Altium

MIGRATION GUIDE

Autodesk Eagle to Altium Designer Develop

EAGLE

ALTIUM DESIGNER DEVELOP

Contents

Smooth Transition from Autodesk Eagle PCB to Altium Designer Develop

For the purpose of this migration guide we are going to focus on importing Eagle PCB Designs, if you require other files migrated, please refer to their specific guide.

Legacy Translation

Before You Start to Import

This guide will show you how to import data from your legacy system into Altium Designer Develop. What it will not ask is "why do you want to do it?" Customers frequently ask about data import facilities before we go forward however, we need to ask ourselves why.

”

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

- Does the data include all necessary information to create a robust representation within Altium?
- Is it easier to "draw a line in the sand" and start new designs in the new system 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 allowed after importing?
- Do you need to take advantage of additional features of Altium once imported and how do you plan to achieve this?

”

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,17 and error is present in your library?

So, after considering these points you should read on if you still wish to import your legacy data into Altium.

Preparing to Migrate Your Legacy Data

Never forget that migration of data follows the “garbage in.... more garbage out” principle. So, we need to consider four distinct phases of the process, this is best shown in the flow diagram below: -

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

3. Import data

- Save «raw» imported data
- Integrity check

4. Post import

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

Step 1: Prepare Original Data - Pre-Translation Tidy Up

It is prudent to clean and tidy your design within your legacy system before trying to export. We’ve put together a checklist to help you:

Schematic considerations

- Does this design have single pin components representing power objects?
- Connectors are represented as one-gate-per-pin with over 256 “gates”
- Keep a note of the ambiguous connectivity in the design (For example - multifunctional pins, hidden pins, or implicit connections).
- Make sure to understand which nets in the design are local(defined at the document level) in nature.
- Do the schematic symbols call up the correct PCB footprints?
- Does the schematic match the PCB?

PCB considerations

- A large number of graphical objects like mechanical drawing or non-ECO-registered drawing primitives need to have assigned in the documentation layers.
- Make a note if the design has star point earth.
- Deliberate DRC violations (For example, allowed short circuits and net-ties).
- Objects extending beyond the environment
- Knowledge of correct PCB layer assignment is required in order to map the imported layers with existing layers in Altium Designer Develop.
- Do the auto-named nets match with the schematic?

Library considerations

- Schematic symbols matching with PCB footprints
- Correct supply chain information and BoM parameters
- Associated 3D models cannot be imported for eagle-based libraries and one can import the respective STEP models in Altium later on.
- Correct representation of custom pads, copper shapes, solder mask and resist.

Step 2: Save Data in a Suitable Format

Supported Version and File Formats

The following table details the versions of all Eagle design formats that can be migrated into Altium Designer Develop. Note that this list is updated all of the time, so please check with us for specific systems and versions before undertaking migration.

TYPE	SYSTEM	VERSION	FORMAT
PCB	Eagle	6.4.0	(*.BRD)
SCH	Eagle	6.4.0	(*.SCH)
LIB	Eagle	6.4.0	(.*LBR)

Altium Designer Develop's EAGLE Importer is able to import EAGLE design files saved with EAGLE version 6.4.0 (or later). These are XML-format in nature – EAGLE binary-format design files cannot be imported directly using the EAGLE Importer. For these older, binary version design files, it is advised to save them in this later (XML) format, through your EAGLE software, before attempting to import them into Altium Designer Develop.

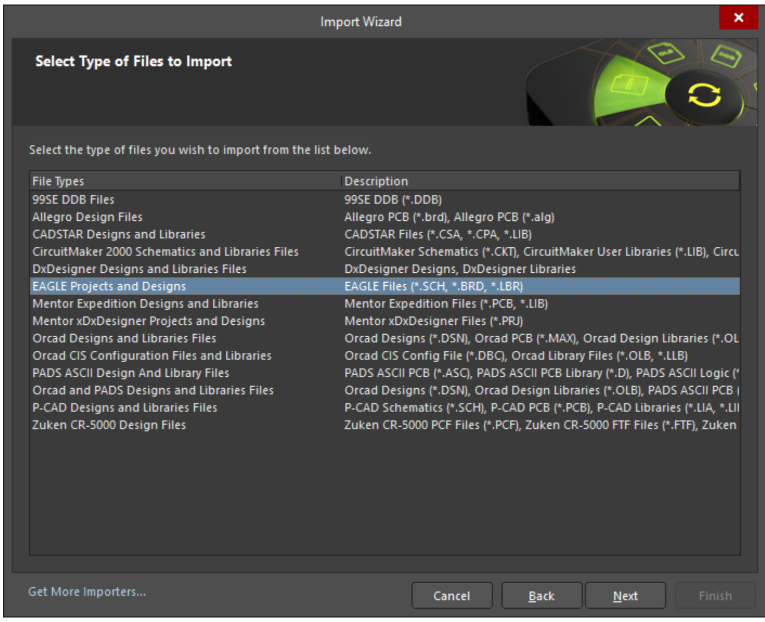
PCB files translate as follows:

- Eagle PCB Design files (*.brd) translate to Altium Designer Develop PCB files (*.PcbDoc).
- Eagle SCH Design files (*.sch) translate to Altium Designer Develop PCB files (*.SchDoc).
- Eagle Library Design Files (*.lbr) translate to Altium Designer Develop PCB Libraries and Altium Designer Develop Schematic Libraries (*.PcbLib & *.SchLib).

Step 3: Import Data into Altium

Using the Import Wizard for Eagle Design Files

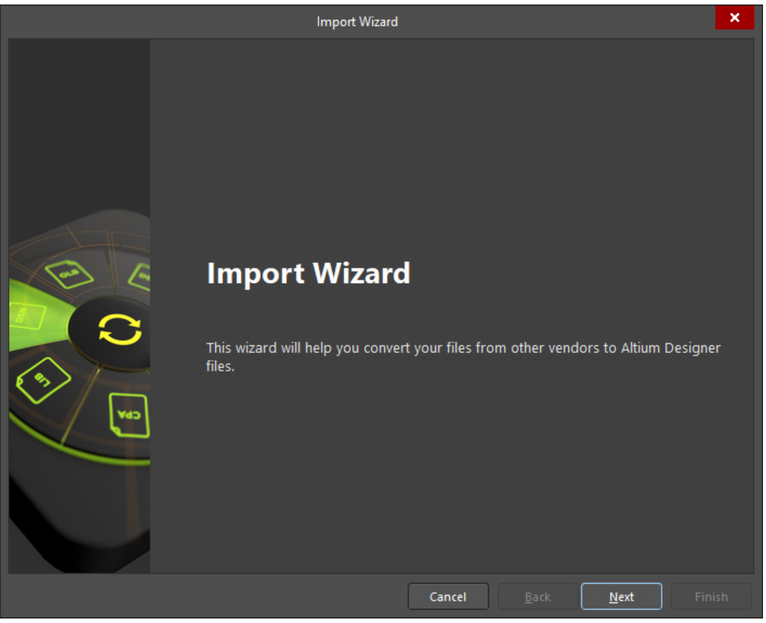
The Import Wizard can be launched from the Altium Designer Develop File menu. Choose the **EAGLE Projects and Designs** from the list of file types as shown in the screenshot below. On the “Importing Eagle Design Files” screen, click the Add button to choose the Eagle design files. Multiple files can be translated at the same time. Step-by-step instructions on how to use the Import Wizard are to follow.



Starting the Import Wizard for Eagle Design files

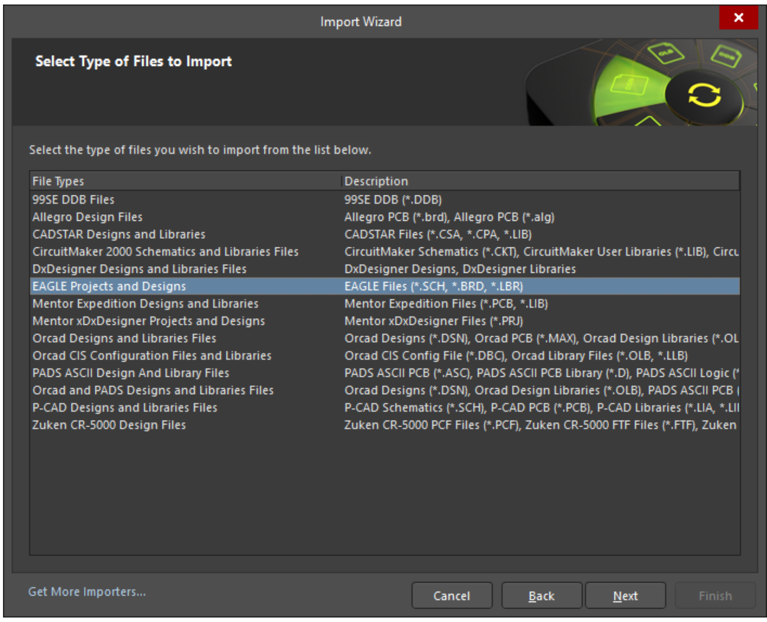
> NOTE:
> You can also drag your Eagle Design Files into Altium Designer Develop’s Projects Panel, this will automatically launch the wizard in Eagle Import mode.

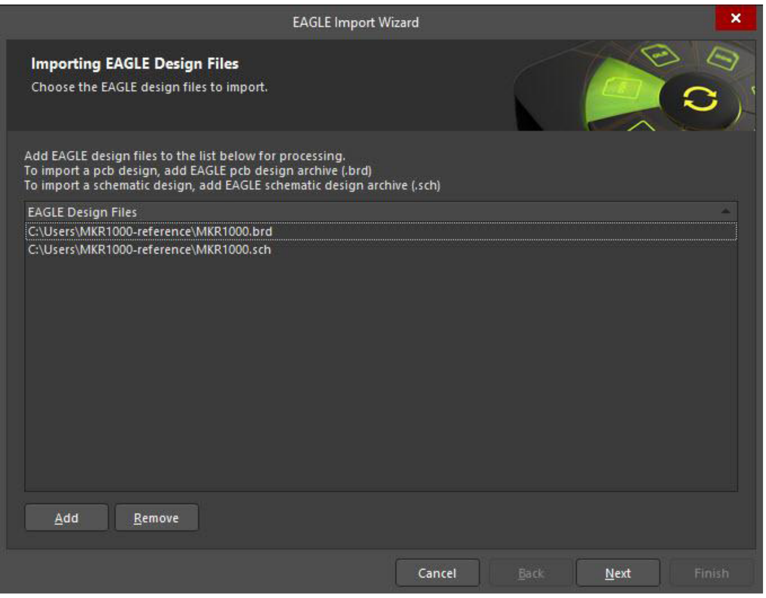
Step-by-Step Import Instructions on Importing Eagle Design files.



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

- Select Type of Files to Import→**Eagle Projects and Designs**

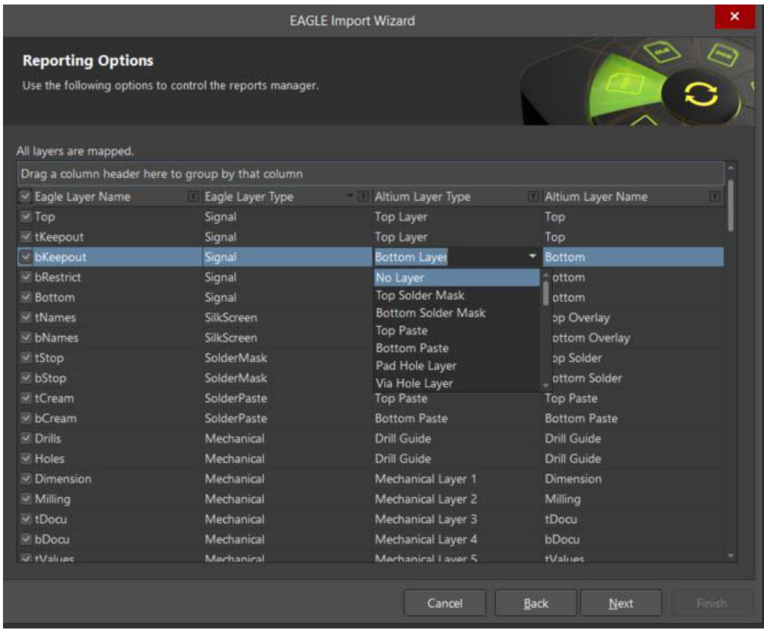


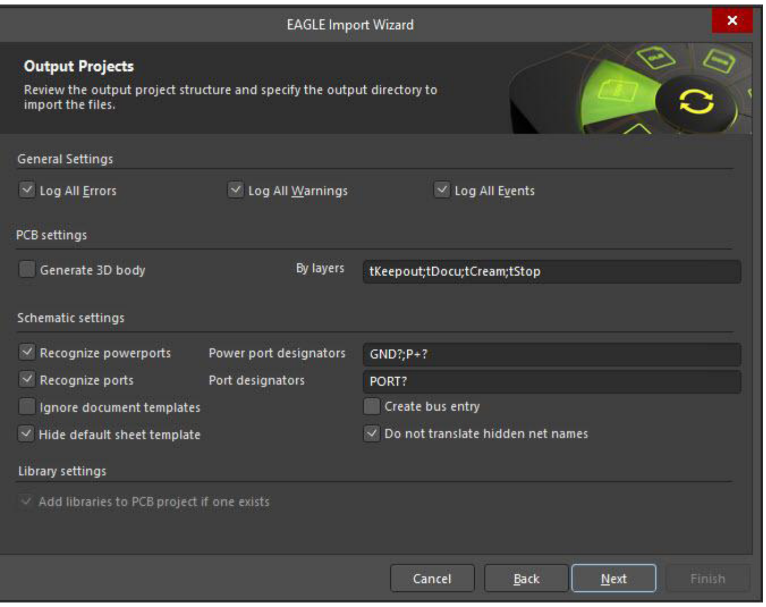


- Add the file(s) to be translated. In this case, 'MKR1000.brd' and 'MKR1000.sch' have been used.

> NOTE:
> You can add as many .brd and .sch files at this point, however, if you add files of a different file name, separate projects will be created. Once you have chosen your design files, the next screen will allow you to import any library files you have.

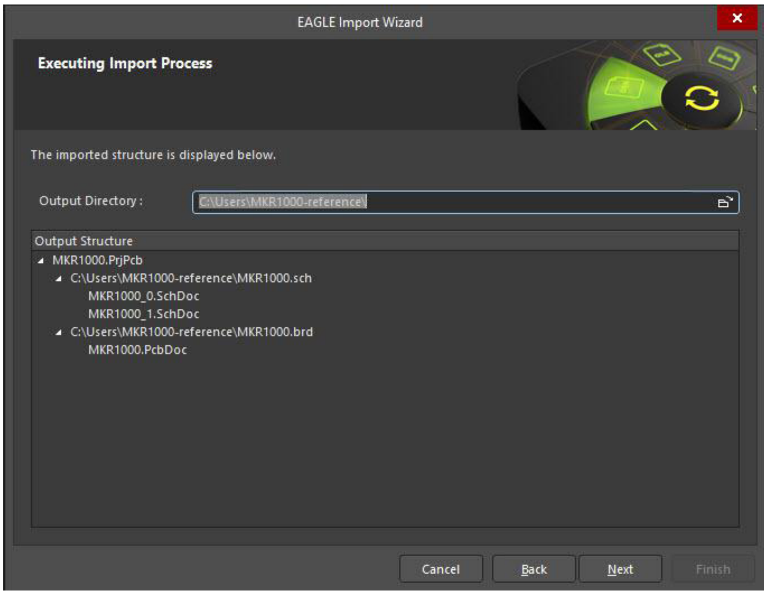
Using the 'Reporting Options' page map the layers for the PCB by assigning a suitable layer type and layer name to be Signal/Solder/Paste/Silkscreen/Mechanical

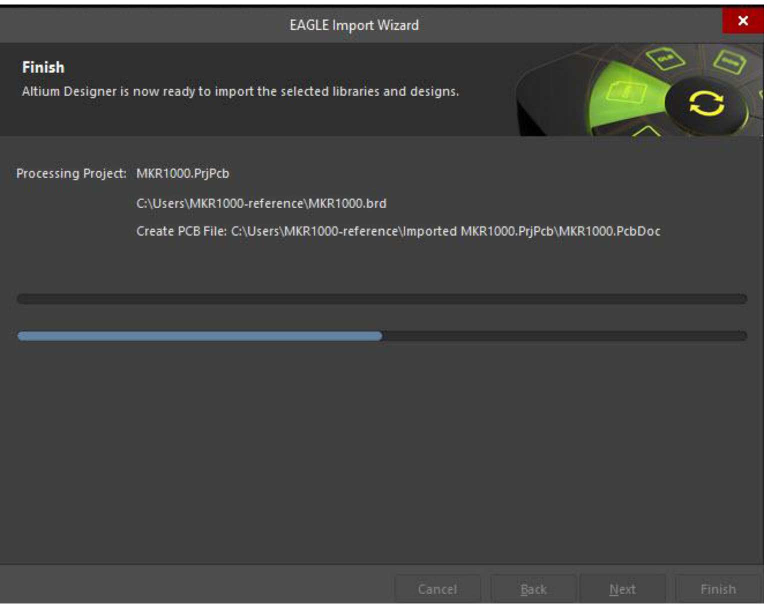




- Use the 'Output Project Structure' page to enable or disable the settings for logging all errors, all warnings, and events respectively. You can also set up Schematic Settings here.

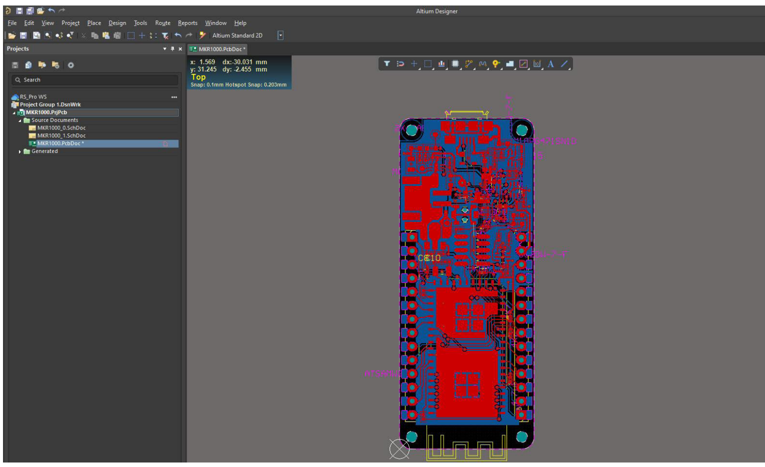
- A preview of the files being translated and their output directories are then shown. You can change the main output directory at this point if desired.

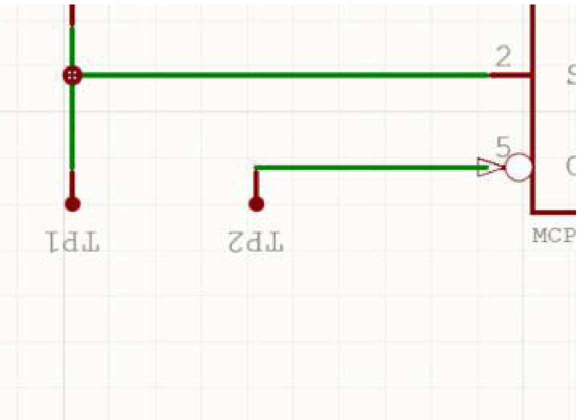


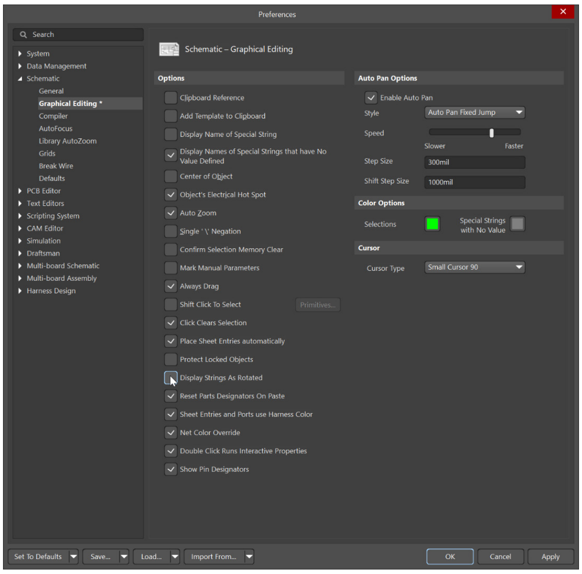


- Click the final Next button and the Import Wizard will take care of the rest.

- Congratulations, your design has now been imported into Altium Designer Develop! Follow the Post Import Tidy Up checks to ensure the design has been fully checked and verified.







NOTE:
In case the imported text on schematics is inverted after design import, simply click on the gear button at the top right corner and Preferences>Schematic>Graphical Editing disable the option 'Display Strings as Rotated'. An example is shown above.

Step 4: Post Import Tidy Up

We've put together a verification checklist for you:-

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

Reference Tech-Doc - Post Import Consideration

Getting Help

Main article: Documentation and Help

The best way to learn is through doing, Altium and Altium Designer Develop provide a number of ways to help you do that:

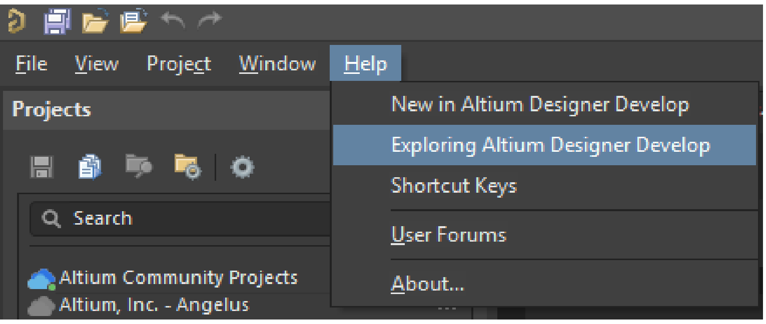
- Press F1 with cursor over any object, editor, panel, menu entry, or button to open a brief description in your web browser
- Use shortcut Shift+F1 while running a command for a list of shortcuts you can use in that command.
- Altium Designer Shortcut Keys
- Search the Altium Documentation, on the Altium website.
- Visit the Video Library where you can watch over 150 short training videos, each detailing the exact steps needed to complete a task.
- Activating & Managing Your Altium 365 Workspace

Image illustrating pressing Shift+F1 during active interactive routing command

See Also

Below are references to other articles and tutorials in the Altium Designer Develop Documentation Library that talk more about the conceptual information as well as walk you through specific tasks. Remember, you can also browse through the Help contents, and use F1 and What's This at any time in a dialog for more details.

- For more PCB project options, refer to the 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.



A great place to start your journey through all the new possibilities with your Altium Designer Develop installation. On the top left of Altium Designer Develop you can find the **Help » Exploring Altium Designer Develop link**

From here, you can easily access the Documentation where the "Getting Familiar with the Altium Design Environment" category will ease your beginning into using Altium.

Contact

Website
www.altium.com

Phone: +1 858-864-1500

Head Office

Address:
4225 Executive Square, Suite 800,
La Jolla, CA 92037