BoseResearch
INNOVATIVE TECHNOLOGY THROUGH RESEARCH

PCB Design Rules
for
Manufacturability, Serviceability and Testability

of

600 W AC-DC Converter

for

HUNTING

Revision History

Ver. Rev	Date	Revision details	Done by	Checked by	Approved by
1.0	9 Apr 2010	First Release	GHR	AMM	SB
2.0	10 Aug 2016	Updated with latest changes in IPC Standards	GHR	AC	DCB

CONTENTS

1. INTRODUCTION

1.1 Purpose

This document lists the Printed Circuit Board (PCB) Layout design considerations to be followed for the project. These design considerations draw reference to **IPC** (Association Connecting Electronic Industries) Reference Standards for producing Rigid Printed Board Assemblies and adopting Customer's manufacturing practices while also suggesting improvements towards achieving smaller, cost effective, faster and better quality products. with an aim to achieve with the pictures, 3D pictorial representation.

1.2 Scope

The scope of this document is limited to the requirements of this project in particular. For improved clarity, pictures and pictorial representation have been used.

1.3 Intended Audience

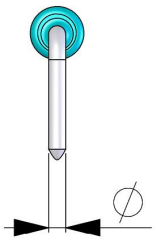
Customer's Design, Purchase and Production team, Project Manager, Test Engineers and Design Engineers at Bose.

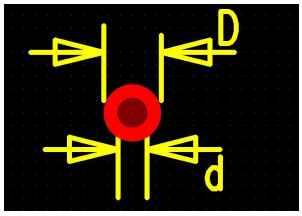
1.4 References

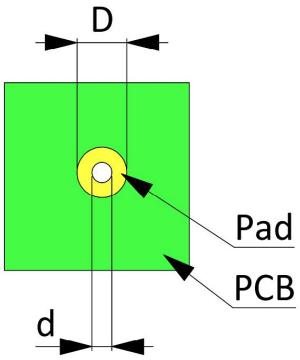
Sl. No.	Description	IPC Ref. No. with Release
1	Generic Standard on Printed Board Design	**IPC-2221B**
2	Generic Requirements for Through-Hole Design and Land Pattern Standard	**IPC-7251**
3	Generic Requirements for Surface Mount Design and Land Pattern Standard	**IPC-7351**
4	Stencil Design Guidelines	**IPC-7525**
5	Surface Mount Design and Land Pattern Standard	**IPC-SM-782**
6	Acceptability of Electronic Assemblies	**IPC-A-610D**

2. FORMING AND MOUNTING OF THROUGH HOLE COMPONENTS:

2.1 Standard pad assignment for through hole components

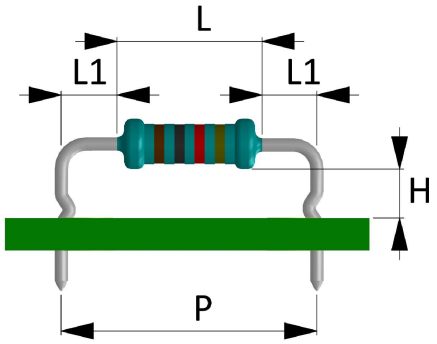


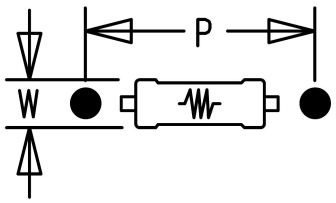




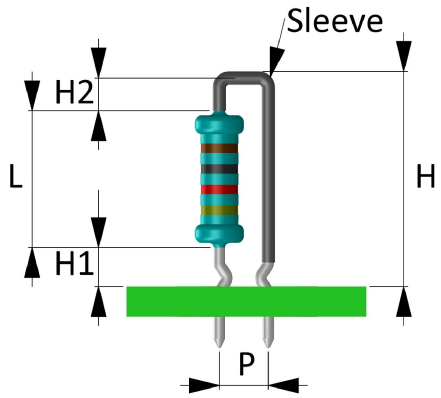
- d = Hole Dia = (Ø + 0.25 mm) **(Level A : Max. lead to hole ratio as per IPC 7251)**
- D = PAD Dia = (2 x d)
- D = Solder Mask Dia = D (PAD Dia) **(as per IPC 7251)**

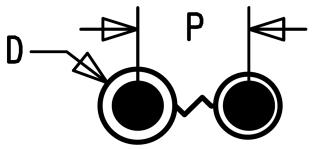
2.2 Special Forming and mounting of Horizontal & Vertical through hole resistors





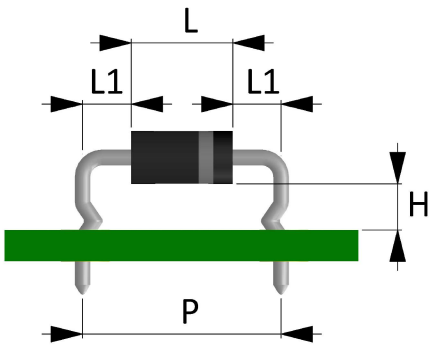
- L = Depends on Wattage
- L1 = 2 mm (Level B) **(as per IPC 7251)**
- H = 0.7 mm minimum and 1.5 mm maximum **(for air flow to dissipate heat as recommended by IPC-A-610D**
- P = Pitch = L + L1+L1 Round off to Multiple of 2.54 mm
- W = As per The Manufacturer data sheet including maximum tolerance

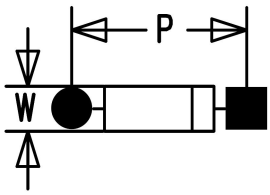




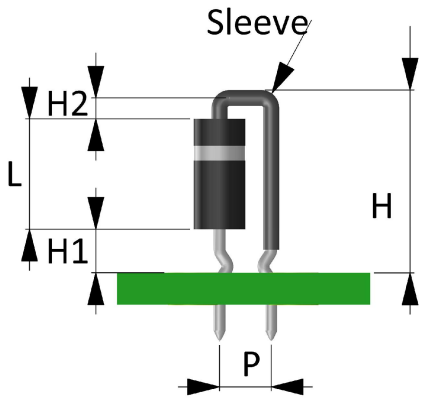
- L = Depends on Wattage
- H1 = 0.25 mm minimum and 2 mm maximum (**as recommended by IPC-2221B**)
- H2 = 2 mm (Level B) **(as per IPC 7251)**
- H = 15 mm Maximum Height of vertical Mounting from the Board **(as per IPC 2221B)**
- P = Pitch = D /2 + Multiple of 2.54 mm (As per IPC 7251)
- D = Depends on Wattage
- Check that when pushed with 10N force, the component in this position does not violate any clearance and creepage distance required as per safety standard IEC 60950-1 or other applicable standard

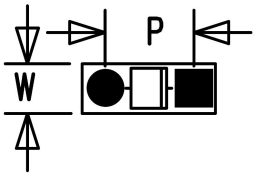
2.3 Special Forming and mounting of Horizontal & Vertical through hole Diodes





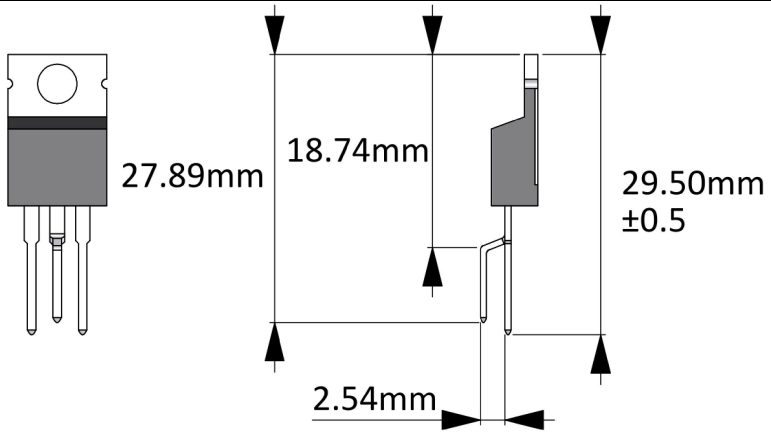
- L = Depends on Packages
- L1 = 2 mm (Level B) **(as per IPC 7251)**
- H = 0.7 mm minimum and 1.5 mm maximum **(for air flow to dissipate heat as recommended by IPC-A-610D**
- P = Pitch = L + L1+L1 Round off to Multiple of 2.54 mm
- W = As per The Manufacturer data sheet including maximum tolerance



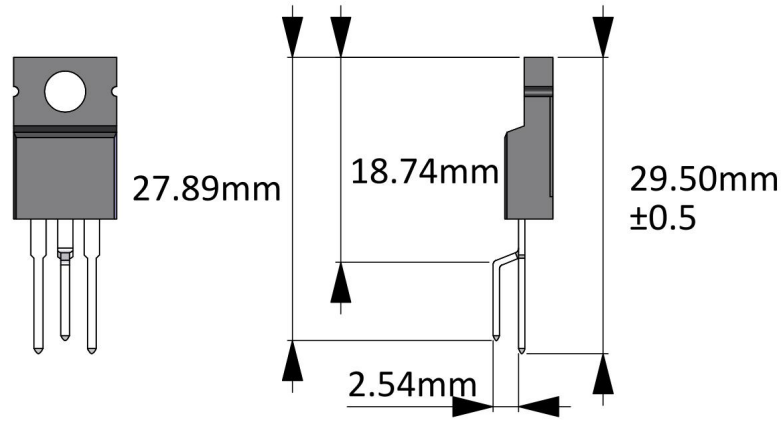


- L = Depends on Package
- H1 = 0.25 mm minimum and 2 mm maximum (**as recommended by IPC-2221B**)
- H2 = 2 mm (Level B) **(as per IPC 7251)**
- H = 15 mm Maximum Height of vertical Mounting from the Board **(as per IPC 2221B)**
- P = Pitch = W /2 + Multiple of 2.54 mm (As per IPC 7251)
- W = Depends on Package
- Check that when pushed with 10N force, the component in this position does not violate any clearance and creepage distance required as per safety standard IEC 60950-1 or other applicable standard

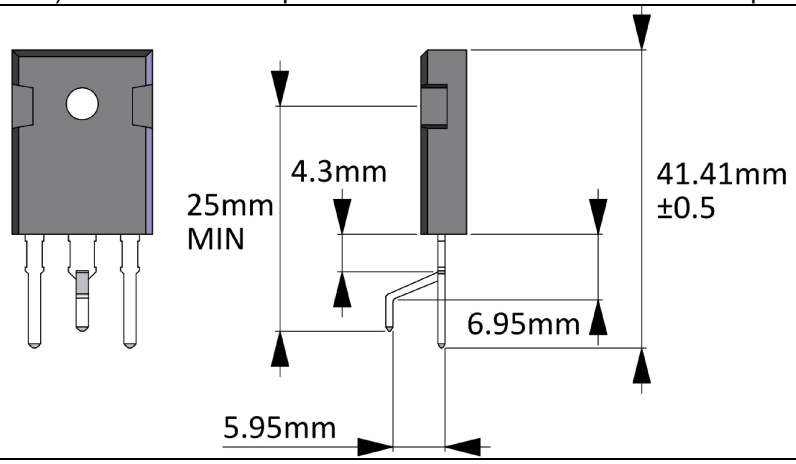
3. FORMING AND MOUNTING OF DEVICES:



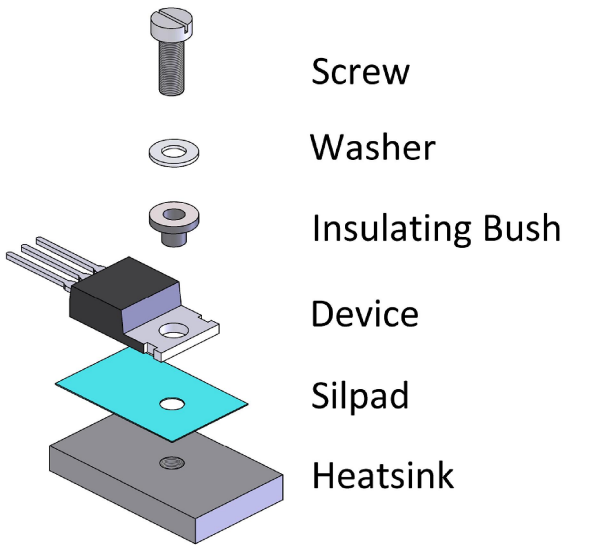
TO 220 Device (Normal Pack) mark clearances- picture with sleeve if clearance not adequate

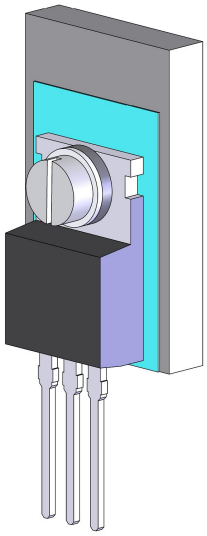


TO 220 Device (Iso Pack) mark clearances- picture with sleeve if clearance not adequate

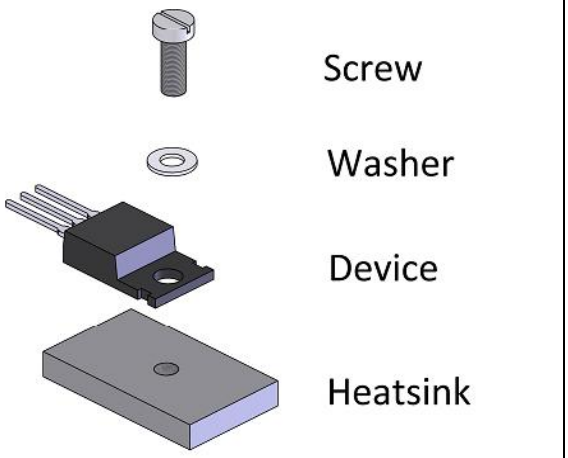


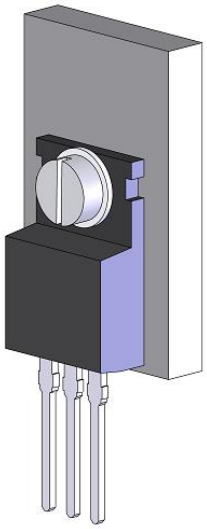
TO 247 (ISO Pack) mark clearances- picture with sleeve if clearance not adequate



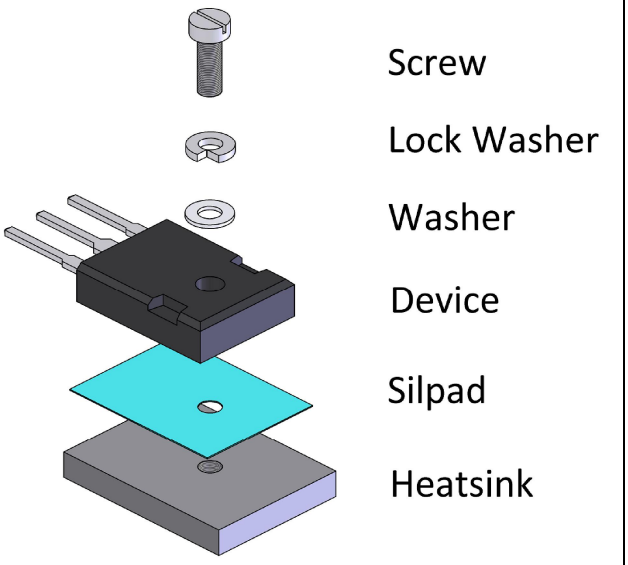


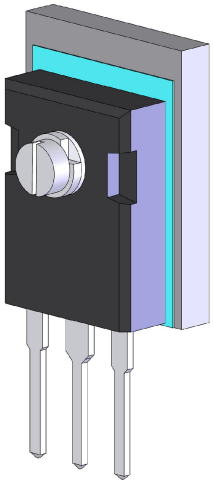
- Direct mounting of TO-220 Device (Normal pack)



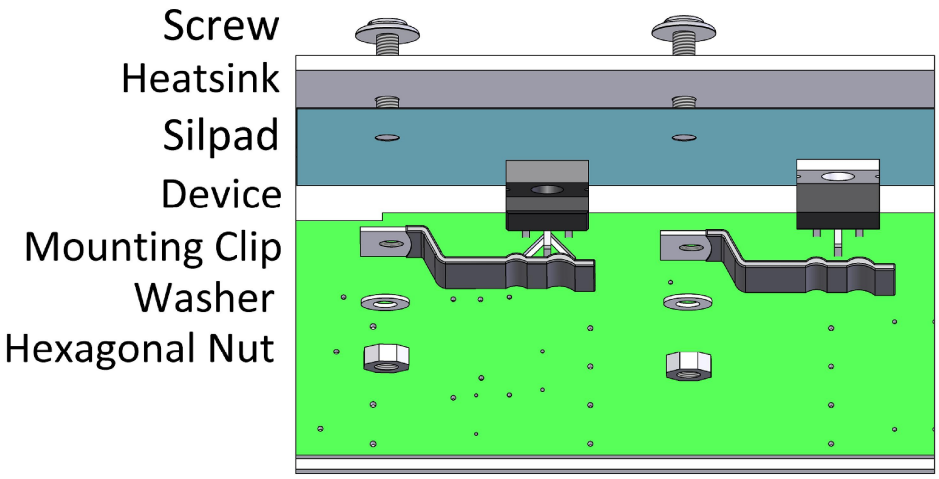


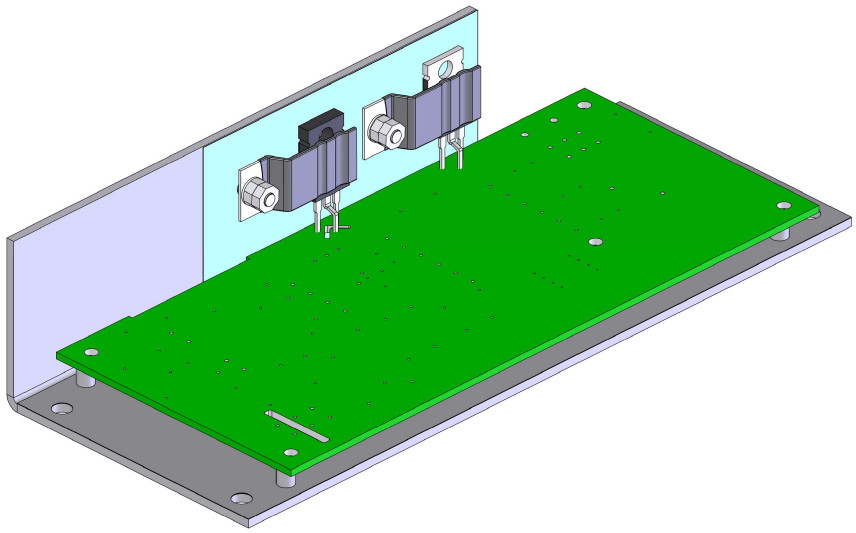
❖ Direct mounting of TO-220 Device (Iso pack)





❖ Direct mounting of TO-247 Device.





❖ Device mounting With clips

4. MOUNTING OF SMD COMPONENTS :

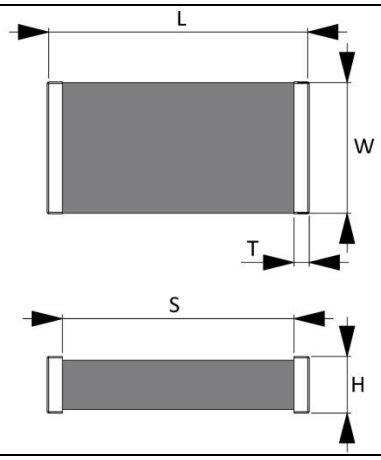
A. SMD components would be preferably placed on the solder side of the printed circuit board (PCB) provided these are suitable for wave soldering. Most of the SMD packages (0805, 1206, 1210, 2010, 2512, SOT-23, SMA, SMB, SMC, SO-IC, TSSOP) are suitable for wave soldering except 1812, DPACK, D2PACK, MLP, QFN. SOT-223 packages of some manufacturers are not recommended for wave soldering.

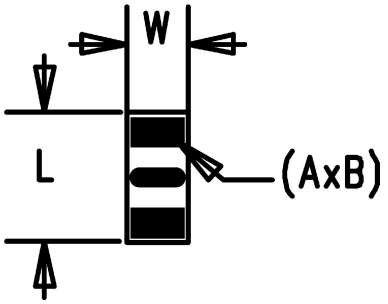
B. It is also possible to place SMD components on the component side of the PCB due to paucity of space or if component manufacturer does not recommend wave soldering of a particular type of SMD component like the 1812 capacitors of some values/ vendors or

SOT 223 devices. Also, DPACK, D2PACK, MLP and QFN packages will be placed on component side of the PCB as they need to be Reflow soldered.

C. The PCB decal and dimension used for different SMD packages for both Reflow and Wave soldering processes are shown in **Table.2** below.

5. Table.2: PCB Outlines of SMD Components for Different Soldering Processes IPC-7351





Sizes	L	W	T	S	H
0805	2.15mm	1.4mm	0.6mm	0.9mm	0.6mm
1206	3.45mm	1.75mm	0.7mm	1.95mm	0.75mm
2010	5.2mm	2.7mm	0.85mm	3.5mm	0.7mm
2512	6.5mm	3.3mm	0.85mm	4.8mm	0.75mm

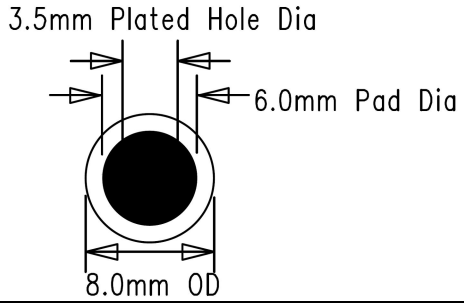
Sizes	L	W	A	B
0805	3.3 mm	1.7mm	0.9mm	1.2mm
1206	4.4 mm	2.1 mm	1.05 mm	1.8 mm
2010	6.6 mm	3 mm	1.2 mm	2.7 mm
2512	7.9 mm	3.5 mm	1.2 mm	3.2mm

Glue spot for wave soldering components As per IPC-7525

6. Mounting Holes

A. Minimum of four mounting holes will be placed on the PCB for Mechanical Stability. The maximum count of these mounting holes will depend on design requirements.

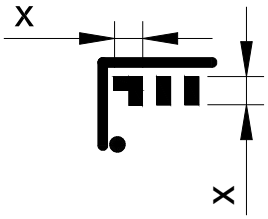
B. PCB Mounting Hole: 3.5 mm-plated hole, 6 mm pad dia, 8 mm OD



PCB Mounting Hole

7. Solder Thieves

The use of solder thieves (areas of metallization in addition to, or attached to, the downstream pair of solder lands of the IC footprint) is recommended for wave soldering as they reduce the likelihood of solder bridging on these lands. This is a L-Shaped pad of size equal to the length of other pads.



Size of the solder thieves pad

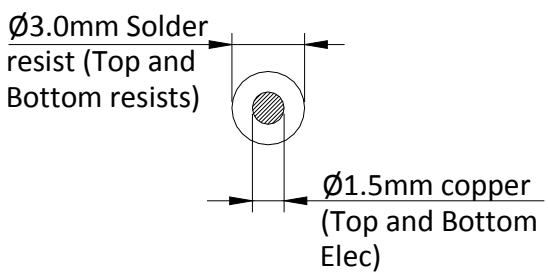
8. Fiducials

A. The edge of the fiducial should be no closer to the board edge than the sum of 4.75 mm and the minimum fiducial clearance required. If less than this sum, a board handling fixture may be required.

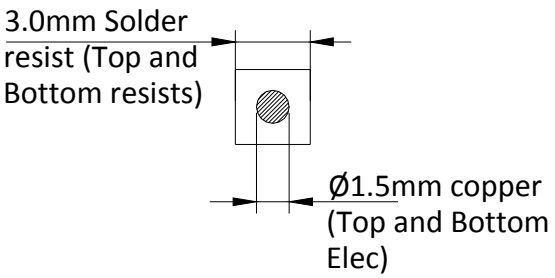
B. Fiducials are needed for an auto placement machine to reference a particular point on a PCB.

C. A Fiducial is defined as a 1.5mm Pad with a 3.0mm solder mask opening around it. Two types of Fiducials that could be used are shown below As per IPC-7351

D. At least Three Fiducials shall be placed near the mounting holes for PCB of any Dimensions.

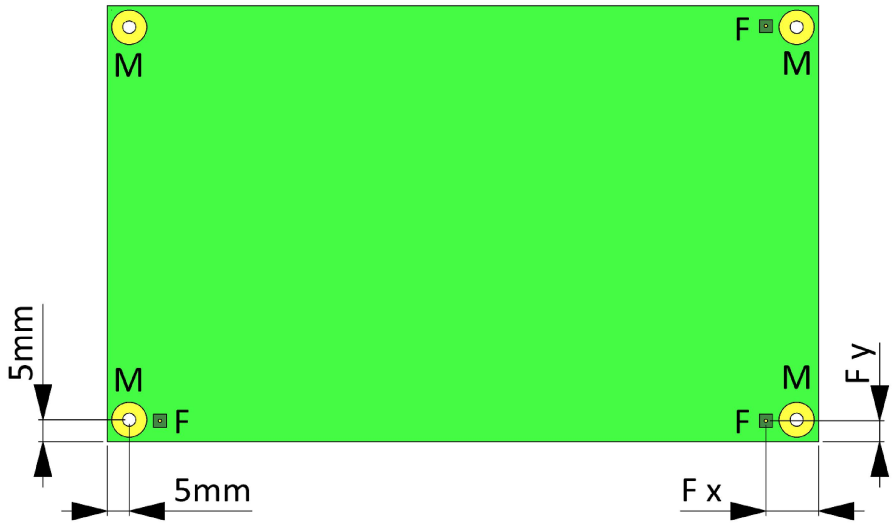


Fiducial Type 1

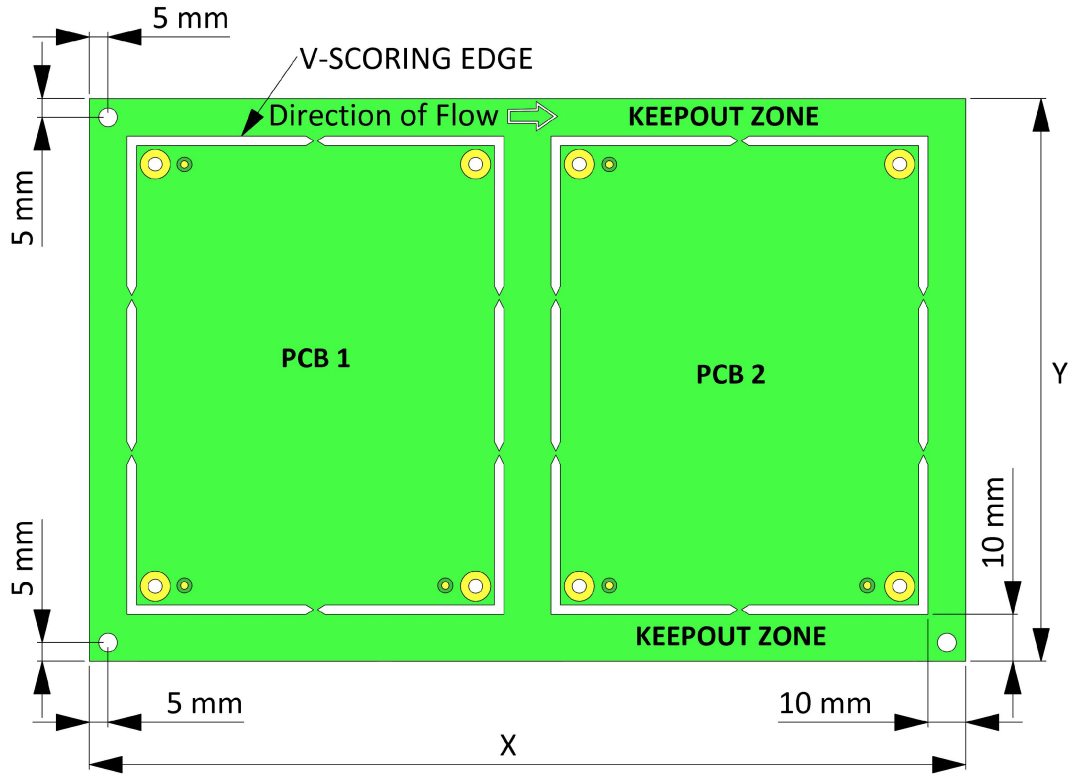


Fiducial Type 2

M - Mounting Hole
F - Fiducials



Fx & Fy = Should be multiply 4.75 mm from board edge
Mounting holes 5 mm from board edge **As Per IPC7351**



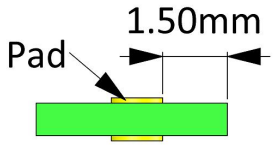
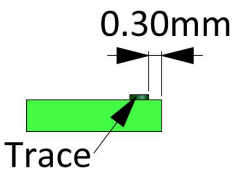
Board Size and Panel Construction as per IPC7351
The keepout zone defined in this illustration is typical for in-line assembly automation using reflow and wave solder processes.

9. STANDARD RULES FOR PCB DESIGN:

9.1 Clearance between PCB edge and the nearest Component or copper trace:

A. Min. distance from PCB edge to Copper = 0.3 mm or as given by Customer.

B. Min. Distance from PCB edge to component Pad For soldering testing should be atleast 1.5mm.

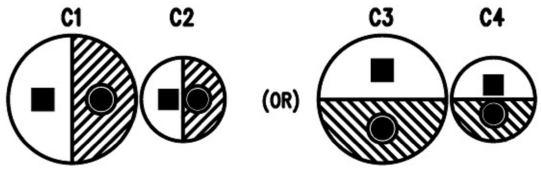
	
Min. distance from PCB edge to component Pad 1.50 mm As per **IPC2221**	Distance of PCB edge to Trace 0.3mm

9.2 Component placement in rows / columns

For improved aesthetics of the finished PCB, components must be preferably arranged neatly in rows / columns with same orientation where possible. A suitable grid will depend on the density of the PCB.

A. Through Hole Components:

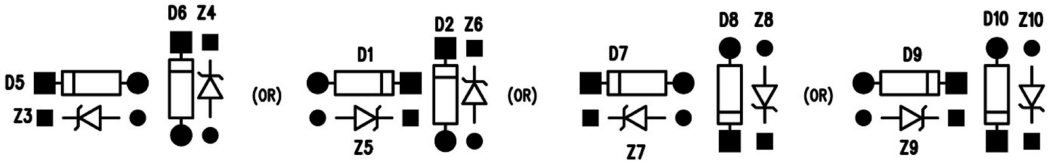
All electrolytic capacitors on the PCB shall be placed with the same orientation as shown below.



Electrolytic capacitor orientation

B. Diodes and Zeners:

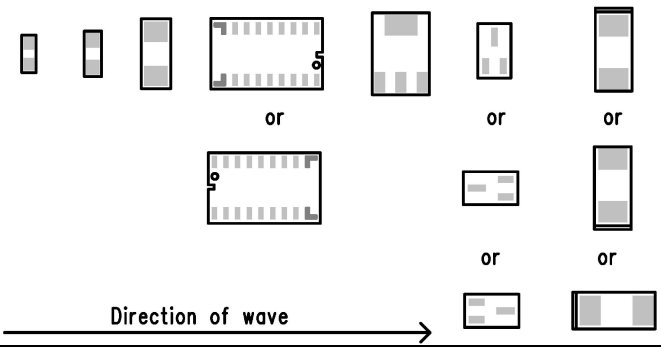
All through hole diodes and Zeners shall be placed with same orientation or mutually perpendicular orientation. SMD diodes and Zeners shall be placed in only one direction.



Through Hole Diodes orientation

C. SMD Components:

All SMD components with same package shall have the same orientation for the entire PCB. As Per IPC-2221B



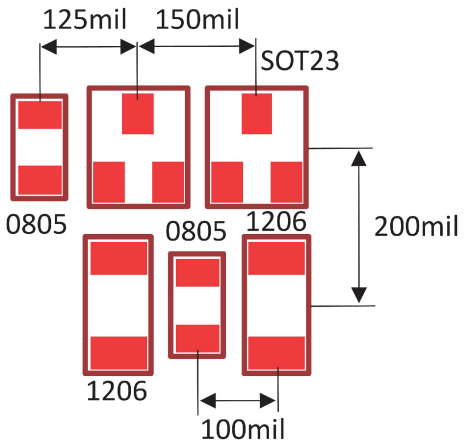
SMD components orientation

D. Grid spacing for SMD Components placement and Alignment:

The SMD components of same package will be aligned as grids in rows and columns with fixed distance between them. This would ensure good wave soldering. The possible alignment patterns are shown below:

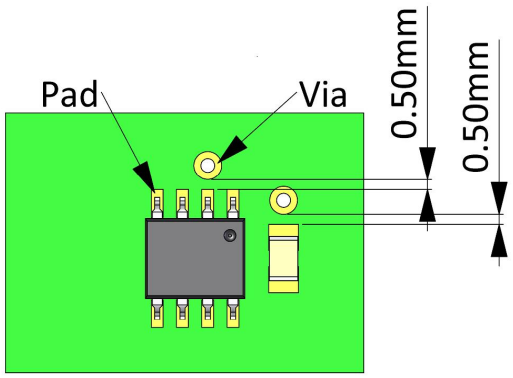
As per IPC 7351

Placement Grid 0.635 mm As per IPC-2221

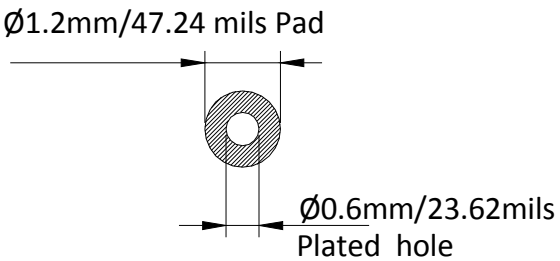
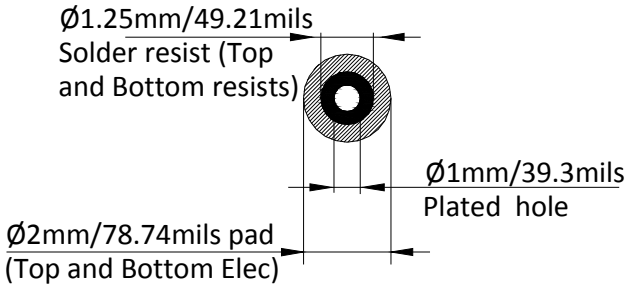


SMD Alignment patterns

10. Placement of Vias and their dimension:



Vias near SMD component pads on Wave and reflow solder via should be 0.5 mm far as per **IPC7351**

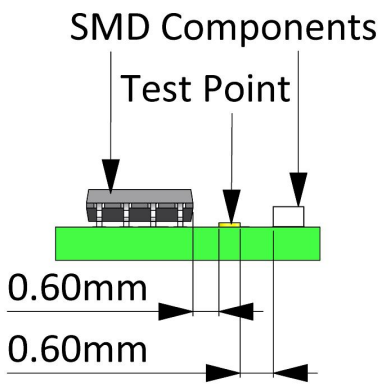
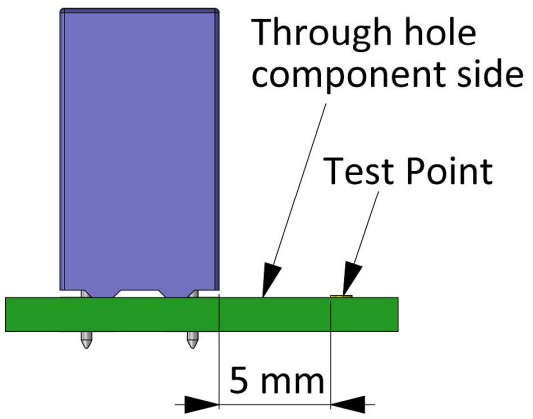
	
Standard Via	Additional Via for POWER Traces

A. Via dimension 0.6/1.2 mm Via Tented A via with a dry film mask material applied bridging over the via as per IPC-4761

B. All multiple PTHs on power trace for current reinforcements 2/1 Via masked with opening of 0.25 mm for Via filling while wave solder.

11. Test Points

Test points shall be provided based on the importance of a particular signal that would help in debugging and troubleshooting. The number of test points shall be provided on the above consideration on a best effort basis and not for every net on the PCB.

	
SMD component to Test point 0.6 mm Clearance as per IPC2221	TH component to Test point 5 mm Clearness as per IPC2221

12. High Current Reinforcements

A. Multiple plated through holes (PTH) of 1.25 mm pad Diameter and additional VIA for Power Traces of 1.25 mm pad Diameter shall be placed surrounding the Pads of the components carrying high currents where reinforcements are essential. The PTHs along with the pad will be covered with copper.

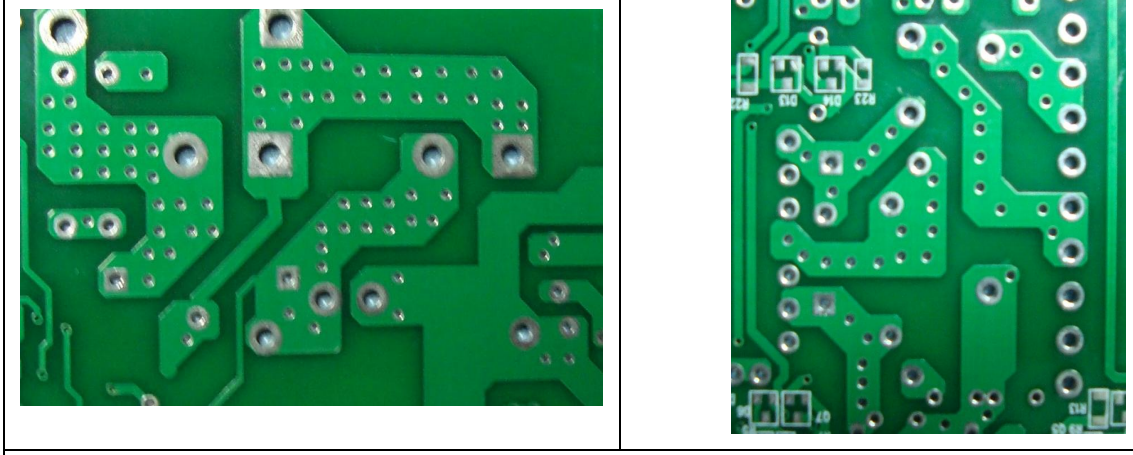


Fig.8a. Placement of Multiple PTHs

B. Copper Rods shall be soldered on solder side of the PCB traces where high current reinforcements are required as shown in the picture below.

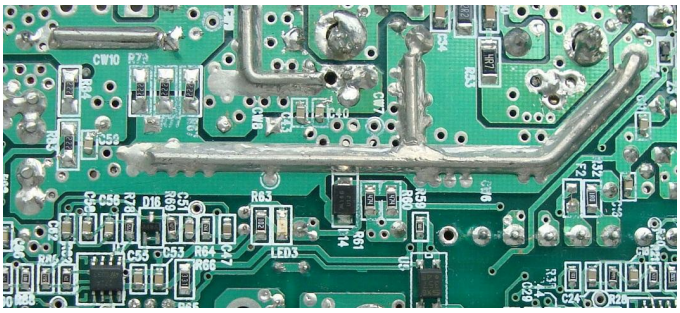
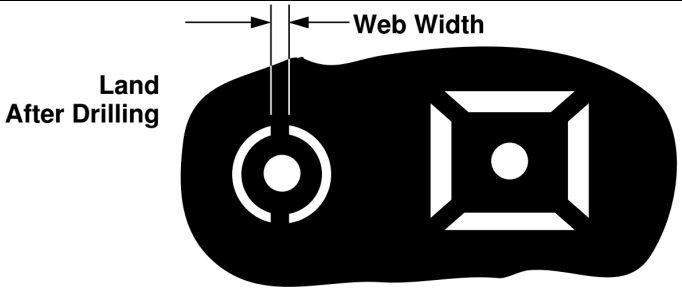


Fig.8b. Copper rods are soldered on power traces carrying high currents

13. Thermal Relief:

Thermal Relief in Conductor Planes The relationship between the hole size, land and web area is critical. Typically, divide 60% of the minimum land area diameter by the number of webs desired to obtain the width of each web in accordance with the following example:



As Per IPC2222

Thermal Relief Calculation
Total thermal width = 60% of land size
For example Land size 2 mm
= 0.6 x 2 mm
= 1.2 mm

Original Web Size Calculation
4-web width = 1/4 of total thermal width x Thermal Width
=1.2÷4 = 0.3 mm
= 0.3 x 1.2 mm
Each Web is =0.36 mm

14. Standard Clearances

A. Standard minimum clearances:

The standard minimum clearances shall be as tabulated below:

Table 3. Standard minimum clearances

Clearance items	Minimum clearance
PAD to PCB edge	1.5 mm or 60mils
Comp to PCB edge	1 mm or 40mils
Copper to PCB edge	0.3mm or 12mils
Copper to Copper	0.5mm or 20mils
Pad to Copper	0.5mm or 20mils
Route to Copper	0.5mm or 20mils
Via to Copper	0.5mm or 20mils
Route to Pad	0.3mm or 12mils
Route to Route	0.3mm or 12mils
Test point to Via	0.5mm or 20mils
Via to Via	0.5mm or 20mils
Via to SMD Pad	0.5mm or 20mils

a) **Applicable Safety Clearance and Creepage Distances on PCB:**

Table .4. Applicable Safety Clearance and Creepage Distances on PCB as per IEC 62368-1

Circuit	Clearance between	As per IEC 62368-1 & IEC-60950-1		
		Insulation	**Clearance (mm)**	**Creepage Distance (mm)**
Primary Circuit from Input (with transients) to MOV	Line and Ground	Functional	1.5	3.2
	Line and Earth	Basic/ Supplementary	2.0	3.2
	Ground and Earth	Basic/ Supplementary	2.0	3.2
Primary Circuit after MOV upto Bridge	Line and Ground	Functional	0.8	3.2
	Line and Earth	Basic/ Supplementary	1.5	3.2
	Ground and Earth	Basic/ Supplementary	1.5	3.2
Primary Circuit after Bridge upto Transformer	Line and Ground	Functional	3.0	-
	Line and Earth	Basic/ Supplementary	-	3.2
	Ground and Earth	Basic/ Supplementary	-	3.2
Across Transformer	Primary and Secondary	Reinforced	6.4	6.4
Secondary Circuit 500 V DC	Line and Ground	Basic/ Supplementary	0.8	5.0
	Line and Earth	Basic/ Supplementary	0.8	5.0
	Ground and Earth	Basic/ Supplementary	0.8	5.0

PCB REQUIREMENT SPECIFICATION

Item	Description	Reference
Product	AC to DC switch mode converter	
Dimension	PCB dimensions of 153 mm x 68 mm mounted on an aluminum L Bracket as per Customer drawing Sk56394.pdf. Maximum component height above PCB shall be 30 mm and no components on bottom side. TH Component pin to sheet metal distance > 3.2 mm	
Laminate Material	Double Sided, 1.6 mm thick FR4/94V-0 Tg 150^{0}C glass epoxy 17.5 µ Base Copper thickness of laminate material meeting IPC Class-2.	IPC 2221B
Track width **Spacing** **Drill Size**	Min 0.254 mm / 10 Mils Min 0.304 mm / 12 Mils Min 0.6 mm / 23.67 Mils	
Signal Via dimension	Finished hole dimension after min. 20-25 µ copper buildup shall be 0.6 mm ± 0.1 mm as per Customer standard.	IPC 2221B
Signal Via filling non-conductive	Vias can be plugged with non-conductive UL approved material (Flammability class V-2 & RTI atleast 130^{0}C) as per type IIIb of IPC 4761	IPC 4761
Multiple PTH dimension	Finished hole dimension after min. 25 µ copper buildup shall be 1.0 mm ± 0.1 mm as per Customer requirement.	IPC 2221
Multiple PTH filling conductive	Vias can be plugged with conductive material similar to CB100 from Dupont as per type xx of IPC 4761 as per Customer requirement.	IPC 4761
Non Plated Cutouts	Dimension as per Top silk Gerber with corner radius 0.5 mm, must have smooth finish	
Solder Mask	SMOBC, PISM with green color masking	
Legend	White colour legend (sharpness required) 1.5 mm line Thickness. No printing on production PCBs as per Customer requirement.	
Finish	35 µ finished copper with Lead free HASL shinning finish with tin layer of minimum thickness 1 µm	
Checking	FPT , Complete with Optical and ICT testing	
UL Marking	Yes	

15. FABRICATION : CAM OUTPUT FILE Name

15.1 BOSE Fabrication CAM

Gerber / CAM	Layers in PCB	Items	Remarks
Top Silk	Top Silk	Board Outline	
		2D Line	any drafting item drawn
		Ref Des	
		Text	any drafting item drawn
		Outlines	Component outlines
	Top Code	Text	Project Code
Bottom Silk	Bottom Silk	Board Outline	
		2D Line	any drafting item drawn
		Ref Des	
		Text	any drafting item drawn
		Outlines	Component outlines
	Bottom Code	Text	Project Code
Top Mask	Top Solder Resist	Pads	Pads, Fiducials, Badmarker
		2D Line	Optional for copper reinforcement soldering on traces
		Vias	
		Text	Layer Name outside the board
		Test Points	
	Top Isolation	2D Lines	Optional used in Bose gerbers only
	Corner	2D Lines	Board corners
	Top Code	Text	Project Code (Optional)
Bottom Mask	Bottom Solder Resist	Pads	
		2D Line	Optional for copper reinforcement soldering on traces
		Vias	
		Text	Layer Name outside the board
		Test Points	
	Bottom Isolation	2D Lines	Optional used in Bose gerbers only
	Corner	2D Lines	Board corners

Gerber / CAM	Layers in PCB	Items	Remarks
Top Elec	Top Elec	Pads	Pads, Fiducials, Badmarker
		Traces	
		Vias	
		Copper	
		Text	Layer Name outside the board
		Test Points	
	Corner	2D Lines	Board corners
	Top Code	Text	Project Code (Optional)
Bottom Elec	Bottom Elec	Pads	Pads, Fiducials, Badmarker
		Traces	
		Vias	
		Copper	
		Text	Layer Name outside the board
		Test Points	
	Corner	2D Lines	Board corners
NCDRILL_PTH		TH Vias	Board outline is selected automatically by software
		Plated pins	
NCDRILL_NPTH		Non plated pins	Board outline is selected automatically by software
FAB	Drill Drawing	Pads	
		Vias	
		Text	Layer Name outside the board
	Dimension	2D Lines	
		Text	
	Cutout	2D Lines	
		Text	Dimension and any other information like PTH & NPTH

15.2 BOSE Assembly CAM

Gerber / CAM	Layers in PCB	Items	Remarks
Top Assembly TH	Top Assembly TH	Board Outline	
		2D Line	any drafting item drawn
		Ref Des	
		Text	any drafting item drawn
		Outlines	TH Component outlines
	Top Code	Text	Project Code
Top Assembly SMD	Top Assembly SMD	Board Outline	
		2D Line	any drafting item drawn
		Ref Des	
		Text	any drafting item drawn
		Outlines	SMD Component outlines
	Top Code	Text	Project Code
Bottom Assembly	Bottom Assembly	Board Outline	
		2D Line	any drafting item drawn
		Ref Des	
		Text	any drafting item drawn
		Outlines	SMD Component outlines
	Bottom Code	Text	Project Code
Top Paste	Top Paste	Pads	
		Text	Layer Name outside the board
	Corner	2D Lines	Board corners
	Top Code	Text	Project Code (Optional)
Bottom Paste	Bottom Paste	Pads	
		Text	Layer Name outside the board
	Corner	2D Lines	Board corners
Bottom Glue	Bottom Glue	Outlines	
		Text	Layer Name outside the board
	Corner	2D Lines	Board corners

15.3 BOSE Assembly CAM

Pick & Place Files	Side	Parts
Pick & Place Top_TH	Top	Thru Pin
Pick & Place Top_SMD	Top	SMT
Pick & Place Bottom	Bottom	SMT

****