6 tips for transferring a PCB schematic to layout

Mahmoud Wahby - March 27, 2014

PCB Design Best Practices: Six things to consider when transferring a PCB schematic to layout. All the examples in this article are developed using the NI Multisim design environment, however the same concepts apply when using different EDA tools.

Initial Schematic Transfer
Transferring the schematic to the layout environment through the netlist file will also transfer the part information, netlist, layer information, and initial trace widths settings.

Here are some recommended steps for preparing for the layout phase:

1. Set the Grids & Units to desirable values. For finer placement control on objects and traces, the Part grid, Copper grid, Via grid and SMD grid can be set to 1 mil.
2. Set the Board outline clearance and Via support as required. PCB Manufacturers may have specific minimum or nominal recommendations for blind and buried via settings.
3. Set the Pads/Vias options according to the PCB manufacturer. Smaller vias with Drill diameter of 10 mils and Pad diameter of 20 mils can safely be used with most PCB manufacturers.
4. Set the design rules as desired.
5. Set custom shortcuts for the Favorite layers to quickly allow layer changes (and via creation) while routing.

Handling Errors in Schematic Transfer
One error that is common during the schematic transfer is nonexistent or incorrect footprint assignments. Some things to note:

- If a part in the schematic is missing a footprint, a warning message will indicate that the virtual component will not be exported. In this case, no default footprint information is transferred to layout and the component is simply removed from the layout.
- If a footprint is transferred but is not correctly matched to a valid footprint shape, a warning message will occur during the transfer to indicate the mismatch.
- Correct the footprint assignments in your schematic or create a valid footprint for any parts. When corrected, perform a forward annotation step to update and synchronize the design information.

Updating the Design through Annotation
Annotation is the transfer of design changes from schematic to layout, or layout to schematic. Both back annotation (layout to schematic) and forward annotation (schematic to layout) are critical to maintain design accuracy.

In order to safeguard the completed work, backup and archive the current revision schematic and
layout files prior to any major forward or back annotation steps.

Do not make simultaneous changes in both the schematic and the layout. Make changes in one part of the design only (either in the schematic or layout) and then perform the correct annotation step to synchronize the design data.

**Renumbering parts**
Part renumbering is a feature allowing the components on the PCB to be renumbered in a specific sequence. Reference designators should be ordered from top to bottom and in a left to right direction on the PCB. This increases the ease of locating parts on the boards during assembly, test, and troubleshooting.

**Handling Last Minute Part or Netlist Changes**
Last minute PCB part or netlist changes are undesirable but sometimes are needed due to part availability or detection of last minute design errors. If changes are needed in components or the netlist, these should be originated in the schematic and then forward annotated to the layout tool.

Some tips:

1. If adding a new part (such as a pull-up resistor on an open-drain output) after the layout commences, add the resistor and net to the design from the schematic. After forward annotating, the resistor will be shown outside the board outline as an unplaced component and the ratsnest will be shown indicating the connecting nets. Place the component inside the board outline and route as normal.

2. Back annotation works best with reference designator changes, such as post layout renumbering.

**Highlight Selection for Locating Parts**
While laying out the PCB, a technique for viewing a specific component or trace in the schematic is to use a 'Highlight Selection' feature. This feature allows you to select a component or a trace (or multiple objects) and reviewing where they lie in the schematic.

This is especially useful when matching up bypass capacitors to their intended IC connections. Conversely, it is possible to locate particular components or traces in the layout when viewing the schematic.

**Also See:**

- 5 tips for PCB schematic file management
- 6 tips for choosing PCB components
- Tips for PCB design management