



Altium

How to Simplify PCB Design



HOW TO SIMPLIFY PCB DESIGN

1. HOW TO SIMPLIFY AND AUTOMATE YOUR PCB ECO WORKFLOW
2. HOW TO SIMPLIFY ROUTING WITH PIN SWAPPING
3. HOW TO SIMPLIFY OUTPUT GENERATION WORKFLOW
4. HOW TO SIMPLIFY CIRCUIT REPLICATION WITH MULTI-CHANNEL DESIGN

**1**

How to Simplify and Automate your PCB Eco Workflow

How to Simplify and Automate your PCB Eco Workflow

Component links can increase productivity by automating ECO procedures.

Altium Designer enables automation when generating ECOs by unifying schematic editor and PCB layout with component links that seamlessly connect the two.

To establish the connection between the schematic and the PCB, Altium Designer automatically assigns a Unique ID to any symbol placed in design.



[See original content here](#)

How to Simplify and Automate your PCB Eco Workflow

Component links can increase productivity by automating ECO procedures.

With component links, you can:

- Automatically transfer data back-and-forth from schematic to PCB layout.
- Easily perform ECOs without having to import and export your design.
- Handle all aspects of your design in a single, unified environment.



[See original content here](#)

**2**

How to Simplify Routing with Pin Swapping

How to Simplify Routing with Pin Swapping

Optimal component placement goes a long way towards minimizing crossover connection lines. However, crossovers can never be completely avoided.

It is common for PCB designers too, wherever electrically possible, swap net assignments from one device pin to another eligible device pin. Similarly, sub-parts within a common package can be swapped to reduce crossover connections. Pin swapping relies on the fact that nets of two different physical pins can be swapped without having any negative impact on the electrical functionality of the design.

Device pin and sub-part swapping greatly help in reducing the overall number of crossover connections in a PCB. Furthermore, once pin or part swaps are made within the PCB design, the schematic must be updated to reflect the changes and remain synchronized with the PCB layout.



[See original content here](#)

**3**

How to Simplify Output Generation Workflow

How to Simplify Output Generation Workflow

Handing the design to manufacturing is a significant part of the project organizing and assembling the set of files that must be provided to the manufacturer for fabrication and assembly.

Output Generation - The typical files required are the Bill of Materials (BOM), Gerber files, NC drill files, ODB++ files, IPC-2581 files, pick and place files, and assembly drawings. It's important to establish a consistent work flow, so that it is easy to locate, organize, and assemble the required documentation.

Standardizing Outputs - An Output Job Configuration file can be added to any project and becomes the one source where all required outputs can be generated. Setup an Output Job Configuration file that has all the required outputs ready for generation.

Configuring and Reusing Outputs - Many of the delays involved with the PCB design to manufacturing process can be avoided with accurate data and documented notes for fabrication and assembly.



[See original content here](#)

**4**

How to Simplify Circuit Replication with Multi-channel Design

How to Simplify Circuit Replication with Multi-channel Design

The main advantage of multi-channel design is that any change to the underlying circuit need only be made once. And that change will immediately be seen in every instance.

Flat Designs Using Multiple Sheets

Step 1 - Schematic Creation Replication - Start by creating the initial circuit on the first schematic sheet of a PCB project and then add a second, empty schematic sheet to the Project.

Step 2 - Project Options Setup - The next step is to set the Project Options to automate the Component Class and Room generation.

Step 3 - PCB Layout - The final step is to create and save a new PCB file, then use Design/Import Changes to populate the board.



[See original content here](#)

How to Simplify Circuit Replication with Multi-channel Design

The main advantage of multi-channel design is that any change to the underlying circuit need only be made once. And that change will immediately be seen in every instance.

Flat Designs Using a Single Sheet

A second multi-channel situation makes use of a much smaller circuit, copied and pasted many times within the same sheet. Though, this method requires a few more manual steps in order for Copy Room Formats to function correctly.



[See original content here](#)



ALTIUM
DESIGNER18



HOW TO SIMPLIFY PCB DESIGN

Altium Designer 18

New to Altium Designer?
Don't be!

Try the latest release and get time saving workflows and enhanced team
collaboration options today at:

www.Altium.com/free-trials



Altium

Thanks for your attention!