MIGRATION GUIDE

cādence[®]

Migrating from PADS to **OrCAD X**

Contents

Overview	2
Import Your PADS Schematic Data to OrC	AD X Capture3
Import Your PADS PCB Design into OrCAD	X Presto PCB Editor 4
Synchronize the Schematic to the Migrate	ed PCB7
Importing Your PADS PCB Libraries into O	rCAD X PCB Editor9
Next Steps	

Overview

Choosing the right PCB design solution is never an easy task. No matter if you are a startup company looking for tools to develop your next innovative electronic product or a large enterprise wanting a better solution to improve the productivity of your design team, selecting a PCB solution can be a daunting task. No one wants to get 75% of the way through a design to find out that the software you selected is not going to achieve what you need to accomplish.

Before you select a PCB design software package, there are many performance and capability aspects you should consider first:

- > Does the capabilities of the application and its technology meet your design requirements?
- Does the design software licensing fit within your budget?
- What level of support can you expect? Will you be able to get quick responses to your questions and access online tutorials? Is local help available?
- Can the application scale with your needs? As designs are getting more and more complex, will the capabilities of the tool adjust accordingly?
- How many other companies in your industry are using this tool and what is their feedback?

OrCAD X[®] offers an excellent solution for individual designers, small design teams, and large enterprises. OrCAD X offers constraint-driven design, advanced auto/interactive routing, high-speed design, DFM, dynamic shape technology, and much more, helping you deliver high-quality, first-time-right designs in the shortest timeframes. You can be confident that you will have the right solution and technologies at an affordable price to meet all of your design challenges today and tomorrow. Here are five of many reasons why:

- 30 years of innovation and leadership in the industry
- Affordable price and flexible purchase models
- Cutting-edge technologies
- Ecosystem empowered
- Industry's best customer support

Cadence[®] and OrCAD X provide the only full scalable PCB design solution on the market that can seamlessly grow with your needs. OrCAD X products are backed by Cadence and their network or certified Cadence Channel Partners (CCP). Get help when you need it by phone or email from local, knowledgeable PCB design professionals.

Like many companies selecting OrCAD X, you have existing or legacy designs you need to convert or translate into OrCAD X. The good news is that OrCAD X is supplied with an integrated and proven PADS translator built in. This guide will walk you through the steps and process involved in getting your design IP into the OrCAD X format so you can start realizing the advantages of moving to OrCAD X!

Import Your PADS Schematic Data to OrCAD X Capture

Step 1 - Export Schematic Data from PADS

In DxDesigner export your PADS schematic to EDIF 2.0 format using the EDIF Interfaces window:

EDIF Interfaces		×
EDIF Schematic Write	r	
Schematic/Symbol	vc709	
Project	F:\EVM_Demo\VC709\VC709.prj	Browse
Output	F:\EVM_Demo\VC709\vc709.eds	Browse
Configuration file		Browse
Level		
Convert Design H	lierarchically Alias	
Translate Symbo	I	
		^
<		>
	OK Cancel	Help

Note: If you are using PADS Logic you will need to convert your design into DxDesigner first using the Mentor-supplied conversion utilities, then you can proceed with Step 1 of the conversion process.

Step 2 - Import Schematic Data to OrCAD X Capture

Once the EDIF file is created, launch OrCAD X Capture and select the "File » Import » EDIF" command from the top menu.

💽 0	rCAD X Capture CIS-[/ - (SCHE	MATIC1 : PAG	E1)]										
File	Design Edit View To	ools Place	PCB SI Analysis	PSpice	Access								
	New		ちさ掛	. 77									
B	Open	•	art Page × /-	(SCHEMAT	IIC1 : PA								
	Close	Ctrl+F4											
	Save	Ctrl+S											
						In	nport Design					×	K
	Archive Project					The second se							
						ľ	Open						
			Import Selection							Bro	wse		
	Export		PSpice				Save As						
	Print Preview									Bro	wse		
	Print	Ctrl+P	PDIF				Configuration c:\cadence\	ifile ∖sob 23.1	\tools\car	pl Bro	wea		
	Print Setup		Design XML										
	Print Area		Library XML						ок	Cance		Help	
	1 bigeasydriver_v16a.opj		Altium Schemat	c Translato	ır								Ĺ
	2 BIGEASYDRIVER_V16A.OLB												
	3 bigeasydriver_v16a.opj		Eagle Schematic	Iranslator	·								
║ _	4 c:\users\\howto.opj												
	5 C:\USERS\\SYMBOLS_ALL	.OLB											
	6 FAULT-DETECTOR.opj												
	Exit												

Select your EDIF file, your destination output folder, and make sure you select the correct configuration file. Click **OK** to begin the translation. Once the EDIF file has translated into OrCAD X Capture, some text cleanup may be required.

Import Your PADS PCB Design into OrCAD X PCB Editor

Step 1 - Preparation

In the PADS PCB application, export your PCB database to the most recent ASCII format. The newer version of the ASCII can be exported using PADS version 5 or later by choosing the menu items: **"File » Export » ASCII"**

This creates an ASCII database (*.ASC) that will be used for translation into OrCAD X Capture format. Refer to the PADS documentation if more detail on exporting an ASCII database is required.

Note: Only copper pours in PADS defined as positive data are recognized by the translator.

Step 2 - Create a Blank Board

This step will ensure the translated PCB file is placed in the correct location. Create a new board from the home screen in the OrCAD X PCB Editor. You can also use the **"File » New >> Board"** command from the top menu bar.

In the New Board window, navigate to the folder where you want the translated PCB to be saved. Enter a board name; this name is just a placeholder and can be changed after the import completes.

OrCAD X Presto Profession	nal	
File Edit View Help		
New	×	Board
🛅 Open	Ctrl+O	Footprint
Open Recent	•	
Symphony Connect		
🖶 Save		
📔 Save As		
📔 Save All	Ctrl+S	
Import		

In OrCAD X Presto PCB Editor, choose "Import » Translators » PADS" from the top menu bar:



Browse to the **.ASC** file to be translated, create an options file, and select an Output Design folder.

Step 3 - Layer Mapping to OrCAD X PCB Editor

Select your ASCII database for Import, enter an options file name and select **"Show options dialog"** to create your options file, then select **"Translate"**:



Map PADS layer numbers to the corresponding OrCAD X Presto PCB Editor layers (PCB footprint, board data layers, etch):

LINE	COPPER TEXT DECAL F	AD VIA		
Layer	Layer Name	Class	Subclass	^
0	All layers	BOARD GEOMETRY	ALL	
1	Primary Component Side	ETCH	TOP	
2	Ground Plane	ETCH	Ground Plane	
3	Power Plane	ETCH	Power Plane	
4	Secondary Component Side	ETCH	BOTTOM	
5	Layer_5	UNUSED		
6	Layer_6	UNUSED		
7	Layer_7	UNUSED	•	
8	Layer_8	UNUSED	-	
9	Layer_9	UNUSED		
10	Layer_10	UNUSED		~
<				>
Class:	BOARD GEOMETRY	Sub Class:		~
1				
utomatio	c solder layer creation) o not create Teardrops	;
Crea	ate solder layers. Mils to oversi	ze: 0	Create Dynamic Shapes	
hrough	pin thermal/anti pad creation			
creat	e thermal/anti pad	Thermal pad	oversize: 10.000	mils
	10.000	Spoke width:	10.000	- ile

After mapping, the Command window will show the translation progress. Once the translation finishes, the translated **.BRD** file will be accessible in the folder specified in the Output Design field.

Note: In some cases, especially for larger boards, it might seem like PCB Editor has stopped working. Do not close it as tests on large boards have shown translation times taking over five minutes, although this is unusual.

Note: Make sure to check the log file after the translation completes. The log file will contain all the text shown in the Command window once the translation completes; this file can be found in the root folder as the translated **.BRD file**.

The translated PCB layout will appear in the main editor window inside the new drawing that was created in Step 2. Save the **.BRD file** before continuing.



Synchronize the Schematic to the Migrated PCB

Step 1 - Creating a Netlist in OrCAD X Capture

From OrCAD X Capture, select "PCB » Update Layout".



Select the translated PADS board file "OK" to create the netlist and accept saving the project and creating an allegro subdirectory.

New Layout		×
Create New Layout and Associate in Pro	oject	
PCB Layout Folder	allegro	
Input Board File		
Board	allegro\bigeasydriver_v16a.brd	
	Ok Cancel He	lp

Note: Once translated, the OrCAD X board database will have _allegro in the file name differentiating it from the PADS board database.



Step 2 - Check Design Status

Check the Properties Panel for the design status of the project. There may be Unrouted Nets or Connections.



Step 3 - Check Physical and Spacing DRC Constraints

From OrCAD X Capture, start the Constraint Manager from the **"PCB » Constraint Manager"** menu item. Check and verify the physical and spacing DRC rules.

Update the online DRC using the toolbar icon.

Step 5 - Change to Preferred Colors

Changing the colors to suit your preferences is easy. Select the color and visibility toolbar icon and change your color preferences accordingly.

Importing Your PADS PCB Libraries into OrCAD X PCB Editor

Invoke the library translator from within OrCAD X Presto PCB Editor using the **"Import » Translators » PADS Library..."** command from the top menu.

Fil	e Edit	View	Setup	Tools	ECO	Manufa	acturing	Reports	Help
	New					۲			
	Open				(Ctrl+O			
	Open Rec	ent				۱.			
	Open Des	ign Direc	tory						
	Symphony	y Connec	t						
	Save BigE	asyDriver	_v16a_alle	egro.brd	(Ctrl+S			
P	Save As								
	Import					•	PADS		
	3D Export						PADS	Library	
	Reset Defa	ault Licer	ise				Altiun	n PCB	
	IP Protect	ion Locki	ng				Eagle	PCB	
X	Exit					Alt+F4			

🞯 OrCAD X PCB Professional

Use the OrCAD X library translator to browse to the folder where your PADS package and decal libraries are located, and specify your desired output folder for the migration. Type in the file name for the options file you wish to create, select the **"Show Options Dialog"**, and then **"Translate"**:

Ø PADS LIBRARY TRANSLATOR − □				
PADS Library Directory:				
Options File:				
Output Directory:				
Show options dialog				

Select the options below:

PADS Layout Library to Allegro Translator Options

PADS to Allegro layer mapping								
COPPER DECAL PAD								
Layer Class Subclass								
0 PACKAGE GEOMET SILKSCREEN_TOP 1 PACKAGE GEOMET SILKSCREEN_TOP 20 UNUSED - 27 UNUSED -								
Class: V Sub Class: V								
Create package symbol files Create drawing files for custom padstacks								
Automatica ender (aaste laver graatien								
Create solder/paste layers Solder oversize: 0.050 mils Paste oversize: 0.000 mils								
Through pin thermal/anti pad creation Thermal pad oversize: 0.250 mils								
Antipad oversize: 0.250 mils Spoke width: 0.350 m	112							
OK								

The OrCAD X Presto PCB Editor library footprints and padstacks will be available in the specified folder as .DRA files.

Next Steps

Now that you know how easy it is to move to OrCAD X Presto PCB Designer, are you ready to learn more about the exciting OrCAD features and technologies which will help you improve your design productivity? Here are some resources you can leverage to learn more about OrCAD X technologies.

What's New in OrCAD X

Want to know what are the new features in the latest OrCAD X release? Check out what's new.

Customer Testimonials

See how companies leverage OrCAD X to bring their products to market on time and budget. Read OrCAD X customer stories.

Product Information

Need more videos, application notes, or datasheets to dive deeper into the OrCAD X technologies? <u>View OrCAD X product</u> pages.

If you have any questions about migration or OrCAD X, please do not hesitate to contact your local Cadence Channel Partner at https://www.cadence.com/en_US/home/alliances/channel-partner.html

cādence[°]

Cadence is a pivotal leader in electronic design and computational expertise, using their Intelligent System Design Strategy to turn design concepts into reality. Cadence customers are the world's most creative and innovative companies, delivering extraordinary electronic products from chips to boards to systems in the most dynamic market applications. www.cadence.com

© 2020 Cadence Design Systems, Inc. All rights reserved worldwide, Cadence, the Cadence logo, and the other Cadence marks found at www.cadence.com/go/trademarks are trademarks or registered trademarks of Cadence Design Systems, Inc. All other trademarks are the property of their respective owners. 07/24 DB/CPG/MG-ORCD-PDS/PDF