

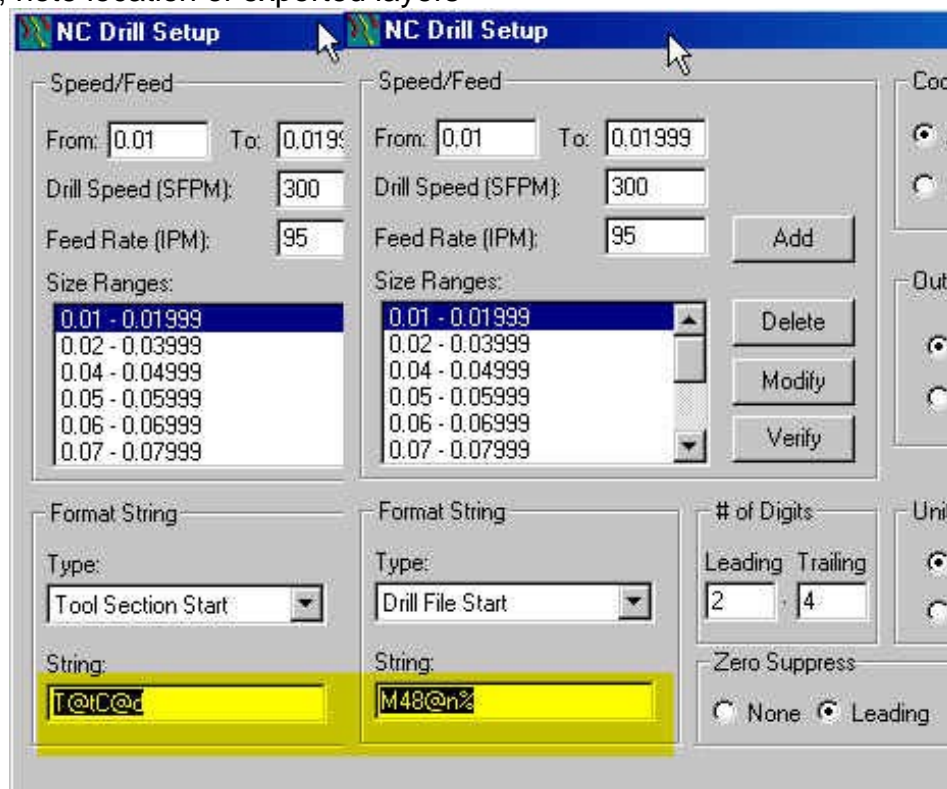
Netlist Comparison in CAM350

Gerber and Drill export from Power PCB

In Power PCB

File|CAM

- 1) If you use augment on the fly, hit regenerate before exporting Gerbers to insure flashed endpoints.
- 2) Use offset as your default method of registration to insure all layers are aligned. Use the same offset distance for the Drill file as well.
- 3) In the device setup for drill files, under Format String, choose Drill File Start. Type in this string M48@n%
- 4) In the same dialogue, choose Tool Section Start. Type in this string T@tC@d
Note: If units to be used are English, then units under Setup|Preferences should be set to inches, not mils.
- 5) Choose layers to export, Note: Only electrical and drill layers are required at this Point.
- 6) Run, note location of exported layers



Generation of an IPC Netlist from Power PCB

In Power PCB, load the PCB File.

Tools/CAM350

- 1) Use defaults, Write down the location of .CAM file (Note: There is a bug in older versions of Power PCB that will crash the application if you change the path.)
- 2) You have 2 choices; create the file only, or create and launch CAM350.
- 3) once the file is loaded in CAM350,note the location of a pin 1 or tooling hole to issue proper registration to the Gerber layers.

In CAM350, File|Export|Netlist

- 1) Choose IPC-D-356 or IPC-D-356A. IPC-D-356 is ok for the purposes of Netlist comparison in most cases.
- 2) Note location of the saved file.

Importing Gerbers into CAM350

In CAM350

- 1) File|Import|Auto Import, choose directory which contains the Gerber files
- 2) Hit Next
- 3) If format other than 2.4 English or 3.3 Metric was used for the drill file, you will need to manually change those under format.
- 4) Verify that the correct files are to be loaded, and hit Finish.
- 5) If drill string above was not used in Power PCB, and you get an error reading the drill sizes, run drill script pads_drill.scr to identify non-plates and correct the actual drill sizes.
- 6) Drill sizes and plating characteristics can be verified under Tables|Nc Tool Tables.
- 7) Insure endpoints are flashed, by using contrasting colors in the layer bar.
- 8) In Tables|Layers, label the copper layers
- 9) Go to Info|Query|All, and select the same Pin 1 or tooling hole to insure Gerbers are in the same location as the IPC-d-356 Netlist.

Extracting the Netlist from the Gerbers

- 1) Go to Utilities|Netlist Extract, and accept the defaults.
- 2) Hit OK

Importing and Comparing the IPC-d-356 Netlist to the Gerber extracted Netlist.

- 1) Go to File|Import|IPC, and choose the correct format, 356 or 356a depending on what you chose when outputting the CAD Generated IPC Netlist.
- 2) Go to Analysis|Nets|Compare External Nets to compare the two Netlists.
- 3) Errors are viewed in the Error Control Bar, Click View to see all of the errors or to jump to a specific error.
- 4) If you exit the error control bar, you can retrieve it through Info|Find|Net Compare Errors.