MIGRATION GUIDE

Cadence OrCAD Schematic and PCB to Altium Designer Develop

Contents

Making a Smooth Transition from OrCAD Capture to Altium Designer Develop

In this guide, we'll focus on importing OrCAD Capture designs. Please refer to specific guides if you need to migrate other files.

Legacy Translation

Before You Begin

This guide will show you how to import data from your legacy system into Altium Designer Develop. Before importing all of your legacy data, it's important to ask yourself, "why do I need to import my legacy data?" Below are some of the most common considerations to consider when deciding whether to import legacy data into Altium Designer Develop.

"

We have 20 years of data, and I don't want to leave it behind.

- Does the legacy data include critical information for creating robust designs within Altium?

- Is it easier to start new designs rather than re-work the imported data to make it usable?

"

We have some 'golden' designs that we need to bring into Altium Designer Develop so that we can up-issue them.

- How do you plan to verify the imported data?

- How much re-work is going to be needed after importing?

- Do you need to take advantage of additional Altium Designer Develop features once the data is imported?

"

Our service bureau doesn't use Altium, but they can import Altium data. Our Altium system can import their data. Is this a reliable way to pass designs between our companies?

- How do you maintain library integrity?

- Who has design authority?

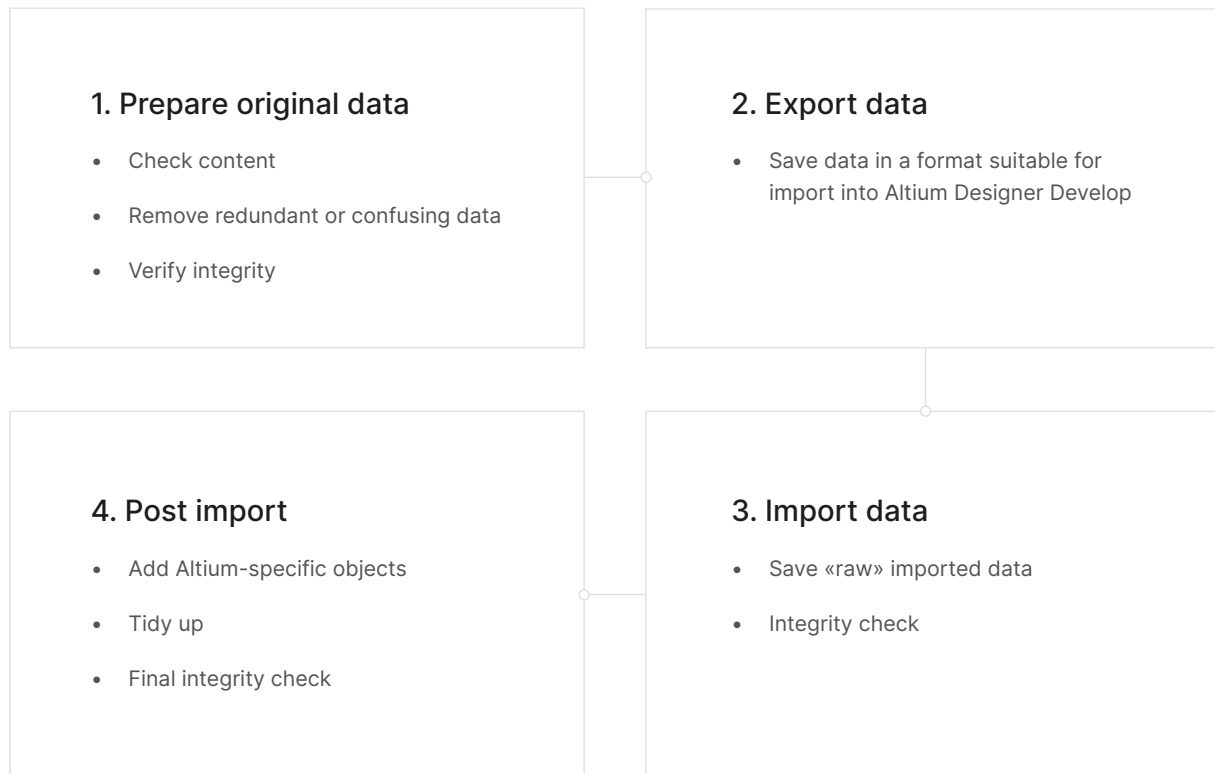- How do you plan to verify the data?

"

We have a library of trusted parts which we'd like to bring into Altium.

- Are there any exotic parts that you may have trouble representing within Altium?

- Do you need to add Altium-specific features (like 3D models) for every part?

- How much redundancy, duplication, and error is present in your library?

After reviewing these considerations, read on to learn how to import your legacy data into Altium Designer Develop.

Altium

Preparing to Migrate Your Legacy Data

There are four phases to migrating legacy data.

1. Prepare original data

- Check content
- Remove redundant or confusing data
- Verify integrity

2. Export data

- Save data in a format suitable for import into Altium Designer Develop

4. Post import

- Add Altium-specific objects
- Tidy up
- Final integrity check

3. Import data

- Save «raw» imported data
- Integrity check

Step 1: Prepare Original Data

It is prudent to clean up your design before attempting to export. Below is a checklist of data cleanup best practices to help you as you prepare your data:

Schematic considerations:

- Individual pin connectors (example: Block connectors) in Capture must be modified at the Capture tool level before migration. Alternatively, you can replace pin connectors in Altium Designer Develop post-migration.
- Connectors should be represented as one-gate-per-pin with over 256 "gates."
- Ambiguous connectivity for reuse blocks must be broken apart in Capture.
- Remove hidden pins or implicit connections.
- Local net names placed at wire intersection must be placed at correct locations (away from intersection point)
- Are schematic symbols mapped to correct PCB footprints or not?
- Schematic and PCB are in sync?

Library considerations:

- Do the schematic symbols match with PCB footprints?
- Does the library contain correct supply chain information and BoM parameters?
- Does 3D information need to be imported? If yes, then the height attribute should be assigned in OrCAD Capture.

Altium

Step 2: Save Data in a Suitable Format

Supported Version and File Formats

The following table shows Capture PCB Design Formats and versions that can migrate into Altium Designer Develop. This list is updated regularly, so please check with us before starting a new migration. Refer to this link for updated information.

| TYPE	SYSTEM	VERSION	FORMAT
Schematic	OrCAD Capture	17.4	ASCII (.dsn)
Schematic Library	OrCAD CIS	17.4	ASCII (.olb)
Legacy PCB	OrCAD Layout	V9.2.3	ASCII (.max)

The following file types are supported:

- OrCAD Capture (*.DSN) files

- OrCAD Schematic Library (*.OLB) files

- OrCAD Layout (*.MAX) files

## Phase 3: Import Data into Altium Designer Develop

Using the Import Wizard for OrCAD Files

Launch the Import Wizard from the Altium Designer Develop File menu. Choose the **OrCAD Designs and Libraries Files** option, as shown below. On the "Importing OrCAD Designs" screen, click the Add button to choose OrCAD design files. You can translate multiple files at the same time. Step-by-step instructions on using the Import Wizard are to follow.
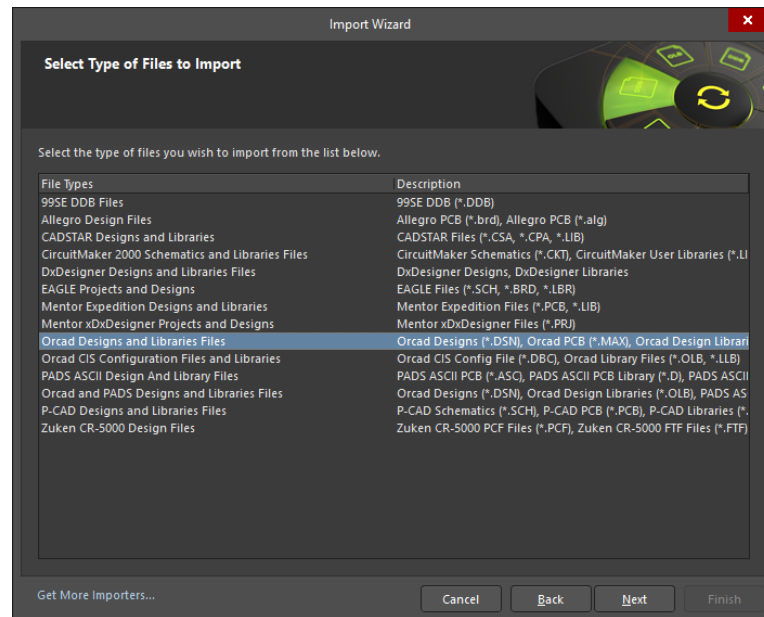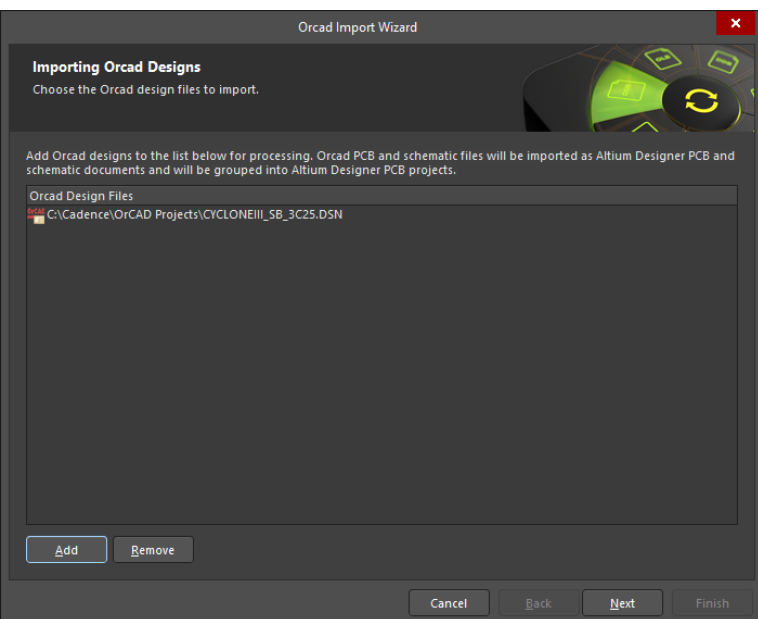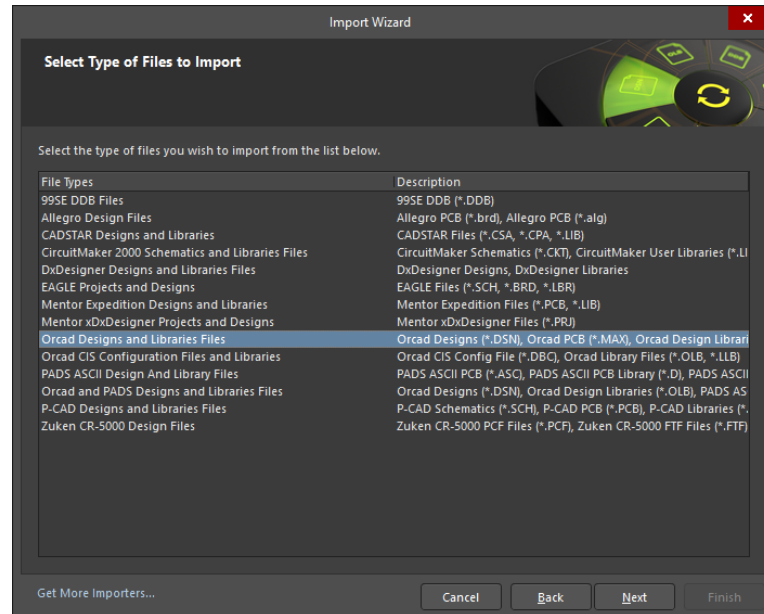

Starting the Import Wizard for OrCAD files

Step-by-Step Import Instructions on Importing an OrCAD Design file (.DSN)
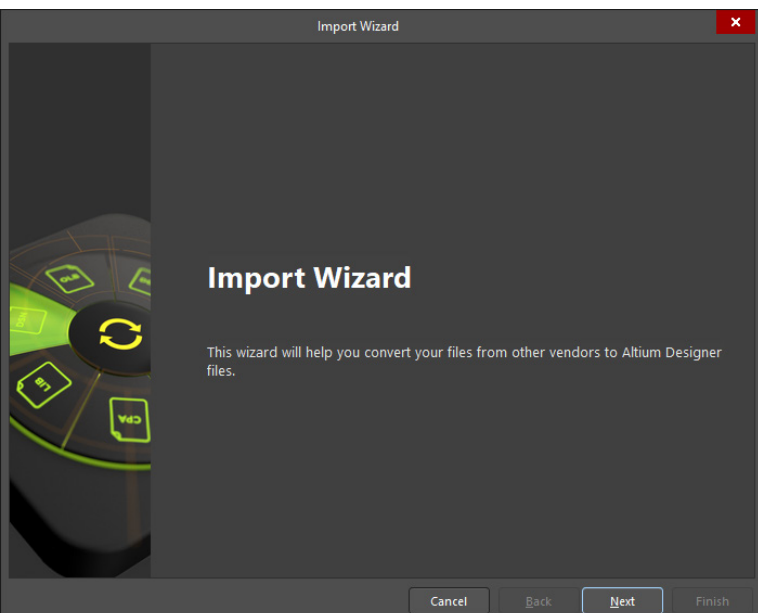
- Start the Import Wizard with **File » Import Wizard**

- Select Type of Files to Import→**OrCAD Designs and Libraries Files**

- Add the file(s) to be translated. In this case, the "CYCLONEIII_SB_3C25.DSN" file has been used.
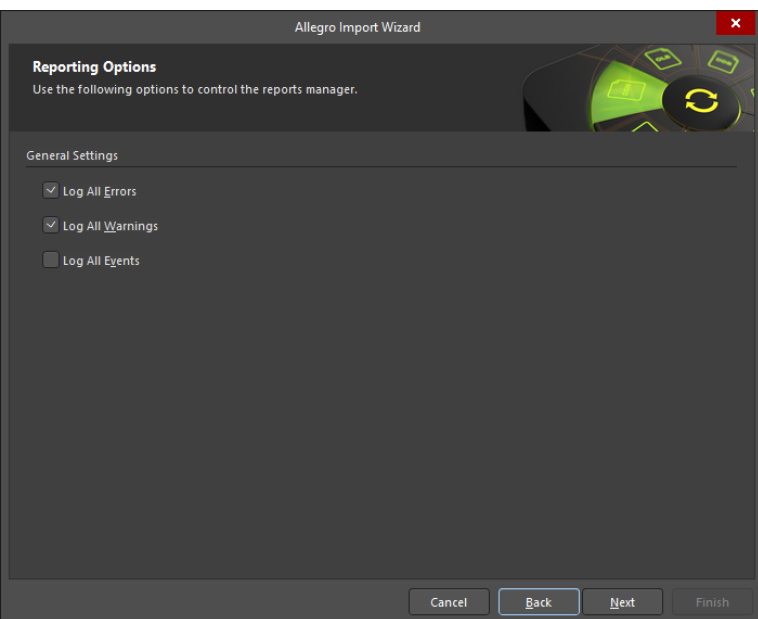

NOTE:

You can add as many OrCAD Design files as you like at this point. However, if you add files of a different name, Altium Designer Develop will create separate projects.
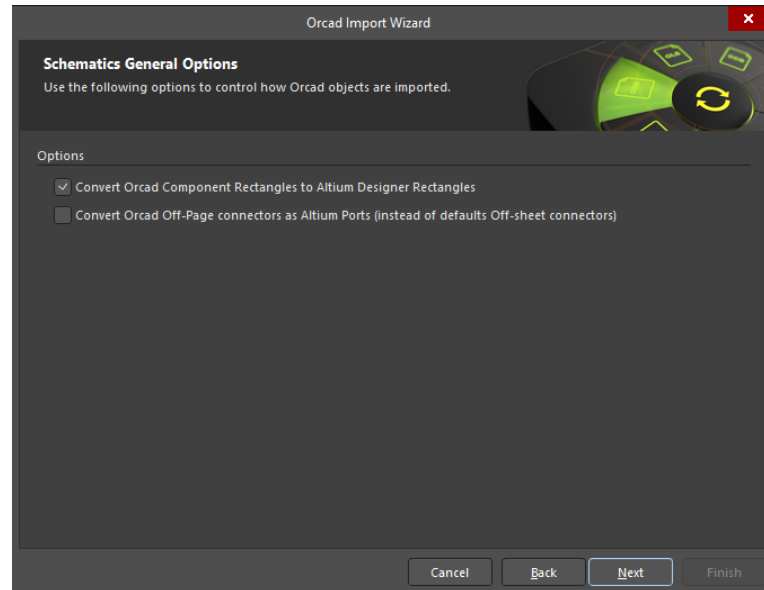
Add any Schematic or PCB Libraries to import (if available).
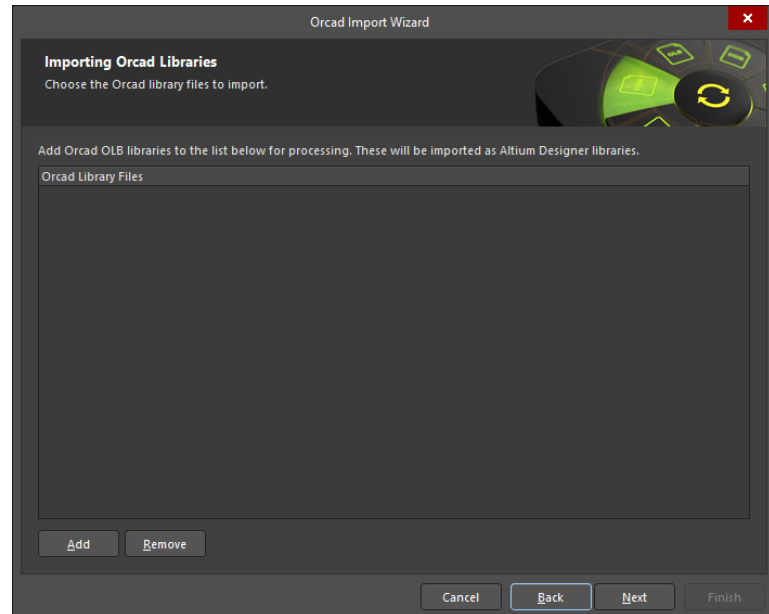
- Set the options for what level of reporting is done after the translation has completed.

- Select whether you want to convert OrCAD component rectangles to Altium Designer Develop rectangles and use Off-Page connectors of OrCAD as Altium Ports.

- Set your desired Schematic Sheet options, including how Title Blocks are imported and Pin-to-Pin spacing options.
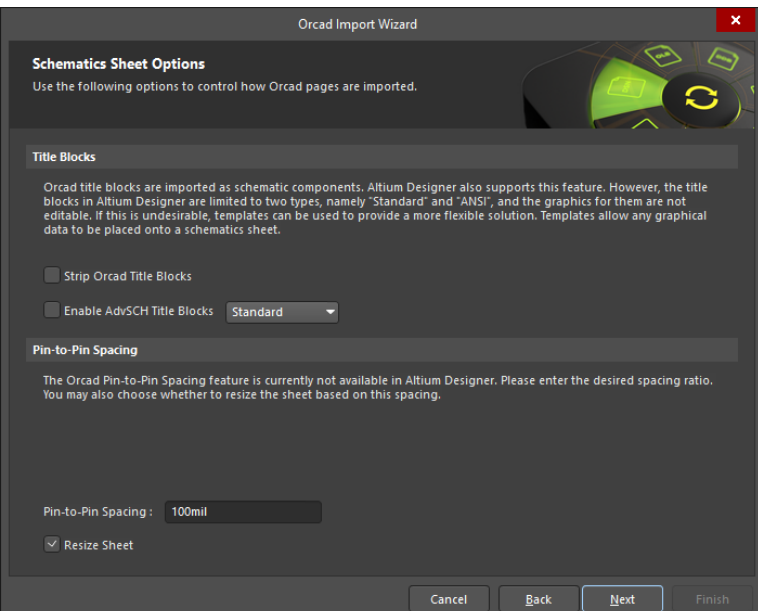
- Set your Schematics Parameter and Net Connectivity Options. Here you can choose to Auto-position the Parameters, and also Disable "Mark Manual Parameters".
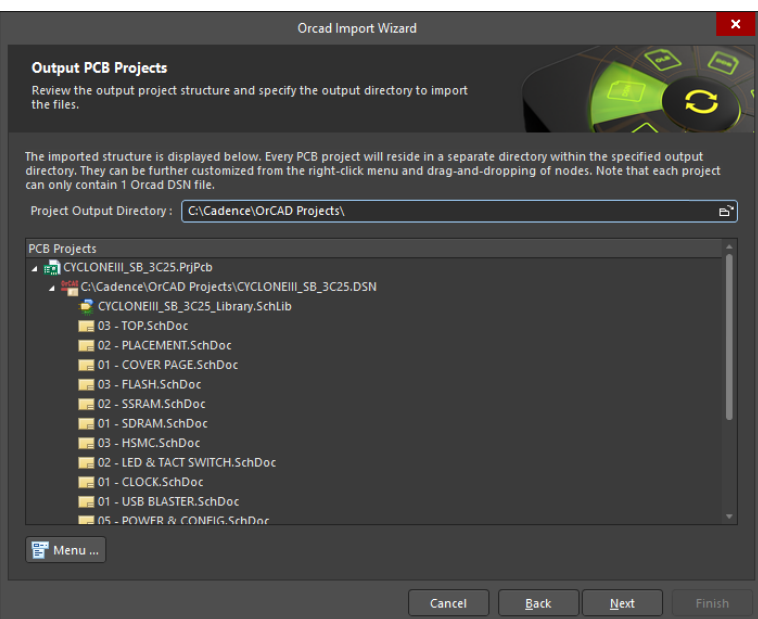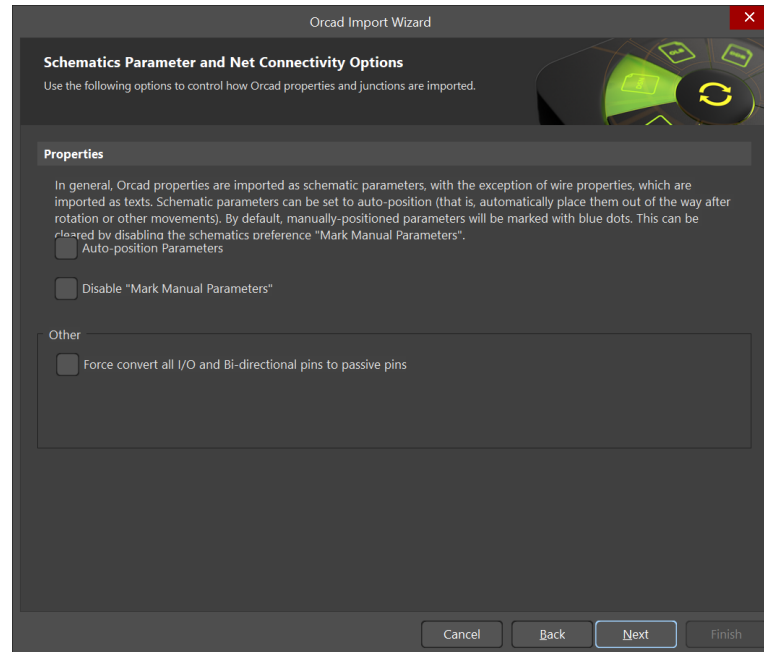
- You will see a preview of the files being translated and their output directories. You can change the main output directory if desired.

- Click the final Next button. The Import Wizard will take care of the rest.

- Congratulations, your design has now been imported into Altium Designer Develop!

- Follow the Post Import Tidy Up checklist to ensure the design has been fully reviewed and verified.
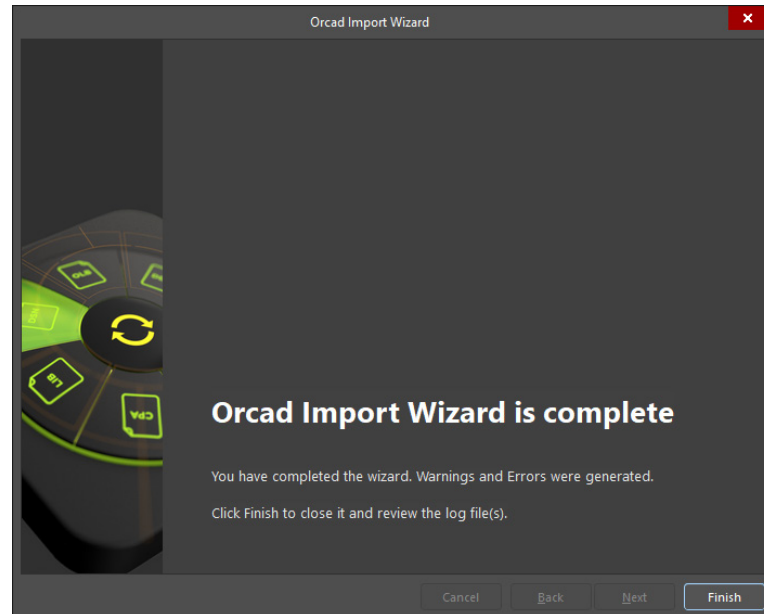
Step 4: Post Import Tidy Up

Once your migration is complete, we recommend checking your design to ensure that all data is transferred as expected. Below is a list of key checks to perform post-migration:

Physical check

- View » Fit Document

- Board shape and cutouts

Electrical check

- Netlist

Rules

- Have all rules been imported

- DRC check

- Check settings for polygons - Island removal, min primitive size

- Thermal reliefs, direct connect

- Check power plane settings

- Power plane Pull-back

- Solder mask, Paste mask rules

- Via Tenting

- Testpoint assignments

Power check

- Nets

- Planes

- Polygons

Documentation check

- Layers

- Text/Strings

- Legends

PCB reports

- Number of components/nets

- All nets routed

Making a Smooth Transition from Cadence Allegro/OrCAD PCB Editor to Altium Designer Develop

In this guide, we'll focus on importing Allegro/OrCAD Layout PCB designs. Please refer to specific guides if you need to migrate other files.

Step 1: Prepare Original Data

It is prudent to clean up your design before attempting to export. Below is a checklist of data cleanup best practices to help you as you prepare your data:

PCB considerations:

- Large numbers of graphical objects like mechanical drawing or non-ECO-registered drawing primitives must be migrated on documentation layers.

- Star point grounding check.

- Remove deliberate DRC violations.

- Remove objects extending beyond the environment.

- Known PCB layer assignments need to map correctly with existing Altium Designer Develop PCB, especially with the power plane or signal layer.

- Do the auto-named nets match with the schematic?

Library considerations:

- Does 3D information need to be imported? If yes, then the height attribute should be assigned in Allegro.

- Correct representation of custom Allegro, copper shapes, solder mask, and resist?

Step 2: Save Data in a Suitable Format

Supported Version and File Formats

The following table shows the versions of all Allegro design formats that can be migrated into Altium Designer Develop. This list is updated regularly, so please check with us before starting a new migration. Refer to this link for updated information.

TYPE	SYSTEM	VERSION	FORMAT
Schematic	Allegro	up to 17.4	ASCII (*.alg)

The **Altium Designer Develop Import Wizard** handles both Allegro PCB Design files (*.brd) and Allegro ASCII Extract files (*.alg). If you have Allegro PCB Editor installed, you can directly translate Allegro PCB Design files (*.brd) into Altium Designer Develop PCB files (*.PcbDoc)

Altium

PCB files translate to Altium Designer Develop as follows:

ALLEGRO FILE TYPE	ALTIUM DESIGNER DEVELOP FILE TYPE
Allegro Binary PCB Design files (*.brd)	Altium Designer Develop PCB files (*.PcbDoc)
Allegro ASCII Extract Files (*.alg)	Altium Designer Develop Schematic file (*.SchDoc)

Using ASCII Conversion to Import Without Allegro Installed on the Same Workstation as Altium Designer Develop

An Altium Designer Develop workstation with no licensed Allegro installation can import Allegro ASCII Extract files (*.alg). The following procedure enables a Licensed Allegro user to convert Allegro binary *.brd files to Altium compatible *.alg files. However, you must run this conversion on an Allegro-licensed machine.

1. Locate the following two files in the \System folder of your Altium Designer Develop installation (Summer 08 or later):

   - Allegro2Altium.bat

   - AllegroExportViews.txt

1. Copy the two files to the folder containing the *.brd file.

2. Open a command prompt, navigate to the folder containing these files and type the following: Allegro2Altium your_file.brd (your_file = file name)

3. Surround your filename with double quotes if the filename contains spaces, i.e., Allegro2Altium «yourfile.brd»

4. Your Allegro-licensed machine will create an ASCII file will create an ASCII file and place it in the folder with a *.alg extension. Copy the *.alg file to the Altium Designer Develop workstation and import using the Import Wizard.
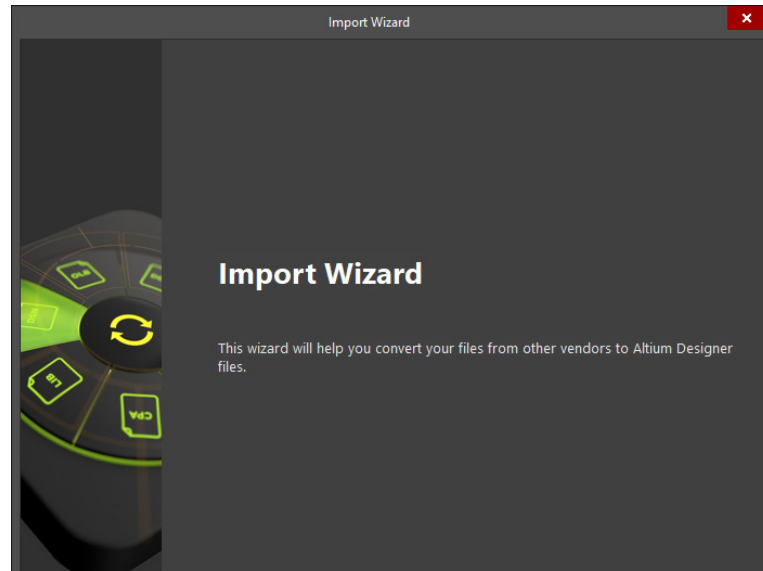
Step 3: Import Data into Altium Designer Develop

Using the Import Wizard for Allegro Design Files

You can launch the Import Wizard from the Altium Designer Develop **File** menu. Follow the step-by-step instructions on how to use the Import Wizard throughout your data migration.

**Step-by-Step Import Instructions on Importing an Allegro ASCII PCB Design file.**

- Start the Import Wizard with **File » Import Wizard**

Starting the Import Wizard for Allegro files

NOTE:
You can also start the Import Wizard by dragging your Allegro Design Files into Altium Designer Develop's Projects Panel. This action will automatically launch the wizard in Allegro Import mode.
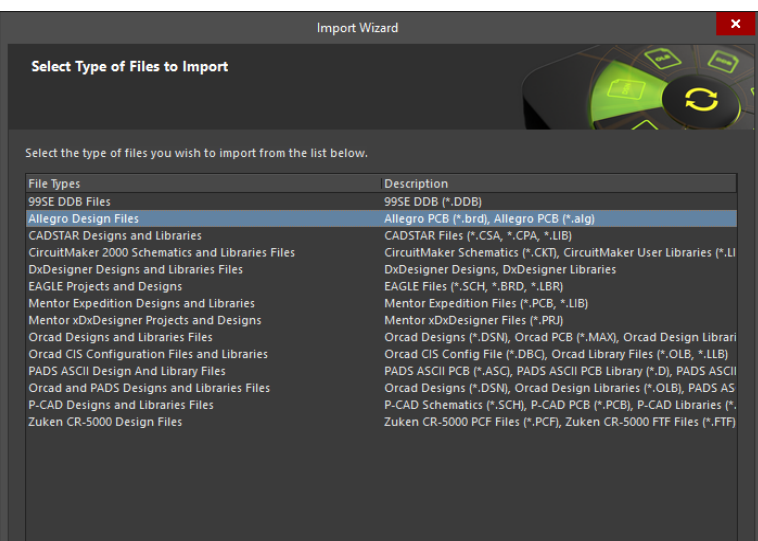

- Select Type of Files to Import→**Allegro Design Files**

- Add the file(s) to be translated. In this case, 'CycloneIII_SB_3C25.ALG' has been used.
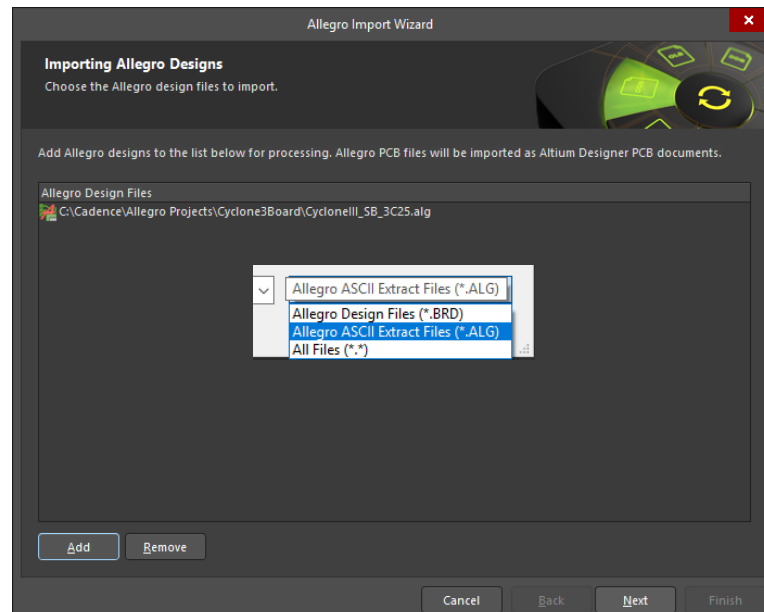

NOTE:
You can add as many *.brd and *.alg files as you like at this point. In the file browser, change the file types pull-down to choose .brd or .alg files, depending on what you want to import. However, the Import Wizard will create separate projects if you add different file types.
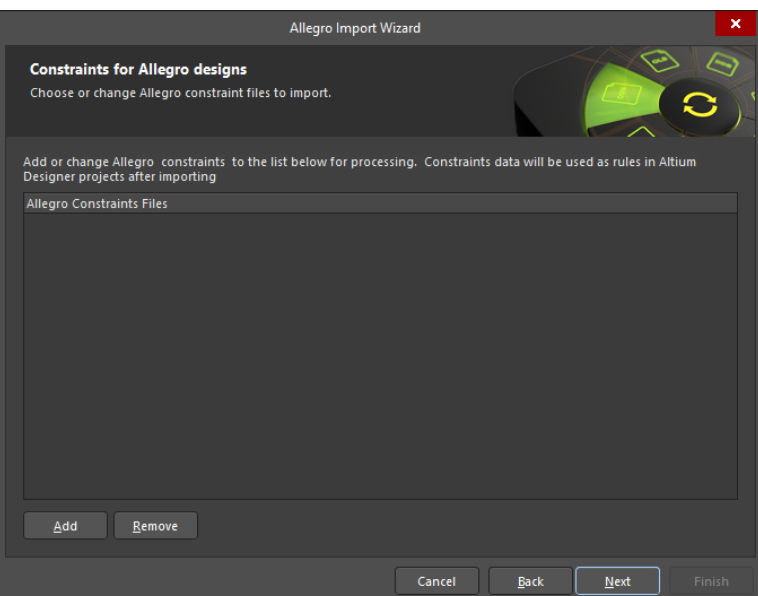

- Add Constraint File or Footprint → If available and if you would like to add.
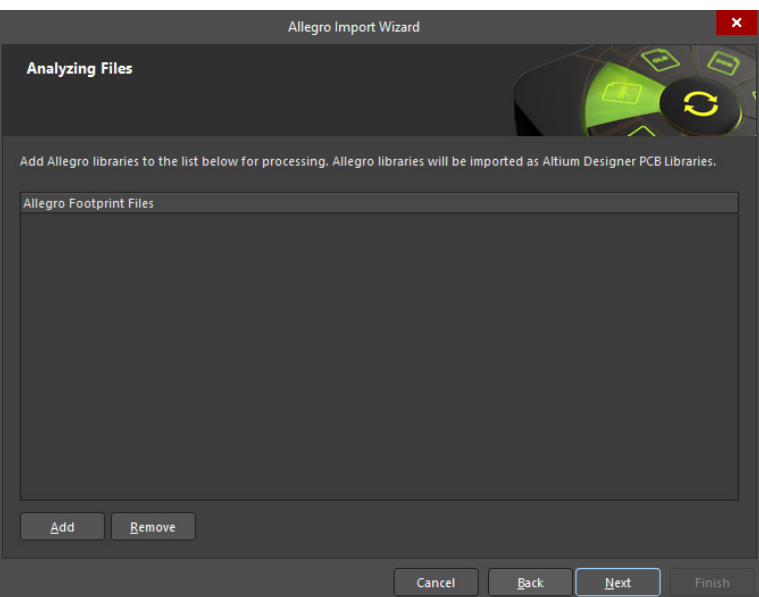
- Use the Reporting Options page to enable or disable the settings for logging all errors, warnings, and events, respectively.
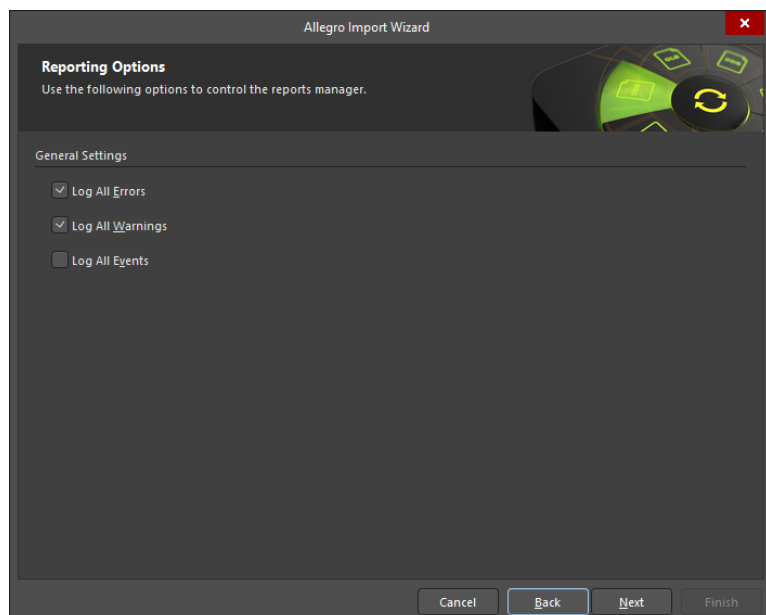
- The Import Wizard presents users with several options to control the translation from Allegro to Altium Designer Develop. **Specify Polygon Connect** and **Plane Connect** options for the PCB import process appear at this stage. Enable the **Import Auto-Generated Copper Pour Cutouts** option to import the auto-generated voids in Allegro PCB Editor as cutouts when the file is translated. The default options are displayed below.
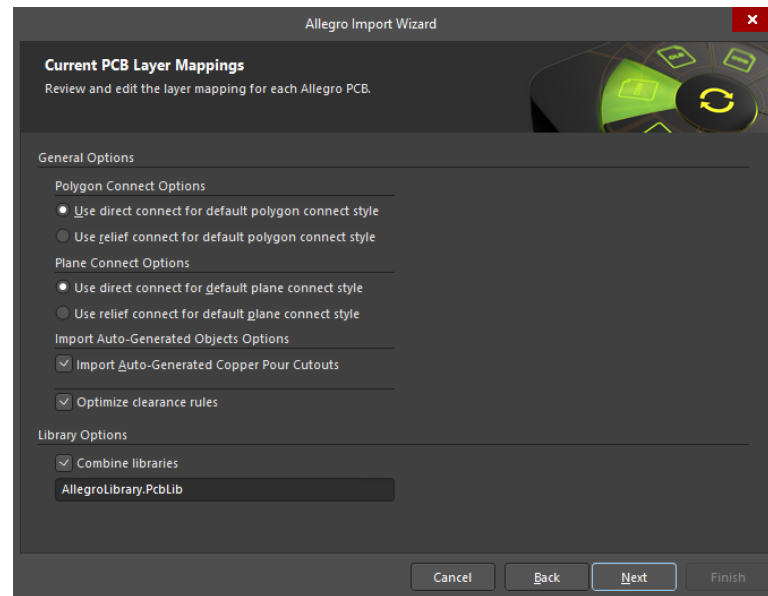
Layer Mapping

The Import Wizard provides the default mapping to build the layer mapping for each PCB. You can customize layer mapping for each design you choose to import. You may wish to import multiple Allegro PCB designs and map the same Allegro layer to the same Altium Designer Develop layer. You can set your layer mapping once and use this configuration for all of your import files if you want.

Setting layer mapping is an advantage because batch layer management can save time when importing multiple designs. The disadvantage to using layer mapping is that Default Layer Mapping is not always intelligent with differing structures in designs and may require some manual changes.
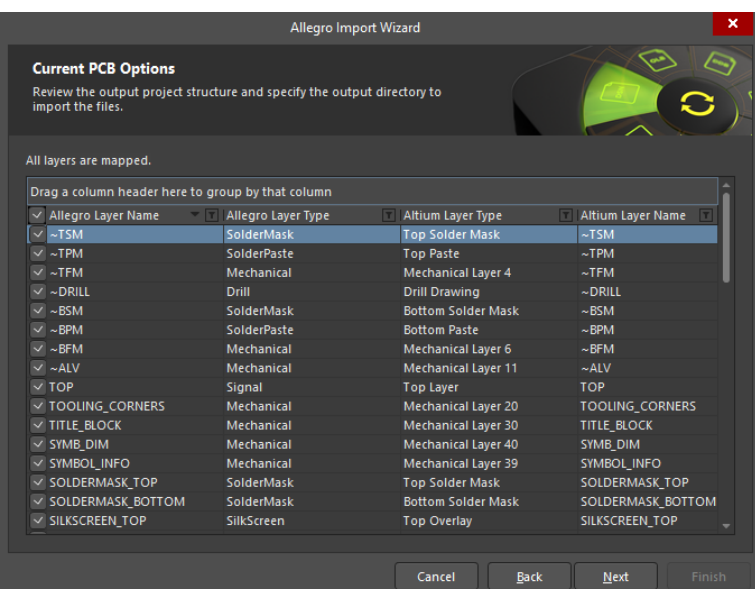
Right-click on the Altium Designer Develop Layer Mapping list (or use the dropdown) to manipulate the layer mapping of Allegro PCBs to Altium Designer Develop PCBs. You can even disable the layers which are not required on the Altium Designer Develop side or are not relevant. By default, the Import Wizard will map all layers.

You can use the **Load and Save Layer Mapping Configuration Files** command using the Load Layer Mapping and Save Layer Mapping menu items, respectively, to quickly apply layer mapping for Allegro and Altium Designer Develop layers.

Altium Designer Develop supports the default 30-layer mode and the expanded 250-layer mode for Allegro PCB files. If the user customizes the default layer mappings, those settings can be saved to a configuration file so that you can quickly reuse those mappings for subsequent translations.

- You will now see a preview of the files selected for translation, along with their output directories. You can change the main output directory at this point if desired.
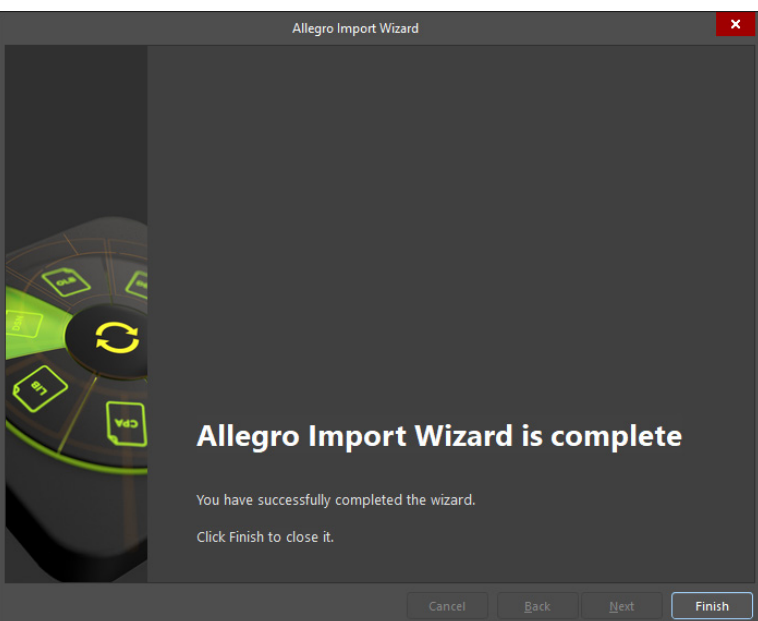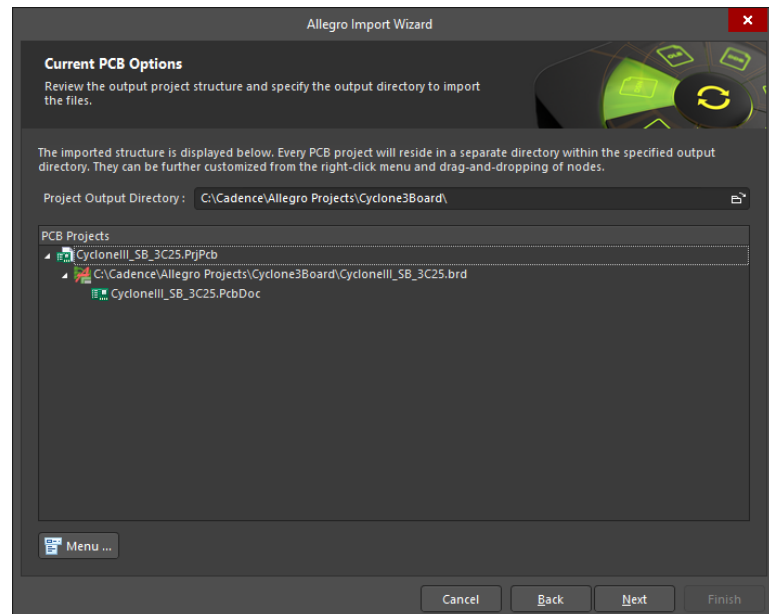

- Click the final Next button, and the Import Wizard will take care of the rest.

- Congratulations, your design has now been imported into Altium Designer Develop!

- Follow the Post Import Tidy Up checklist to ensure the design has been fully reviewed and verified.

- Make it to where you can view the link 'Bringing Together Imported Schematic and PCB Designs' guide.

Getting Help

There are many ways to learn more about Altium Designer Develop:

- F1 over any object, editor, panel, menu entry, or button to open a brief description in your web browser

- Press Shift+F1 or ~ key while running a command for a list of shortcuts you can use in that command.
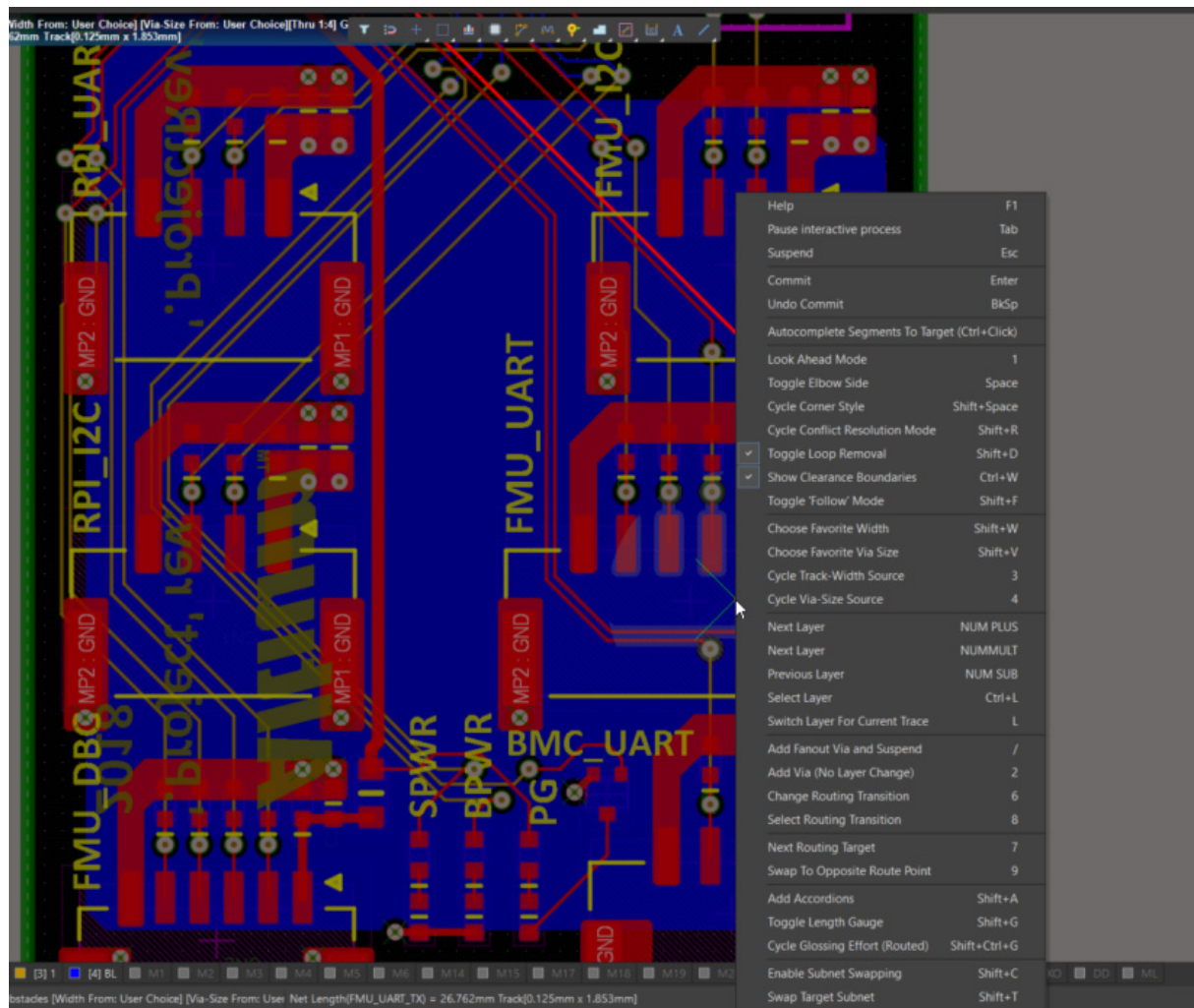
- Access our complete Altium Designer Develop Documentation online.

- Visit the Altium Video Library where you can watch training on over 150 different topics. Each video walks you through the steps needed to complete a task.

- All Altium Designer Develop subscription levels include easy access to real-time support, user communities, comprehensive product documentation, training videos, and knowledge base of targeted solutions for common user experiences. Contact Altium Support Here.

Additional Resources

Online Resources:

If you're ready to dive deeper into the powerful features of Altium Designer Develop, below is a list of articles that provide information to help you get started.

- For a full overview of the PCB design process with Altium Designer Develop, check out this tutorial: A Complete Design Walkthrough with Altium Designer Develop.

- For a look at the basics of creating components, read A Look at Creating Library Components.

- For a tutorial that steps you through all the basics of editing multiple objects, take a look at Schematic Placement and Editing Techniques in Altium Designer Develop.

- For an overview of How to Start an FPGA PCB Layout in Altium Designer Develop, read How to Start an FPGA PCB Layout For Your Embedded System.

Software Platform Resources:

Another great way to get the most out of your Altium Designer Develop installation is by exploring its help section. You can launch the help menu on the left side of Altium Designer Develop by clicking **Help** and navigating to **Exploring Altium Designer Develop**.
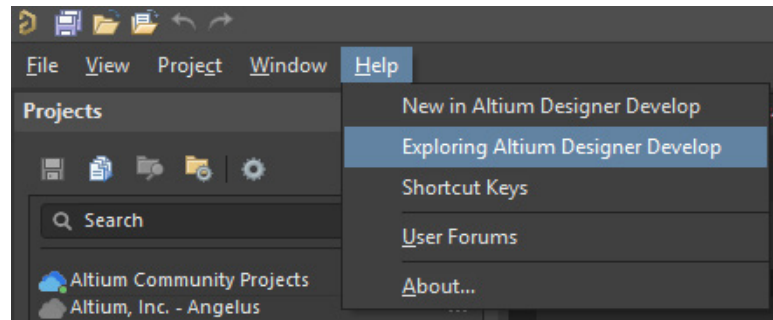
Appendix: Translating OrCAD in Altium Designer Develop

Sheet Structure in OrCAD Capture's Project Manager

Like Altium Designer Develop, OrCAD Capture supports flat and hierarchical designs. Both software tools use a block-like symbol to define a sheet-to-sheet structure.

• Altium Designer Develop uses Sheet Symbols

• OrCAD Capture uses Hierarchical Blocks

The sheet symbol references the lower-level schematic in Altium Designer Develop and OrCAD Capture. In Altium Designer Develop, the lower level schematic is simply another schematic sheet. However, the Capture schematic references can be more complex.

Capture has two layers of design partitioning that affect hierarchy.

• **Schematic documents**: these appear as folder icons in Capture's Project Manager

• **Pages:** These appear as schematic sheet icons.

Each Capture schematic can contain one or more pages. The Capture hierarchical block points to the schematic below, which means the block can reference circuitry divided over multiple pages.

Typically a flat design in Capture is one schematic (folder), with the design created on as many pages as required in that schematic (folder). For a hierarchical design, the hierarchical block symbol (or part with an attached schematic sheet or model) is used to partition the major functional regions of a design.

Each hierarchical block or VHDL model represents a unique design module for a simple hierarchy. The Hierarchy tab in OrCAD Capture's Project Manager displays a simple hierarchical design as a tree of schematic pages. The root module is called the schematic folder or VHDL entity at the top of a hierarchy, which directly or indirectly refers to all other modules in the design. In the OrCAD Project Manager's File tab, the root module has a backslash on its folder icon, shown below. The root module folder and any other module folder can contain as many schematic pages or VHDL models as required.
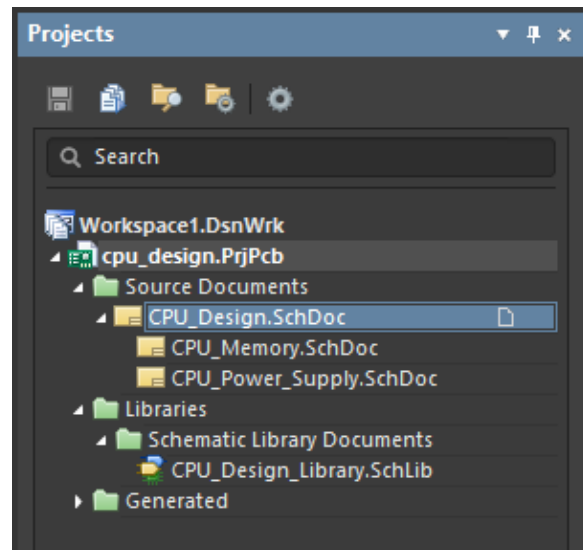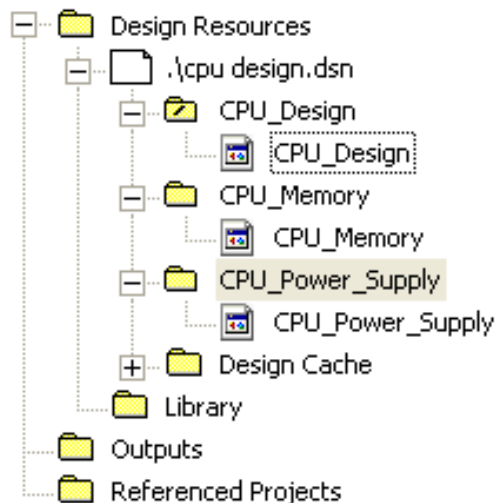
Project Panels - On the left is the structure of an example CPU_Design project as it would appear in OrCAD Capture. On the right is the same project CPU Design after import in the Projects panel of Altium Designer Develop.

Defining Net Connectivity

In OrCAD Capture, net connectivity uses net aliases, off-page connectors, hierarchical blocks, hierarchical ports, and globals. Nets between schematic pages within a single schematic folder are connected through the off-page connectors, while the hierarchical blocks and ports connect the nets between the schematic folders. Globals are used to connect power/ground nets throughout the design.

Altium Designer Develop uses a similar set of net identifiers to create net connectivity. Within a schematic sheet, you can use Wires and Net Labels. Between schematic sheets, nets in a flat design are typically connected using Ports, but Off-Sheet Connectors are also available. Nets in a hierarchical design are connected from a Port on the lower sheet to a Sheet Entry of the same name in the sheet symbol that represents the lower sheet. Power/ground nets are connected using Power Ports.