

Creating a Tooling Package

Before submitting your board designs, please review the following items:

Fabrication Data

- Inner/Outer Layer Data -- please include all copper layers
- Solder Mask Layers
- Legend Layers
- Aperture List -- when applicable
- Drill file(s) Excellon 1 or 2 format, -- include units (inch or mm), file format (e.g., 2:3, 2:4) tool codes and XY coordinates (ASCII or EIA format)
- IPC 356A netlist or Mentor Graphics neutral file. Include netlist exceptions list for known design shorts or opens.
- Readme file containing design layers descriptions, any other special instructions and a primary engineering contact

Fabrication Drawings

- Drawings in Gerber RS-274X, HPGL1 or 2, Adobe PDF, DXF format, AutoCad, Tif/Tiff, Postscript or Hard Copy
- Board Outline Dimensions -- cutouts, chamfers, radii, bevels, scores, etc.
- Dimensions -- either from a reference hole to a corner or to two sides of the board outline
- Hole Chart -- hole symbols on the drawing and finished hole sizes
- Material Requirements
- Finished Board Thickness & Tolerance
- Layer Stack-up Order
- Controlled Impedance Requirements. Identify layers to be controlled and specific trace/space geometries to be controlled (design these traces unique from other non-controlled trace widths if possible).
- Dimensioned Array Drawing -- if applicable
- Special Instructions -- if applicable

Preferred Data Formats

"IF POSSIBLE SEND ONLY ONE FORMAT"

- Valor ODB++
- Gerber RS-274X
- Gerber RS-274-D with Aperture Wheel

Technical Support Contacts

John Tebbe – Chaska, MN -- (952) 442-2189