

Disclaimer

This document explains how to convert a PADS decal and place it into OrCAD layout. This document is provided as a convenience to our customers to aid in the creation of footprints for their design. Samtec does not presently provide OrCAD libraries for our parts, and therefore is not liable for any conversion issues between PADS and OrCAD. We strongly recommend the user to review the decal for accuracy of attributes, layer information, and scaling after conversion.

If you do not own a fully functional demo version or licensed PADS seat and need an OrCAD decal extracted from PADS, a request can be submitted to Samtec at <mailto:PADS@samtec.com> for the ASCII or .max file.

Samtec does provide standard PADS footprints available free of charge from our website at http://www.samtec.com/standard_products/product_information/pad_schematics.aspx.

Introduction

The first and most problematic concern is the fact that OrCAD will only accept an ASCII file that is in PADS ASCII version 1.5. The limitation, with PADS, will only allow an export of a part decal or part name up to 16 characters for version 1.5. Due to the nature of Samtec's numbering system, it is not possible to directly export a Samtec decal as a Version 1.5 ASCII file as most of our part numbers exceed 16 characters. The part decal AND the part name must be changed to a name that has 16 characters or less.

PADS stores and extracts all parts from libraries. Libraries can contain one to hundreds of parts for layouts. As most customers only desire one part for their purposes, we have created libraries that are named by the connector part number and only contain that one part. Additional parts can be added to these libraries or left as is. A library consists of four files having extensions of .pt4, .pd4, .ld4, and .ln4. After downloading a part library from the website and unzipping it, these four files must be placed in the 'libraries' directory for PADS. This directory is in the 'My Documents\PADS Projects\Libraries' folder. After which you add this library to the already existing list which tells PADS this new library exists and for PADS to look at it. If not, you cannot pull the part into your design.

A good plan is to first create an additional library where you can save the renamed part. If not desired, you can simply save the part in the downloaded part library or any other library you wish.

IF YOU WANT TO ADD A NEW LIBRARY:

File...
 Library...
 New Lib...
 Type the name of the library you wish you use and hit 'Save'
 Close.

Now, go into PADSLayout.

Whether or not you created a new library or are going to use an already existing library, you need to tell PADSLayout to look at that library. Perform the following....

File...
 Library...
 Lib. List...
 Add...(From the list, click on the library where the part is located.)
 Press Open.
 Press OK.
 Close.

Now, after hitting the close button, you will automatically be returned to the layout screen.

PADS Decal conversion to OrCAD, Version 1

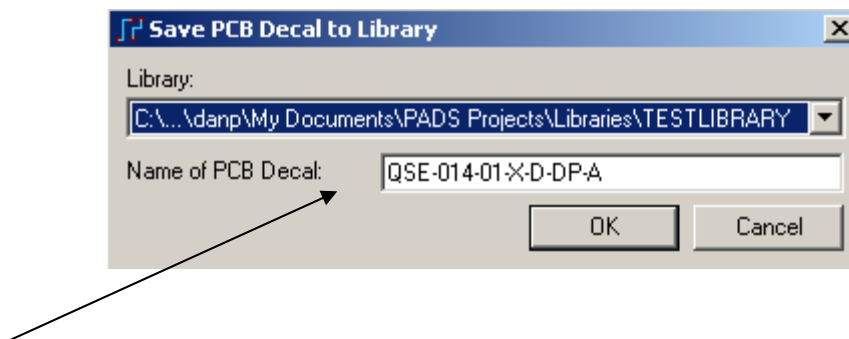
Tools...
Decal Editor

File..
Open Decal

Now, select the component you wish to convert to an ASCII and that decal will be brought to the screen.

File...Save Decal As...

You will be prompted by:



Change the Name of the PCB Decal to whatever you choose, but it must be 16 characters or less.

You also will need to select the library in which to save this part. I created a new library, TESTLIBRARY and added to the list for lookup.

Click OK.

You will then be prompted with....

Would you like to create a new part type?

Click YES. Type in a name for the part. It is suggested the name be the same as the decal.

File... Exit Decal Editor

You will return to PowerPCB and be prompted by:

Click OK.

PADS Decal conversion to OrCAD, Version 1

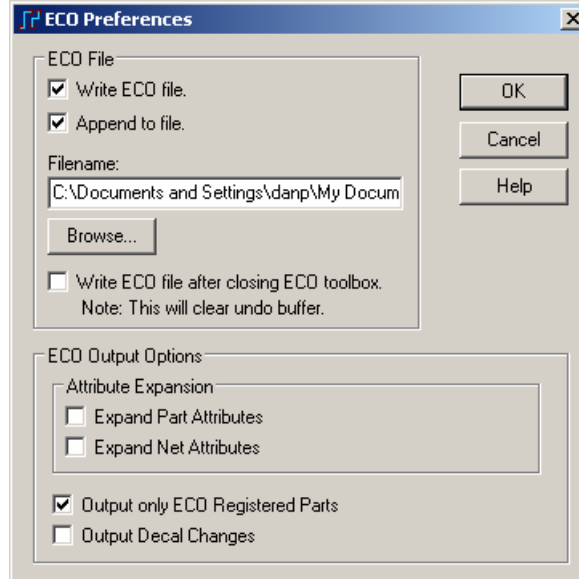
You have now created a new decal ready for ASCII export.

Now you need to bring in the decal to PADSlayout.

Depress the ECO button...



You will be prompted by this pop up...depress OK.



PADS Decal conversion to OrCAD, Version 1

Now you need to add the component. Depress the "Add Component" button



You will be prompted by a "Get part from library" pop up. Select the part you just created. Do not open the original decal as the file name could be longer than 16 characters. You will have to select the appropriate library. Note: You may have to type in an asterisk '*' and hit Apply to get the decals to list.

When you have selected the decal, hit Add. You may be prompted to type in a reference designator. This is of no consequence and can be anything. "J" is usually used for a connector.

Depress "Close".

You will see the outline of the connector move as you move your cursor. Depress the mouse button to put down the connector. Afterwards hit the "ESC" button.

Right click...select components.

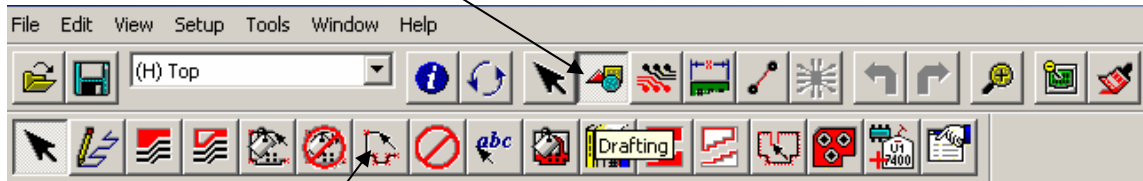
Now, select the entire part by depressing the cursor on the component and view the part name and decal name in the bottom left hand corner of the screen. You should see the part name and decal name you used. If not, the proper transfer did not occur.

If this is completed, one more step is necessary for ASCII export. OrCAD has a requirement that there be a board outline around the parts exported. Otherwise, the import will fail and the proper .max file will not be created.

CREATING A BOARD OUTLINE.

Zoom out enough so that you can view the entire connector. Hit "Ctrl + B".

Select the Drafting button.



Now, depress the Board outline and Cutout button.

Moving the cursor in the layout area, right click, and select the rectangle from the menu.

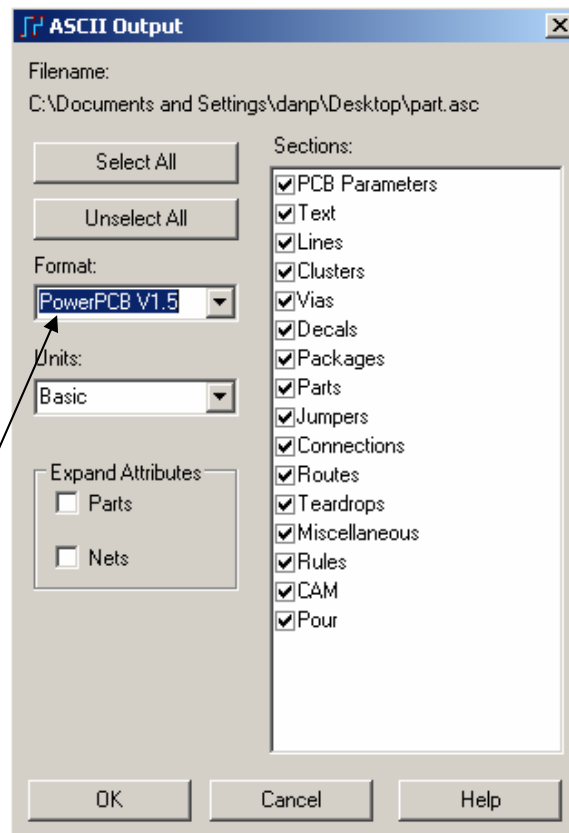
You are now going to draw a rectangle around the entire connector. All that matters is that the rectangle is bigger than the connector. One click will start the rectangle. No need to hold down the cursor. Just click once and let go. Move the cursor to the opposite corner and click again. There will now be a board outline around the connector.

Now we are ready to export the ASCII file.

EXPORTING A PADS DECAL (FOOTPRINT)

In PADSLayout...File...Export...

Select location and name of the file. Hit Save. You will then see...



Select PowerPCB V1.5 from the pull down menu.

Depress the 'Select All' button as well.

Hit OK.

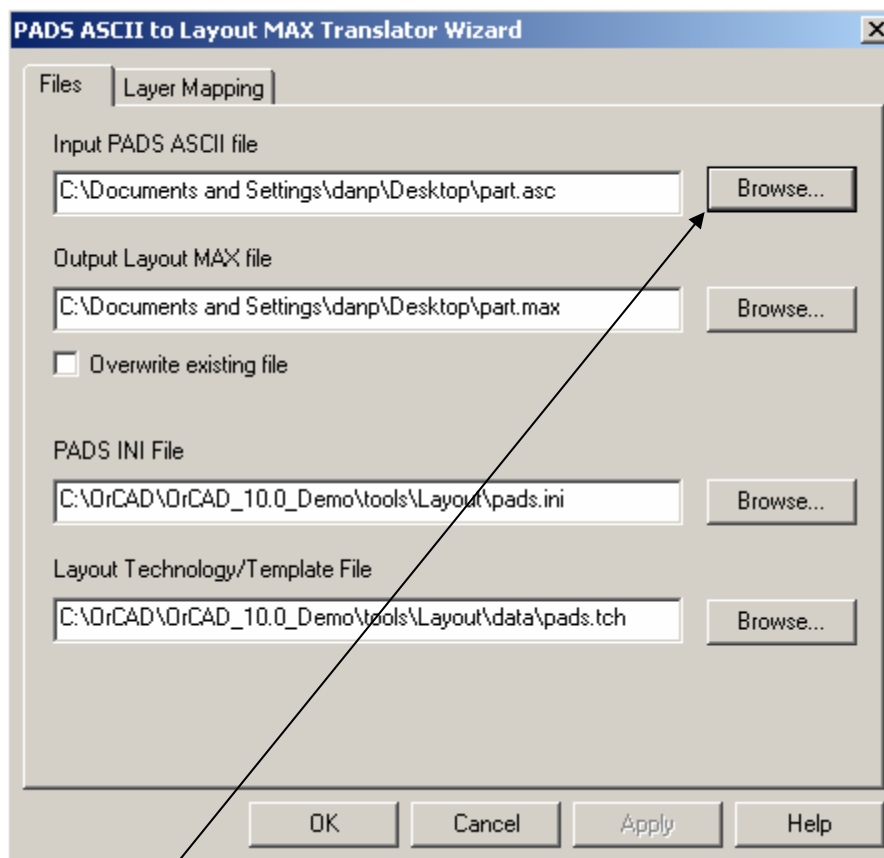
You will now have the proper ASCII file to pull into OrCAD.

IMPORTING FOOTPRINT INTO ORCAD

Open OrCAD.

File...Import...PADS PCB

You will then see...



Hit the 'Browse' button and select the ASCII file you just saved.

Now hit OK.

If everything goes well, you should see 'Processing Complete'.

The window will not close.

You will now have the .max file from which to open via OrCAD.

PADS Decal conversion to OrCAD, Version 1

File...Open...

Select the .max file you just saved and Open.

You should now see the decal you desire.