MIGRATION GUIDE

cādence°



Migrating from Altium to **OrCAD X**

Contents

Overview



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:

- > Do 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 the 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 fully 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 Altium Designer 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!

Translating Altium Designer .SchDoc Files to OrCAD X Capture

Before you start translating your Altium Designer schematic data into OrCAD X Capture format, the schematic must be saved in ASCII format. This can be done with the Save As... command in Altium Designer. This will replace the original binary file by its ASCII equivalent. The file extension will stay the same.

Save As						Х
← → · ↑	> This PC > C: > Users >		~	S (Search Users	
Organize 🔻 🛛 Ne	ew folder				=== -	?
💻 This PC	Name	Date modified	Туре	Size		
3D Objects	All Users	12/7/2019 1:30 AM	File folder			
Deckton	Default	8/7/2021 10:58 AM	File folder			
Desktop	Default User	12/7/2019 1:30 AM	File folder			
	Public	8/15/2021 8:50 PM	File folder			
Downloads						
Music						
Pictures						
Videos						
E C:						
i Network						
	*					
File name:	Sheet1.SchDoc					\sim
Save as type:	Advanced Schematic binary (*.SchDoc)					~
	Advanced Schematic binary (*.SchDoc)					
∧ Hide Folders	Advanced Schematic ascii (*.SchDoc) Schematic binary 4.0 (*.sch)					
	Advanced Schematic template (*.SchDot)					

Project Structure

Your Altium Designer schematic files can be translated only if they are embedded in a PCB project **(*.PrjPCB)**, which manages the design documents needed to manufacture a PCB design. A valid project structure file (*.**PrjPCBStructure**) is also required. In older versions of Altium Designer, this structure file is generated after compiling the PCB project and references the individual schematic sheets (*.**SchDoc**).

Name	Туре	Size
History	File folder	
📾 Migrate.PrjPcb	Altium PCB Project	36 KB
Migrate.PrjPcbStructure	PRJPCBSTRUCTURE File	1 KB
E PCB1.PcbDoc	Altium PCB Document	155 KB
E Sheet1.SchDoc	Altium Schematic Document	215 KB
Sheet2.SchDoc	Altium Schematic Document	486 KB

Each schematic page ***.SchDoc** is saved in ASCII format.

Project file *.**PrjPCB** acts as project master.

File *. PrjPCBStructure has references to schematic pages (relative path).

File *.PcbDoc is only needed when PCB has to be translated.

The .**PrjPCBStructure** file can be generated in two different ways depending on your version of Altium Designer.

Altium Designer 19 and Earlier

If your .PrjPCBStructure file does not exist, follow these steps to create the file:

Select "File » New » Project" from the menu, the New Project dialog will open. From the list of available project types, choose PCB Project.

Add the schematic documents (*.SchDoc) to the project

Compile the project (e.g. "RMB » Compile PCB Project")

Altium Designer 20 and Later

Starting with Altium Designer 20, all .**PrjPCBStructure** files are generated automatically at certain points in the design process. You can force creation of a .PrjPCBStructure file with these steps:

Select "File » New » Project" from the menu, the New Project dialog will open. From the list of available project types, choose PCB Project.

Add the schematic documents (*.SchDoc) to the project

Save the project (e.g. "RMB » Save")

Close the project and navigate to the folder containing the project

As soon as you close the project, Altium Designer will create your .PrjPCBStructure file.

For more information about setting up PCB projects, refer to the documentation for your version of Altium Designer.

Import Your Altium Designer Schematic Data to OrCAD X Capture

Step 1 - Import the Altium Schematic

In OrCAD X Capture, Click on "File » Import » Altium Schematic Translator" to launch the Altium-Capture translator. Browse to the ACSII file to be translated (*.PrjPcb) and specify the output directory for the OrCAD X Capture project.

Altium Schematic Translator		×
PrjPCB File:	C:/Users/Migrate/Migrate.PrjPcb	
Output Directory:	C:/Users/Migrate/out	
Frame Size:	CUSTOM	•
Library Optimization:	Create Individuall Symbol Definitions	•
Translate	Viewlog Close	Help

There are two options in this dialog:

Frame Size - This will define the sheet size and layout in the translated schematic sheets. Standard ANSI and ISO sheet sizes can be selected from this dropdown menu.

Library Optimization - When the "Create Individual Symbol Definitions" box is checked, the translator will generate a separate symbol definition for each instance of a component by adding a suffix. For example 0805_1, 0805_2, 0805_3 and so on. This is to account for instance-specific footprint modifications.

Once the appropriate options are selected and the destination folder is specified, click the Translate button to begin the migration process. The translator will convert the set of schematic sheets into a .**DSN** file that can be opened in OrCAD X Capture.

Once finished, open the .DSN file by adding it to a new project, and check the translated schematic for errors.

Import Your Altium Designer Board Data to OrCAD X Presto PCB Editor

Step 1 - Running the Altium PCB Translator

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



Browse to the ASCII *. PcbDoc file to be translated; see the note below on creating individual symbol definitions.

Altium PCB Translator			-		×
ASCII PcbDoc File:]
 Create individual sym Cleanup dangling cling Skip shape based team 	nbol definitions nes rdrops				
Translate shapes	Static	-	~		
Pad touch connections	Connect by cline	-	Min width	6.00	
Translate View	log Close	ļ		Help	

Note: When the "Create Individual Symbol Definitions" box is checked, the translator will generate a separate symbol definition for each instance of a component by adding a suffix. For example 0805_1, 0805_2, 0805_3 and so on. This is to account for instance-specific footprint modifications within the Altium Designer design. In OrCAD X Presto PCB Editor, separate symbol definitions are beneficial when libraries are exported to disk.

By default this option is unchecked, which means that the translator will create one symbol definition only for a given Altium Designer footprint. Use this option only if instance-specific changes have been made in Altium .**PcbDoc** file.

Click "Translate" to begin the conversion process. The Command window will show the translation progress. Once the translation finishes, the translated .**BRD** file will be accessible in the "out" folder, which will be located in as the original Altium .**PcbDoc** file. **Note**: In some cases, especially for larger boards, it might seem like the 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. Save the .**BRD** file before continuing.



STEP 2 (Optional) - Translating the Altium Symbols into an OrCAD X Library

In OrCAD X Capture, Navigate to the Project Hierarchy, under the Design File Folder, there is a Design Cache. This contains one instance of every schematic symbol used in this design. To create a Library for OrCAD X using these files. Navigate to **File > New > Library**

	PCB									
-	Fie therarchy									
	Page1									
	🔳 📻 [23] - SENSORS, CAN, IR									
	Page1									
	E - F [24] - HEADERS, UART									
	Page1									
	[25] - LEDS, BUTTONS								A CONTRACTOR OF	
	Page1		OrCAD X Cap	oture Cl	S-[/Desi	gnator_	10 - ([03]	- CPU	 DDR3, DDR3 	MEM :
			_			-				
	Pagel	File	Design	Edit	View	Tools	Place	PCB	SI Analysis	PSnice
	Page1	1 115	besign	Luit	vicuv	10013	Thatee	100	STAnalysis	1 Spice
	E [28] - PWR IN MECH DOC	-							D	
	Page1		New						Project	Ľ
	E [29] - POWER SEQUENCING									1
	Page1		Open						Design	
	= I [30] - REVISION HISTORY	d								\
	Page1		Close			0	trl+F4		Nhran/	
	🚊 🔚 Design Carche		Close				411-14		and by	
	0471554001_J8 : C:_PE\ORCADX_INTRO_MOVIES\/								1000 51-	
	especial and the state of the second and the state of the second		Sava				Ctrl+S		VHDLFile	
	1 84053 5 133, CA RE ORCADY, INTRO, MOVIES		Save				Cuita			
	2N7002BKS 115 O11 · C1 PE\ORCADX INTRO MO								Verilog File	
	2N7002BKS.115 Q8: C:\ PE\ORCADX INTRO MOV		Check and	d Save						
	2N7002BKW,115_Q12 : C:_PE\ORCADX_INTRO_M(Text File	
	2N7002BKW,115_Q13 : C:_PE\ORCADX_INTRO_M(Save As						reactine	
	2N7002BKW,115_Q3 : C:_PE\ORCADX_INTRO_MO								DCalles Liberry	
			Save Proje	of Ac					PSpice Library	
	2N7002BKW,115_Q5 : C:_PE\ORCADX_INTRO_MO		Save Proje	ACC AS				-		
	2N7002BKW,115_Q6: C:_PE\ORCADX_INTRO_MO									
	2N7002BKW 115 09 C1 PE\ORCADX INTRO MO									

Now select all the parts in the design cache, then **Copy** and **Paste** them into your new Library.





Synchronize the Schematic to the Migrated PCB

Step 1 - The Design Sync Setup

In OrCAD X Capture, click on "PCB » Design Sync Setup"



If you are Exporting to OrCAD X Presto PCB Editor, Select **OrCAD X Presto** from the Select Layout Tool Dropdown. What this means is that if you select the open layout option, the OrCAD X Presto PCB Editor will open.

Design Sync Setup	×					
✓ Layout Tool and Design Sync Options						
Layout Folder	allegro					
Allow Etch Removal	No 👻					
Create User Defined Properties	No 👻					
Select Layout Tool	OrCAD X Presto					
✓ Placement						
Place Change Component	Always -					
Ignore Fixed Property	No 👻					
✓ Constraints						
Constraints	Changes Only 👻					
Show Difference Report						
✓ PCB Netlist						
Configuration file	C:\Cadence\SPB_23.1					
	Ok Cancel Help					

Select **OK**.

Step 2 - Creating the Netlist in OrCAD X Capture

In OrCAD X Capture, select "**PCB** » **Update Layout**".

PCB	SI Analysis PSpice Accessor	New Lavout		×
	New Layout			
	Update Layout	Create New Layout and Associate in Pr	oject	
	Update Schematic	PCB Layout Folder	allegro	
.	Constraint Manager	Input Board File	-	
	Configure PCB Power Nets	Board	allegrolopenrey v1i1 brd	
₽\$	Design Rules Check		allegiolopentex_vitt.bid	
	Design Sync Setup			
	View Reports		Ok Cancel Help	

In the Update Layout dialogue, chose the export folder, name your board, if you have an Input Board File from which you want to update the netlist

Press **OK**, this will Netlist your design and push the netlist to the layout tool chosen.

Step 3 - Check Design Status

Check and update the status of your design within the **Properties Panel** under the **Status** section.



Step 4 – 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.

The **Status** section in the **Properties Panel** displays all DRC, and you can navigate to specific DRCs from this section.

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.

Next Steps

Now that you know how easy it is to move to OrCAD X, are you ready to learn more about the exciting OrCAD X 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 customer stories.

Product Information

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

If you have any questions about the migration or in general about OrCAD X, please do not hesitate to contact your local Cadence Channel Partner at http://orcad.com/about/contact-us.

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

© 2024 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. 05/24 DB/CPG/MG-ORCD-ALTM/PDF