



Pulsonix Design System
Step-by-step Guide To
Protel Design & Library Conversion

Copyright

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.

Protel is a trademark of Altium.

All other trademarks are acknowledged to their respective owners.

Pulsonix, a division of WestDev Ltd.

Printed in the UK. Issue date: 26/03/2014 iss 3

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 PROTEL INFORMATION	4
Overview	4
Which versions of Protel are supported?	4
File extensions expected	4
Protel databases.....	4
Filenames	5
CHAPTER 2. EXPORTING SCHEMATIC DESIGNS	6
Exporting Schematic designs From Protel into Pulsonix	6
Importing Protel Schematic designs into Pulsonix	8
CHAPTER 3. EXPORTING SCHEMATIC SYMBOLS.....	10
Exporting Schematic Symbols From Protel into Pulsonix.....	10
Importing Protel Schematic Symbols into the Pulsonix libraries	12
CHAPTER 4. EXPORTING PCB DESIGNS	15
Exporting PCB Designs From Protel into Pulsonix.....	15
Importing Protel PCB designs into Pulsonix	16
CHAPTER 5. EXPORTING PCB FOOTPRINTS	18
Exporting PCB footprints From Protel into Pulsonix	18
Importing Protel PCB Footprints into Pulsonix.....	20
CHAPTER 6. IMPORTING PARTS.....	22
Importing Parts From Protel into Pulsonix	22

Chapter 1. Exporting Protel Information

Overview

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

Which versions of Protel are supported?

Protel 98, 99SE and Protel DXP ASCII files are supported.

File extensions expected

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

Protel 98 and 99SE

- For Protel Schematics – .prj
- For Protel Schematic Library Symbols – .lib
- Parts – in Protel Parts and Symbols are the same thing.
- For Protel PCB Layout – .pcb
- For Protel PCB Footprint libraries – .lib
- In Protel, PCB footprints don't have any Part information.

Protel DXP

- In Protel DXP Schematic design - .schdoc
- Protel DXP projects are - .eproject
- In Protel DXP PCB design - .pcbdoc

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

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

Protel databases

Protel designs and libraries are held in databases. The databases must be opened and then the designs or libraries that you wish to export must be opened individually.

Filenames

Quite often Protel will not save the file with the name that you asked it to. The name will have either the words "Backup of " or "Copy of " added at the start of the name that you chose. This is not a problem when saving libraries or PCB designs, but when importing multiple design file Schematic designs into Pulsonix the filenames have to be correct, so you may need to rename these files to their correct name using the File Explorer.

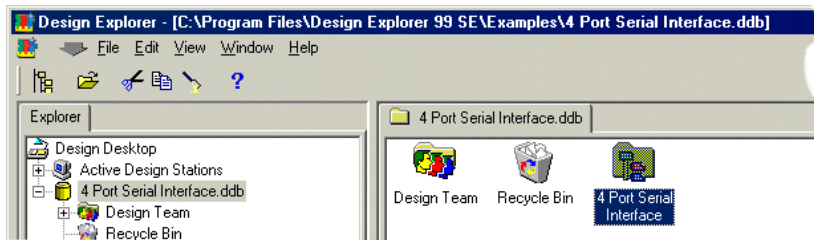
Chapter 2. Exporting Schematic Designs

Exporting Schematic designs From Protel into Pulsonix

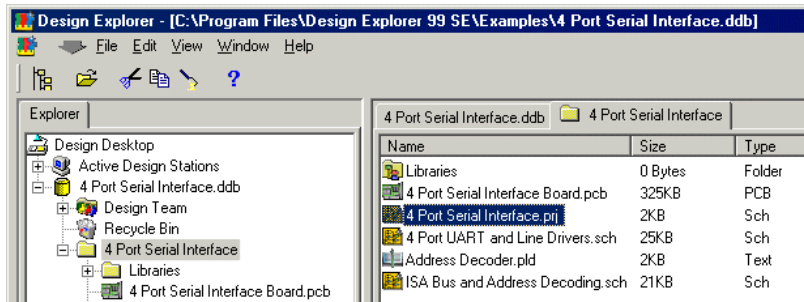
Schematic designs can be made up of many design files, which may be controlled by a design project file. To successfully transfer a schematic design all the design files must be exported. If there is a project file controlling the Schematic design and you export this and all the other schematic design files with their correct name, then the whole Schematic design can read into Pulsonix just by opening the ASCII version of the project file.

► Exporting Schematic designs or Projects

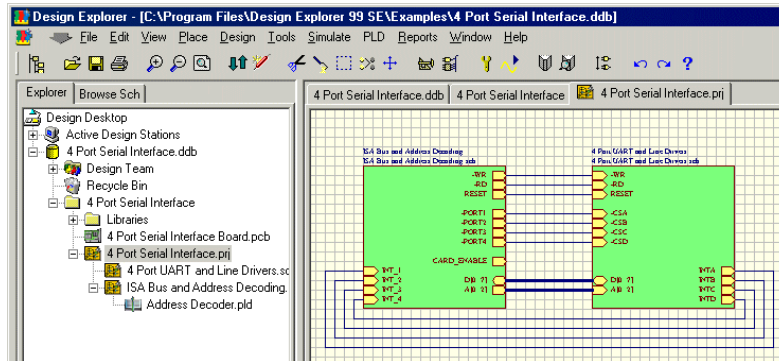
1. In Protel, from the **File** menu select **Open**
2. Choose the database to open (*.ddb)
3. The design Explorer will open.



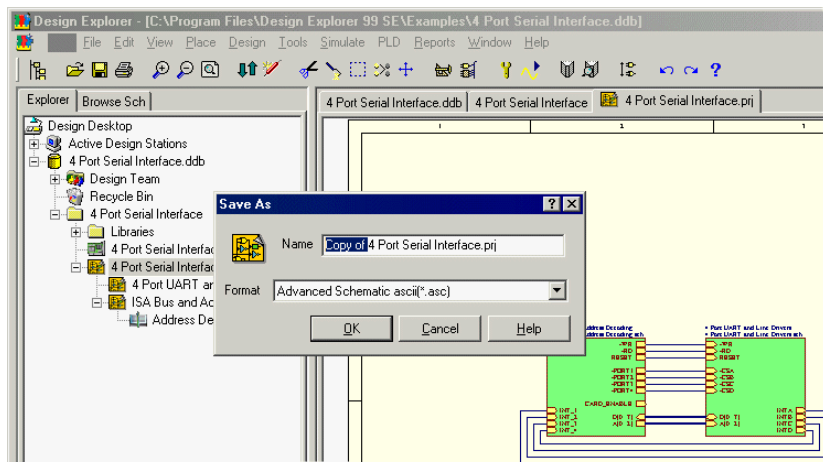
4. Double click on the design name, e.g. **4 Port Serial Interface**
5. The design window will open.



6. Select the **Project** file (.pri), this will contain all the sheets for the design
7. The top level design will open.



8. From the **File** menu, select **Save As**

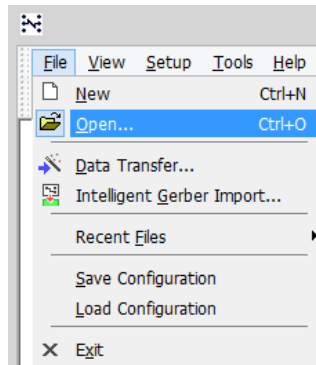


9. The **Save As** dialog will allow you to edit the file **Name** and the **Format**.
10. Edit the filename to remove the words ‘**Copy of**’.
11. From the drop down list, change the format to be **Advanced Schematic ascii (*.asc)**
12. Click **OK** to export the design.
13. The resultant file will be saved into the project folder but will require editing to remove the words ‘**Backup of..**’
14. This completes the export of the Schematic Design file.

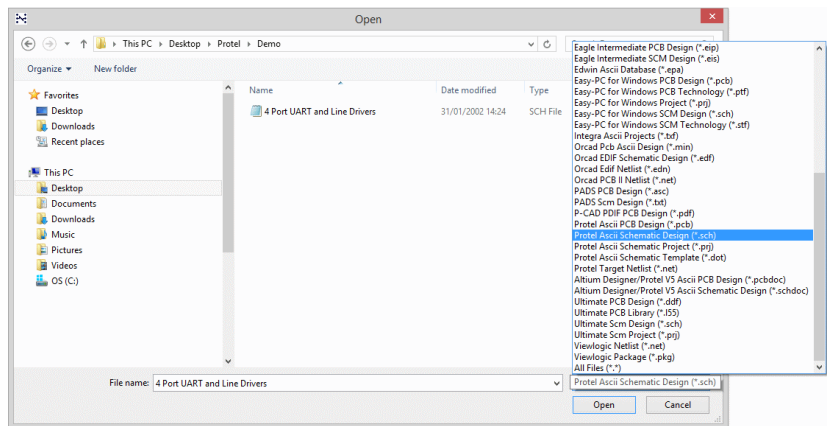
Importing Protel Schematic designs into Pulsonix

► To import the Schematic file into Pulsonix

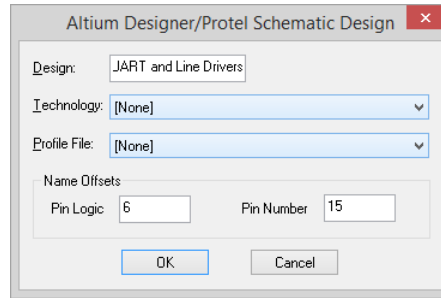
1. In Pulsonix, from the **File** menu select **Open**.



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 **PROTEL ASCII Schematic Project (*.prj)**
4. Select your design file and click **Open**.



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



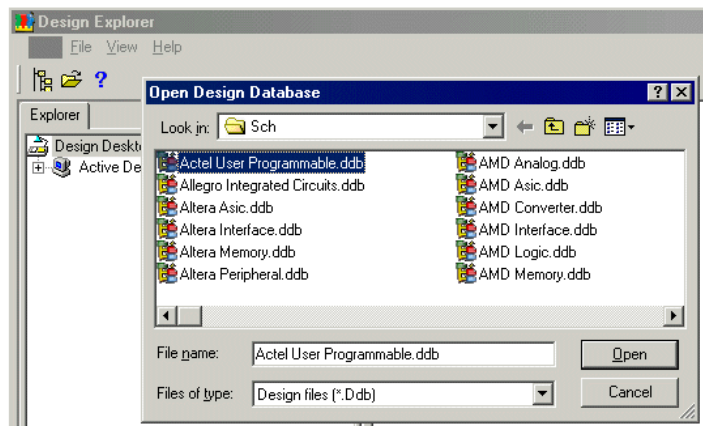
6. If you hadn't selected the File of type list to be the Protel filter previously, Pulsonix will still automatically detect the file format and import it in the correct manner.
7. The target **Design** name is presented. You can edit this to the required name.
8. For **Technology** you don't need to 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. The **Profile File** is used to contain predefined drawing borders and suchlike. Generally speaking, these are contained within the Protel design so they are not required at this stage. You can add them after in Pulsonix anyway.
10. If your design does not use the default **Pin Logic** and **Pin Number** Offset, you can specify your values in the **Name offsets** dialog. The units used are thou.
11. A progress message will appear followed by the design that will automatically open in Pulsonix.
12. If there are any errors the design will not open. If errors can be corrected in Protel 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

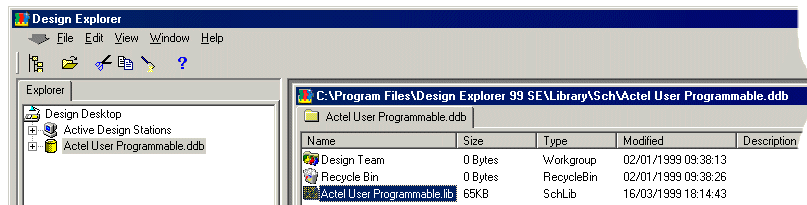
Exporting Schematic Symbols From Protel into Pulsonix

► Exporting Schematic Symbols

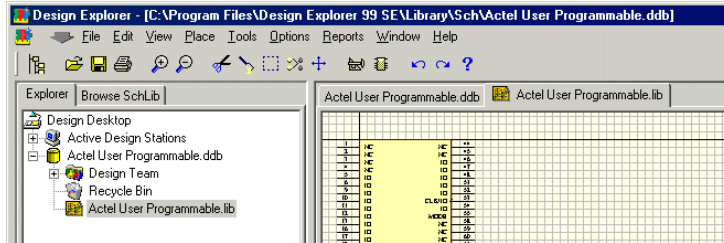
1. In Protel, from the **File** menu select **Open**
2. Browse to the library folder required



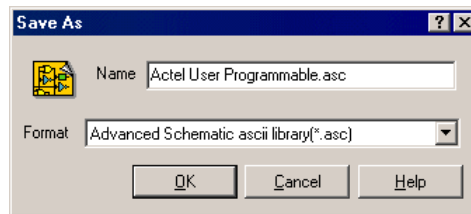
3. From the Open dialog select the library database required and click **Open**.
4. This procedure will need to be done for every library required for conversion, they cannot be done as a batch.
5. The opened library will appear in the browser.



6. Double click on the library filename to open it.



7. You should ensure that you are using the **EDIF** dialog.
8. From the **File** menu, click **Save As**.

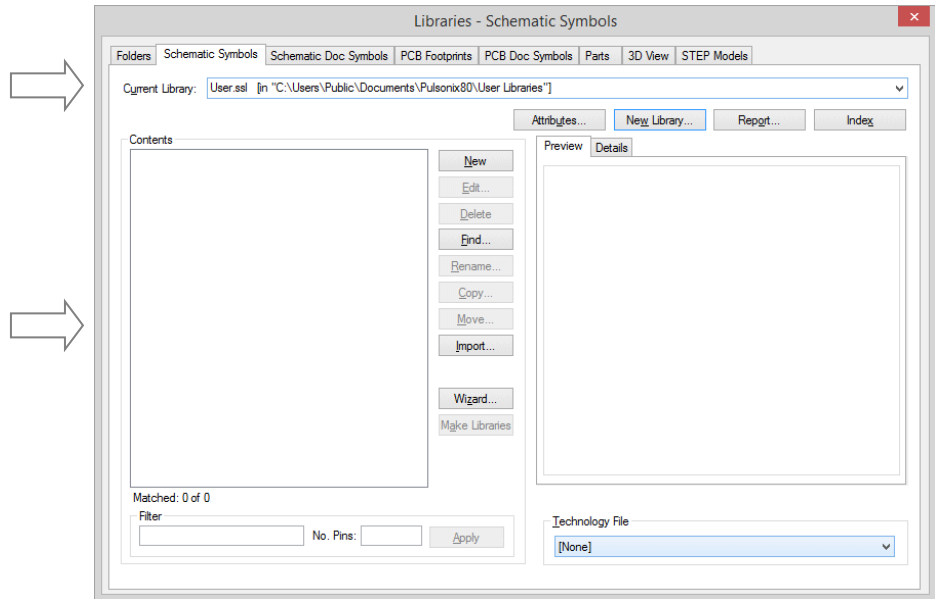


9. Edit the **Name** to that required.
10. On **Format**, from the drop down list box select **Advanced Schematic ascii library (*.asc)**.
11. Click **OK** to start the export.
12. This completes the export of the Schematic Symbol file.
13. You must open and convert each of the symbol libraries that you require to use in Pulsonix.

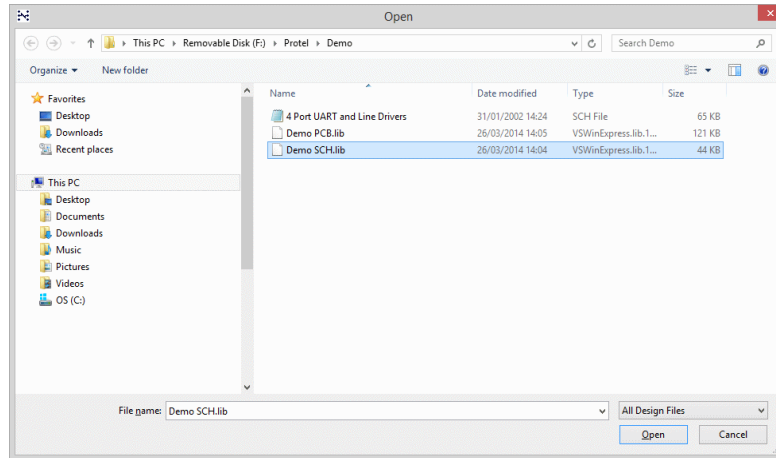
Importing Protel Schematic Symbols into the Pulsonix libraries

► To import the Schematic file into Pulsonix

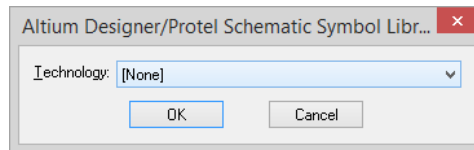
1. In Pulsonix, from the **Setup** menu select **Libraries**.
2. The **Library Manager** is displayed.
3. Select the **Schematic Symbols** tab.



4. From the current library drop down list, select the **User.ssl** library. You can also create your own library if you like. The **Contents** list will be empty at this point.
5. Click the **Import** button.
6. Select the file for import. This will be the **.asc** file that you created in the previous procedure.

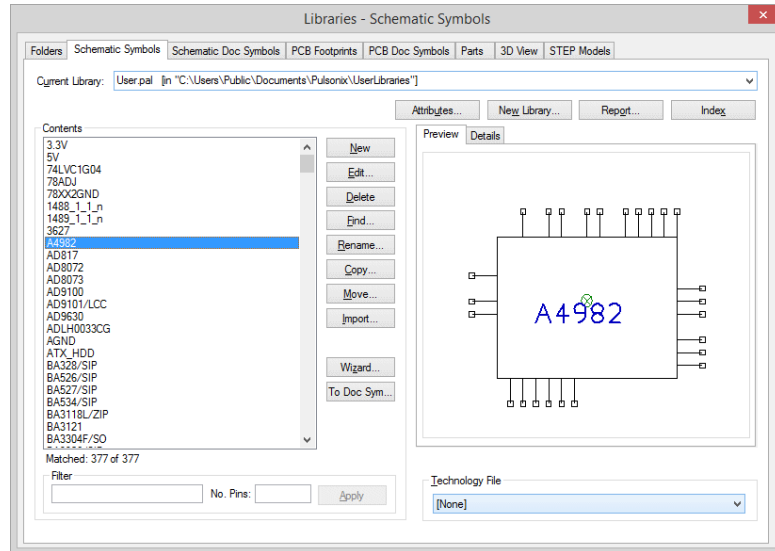


7. There is no need to change the **File of type** unless you require a finer filtering of the list of files for selection.
8. Pulsonix will automatically detect the file type being imported.
9. The file type will be acknowledged which you confirm with **OK**.



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

14 Exporting Schematic Symbols



13. 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.
14. 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.
15. 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.

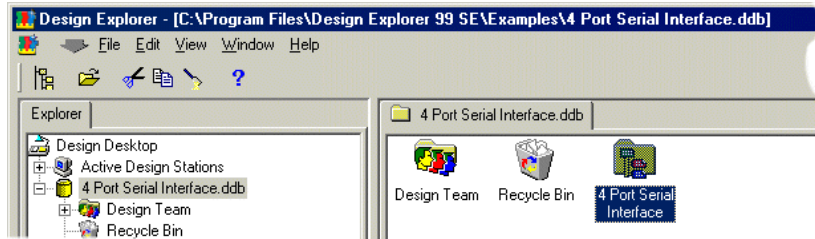
Note: For multiple file import, you can also use the Data Transfer Wizard.

Chapter 4. Exporting PCB Designs

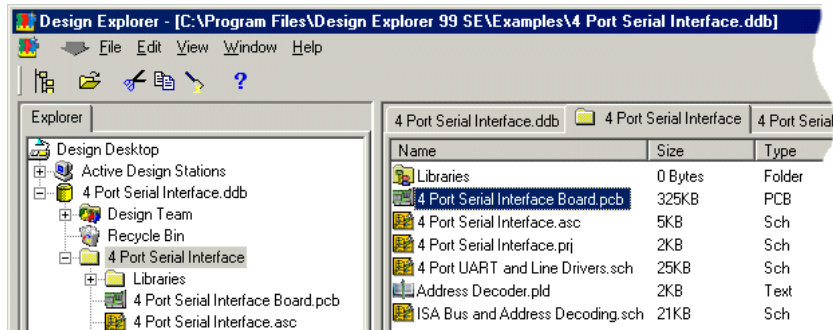
Exporting PCB Designs From Protel into Pulsonix

► Exporting PCB Designs

1. In Protel, from the **File** menu select **Open**
2. Choose the database to open (*.ddb)
3. The design Explorer will open.

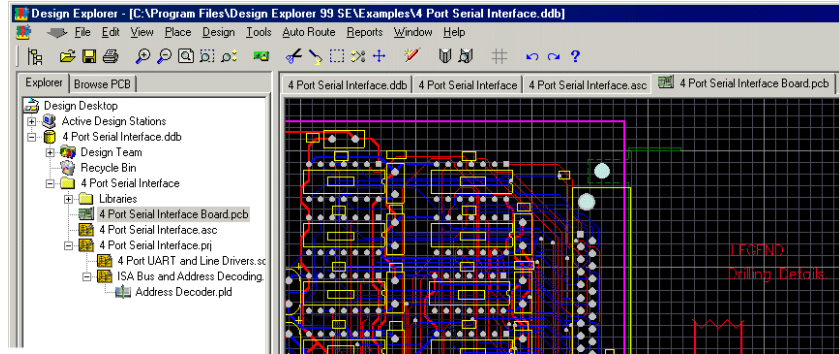


4. Double click on the design name, e.g. 4 Port Serial Interface
5. The design window will open.

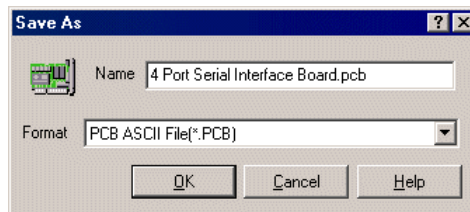


6. With the Design explorer displayed, double click on the design name of the .pcb file.
7. The PCB design will open.

16 Exporting PCB Designs



8. From the **File** menu, select **Save As**.

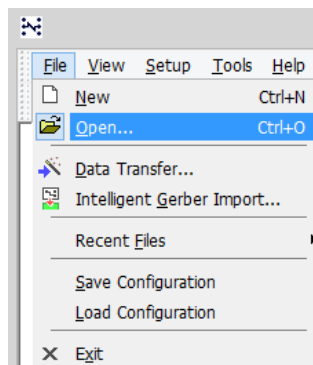


9. Edit the filename to that required.
10. Using the drop down list, change the Format to be **PCB ASCII File (*.PCB)**
11. Click **OK**.
12. This ASCII file will be saved to the same location as the original design file.
13. You are ready to import the ASCII file into Pulsonix.

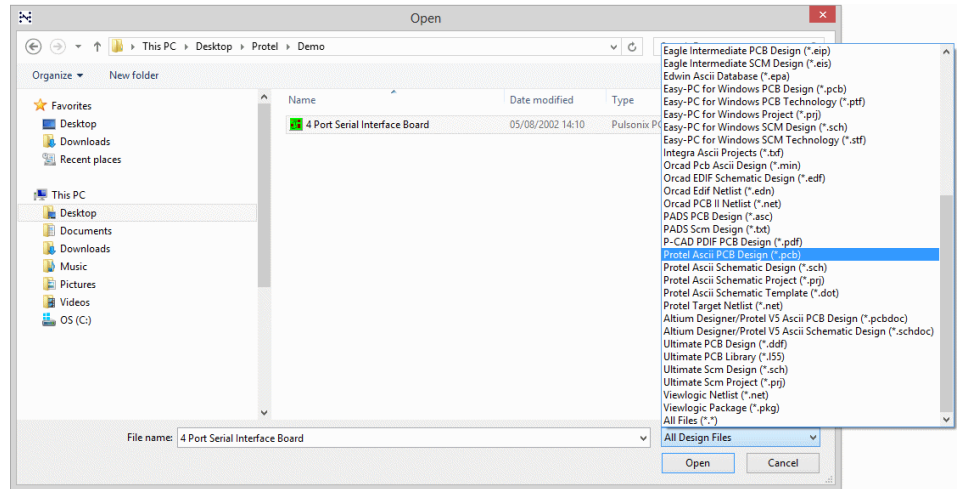
Importing Protel PCB designs into Pulsonix

► To import the PCB file into Pulsonix

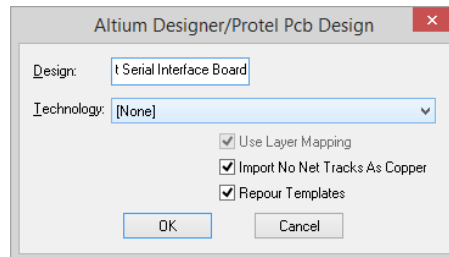
1. In Pulsonix, from the **File** menu select **Open**.



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 **PROTEL Ascii PCB Design (*.pcb)**
4. Click **Open**.
5. The **Protel PCB Design** import dialog is displayed.



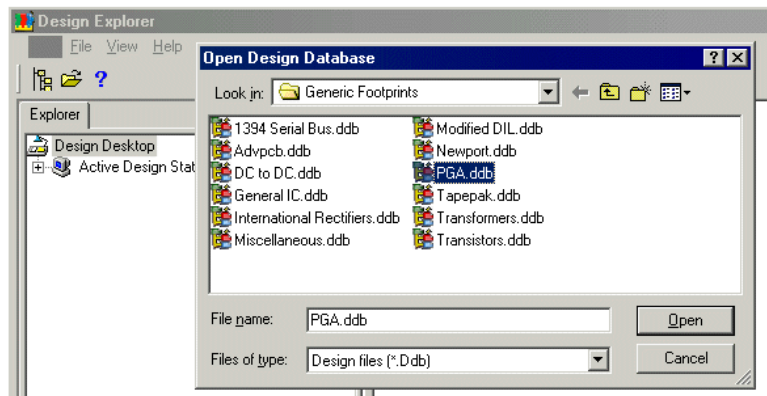
6. The target **Design** name is presented. You can edit this to the required name.
7. For **Technology** you don't need to select a file, so select **[None]**. However, if you wish to apply a new set of colours or styles to the incoming design, select a Technology file from the drop down list.
8. A 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 Protel 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 5. Exporting PCB Footprints

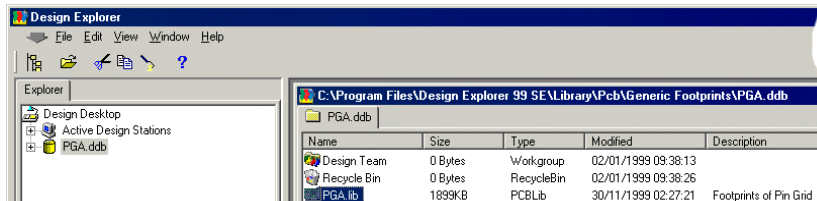
Exporting PCB footprints From Protel into Pulsonix

► Exporting PCB Footprints

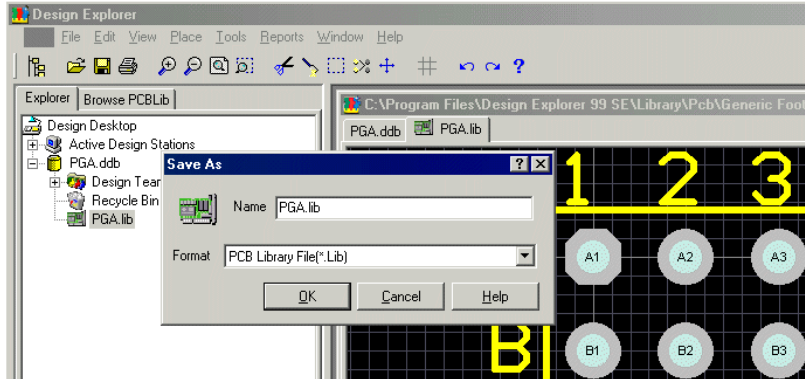
1. In Protel, from the **File** menu select **Open**
2. Browse to the library folder required



3. From the Open dialog select the library database required and click **Open**.
4. This procedure will need to be done for every library required for conversion, they cannot be done as a batch.
5. The opened library will appear in the browser.



6. Double click on the library filename to open it.
7. From the **File** menu, click **Save As**.

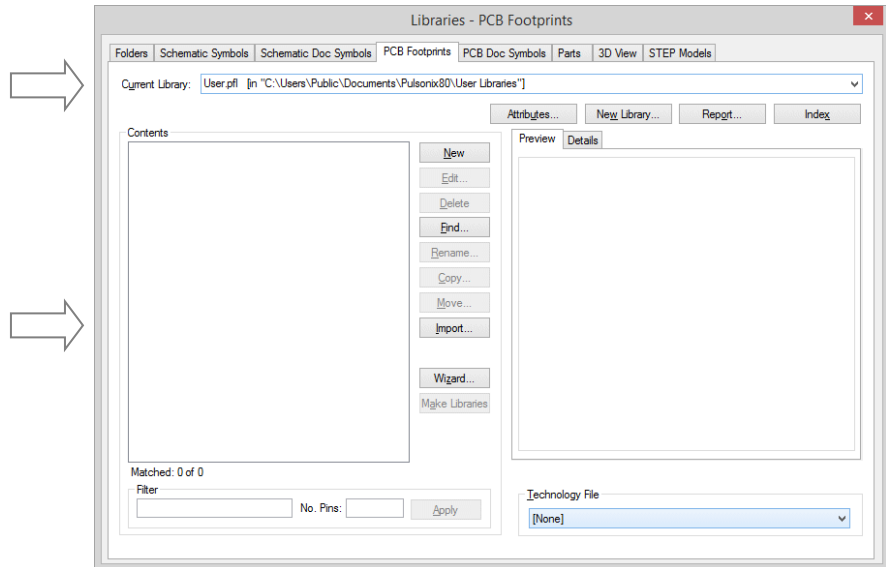


8. Edit the **Name** to that required.
9. On **Format**, from the drop down list box select **PCB Library file (*.lib)**.
10. Click **OK** to start the export.
1. This completes the export of the PCB footprint
2. Once completed, a report is displayed. Click **OK** to exit.

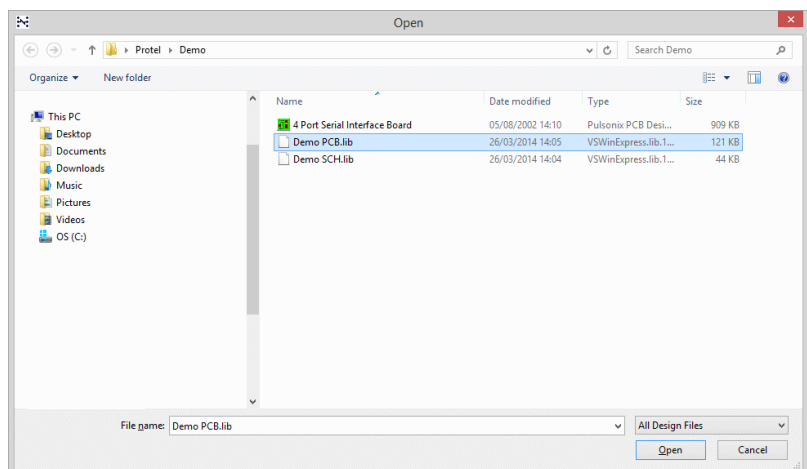
Importing Protel PCB Footprints into Pulsonix

► To import the PCB footprint file into Pulsonix

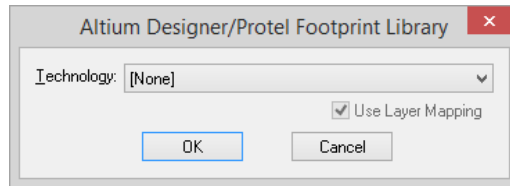
1. In Pulsonix, from the **Setup** menu select **Libraries**.
2. The **Library Manager** is displayed.
3. Select the **PCB Footprints** tab.



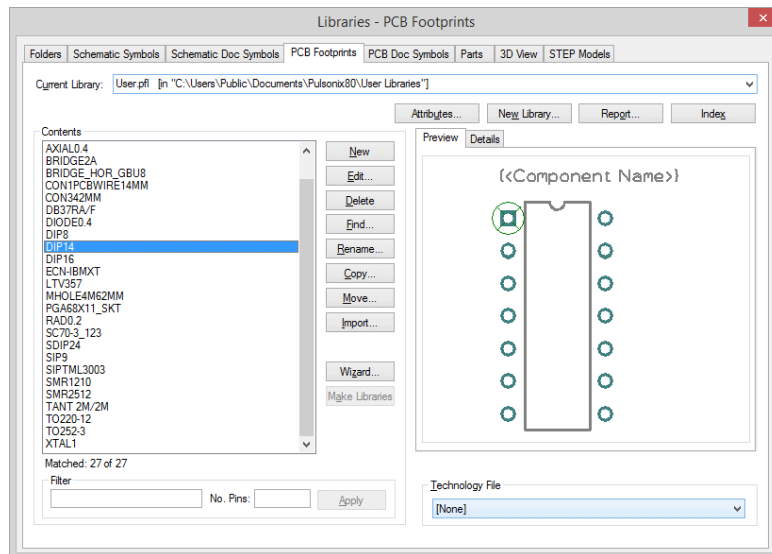
4. From the current library drop down list, select the **User.pfl** library. You can also create your own library if you like. The **Contents** list will be empty at this point.
5. Click the **Import** button.



6. Select the file for import. This will be the **.lib** 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

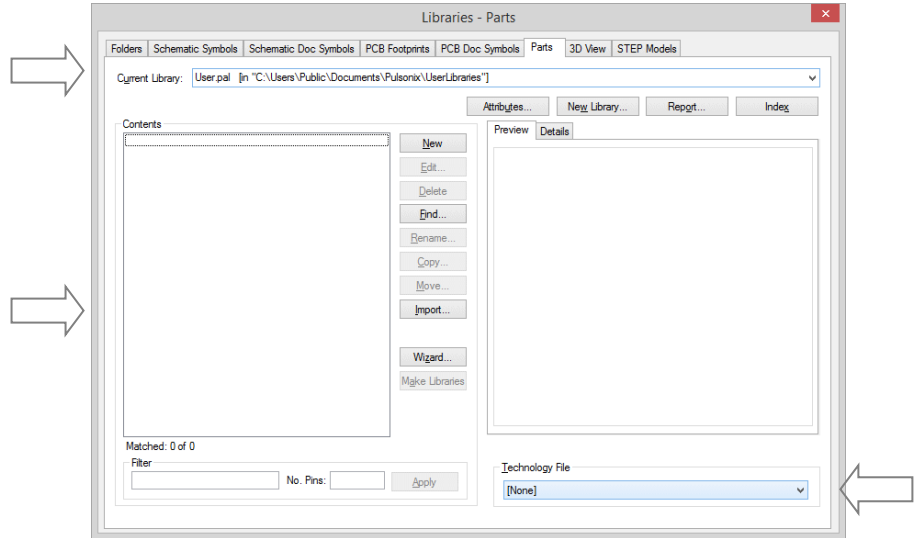
Importing Parts From Protel into Pulsonix

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

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

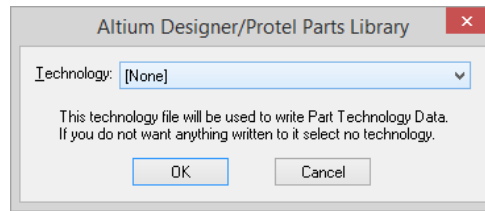
► To import the Parts files into Pulsonix

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

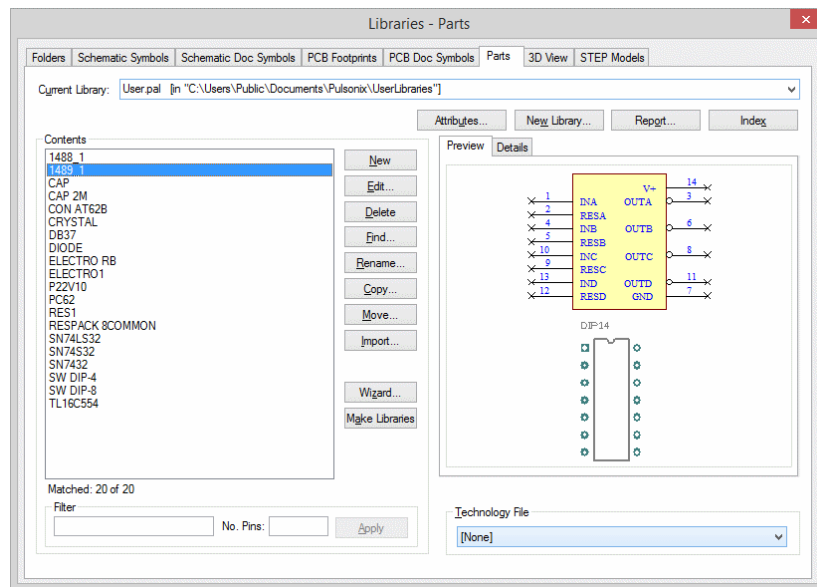


2. The **Library Manager** is displayed.
3. Select the **Parts** tab.
4. From the current library drop down list, select the **User.pal** library. You can also create your own library if you like. The **Contents** list will be empty at this point.
5. Click the **Import** button.
6. Select the file for import. This will be the **.asc** 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. One isn't required for Protel files as it is already built-in.



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.