APPLICATIONS NOTE
Speedline | MPM
16 Forge Park
Franklin, MA 02038-3137
jperault@speedline.cookson.com
January 4, 1999

Paste in Hole Printing

Paste in Hole, Through Hole printing or Intrusive reflow – printing is quickly becoming a cleaner, more effective alternative to wave soldering as a method for connecting through hole components. This note is a quick reference of the many papers written on this subject and outlines the necessary design criteria for successful printing.

Board / Component design

Correct process designs are very important to the success of through hole printing. If the hole to lead ratio is too large it will require very large stencil apertures, a thick stencil or a large percentage hole fill to obtain an adequate joint. For circular leads the hole diameter should be made approximately 0.012" larger than the pin diameter plus its diameter tolerance, for rectangular pins the hole should be made 0.010" larger than the largest diagonal plus its tolerance. This hole to pin size difference allows for solder paste out-gassing and will help reduce voids. This difference also allows for some placement tolerance for auto-insertion machines.

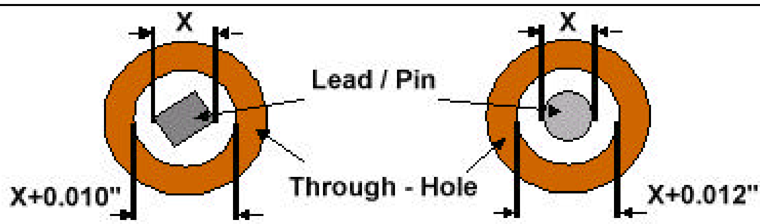


Figure 1: Where X is the maximum diagonal or diameter of the Lead or Pin

Other component design criteria that must be considered are that the components must be able to tolerate reflow temperatures between 220° to 235°C, and the components must also have a standoff of at least 0.010" to clear the paste deposit that is printed.

Calculation of the required Solder Paste Volume

The volume of solder paste, V_{Paste} required is determined by calculating the volume difference between the through hole V_{Hole} and the lead, V_{Lead} and then accounting for the part of the paste which shrinks due to the fact that paste is 50% metal by volume. The formula is:

$$\text{Volume of Solder Paste} = (V_{Hole} - V_{Lead}) \times 2$$

Where the volume of the hole is calculated by the formula for a cylinder:

$$\text{Volume of a Cylinder} = \pi \frac{D^2}{4} h$$

Where D is the Diameter of the Through Hole and *h* is the height of the cylinder or the board thickness in this case, and the Volume of the Lead is calculated by the formula of a brick or that of a cylinder.

$$\text{Volume of a Brick} = \text{Length x Width x } h$$

where *h* is the amount of the lead contained in the hole (i.e. board thickness). Therefore, if you are using a circular lead in a circular hole the equation for the Volume of Solder Paste can be simplified by:

$$\text{Volume of Solder Paste} = 2 \times \left(\pi \frac{D_{Hole}^2 - D_{Lead}^2}{4} h \right)$$

Certain automatic inspection machines require that these leads have a fillet on the bottom in order to accurately inspect the joints. This additional paste is calculated as follows and is added to the Volume of solder paste found above.

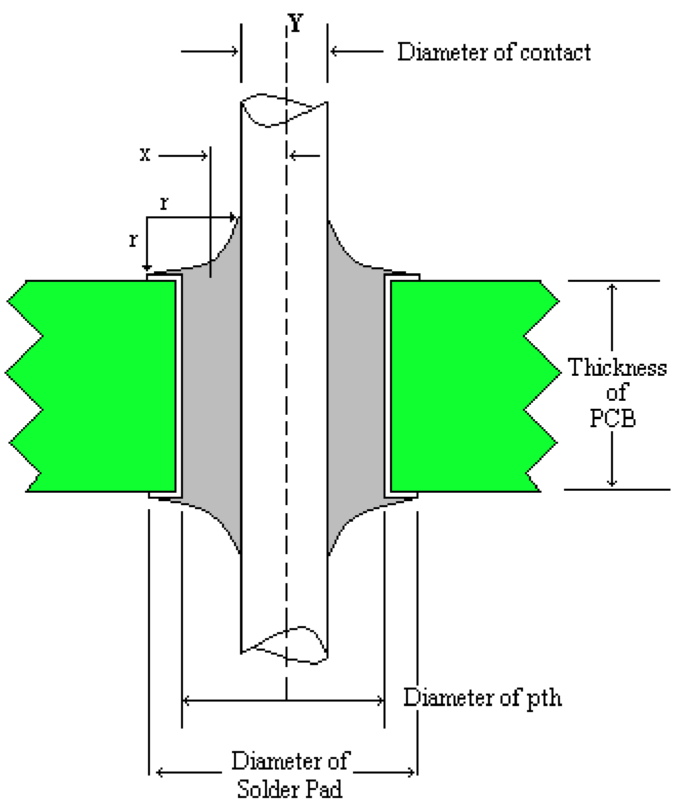


Figure 2: Through Hole Cross Section

<u>Fillet Solder Volume</u>
This is the volume of solder that forms the fillet in and around the lead. It does not include the solder in the Plated Through-Hole.
The area of the fillet is:

$$A = 0.215\, r^2$$

The location of the center of gravity for a fillet is:

$$X = 0.2234\, r + a$$

where “a” is the radius of the contact.

The path of the fillet is a circle, therefore the:

$$\text{length of path} = 2\pi X$$

Therefore the Volume of the fillet is:

$$V_{Fillet} = 0.215\ r^2 \times 2\pi\ (0.2234 \times r + a)$$

And,

$$\text{Required Paste Volume} = (V_{Hole} - V_{Lead} + V_{Fillet}) \times 2$$

Determination of the Aperture Size

The stencil thickness should be designed primarily for the finest pitch devices on the board. The smaller the aperture usually results in the smaller the thickness. Nominal stencil thickness usually ranges between 0.005" to 0.008". Once this dimension is determined the process engineer can use the Required Paste Volume found earlier to back calculate the aperture opening. This aperture will almost always require some amount of over printing and printing on the board's solder mask. The illustration below shows some common techniques. Keep in mind that the process engineer can design any aperture geometry that is needed to fit the amount of paste necessary.

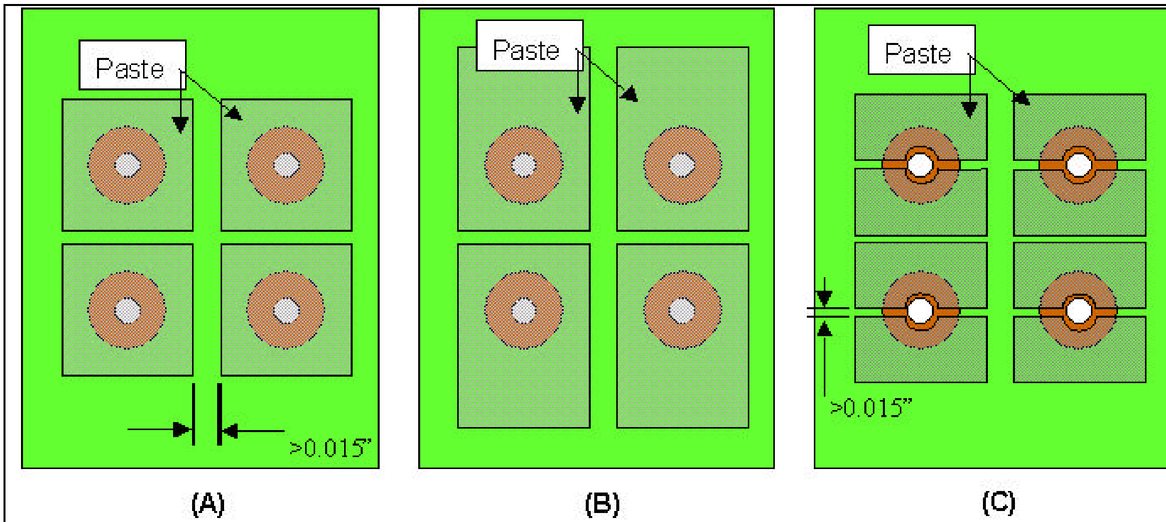


Figure 3: Illustration of Paste depositions

If the aperture size required is too large for the correct amount of solder paste because the hole to pin ratio is too large the engineer may want to investigate looking in to a new component, if the spacing between the holes is too small there are only 2 alternatives the process engineer can look at. Depend upon a percentage of hole fill to compensate for additional paste required, or use a multi – level or step stencil for added thickness around the through hole component(s).

Percentage of Hole Fill

The addition volume required is the difference between the volume of the maximum allowable aperture size and the Volume of Paste required:

Additional Volume of Paste = $V_{Paste} - V_{Aperture}$

The percentage of Hole fill required is:

$$\text{Hole Fill \%} = \left(\frac{\text{Additional Volume}}{V_{Paste}} \right) \times 100$$

Note, a lot of the material in the hole will be pushed out by the pin upon insertion. It is not a really good assumption that a consistent, repeatable amount of paste will stick to the pin every time, this is very paste dependent. The paste's viscosity imparts hole fill and as the viscosity changes over time the percent of hole fill will vary. Viscosity is also effected by temperature and humidity, which will also cause variations in hole fill.

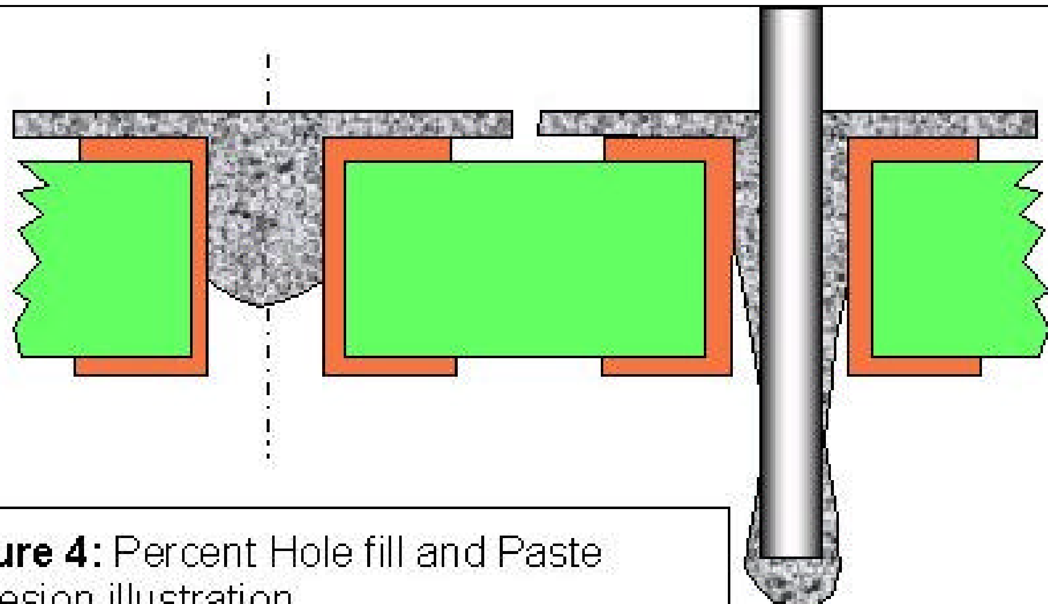
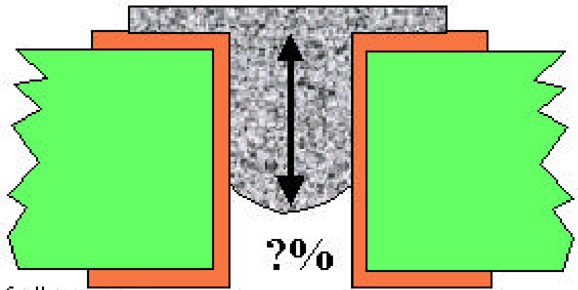


Figure 4: Percent Hole fill and Paste adhesion illustration.

For Example



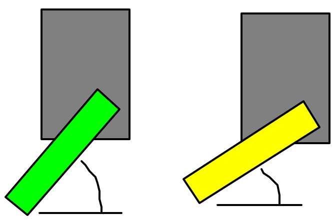
Given the following:
Hole Diameter = 0.027" Board Thickness = 0.062"
Pin Diameter = 0.018" Stencil Thickness = 0.006"
Maximum Allowable Aperture Size = 0.055"x 0.055"

V_{Paste} = 39443 mils^3
$V_{Aperture}$ =18150 mils^3
$V_{Additional}$ = 21293 mils^3

Percent Hole Fill Required = 54 %

Full or partial filling the hole, when necessary, may require special printing parameters. The following is a list of possible parameter changes required and the additional effects that need to be noted:

Parameter / Material Change	Effects
Lower the Squeegee Speed	• Push more paste into ALL apertures • May require lowering Wipe Frequency (e.g. decrease from wiping every 10 boards to wiping every 5) • May increase tendency to bridge on finer pitch devices. • Will increase cycle time.
Increase Pressure	• Push more paste into ALL apertures • May require lowering Wipe Frequency (e.g. decrease from wiping every 10 boards to wiping every 5) • May increase tendency to bridge on finer pitch devices. • Will increase wear on blades and stencil.
Use Polyurethane Squeegee material	• Polyurethane material will force more paste into holes but will also scoop from apertures.

	• Polyurethane will introduce larger volume variations on N-S vs. E-W apertures on QFPs. • May require lowering Wipe Frequency (e.g. decrease from wiping every 10 boards to wiping every 5) • May increase tendency to bridge on finer pitch devices. • The lower the durometer (hardness) of the blade the more the blade will flex and the more the blade will push paste into the hole as well as increasing the amount of scooping.
Lower attack angle of blades 	• This increases the force vector of the paste into apertures. • Polyurethane will introduce larger volume variations on N-S vs. E-W apertures on QFPs. • May require lowering Wipe Frequency (e.g. decrease from wiping every 10 boards to wiping every 5) • May increase tendency to bridge on finer pitch devices.
Increase Vacuum	• On smaller holes this may help pull additional paste deeper into the hole. • On larger holes this may be a problem and result in the vacuum pulling the paste completely out of the hole. • See note on Cleanliness of the machine.

Cleanliness of the machine when filling through holes
When filling large holes or when using vacuum to assist this, paste will often be pulled through the board and onto the below supports or onto the machine table. This needs to be addressed because it can cause machine repeatability issues and downtime, as well as board shorts. When paste is pulled through the board and gets deposited onto the fixture or board supports, the next boards through the machine may have end up with paste residue attached to the bottom (supported) side which, after reflow may cause shorts.

Special board supports can be made to incorporate pockets underneath the through holes that will serve as receptacles for paste that gets forced through the hole. Or special stencil designs can be used (partial donuts) to block the holes from the paste upon printing. The outer edge of these must come in contact with the PTH pad to effectively reflow into the hole. (**See Figure 3 (C)**).

As you can see there are a many things that can be done to assist in filling the holes, but they all can effect the process and the finer pitch printing results negatively. Ideally, the process engineer should try to design both the board and the components to minimize the requirements for filling the hole.

Using a Step-Stencil

Technologies today have introduced better stencil fabrication methods which have resulted in better paste release and reducing the need for step stencils. The maximum step that should ever be used is 0.002" . When larger steps are used the need for increased pressure, larger keep outs and use of Polyurethane blades are required. All of these items may result in poor printing results. The keep out is the additional area around the fine pitch devices that is required to allow the squeegee to conform into the stencil reduction area.

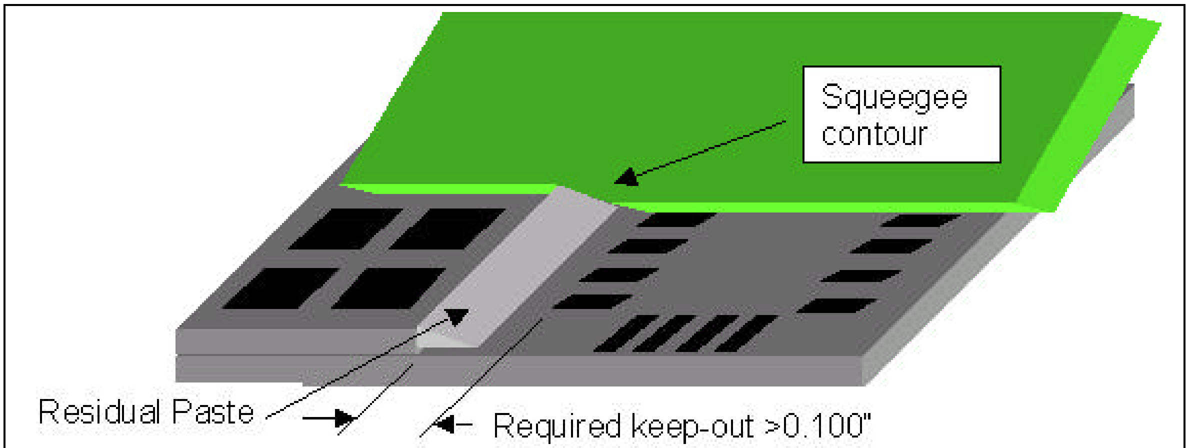


Figure 5: Step Stencil Illustration

The required keep-out may vary some due to the rigidity of the squeegee blade. Stiffer blades will require a larger keep-out. Another concern is that the residual paste that gets left behind in the corner of the step will tend to harden over time. This paste may break away and may get re-deposited and clog other apertures leading to defect in printing. The use of step stencils should be used only as a last resort.

Results – Acceptable Joints

As seen above there are two formulas used to determine the correct amount of solder paste, one takes into account a fillet and the other does not. At times the design criteria above will not allow for "ideal" joints (i.e. fillets on both ends), due to component and through hole centerline spacing, hole size, etc…, because of this the user should refer to the IPC specification for acceptable joints (IPC-A-610A).