OrCAD™
CADENCE PCB SOLUTIONS

Migration Guide

Migrating from Altium to OrCAD PCB Designer

Table of 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 first product, a large enterprise wanting to improve the productivity and efficiency of your design team, or somewhere in between, selecting a PCB solution can be a daunting task. Nobody wants to get 75% of the way through a design to find that the software you selected is not going to achieve what you need it to do.

There are many aspects you have to consider when choosing your PCB Design solution, including:

• Do the capabilities of the offering and its technology meet your design requirements?

• What's the cost of the software? Does it fit within your budget?

• What's the support service like? Will you be able to get quick responses to your questions and access online tutorials?

• Can its technology and solution 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?

With OrCAD® PCB Design Solutions, you can be confident that you will have the right solution and technologies to meet all of your design challenges today and tomorrow. Here are five of many reasons why:

Reason 1: Security of Scalability for Your Future Design Challenge

Cadence and OrCAD provide the only full scalable PCB design solution on the market that can seamlessly grow with your needs. Invest for the future with a tool that can grow as your business does.

Reason 2: Affordable Price and Flexible Purchase Models

OrCAD PCB design is not a "one-size fits all" enterprise. Unlike other vendors monolithic pricing models, OrCAD provides flexible pricing tiers enabling design teams to access the capabilities they need for the price they can afford.

Reason 3: Cutting-edge Technologies

OrCAD 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.

Reason 4: Fully Customizable and Ecosystem Empowered

Open APIs allow you to extend and customize your design environment to meet your unique needs. Connect to external systems, generate custom reports, even customize the UI. It is all possible with OrCAD.

Reason 5: Industry's Best Customer Support

OrCAD 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 PCB Design Solutions, you may have existing or legacy designs you need to convert or translate into an appropriate design format. The good news is that these OrCAD tools are supplied with an integrated and proven Altium design translator built in.

This guide will walk you through the steps and process involved in getting your design IP into the OrCAD format so you can start realizing the advantages of moving to OrCAD!

Prerequisites of Translating Altium Designer Schematic Data to OrCAD Capture Format Data

STEP 1 - Prepare ASCII Schematic

Before you start translating your Altium Designer schematic data into OrCAD Capture format, the schematic has to be saved to ASCII format within Altium Designer. This will replace the original binary file by its ASCII equivalent. The location and file extension stay the same.
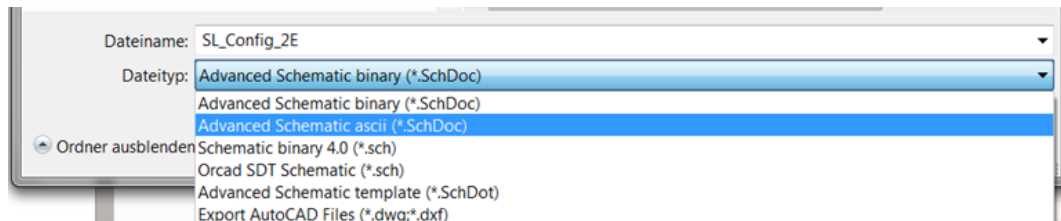
Your Altium Designer schematic data can be translated only if they are embedded in a PCB project (***.PrjPCB**), which manages the design documents needed to manufacture a PCB design.

A valid structure file (***.PrjPCBStructure**) is needed. It will be generated after compiling the PCB project and references the individual schematic pages (***.SchDoc**).

If the project file does not exist, you can follow the steps below:

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

2. Add the schematic documents ***.SchDoc** to the project

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

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

Project Structure example

- 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.

# Import your Altium Designer Schematic Data to OrCAD Capture

STEP 1 - Run the Altium Designer schematic translator

In OrCAD Capture, Click **File >> Import Altium Schematic** to launch Altium-Capture translator.

Browse to the ACSII file to be translated (***.prjPCB**) and specify the output directory for the OrCAD Capture project. Click on Translate button.
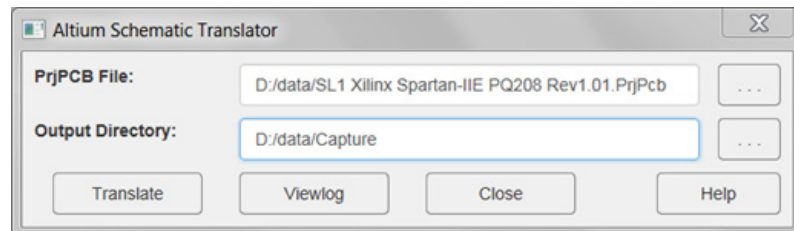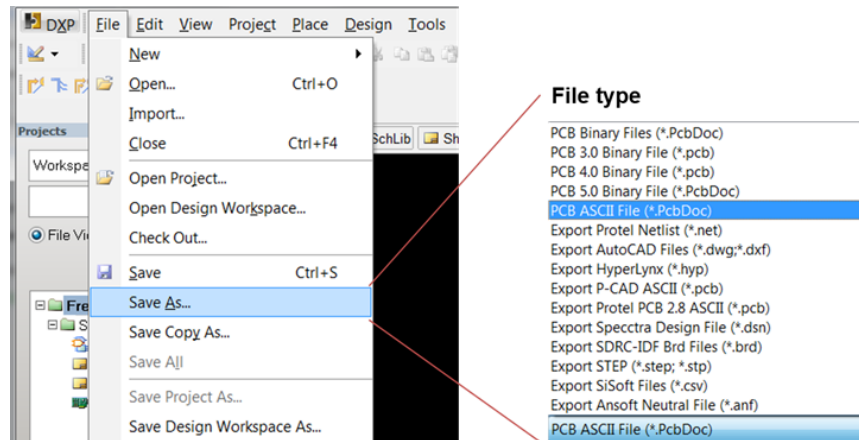

Once finished, open the project and check schematic for translation errors.
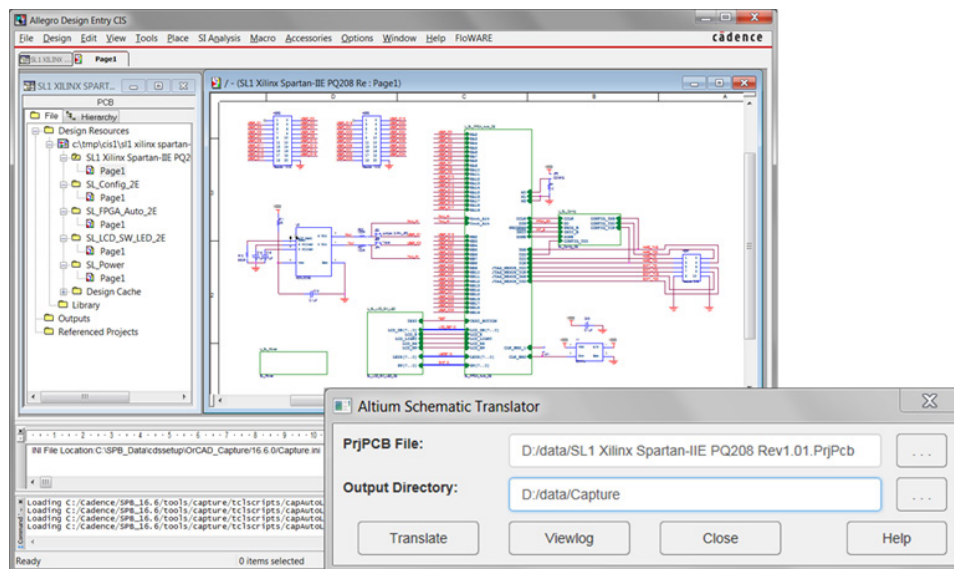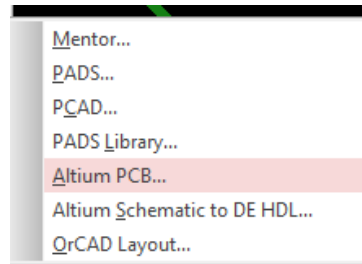
Import Your Altium PCB Design into OrCAD PCB Editor

STEP 1 - Preparation

Before you start translating your Altium PCB design data into OrCAD PCB Editor, PCB design data has to be saved as a PCB ASCII File (***.PcbDoc**) within Altium PCB Designer.
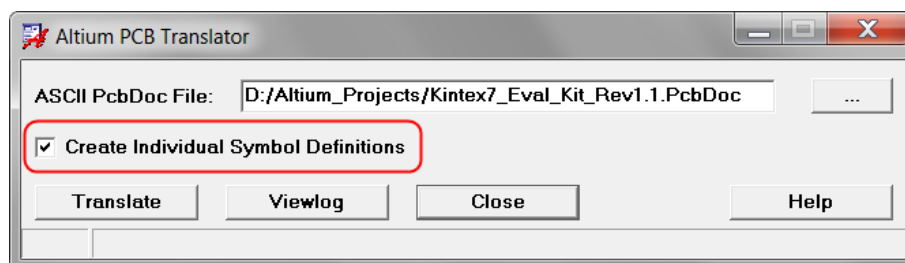

STEP 2 - Running the Altium PCB Translator

In OrCAD PCB Editor, under the file menu, choose **Import >> CAD Translators >>Altium PCB**.

Browse to the ASCII *.PcbDoc Altium Designer design file to be translated; see 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 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 the Altium tool design.

Synchronize the schematic to the Migrated PCB

If you plan to translate a complete Altium Designer project, including both schematic and PCB, after you have translated the schematic into OrCAD Capture, run **Tools >> Create >> Netlist** and specify an empty database. Then open this database with OrCAD PCB Editor.

Within OrCAD PCB Editor run **Altium PCB translator** and navigate again to the ***.prjPCB** file. The translator will retain netlist and device logic and translate only the remaining data in order to complete the board.

Note: In order to get the right design hierarchy structure in translated designs, it is mandatory to provide a ***.PrjPcbStructure** file along with ***.PrjPcb**.

If this file is not supplied, the translator does not know which is the top/root drawing in your project. If the case structure file is not present, you can set the root drawing in the translated schematic before creating a netlist.
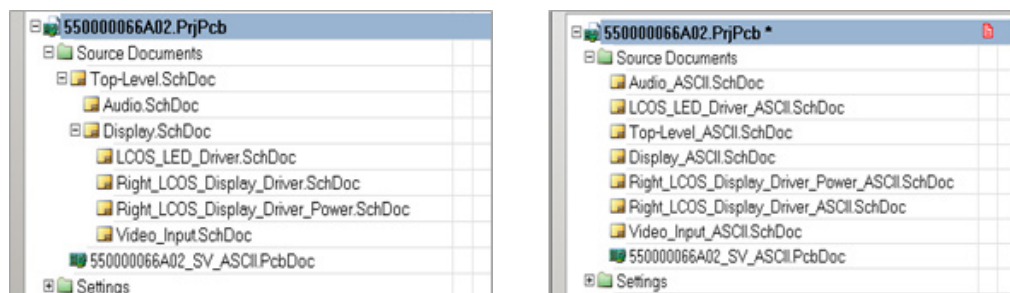
Next Steps

Now that you know how easy it is to move to OrCAD 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 technologies.

What's New in OrCAD

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

Customer Testimonials

See how real companies are leveraging OrCAD to bring their products to market on time and on budget. View OrCAD customer stories.

Product Information

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

Ready to Talk?

The OrCAD product line is supported by a worldwide network of Cadence Channel Partners (CCP). Would you like to talk with our channel partners? Contact us now!

The OrCAD product line is owned by Cadence Design Systems, Inc.