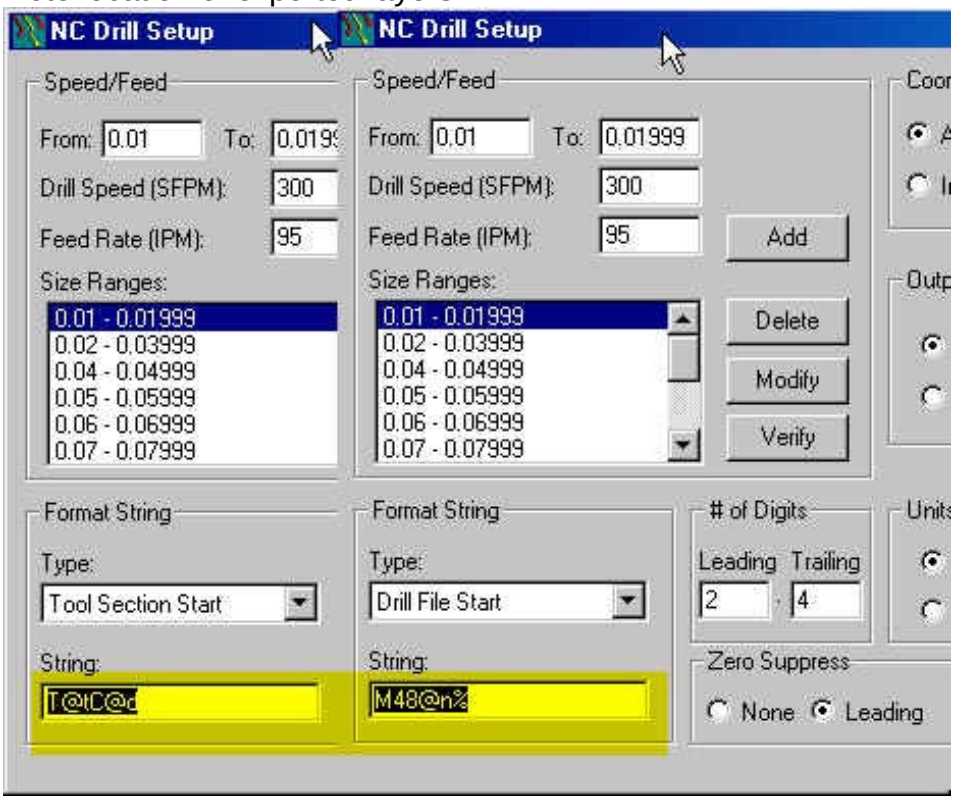
Netlist Comparison in CAM350

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.