



SAE Default Manufacturing Specifications

Scope:

When manufacturing specifications are not provided by the customer, we will use this specification.

Note:

All PCBs will be built and inspected to the following IPC Standards:

IPC-A-600 Class II Acceptability of Printed Boards

IPC-6012A Class II Qualification & Performance for Rigid Printed Boards

I. GENERAL:

Order of precedence

1. Any notes on the Purchase Order
2. Customer provided drawing
3. Customer supplied specification
4. This specification

There shall be no functional deviation from customer data without customer approval.

All PCB's will be manufactured with standard FR4 UL approved material.

Each board will be marked with the vendor logo, UL designator and date code with the date code displayed (WWYY).

1st choice: Top silkscreen

2nd choice: Bottom Silkscreen

1st choice with no legend film: Etch

Default stack up is 1oz Inner Layer and ½ oz outerlayer copper and a balanced foil construction. If spacing is less than .006 on Inner Layer, ½ oz copper will be used. Overall thickness will be .062 +/- .006.

II. MANUFACTURING:

Plating:

If not specified boards will be vertically Hot Air Leveled using 63/37 Tin Lead solder and will be covered and solderable.

Plating standard thickness':

- Immersion Gold: 2-5 Micro Inches (Selective and SMT)
- Hard gold: 30-50 Micro Inches (Tab Plating)
- Nickel: 100-130 Micro Inches

Screening:

Soldermask will be Green Liquid Photoimageable mask applied to both sides.

Silkscreen will be white, permanent, non-conductive epoxy ink. Minimum line width for silkscreen will be .006.

Soldermask minimums:

- Dam Width: .003 (Black Soldermask .005)
- Clearance: .005

Physical Dimensional Tolerances:**Rout:**

- External Board: +/- .010
- Internal Features: +/- .005

Required Spacing for Scoring

- on 15° blade = 0.015" minimum
- on 30° blade = 0.025" minimum

Capability Matrix

PCB FEATURE	DESIGN FOR CAPABILITY
Minimum Trace	.004
Minimum Space	.004
Pad To Ground Spacing	.008
Split Ground Plane Spacing	.008
Ground Clearance Over Drill	.016 over drilled (fin. + .005)
NPTH To Copper Clearance	.005
Preferred Aspect Ratio	4:01
Smallest Via Hole	.008
Min. Pad Size Over Hole Size	.008 over drilled (fin. +.005)
Min. Solder Mask Clearance	.005 overall, .0025 side
Min. Solder Mask Web Width	.003 (.008 copper to copper) .005 Black SM (.010 cu to cu)
Min. Cu Edge Clearance	.010
Via Hole Size Tolerance	+.003/ - hole size