PULSONIX

Pulsonix Design System

Step-by-step Guide To
OrCAD Design & Library Conversion

Copyrights

Copyright © WestDev Ltd. 2000-2014
Pulsonix is a Trademark of WestDev Ltd. All rights reserved. E&OE

Copyright in the whole and every part of this software and manual belongs to WestDev Ltd. and may not be used, sold, transferred, copied or reproduced in whole or in part in any manner or in any media to any person, without the prior written consent of WestDev Ltd. If you use this manual you do so at your own risk and on the understanding that neither WestDev Ltd. nor associated companies shall be liable for any loss or damage of any kind.

WestDev Ltd. does not warrant that the software package will function properly in every hardware software environment.

Although WestDev Ltd. has tested the software and reviewed the documentation, WestDev Ltd. makes no warranty or representation, either express or implied, with respect to this software or documentation, their quality, performance, merchantability, or fitness for a particular purpose. This software and documentation are licensed 'as is', and you the licensee, by making use thereof, are assuming the entire risk as to their quality and performance.

In no event will WestDev Ltd. be liable for direct, indirect, special, incidental, or consequential damage arising out of the use or inability to use the software or documentation, even if advised of the possibility of such damages.

WestDev Ltd. reserves the right to alter, modify, correct and upgrade our software programs and publications without notice and without incurring liability.

Microsoft, Windows, Windows NT and Intellimouse are either registered trademarks or trademarks of Microsoft Corporation.

OrCAD is a trademark of Cadence Design Systems Inc.
All other trademarks are acknowledged to their respective owners.

Pulsonix, a division of WestDev Ltd.
Printed in the UK. Issue date: 20/03/2014 iss 4

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
Web www.pulsonix.com

Contents

CONTENTS **3**

CHAPTER 1. EXPORTING ORCAD INFORMATION **4**

Overview 4

File Extensions Expected 4

CHAPTER 2. EXPORTING SCHEMATIC DESIGNS **5**

Exporting Schematic designs From OrCad into Pulsonix 5

Importing OrCAD Schematic designs into Pulsonix 8

CHAPTER 3. EXPORTING SCHEMATIC SYMBOLS **10**

Exporting Schematic Symbols From OrCad into Pulsonix 10

Importing OrCAD Schematic Symbols into the Pulsonix libraries 13

CHAPTER 4. EXPORTING PCB DESIGNS **15**

Exporting PCB Designs From OrCad Layout into Pulsonix 15

Importing OrCAD Layout PCB designs into Pulsonix 17

Importing OrCAD PCB Designer and Allegro PCB designs into Pulsonix 18

CHAPTER 5. EXPORTING PCB FOOTPRINTS **19**

Exporting PCB footprints From OrCad Layout into Pulsonix 19

Importing OrCAD PCB Footprints into Pulsonix 21

CHAPTER 6. IMPORTING PARTS **24**

Importing Parts From OrCad into Pulsonix 24

Chapter 1. Exporting OrCAD Information

Overview

The conversion between OrCAD Schematic Capture, OrCAD Layout and OrCAD Libraries to the Pulsonix equivalent is by use of ASCII format files. These files can be exported from OrCAD using the appropriate facility. OrCAD does the formatting of these ASCII files automatically which the Pulsonix product can then read in.

File Extensions Expected

From the OrCAD, the following file extensions for the ASCII files are expected:

- For OrCAD Schematic Capture – .edf
- For OrCAD Schematic Library Symbols – .edf
- Parts – in OrCAD Parts and Symbols are the same thing.
- For OrCAD PCB Layout – .min
- For OrCAD PCB Layout Footprint libraries – .min
- In OrCAD PCB Layout, footprints don't have any Part information.
- For OrCAD PCB Designer Professional - .brd

Pulsonix Parts libraries are created using the OrCAD Schematic Symbol files or Schematic Design files. Parts cannot be created from OrCAD footprint libraries. Then can however be created using OrCAD PCB Layout designs.

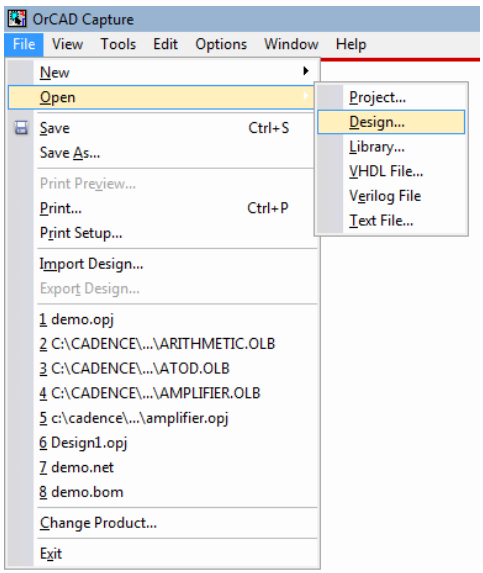
Note: You can also import OrCAD Schematic and PCB ASCII designs to create Pulsonix libraries if the Symbol or Footprint libraries are not available.

Chapter 2. Exporting Schematic Designs

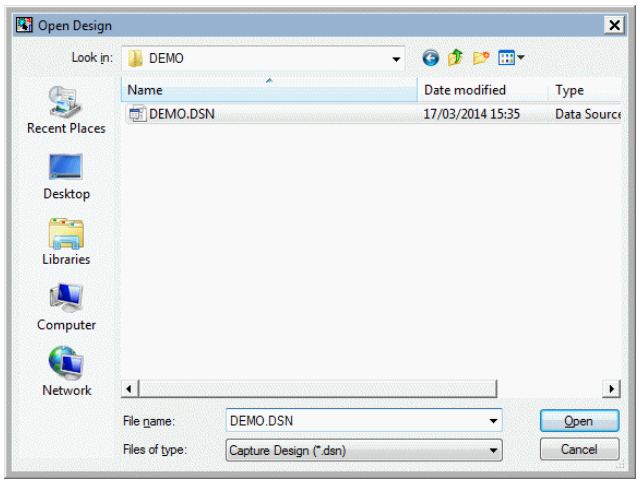
Exporting Schematic designs From OrCad into Pulsonix

▶ Exporting Schematic designs

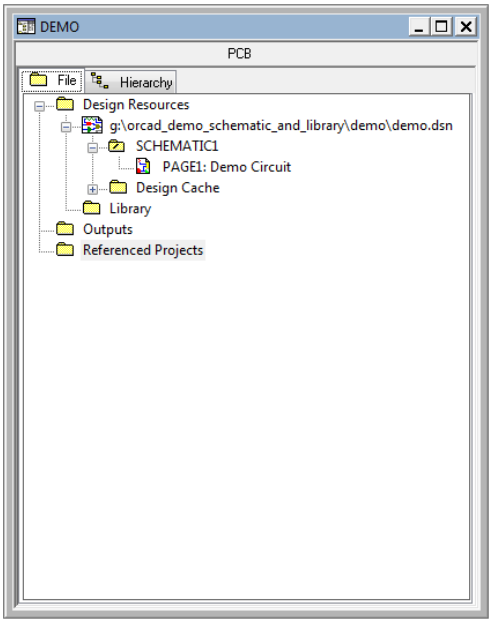
1. In OrCAD Capture, from the **File** menu select **Open>** and **Design**



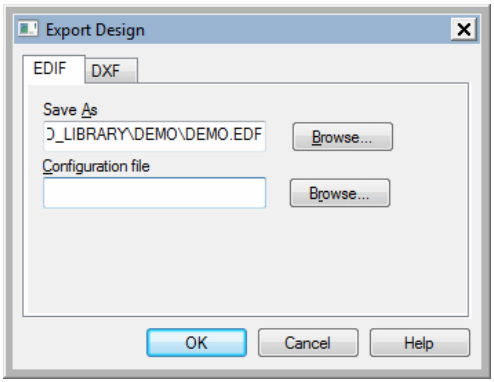
2. From the **Open** dialog, select the design required (the one shown with the .DSN file extension) and click **Open**.



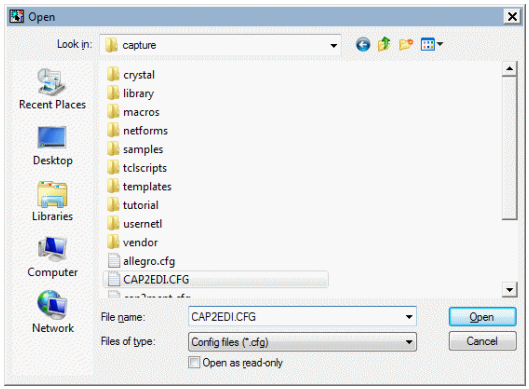
3. The opened design will appear in the browser.



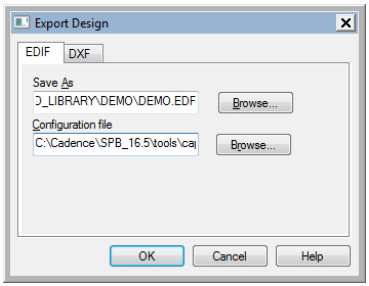
4. From the **File** menu, select **Export Design**
5. You should ensure that you are using the **EDIF** dialog.



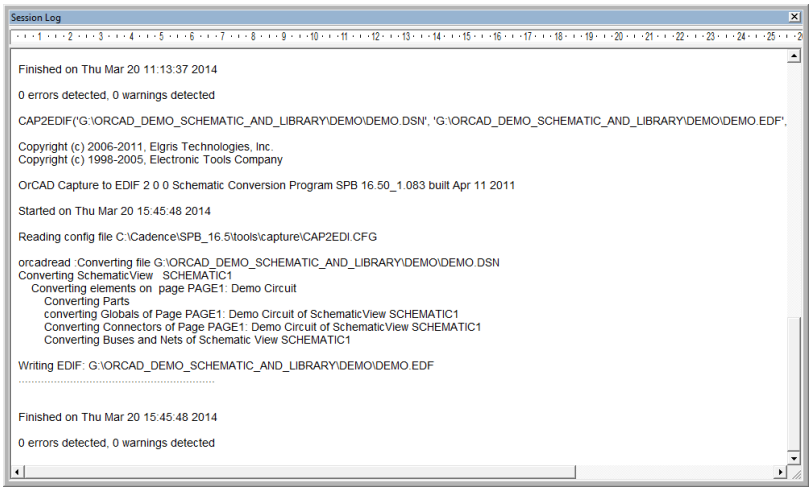
6. Two files are required. The **Save As** entry will automatically select the design file that you have open and will give it a file extension of .edf. You may need to scroll the box using the right arrow to see the full filename. You can type in any name of your choosing for the file to export.
7. A **Configuration file** is also required.
8. Click on the **Browse** button.



9. Navigate to the Capture installation folder which contains the supplied mapping file **CAP2EDI.CFG**
10. Click **Open** to select it.
11. The completed **Export Design** dialog will now look like this below.



12. Click **OK** to start the export.
13. One completion, the **Session Log** window will report the successful export of your design.

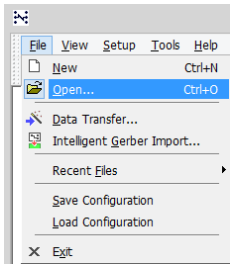


14. This completes the export of the Schematic Design file.

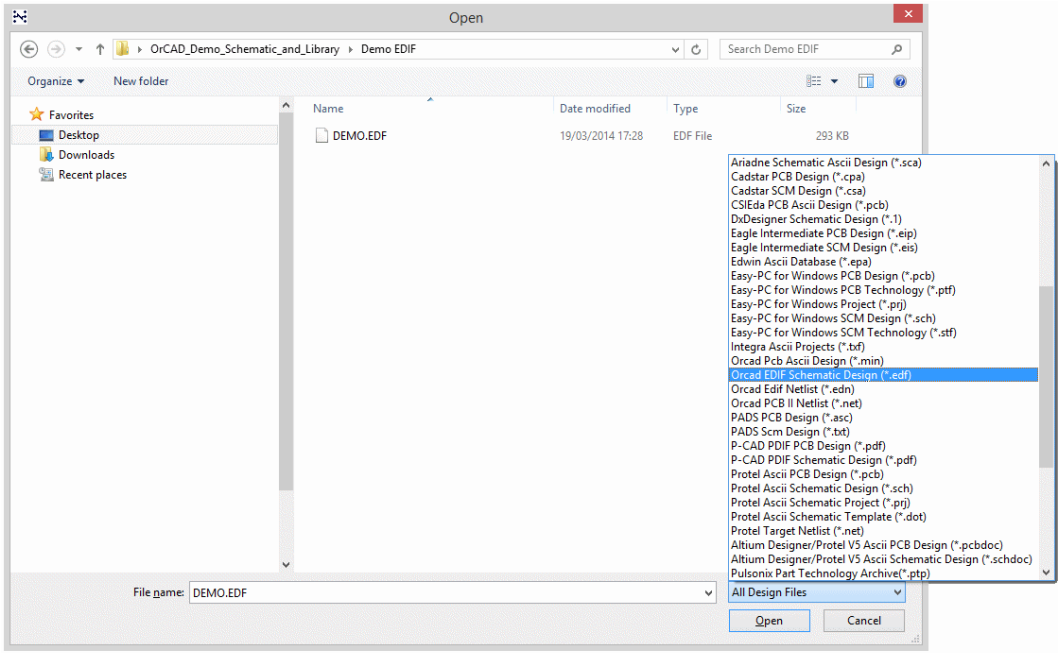
Importing OrCAD Schematic designs into Pulsonix

▶ To import the Schematic file into Pulsonix

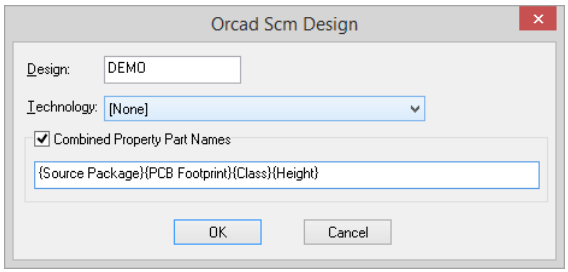
1. In Pulsonix, from the **File** menu select **Open**.



2. From the Open dialog browse to the file that you have just created from the procedure previously.
3. To help you refine the selection, drop down the **Files of type** list and select **ORCAD EDIF Schematic Design (*.edf)**
4. Select your design file and click **Open**.



5. The import **OrCAD SCM Design** dialog opens.



6. If you hadn't selected the File of type list to be the OrCAD filter previously, Pulsonix will still automatically detect the file format and import it in the correct manner.

7. The target **Design** name is presented. You can edit this to the required name.

8. For **Technology** you don't need to select a file, so select **[None]**. However, if you wish to apply a new set of colours or styles to the incoming design, select a Technology file from the drop down list.

9. For annotated OrCAD Schematics, there is the ability to have part names made up of its properties. Check the **Combined Property Part Names** dialog and enter the property name for which you want the part name to be made up of. The individual properties should be enclosed in curly brackets and in the order that you want. If the combined name is greater than 31 characters it will be truncated.

10. A progress message will appear followed by the design that will automatically open in Pulsonix.

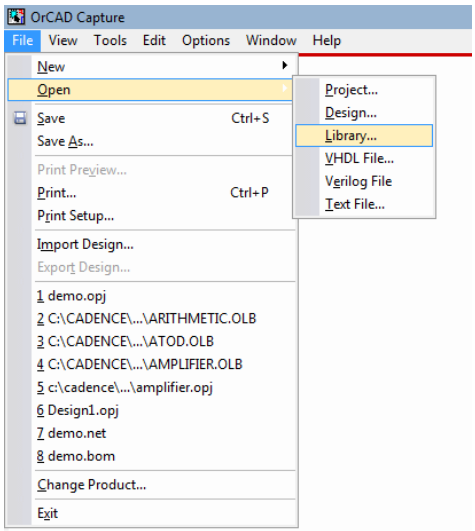
11. If there are any errors the design will not open. If errors can be corrected in OrCAD then do so and re-export the design again. If the errors cannot be resolved, contact our sales office or your local representative who will assist you.

Chapter 3. Exporting Schematic Symbols

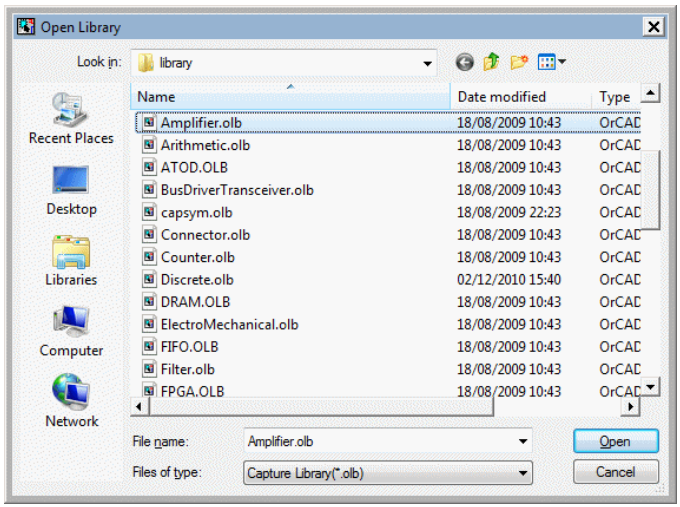
Exporting Schematic Symbols From OrCAD into Pulsonix

▶ Exporting Schematic Symbols

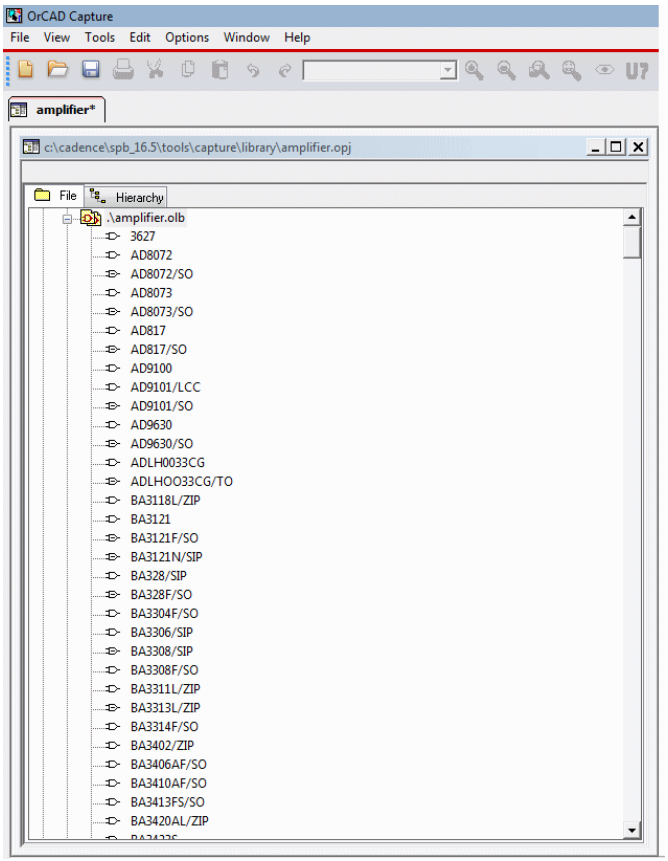
1. In OrCad Capture, from the **File** menu select **Open**> and **Library**



2. From the **Open** dialog, select the library required (with the .olb file extension) and click **Open**.

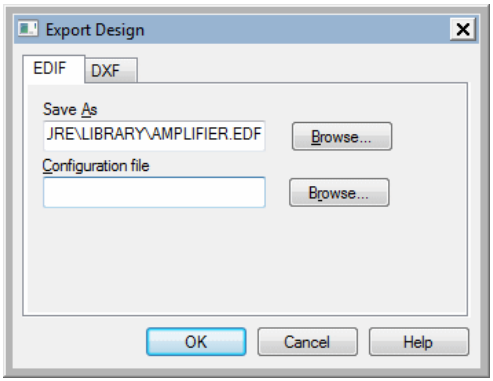


3. The opened library will appear in the browser.



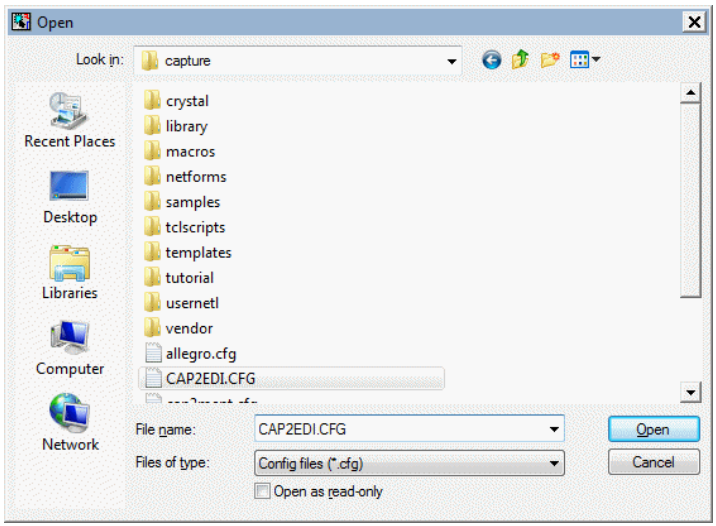
4. From the **File** menu, select **Export Design**

5. You should ensure that you are using the **EDIF** dialog.

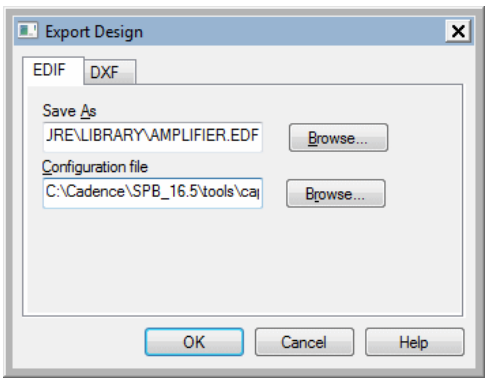


6. Two files are required. The **Save As** entry will automatically select the library file that you have open and will give it a file extension of .edf. You may need to scroll the box using the right arrow to see the full filename. You can type in any name of your choosing for the file to export.

7. A **Configuration file** is also required.
8. Click on the **Browse** button.



9. Navigate to the Capture installation folder which contains the supplied mapping file **CAP2EDI.CFG**
10. Click **Open** to select it.
11. The completed **Export Design** dialog will now look like this below.

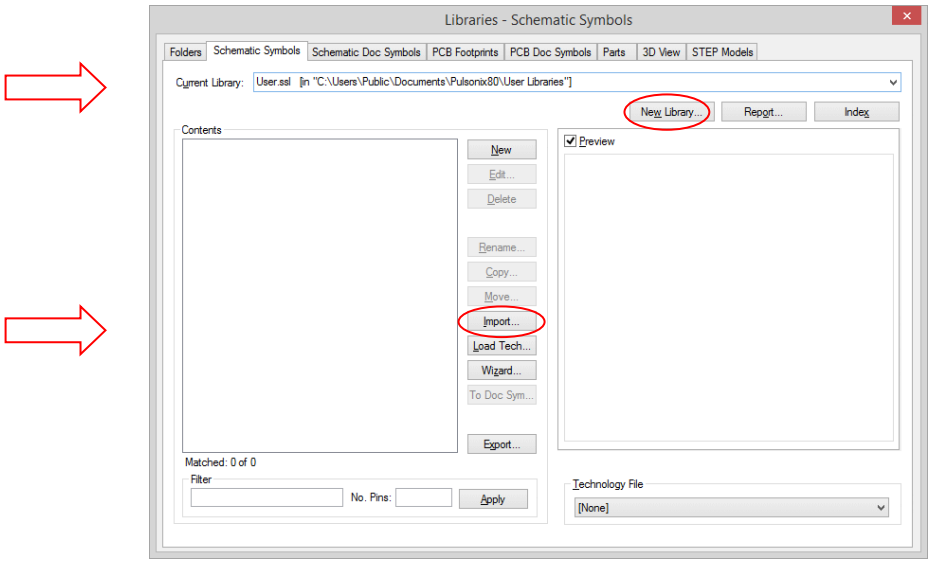


12. Click **OK** to start the export.
13. One completion, the **Session Log** window will report the successful export of your symbol library.
14. This completes the export of the Schematic Symbol file.
15. You must open and convert each of the symbol libraries that you require to use in Pulsonix.

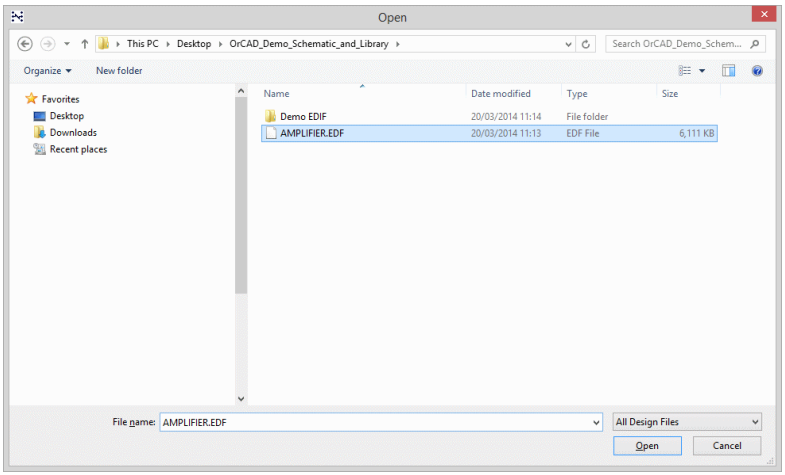
Importing OrCAD Schematic Symbols into the Pulsonix libraries

▶ To import the Schematic file into Pulsonix

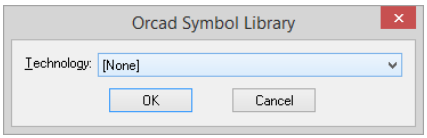
1. In Pulsonix, from the **Setup** menu select **Libraries**.
2. The Library Manager is displayed.
3. Select the **Schematic Symbols** tab.



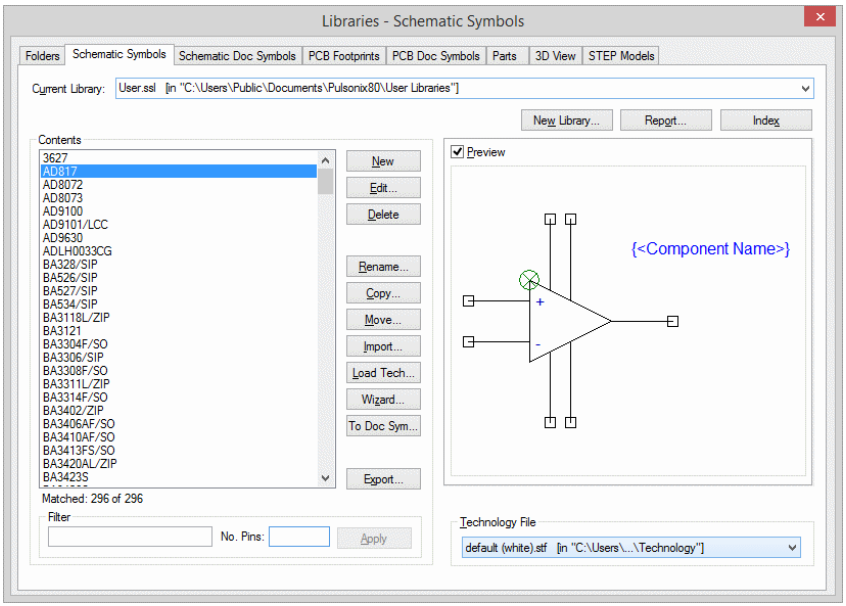
4. From the current library drop down list, select the **User.ssl** library. You can also create your own library if you like (Click New Library Button). The **Contents** list will be empty at this point.
5. Click the **Import** button.



6. Select the file for import. This will be the **.EDF** file that you created in the previous procedure.
7. There is no need to change the **File of type** unless you require a finer filtering of the list of files for selection.
8. Pulsonix will automatically detect the file type being imported.



9. You can enter a technology file if required. If you require a white background then select the Pulsonix **Default [White].stf** technology file.
10. Click **OK** to import the file.
11. When the import has completed successfully the **Library Manager** will show the Schematic Symbols dialog again.



12. From this dialog you can verify that all the symbols have been imported by looking in the **Contents** list and by the number that show as **Matched** after it.
13. To Preview any of the symbols, click the symbol required and it will appear in the **Preview** window. You must have the Preview check box selected to see the symbol.
14. By selecting a **Technology File** in from the drop down list you can view the symbols using the white background. This file will also be used when the symbol is subsequently edited as well.

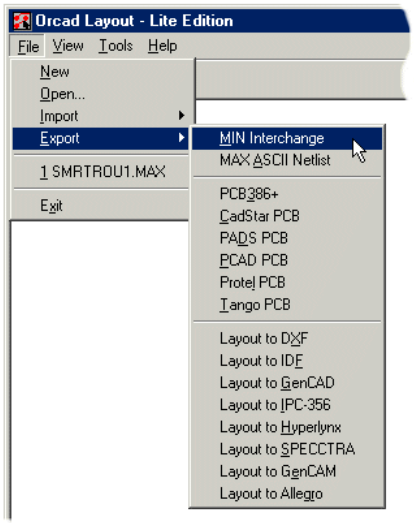
Note: For multiple file import, you can also use the Data Transfer Wizard.

Chapter 4. Exporting PCB Designs

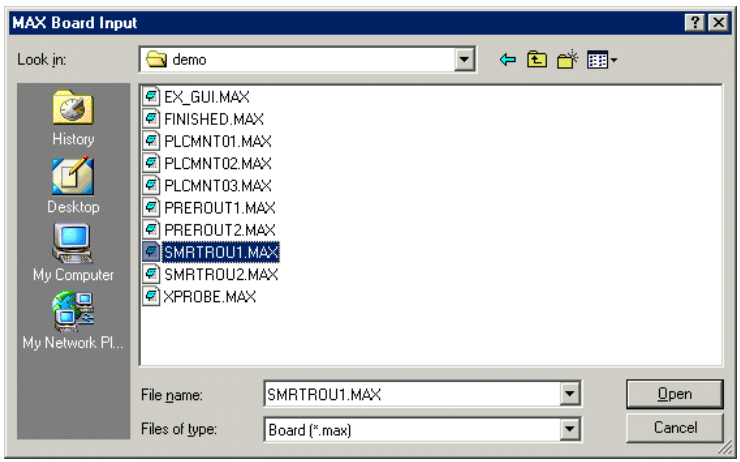
Exporting PCB Designs From OrCad Layout into Pulsonix

▶ Exporting PCB Designs

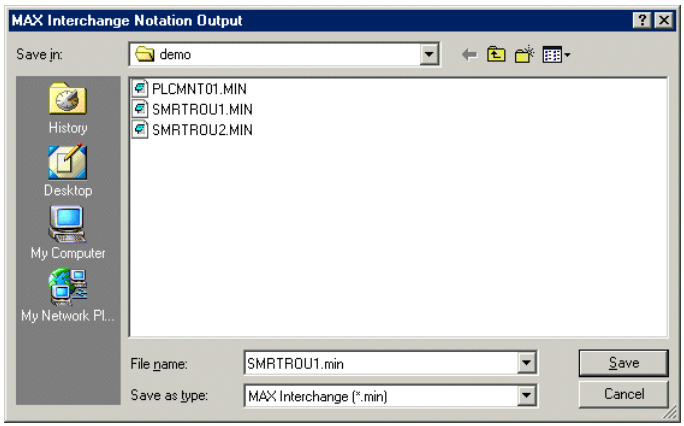
1. In OrCad Layout, from the **File** menu select **Export>** and **MIN Interchange**



2. From the **MAX Board Input** dialog, select the design required (with the .MAX file extension) and click **Open**.

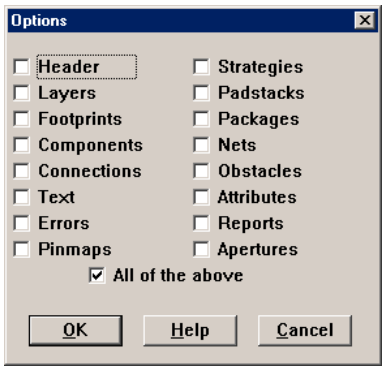


3. You are now required to select the output filename.



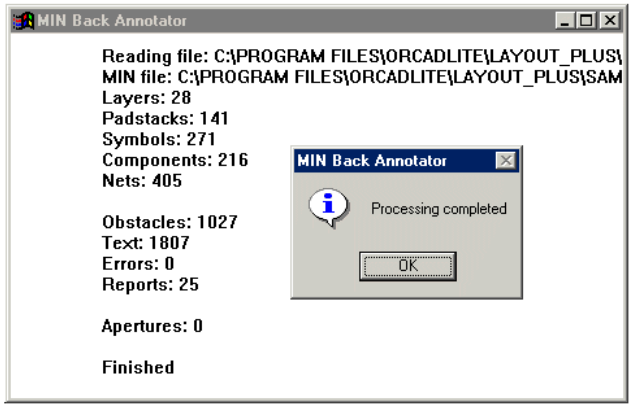
4. This will have the file extension **.min**

5. You are presented with the **Options** dialog. Ensure that the box **All of the above** is checked on to select the categories available.



6. Click **OK** to start the conversion.

7. Once completed, a report is displayed. Click **OK** to exit.

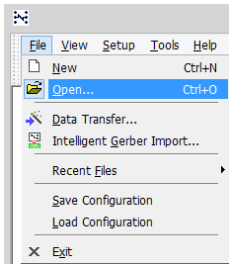


Importing OrCAD Layout PCB designs into Pulsonix

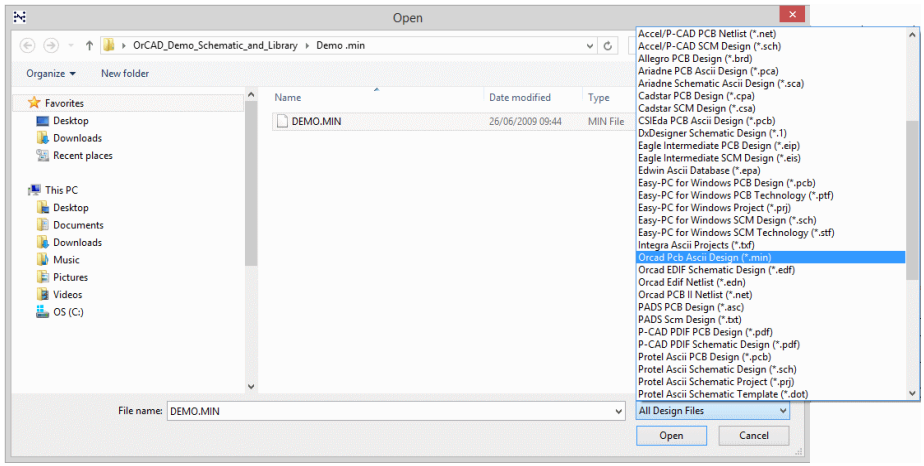
If you wish to import designs from **OrCAD PCB Designer**, please refer to the following page.

▶ To import the PCB file into Pulsonix

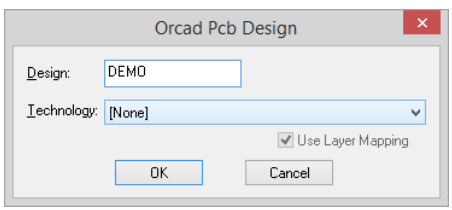
1. In Pulsonix, from the **File** menu select **Open**.



2. From the Open dialog, browse to the file that you have just created from the procedure previously.



3. To help you refine the selection, you can drop down the **Files of type** list and select **ORCAD PCB Design (*.min)**
4. Click **Open**.
5. The **OrCAD PCB Design** import dialog is displayed.
6. The target **Design** name is presented. You can edit this to the required name.



7. For **Technology** you don't need to select a file, so select **[None]**. However, if you wish to apply a new set of colours or styles to the incoming design, select a Technology file from the drop down list.
8. A progress message will appear followed by the design that will automatically open in Pulsonix.
9. If there are any errors the design will not open. If errors can be corrected in OrCAD then do so and re-export the design again. If the errors cannot be resolved, contact our sales office or your local representative who will assist you.

Importing OrCAD PCB Designer designs into Pulsonix

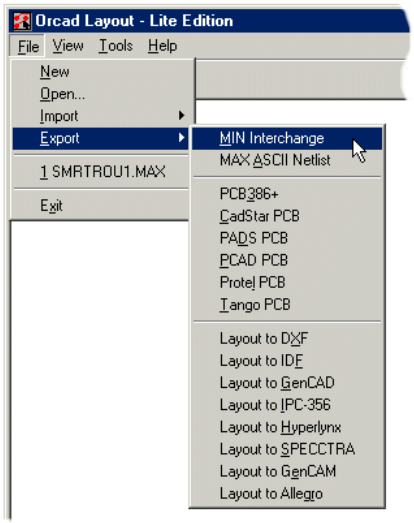
1. In Pulsonix, from the **File** menu select **Open**. (Pulsonix will automatically recognise the OrCAD PCB Designer .brd files when they are opened).
2. Pulsonix will detect that you have not done this before, and will ask you to browse to the extracta.exe. This is likely to be in the tools\pcb\bin sub folder of your OrCAD installation.

Chapter 5. Exporting PCB Footprints

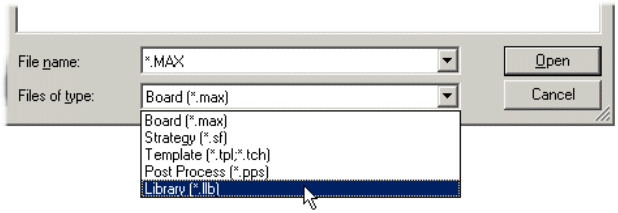
Exporting PCB footprints From OrCad Layout into Pulsonix

▶ Exporting PCB Footprints

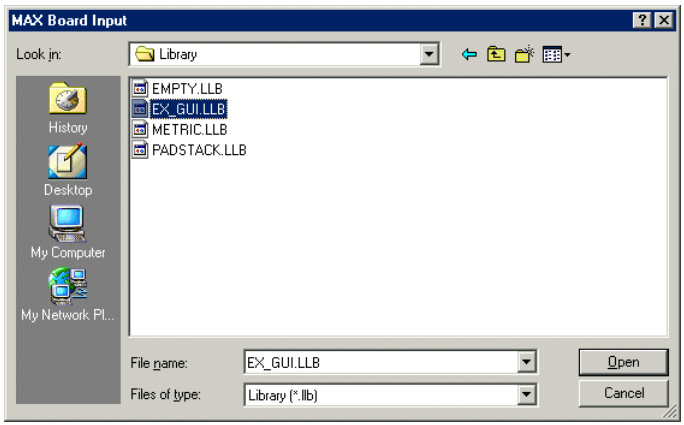
1. In OrCad Layout, from the **File** menu select **Export>** and **MIN Interchange**



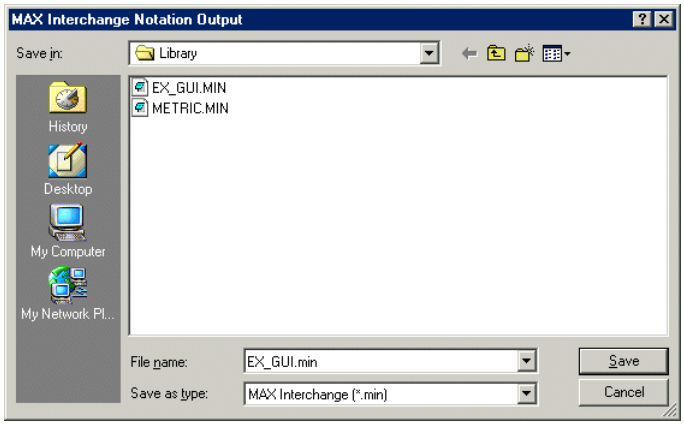
2. The **MAX Board Input** dialog is displayed.
3. Using the **File of type** drop down list, select **Library [*.llb]**



4. From the dialog, browse to find the required design and click **Open**.

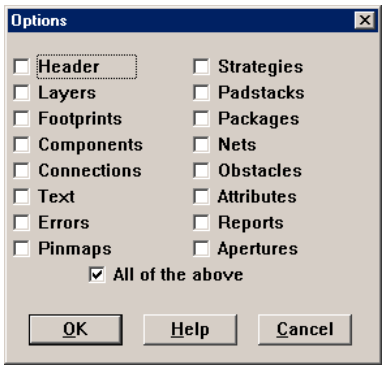


5. You are now required to select the output filename.



6. This will have the file extension **.min**

7. You are presented with the **Options** dialog. Ensure that the box **All of the above** is checked on to select the categories available.

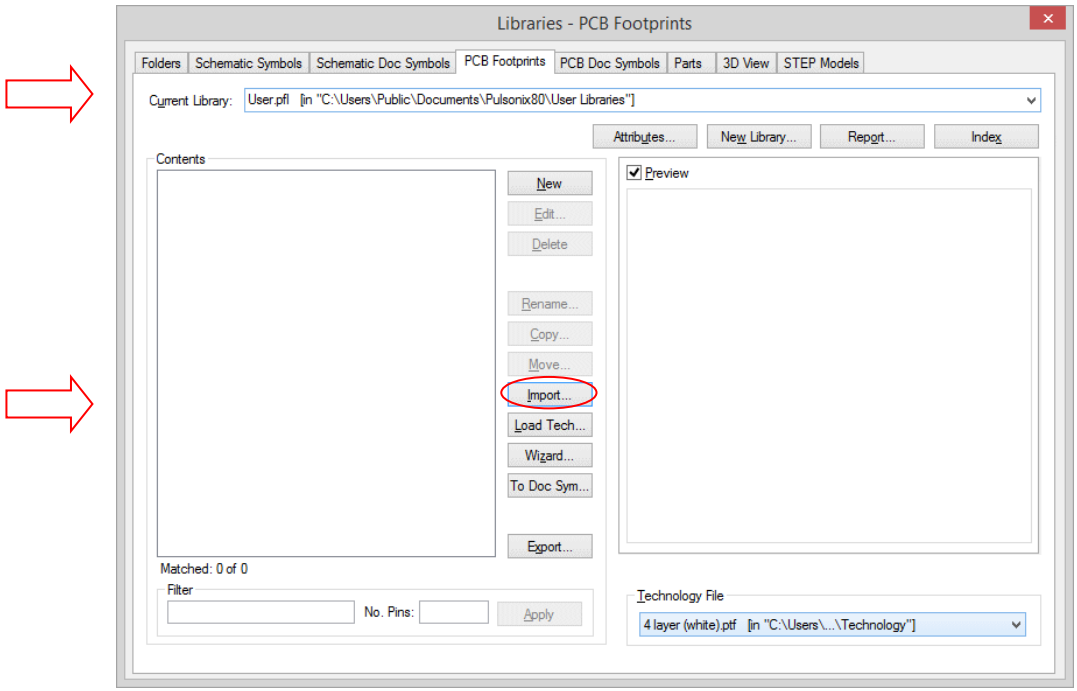


8. Click **OK** to start the conversion.
9. Once completed, a report is displayed. Click **OK** to exit.

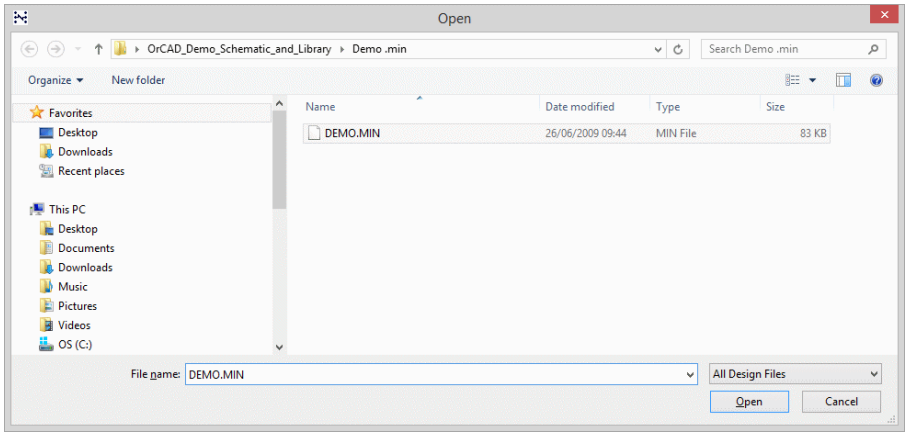
Importing OrCAD Layout PCB Footprints into Pulsonix

▶ To import the PCB footprint file into Pulsonix

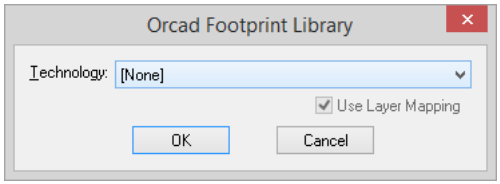
1. In Pulsonix, from the **Setup** menu select **Libraries**.
2. The Library Manager is displayed.
3. Select the **PCB Footprints** tab.



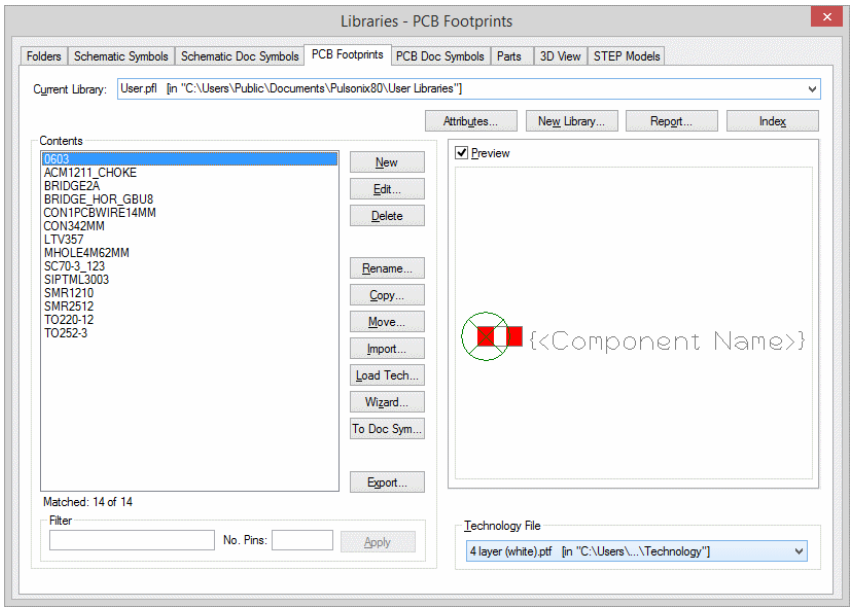
4. From the current library drop down list, select the **User.pfl** library. You can also create your own library if you like (Click the New Library Button). The **Contents** list will be empty at this point.
5. Click the **Import** button.



6. Select the file for import. This will be the **.MIN** file that you created in the previous procedure.
7. There is no need to change the **File of type** unless you require a finer filtering of the list of files for selection.
8. Pulsonix will automatically detect the file type being imported.



9. You can enter a technology file if required. For PCB Footprints, the import file contains everything it needs so leave the selection set to **[None]**.
10. Click **OK** to import the file.
11. When the import has completed successfully the **Library Manager** will show the PCB Footprints dialog again.



12. From this dialog you can verify that all the symbols have been imported by looking in the **Contents** list and by the number that show as **Matched** after it.

13. To Preview any of the footprints, click the symbol required and it will appear in the Preview window. You must have the Preview check box selected to see the symbol.

14. By selecting a Technology file in from the drop down list you can view the footprints using the white background. This file will also be used when the footprint is subsequently edited as well.

Note: For multiple file import, you can also use the Data Transfer Wizard.

Chapter 6. Importing Parts

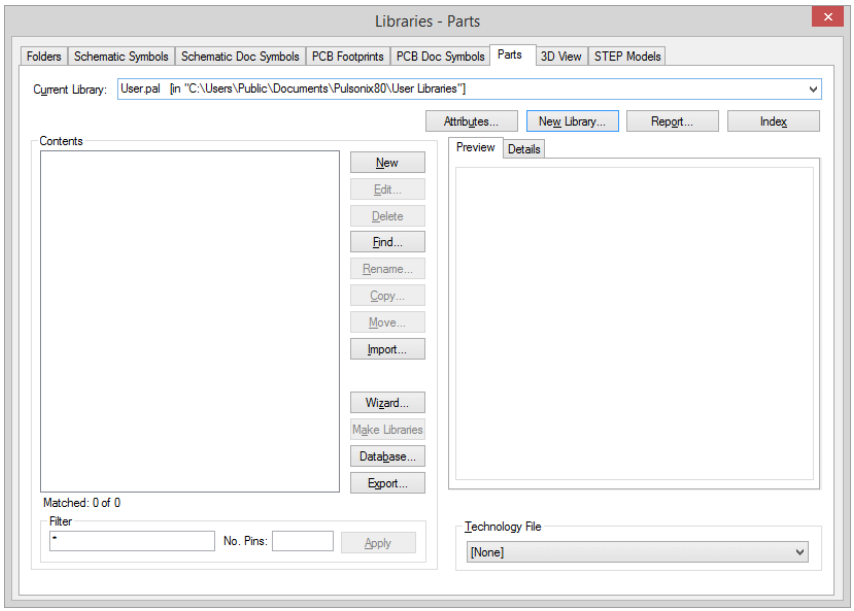
Importing Parts From OrCAD Layout into Pulsonix

In order to be able to use the Schematic Symbols and PCB Footprints imported from OrCAD in Pulsonix, you will also need to create a Part entry for them.

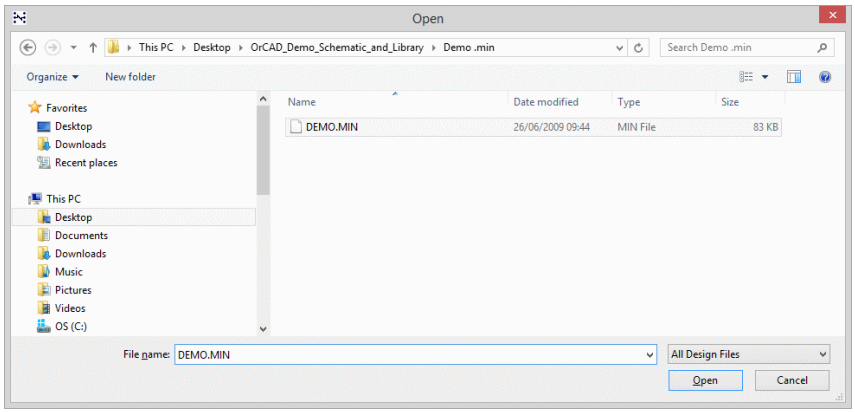
Part information in OrCAD is only contained within the Schematic Symbols (or Schematic designs), so you will need to import the Schematic Symbols to create these entries.

▶ To import the Parts files into Pulsonix

1. From the **Setup** menu select **Libraries**.



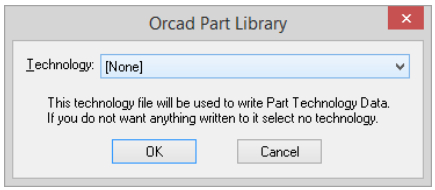
2. The **Library Manager** is displayed.
3. Select the **Parts** tab.
4. From the current library drop down list, select the **User.pal** library. You can also create your own library if you like. The **Contents** list will be empty at this point.
5. Click the **Import** button.



6. Select the file for import. This will be the **.EDF** file that you created in the previous procedure (Exporting Schematic Symbols).

7. There is no need to change the **File of type** unless you require a finer filtering of the list of files for selection.

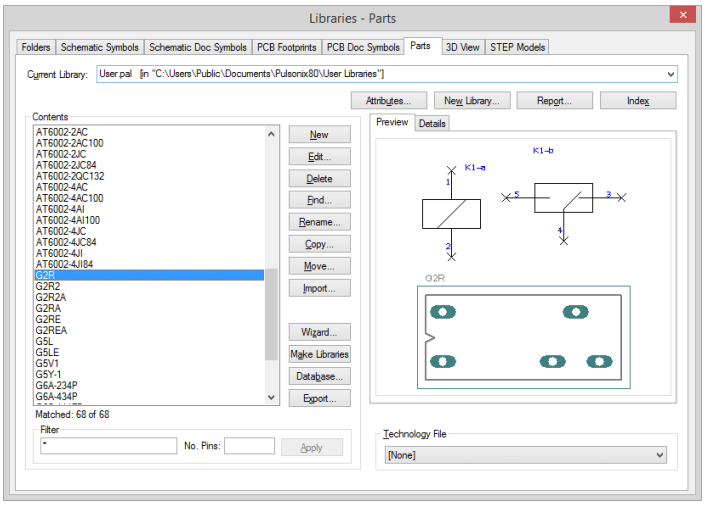
8. Pulsonix will automatically detect the file type being imported.



9. You can enter a technology file if required. The **Technology** is used to preload attribute names and predefined net names. It is not required in this instance.

10. Click **OK** to import the file.

11. When the import has completed successfully, the Library Manager will show the Parts dialog again.



12. From this dialog you can verify that all the Parts have been imported by looking in the **Contents** list and by the number that show as **Matched** after it.
13. The **Details** window will show the **PCB footprint** name and **Schematic Symbol** name for that **Part**.
14. You can verify that the relevant Symbols and PCB Footprint are correct by editing the Part.
15. If the Schematic Symbol has the PCB Footprint name associated with it, the conversion mechanism will pick up the Footprint name and will present it in the Parts dialog. Where the name association doesn't exist, you will need to add the relevant Footprint by editing the Part definition.

Note: For multiple file import, you can also use the Data Transfer Wizard.