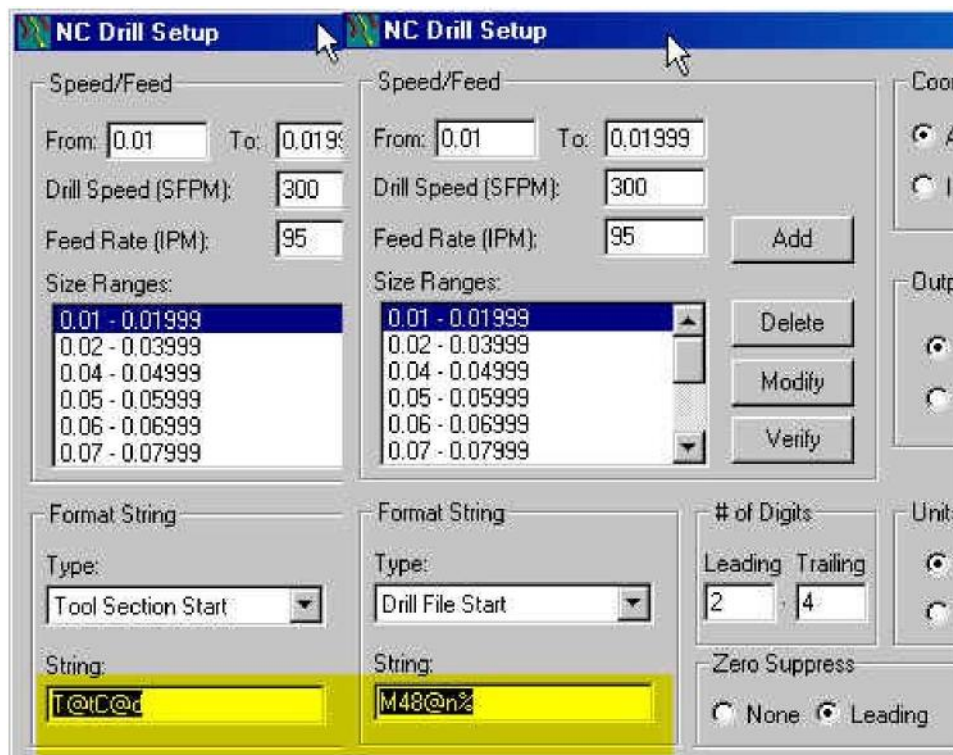




Netlist Comparison in CAM350

Gerber and Drill export from Power PCB

- 1) If you use augment on the fly, hit regenerate before exporting Gerber layers 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. Recommended is X10 Y10 if working in inches.
- 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, and Run. Note: Only electrical and drill layers are required at this point.





Generation of an IPC Netlist from Power PCB

In Power PCB, load the PCB File.

- 1) Tools/CAM350
- 2) Use default settings.
- 3) You have 2 choices; create the file only, or create and launch CAM350.
- 4) 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.
- 2) Note location of the saved file.

Alternative Methods of creating IPC-D-356 netlists.

ASCII method

- 1) Export ASCII file from Power PCB (File|Export)
- 2) Import ASCII into CAM350 (File|Import|Cad Data|Power PCB)
- 3) Export Netlist as above

Script Method (Validity not verified)

- 1) Use the script that is available at many Power PCB related sites on the Internet to create the IPC-D-356 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 offset was used in Power PCB, move the gerbers and drill files to the original location.
For Example: if an offset of X10 and Y10 was used, you must select all the gerbers and drills and move them x-10 and y -10, by using the relative option in the coordinate pop-up.
- 6) If drill string above was not used in Power PCB, and you get an error reading the drill sizes, run the script "pads_drill.scr" to identify non-plates and correct the actual drill sizes.
- 7) Drill sizes and plating characteristics can be verified under Tables|Nc Tool Tables.
- 8) Insure endpoints are flashed, by using contrasting colors in the layer bar.
- 9) In Tables Layers, label the copper layers as to type. (Top, Bottom, etc...)

Extracting the Netlist from the Gerbers

- 1) Go to Utilities|Netlist Extract, and accept the defaults.
- 2) Click 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) Under Settings|View Options, choose view imported nets
- 3) Net points should be registered to the center of pads
- 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.