General Manufacturability Guidelines

These international guidelines developed by the Association Connecting Electronics Industry or IPC have helped standardize the assembly and production of electronic equipment and assemblies. Following these guidelines will allow for a smoother quoting and manufacturing process.

Bill of Materials (BOM)

A normal BOM is used to purchase parts, an Assembly BOM has more production information in it and is critical for quick-turn and time-sensitive jobs.
The Assembly BOM should be saved in the Microsoft Excel Format.

Please ensure there is a manufacturer's part number on every item in the BOM.

While not required it can be helpful to list acceptable substitutions for components in the BOM in case of potential conflict. Please indicate if a component is "critical" and no substitutions are allowed.

Assembly BOM Requirements

- Assembly Name & Revision Number
- Microsoft Excel format
- Quantity of Manufacturer's Part Number
- Reference Designators of Components
- Manufacturer's Part Numbers for each component
- Indicate if any components can't be substituted
- Component Value
- Component Package/decal
- Part parametric information
- Short Description
- Bare Board Part Number & Revision Number

Gerber File Requirements

Include complete industry-standard Gerber files. Gerber files are a guide to building a stacked layered PCB. The companion aperture files specify which tools to use when making your PCB.

Gerber File Requirements

- Gerber file format RS-274X
- Include ASCII pick-and-place files
- Include NC Drill Files & Drill Chart

The **RS-274X** standard is preferred for Gerber files as this file format automatically assigns the D-Code aperture settings and includes other helpful meta-information.

The ASCII pick-and-place file should contain accurate component placement information.

Remember to include the XY data file as this information is critical to SMT placement.

Customer supplied component requirements

- The customer is responsible for providing RoHS compliant Bill of Materials if the assembly requires it. If required, Pro-Active Engineering will build assemblies in accordance to RoHS compliant processes.
- It is recommended that the customer send extra parts due to leaders and/or attrition during the PCB assembly process. Pro-Active does not experience much loss, but to be safe Pro-Active would appreciate having a small amount of overages to avoid any delays in assembly.
- Lead time may be impacted if the proper leader and/or attrition is not supplied. If you have expensive components or a minimum quantity of any component on hand, please contact your sales representative for details.
- Parts that are properly shipped in anti-static carriers or in moisture sensitive packaging will be treated with care while being inventoried, placed, and resealed. Any unused Customer supplied parts are returned with your assembly.
- Component Packaging Requirements
    - Each component shall be individually packaged, labeled with the BOM line item number, customer part number and/or manufacturers' part number.
    - The following are acceptable packages of consignment / customer supplied components:
        - SMT Reels
        - Cut Tape
        - Thru-Hole bulk or tape
        - Tubes / Trays
        - Digi-Key Reels
    - SMT components, not supplied on reels, are required to be on one continuous strip of tape.
- **Bare PCB's**
    - We recommend having a silkscreen on your PCB that indicates polarity and 1st-pin markings.
    - All bare PCB's require a ¼ inch rail on all sides to accommodate machine handling.
    - A minimum of two fiducials are required for accurate component alignment and placement.
    - Minimum Array Size is 3" x 3"
    - Maximum Array Size is 17" x 17"
- **Overage Guideline**
    - Passive Devices (Resistors, Capacitors, and Diodes)
        - 0603 and larger footprints: Add 10% and/or 20 pieces to build quantity. One continuous tape strip required (Splicing of tape is not permitted).
        - 0402 and 0201 footprints: Add 50 % and/or 50 pieces to build quantity. One continuous tape strip required (Splicing of tape is not permitted).
        - All others add 20% overage.
    - IC's:
        - Under $1.00 - Add 5% overage, or one piece. Whichever is greater.
        - $1.00 - $5.00 - Add one piece.
        - Over $5.00 - No extra is required.
        - If IC's are in tape media, One continuous piece of tape is required (Splicing of tape is not permitted),
    - Thru-Hole
        - A minimum of one thru-Hole or one percent whichever is greater. (Large volume jobs add to 2-3% overage on inexpensive items .50 cents and less)

PCB Fabrication Specifications

To best meet the needs of each
individual customer, we offer a wide range of options for PCB Fabrication.

If not specified Pro-Active Engineering will default to the specifications indicated in the right column in the table below this text.

PCB Fabrication Specifications	Default Settings
Board Size:	
PCB Layers:	
PCB Material (Please Specify)	Default: FR4
TG Rating:	Default: 130 or Higher
Thickness:	Default: 0.062”
Finished Outer Layers Copper:	Default: 1oz
Finished Inner Layers Copper:	Default: 1oz
Solder Mask Color:	Default: GREEN
Silk Screen Color:	Default: WHITE
Finish/Plating:	Default: LF-HASL or ENIG
RoHS Compliance Required:	Default: Yes
Class 2 Manufacturing Acceptable: (Class 3 optional)	Default: Class 2
Any other items to note about PCB?	Impedance Control Required, Blind/Buried Vias, Specific Stack-up Specs?

CAD File Formats

We can accommodate virtually all CAD files from nearly all major software packages.
Below you will find a list of all CAD file software programs and the file extension these files are saved in.
If you have any concerns about a specific software package, we'll be glad to help you at quoting@proactivepcb.com 608-837-7838.

CAD System	Standard Extension	CAD System	Standard Extension
Accel P-CAD	.pdf	McCAD	.ipc
Accel P-CAD	.ipc	McCAD	.cad
Accel P-CAD	.pcb	McCAD	.tbs
Altium Designer	.pro	MentorBoardStation	.neu
Altium Designer	.pcb	MentorBoardStation	.txt
Altium Designer	.pcbdoc	Mentor Expedition	.cad
AutoCAD	.dxf	Mentor Expedition	.ipc
Bartels Auto Engineer	.ipc	ODB++	varies
Bartels Auto Engineer	.cad	OrCAD Masstek	.min
Cadence-Valid Allegro	.ipc	OrCAD Masstek	.cad
CADSTAR	.paf	PADS	.asc
CADSTAR	.cdi	P-CAD	.pdf
CIRCAD	.cad	P-CAD	.ipc
CIRCAD	.ipc	P-CAD	.pcb
ComputerVision	.cvi	Pulsonix	.cad
Diptrace	.asc	Protel	.pcb
Eagle CAD	.ipc	Protel	.ipc
Eagle CAD	.cad	SCI Card	.cii
Eagle CAD	.fatf	Seetrax Ranger	.cad
Easy-PC	.cad	SuperMax ECAD	.ipc
EE Designer	.ala	Tango (Protel)	.pcb
Fabmaster	.fatf	Theda	.tl
GENCAD	.cad	ULTIboard	.cad
GENCAM	.gcm	ULTIboard	.ipc
Gerber	.ger/grb/art/pho	Veribest	.vrb
HPGL	.hpg	Veribest	.cad
Incases	.tl	Visula	.paf
Intergraph	.lst	X-Y Component Center	.xyr
KiCAD	.cad	Zuken	.paf