



## Recommended Post Processing Procedure using Eagle Layout

Note: The Eagle Tutorial and Manual .pdf files are recommended reading.

1. Create Schematic

2. Layout Board

It is PEDL's recommendation that the following endmill and drill sizes be as closely adhered to as is possible to ensure consistency between the designed board and the actual finished board.

Mills: .011", .015", .020", .031" (Choose .031" ground plane isolation if possible)

Drills: .032", .040", .0595"

If larger holes are required for mounting certain components a pilot hole will be substituted, to allow for precise locating of hole after milling is complete.

3. When adding text to the board, do so on a separate layer (#46 Milling is recommended). This will allow for a much better-looking and easier to create text on the finished board as the letters will be milled from the copper, as opposed to the copper around the text being removed.

4. CAM generation of "Gerber RS274X" file for milling:

File/Open/Job Gerb274X

Layers: one side per file, e.g. top, vias, pads (.cmp is for component side)

5. Drill tool file generation; ULP button to run DRILLCFG.ULP

6. CAM generation of "Excellon" Drill & holes file;

File/Open/Job Excellon

It is of great importance that the drill file be in the Excellon (.ncd, .dr\*) format – if it is not the mill will be unable to drill the component lead holes. These files will be small (~2k) in comparison to the Gerber files (.cmp) created.

In ISOpro:

7. Import Gerber file into 1<sup>st</sup> layer.

8. Import drill file (\*.drc) into 2<sup>nd</sup> layer.

If errors result (ie. the holes do not line up with the traces, or of the wrong scale), adjust the digits of precision. The defaults are usually reliable, but Integer = 2 and Decimal = 4 or Decimal = 3 are also good choices.

11. Import text label Gerber file (#46 Milling from step #3)

12. From here it is standard ISOpro procedures for isolating, rubbing out, milling, etc.