



For a definition of terms and more information (underlined text), check out our [PCB Glossary](#). Accepted "PCBpro.com" file formats: Gerber with Excellon Drill file, or Ivex '*.brd' file. All files should be sent in a zip-format. A [readme file](#) is recommended.

Information highlighted in **BLUE** should be in the board layout design to ensure a working board.

Category	Description	PCBpro Capability
Files can be	Gerber files - preferred format	274-X (embedded apertures)
	Gerber files - acceptable format	274-D, with one aperture list
	Gerber files - extra cost possible	274-D, with multiple aperture lists
	Excellon drill file	X & Y coordinates, with tool sizes included.
Material is FR4	Bow and twist	0.0075"
	Dielectric spacing	0.005" min
	Starting thickness	within +/- 0.007"
Layer Registration	Layer to layer	within .010" (+/- .005" each layer)
Copper Plating	Minimum in holes	0.0008"
	Minimum on surface	0.0008"
Inner layers	Minimum clearances	Hole size + .025"
	Minimum back set	Copper .025" from cut line
	Minimum pad size	Hole size + .018"
	Minimum traces and spacing	As per order form (ex. 0.006" with 1-oz. copper)
Outer layer	Minimum back set	Copper .010" from cut line
	Minimum pad size (non-via)	Hole size + .017"
	Minimum via pad size	Hole size + 0.010"
	Minimum traces and spacing	As per order form (ex. 0.006" with 1-oz. copper)
Multi Layers	1-4 layers	0.031", 0.047", 0.062", 0.093", & 0.125" Thickness 1,1.5, & 2.5oz Finished Copper Weight
	5-8 layers	0.062", .093", & .125" Thickness 1,1.5, & 2.5oz Finished Copper Weight
Traces	Finished trace size	Within 20% of artwork
	Trace size .006"-.009"	1 oz, Finished Copper Weight
	Trace size .010"-.013"	1, & 1.5oz, Finished Copper Weight
	Trace size > .013"	1,1.5, & 2.5oz Finished Copper Weight
Solder Finish/Holes	Hot Air Leveling (used with solder mask)	0.0001"- .0015"
	Reflow (used without solder mask)	
	Solder filled holes	Holes drilled less than .020" may be plugged
Solder mask	Mask registration tolerance	Within +/- .005"
	Nominal thickness	0.0004"
	Mask film clearance	Pad size + smallest board spacing
Silkscreen	Marking / text	+/- .010" registration
		Min. 0.007" line required to be visible on board
Electrical Testing	Continuity	10-50 ohm (50 ohm std.)
	Isolation	10-100 Mohm (10 Mohm std.)
	Voltage	10-250V (100 V std.)
	Minimum SMT pitch	0.020"

Drilling	Sizes from 0.010"-0.016" (0 .010" our min.)	0.031", 0.047", 0.062", thickness
	Sizes from 0.017"-0.019"	0.031", 0.047", 0.062", & 0.093" thickness
	Sizes > 0.019"	0.031", 0.047", 0.062", 0.093", & 0.125" thickness
	Finished hole size	Within +/- 0.004" of drill file sizes. Please be aware that below 0.020" may plate shut.
	True position	0.005"
Board Routing	Tolerance board edge & cutouts	Within +/- .010"
	Maximum board size	14" X 18"
	Minimum board size	0.25" X .25"
Tab Routing	Space between boards	0.1", you send a one up image, we do the rest.
	Boarder around pallet	0.5"
	Number of tabs per board	4
	Overall panel size (view sample)	Varies with board size, approx. 12" x 14"
V-Scoring (30 degree)	Space between boards	0.0"
	Boarder around pallet	0.5"
	Overall panel size (view sample)	Varies with board size, approx. 12" x 14"
	Minimum Board Size	1.5" x 1.5"
	Minimum back set	Copper 0.030" from cutline
Gold fingers (30 degree)	Gold plated edge connectors (30 d bevel)	30 micro inch Au, over 100 microinch Ni minimum
Standards	IPC-600 Pictorial guide	We use Class II to determine defects
	UL (Underwriters Laboratory)	When selected on order form

Rev. 1020040809-2



PCBpro PCB Manufacturing Capabilities
<http://www.pcbpro.com>