MIGRATION **GUIDE**

cādence

Migrating from Altium to OrCAD X

Contents

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:

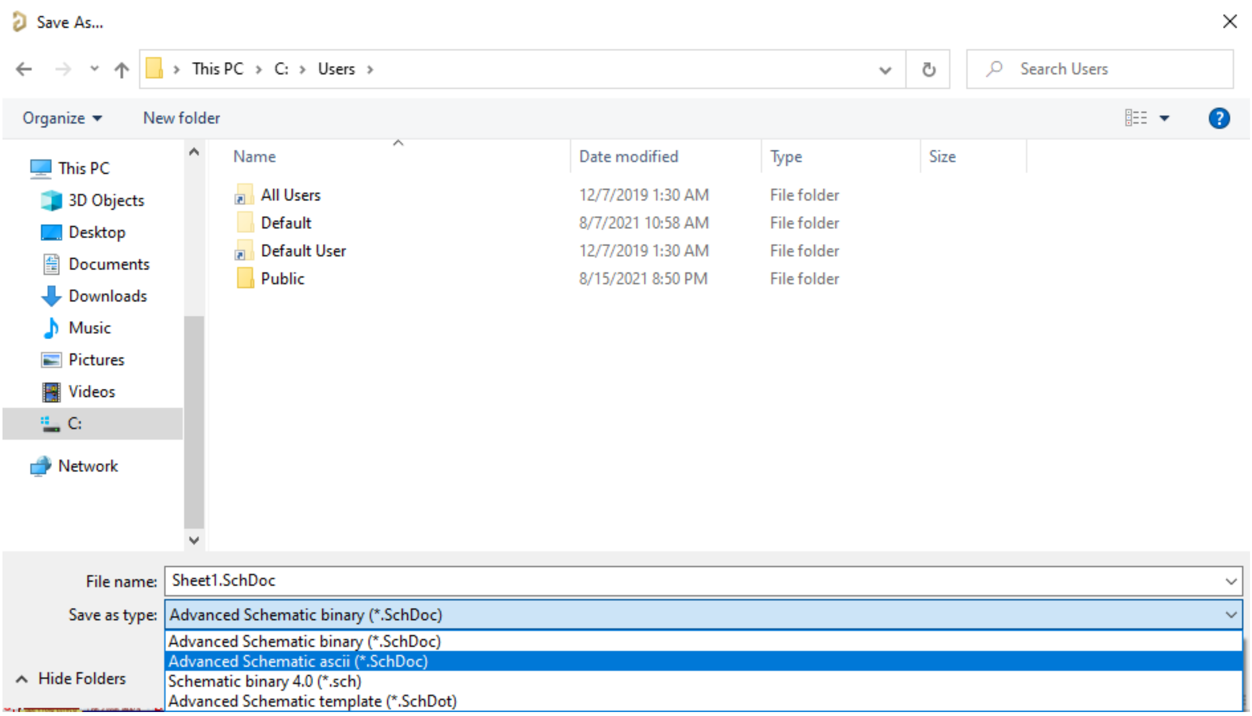
- 30 years of innovation and leadership in the industry
- Affordable price and flexible purchase models
- Cutting-edge technologies
- Ecosystem empowered
- Industry's best customer support

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!

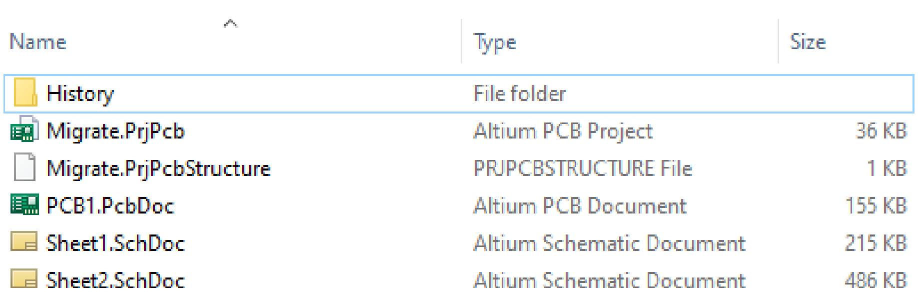
Translating Altium Designer .SchDoc Files to OrCAD X Capture

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.



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**).



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

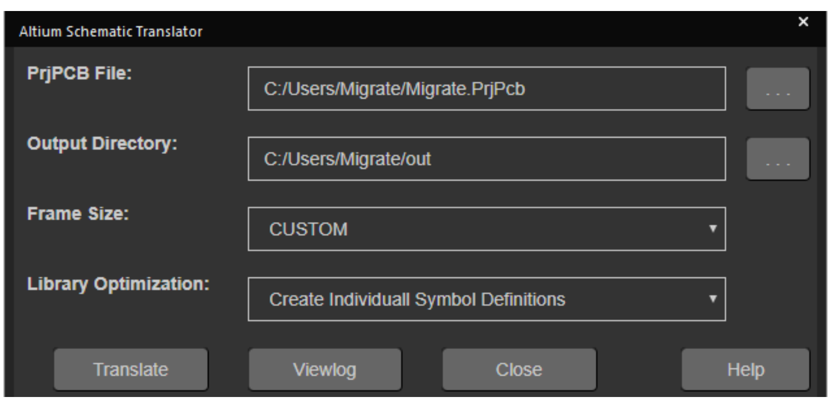
As soon as you close the project, Altium Designer will create your .**PrjPCBStructure** file.

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.



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.

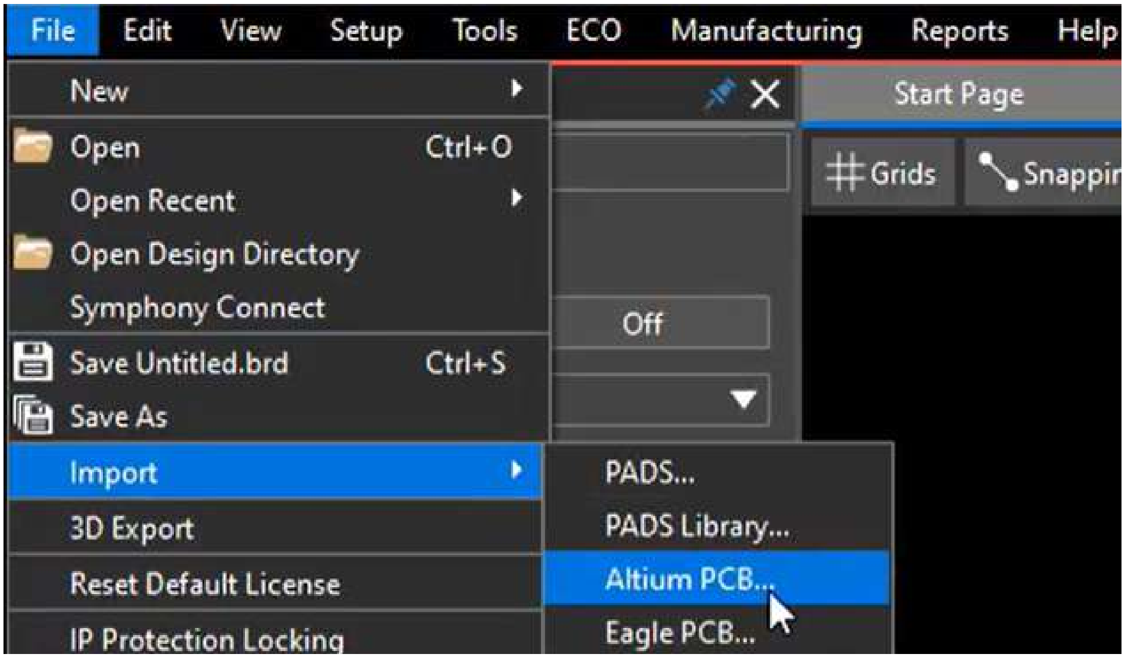
Once the appropriate options are selected and the destination folder is specified, click the Translate button to begin the migration process. The translator will convert the set of schematic sheets into a .**DSN** file that can be opened in OrCAD X Capture.

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

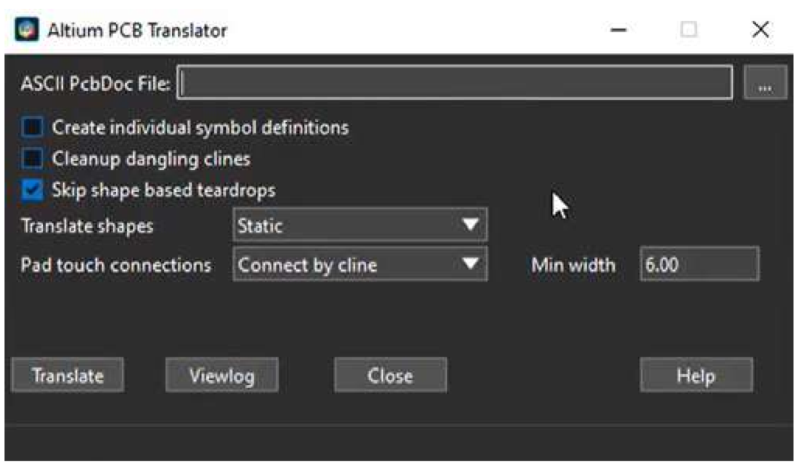
Import Your Altium Designer Board Data to OrCAD X Presto PCB Editor

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.



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.

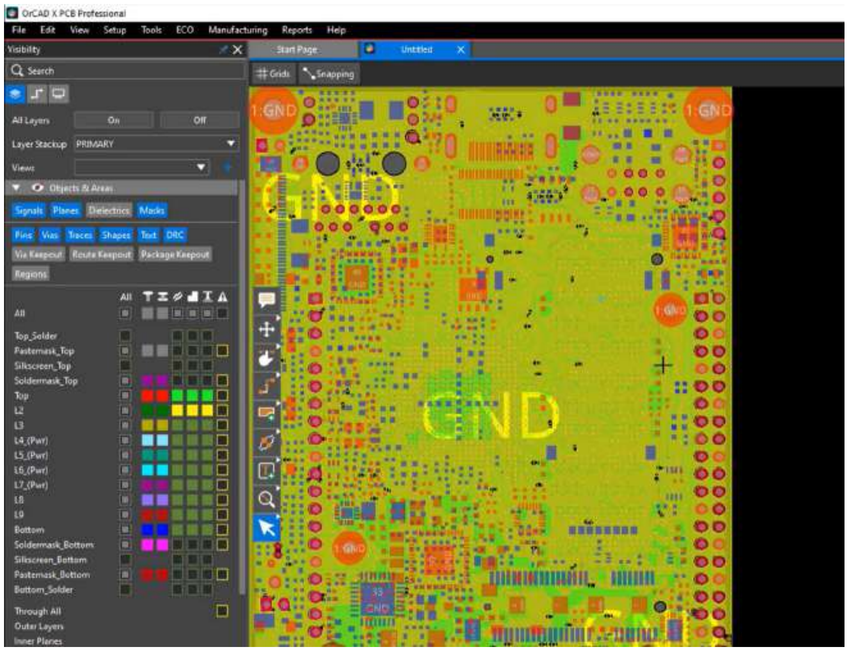
By default this option is unchecked, which means that the translator will create one symbol definition only for a given Altium Designer footprint. Use this option only if instance-specific changes have been made in Altium .**PcbDoc** file.

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.

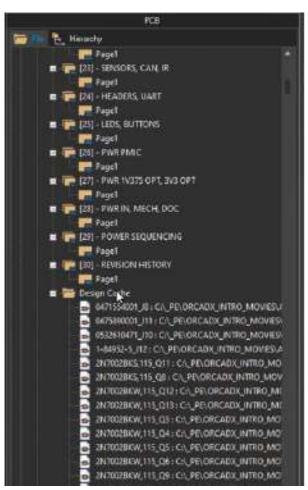
Note: Make sure to check the log file after the translation completes. The log file will contain all the text shown in the Command window once the translation completes; this file can be found in the root folder as the translated .BRD file.

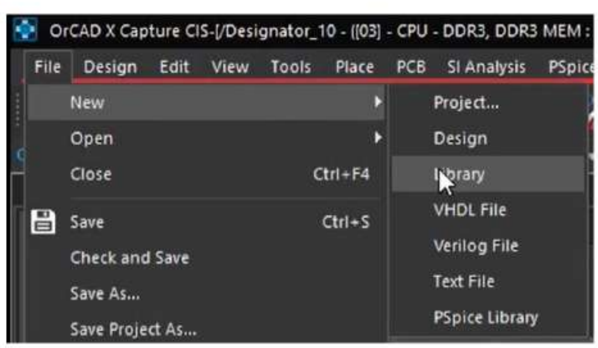
The translated PCB layout will appear in the main editor window inside the new drawing. Save the .**BRD** file before continuing.



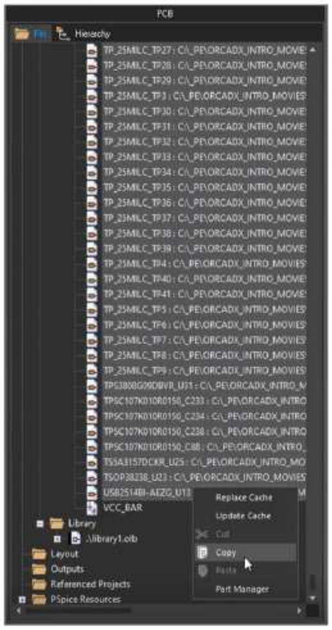
STEP 2 (Optional) – Translating the Altium Symbols into an OrCAD X Library

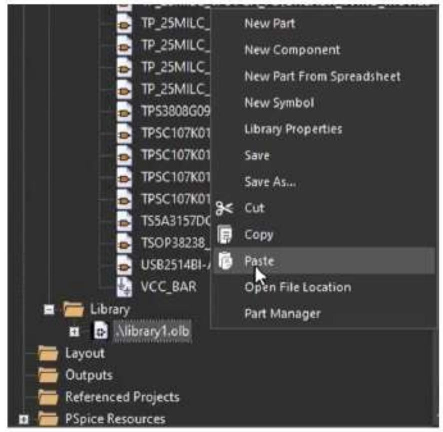
In OrCAD X Capture, Navigate to the Project Hierarchy, under the Design File Folder, there is a Design Cache. This contains one instance of every schematic symbol used in this design. To create a Library for OrCAD X using these files. Navigate to **File » New » Library**





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

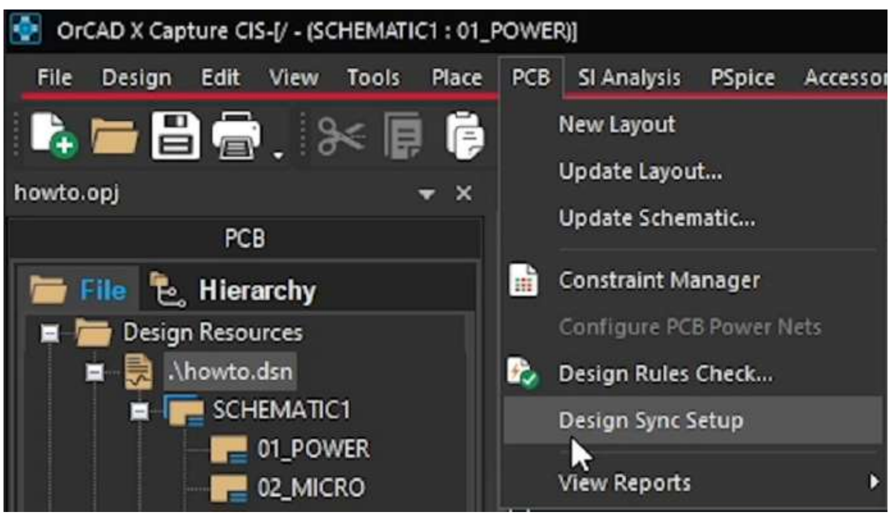




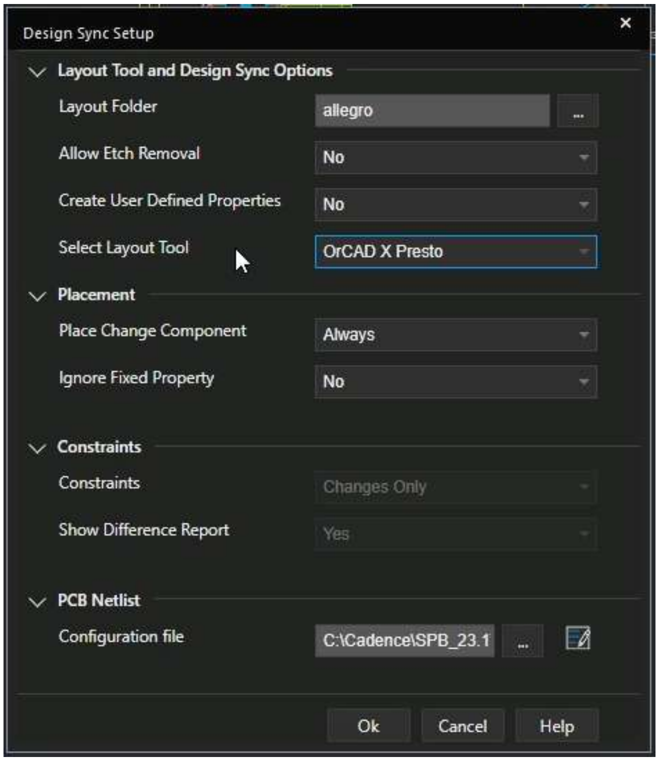
Synchronize the Schematic to the Migrated PCB

Step 1 – The Design Sync Setup

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



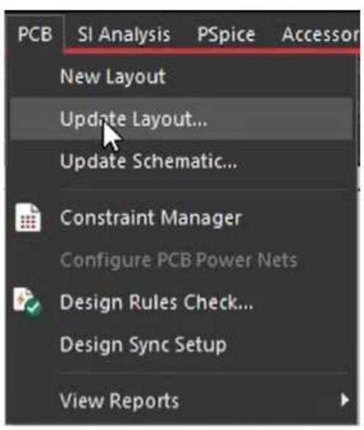
If you are Exporting to OrCAD X Presto PCB Editor, Select **OrCAD X Presto** from the Select Layout Tool Dropdown. What this means is that if you select the open layout option, the OrCAD X Presto PCB Editor will open.

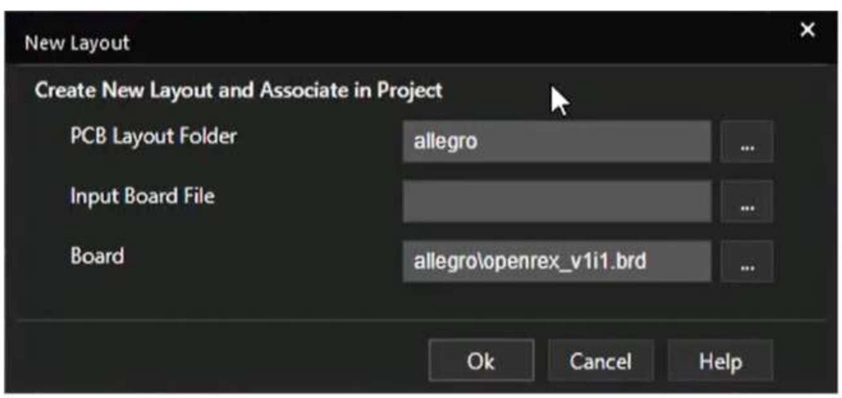


Select **OK**.

Step 2 – Creating the Netlist in OrCAD X Capture

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



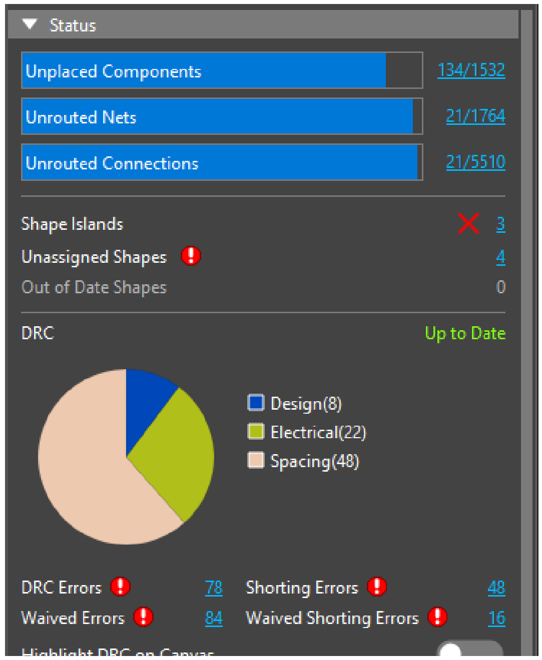


In the Update Layout dialogue, chose the export folder, name your board, if you have an Input Board File from which you want to update the netlist

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**

 05/24 DB/CPG/MG-ORCD-ALTM/PDF