

Altium
Designer



EAGLE Migration Guide



Next Generation Electronic Design Solutions

This application note shows how to treat Design Files in Altium Designer being exported from CadSoft's EAGLE CAE Tool in the Protel/Altium format. It will mention as well some differences in the Tools which will help you to get a high quality conversion. The result will be a project having no limitations in comparison to native Altium Designer Files.

The conversion will follow these Steps which are described in detail in the sections following:

- 1) Export design data from EAGLE
 - Schematic
 - PCB

- 2) Create Project in Altium and add design Data
 - Adjust Project settings
 - Add Schematic(s)
 - ◆ Do manual adjustments
 - Add PCB
 - ◆ Do manual adjustments

- 3) Add finish to Altium Designer Project
 - Synchronize Design Data
 - Save revised files

1) File export

Exporting Schematics and PCBs (*.Sch and *.Brd design data) from EAGLE to the Protel/Altium Format is quite easy. You simply have to open the according EAGLE project or the individual design file and start the appropriate ULP either from the menu or from the workspace icons.

Matching ULPs like eagle2AD_sch.ulp and export-protelpcb.ulp can be found on CadSoft's webpage or in Altium's forums.

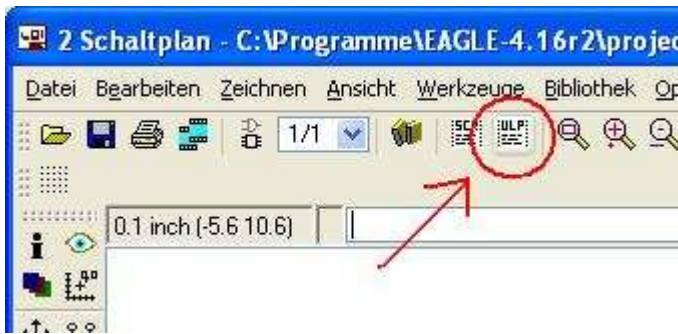


Figure 1: Starting an ULP

To get a result easy to work with it is wise to create a projects folder which collects all projects data.

Schematic export

To export the schematic(s) use the appropriate ULP and you will receive as many schematic Documents as your design does obtain schematic pages. They will be numbered like those in the original EAGLE schematic. As Altium Designer uses 10mils as minimum grid in the schematic, please make sure to obey this in EAGLE as well.

PCB export

Exporting PCB is done by another ULP and needs some manual care: the layer assignment has to be reprogrammed if the default setup is not suitable. Watch out for the layer naming section in the ULP:

```
// Layers Naming
B.layers (L) layer [L.number] = L.name;

layer [LAYER_TOP]      = "TOP";
layer [LAYER_BOTTOM]  = "BOTTOM";
layer [LAYER_PADS]    = "MULTILAYER";
...
```

Due to different system capabilities, watch out for elements where use is made of flat caption for track or circle ends, Altium Designer will not support those and replace them with round caption automatically. The ULP will generate the export in Protel format, Altium is capable of reading this directly but it is recommended to resave it later on in native Altium binary format. You will be guided through this.

2) File Import, generating Altium Designer Project

After the generation of the Schematic document(s) and the PCB, the files can directly be loaded and used by Altium Designer. To be able to continue development or revise your EAGLE project and use the full capabilities of Altium Designer some further steps should be done. Summarized you have to generate a Project. Guidance is provided in this section.

Generating Project

After Altium Designer is started, you have to create a new PCB project. This can be done in multiple ways, one is to use the menu **File >> New >> Project >> PCB Project**. In the Project panel a new Project should pop up, its name will be adjusted when we save it.

In the **Project Options** Dialog that can be called by **Project >> Project Options** some limitations should be adjusted to enable Altium Designer to work in a similar way to EAGLE.

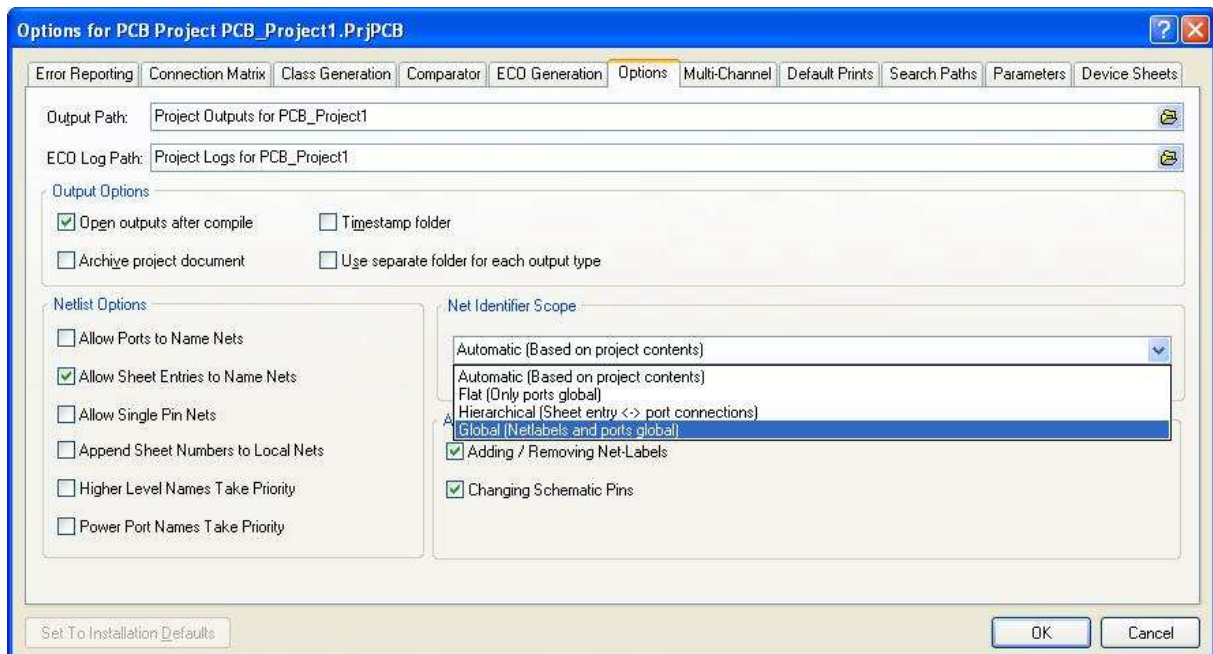


Figure 2: Project Options

Uncheck **Component Classes** and **Generate Rooms** in the **Class Generation Tab**. Set the **Net Identifier Scope** in the **Options Tab** to **Global**.

All other Options can stay with their default setup.

After closing the Project Options Dialog the already exported schematic and PCB data has to be added to the project. For doing this right-click on the project in the **Projects Panel** and choose **Add existing to Project ...**. In the appearing Dialog choose the relevant data and open them. As usual in windows, multiple documents can and should, be chosen at a time.

Maintaining Schematic

Due to some architectural differences in Altium Designer and EAGLE now some design corrections have to be made manually:

Reset the components unique IDs. This is done quite simply, just start **Tools >> Convert >> Reset Component Unique IDs** in the schematic editor. Those IDs are the link between components in the schematic and the PCB, for this the IDs in the PCB will later on be synchronized to those from the schematic.

Look for automatic junctions and get rid of them. Altium Designer automatically adds junction points to T-junctions. In the default setup those junctions are solid blue circles, other than the manual placed ones being red.

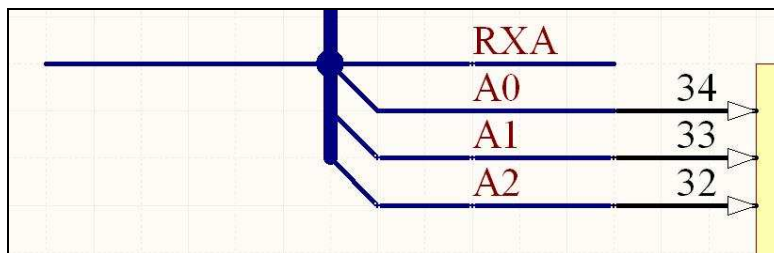


Figure 3: Automatic junction causing shortcut

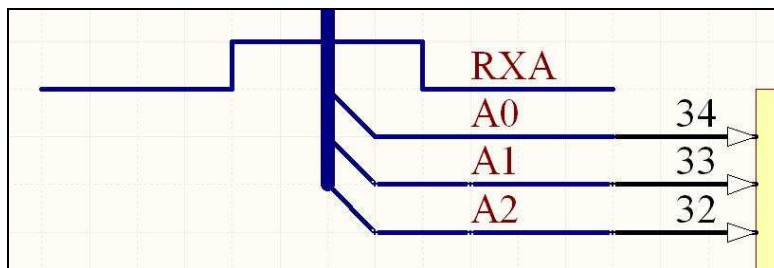


Figure 4: Fix for the unintended shortcut

To get a proper synchronization, make sure no floating **net labels** are around. For this you can use the ERC which will be started when you compile the Project (right mouse-click on the project in the Projects Panel >> Compile ...). If the messages panel does not open automatically, it can be activated over the menu **View >> Workspace Panels >> System >> Messages**.

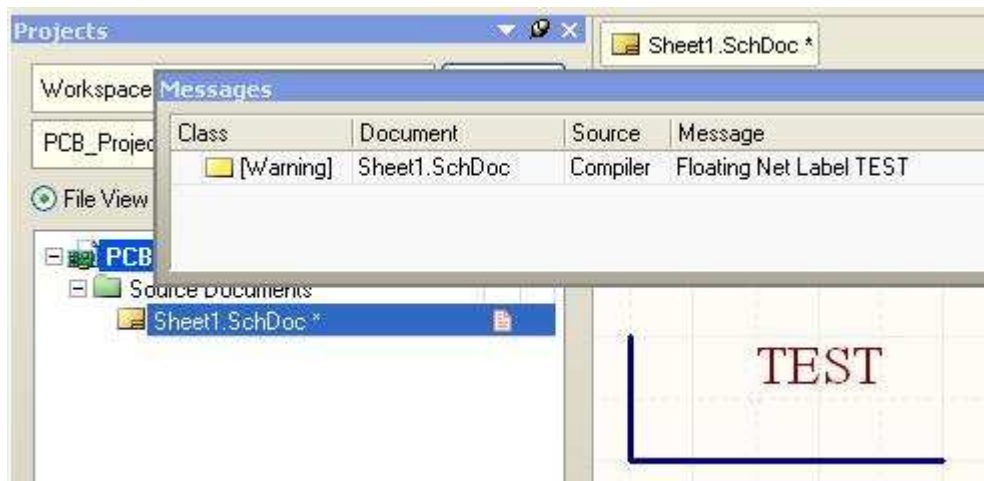


Figure 5: Floating Net label and according Warning

Beside the possible floating net label messages you can and should have a view to all other messages and resolve the issues.

Regarding two design objects you can spend some additional but optional work: **Power Ports** and **Schematic Templates** (EAGLE: frames). In EAGLE those are common parts, in Altium Designer they get a special treatment. To get rid of some warnings later on when doing the synchronization between schematic and PCB you can replace the EAGLE power parts by Altium's power ports and remove the schematic frame. Again, the adding of a native Altium schematic template is optional.

Maintaining PCB

When you open the PCB File the **Protel Import Wizard** will start automatically. This is because the PCB export script writes in the Protel ASCII format. Simply follow the instructions; the correct board shape can be adjusted after import as well.

Polygons, Rules and component classes are not exported. So, if necessary redefine those items.

As mentioned before, the link between schematic and PCB components is defined by so called Unique IDs. Those will be defined for the imported PCB through the function **Project >> Component Links ...**. Use the Button **Add Pairs matched by** (Designator checked) and **Perform Update** to do the PCB update. If there are single components left on the schematic side you might not have replaced the power ports and schematic frames. You do not have to care about them mandatory, just ignore the warnings.

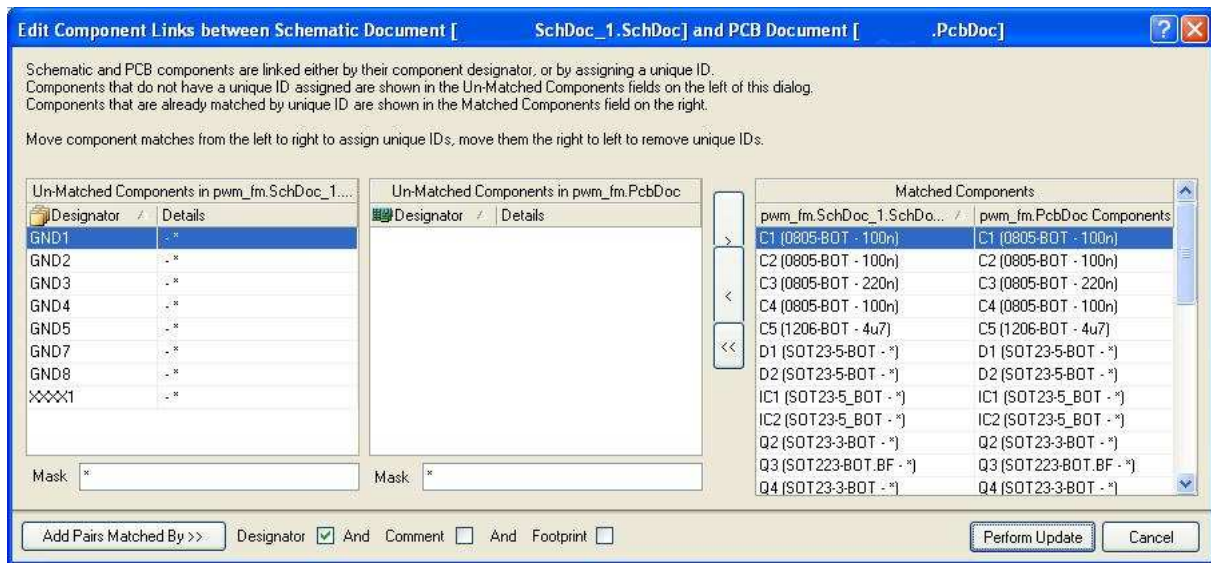


Figure 6: Linking components, some single components stay

Now that you have spent some time in reworking the PCB you should save it in native Altium binary format. For this right-click on the PCB in the Projects Panel and choose **Save as ...** . Adjust PCB Binary Files and save the file, the old PCB data will automatically be replaced in the Project.

3) Synchronizing Schematics and PCB, save your work

Last two steps are synchronizing the project and saving it.

The synchronization can be started from the schematic editor (**Design >> Update PCB Document ...**) or the PCB editor (**Design >> Import Changes From ...**).

After this step the Project must be saved, best location is the folder where the schematic and PCB data already have been stored. To do so, right-click on all documents showing the red symbol for unsaved changes and save them.

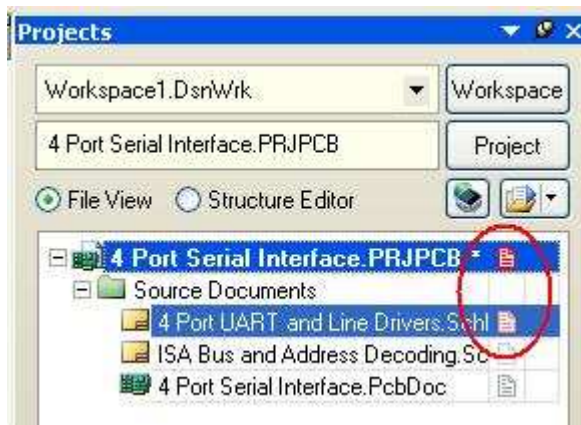


Figure 7: Unsaved documents

Now you are done.

Last but not at least:

This process only works for design data, not for libraries. But you can easily extract libraries from an Altium Designer Project when you activate a schematic page of the project and start the **Design >> Make integrated Library** function.