

North Tech Services

Tel : +27-(0)21 5933560/1

Fax: +27-(0)21 593 2250

email: info@ntspcb.co.za

Manufacturing Capabilities:

Specification	Value
Track-to-Track Clearance	0.20mm (8 mil)
Track Width	0.20mm (8 mil)
Minimum feature to board outline	0.25mm (10 mil)
Silkscreen width	0.20mm (8 mil)
Board outline	0.20mm (8 mil)
Minimum Annular ring size	0.20mm (8 mil)
Solder mask expansion	0.15mm (6 mil)
Minimum via hole size	0.5 mm (20mil)
Minimum hole size	0.6 mm (24mil)
Hole diameter tolerance	<0.05mm (2mil)
Maximum single sided board size	500mm x 400mm
Maximum through hole plated board size	500mm x 400mm

Depending on the design, certain manufacturing guidelines can be exceeded, should you require a printed circuit board with specifications not listed above, please enquire prior to your order.

Product Specifications:

Product	Single Sided, Double Sided Through hole plated
Material	FR4
Copper Thickness	35 micron ; 70 micron
Copper Laminate Thickness	0.8mm 1.0mm 1.6mm 2.4mm
Solder Mask	Dark Green, Blue
Silkscreen Legend	White
Edge Profile	V-score or smoothed
Board Supply	Panel or individual
Inspection	Optical
Single Sided Finish	Roller tin/lead, SMOBC (solder mask over bare copper)
Through Hole Plated Finish	Plated tin/lead
Film imaging resolution	2000dpi - 2400dpi
Data files	RS274-X, RS274-D

Design Guidelines:

Guideline	Minimum	Standard	Recommended
Track-to-Track Clearance	0.20mm (8mil)	0.25mm (10 mil)	0.30mm (12 mil)
Track Width	0.20mm (8 mil)	0.25mm (10 mil)	0.30mm (12 mil)
Minimum feature to board outline	0.25mm (10 mil)	0.30mm (12 mil)	0.30mm (12 mil)
Silkscreen width	0.20mm (8 mil)	0.25mm (10 mil)	0.25mm (10 mil)
Board outline width	0.20mm (8 mil)	0.20mm (8 mil)	0.20mm (8 mil)
Minimum Annular ring size	0.20mm (8 mil)	0.25mm (10 mil)	0.25mm (10 mil)
Solder mask expansion	0.15mm (6 mil)	0.15mm (6 mil)	0.15mm (6 mil)
Via hole size	0.5 mm (20mil)	0.6 mm (24mil)	0.7mm (28mil)

Date Files required:

No:	Name	Description
1.	Drill Drawing	Contains graphical symbols and a drill tool size table
2.	Drill Guide	Contains positions or targets of drill holes
3.	NC Drill File	Contains a drill program for cnc machines
4.	Top Silkscreen *	Contains the component outlines and/or values
5.	Bottom Silkscreen *	Contains the component outlines and/or values
6.	Top Solder Mask *	Contains areas where the solder mask ink should not cover
7.	Bottom Solder Mask *	Contains areas where the solder mask ink should not cover
8.	Top Layer *	Contains the track layer and pads/vias for the top layer
9.	Bottom Layer	Contains the track layer and pads/vias for the bottom layer

* optional depending on your printed circuit board requirement.

All layers are viewed from the top of the printed circuit board, do not mirror any layers.
Check all gerber files with a gerber viewer before submitting them for manufacturing.