the circuit ready for simulation.
    Because of the unique way in which Pulsonix Spice has been developed, the circuit entered is suitable for simulation <u>and</u> can be run directly into Pulsonix PCB design editor with no further interaction of the Parts!

2. The simulator. This is a 'number cruncher' that takes a netlist as it's input and generates a - sometimes very large - data file containing the results. The netlist contains a description of your circuit along with commands to instruct the simulator exactly what analysis to perform. The data file contains all the voltages and currents in your circuit for later plotting or analysis.
    The simulator is based on SPICE3 developed by the University of California at Berkeley and XSPICE developed by the Georgia Technical Research Institute. Although based on these programs, Pulsonix Spice has received extensive further development to improve its overall performance.

3. The waveform viewer. This part of the program plots the results of the simulation using the data file created by the simulator. It can do this while the simulator is running or on demand after the simulation has completed. As well as plotting facilities, the waveform viewer can also perform measurements on the results such as RMS, peak-peak, frequency etc.

To run Pulsonix Spice you will also require the Pulsonix Schematics package. This is used to hold the Spice model properties, create the circuit and to generate the netlist, which is simulated.

Installation Of The Evaluation Software

Standard Installation Folder

As we explained at the beginning of this guide, and during the installation, the evaluation version of Pulsonix Spice will install by default into C:\Program Files\Pulsonix\Pulsonix-Spice. Other folders are created within this, for example, Spice example files under the main Pulsonix Examples folder and libraries, '**Examples\Spice**' and '**Pulsonix\Master Libraries**' respectively.

To install the evaluation version of Pulsonix Spice, first ensure that all other applications have been closed, and relevant data saved. Insert the CD-ROM into the CD drive and wait for a moment. The default setup of Windows enables the *autorun* facility to be automatically used. If your system has the autorun facility disabled or it simply hasn't worked, select **Run** from the **Start** menu, and type D:\setup.exe (replacing the D: with the actual CD drive letter, if necessary, then <Enter>.

The Pulsonix **Setup** screen will now run. You will be lead through the installation process using a wizard and instructions. When requested, use the demo license to try the product.

If at any time you click on **Cancel**, the window that appears gives you the chance to **Continue** or **Exit** the installation.

When the CD contents have been copied, messages will briefly display indicating that registry entries have been created, and icons copied. The selected folder will appear briefly as the icons are added, and a 'Setup Complete' window appears. You can choose whether to immediately launch the program, but in any case, the final step is to click on **Finish** to complete the installation.

Running The Evaluation Software

Starting Pulsonix

To start this evaluation you first need to run up Pulsonix. Click on the **Start** button on the taskbar (at the bottom of your screen). Select **Programs**, **Pulsonix** and **Pulsonix**.

The application will now start running, Pulsonix is now ready to use, as is the Pulsonix Spice simulator (you must also have installed the Pulsonix Spice product during the main installation).

It is assumed that you understand how to run Pulsonix, but where specific actions are required to run the simulator, these will be documented as specific instructions.

Load An Example Design

There are sample designs supplied in the Pulsonix folder under Examples, **…\Pulsonix\Examples\Spice** These are sorted in circuits of similar functionality. There is also a **Tutorial** folder (under **…\Examples\Spice),** which contains circuits used in this guide. This is the full set of circuits as supplied with the 'real' product but not all of them will run with the Pulsonix Spice demonstration mode software, some will be too large in size. All of the circuits supplied in the **Tutorials** folder will run in the demo mode.

Tutorial 1. A simple ready to run circuit

This tutorial demonstrates a basic simulation on a ready to run to circuit. All you need to do is load the circuit and then run it. We will then make a few changes to the circuit and make some extra plots.

▶ **Simulating Tutorial 1**

1. Click on the **File** menu, click on **Open**. Open Tutorial 1, usually located **C:\Users\Public\Documents\Pulsonix\Examples\Spice\Tutorials**

2. A Schematic window will open with the following circuit:
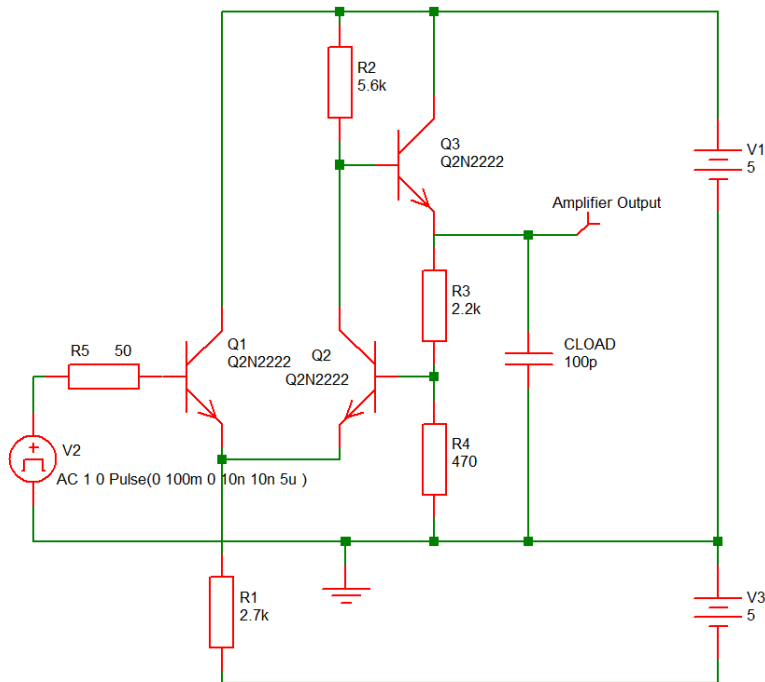
3. This is a simple feedback amplifier designed to amplify a 100mV pulse up to around 500mV. The basic requirement of the design is that the pulse shape should be preserved, DC precision is important but is not critical. The above is the first attempt at a design and has not yet been optimised.

4. This example circuit has been setup to be 'ready to run'.

5. To start the simulation, from the **Simulation** menu, select **Simulate Design** or press the **F9** key on the keyboard. The simulation will not take long, on most machines less than a second.

6. A window will open showing a graph of the output voltage:

Amplifier Output (tran1)
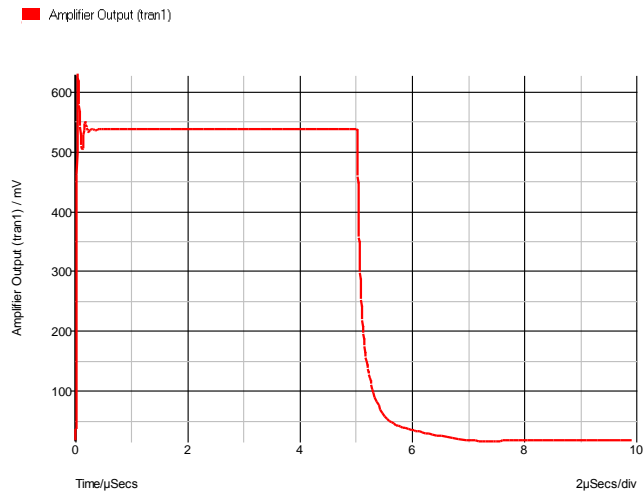
7. As you can see, the amplifier doesn't work all that well. There are two problems.
   a) There is substantial ringing on the rising edge, probably caused by the capacitative load.
   b) The falling edge is somewhat sluggish.

   The sluggish falling edge is caused by the absence of any standing current in the output emitter follower, Q3. To rectify this, we will place a resistor from the emitter to the -5V rail. The resulting schematic is shown below (R6 has been added and is circled):
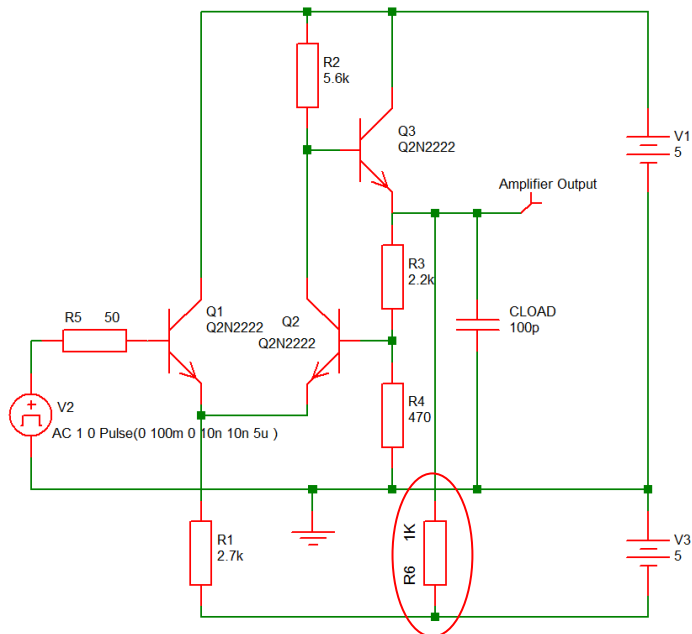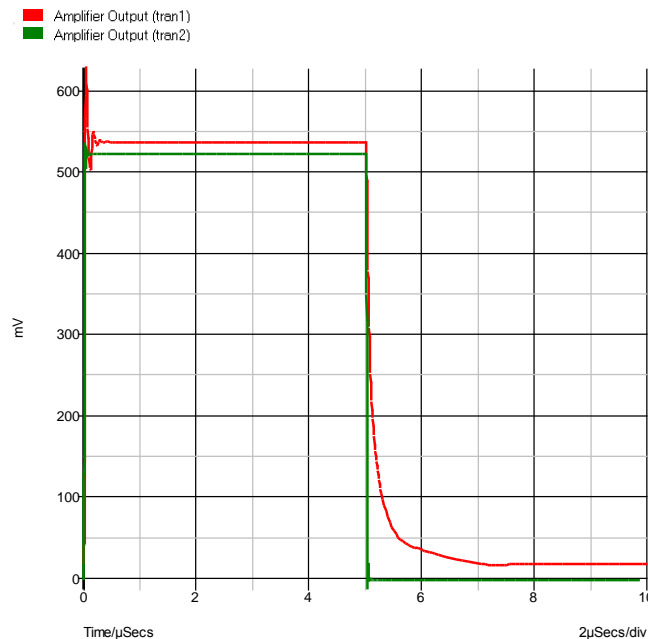

▶ **To make the modifications**

1. Ensure that the Parts popup toolbar is displayed, if not right click on one of the toolbars and select **Parts** from the shortcut menu. From this menu you can also switch on the Simulation toolbar if not already displayed.

2. From the **Parts** toolbar, select the **Passives** button to popup the **Passives** toolbar. Select the **Resistor (Box Shape)** button, or if you prefer, the **Resistor (Z Shape)** button.

3. A resistor symbol will appear. Move the resistor into the location shown in the diagram above. Left click to drop it in place.

4. Another resistor symbol will appear. Cancel this by pressing the Escape (ESC) key on the keyboard.

5. The resistor will be automatically named to R6, the next available number in the'R' sequence.

6. Now connect the resistor.

7. Drag a connection from each of the terminals on the resistor to connect it into the circuit. To be sure you have made the connection, look out for the finish cursor (f) when the connection is over a legal connection point.

8. Alternatively, use the Insert Connection option from the menu or toolbar to make the connections.

9. Re run the simulation by pressing **F9**. The graph will now be updated to look like this:

The graph will now be updated to:

The green trace(the cleaner of the two) is the output after the modification. As you can see, the problem with the trailing edge has been fixed and the ringing is much improved.

Now let's have a look at the ringing in more detail. To do this, we need to zoom in the graph by adjusting the limits of the axes. There are two ways of doing this. The quickest is to simply drag the mouse over the region of interest. The other method is to manually enter the limits using the **edit axis** dialog.

▶ **To zoom with the mouse**

4. Make sure that the graph window is selected by clicking in its title bar.

5. Place the cursor at the top left of the region of interest i.e. to the left of the y-axis and above the top of the red curve.

6. Press the left mouse key and while holding it down, drag the mouse to the bottom right of the area you wish to zoom in. You should see a rectangle appear as you drag the mouse.

Release the mouse key, you should see the following:
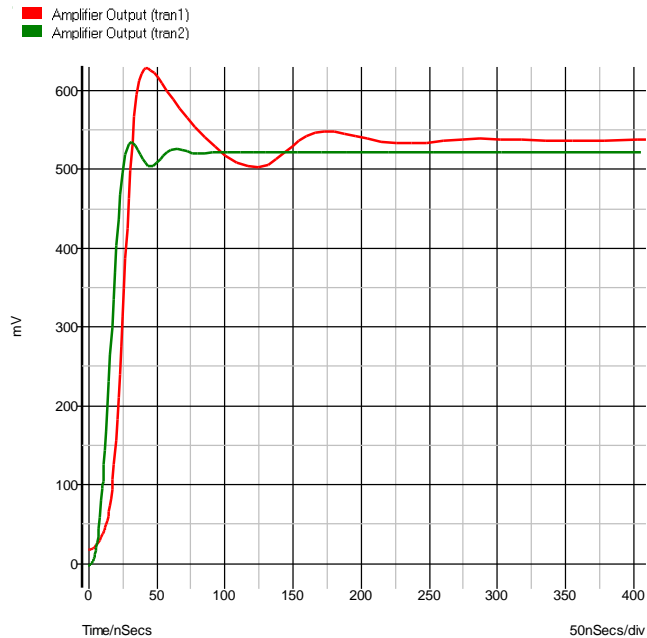
If you don't get it quite right, press the **Undo** (Zoom) button on the toolbar to return to the previous view.

You can improve the ringing by adding a small phase lead in the feed back loop. This can be done by connecting a small capacitor between the emitter of Q3 and the base of Q2. There isn't room to add this tidily at present, so first, we will move a few components to make some space.

▶ **To make the space required**

1. On the Schematic design, drag a selection window over the region shown in dotted lines on the diagram below:
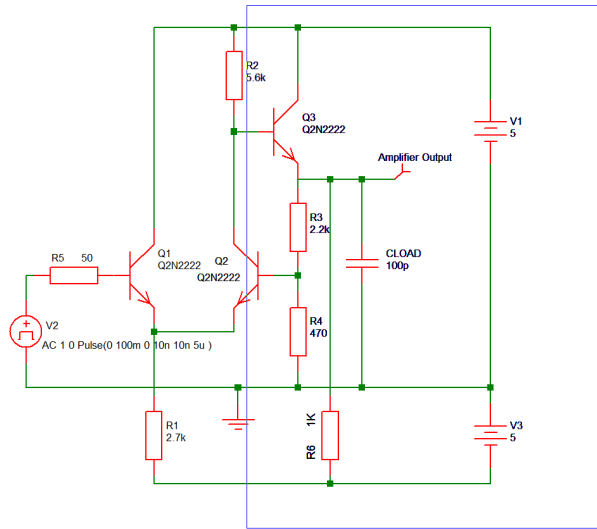
2. As you drag the mouse, a rectangle will appear.

3. Release the mouse. The area enclosed will change to the selected colour. If extra connection segments are selected, they can be removed from the selection using Ctrl-Click, or by ticking the 'Select if Completely Framed' option in the Options > Interaction dialog, available from the Tools menu or by pressing the 'O' key.
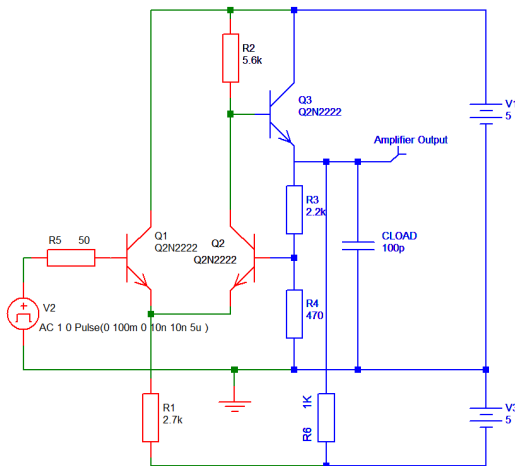

4. The selected connections and Components can be moved by dragging the selection. Move it to the right to give you space on the right of Q2.

5. Move it by a number of grid spaces and release it.

6. Click in 'free' space to deselect the group. You will now see:
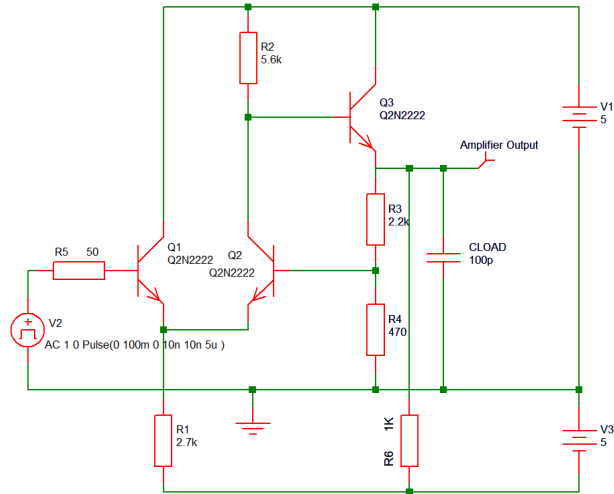
7.   Connect in the capacitor C1 as shown below using a similar procedure as for the resistor R6. Select the Capacitor from the **Parts** popup toolbar.
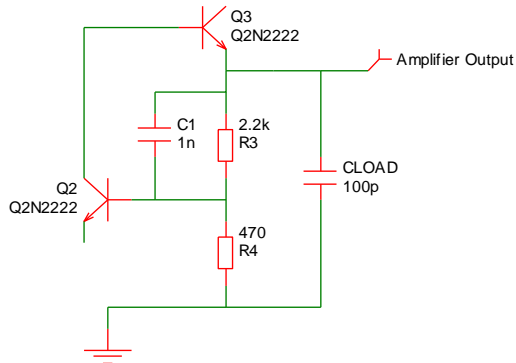

8.   1nF is obviously far too high a value so we will try 2.2pF.

▶ To change the Capacitor value

1.   To change the component's value first select C1. To do this, left click on it.

2.   Either press **F7** or right-click on the Component C1 and from the shortcut menu, select **Edit Spice Value/Model...**. You will see the following dialog appear:

3. You can type the new value in directly in the **Value** box or you can select a value using the mouse alone with the up and down arrow buttons.

4. Change the value to 2.2pf and press OK.

5. Now re-run the simulation by pressing F9.

6. This is the result you should see:
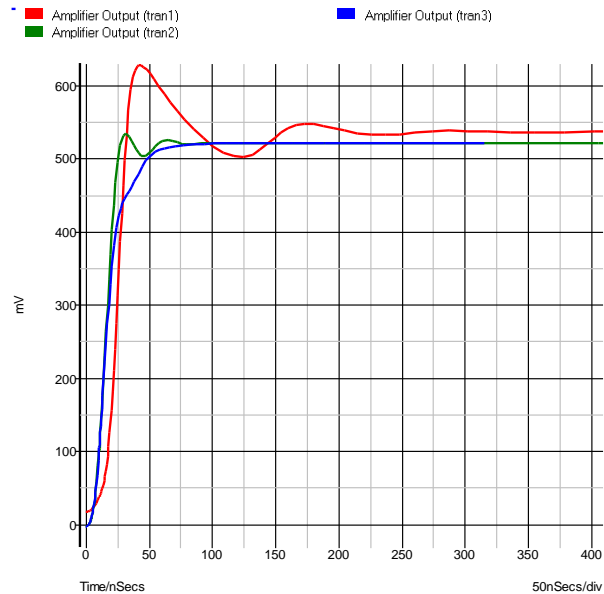
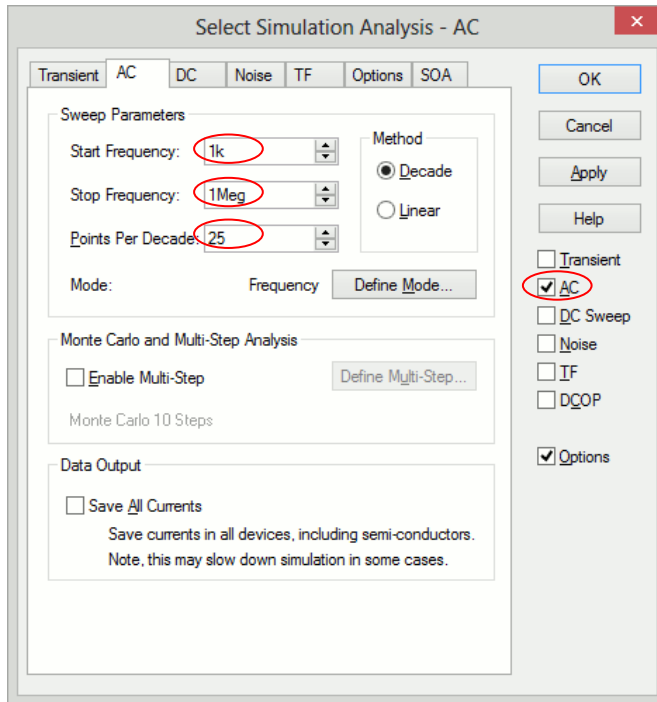The blue curve is the latest result. This is now a big improvement on our first attempt.

We will now round off tutorial 1 by introducing AC analysis.

AC analysis performs a frequency sweep over a specified frequency range.

Introducing AC Analysis

▶ **To make the modifications**

1. In the Schematic window, select the **Simulation** menu and select **Simulation Parameters**.

2. Select the AC tab, ensure that the **AC** option is ticked on the right hand side tick options (this allows for the simulation to run for ticked option), **Start Frequency** is set to **1K**, the **Stop Frequency** is set to **1Meg** and that **Points Per Decade** is set to **25**. The **Decade** radio button should be selected.

3. Click **OK** to close the dialog and accept the values.

Now run the simulation again by pressing **F9**. The following graph appears.
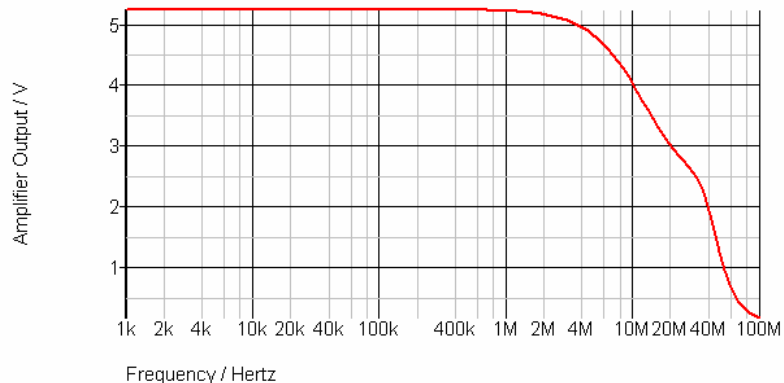
Tutorial 2. A Simple SMPS Circuit

In this tutorial we will simulate a simple SMPS switching stage to demonstrate some of the more advanced graphing and waveform analysis facilities available with Pulsonix Spice.
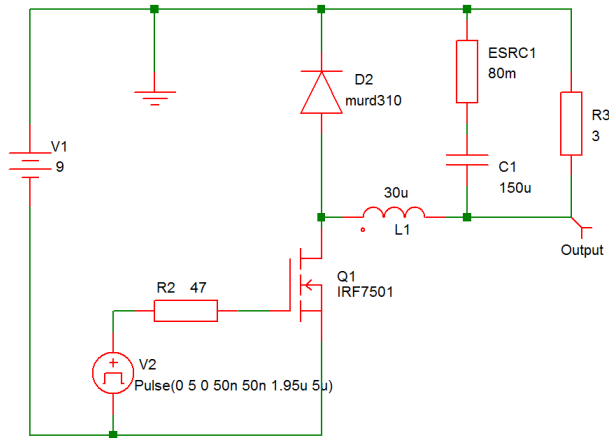
You can either load this circuit from **…Pulsonix\Examples\Spice\Tutorials\Tutorial2** or alternatively you can enter it from scratch. The latter approach is a useful exercise in using the schematic editor to add SPICE components.

▶ **To simulate**

1. If creating the design from scratch, place the Components and connections as shown above.

2. The "Output" probe (on the right side of the design) can be selected from either the **Parts** toolbar and **Probes** popup toolbar using the **Voltage Probe** button, or from the **Simulation** menu and **Insert Fixed Probe, Voltage Probe.**
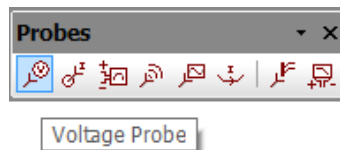
3. The voltage probe uses a single pin symbol, so you can release it on top of the connection required and it will automatically 'weld' itself to this net making itself fully connected.

4. After placing the output probe, select it, then press **F7** to edit its label. Enter the name **Output** to identify the curve when simulated. Enter the text in the **Curved Label** box. All the other options may be left at their defaults.
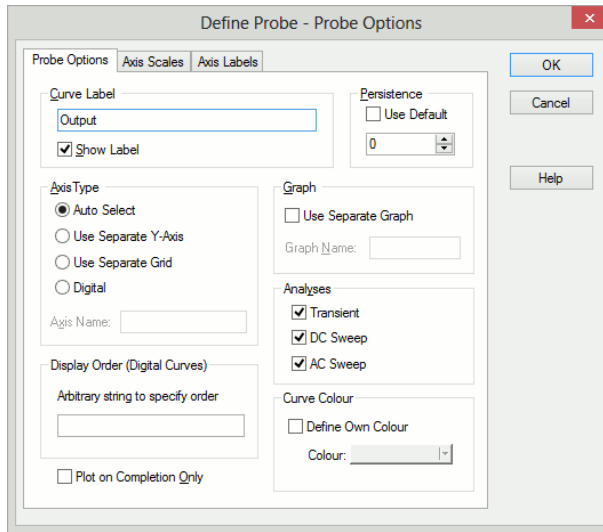
5.   For the pulse source **V2**, you should use a **Universal Source**.

6.   From the **Sources** popup toolbar, select the **Universal Source** button.

7.   When added, select V2 and press **F7**. Edit the settings as shown below:
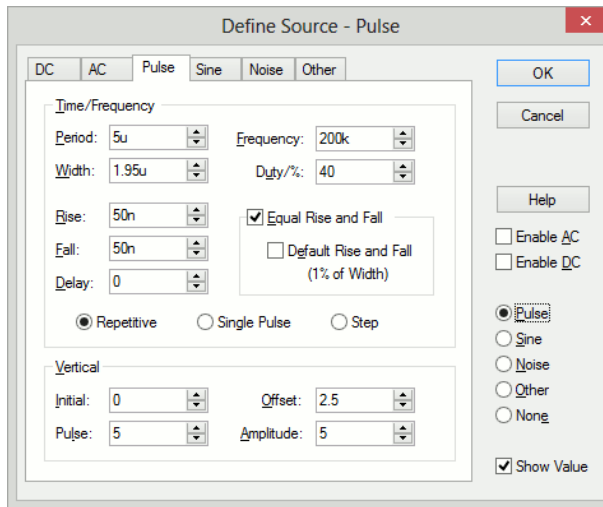
8.   This sets up a 200kHz 5V pulse source with 40% duty cycle and 50nS rise time.

9.   Set up the simulation parameters by selecting the **Simulation** menu and **Simulation Parameters** option.

10.   Check the **Transient** check box on the right hand side of the dialog. In Transient Parameters, set the **Stop Time** to **1m**.
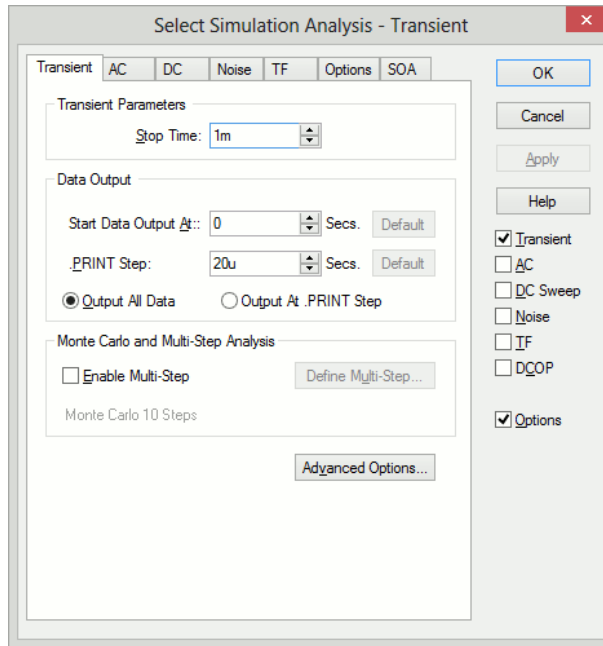
11. In the **Integration Method** section, check the radio button **Gear**. This improves the simulation results for this type of circuit. You will still get sensible results without checking this option, they will just be a little better with it.
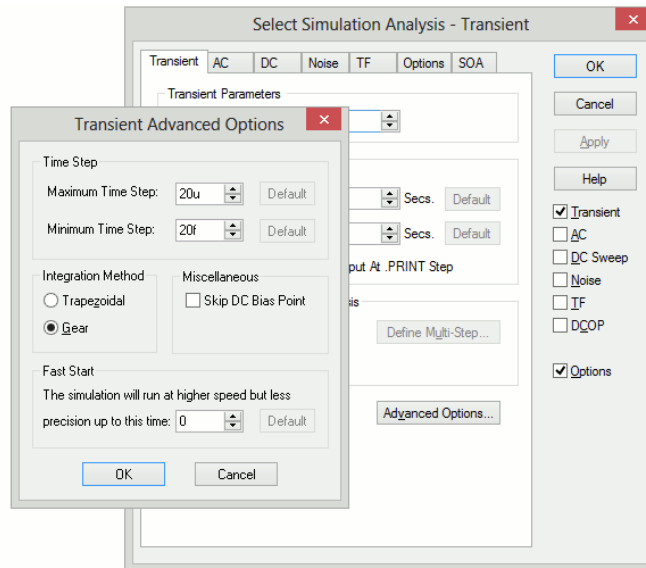
12. If you have any Pulsonix Spice graph windows open, you should now close them. Once you have loaded or entered the circuit, press **Simulate Design** from the **Simulation** menu to start the simulation. This is the graph you will see
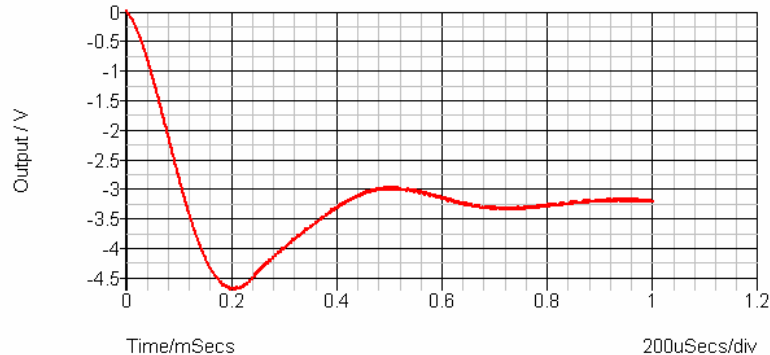
The circuit is the switching stage of a simple step-down (buck) regulator designed to provide a 3.3 V supply from a 9V battery. The circuit has been stripped of everything except bare essentials in order to investigate power dissipation and current flow. Currently, it is a little over simplified as the inductor is ideal. More of this later. We will now make a few measurements.

Lets have a look at the inductor current. We could have viewed the inductor current by placing a current probe on the pin of the device before starting the simulation. Here, instead, we will plot the device current *after* the completion of the simulation.

▶ **To measure power dissipation**

1.   From the Schematic design, click on the **Simulation** menu and **Random Probe**. With the Random Probe cursor shown in design, right click the mouse.

2.   From the shortcut menu, select **User Defined Probe**.

3.   From the dialog, select the **Probe Type** to be **Power In Device** and the **Probe Function** to be **Actual**.
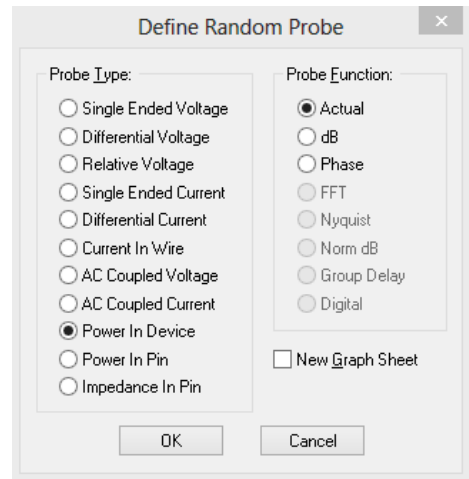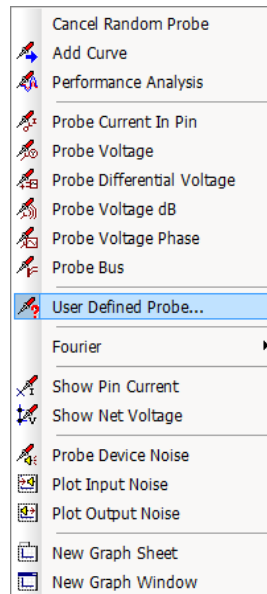
4.   Select the **New Graph Sheet** check box.

5.   Click **OK** to confirm the selection.

6.   Left click on Q1 to use the probe.

7.   This is what you should see:
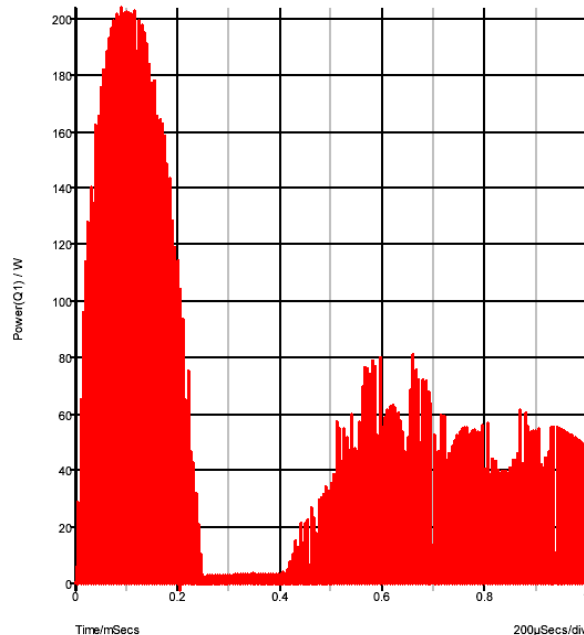
8.   This shows a peak power dissipation of 200W although you are probably more interested in the average power dissipation over a specified time.

▶ **To display the average power dissipation over the analysis period**

1.   In the graph window, use menu Measure|Mean.

2.   This should display a value of 517 mW. This is the average power over the whole analysis period of 1mS. You can also make this measurement over any period you select using the cursors.

3.   Zoom in the graph at a point around 100uS, i.e. where the power dissipation is at a peak.

4.   Switch on graph cursors using the Cursors|Toggle On/Off option on the right mouse menu. There are two cursors represented by cross-hairs. One uses a long dash and is referred to as the main or measurement cursor. When first switched on the reference cursor is positioned to the left of the graph and the main to the right.

5.   Position the cursors to span a complete switching cycle. There are various ways of moving the cursors. To start with the simplest is to drag the vertical hairline left to right. As you bring the mouse cursor close to the vertical line you will notice the cursor shape change. See "Graph Cursors" for other ways of moving cursors.

6.   Press F3 or select analysis menu (right click in legend panel) More Functions... From the tree list select **Measure|Transient|Mean|Cursor Span**:
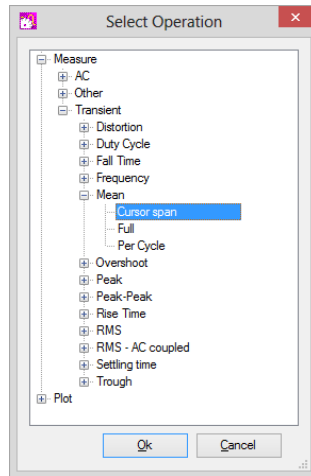
You should see a value of about 2.8W displayed. This is somewhat more than the 517mW average but is still well within the safe operating area of the device. However, as we noted earlier, the inductor is ideal and does not saturate. Lets have a look at the inductor current.

▶ **To use the Random Current Pin Probe**

1.   From within the Pulsonix Schematic window, select the **Simulation** menu.

2.   Select **Random Probe**. Right click, and from the shortcut menu select **New Graph Sheet**. A new simulation graph will appear.

3.   Right mouse click again to display the same shortcut menu, this time select **Probe Current In Pin**.

4.   Click on the left pin of the inductor L1 (marked pin 'P'). This is what you will see in the graph window:
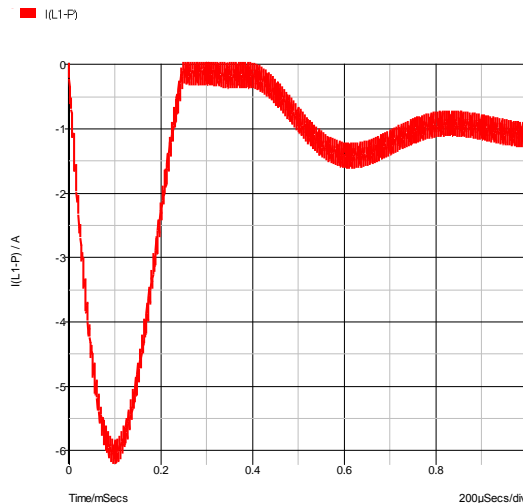
This shows that the operating current is less than 1.5A but peaks at over 6A. In practice you would want to use an inductor with a maximum current of around 2A in this application; an inductor with a 6A rating would not be cost-effective. We will now replace the component, with something closer to a real inductor.

▶ **To replace inductor L1**

1.   Press the **Escape** key to exit Random Probe mode.

2.   Left click to select the inductor component L1 in the schematic design.

3.   Click the **Del** key on the keyboard.

4.   The two connections which were previously attached to L1 will now be left 'hanging' in free space.

5.   If the **Part Browser** is not already running, select it from the **View|Dockable Windows** menu. Make sure that Spice category is displayed, if not use **Change Part Category** from the right hand menu and add Spice category to the list.

6.   From the Spice Category list, select **Passives**, then **Saturable Transformer**. Drag off the Part Browser view to add this device to the design.

7.   You will be presented with the Define Saturable Transformer/Inductor dialog:
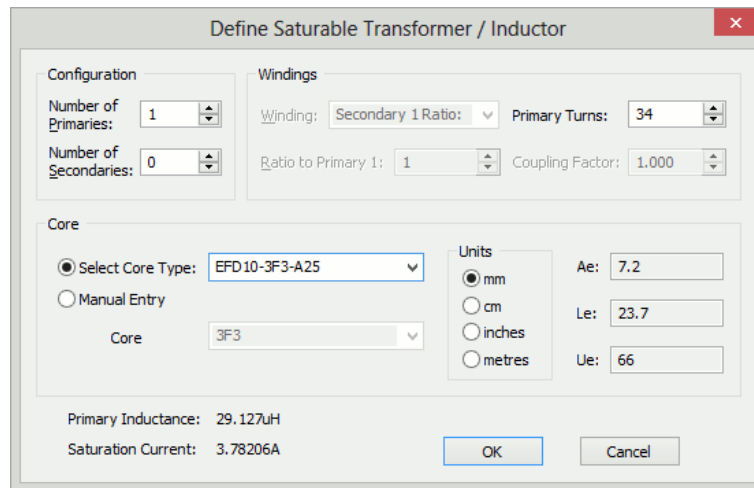
8.   Select number of Primaries to 1 and number of Secondaries to 0 for and inductor.

9.   Select **EFD10-3F3-A25** in the **Select Core Type:** drop down list. This is part number for a Philips ferrite core. Enter 34 in the **Primary Turns** edit box. **Units** should be mm. All other information will be automatically calculated.

10.   Drag it into the space previously occupied by L1, roate and mirror as necessary to make sure pin 1 is on the left.

11.   Press **Esc** to cancel the addition of a second copy of this component.

12.   If you placed the inductor in exactly the same location as L1 it will 'auto weld' attaching the connections back to the two terminals of the new component. You will now not see pin crosses in the design.

13.   Press **F9** to simulate again.

14.   Select the graph sheet that displayed the inductor current by clicking on its tab at the top of the graph window. Now select the simulation menu in the Pulsonix Schematic editor.

15.   Select Random Probe. From the shortcut menu select Probe Current In Pin.

16.   Click on the left hand inductor pin again. You will notice the peak current is now in excess of 45A. This is of course because the inductor is saturating. You can also measure the peak power over 1 cycle.

▶ **To measure the peak power**

1.   In the graph window, select the graph sheet tab with the power plot.

2.   In the schematics window use **Random Probe** and, from the shortcut menu, select **User Defined Probe**.

3.   From the dialog, select the Probe Type to be Power In Device, the Probe Function to be Actual and uncheck the New Graph Sheet box.

4.   Left click on the Q1 component.

5.   In the graph window, zoom out to see the full graph using the **Fit Window** button from the toolbar.

6.   Zoom in on the peak power.

7.   Position the cursors to span a full cycle. (The cursors are currently tracking the first power curve. This doesn't actually matter here as we are only interested in the x-axis values. If you want to make the cursors track the green curve, you can simply pick up the cursor at its intersection with the mouse and drag it to the other curve).

8.   We now have two curves on the graph so we must select which one with which we wish to make the measurement. To do this check the box.
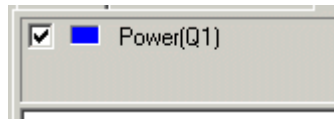
9.   As before, press F3 in the graph window, then select Measure|Transient|Mean|Cursor Span. A value of about 11-12W should be displayed, rather substantially higher than before.

Tutorial 3. Installing 3rd Party Models

In this tutorial, we will install a device model library. For this exercise, we have supplied a model library file - TUTORIAL3.MOD - with just two devices. These are:

> SXN1001 - an NPN bipolar transistor
>
> SXOA1000 - an opamp.

Both are totally fictitious.

You will find this file in the tutorial folder **Examples|Spice|Tutorials|Tutorial3.mod**. There are two aspects to installing a model. Pulsonix Spice needs to know where within your file system, the model is located. If the model is to be listed in the Pulsonix Schematics system using the **Insert Part By Model** option, then Pulsonix Spice also needs to know what part to use for it in the Schematic and what category it comes under.

▶ **To Insert Part By Model**

1. Open windows explorer or click on My Computer or open other file manager of your choice.

2. Locate TUTORIAL3.MOD in *EXAMPLES*\Tutorials (see "Examples and Tutorials – Where are They?" Pick the file up and drop into the Pulsonix Spice command shell. That is, drop it in the window where Pulsonix Spice messages are displayed. If you can't see the command shell because it is obscured, select any Pulsonix Spice window then press the space bar.

A message box will appear asking you to confirm you wish to install the file. Click OK.

The message "Making device catalog. This may take some time, please wait…." will be displayed.

At this stage, Pulsonix Spice knows where to find our fictitious models. You will find that it also knows about the NPN transistor as the following demonstrates:

- In the Schematics window, open a new Schematic design.

- From the Simulation menu use **Insert Part Using Model.**

- From the dialog select **NPN** from the left hand list.

- In the model list search for our new transistor, SXN1001. Using the filter string "SX*" will help.

- Select the device and press **OK** to place it on the Schematic sheet.

Without you having to tell it, Pulsonix Spice already knew that the SXN1001 is an NPN transistor. This is because it is a *primitive* device defined using a .MODEL control. Such devices are built in to the simulator and what the device is can be determined from the model contained in TUTORIAL3.MOD.

The other device in the model library is an op-amp and is defined as a *sub-circuit*. This is a module made up of other components, in this case BJT's, diodes, resistors and current sources. If you repeat the above procedure for the opamp device - SXOA1000 - by looking under the Op-amps category, you will find it is not there.

In order to arrange for the device to be listed in the **Insert Part Using Model**, you need to associate the model with a Pulsonix Part.

▶ **To associate Models with Parts**

1. In the simulator **command shell**, from the **File** menu, select **Model Library>** and **Associate Models and Parts...**

2. You should see the opamp - **SXOA1000** - listed in the box on the left under the ***unknown*** drop down list, click on it. This is what you will see:
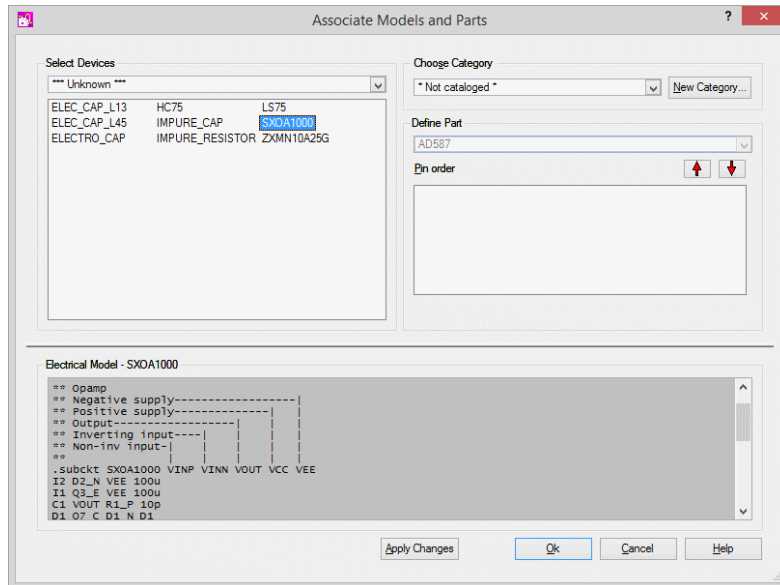
3. You must now supply a category and a Part:

4. From the list box immediately below the "Choose Category" label, select "Op-amps"

5. From the list box below the "Define Partl" label, select "Opamp" from the list of Parts with the same number of pins as your model.
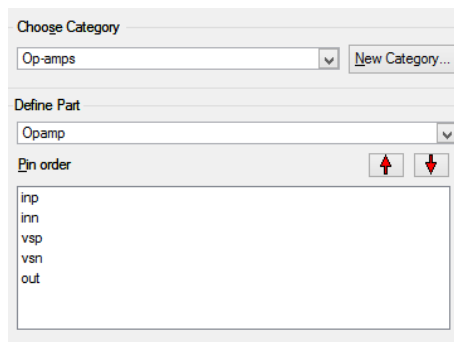
We have not quite finished yet. Our new op-amp has the wrong pin out for the Schematic Part. The pin order for the Part is shown in the third box down on the right and is as follows:

P	Function
i	*Non-inverting input*
i	*Inverting input*
v	*Positive supply*
v	*Negative supply*
o	*Output*

The text at the bottom of the dialog box shows the actual sub-circuit definition. Fortunately, it has been annotated with the function of each of the sub-circuit's terminals. (This is in fact usually the case with third-party models). As you can see, the output terminal is in the wrong place. We can change the pin order using the "Pin order" up and down buttons:

1. Select "out" in the pin order box.

2. Click twice on the up button so that it is positioned between "inn" and "vsp"

This is what you should now have:
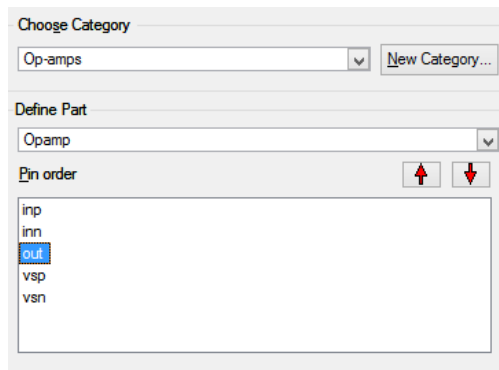
Now to finish:

1. Press **Apply Changes** button

2. Press **OK**.

You will now find our op-amp listed under the "Op-amps" category in the Schematic **Insert Part Using Model** dialog.

Notes: You will not always need to execute the above procedure to associate models and parts even for sub-circuit devices. Pulsonix Spice is supplied with a data base of over 30,000 devices that are already associated. These are devices for which SPICE models are known to be available from third party sources. This database is in the file ALL.CAT which you will find in the Pulsonix Spice installationsupport folder. The information you enter in the associate models dialog is stored in a file called USER.CAT which you will find in the application data directory folder.

Pulsonix Spice Help

If you need more detailed information on any of the functions offered by Pulsonix Spice, simply select the **Help** menu and **Contents**. You can either select a topic by double clicking it in the **Contents** tab, or search for a topic by typing the first few letters of a keyword in the **Index** tab.

From within the **Simulator**, the **Help** menu also contains online manuals for the **Simulator Reference Guide**, the **Device References** and the **Script Users Guide**. From within Pulsonix itself, using the **F1** key from any of the simulator dialogs will display the online help for that topic.

Explore for Yourself

Now that we have taken you through the design process from Schematic to simulation, it is time for you to carry on exploring the facilities of this program on your own. As save is inhibited in Pulsonix, you will not change permanently any of the sample files provided or your own designs. If you have used the Pulsonix Spice demo with a working copy of Pulsonix, you will be able to explore more and try different iterations of the design, but don't forget there are demo limits, which come into play.

Technical Support

If you have any problems installing or running this evaluation, you can contact our Technical Support department by telephone on +44 (0)1684 296 570 or by e-mail to support@pulsonix.com Alternatively, contact your local distributor or sales representative and request assistance.

Web Site Access

www.pulsonix.com

Our web site www.pulsonix.com keeps you fully up-to-date with current developments and new product information. The site is fully accessible and not restricted in any way, please feel free to visit and browse at any time.