SMT STENCIL DESIGN AND CONSIDERATIONS

Cuong Tran

Process Engineer

Table of Content

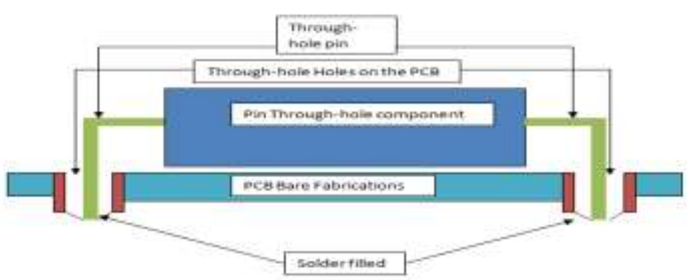
Subject…………………………………………………………………………………………………Page

A BRIEF DISCUSSION OF PCBA TECHNOLOGY

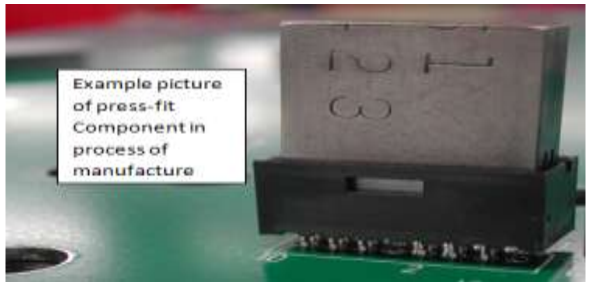
Introduction:

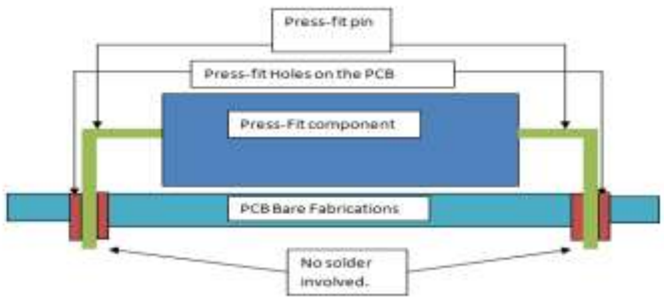
PCBA (Printed Circuit Board Assembly) is a segment of printed circuit board technology. This segment of printed circuit board industry is concentrated in assemble all the pieces of electronic industry to one piece before output them to market. This segment covers: interconnection technology, package design technology, system integration technology, board and system test technology…etc. However, in a very brief and short description, PCBA is the segment that concentrated in assembly all electronics and electro mechanical components on the surface of a PCBA using metallic bonding such as: pin through-hole's solder, surface mount solder, or press fit interconnection.

Pin through-hole technology is the technology where components are soldered on the board using male-female type connections. PCB (printed circuit board) bare fabrication will provide holes that connected to all the internal circuit of the fabrication. On the other hand, the components that will be assembled on this through-hole fabrication have male-pin type that make by electronics packaging industry. When PCBA process applies, the components that have male-pin will be placed in the through-hole fabrication and then solder them together by selective wave, selective wave fixture, or dip in the liquid solder to form interconnection joins. The picture below should illustrate the summary of this type of technology:

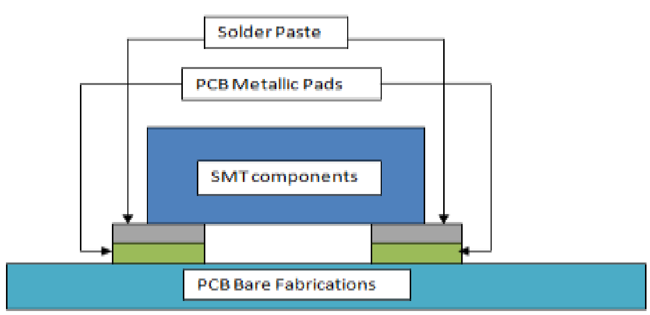


Similar to pin through-hole technology is press-fit components design. Press-fit technology deviate from the idea of through hole component's design, but not using solder to join the component to the PCB fabrication, but have the same strength as pin through structure. The only different between these two technologies are: 1) the male pin will be make to fit the female hole on the PCB fabrication. 2) In order to achieve join structure, press-fit will use pressure to put the male pin into the female hole on the fabrication in order to complete the joining. 3) press-fit technology does not use any solder, but design in very fit and tight dimensions between the holes of PCB fabrication and the component male pin. 4) All the defects of this technology will come from the accuracy of the dimensions, the support tooling, and the support equipment. The picture below should illustrate the summary of this type of technology:





Last but not least, the current and widely use technology now is SMT (Surface Mount Technology). Briefly, we can use the name of this technology to understand its use. Mounting or solder the components on the surface of the PCB fabrication is SMT. Different between SMT and pin through-hole technology is: while pin through-hole using liquid solder, SMT use solder in the physical form of paste and then melt this paste in a heat oven to form the solder join. This method of forming solder join gives SMT the advantage below: improves the time of production, improves production capacity in term of quantity of complete product, increases the density of components that can be mounted on the PCB fabrications, and it helps the PCBA technology to produce products with smaller and smaller size. The picture below should illustrate the summary of this technology:



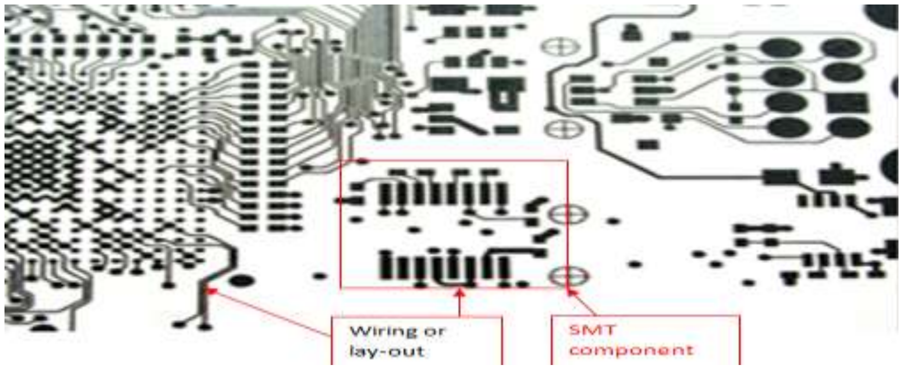
This document will concentrate to discuss how to optimization and design of one of the most important tool in SMT, SMT stencil. This tool is a printing template to deposit solder on the surface of the SMT PCB fabrication. Although SMT stencil is just a tool that help to complete SMT manufacture process, but it plays a very critical role in achieving high quality SMT product. We will discuss a little more detail of SMT technology which will lead to how design a quality stencil for SMT process.

DISCUSSION OF SMT TECHOLOGY

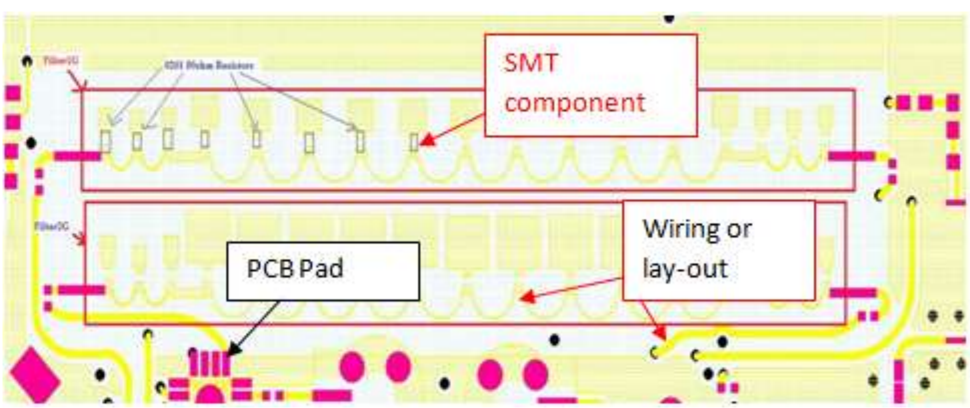
As we discussed in the paragraphs above, SMT technology is the technology that mount components on the surface of PCB fabrication. This is different from pin through-hole and press-fit technology where we mount the components through the body of PCB fabrication. However, in reality, these technologies often use or design concurrent altogether because of their advantages and disadvantages base of the need of applications and availabilities of components. For understand more about SMT we need to understand the process components of this technology. We can distinguish them by the description below: 1) SMT Printing, 2) SMT PCB, 3) SMT Placements, and 4) SMT Reflow. However, for stencil design purpose, we will concentrate on SMT printing and SMT PCB fabrication.

First of all, we need to understand what are differences between SMT components in comparison with other process components? At the beginning of this document, we discussed about pin through-hole and press-fit technology. These two technology have the similarity is that they are both mount through the body of the PCB fabrication, but they were different because one technology uses solder to mount, and the other technology uses pressure and physical dimension to mount. SMT is completely different from these two types of PCBA process because it is only mounted on the surface of the PCB fabrication. This is why it introduced three other terms: Components termination, PCB pads, and solder paste. Component terminations are metallic area that attaches directly on the SMT components for the purpose is that it will be the bonding area that connected with the PCB fabrication. In relation, PCB pads are metallic area on the PCB fabrication surface to serve the purpose of bonding PCB interconnection to the component interconnection using SMT process. In order to attach or bond the component's termination and the PCB pads, solder paste or SMT epoxy are used to help these metallic chemical bonding. The relationship between these three terms is generally understood as the following: PCB lay-out, Component Termination design, and PCBA bonding.

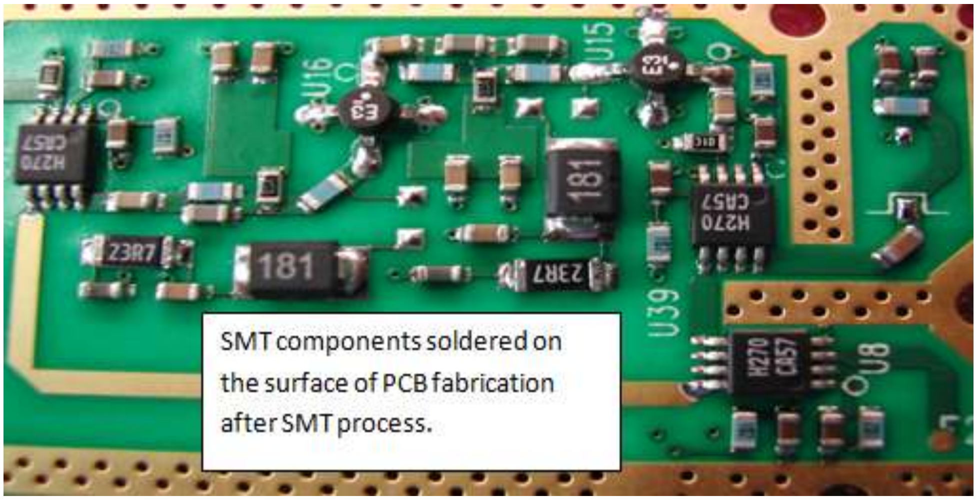
PCB fabrication is the PCBA component that lay-out all the wiring of electronic circuits. PWB (printed wired board) and PCB are similar technologies that condense complex physical wiring of circuit design into an organized manner that will give the same function as physical wiring. The relationship of SMT component, PCB lay-out, and PCBA process are discussed in two steps below:



1) While these wiring hide within the body of the PCB fabrication, each of the circuit connection will represent on the surface of the PCB fabrication by PCB pads. In other words, PCB pads are metallic surface that directly connected to the wiring within the body of the PCB fabrication. The PCB pads are design base on the external components pins or termination that will assemble on the PCB surface. These components has its own function to the circuit, when they are assemble on the PCB surface, they will connect to the internal wiring of the PCB and complete the circuit design. Thus, if a designer use similar wiring and different component can create different assembly and different circuit design. This is the strongest advantages of the PCB lay-out in comparison with physical wiring. The picture below illustrated the SMT PADs and SMT component placing on the PAD:



2) Between the component terminations and the PCB pad is the solder paste. After the solder paste melted, it will form a solid solder join like the picture above, which will connect functional components to the internal circuit wiring of PCB fabrication to complete the circuit. the picture below should illustrate the idea:

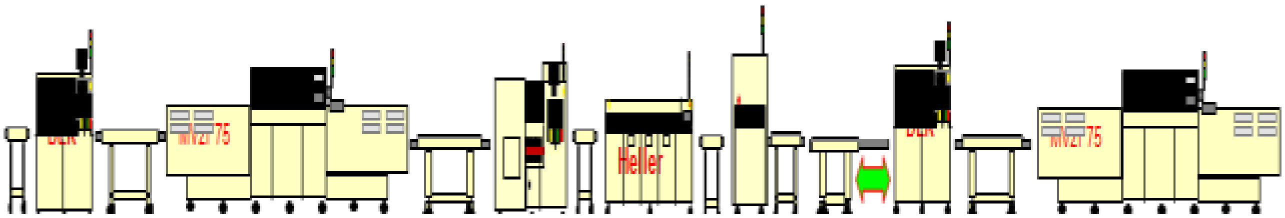


SMT PROCESS AND BASIC EQUIPMENTS

SMT processes and basic equipments are created to achieve the manufacture capabilities of SMT technology. In this section, we will briefly discuss some basic equipments and equipments lay-out to perform manufacture of SMT. This discussion also refer to some basic process that apply to SMT manufacturing.

Basic of SMT Equipments Lay-out:

Deviate from the idea of an assembly line, SMT equipments are lay-out in line. Each equipment will provides some of the basic function and basic process requirement. First of all, let summary SMT line lay-out. The general SMT line lay-out will have these basic equipments that follow SMT process sequences: ***1) SMT printer, 2) SMT Transport Rail Unit, 3) SMT Automated Paste Inspection unit (Optional), 4) SMT Chip Shooter, 5) SMT Transport Rail Unit, 6) SMT IC Placer, 7) SMT Transport Rail Unit, 7) SMT Oven, 8) SMT End Transport Unit, 10) SMT Optical Inspection: 5DX, 3DX, X-Ray, or AOI (Optional), and 11) SMT In process QA Station***. We will briefly describe in the following sections about each of equipment because they will introduce some of the basic processes of SMT, and the picture below should illustrate the line lay-out described above:



1) SMT Printer: use to print solder paste on the surface of PCB fabrication. The main tool for this equipment is a SMT stencil with special aperture cut base on the pad lay-out of each PCB fabrication. This meant that all will be unique for each assembly and each PCB side defend on assembly. We will discuss more about how important is a SMT stencil and how we design them in later discussion of this document.

2) SMT Transport Rail Unit: use to transport PCB fabrication along the line from one machine to another. Some time, the equipment itself have build in transport rail, but additional transport rails are used ensure smother process.

3) SMT Automated Inspection Unit (Optional): this is optional equipment because it is not required and applied to all SMT line. Some contract manufacture use, semi automated inspection tool to perform the quality inspection, and while the other contract manufacture might use automated inspection. This type of SMT equipment is to help inspect quality of printing process. This type of equipment will inspect on alignment of the printing, solder paste height, solder paste volume, and other solder paste printing defects. This is optional equipment, but they are very important in SMT process and SMT quality because more than 70% of solder defects begin from SMT printing.

4) SMT Chip Shooter: some company might refer this machine as passive placement base on their process. However, Chip Shooter is more accurate term because this machine will place small IC such as: SOP, TSOP…etc. Most of the time, this machine will place both small IC and passive components such as: capacitors, resistors, inductors, op-amp, oscillators…etc. This machine will place components with really high speed, and this is the main reason that it used to place only small IC and passive components.

5) SMT IC Placer: use to place bigger and more complicate IC such as: QFN, BGA, LBGA…etc. This machine is armed with more optical and alignment capabilities. It can be programmed to place IC with more termination pins and more complicated pad layout components. It usually place part with slower speed because these part require higher accuracy of SMT placement.

6) SMT Oven: the ideal is like a regular baking oven with extreme high temperature that able to melt solder into liquid. After solder paste printed on the PCB fabrication, and the placing all SMT components is done, then the assembly board will send over this oven to

melt the solder paste and create the solder joints between component terminations and PCB pads.

7) Optical Inspection Units: these units are varies in capabilities and purpose because they more concentrate on specific inspection after the solder joints came out of the oven. They are important but optional because these equipments are purpose to enhance quality inspection only. Many companies still use visual inspection as the main QC methods.

8) QA and QC station: this is human operate table or station, where visual inspection performs not by machine but human. This station also purpose to ensure defect are recorded and fixed before moving further in PCBA processes.

The Role of SMT Stencil in Surface Mount Technology:

SMT printing begins the SMT production processes. This is the reason that it becomes the first most important process to achieve high quality in SMT production. Although to achieve high quality in SMT printing requires other process control such as: good condition equipment, good printing parameter and set-up, good printing support, good print cleaning, good printing operator, and good print tooling, but good printing stencil plays one of the most important role in the achievement of high quality SMT PCBA process. This is why many PCBA process engineer and project engineers concentrate to produce a good SMT stencil for printing process because it is the main tool that cannot be maneuvered or chance unless getting a new one. At the least, if SMT production start with a good designed SMT stencil, it will help to minimize at least 60% to 70% SMT defects besides process handling, SMT placement, and SMT reflow. Many of PCBA engineer believe that if they can have a very good stencil design, they can prevent 60% to 70% of SMT solder defects for the whole SMT process. This has triggered many discussions of how to design a good SMT stencil in the PCBA industry. Each of discussion provides different observations that help engineers to design SMT stencil. This short document will provide an observation that will help to inform readers on how a SMT stencil design begin and complete? What are requirements of a good SMT stencil design? What is the optimization method? What are factors that help to produce a good SMT stencil design? And how achieve this goal with engineering approach?

SMT STENCIL DESIGN, PREPARATION, AND CONSIDERATIONS

Many technical papers and researches introduce different approach of achieving good SMT stencil design. In this short writing, the writer would like to introduce a systematic approach that helps engineer or designer to review, to prevent, and to improve their stencil design. This document also includes the discussion of how to minimize the SMT printing defects such as: over print, under print, solder bridge…etc.

PREPARATION AND CONSIDERATIONS

In order to design a good SMT stencil, the writer would like to introduce a systematic approach to achieve this goal. This system consist of four steps: CAD or GERBER review, BOM review, Standard Suggestion Review, and PCB fabrication review. We will discuss each steps and its function:

1) CAD or GERBER Review: CAD or GERBER is always required because this information is not only used to make stencil but for SMT placement process it is required to help SMT programming. Usually, this information always provided.

2) BOM Review: many engineer might not think of this step as a relevant step, but it play a very important role for stencil design process because of the information that provide in the BOM. The information usually available provide from BOM: part packaging type, part packaging size, and process type.

3) Standard and Suggestion Review: some of the packages have special suggestion for stencil design from manufacture, stencil design also have certain standard design, for some companies also available their own stencil design guideline, and at review standard will help the designer to set his or her mind on certain type of stencil design such as NO-CLEAN or CLEAN design, LEAD or LEAD-FREE solder printing design.

4) PCB Fabrication Review: optional but the most important step. The solder sample is not always available to the stencil designer, but it provides many missing information that not provide in CAD, GERBER, BOM, or Standard Guideline. The information provides by sample PCB fabrication include: variation in dimension of PCB pad from GERBER to actually fabrication, is there any DFM that normally not see by CAD or GERBER, and how this actual PCB fabrication impact on the stencil design.

SMT STENCIL DESIGN

This stencil design suggestion will base on IPC-7525 Rev L that released in 2005 and IPC-7095 Rev A that released in October 2004. The ideas and formulas mention in this document base on suggestion and analysis of these two standards above. We will begin by looking at general guideline suggested by IPC-7525 Rev L, and then discuss in detail with analysis of IPC-standard and then move forward to specific application of these analysis.

IPC-7525 Rev L General Guideline and Formulas:

According to this IPC-7525L standard, two factors that help a designer to calculate and estimate the quality of his stencil design are: Aspect Ratio and Area Ratio. However, we also introduce another factor that not directly mention in IPC document is solder paste volume estimation.

1) ***Aspect Ratio:*** is the calculation of percentage that solders paste release through an aperture onto the surface of PCB fabrication or PCB pad. This is a simplified version of area ratio. This relationship is simply compared width of the PCB pad and thickness of stencil foil. The standard suggests that good aspect ratio is at least greater than ***1.5 or understood as 150%*** solder release on the PCB pad through a aperture when the printing squeeze sweep the solder over the aperture. This is the simplified version of area ratio when the length is much greater than the width so the standard suggested the formula below:

$$\boldsymbol{Aspect\ Ratio = \frac{Width\ of\ Aperture}{Thickness\ of\ stencil\ foil}}$$

$$Width\ of\ aperture = W\ and\ Thickness\ of\ stencil\ foil = T$$

$$Sometime\ W = D\ in\ round\ PCB\ pad, where\ W = Width\ of\ the\ pad, and\ D = Diameter\ of\ the\ circular\ pad\ to\ estimate\ aspect\ ratio$$

$$\boldsymbol{Aspect\ Ratio = \frac{W}{T}}$$

2) ***Area Ratio:*** is the ratio between the aperture walls and the PCB pad. This calculates the percentage of solder release on the PCB pad with certain aperture design. The standard suggests the good solder release is greater or at least ***0.66 (66%).*** The description of this calculation below:

$$\mathbf{Area\ Ratio = \frac{Area\ of\ the\ pad}{Area\ of\ the\ aperture\ walls} = \frac{L * W}{2 * (L + W) * T}}$$

$$L = The\ lenght\ of\ the\ pad$$
$$W = The\ Width\ of\ the\ pad$$
$$T = The\ thickness\ of\ stencil\ foil$$

3) ***Theoretical Solder Paste Volume:*** this idea was not discuss detail in IPC standard and no special suggested formula, but it played a very important role in solder paste quality control. *Theoretical Volume is the volume of solder paste intended to deposit on the PCB pad*. When an aperture designed, there is certain amount of solder paste that intended to deposit on the PCB pad, and this intended volume is calculated by normal volume theory. Using solder paste volume comparison between theoretical volume and the actual volume deposited on the PCB pad can show how accurate and how good solder release and deposit after printing process. For example, if an aperture size 10x10 mils with a stencil thickness is 5 mils then the theoretical volume is 500 $(mils)^3$. For some reason, the measure volume of this aperture is 300 $(mils)^3$, then it raises the question why this volume is too far different from the intended or theory volume. The stencil designer or the engineer can go back and check if the aperture's cut was accurate as he designed, or was other factors that caused these two volumes too far different from each other. Thus, understand theoretical volume can help

control, improve, and trouble shoot SMT printing process. The theoretical volume is thought as the following formula:

$$\textbf{Theoretical Solder Paste Volume} = (\textbf{Area of the aperture} * \textbf{Thickness of stencil})$$

Stencil Design's Formula and Apertures:

$$Area\ Ratio = \frac{Area\ of\ the\ pad}{Area\ of\ the\ aperture\ walls} = \frac{L * W}{2 * (L + W) * T}$$

$$L = The\ lenght\ of\ the\ pad$$
$$W = The\ Width\ of\ the\ pad$$
$$T = The\ thickness\ of\ stencil\ foil$$

$$Theoretical\ Solder\ Paste\ Volume = (Area\ of\ the\ aperture * Thickness\ of\ stencil)$$

Analysis of Standard Area Ratio of IPC-7525A:

When look at the suggest formula above, we see that this formula bases on rectangular aperture because:

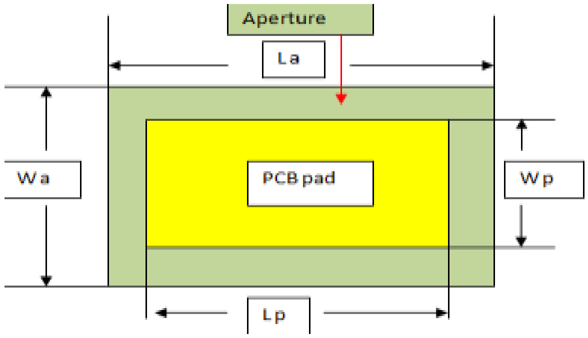
1) ***Numerator is the area of rectangle formula when L*W = Area. This suggested that this formula bases on rectangular pad.***
2) ***Denominator is 2*(L + W)* T or we can thought this as (T * Perimeter of rectangle).***
3) ***If I use this analysis and look at the formula one more time, it would be like this:***

$$Area\ ratio = \frac{Area\ of\ rectangular}{Perimeter\ of\ rectangular * Thickness\ of\ stencil} = \frac{L * W}{2 * (L + W) * T}$$

This wording of IPC formula does not change the formula, but give use more analysis options because in reality, rectangular pad is not the only type of pad shape use on the PCB, and thinking this way will help to calculate the other pad's geometric. Before we discuss further in specific formula apply for each PCB pad shape, we need to understand this three terms: 1) ratio 1:1, 2) extend 1:1, and 3) reduce 1:1.

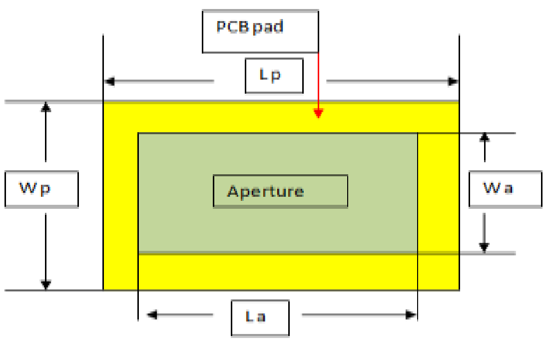
1) ***Area ratio 1:1*** means that the aperture will cut or open exact dimensions of the PCB pad.

2) ***Area Extend 1:1*** means that the aperture will cut or open by extend each dimension of the PCB pad one. The picture below should illustrate the idea:



$$L\ a\ (width\ of\ aperture) = L\ p\ (length\ of\ PCB\ pad) + 1$$
$$W\ a\ (width\ of\ aperture) = W\ p\ (Width\ of\ the\ PCB\ pad) + 1$$

3) ***Area reduce 1:1*** means that the aperture will cut or open after reduce all dimensions of PCB pad to 1 unit. The idea should illustrate in the picture below:



$$L\ a\ (width\ of\ aperture) = L\ p\ (length\ of\ PCB\ pad) - 1$$
$$W\ a\ (width\ of\ aperture) = W\ p\ (Width\ of\ the\ PCB\ pad) - 1$$

Formulas for Popular Aperture:

In this following section, we will discuss in detail of some popular aperture use in the industry. We will assume that these formulas will not apply for not 1:1 design.

1) Rectangular Pad with Rectangular Aperture:

This formula applies for rectangular pad with the design of rectangular aperture:

$$= \frac{\textbf{Area of the rectangular Pad}}{(\textbf{Perimeter of rectangular aperture} * \textbf{Thickness of stencil foil})}$$

$$\textbf{Area ratio according to IPC} = \frac{\boldsymbol{L * W}}{\boldsymbol{2 * (Lm + Wm) * T}}$$

$$Lm = L \text{ in } 1:1 \text{ ratio, other case } Lm \neq L$$

$$\text{Theoretical volume for rectangular aperture} = Lm * Wm * T = \text{thickness of stencil foil}$$

2) ***Square Pad with Square Aperture:***

This formula applies for square PCB pad with square aperture design:

$$\text{Area ratio of square aperture} = \frac{\text{Area of square pad}}{(\text{Perimeter of spuare aperture} * \text{Thickness of stencil foil})}$$

$$\text{Area ratio for square aperture} = \frac{L^2}{4Lm * T}$$

$$L = \text{Lenght of the PCB square pad}$$

$$T = \text{Thickness of stencil foil}$$

$$Lm = L \text{ in } 1:1 \text{ ration, but for other case } Lm \neq L$$

3) Circular Pad with Circular Aperture:

This formula applies for circular PCB pad with circular aperture design:

$$\text{Area ratio of circular aperture} = \frac{\text{Area of the circular pad}}{(\text{Circumference of the circular aperture} * \text{Thickness of stencil foil})}$$

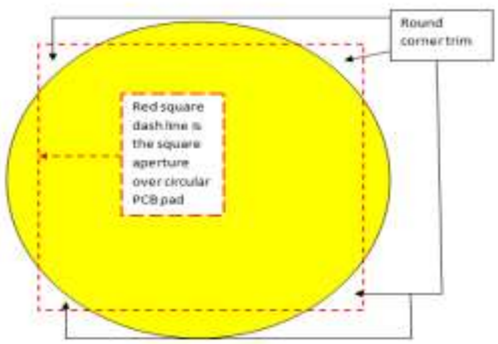
$$\textbf{Area ratio of circular aperture} = \frac{\pi * (\frac{D}{2})^2}{\pi * Dm * T}$$

$$D = Diameter\ of\ the\ PCB\ pad$$

$$Dm = D\ in\ 1:1\ ratio, other\ case\ Dm \neq D$$

$$\textbf{Theoretical volume of circular aperture} = \pi * \left(\frac{Dm}{2}\right)^2 * T = \textbf{Thickness of stencil foil}$$

4) <u>*Circular Pad with Square Aperture with round corner trim:*</u>



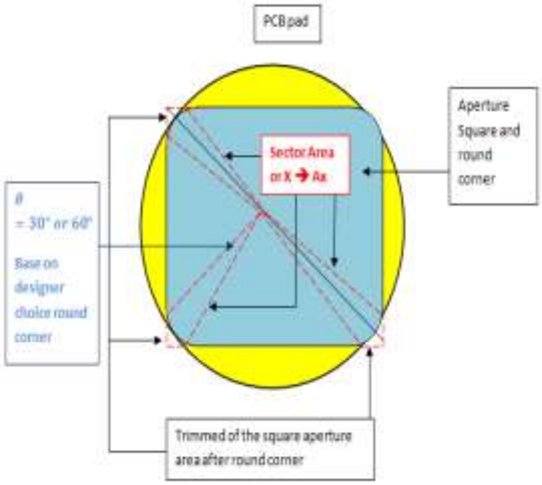
Generally, we can use the formula below to estimate area ratio:

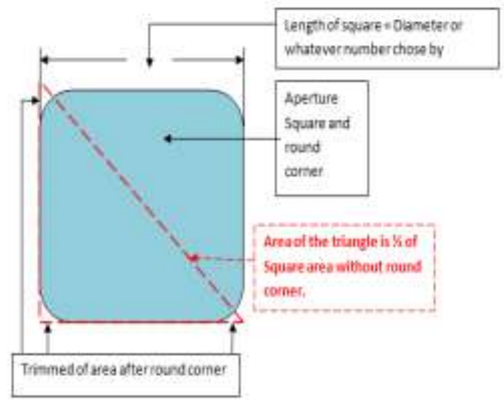
$$\textbf{Area ratio for square aperture over round pad} = \frac{\pi * \left(\frac{D}{2}\right)^2}{(Lm)^2 * T}$$

$$D = diameter\ of\ the\ round\ pad$$
$$Lm = D - some\ units\ chose\ by\ the\ stencil\ designer$$

In some most of the case, the four corners of the square will be trim off to prevent excess printing solder that might create solder balls. We will discuss in detail for this aperture design.

Explanation of this formula:





These two diagrams help to explain the estimation of round corner square aperture over the circular PCB pad. First let analyze the diagrams, the length of square equal to the diameter of the circle or this length can be any number that the designer choose. For this analysis, we will use the length of square equal the diameter.

A) ***Analysis of above diagrams and theoretical volume:***

$$Ls = D, where\ Ls = length\ of\ square\ and\ D = Diameter\ of\ the\ circular\ pad.$$

$$\boldsymbol{At} = \frac{\boldsymbol{Ls^2}}{\boldsymbol{2}}, where\ At = Area\ of\ triangle\ before\ round\ corner\ trimmed\ and\ Ls^2 = Area\ of\ the\ square\ aperture\ before\ trimble.$$

$$Ax = \left(\frac{\theta}{360}\right) * \pi * \left(\frac{Ls}{2}\right)^2$$

$$Ax = area\ of\ the\ round\ corner\ with\ the\ angle\ \theta = 60° or\ 30°\ depend\ on\ the\ choice\ of\ designer.$$

<u>For example:</u> If a designer want the square aperture with round corner over a circular PCB pad with the diameter = 10 mils. He or she can give the instruction below:

- Open Square aperture with the length of 10x10 and round corner 3 or 6. The "3 or 6" means 30 degree or 60 degree for the central angle of round corner section.
- Thus, the area of the trimmed corner for the triangle in the above picture is:

$$Area\ of\ trimmed\ corner\ \frac{1}{2}Ay = At - 2Ax$$

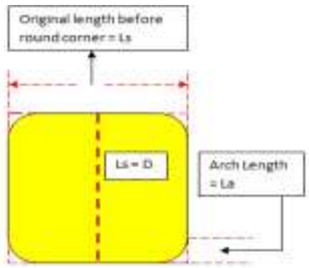
- The total round corner areas were trimmed:

$$Total\ area\ were\ trimmed\ off\ from\ square\ aperture = (Ls^2) - 4Ay$$

$$\boldsymbol{Theoretical\ Volume\ for\ round\ corner\ square\ aperture = (Ls^2 - 4Ay)*T}$$

$$T = thickness\ of\ stencil\ foil$$

Please look at the analysis picture below:



These are calculations for each factor in the perimeter formula:

$$Arch\ Length\ that\ become\ round\ corner = La = \left(\frac{Ls}{2}\right) * \theta$$

Thus, the perimeter:

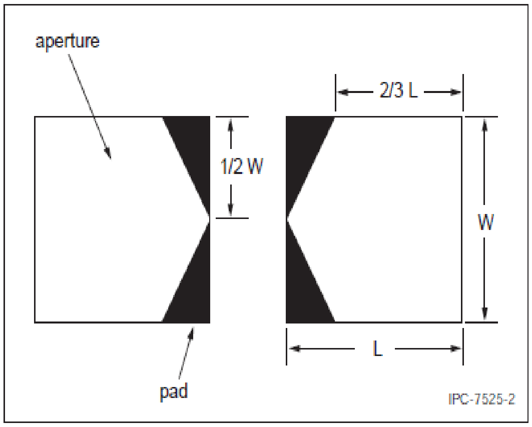
$$Perimeter\ of\ the\ square\ round\ corners = Ps = 4Ls - (4La)$$

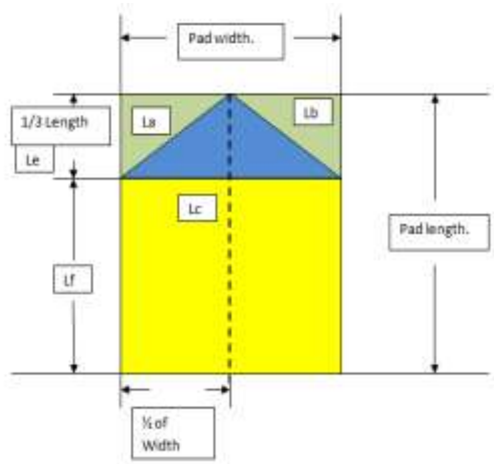
B) <u>Area Ratio for Round Corner Square Aperture over circular PCB pad:</u>

$$\boldsymbol{Area\ ratio\ for\ round\ corner\ square\ aperture = \frac{\pi * \left(\frac{D}{2}\right)^2}{Ps * T}}$$

$$D = Diameter\ of\ the\ circular\ PCB\ pad$$
$$Ps = Perimeter\ of\ the\ Square\ and\ Round\ corners$$
$$T = Thickness\ of\ the\ stencil\ foil$$

5) <u>Rectangular Pad with Home-base Aperture</u>





Home-base aperture is suggested on the right picture by IPC standard. On the left, is the analysis picture base of this suggestion because it is critical to our discussion of home-base aperture's shape?

$$Perimeter\ of\ triagle = La + Lb + Lc$$
$$Perimeter\ of\ rectangle = (Width\ of\ pad + Lf) * 2$$

If I use the left picture to calculate, my relative length La, Lb, Lc lengths are understand as follow:

$$\sqrt{La^2} = (\frac{1}{3}L)^2 + (\frac{1}{2}Lc)^2$$
$$\sqrt{Lb^2} = (\frac{1}{3}L)^2 + (\frac{1}{2}Lc)^2$$
$$Lc = width\ of\ the\ pad$$
$$Lf = \frac{2}{3}length\ of\ the\ pad$$
$$Le = \frac{1}{3}of\ length\ of\ the\ pad$$

Perimeter of the home-base aperture is:

$$Perimeter\ of\ homebase\ aperture = \sqrt{La} + \sqrt{Lb} + Lc + Lf$$

Area ratio of the home-base:

$$Area\ ratio\ of\ homebase\ aperture = \frac{Area\ of\ rectangular\ pad}{Perimeter\ of\ homebase * Thickness\ of\ stencil\ foil}$$

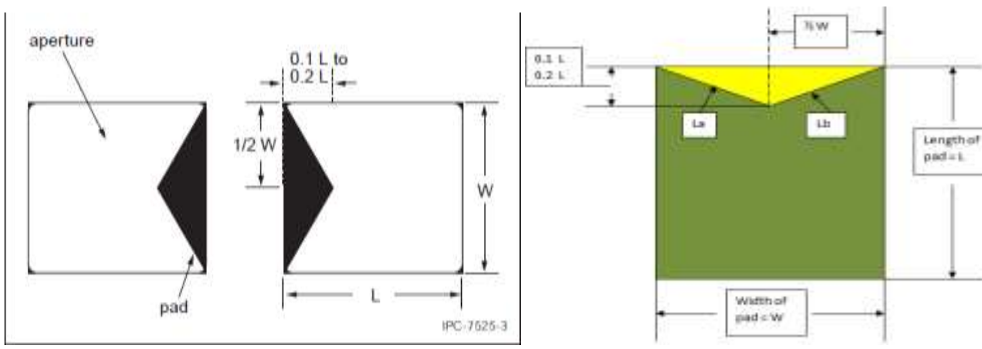
$$\boldsymbol{Area\ ratio\ of\ homebase = \frac{L * Lc}{(\sqrt{La} + \sqrt{Lb} + (2 * lf) + Lc) * T}}$$

<u>Theoretical volume for home-base aperture:</u>

$$Theory\ volume\ for\ homebase\ aperture = (Area\ of\ triangle + Area\ of\ rectangle) * Thickness\ of\ stencil\ foil$$

$$\boldsymbol{Theoretical\ volume\ for\ homebase\ aperture = \left(\frac{1}{2} le\right) * Lc * T}$$

6) *<u>Rectangular Pad with Bow Tie Aperture</u>*



The picture on the right is the suggestion by IPC, and the picture on the left is the analysis base on this suggestion. Thus, the relative calculations are:

$$\sqrt{La^2} = (0.1\ Length\ of\ pad\ or\ 0.2\ length\ of\ pad)^2 + (\frac{1}{2} width\ of\ pad)^2$$

$$\sqrt{Lb^2} = (0.1\ length\ of\ pad\ or\ 0.2\ length\ of\ pad)^2 + (\frac{1}{2} width\ of\ pad)$$

Thus, perimeter of bow tie or inverted home base aperture:

$$Perimeter\ of\ bow\ tie = \sqrt{La^2} + \sqrt{Lb^2} + W + 2L$$

Area ratio calculation as follow:

$$Area\ ratio\ for\ bow\ tie = \frac{area\ of\ rectangular\ pad}{perimeter\ of\ bow\ tie}$$

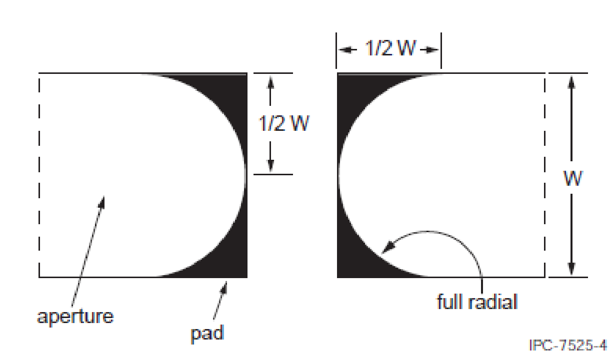
$$\boldsymbol{Area\ ratio\ for\ bow\ tie = \frac{L * W}{\sqrt{La^2} + \sqrt{Lb^2} + W + 2L}}$$

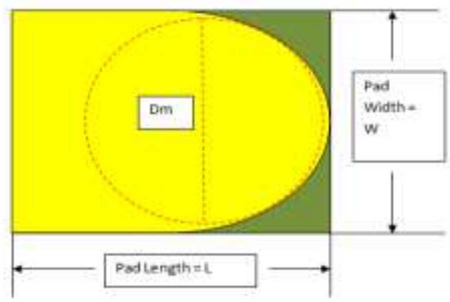
Theory Volume for bow tie:

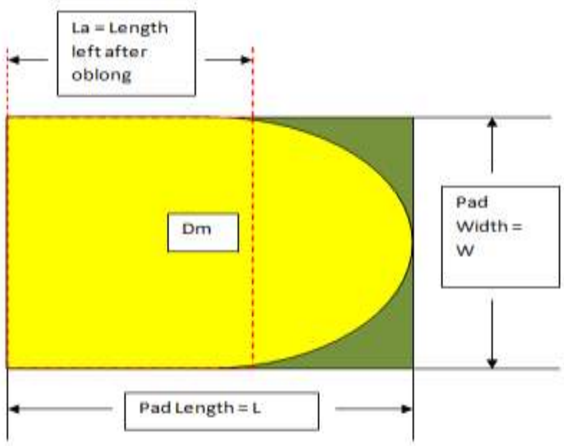
$$Theory\ volume\ for\ bow\ tie\ aperture = (Area\ of\ rectangular\ pad - bow\ tie\ area\ or\ triangle\ area) * Thickness\ of\ stencil\ foil$$

$$\boldsymbol{Theoretical\ volume\ for\ bow\ tie = \left((L * W) - \left(\left(\frac{1}{2}(0.1L\ or\ 0.2L)\right) * W\right)\right) * T}$$

7) Rectangular PCB Pad with Single Side Oblong Aperture:







The picture on the right is oblong design suggested by IPC, and on the right is the analysis diagram of this suggestion. Thus, base on this analysis, oblong design calculations as follow:

Factors:

Dm = Diameter of imagination ➔ Dm = Width of the Pad = W

One side oblong is haft of the circle added on one side of the pad as aperture shape. Thus, relative length calculation as follow:

$$Dm = W$$

$$La = L - Dm\ or\ L - W$$

Perimeter of the oblong:

$$Perimeter\ of\ the\ one\ side\ oblong = \left(2La + W + \frac{1}{2}\ of\ the\ circumference\ base\ on\ W\right)$$

$$\frac{1}{2} circumference\ base\ on\ length\ W = \frac{1}{2}(\pi * W)$$

Perimeter of the whole oblong shape is calculated as follow:

$Perimeter\ of\ one\ side\ oblong$= (La*2) + W + $(\pi * W)$

Area ratio of one side oblong:

$$Area\ ratio\ of\ one\ side\ oblong\ aperture = \frac{Area\ of\ the\ rectangular\ pad}{perimeter\ of\ one\ side\ oblong * Thickness\ of\ the\ stencil\ foil}$$

$$\boldsymbol{Area\ ratio\ of\ one\ side\ oblong = \frac{L * W}{\left(2La + W + \left(\frac{1}{2}(\pi * W)\right)\right) * T}}$$

Theory Volume for one side oblong:

$$Theory\ volume\ for\ one\ side\ oblong = (rectangular\ area\ base\ on\ La + half\ of\ circle\ area\ base\ on\ W) * Thickness\ of\ the\ stencil\ foil$$

$$\boldsymbol{Theoretical\ volume\ for\ one\ side\ oblong\ aperture = \left((La * W) + \left(\frac{1}{2}\left(\pi * \frac{1}{2}W^2\right)\right)\right) * T}$$

8) <u>Rectangular Pad with Double side oblong aperture:</u>

In similar to the one side oblong, but the different are: 1) La calculation will change and 2) instead of half circumference in the perimeter calculation, now is full circumference because both side of aperture will add in oblong. Thus, the relative calculations are:

$$Dm = W$$

$$La = L - (2Dm)\ or\ L - 2W$$

Perimeter of the oblong:

$$Perimeter\ of\ the\ one\ side\ oblong = (2La + Full\ \ circumference\ base\ on\ W)$$

$$circumference\ base\ on\ length\ W = (\pi * W)$$

$$Perimeter\ of\ double\ side\ oblong = (2La + (\pi * W))$$

Area ratio of double side oblong:

$$Area\ ratio\ of\ double\ side\ oblong = \frac{area\ of\ rectangular\ pad}{perimeter\ of\ double\ side\ oblong * Thickness\ of\ stencil}$$

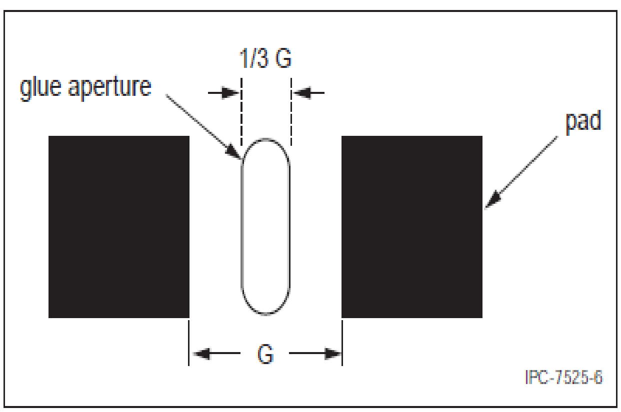
$$\boldsymbol{Area\ ratio\ of\ double\ side\ oblong = \frac{L * W}{\left(2La + (\pi * W)\right) * T}}$$

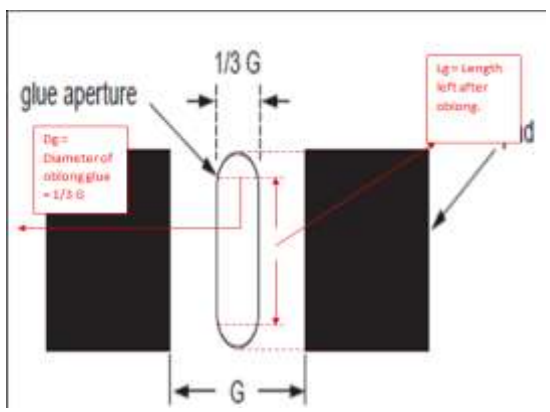
Theoretical volume for double side oblong:

$$Theory\ volume\ for\ double\ side\ oblong = (area\ of\ rectangular\ base\ on\ La + Area\ of\ circle\ base\ on\ W) * T$$

$$\boldsymbol{Theoretical\ volume\ for\ double\ side\ oblong = \left((La * W) + \left(\pi * \left(\frac{1}{2}w\right)^2\right)\right) * T}$$

9) <u>Glue Design Aperture:</u>





This is a suggestion design aperture for glue. On the left is IPC suggestion, and on the right is analysis base on this suggestion. Thus, the aperture shape suggested is a double side oblong. However, the relative calculation is a little bit different. Thus, detail explanation is below:

$$G = Gap\ between\ two\ pads$$

$$Dg = Diameter\ of\ the\ glue\ aperture\ that\ use\ to\ calculate\ oblong = \frac{1}{3}G$$

$$Lg = is\ the\ length\ length\ after\ oblong = W\ of\ the\ pad - 2Dg$$

$$W = Width\ of\ the\ pad$$

Calculation detail of this glue design:

Perimeter of the oblong glue aperture:

$$Perimeter\ of\ the\ oblong\ glue\ aperture = 2Lg + (\pi * Dg)$$

$$Area\ ratio\ of\ the\ glue\ aperture = \frac{Width\ of\ the\ pad * gap\ of\ the\ pad}{perimeter\ of\ the\ oblong\ glue\ aperture * Thickness\ of\ the\ stencil\ foil}$$

$$\boldsymbol{Area\ ratio\ of\ the\ glue\ aperture = \frac{W * G}{\left(2Lg + (\pi * Dg)\right) * T}}$$

$$Theory\ volume\ for\ glue\ aperture = (area\ of\ rectangular\ base\ on\ length\ Lg + circle\ area\ base\ on\ Dg) * T$$

$$\boldsymbol{Theory\ volume\ for\ glue\ aperture = \left((Lg * G) + \left(\pi * \left(\frac{Dg}{2}\right)^2\right)\right) * T}$$

NOTE: *we are questioned why using width of the pad and the gap for the pad area instead of using length of the multiply the width of the pad. The explanation is that this glue aperture will print in the area between two pads. This is why gap between to pads was used as factor and the width of the pad will replace the length of the pad for this case. Glue design is not often use in the normal SMT production process. Most of the time, when glue application needed, a glue dispenser equipment is use to replace printing stencil because printing solder past and glue does not happened in the same time. The printing process of glue has to be separated from solder printing otherwise, if some of the glue or epoxy contaminate the solder paste, it will create contamination defects for SMT solder joints. Thus, this glue application is seldom use stencil printing technology because it usually replace with dispenser equipment to reduce solder joints defects and process complication.*

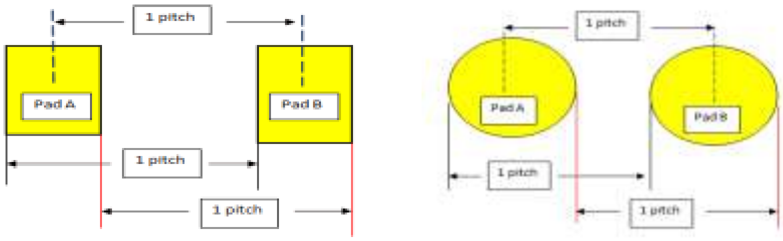
STANDARD APERTURE MODIFICATIONS:

After we discussed several basic formulas for aperture above, the question is how are they applying in SMT stencil design? In reality, after practice from time to time, the stencil design begin to have more experiences, the modification number will be appear in his/her mind and these formulas above will only use for quality improvement analysis. However, they provide deeper understanding of relationship between PCB pad layout, Component layout, and SMT stencil design into another level when it can help to determine or to improve DFM problem. In the following sections, we will use several popular component types to discuss SMT stencil design and application for sample formula above.

<u>Fine Pitch Components and Stencil Aperture Suggestion:</u>

This section will discuss how to design fine pitch component base information given by manufacture data sheet, CAD or GERBER, BOM descriptions, and SMT applications.

1) ***<u>Definition of a Pitch:</u>*** There are three popular definitions that will illustrate below, but we will choose only one for the purpose of our discussion.



Regardless of any kind of PCB pad shape, the three definitions are: right edge of Pad A to the right edge of pad B is one pitch, left edge of Pad A to left edge of pad B is one pitch, and the center point or midpoint of pad A to the center or midpoint of pad B is the pitch. However, look at these three definitions we will think of a pitch in term of formula below:

$$\boldsymbol{A\ pitch = Width\ of\ the\ pad + The\ Gap\ between\ two\ pads}$$

For ideal design and condition:

$$\boldsymbol{Width\ of\ the\ pad = The\ Gap\ between\ two\ pads}$$

2) ***<u>Type of components:</u>*** type of components that need consideration of these ideas are BGA, LBGA, PBGA, CSP Lead Less, QFN, SMT Connectors…etc.
3) ***<u>Definition of Fine Pitch:</u>*** Generally, any component that have .5 mm = 19.68 mil or 20 mils pitch components to the low pitch are consider fine pitch.
4) General Rule of Fine Pitch Aperture: Always try to get back to the ideal condition that width of the pad should equal to the gap between two pads. However, always consider or calculate the aspect ratio, area ratio, and theoretical volume if need to verify the aperture design. Designers should not forget that IPC-7525L suggested that aspect ratio should equal or greater than 1.5 and area ratio should equal or greater than 0.66. Designers should use these suggestions as the limit control for their aperture design. The following table should help to consider the aperture design, and let not forget these are guideline only, and in case by case study, actual calculation should help. These suggestions below base on 5 mils thickness stencil.

The stencil technology is laser cut and electro-polish stencil. We will discuss fine pitch only because they are currently popular and have more problems. Any other pitch QFN can use reduce 1:1 and oblong aperture as standard for consideration.

A) For QFN, SMT connector, or other components with rectangular pad with 20 mils:

Pitch (mils)	Units (mm)	Unit (mils)	Width (mils)	Gap	Modifications	Aperture Shape
20	0.5	20	10	10	Can keep width same length	Oblong (single/double depend on pad
			11	9	Reduce width 1 mils	Oblong (single/double depend on pad
			12	8	Reduce width 1 1/2 mils	Oblong (single/double depend on pad
			13	7	Reduce width 2 mil (DFM)	Oblong (single/double depend on pad
			14	6.5	Reduce width 2.5 mil (DFM)	Oblong (single/double depend on pad

B) For BGA with round pad and 20 mils pitch:

Pitch (mils)	Units (mm)	Unit (mils)	Width (mils)	Gap	DFM	Modification/Aperture Shape
20	0.5	20	10	10	NO	Square Round corner Diameter = Length
			11	9	NO	Square Round corner Length = 9
			12	8	Caution	Square Round Corner Length = 8
			13	7	DFM	No suggestion
			14	6.5	DFM	No suggestion

C) For QFN with rectangular pad and 16 mils pitch:

Pitch (mils)	Unit (mm)	Unit (mils)	Width (mils)	Gap (mils)	Modification and aperture
16	0.4	16	8	8	**Double side oblong 1:1**

[redacted]			9	7	**Reduce 1:1 double side oblong**
			10	6	**DFM**
			11	5	**DFM**

D) ***For BGA with round pad and 16 mils pitch:***

Pitch (mils)	**Unit (mm)**	**Unit (mils)**	**Diameter (mils)**	**Gap (mils)**	**Modification and aperture**
16	0.4	16	8	8	**Square round corner with length = 8**
[redacted]			9	7	**Square round corner with length = 7**
			10	6	**DFM**
			11	5	**DFM**

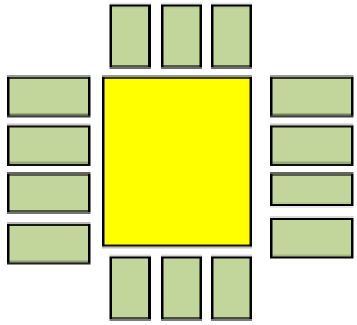
E) ***Other BGA Pitches and Size:***

The other BGA Pitch and size can use these following design considerations:

1) Always check the width and the gap are equal or not
2) Always consider if the lay-out has DFM
3) For BGA that above 20 mils pitch, use standard round 1:1 ratio
4) For BGA with 35 mils pitches to above can consider 6 mils thickness stencil
5) For the gap is 3 mils less than the diameter can be considered as DFM depended on case by case basis.
6) Some time, consider BGA alloy can help improve the solder structure for BGA

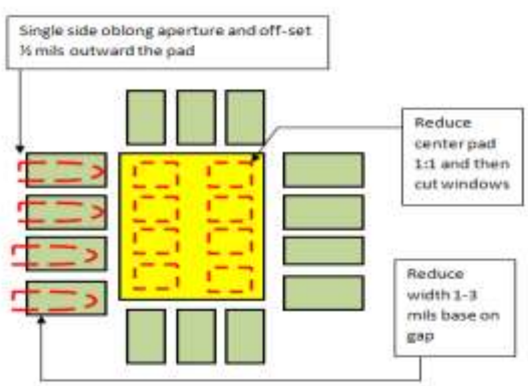
F) ***Lead-Less QFN Package:***

This is a special design component where all the termination leads are hided underneath the package. Many manufacturers suggested different stencil design, and in this document we will suggest a combination of stencil and process design to reduce problem when manufacture this component is PCBA process. The typical component lay-out for this type is illustrated in the picture below and our example is applied for both 20 mil and 16 mil pitch component.



To design stencil for this component type, use the following suggestions:

1) Measure the GERBER: Length, Width, and Gap
2) Compare measurement between the GERBER and the PCB sample fabrication if available
3) Reduce the Length 1-3 mils
4) Reduce the width 1-3 mils defend on gap, use pitch definition to consider how much need to be change in width dimension for aperture
5) Use invert oblong shape where the oblong point to the center ground pad
6) Off-set the aperture ½ mils outward the center of the part
7) Reduce center ground pad 1:1 and then cut windows
8) Stencil technology must use at least laser cut and electro-polish stencil
9) Program SMT placement with very slow or zero pressure placement
10) If possible, provide direct pin support underneath this location when printing SMT
11) Use two cycle cleaning for SMT printing parameter
12) Please look at the picture below to see how the aperture is designed:



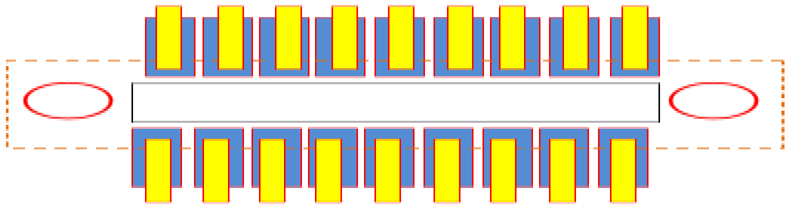
G) Other type of Fine Pitch components:
If we discuss detail how to design each fine pitch component will take a long time to complete. However, there is a great common concept that use for majority of pitch component design is understand how to use pitch definition as the guideline. In the

previous sections, we have discussed pitch formula and some fine pitch component. These discussions base on several main ideas that help designers to designer stencil aperture for all of fine pitch as long as he or she have correct and valuable data. These data are: 1) how big is the gap between two pads, 2) how wide is the width of the PCB pad, 3) how big is the component size, and where on the board that this component will be placed by SMT. These information valuable because:

1) Gap between two pads will help the designer estimate the level of DFM for PCB lay-out and help him to define the pitch of component. When knowing the gap value will help the designer to choose how wide his stencil aperture can be.
2) How wide of is one pad will help the designer to design how much width reduction he can go base on the given PCB lay-out and is the lay-out have DFM problem.
3) How big the component will help the designer to see if this component can be placed by SMT and does is need to have more paste print reduction because the component will be placed by hand.
4) Where the component will be placed on the surface of PCB will help the designer to see what type of thickness stencil can use and is there and additional tool needs to support the printing process.
5) The length was not mentioned because they are less important in pitch components. The general rule of reducing the length is not suggested because it is base on the choice of designer.

The general rules that can apply to design all other pitch components are:

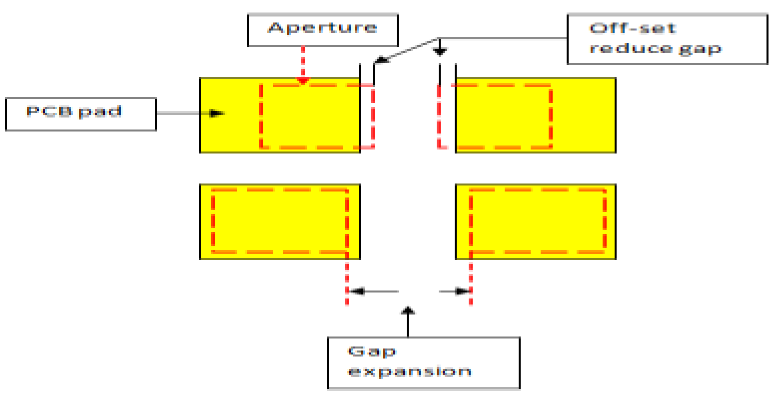
1) Ideal condition, Width of the pad = gap between two pads
2) Reduce width length for aperture from 1 to 3 mils base on the widen of width
3) Helpful to use double of invert single oblong to have more clearance at the head and tail of the pitch component lead. This shape help to prevent solder bridge.
4) Off-set the aperture ½ mils outward the center of the component if needed, usually, this will apply when there is a center pad or center ground pad involve. The gap between all the leads of the component and the center pad should not be bridge unless it is purposed in circuit design and PCB lay-out. Thus, knowing the gap between the center pad and component leads is very important piece of information. Picture below illustrate how off-set look like when print solder paste:



5) For the pad length, designer can keep the same ratio of reduction is 1:1, but recommend that this length should always reduce 1 to 2 mills to prevent the solder printing over the length of the pad.
6) For fine pitch component (between 20 mils to smaller), use laser cut and electro-polish stencil for between aperture opening quality and better solder printing release.
7) In fine pitch component, if the gap is less than two unit of the width of the pad, the designer should caution to consider this lay-out for DFM.
8) With practice, compare the data measure from CAD or GERBER with the data measure from the actual PCB fabrication should help designer to prevent many solder printing defects.

Chip Component: Capacitors and Resistor Design:

1) ***How to change gap in design stencil (Off-set technique):***



The diagram demonstrates how to use aperture off-set or reduction technique to expand or reduce the gap between to PCB pads. Since, we cannot change what designed on the PCB fabrication, this technique will be useful to help expand or reduce gap length. The question is when reducing the gap, some solder will be printing outside the PCB pad and are these solder will become the solder balls or not? The answer is yet. Thus, the designer should not off-set too much outside the PBC pad and it would create solder ball after SMT reflow. The optimum off-set distance is ½ mils for each side.

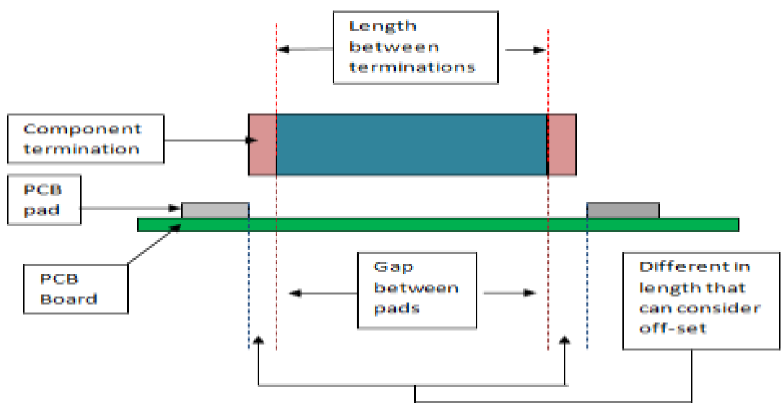
2) ***When are we needed to use off-set? And why are we using off-set?***

A) *When are we needed to use off-set?*

Off-set technique is use to reduce the gap between two pads. This technique is used when the designer afraid of tombstone, solder bridge, or solder wet fill up.

B) *Why are we using off-set?*

For small size passive components such as: 0402, 0201, or 01005 sometime need to off-set the aperture to help the component sit still during reflow. In another word, this technique helps to balance the forces of both pads during reflow and help the tombstone defects. This technique only uses when the gap between two passive PCB pads is larger than standard recommended design. In another word, it is larger than the length between two terminations of the passive chip component. The picture below shows how this relationship applies:



Second, off-set technique uses in QFN, BGA, or fine pitch design is mainly for the purpose of reducing solder-bridge and edge of the lead fillet. Check the suggest aperture design below. Designer should always consider if the lay-out has DFM or not, and this consideration should help him or her to choose appropriate design

Size	Recommended pad width	recommended pad length	Recommended gap	Aperture design
1005	11.0	7.0	6.0	1 to 1
0201	12.0	15.0	9.0	1 to 1

0402	20.0	21.0	12.0	1/3 rule home-base. Shift inward if the gap is more than 12 and reduction if the gap is less than 12
0603	28.0	32.0	23.0	1 to 1
0805	52.0	38.0	23.0	1 to 1
1206	65.0	45.0	60.0	1 to 1
1210	50.0	102.0	60.0	1 to 1

TROUBLE SHOOTING SMT PRINTING PROCESS:

We have not discussed all the SMT components aperture design yet because it is impossible to complete describe each component types and size. However, we have covered the main aspect of the SMT aperture design base on IPC-7525L. This document target to introduce that one the stencil designers understood the interpretations from IPC-7525L standard, it should help them to develop techniques that work for each PCBA because it is always case-by-case study in PCBA stencil design. Research and suggestion from industry will help to enhance knowledge, but it is more help that a stencil designer can detect their design problem. This following section will discuss an optimization method that help to trouble and improve SMT printing. Although SMT stencil is the most important tool for SMT printing, but to achieve high quality, other knowledge of SMT printing should be practice intelligently. Thus, to trouble shoot SMT printing, we should able to answer these following questions when any SMT printing defect occurs: 1) what is the defect? 2) What are possible factors? 3) How good is the stencil cleaning? 4) How good is the printing support? How good is the printing alignment? How good is the aperture design? Is there any DFM? Is there any additional tool need beside the SMT stencil?

<u>In order to answer all these questions, the following trouble shooting sequences suggested:</u>

Define

- Start of Define the printing defect: misalignments, solder bridge, insufficient solder deposit, insufficient solder release...etc.
- Printing problem need to be defined correctly or it will lead to the incorrect implementation

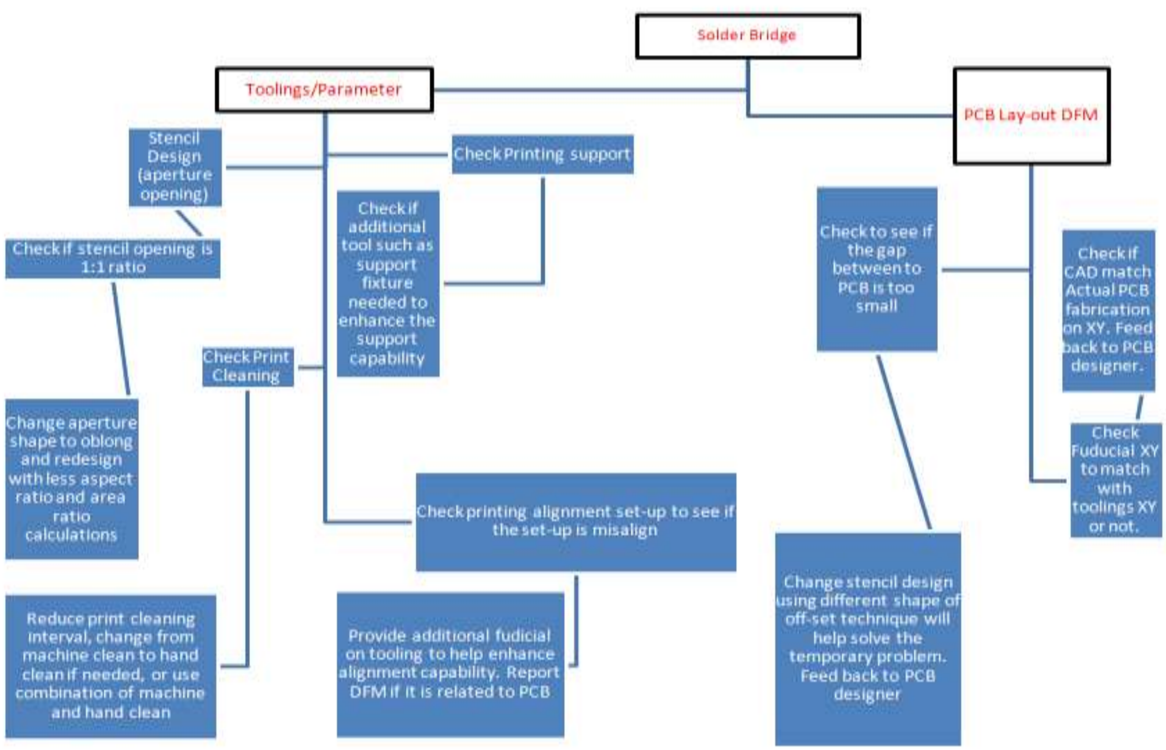
Evaluation

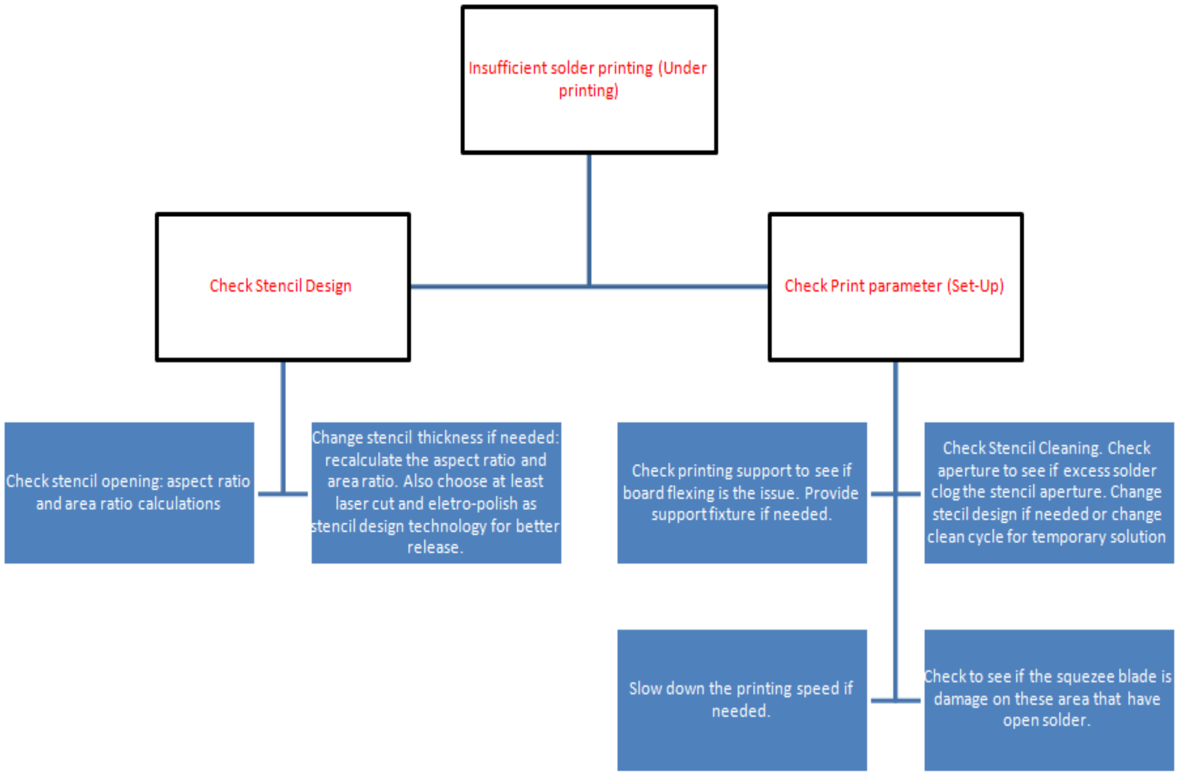
- What are the possible factors?
- Beside printing process, is there any other SMT process that can contribute to the problem (PCB fabrication lay-out or DFM).

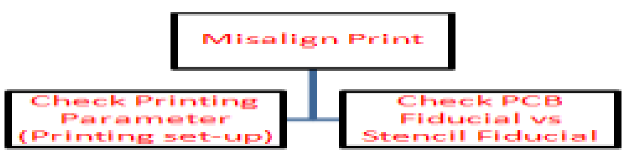
Implementation

- Test the conclusion of which consider as the root cause of the problem
- Reproduce the defect if needed to verify the the factor is the root cause
- Provide solution base on define, evaluation, and test.

<u>Here are some main SMT printing defects:</u>







These are three main stencil printing defects. Other defects that related to stencil printing are solder balls, tombstone, and open solder have discussed by previous aperture design already. When troubleshoot the printing process, the main reference data that manufacture use to estimate the quality of printing are: solder paste volume and solder paste height. These two reference data will help the stencil designer to modify or enhance their design. However, on visual inspections, these three defects above will show is the printing parameter correct? And is there any problem with the design of printing process? Thus, it conclude our discussion of how to design stencil and how to trouble should the performance of the stencil that was designed. Although, we have not complete all the details discussion of the SMT printing and Stencil Design Process, but we have cover most of the main idea how to produce a good stencil design.

<u>References:</u>

IPC-7525L: Stencil Design Guideline, IPC Association Connecting Electronics Industries, May 200, 2215 Sanders Road, Northbrook, IL 60062-6135.

IPC-7095A: Design and Assembly Process Implementation for BGAs, IPC Association Connecting Electronics Industries, October 2004, 3000 Lakeside Drive, Suite 3096, Bannockburn, IL 60015-1249.