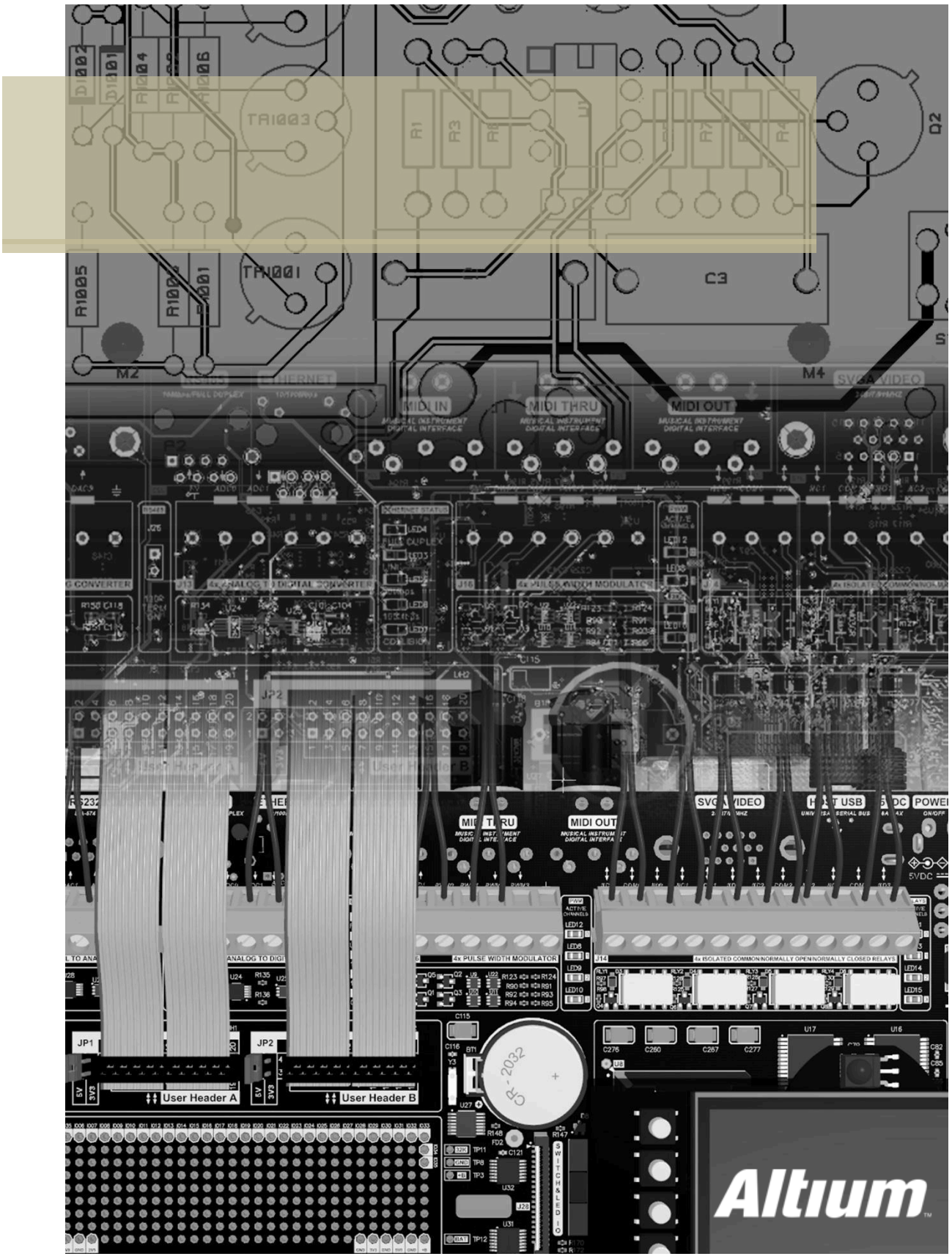

MIDI IN
MIDI THRU
MIDI OUT
4x ANALOG TO DIGITAL CONVERTER
4x PULSE WIDTH MODULATOR
4x ISOLATED COMMON/NORMALLY OPEN/NORMALLY CLOSED RELAYS
JP1
JP2
User Header A
User Header B
CR - 2032
Altium

Table of Contents

Bringing Together Imported Schematic and PCB Designs 1
- Add Existing to Project 5
- Resolving Schematic Errors 11
- Engineering Change Order (ECO) 13
- Resolving Synchronization Errors 14
- Carrying Out a Design Rule Check 14

Introduction

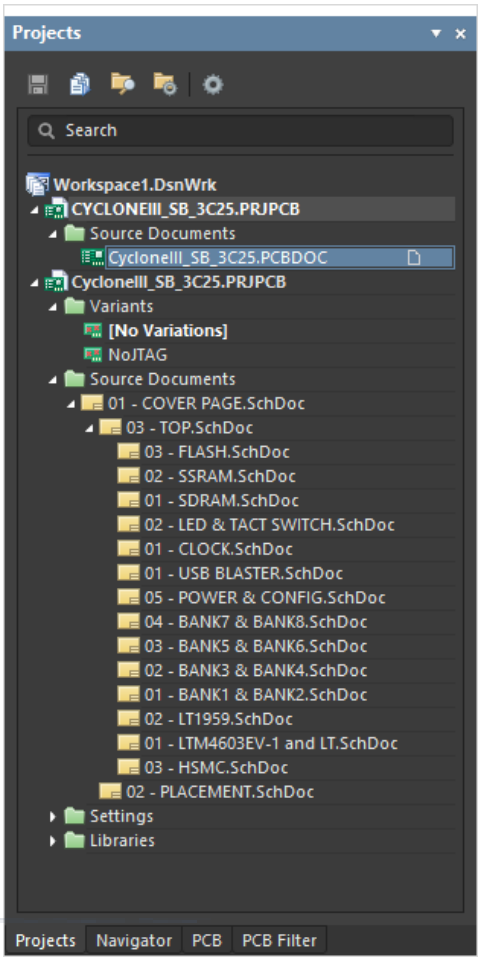
Electronic designs aren’t always created in one single EDA package, for example Schematic Designs could be created in xDxDesigner but the PCB Designs in PADS Layout. If this was the case, then to import the complete design into Altium two separate imports are going to have to be made.

This short guide will show you how to link the separately imported files into a complete PCB Project within Altium Designer and carry out important synchronisation techniques.

Getting Started

After importing you will be left with two separate PCB projects, one containing the PCB Design and the other containing the Schematic documents.

In this example I have imported a PCB design from Allegro and the schematics from OrCAD Capture.



Merging Projects

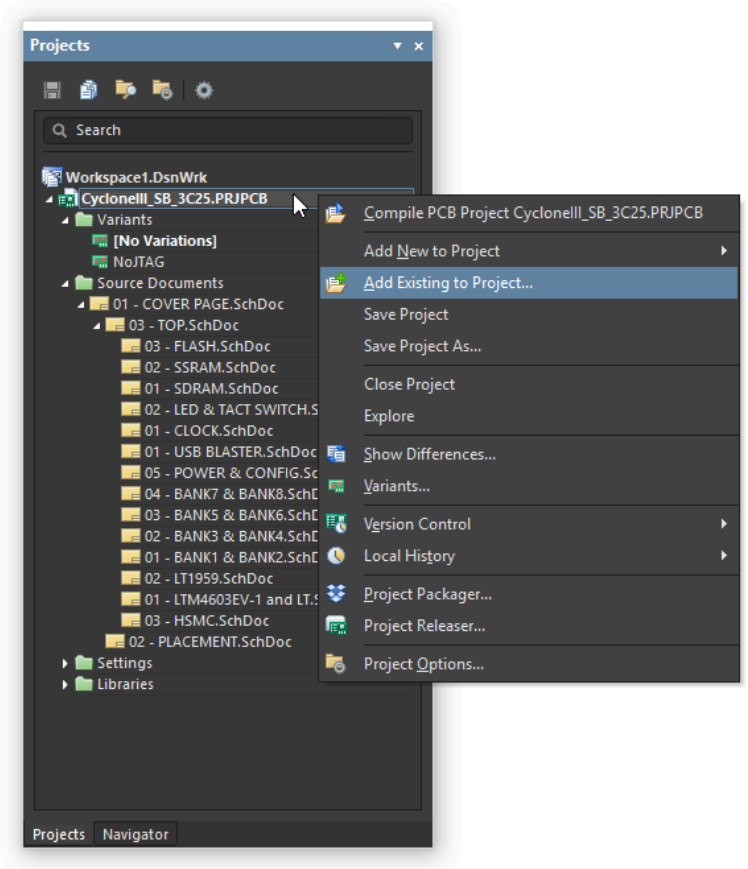
Add Existing to Project

Close either project, I'm going to close the project containing the single PCB document to make less work later.

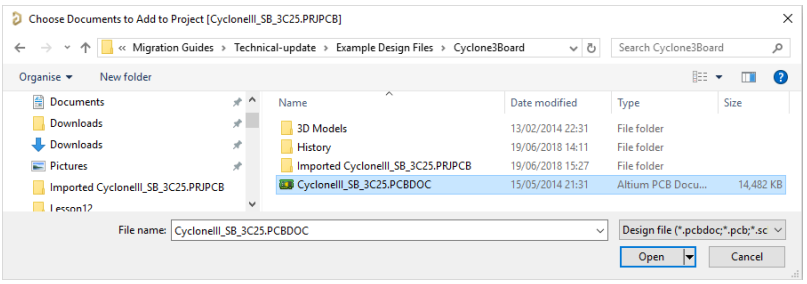


Before you close the project, you may wish to right click the PCB file and go to Explore so you know where the file is saved.

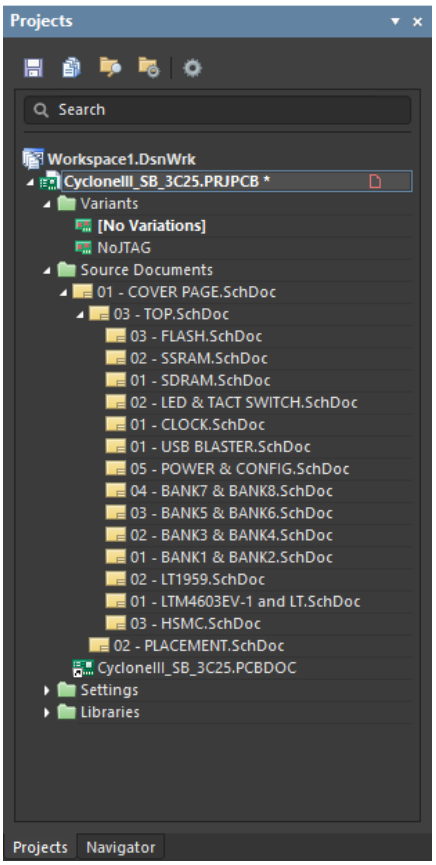
Right click on the remaining project and go to **Add Existing to Project.**



Navigate to PCB File we just closed and add it to our project.



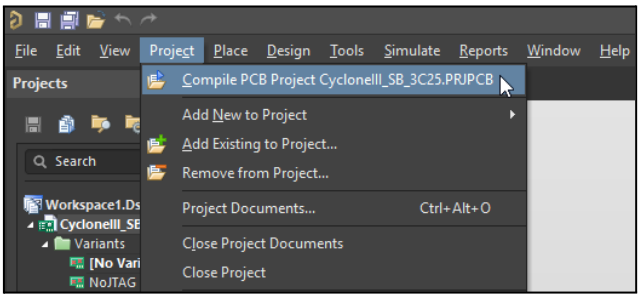
We have now brought a complete design into Altium Designer and successfully linked them into a single PCB Project. However, we aren't finished yet.



If the PCB document is not in the same project folder as the destination project, you will see a small arrow on the file in the projects tab . Ideally, you would move the project files in the same folder directory, i.e. use File » Save As on the document to create a copy in the same location, however it won't cause any issues if they're not.

You should now compile your project by going to **Project » Compile PCB Project** this will generate a list of errors and warnings (if there are any) which you might need to rectify. See Post Import Tidy Up section for more information on Error Checking your Schematics.

When you select **Project » Compile Project** this will determine the connectivity between the sheets. Once complied you will be able to carry out post-compilation activities, such as comparing and showing differences between schematics, parameter managing, parametric navigation of your design, and cross probing back and forth between the schematics and PCB.



Post Import Tidy Up

Referring to the migration guides again, you will remember a section on post import actions. This might be a good time to carry those out, I've included them below. How to carry some of these checks can be found later in the document.

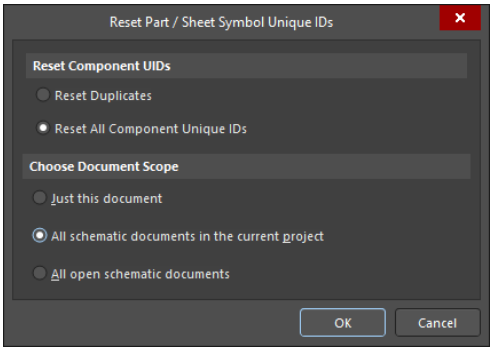
- ❑ Physical check
  - ❑ View » Fit Document
  - ❑ Board shape and cut outs
- ❑ Electrical check
  - ❑ Netlist
- ❑ Rules
  - ❑ Have all rules been imported
  - ❑ DRC check
  - ❑ Check settings for polygons - Island removal, min primitive size
  - ❑ Thermal reliefs, direct connect
  - ❑ Check power plane 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

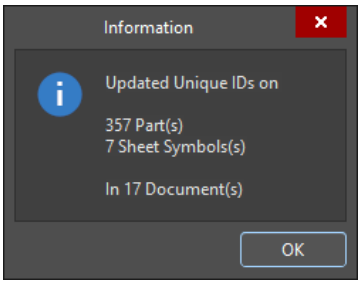
Component Links

In Altium Designer, Unique ID (UID) values allow schematic and PCB objects to remain associated with one another even when their component designators have been modified in one editor.

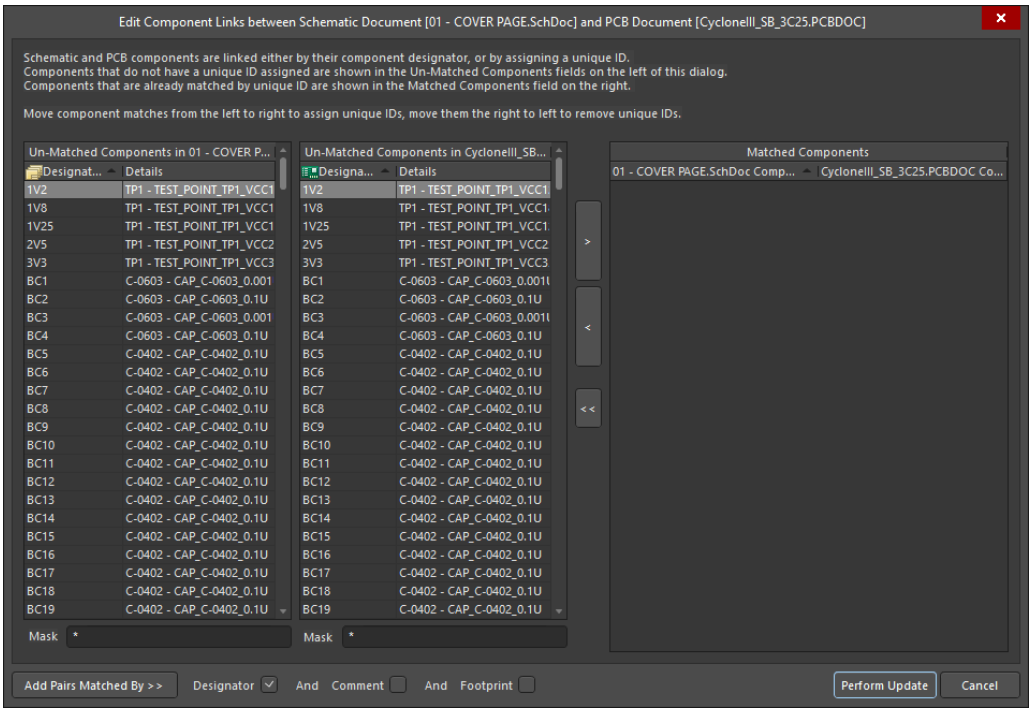
However, you will find that all links between schematic and PCB components are removed when you import from another software. Re-establishing the UID linking is easy, but it must be done based on the designators.

First, reset all Unique ID values on the schematic side, by selecting **Tools » Convert » Reset Component Unique IDs** from the schematic editor menus. Ensure that you reset the Unique ID's for **All schematic documents in the current project.**

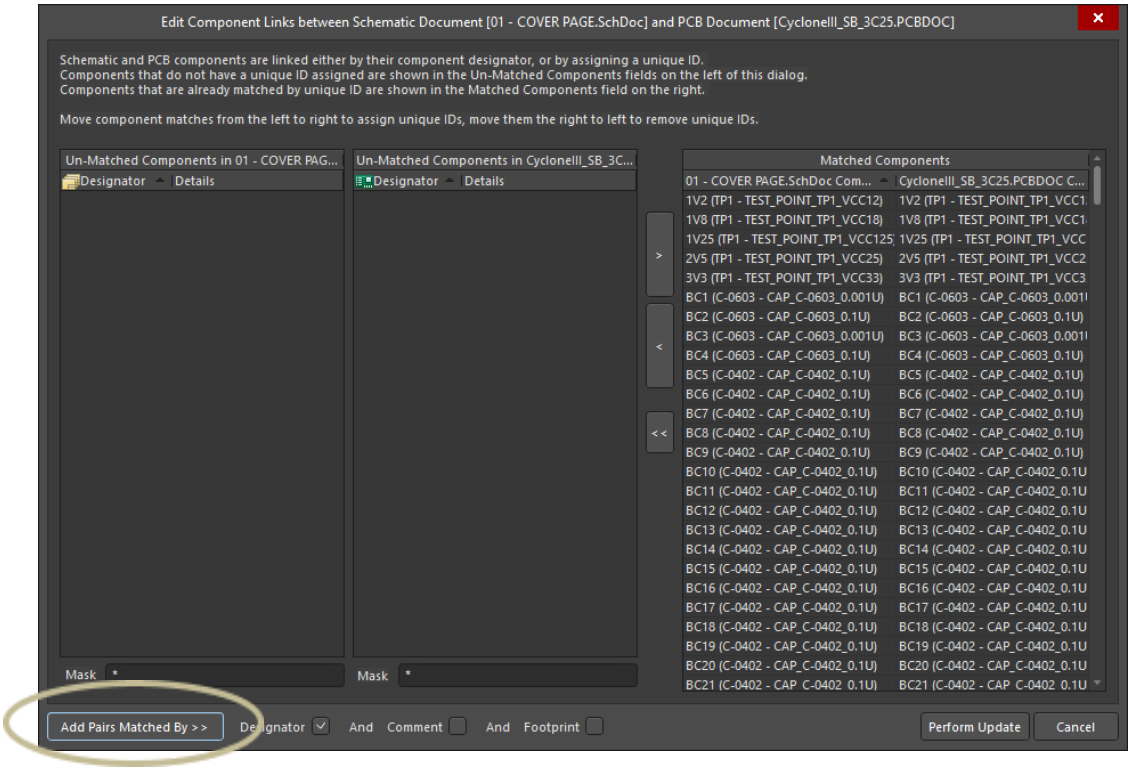




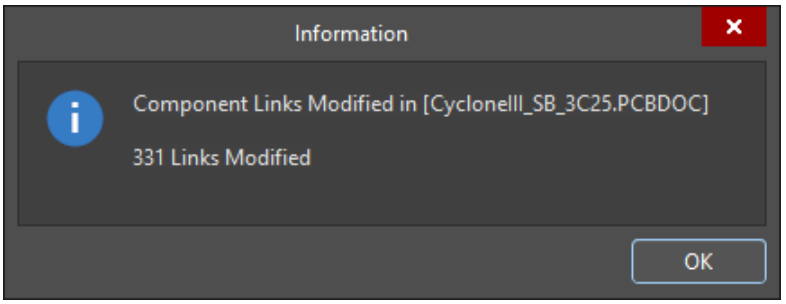
Open the PCB editor and navigate to the Component Links dialog (**Project » Component Links**).



First, add pairs by matching designators (the default correlation).



Then perform the update.

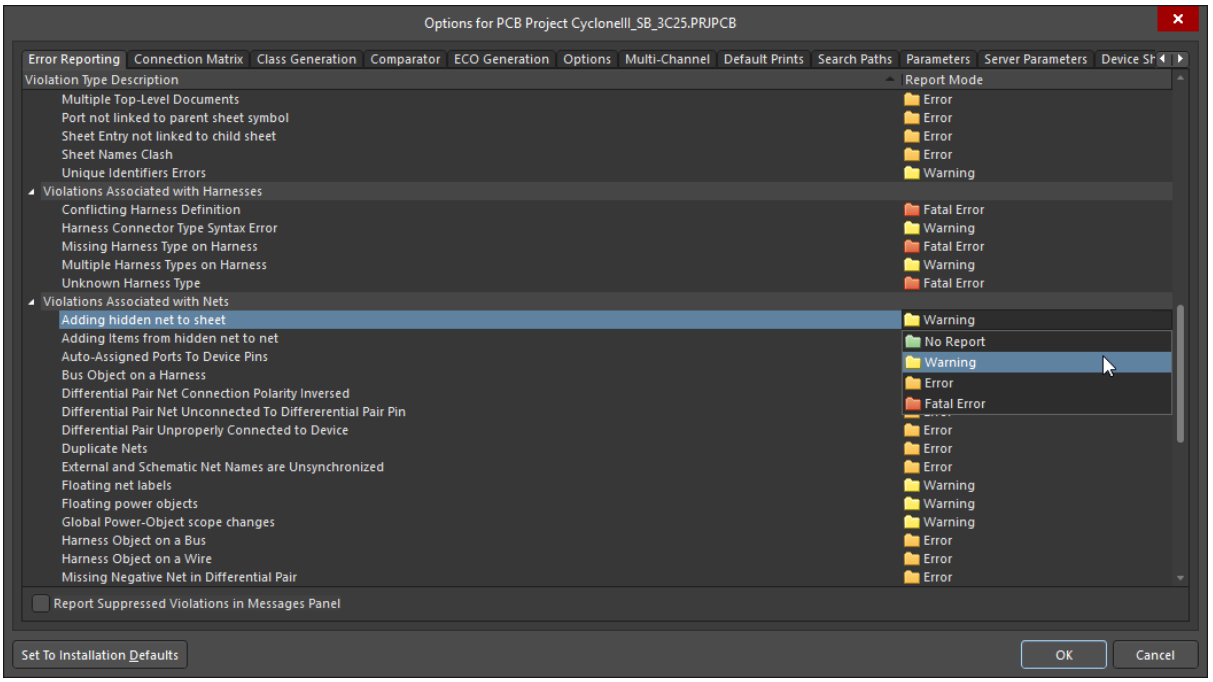


An underlying difference between Altium Designer and other EDA software packages is that establishing links is not a prerequisite to synchronization. If, for example, you skip the sequence described above and simply try running update/import commands on a PCB design you brought into Altium Designer, you will be informed that, although synchronization by Unique IDs has failed, you may still proceed to match by designators. Doing so will not have any effect upon the Unique ID fields in your design, meaning that if you repeat the process, the same status will be reported. Assigning the same Unique ID values to schematic components and PCB footprints is the only way to create persistent links between them.

Error Checking your Schematic

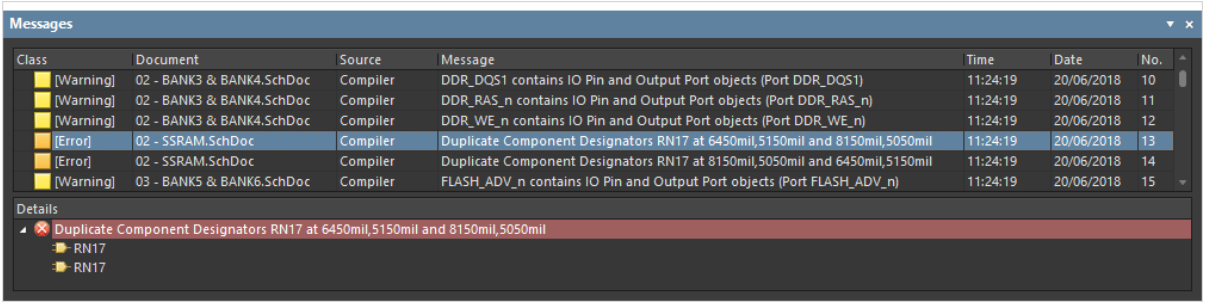
Earlier we ran the compile process to build the sheet to sheet connectivity of our imported project. Running this compile process will also run error checking on the project, which will inform you of any drafting or electrical rule check errors.

Error Checking options can be set under **Project » Project Options » Error Checking** and largely covers your drafting errors.

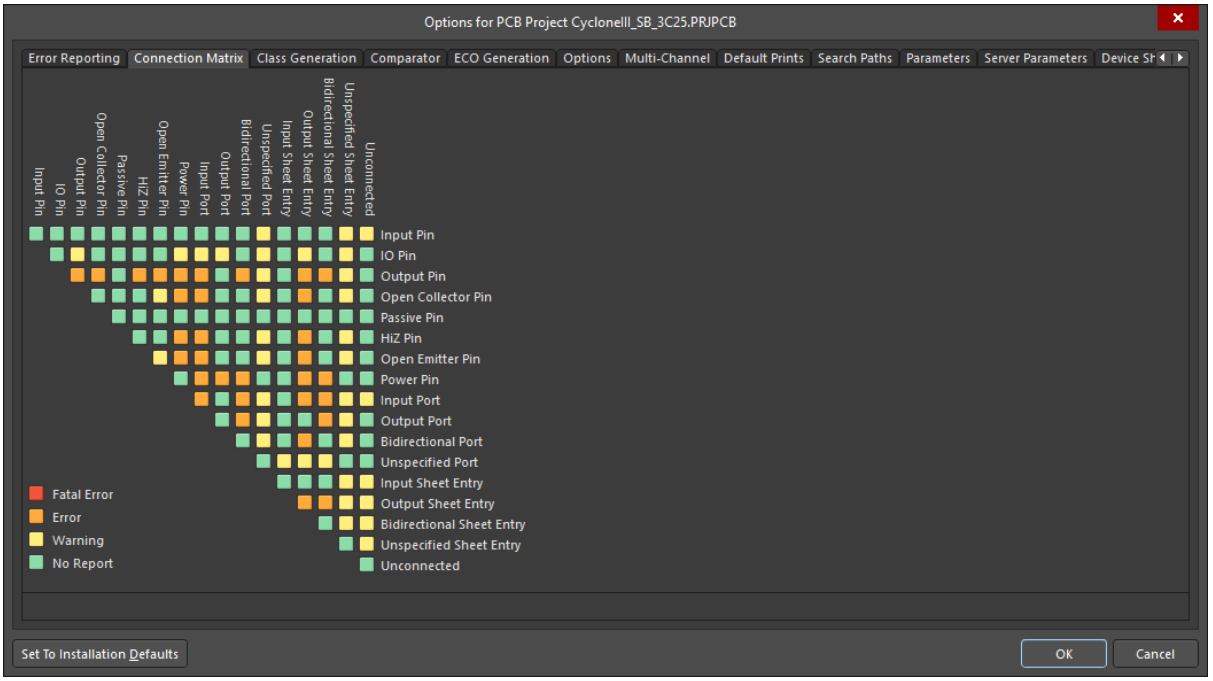


Each violation can be set to the following levels of reporting and so customised to suite your needs

No Report	Provides no feedback about the violation
Warning	Listed as a warning in the messages panel, useful information but doesn't necessarily need fixing
Error	Listed in the Messages Panel, panel will open, if closed, if you have an error.
Fatal Error	Listed in the Messages Panel, panel will open, if closed, if you have an error.



Electrical Rule Checking can be found under **Project » Project Options » Connection Matrix**. Clicking on the coloured blocks allows you to set the reporting severity.



Resolving Schematic Errors

Common errors/warnings you may find.

- Nets with multiple names
  - Often caused by an improper placement of a net label onto another net
  - Use the Details area of the Messages panel to determine the net label and location
  - Double-click the Detail entries to navigate to that item

- Nets with no driving source
  - Can be caused by passive pins diving input pins for example a connector or series passive component
  - Edit the connector/passive component pin to the appropriate electrical types
  - Can be disabled in Project Options » Error Reporting or by placing “No ERC” markers

- Nets containing floating input pins
  - If the pin is not used either tie it to the appropriate power line (e.g. GND) or place a No ERC directive on it.

    Documentation and help on finding and resolving errors can be found here [Compiling and Verifying the Design]

Design Synchronization

Design Synchronization is integrated in Altium Designer without the need for passing a net list. Synchronization in Altium Designer is also bi-directional, allowing you to make annotation changes and component property updates in both directions between your schematic and PCB, in a single operation.

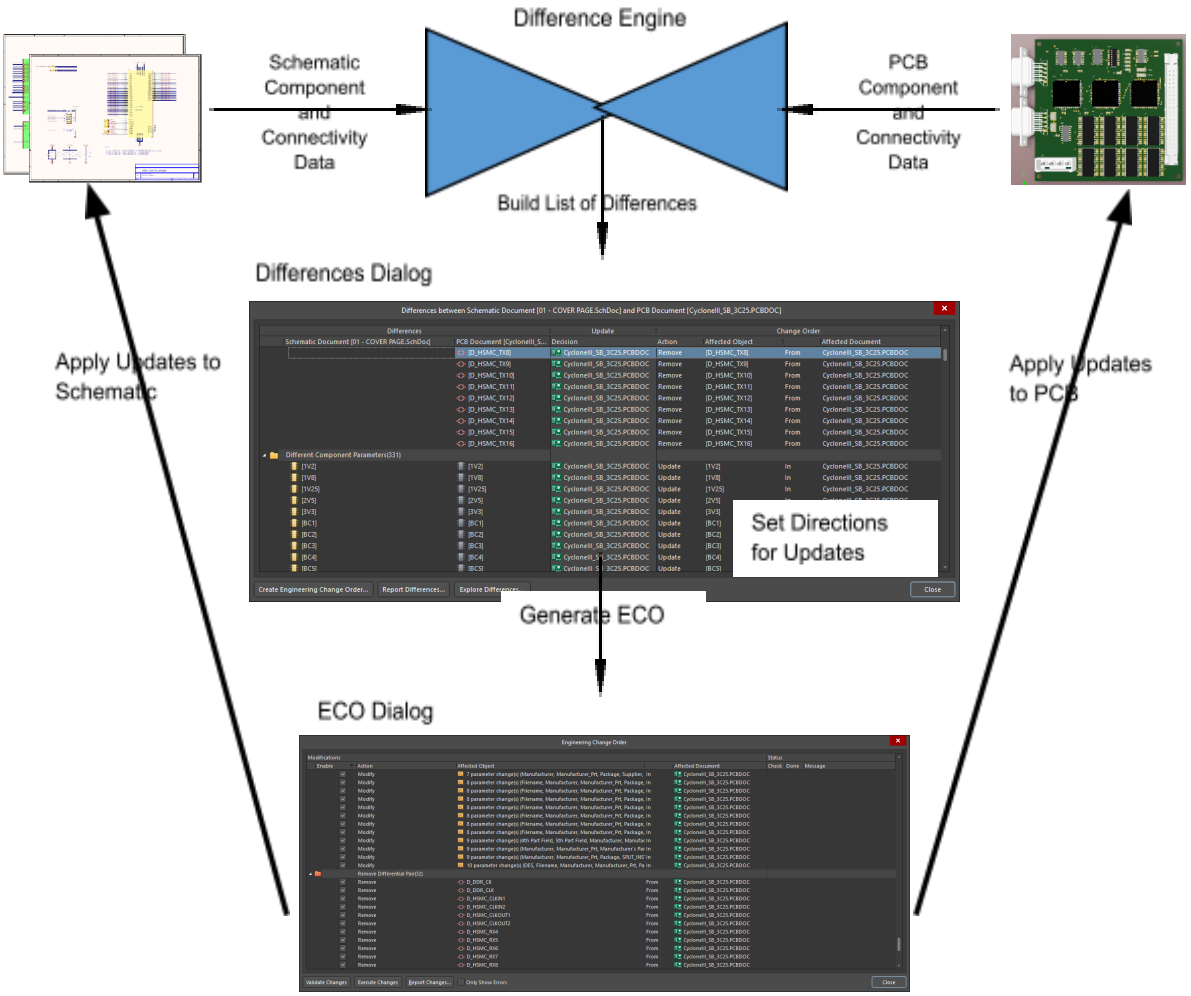


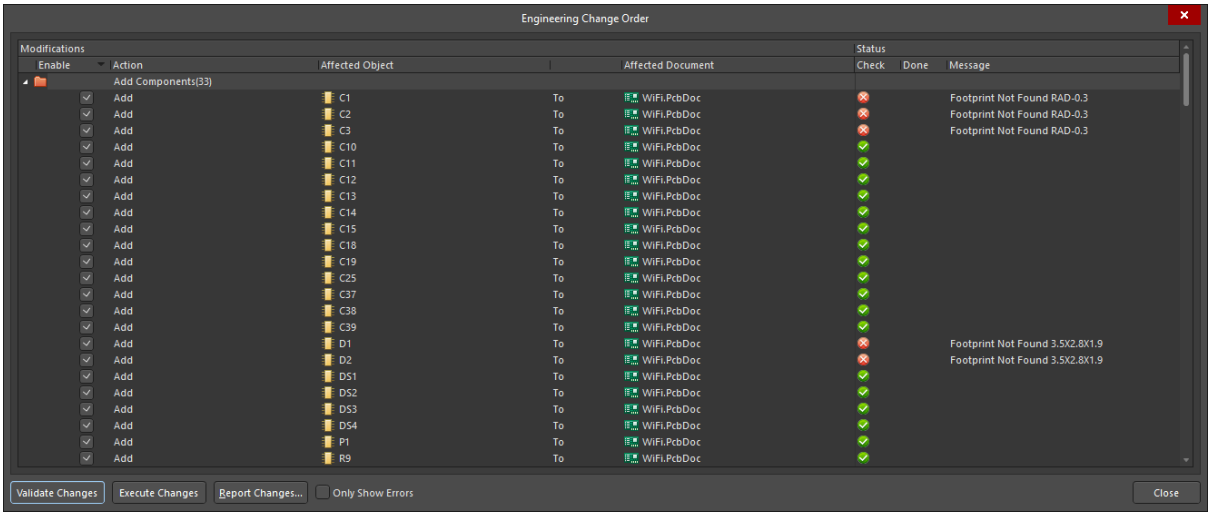
Figure 1. Flow of the synchronization process in Altium Designer

Engineering Change Order (ECO)

Once you've compiled your project, linked Unique ID's and rectified any errors that you can, you should undergo an Engineering Change Order.

The **Engineering Change Order** dialog lists all modifications required to implement changes to one or more design documents, to satisfy the synchronization action requested. For example, you may be synchronizing the PCB document with its source schematics, or updating the parameters of placed components source schematics, with parameter information stored in source libraries or a company database. Whatever the specific action requested, the **Engineering Change Order** dialog allows you to browse, verify and execute the necessary modifications that comprise the ECO.

You should validate the changes first, this validates the ECO before executing it. When an ECO is invalid (a red cross in the **Status** column of the dialog), it means the required ECO is not possible, for example, a component or a footprint could not be found because the required library is not loaded.



In the ECO above, I have some invalid changes which won't be carried out when the ECO is executed. Rectify these errors before executing the ECO.

Resolving Synchronization Errors

Most problems with synchronizing a design generally fall into two categories:

Missing component footprints.

- Common causes:
  - A footprint model is missing from the component information in the schematic.
  - You have forgotten to add the required PCB libraries to the currently available libraries.
  - The component in the schematic does not match any PCB library component.

Footprint pin numbers not matched to schematic pin numbers.

- Common cause:
  - Altium Designer supports user-definable pin-to-pad mapping, the default behaviour is to expect the same number/letter on both sides. Pin-to-pad mapping is defined in the PCB Model dialog (edit the schematic symbol, select the footprint in the Models region of the dialog, and click Edit).

If you get any of these errors, check to see if any of the common issues are a fix for your design.

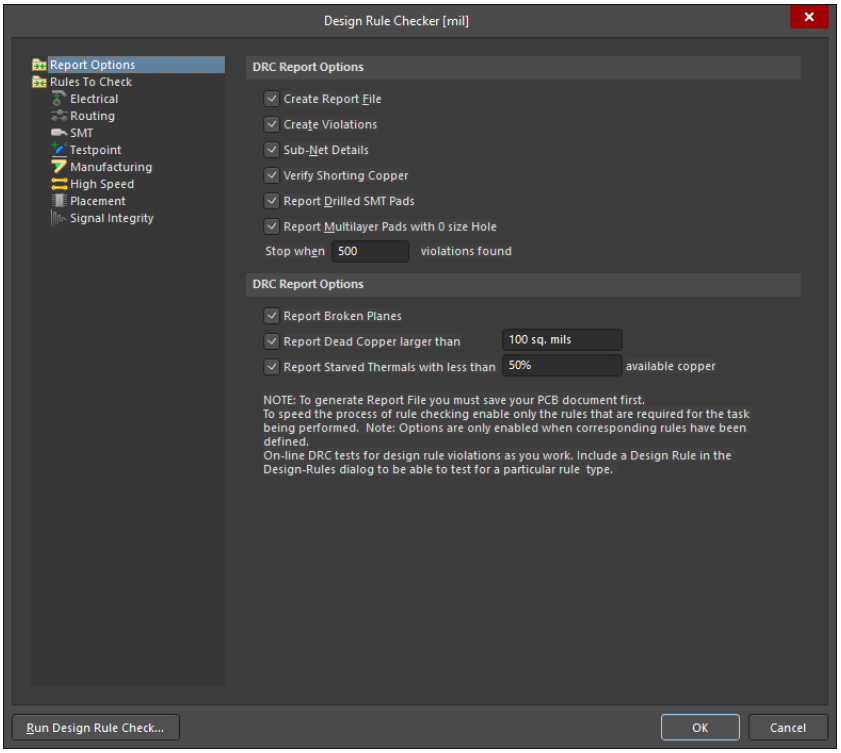
Carrying Out a Design Rule Check

Once you are happy that your schematic and PCB designs are fully synchronised you can continue to carry out a Design Rule Check.

Design Rule Checking (DRC) is a tool that checks both the logical and physical integrity of your design. Checks are made against any or all enabled design rules and can be made online, as you work, and/or as a batch check, with results listed in the **Messages** panel and a generated report. This feature should be used on every routed board to confirm that minimum clearance rules have been maintained and that there are no other design violations. It is particularly recommended that you perform a design rule check after importing a design into Altium from other EDA packages in case any design rules are conflicting.

Some translations my not support the import of your existing design rules, in which case you may need to set new constraints for the imported board. For information on creating design rules please take a look at Constraining the Design - Design Rules

To carry out a DRC, switch to your PCB Document and go to **Tools » Design Rule Check.**



The design rule checker dialog will open, here you can configure the DRC and decide on which rules you would like to be checked.

Click the **Run Design Rule Check** when you are happy with the configuration.

If you checked the '**Create Report File'** option when configuring your DRC, then a verification report will be generated listing all rule violations in your design.

For more information on performing a DRC look at the Design Rule Checking documentation



If you have a lot of **Un-Routed Net Constraint** errors, try running **Design » Netlist » Update Free Primitives from Component Pads** This command is used to resynchronize the net name of the routing primitives to the net name on the pads they connect to.

After resolving any rule violations, your design has now been successfully imported into Altium Designer!

See Also

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

- If you are new to Altium Designer this tutorial takes you through the entire design process, from idea to outputting your manufacturing files. From Idea to Manufacture - Driving a PCB Design through Altium Designer.
- A look at how connectively in handled in Altium Designer. Creating Connectivity.
- For a tutorial that steps you through all 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, read Schematic Placement and Editing Techniques and Placing and Editing PCB Objects.