

# PCB LAB

ELECTRICAL ENGINEERING DEPARTMENT

INDIAN INSTITUTE OF TECHNOLOGY - BOMBAY



MINIMUM DESIGN REQUIREMENT MANUAL

# PCB LAB

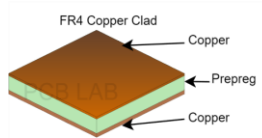
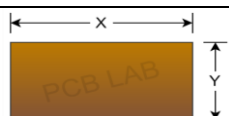
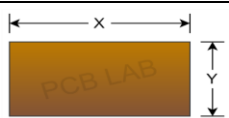
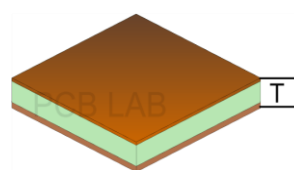
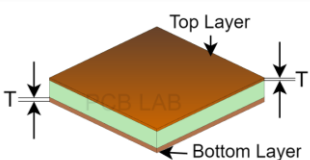
ELECTRICAL ENGINEERING DEPARTMENT, IIT-BOMBAY

## MINIMUM DESIGN REQUIREMENT MANUAL

( By Mr. Yadnik Sonalkar )

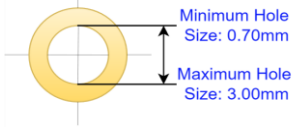
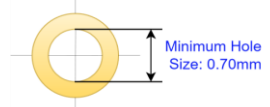
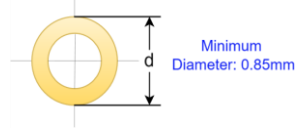
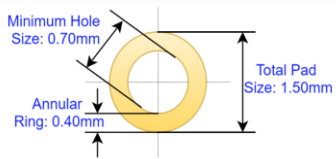



❖ **Minimum design requirements for PCB with Solder Mask and Silkscreen / Legend Print ( Submission of Photo-Film print is mandatory )**

• **PCB Specifications :**

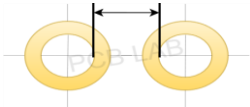
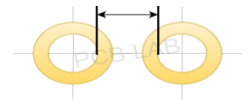
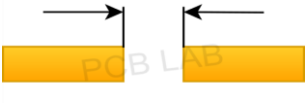
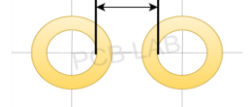






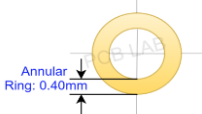
Features	Capability	Notes	Patterns
Layer Count	1,2	The number of copper layers.	-
Material	FR4	For more details, please refer FR4 datasheets.	
Max. Dimension (Xmm x Ymm)	210mm x 160mm	The maximum dimension requirement.	
Min. Dimension (Xmm x Ymm)	4mm x 4mm	The minimum dimension requirement.	
Dimension Tolerance	$\pm 2\text{mm}$	Manual cutting, CNC Routing	-
FR4 Copper Clad Thickness (T)	$\approx 1.6\text{mm}$	For more details, please refer FR4 datasheets. e.g. For the 1.6mm board thickness, the finished board thickness ranges from 1.44mm( $T-1.6 \times 10\%$ ) to 1.76mm( $T+1.6 \times 10\%$ )	
Finished Outer Copper Layer	1oz (35 $\mu\text{m}$ )	Thickness of the copper on the respective layers after fabrication of the PCB board.	
Finished Outer Copper Layer Tolerance	$\pm 2\mu\text{m}$	Varies according to the PCB fabrication setup.	-

**Note :** FR4 datasheets are available on the PCB Labs website.

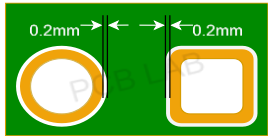
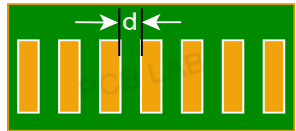
- **Drill / Hole Size :**

Features	Capability	Notes	Patterns
Drill Hole Size	0.70mm (Min.) – 3.00mm (Max.)	Minimum and Maximum drill hole size requirement.	
Drill Hole Size Tolerance	-0.08mm / +0.15mm	e.g. for the 0.80mm hole size, the finished hole size could be between 0.72mm and 0.95mm.	-
Minimum Via hole size	0.70mm	For Single or Double Layer PCBs, the minimum via hole size requirement is 0.70mm.	
Minimum Copper Via diameter (d)	0.85mm	For Single or Double Layer PCBs, the minimum Copper diameter is 0.85mm.	
PTH hole Size	0.70mm – 2.80mm	The PTH hole size will be enlarged to 0.15mm in the fabrication. Please generate and submit separate file for PTH holes. e.g. PTH_drills.drl etc.	-
Pad Size	Minimum 1.50mm	The minimum size of annular ring around plated through hole pads is 0.40mm. If the recommended sizes are not respected then the pad will not be produced properly.	
NPTH Hole Size	0.70mm (Min.) – 3.00mm (Max.)	Please generate and submit separate file for NPTH holes. e.g. NPTH_drill.drl etc.	-
Plated Slots	Not Supported	-	
Non-Plated Slots	Not Supported	-	
Castellated Holes	Not Supported	Half Hole Cut.	


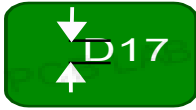
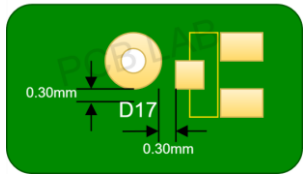
- **Minimum Clearance :**

Features	Capability	Patterns
Hole to hole clearance ( different nets )	1.2mm	
Via to Via clearance ( Same nets )	0.50mm	
Pad to Pad clearance ( Pad without hole different nets )	0.35mm	
Pad to Pad clearance ( Pad with hole different nets )	1mm	
Via to track	0.75mm	
PTH hole to track	0.75mm	
NPTH hole to track	0.75mm	
Pad to track	0.35mm	
Minimum Trace width	0.35mm	
Minimum Trace to Trace Clearance	0.35mm	
Minimum Annular Ring	0.40mm	


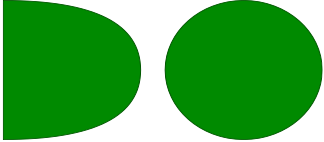
- Solder mask :**

Features	Capability	Notes	Patterns
Solder mask opening / Expansion	0.2mm	The solder mask should have a minimum of a 0.2 mm opening around the pad to avoid any mis-registration.	
Solder Bridge (d)	0.3mm	To have solder mask bridge, the spacing between copper pads edge must be 0.3mm.	
Solder mask color	Green	Liquid photo-imageable solder mask	-
Solder mask thickness	10-15µm	The thickness for solder masks on finished PCB.	-

- Silkscreen / Legend Print :**

Features	Capability	Notes	Patterns
Minimum line width	0.25mm	Characters width less than 0.25mm will be unidentifiable.	
Minimum Text height	1mm	Characters height less than 1.0mm (40mil) will be unidentifiable.	
Silkscreen / Legend Print color	White	Liquid photo-imageable solder mask	-
Pad to Silkscreen	0.3mm	The Minimum Distance Between Pad and Silkscreen is 0.3mm.	

- **PCB Board Outlines :**

Features	Capability	Notes	Patterns
Square, Rectangular board shape	Support	PCB board dimensional cut-out.	
Other than Square, Rectangular board shape	Not Supported	PCB board dimensional cut-out.	

- **PCB Surface Finishing :**

Features	Capability	Notes	Patterns
Surface Finishing Process	Organic Solderability Preservative (OSP) Process	-	-

**❖ Contact :**

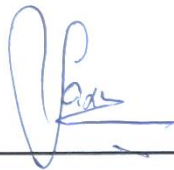
For any query contact us.

Email : [pcblab@ee.iitb.ac.in](mailto:pcblab@ee.iitb.ac.in)

Ph No : +91-22-2159-3599

Website : <https://www.ee.iitb.ac.in/~pcblab/>

Address : PCB LAB ( 1<sup>st</sup> Floor, Room No : 116-A ), Electrical Annex Building,  
Electrical Engineering Department, IIT-Bombay.



---

**Prof. Laxmeesha Somappa**

**In-Charge, PCB LAB**

Electrical Engineering Department,  
Indian Institute of Technology-Bombay,  
Powai, Mumbai, Maharashtra, 400076.



---

**Head of the Department**

**PROFESSOR & HEAD**

Department of Electrical Engineering  
Indian Institute of Technology-Bombay,  
Powai, Mumbai-400076

Electrical Engineering Department,  
Indian Institute of Technology-Bombay,  
Powai, Mumbai, Maharashtra, 400076.