IPC-7352 Pad Stack Naming Convention

PCB Libraries, Inc.

December 3, 2024

IPC-7352 Pad Stack Naming Convention

INTRODUCTION

The pad stack consists of combinations of letters and numbers that represent shape, or dimensions of lands on different layers of printed boards or documentation. The name of the pad stack needs to represent all the various combinations. These are used in combination with the land pattern conventions defined herein according to the rules established in the IPC-2220 Design standards.

The first part of the pad stack convention consists of a land shape. There are six basic land shape identifiers. Note: All alphabetical characters are “lower case”. This helps discriminate numeric values.

Basic Land Shape Letters

r = Rectangle
rr = Rounded Rectangle
cr = Chamfered Rectangle
b = Oblong
d = D Shape (Square on one end and Circular on the other end)
c = Circular
s = Square
rs = Rounded Square
cs = Chamfered Square
u = User Defined Contour (Irregular Shape)

Pad Stack Defaults

- Solder Mask is 1:1 scale of the land size
- Paste Mask is 1:1 scale of the land size
- The Assembly Layer land is 1:1 scale of the land size
- Inner Layer Land is the same shape as the outer layer land
- The Primary and Secondary lands are the same size
- The inner layer land shapes are Circular
- Vias are Circular
- Thermal ID, OD and Spoke Width sizes follow the IPC Level A, B or C
- Plane Clearance Anti-pad size follows the IPC Level A, B or C
- Thermals have 4 spokes
- Mounting Holes are Circular

Note: Every board fabricator’s ability to register solder mask is different. The 1:1 scale solder mask covers the variation, and so long as manufacturers are building to specs such as IPC-6012 that say you can’t have mis-registration of the solder mask.

Illegal characters that cannot be used (Microsoft requirement) include “ ” , ; : / \ [ ] ( ) . { } * & % # $! @ ^ =

Examples utilizing the pad stack naming convention (all values are in metric units)
Note: Every number goes two places to the right and as many as needed to the left of the decimal
Examples: 1150 = 11.50 mm or 11500 µm, 150 = 1.50 mm or 1500 µm, 15 = 0.15 mm or 150 µm

c150h90 where “c” denotes a Circular land with a 1.50 diameter and H denotes a hole size of 0.90
c130_95 Donut pad where “c” denotes a Circular land with a 1.30 OD diameter and 95 denotes the ID diameter
c130_95hn70k147 Donut pad - “c” denotes a Circular land with a 1.30 OD diameter and 95 denotes the ID diameter and hn70 denotes hole, non-plated 0.70 diameter and k147 denotes a keep-out 1.47 diameter
v50h25 where a “v” denotes a via with a 0.50 land (default Circular land) and h denotes a 0.25 hole
s150h90 where “s” denotes a 1.50 Square land and h denotes a hole size of 0.90
s350 where ‘s” denotes a square SMT land size of 3.50
r200_100 where “r” denotes a Rectangular SMT land 2.00 land length X 1.00 land width
b300_150 where “b” denotes a SMT Oblong land size of 3.00 X 1.50
b400_200h100 where “b” denotes an Oblong land size of 4.00 length X 2.00 width and 1.00 hole
d300_150 where “d” denotes land with one circular end and one square end (looks like a D) 3.00 X 1.50
v30h15l1-3 where “v” denotes a 0.30 blind via with 0.15 Hole; 1 is the starting layer, 3 is the end layer

r200_100r5 = Rounded Rectangular 2.00 mm X 1.00 mm X 0.05 mm radius corners
r200_100c10 = Chamfered Rectangular 2.00 mm X 1.00 mm X 0.1 mm chamfered corners
v30h15l3-6 where “v” denotes a 0.30 buried via with 0.15 Hole; 3 is the starting layer, 6 is the end layer

Note: It is assumed that the through-hole pad stack has the same value as the mounted layer size and shape for –

- Inner Layer
- Opposite Side
- Solder Mask
- Solder Paste
- Assembly Layers

It is also assumed that the “Plane Clearance” and “Thermal Relief” data follows the through-hole convention guidelines defined in the IPC-2221 and IPC-2222 standards.

Modifiers that are used when pad stack features are different than the defaults

These are the “Variants” or “Modifiers” that go after the basic pad stack naming convention.
These are used when the User needs to change the pad stack default values either by a different dimension or a different shape. In instances where shapes are different this becomes a two letter code with the modifier first followed by the land shape letter.

n = Non-plated Hole
z = Inner Layer land dimension if different than the land on primary layer
x = Special modifier used alone or following other modifiers for lands on opposite side to primary layer land dimension
t = Thermal Relief; if different than IPC standard pad stack – tid_od_sw for 4 spoke default
m = Solder Mask if different than default 1:1 scale of land
p = Solder Paste if different than default 1:1 scale of land
a = Assembly surface land if different than default 1:1 scale of land
y = Plane Clearance (Anti-pad) if the value is different than the Thermal OD
o = Offset Land Origin
k = Keep-out
r = Radius for Rounded Rectangular Land Shape
c = Chamfer for Chamfered Rectangular Land Shape

Shape change is the last letter in the string prior to the dimension.

Other usage of the pad stack naming convention

USE of letter v: Vias can be named using the pad stack naming convention. Because most vias use lands that are circular in shape, the letter V will be used in place of the letter C in the pad stack naming convention. If this is not true the modifiers can be added after the letter V to signify shape or dimensional changes to this default.

USE of letter w: In addition to Vias the pad stack naming convention can also be used for defining mounting holes. The letter W shall be used to define the mounting hole characteristics and any associated lands used for the surface lands (either plated or un-plated)

Examples of double character modifiers:

ts = Thermal Square; if different than the top side land shape and dimensions
sw = Thermal spoke width
zs = Inner Layer Land Shape is Square (Note: The default is circular)
m0 = No Solder Mask
mxc = Solder Mask Opposite Side Circular
mx0 = Solder Mask Opposite Side No Solder Mask
xc = Opposite Side Circular
vs = Via with Square land
hn = Non-plated Hole

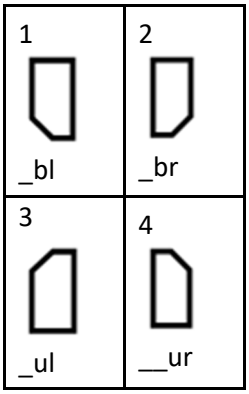
Modifier Example for Through-hole:

s150h90zs150 = where “s” is Square 1.50 land with 0.90 Hole with 1.50 inner (Z) Layer Square land
c150h90zc150 = where “c” is Circular 1.50 land with 0.90 Hole with 1.50 inner (Z) Layer Circular land

Modifier Examples for Vias:

vs50h25 where “vs” denotes a 0.50 Square Via with a 0.25 Hole
v50h25xs70 where “v” is 0.50 Circular Via with 0.25 Hole and 0.70 Square land on opposite side

Chamfered & Rounded corner modifiers are used to indicate which corner(s) are modified.
Order of precedence has been given to the first 4 modifiers.



Modifiers:
bl – bottom left
br – bottom right
ul – upper left
ur – upper right
ulr – upper left & right
blr – bottom left & right
ubl – upper and bottom left
ubr – upper and bottom right

Rounded and Chamfered lands in “one corner” Modifier Examples:

r100_200rbl50 = rectangular land 1.00 x 2.00 with 0.50 radius for rounded corner in bottom left corner

r100_200rbr50 = rectangular land 1.00 x 2.00 with 0.50 radius for rounded corner in bottom right corner

r100_200rul50 = rectangular land 1.00 x 2.00 with 0.50 radius for rounded corner in upper left corner

r100_200rur50 = rectangular land 1.00 x 2.00 with 0.50 radius for rounded corner in upper right corner

r100_200cbl50 = rectangular land 1.00 x 2.00 with 0.50 chamfer for chamfer corner in bottom left corner

r100_200cbr50 = rectangular land 1.00 x 2.00 with 0.50 chamfer for chamfer corner in bottom right corne

r100_200cul50 = rectangular land 1.00 x 2.00 with 0.50 chamfer for chamfer corner in upper left corner

r100_200cur50 = rectangular land 1.00 x 2.00 with 0.50 chamfer for chamfer corner in upper right corner

Examples of a square land with 3 rounded and 1 chamfered corner (for Thermal Pad)

s300p190r25cul50 = square 3.00 land, square paste 1.90, corner radius 0.25, upper-left chamfer 0.50

s300p190r25cbl50 = square 3.00 land, square paste 1.90, corner radius 0.25, bottom-left chamfer 0.50

Chamfered and Rounded Rectangular with all four corners chamfered does not need a corner modifier.

Modifier Examples with Rounded Rectangle Land Shape:

r200_100r50 = rectangular land 2.00 x 1.00 with 0.50 radius for rounded corners in all 4 corners
r200_100c50 = rectangular land 2.00 x 1.00 with 0.50 chamfer for chamfered corners in all 4 corners

Examples of a pad stack with Circular land with hole using various modifiers

c150h90 = Default pad stack with a 1.50 circular land with a 0.90 hole (no modifiers used)

c150hn90 = Default pad stack with a 1.50 circular land with a 0.90 non-plated hole (no modifiers used)

c150h90z140 = Inner layer land is smaller than external lands 1.40 or 0.10 smaller

c150h90z140x170 = Opposite side land is larger than top side land 1.70 or 0.20 larger

c150h90z140x170m165mx185 = Solder mask opening for top and bottom lands 0.15 larger for each

c150h90z140x170m165mX185a200 = Assembly drawing land in 0.50 larger than 1.50 primary land

c150h90z140x170m165mx185a200y300 = Plane clearance anti-pad diameter is 3.00

c150h90z140x170m165mx85 = Solder mask encroachment on opposite land by 0.65 smaller

c150h90m165 = adding a solder mask opening of 1.65 diameter or 0.15 larger than land

c150h90t150_180_40 = Thermal ID 1.50, OD 1.80, Spoke Width 0.40, Anti-pad 1.80

c150h90t150_180_40y200 = Anti-pad 2.00 (because the size is different than the Thermal OD)

c150h90t150_180_80_2 = Spoke Width 0.80 with 2 Spokes

c150h90m165t150_180_40 = Solder Mask 1.65

Examples of a pad stack with Oblong land with Slotted Hole

Sample – **b** = Oblong Land Shape then "**X**" dimension (length) then Underscore **_** "**Y**" dimension (width)

b400_200h300_100 = Oblong land 4 mm length X 2 mm width with slotted hole size 3 mm X 1 mm

b400_200hn300_100 = Oblong land 4 mm X 2 mm with non-plated slotted hole size 3 mm X 1 mm

Examples of a SMT pad stack land using various modifiers

b300_150 = Default pad stack with a 3.00 length and 1.50 width land (no modifiers used)

b300_150m330_180 = Solder Mask is 0.30 larger than the land on all sides

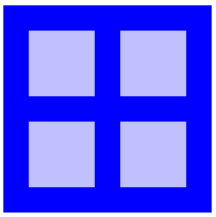
b300_150m330_180p240_140 = Solder Paste is smaller by 0.10 width and 0.60 length

b300_150b-50 = Oblong Land 3.0 mm X 1.5 mm w/Offset Origin negative 0.5mm

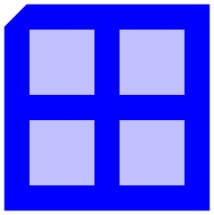
r400_200po430_230 = Rectangle SMT land 4.00 X 2.00 with an Oblong Paste Mask size of 4.30 X 2.30

Example of Thermal Pads for QFN, SON, QFP and SOP

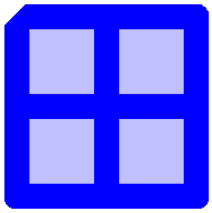
Square Configurations



s480p4s152 = 4.80 mm Square Land with 4 Paste Mask Squares 1.52 mm each



s480p4s152cul50 = 4.80 mm Square Land with 4 Paste Mask Squares 1.52mm each with 0.50 mm Chamfer in Upper Left corner



s480p4s152cul50r25 = 4.80 mm Square Land with 4 Paste Mask Squares 1.52 mm each with 0.50 mm Chamfer in Upper Left corner with 0.25 mm corner Radius

Example of a Mounting Hole

w700h400z520m720 = This is a Plated Through Mounting hole for a #6-32 screw using a 4.00 diameter hole and having a circular 7.00 land on the primary and secondary side of the board, with a solder mask clearance that is 0.20 larger than the 7.20 land. The internal lands are smaller that the external and are also circular 5.20 in diameter.

w700hn400z520m720 = Non-plated version

Example of a Local Fiducial for Fine Pitch SMT Components

c100m200k200 = Circular Land 1.00 with Solder Mask 2.00 with Keep-out 2.00
s100m200k200 = Square Land 1.00 with Solder Mask 2.00 with Keep-out 2.00

Example of Proportional Plated Through-hole Pad Stack

c150h100 = 1.5 mm circular pad with 1 mm hole with 1.5 mm solder mask with 1.5 mm plane clearance with 1.5 mm assembly outline with Thermal Relief w/4 spokes 0.4 mm width with ID 1.5 mm and OD 1.8 mm

Example of Proportional Non-plated Through-hole Pad Stack

c100hn150 = 1 mm circular pad with 1.5 mm hole “non-plated” with 1.5 mm solder mask with 2.35 mm plane clearance with 2.1 mm keep-out