IPC-7251 & 7351 Padstack Naming Convention

INTRODUCTION

The padstack 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 padstack 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 padstack 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

> c = Circular
> s = Square
> r = Rectangle
> b = Oblong
> u = Contour (Irregular Shape)
> d = D Shape (Square on one end and Circular on the other end)

Padstack 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
- 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 padstack 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

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

v30h15l3-6 where "v" denotes a 0.30 buried via with 0.15 Hole; 3 is the starting layer, 6 is the end layer

The through hole "IPC-7251 Padstacks.xls" file should be used as the basis for a new chart in IPC-7251 and IPC-7351B. Note: Draft supporting paragraphs with formula that document the math involved.
It is assumed that the padstack 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 padstack features are different than the defaults

These are the "Variants" or "Modifiers" that go after the basic padstack naming convention.
These are used when the User needs to change the padstack 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 the land convention that matches land on primary layer
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 is a different value than the default
o = Offset Land Origin

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

Other usage of the padstack 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 padstack 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 padstack 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
tt = Thermal with 2 spokes
ttt = Thermal with 3 spokes
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 Examples:

s150h90zs150 = where "s" is Square 1.50 land with 0.90 Hole with 1.50 inner (Z) Layer Square land
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

Examples of a padstack with Circular land with hole using various modifiers

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

c150hn90 = Default padstack 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

Examples of a padstack 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 4mm length X 2mm width with slotted hole size 3mm X 1mm

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

Examples of a SMT padstack land using various modifiers

b300_150 = Default padstack 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

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

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

r400_200po430_230 = Rectangle SMT land 4.00 X 2.00 with a Oblong Solder Paste size of 4.30 X 2.30

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