

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	
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	
Exporting PCB footprints From Protel into Pulsonix	18
Importing Protel PCB Footprints into Pulsonix	
CHAPTER 6. IMPORTING PARTS	
Importing Parts From Protel into Pulsonix	

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.

👫 Design Explorer - [C:\Program Files\Design	Explorer 99 SE\Examples\4 Port Serial Interface.ddb]
📑 🦇 Eile Edit View Window Help	
🏗 😅 🗲 🖻 🍾 📍	
Explorer	4 Port Serial Interface.ddb
🝰 Design Desktop	
🕀 😻 Active Design Stations	
🖻 👘 👸 4 Port Serial Interface.ddb	Design Team Recycle Bin 4 Port Serial
庄 🧑 Design Team	Interface
📕 🚽 🎬 Recycle Bin	

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

ݶ Design Explorer - [C:\Program Files\Design	Explorer 99 SE\Examples\4 Port Ser	ial Interface.d	ldb]
饕 —— Eile Edit ⊻iew Window Help │ 院 译 ✔ 陷 ∖ ?			
Explorer	4 Port Serial Interface.ddb 📃 4 Port	Serial Interface	
🝰 Design Desktop	Name	Size	Туре
😟 🧕 Active Design Stations	🙀 Libraries	0 Bytes	Folder
🖻 👘 4 Port Serial Interface.ddb	4 Port Serial Interface Board.pcb	325KB	PCB
🕀 🦏 Design Team	4 Port Serial Interface.prj	2KB	Sch
	4 Port UART and Line Drivers.sch	25KB	Sch
i⊟i⊒i 4 Port Serial Interface iti⊒i Libraries	Address Decoder.pld	2KB	Text
4 Port Serial Interface Board.pcb	ISA Bus and Address Decoding.sch	21KB	Sch

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







- 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.

Altiu	m Designer/Protel Schematic Design	×
<u>D</u> esign:	JART and Line Drivers	
Technology:	[None]	~
Profile File:	[None]	~
Name Offse Pin Logic	6 Pin Number 15	
	OK Cancel	

- 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

📕 D	esign Explore	: r			
	<u>File</u> <u>V</u> iew	<u>H</u> elp			
	🖻 ?	Open Design	Database		? ×
	Design Deskti	Look jn: 🔂	Sch	- ÷ ÷	➡ ■•
	Active De	Actel User	Programmable.ddb	🎽 AMD Analog.d	
	Ť	1000	egrated Circuits.ddb	🎽 AMD Asic.ddb	
		Altera Asic.		MD Converte	
		Altera Inter		MD Interface.	
		Altera Mem	·	🎽 AMD Logic.ddl	
		🎽 Altera Perip	oheral.ddb	隨 AMD Memory.c	dbt
		•			Þ
		File <u>n</u> ame:	Actel User Programmable.ddb		<u>O</u> pen
		Files of type:	Design files (*.Ddb)	-	Cancel

- 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.

👫 Design Explorer					
- ————————————————————————————————————					
隋 🖻 🗲 🖻 🏷 🣍					
Explorer	💽 🚺 C:\Program File	s\Design Explorer 99) SE\Library\Sc	h\Actel User Program	nable.ddb
🝰 Design Desktop	📃 📄 Actel User Progr	ammable.ddb			
Active Design Stations	Name	Size	Тире	Modified	Description
🗄 👘 💼 Actel User Programmable.ddb	🙀 Design Team	0 Bytes	Workgroup	02/01/1999 09:38:13	1 b oconpaio
	🗑 Recycle Bin	0 Bytes	RecycleBin	02/01/1999 09:38:26	
	Actel User Progra	mmahle lih 65KB	Schlib	16/03/1999 18:14:43	

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.

Save As	• 20 • 20 • 20 • 20 • 20 • 20 • 20 • 20	×
	Name Actel User Programmable.asc	
Format	Advanced Schematic ascii library(*.asc)	1
	<u> </u>	

- 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.

Libraries - Schematic Symbols
Folders Schematic Symbols Schematic Doc Symbols PCB Footprints PCB Doc Symbols Parts 3D View STEP Models
Current Library: User.ssl [in "C:\Users\Public\Documents\Pulsonix80\User Libraries"] v
Attributes New Library Report Index Contents
Import Wigard Mgke Libraries Matched: 0 of 0 Filer No. Pins: Apply Iechnology File [None]

- 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.

) 🍥 🕤 🕆 퉬 🕨 This PC 🔸 Removable Disk	(F:) → Protel → Demo		✓ C Search De	mo)
)rganize 🔻 New folder				8≡ • [
Favorites	Name	Date modified	Туре	Size	
Desktop	4 Port UART and Line Drivers	31/01/2002 14:24	SCH File	65 KB	
🐌 Downloads	Demo PCB.lib	26/03/2014 14:05	VSWinExpress.lib.1	121 KB	
Secent places	Demo SCH.lib	26/03/2014 14:04	VSWinExpress.lib.1	44 KB	
Music					
Pictures Videos OS (C:)					

- 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**.

Altium Des	igner/Protel Schematic Symbol	Libr ×
<u>T</u> echnology:	[None]	¥
	OK Cancel	

- 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.

ders Schematic Symbols urrent Library: User.pal	Schematic Doc Symbols PCB In "C:\Users\Public\Documents\			Parts 3D Vie	w STEP	Models	
Contents 3.3V 5V 7ULVC1604 78ADD 1483_1_1_n 1483_1_1_n 1483_1_1_n 1483_1_1_n 1483_1_1_n AD8073 AD8073 AD8073 AD9101/LCC AD9503 AD9101/LCC AD9503 AD9101/LCC AD9503 AD9101/LCC AD9503 AD9101/LCC AD9503 AD9101/LCC AD9503 AD9101/LCC AD9503 AD9101/LCC AD9503 AD9101/LCC AD9503 AD9101/LCC AD9503 AD9101/LCC AD9503 AD9101/LCC AD9503 AD9101/LCC AD9503 AD9101/LCC AD910/LCC	Â	New Edt Delete End Copy Move Import Wigard To Doc Sym.		Details	A 4 9	Report	
BA3121 BA3304F/SO Matched: 377 of 377 Filter	No. Pins:	Apply	Technol [None]				v

- 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.

🎊 Design Explorer - [C:\Program Files\Design	Explorer 99 SE\Examples\4 Port Ser	ial Interface.c	ldb]
📑 🦇 Eile Edit View Window Help			
🏗 🖻 🗲 🗈 🍾 📍			
Explorer	4 Port Serial Interface.ddb 📃 4 Port :	Serial Interface	4 Port Serial
🝰 Design Desktop	Name	Size	Туре
Active Design Stations	👔 🥦 Libraries	0 Bytes	Folder
🖻 📄 🛑 4 Port Serial Interface.ddb	🕮 4 Port Serial Interface Board.pcb	325KB	PCB
⊡ 🧊 Design Team	🛛 🧱 4 Port Serial Interface.asc	5KB	Sch
Recycle Bin	🛛 🧱 4 Port Serial Interface.prj	2KB	Sch
	🛿 🧱 4 Port UART and Line Drivers.sch	25KB	Sch
4 Port Serial Interface Board.pcb	Address Decoder.pld	2KB	Text
4 Port Serial Interface.asc	ISA Bus and Address Decoding.sch	21KB	Sch

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



8. From the File menu, select Save As.

Save As				? ×
	Name	4 Port Serial	Interface Board.pc	b
Format	PCB AS	CII File(*.PCB)		•
		<u>0</u> K	<u>C</u> ancel	<u>H</u> elp

- 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.

N								
<u>F</u> ile	<u>V</u> iew	<u>S</u> etup	<u>T</u> ools	<u>H</u> elp				
D	<u>N</u> ew			Ctrl+N				
- 🖻	<u>O</u> pen			Ctrl+O				
Save Configuration								
×	E <u>x</u> it							

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

	Open			×
🕒 🎯 👻 🛧 🌗 > This PC 🔸 Desktop	p → Protel → Demo		v C	Eagle Intermediate PCB Design (*.eip)
Organize 🔻 New folder				Eagle Intermediate SCM Design (*.eis) Edwin Ascii Database (*.epa)
🔆 Favorites	^ Name	Date modified	Туре	Easy-PC for Windows PCB Design (*.pcb) Easy-PC for Windows PCB Technology (*.ptf)
E Desktop	4 Port Serial Interface Board	05/08/2002 14:10	Pulsonix P	Easy-PC for Windows Project (*.prj) Easy-PC for Windows SCM Design (*.sch)
🚺 Downloads				Easy-PC for Windows SCM Technology (*.stf)
🔢 Recent places				Integra Ascii Projects (*.txf) Orcad Pcb Ascii Design (*.min)
				Orcad EDIF Schematic Design (*.edf)
📜 This PC				Orcad Edif Netlist (*.edn) Orcad PCB II Netlist (*.net)
Lesktop				PADS PCB Design (*.asc)
Documents				PADS Scm Design (*.txt) P-CAD PDIF PCB Design (*.pdf)
0 Downloads				Protel Ascii PCB Design (*.pcb)
Music				Protel Ascii Schematic Design (*.sch) Protel Ascii Schematic Project (*.pri)
Pictures				Protel Ascii Schematic Template (*.dot)
Videos				Protel Target Netlist (*.net) Altium Designer/Protel V5 Ascii PCB Design (*.pcbdoc)
🔤 05 (C:)				Altium Designer/Protel V5 Ascii Schematic Design (*.schde
				Ultimate PCB Design (*.ddf) Ultimate PCB Library (*.155)
				Ultimate Scm Design (*.sch)
				Ultimate Scm Project (*.prj)
				Viewlogic Netlist (*.net) Viewlogic Package (*.pkg)
	*			All Files (*.*)
File name: 4 Port Serial	al Interface Board		~	All Design Files
				Open Cancel

- 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.

A	ltium Designer/Protel Pcb Design	×
<u>D</u> esign:	t Serial Interface Board	
<u>T</u> echnology:	[None]	~
	🕑 Use Layer Mapping	
	Import No Net Tracks As Co	opper
	Repour Templates	
	OK Cancel	

- 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

👥 Design Explorer	
<u>F</u> ile ⊻iew <u>H</u> elp	Open Design Database ? 🗙
] ╠ ≌ ?	Look in: 🔂 Generic Footprints 💽 🔶 🛍 📅
Explorer	. 🎼 1394 Serial Bus.ddb 🛛 🖉 Modified DIL.ddb
Design Desktop	Advpcb.ddb PGA.ddb C to DC.ddb C PGA.ddb
	General IC.ddb Tapepak.ddb
	international Rectifiers.ddb 🎬 Transformers.ddb
	Miscellaneous.ddb 👘 Transistors.ddb
	File <u>n</u> ame: PGA.ddbpen
	Files of type: Design files (*.Ddb)
	11

- 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.

🔝 Design Explorer					
- — — Eile Edit ⊻iew Window Help					
🏗 😅 🗲 🖻 🍾 📍					
Explorer	💽 C:\Program File	s\Design Expl	orer 99 SE\Libra	ary\Pcb\Generic Foot	orints\PGA.ddb
Besign Desktop	PGA.ddb				
Active Design Stations GA.ddb	Name	Size	Туре	Modified	Description
	🥡 Design Team	0 Bytes	Workgroup	02/01/1999 09:38:13	
	📲 🎯 Recycle Bin	0 Bytes	RecycleBin	02/01/1999 09:38:26	

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

🚹 Design Explorer
File Edit View Place Iools Reports Window Help
Ĩª ≇∎를 ₽₽©¤ ∉≻∷≈+ # ∽~?
Explorer Browse PCBLib
Design Desktop PGA.ddb 💌 PGA.lib
Active Design Stations
E PGA.ddb Save As
🕒 🤬 Design Tear
PGA.lb
Format PCB Library File(*.Lib)
<u>D</u> K <u>C</u> ancel <u>H</u> elp

- 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.

	Libraries - PCB Footprints	
N	Folders Schematic Symbols Schematic Doc Symbols PCB Footprints PCB Doc Symbols Parts 3D View STEP Models	
	Current Library: User.pfl [in "C:\Users\Public\Documents\Pulsonix80\User Libraries"]	
	Attrbytes New Lbray Report Index Corterts	
	Matched: 0 of 0 Fiter No. Pins: Apply Ichnology File [None]	

- 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.

ŧ		Open				
🔄 🎯 👻 ↑ 🁪 ► Protel ► Demo				✓ 🖒 Search De	mo	۶
Organize 🔻 New folder						
	^	Name	Date modified	Туре	Size	
This PC		4 Port Serial Interface Board	05/08/2002 14:10	Pulsonix PCB Desi	909 KB	
Lesktop Documents		Demo PCB.lib	26/03/2014 14:05	VSWinExpress.lib.1	121 KB	
Downloads Pictures Videos S (C.)		Demo SCH.lib	26/03/2014 14:04	VSWinExpress.lib.1	44 KB	
File game: Demo PCB	,lib			✓ All Desig		ancel

- 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.

Altium Designer/Protel Footprint Library	×
Technology: [None]	~
✓ Use Layer Mapp	ing
OK Cancel	

- 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.

				es - PCB Footpri			
olders Sc	hematic Symbols	Schematic Doc Symbols	PCB Footprints	PCB Doc Symbols	Parts 3D View	STEP Models	
Current Lib	rary: User.pfl [in	"C:\Users\Public\Docum	ents\Pulsonix80\	User Libraries'']			¥
_				Attributes	. Ne <u>w</u> Library	r Report	Index
Contents AXIAL0.4	4		∧ Ne	Preview	Details		
CON1PC	HOR_GBU8 BWIRE14MM		Ed		(<com< td=""><td>ponent Name>}</td><td></td></com<>	ponent Name>}	
CON342 DB37RA DIODE0	/F		<u>D</u> el			°	
DIP8 DIP14 DIP16			<u>R</u> ena	me	õ	0	
ECN-IBM LTV357 MHOLE4					0	0	
PGA68X RAD0.2	11_SKT		<u>M</u> ov Impo		0	0	
SC70-3_ SDIP24 SIP9					0	0	
SIPTML3 SMR121 SMR251	0		Wi <u>z</u> a Make L		0	0	
TANT 21 TO220-1 TO252-3 XTAL 1	й/2M 2		v	branes	<u>ہ</u>	•	
	: 27 of 27		¥				
Filter		No. Pins:	Apph		ology File e]		~

- 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.

Libraries - Parts natic Doc Symbols PCB Footprints PCB Doc Symbols 3D View STEP Models Parts User.pal fin "C:\Users\Public\Documents\Pulsonix\UserLibraries" ¥ Attributes.. . Ne<u>w</u> Library... Rep<u>o</u>rt Preview Details New Edit... Delete End... Rename. Copy... Move... Import... Wizard... Make Libraries Matched: 0 of 0 Filter Technology File No. Pins: Apply [None]

1. From the Setup menu select Libraries.

To import the Parts files into Pulsonix

- 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.

Al	tium Designer/P	rotel Parts Library	×
<u>T</u> echnology:	[None]		~
		I to write Part Technology I ten to it select no technolo	
	OK	Cancel	

- 10. Click **OK** to import the file.
- 11. When the import has completed successfully, the **Library Manager** will show the Parts dialog again.

			Libraries	- Parts				
olders Schematic Sy			PCB Footprints PCB Do nents\Pulsonix\UserLibrarie	o ojinoolo	Parts 3D	View STE	P Models	•
Contents				Attributes		w Library	Rep <u>o</u> rt	Index
Contents [1498] [1408]			New Edit Delete Find Rename Copy Move Import	x 1 D(A) 0UTA 3 x 2 RESA 4 3 x 4 D(B) 0UTA 3 x 5 RESA 6 3 x 10 D(C) 0UTA 5 x 10 D(C) 0UTA 5 x 10 D(C) 0UTA 11 DC 0 0 0 11 x 12 RESD 6(A) 0 0 0 0 0 0 0 0 0 0 0			* * *	
Matched: 20 of 20		No. Pins:	Mgke Libraries	Techno [None	ology File	0	0	~

- 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.