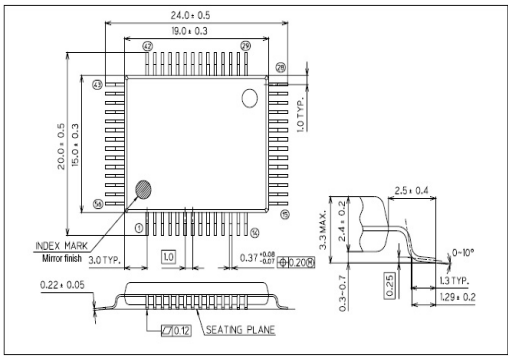
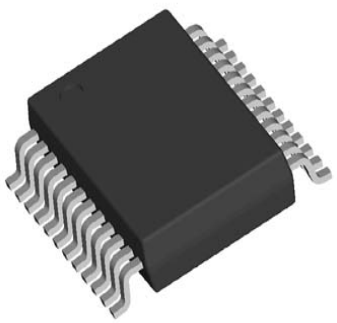
Leaded IC's



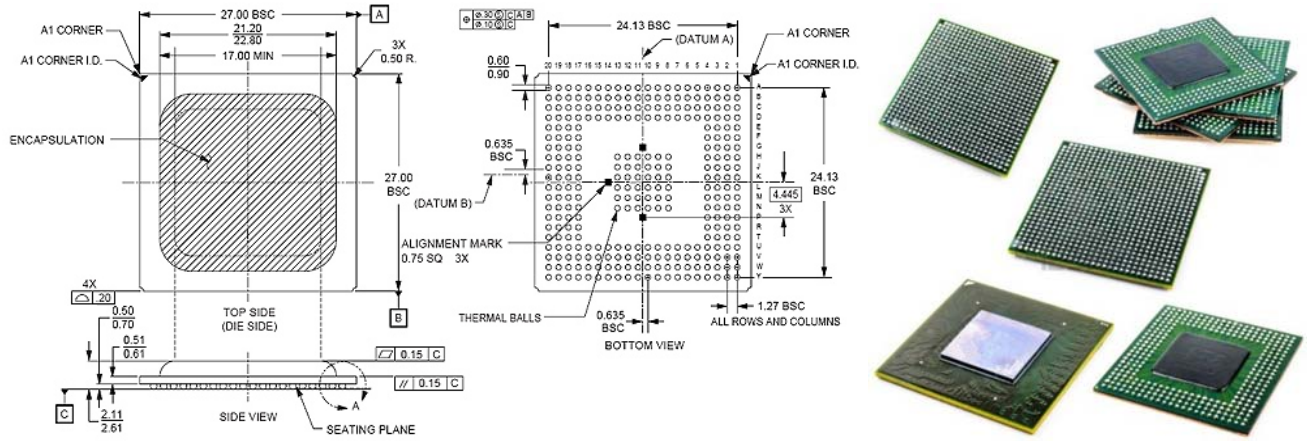


Device	Aperture Length	Aperture Width	Comments or Recommendations	Stencil Thick.
Leaded component > 50 mil (Standard Pitch)	1:1 Oblong or ¼ Radius corners	Min of 12 mil gap between apertures recommended		Any
Leaded component 31 mil (Fine Pitch)	1:1 Oblong or ¼ Radius corners	Max. aperture width = 19 mil Min. aperture width > 9 mil		5, 6 mil
Leaded component 25 mil (Fine Pitch)	1:1 Oblong or ¼ Radius corners	Max. aperture width = 14 mil Min. aperture width > 9 mil		5, 6 mil
Leaded component 20 mil (Fine Pitch)	1:1 Oblong or ¼ Radius corners	Max. aperture width=11 mil Min. aperture width > 8 mil		5 mil
Leaded component 16 mil,	1:1 Oblong	Max. aperture width = 8 Min. aperture width > 7	FG or Eform stencils recommended	3, 4 mil

Common issues with bridging are not selecting the proper stencil thickness and proper stencil reductions. One more item to check is the solder mask resist. Make sure the solder mask is overlaid against the paste mask to check for proper solder resist. Without any solder resists there are more chance of solder flowing or spreading on the PCB which results in bridging.

Note: if there are no solder resist, more reduction may required.

Ball Grid Array

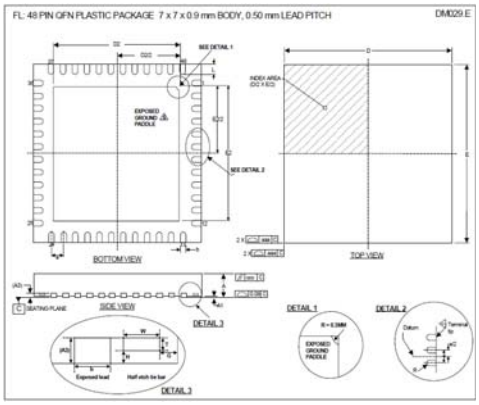


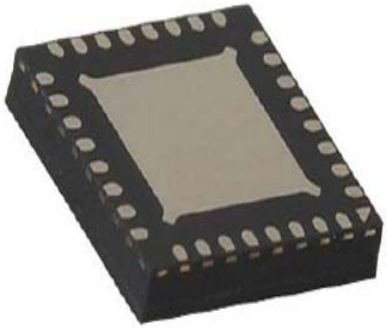
Device	Aperture Length (Ratio to pad)	Aperture Width (Ratio to pad)	Comments	Stencil Thick.
PBGA / CBGA (> 39.4mil)		16 mil min	Square round recommended (keep the pad within the solder mask area)	6 mil
PBGA (= 39.4 mil)		16 mil to 25 mil	Square round recommended (keep the pad within the solder mask area)	6mil
µBGA (31.5 mil pitch)		16 mil to 18 mil	Square round recommended (keep the pad within the solder mask area)	5mil
µBGA (20 mil pitch)		11 mil square round	Square round recommended	4mil
µBGA (16 mil pitch)		9 mil square round	Square round recommended	3, 4mil
µBGA (12 mil pitch)		7 mil square round	Square round recommended	3mil

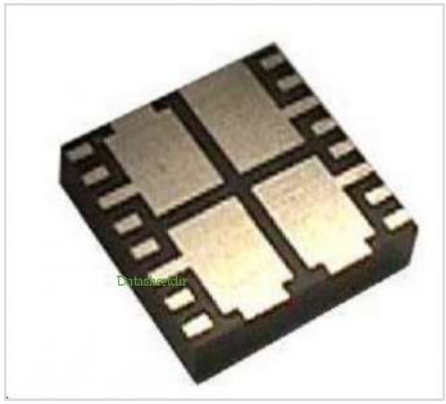
It is very important to choose the correct foil thickness and design the proper pad size. Overprint the micro BGA to get the correct aspect ratio.

Note: if there are no solder resist, more reduction may required.

Leadless packages







Device	Aperture Length (Ratio to pad)	Aperture Width (Ratio to pad)	Comments	Stencil Thick.
16 mil QFN (Quad - Flat-No leads)	Check for solder mask resist and apply reduction accordingly. 7 to 8 mil pad width. Overprint the signal pins or shift by 3-5 mil depends on the length.			3,4
20 mil QFN (Quad - Flat-No leads)	Check for solder mask resist and apply reduction accordingly. 8 to11 mil pad width. Overprint the signal pins or shift by 3-5 mil depends on the length.			4 Mil
25 mil QFN (Quad - Flat-No leads)	Check for solder mask resist and apply reduction accordingly. 8 to12 mil pad width. Overprint the signal pins or shift by 3-5 mil depends on the length.			4,5 mil
31 mil QFN (Quad - Flat-No leads)	8 to16 mil pad width. Overprint the signal pins or shift by 3-5 mil depends on the length.			4,5 mil
Ground pads (i.e. under DFN,QFP, QFN, etc)	50-60% Area Reduction OR 40% X/Y reduction and window pane if necessary. Check if there are open via. If so avoid depositing solder on top of the vias.		Recommend to close the vias before applying solder	

Ground pad designs:

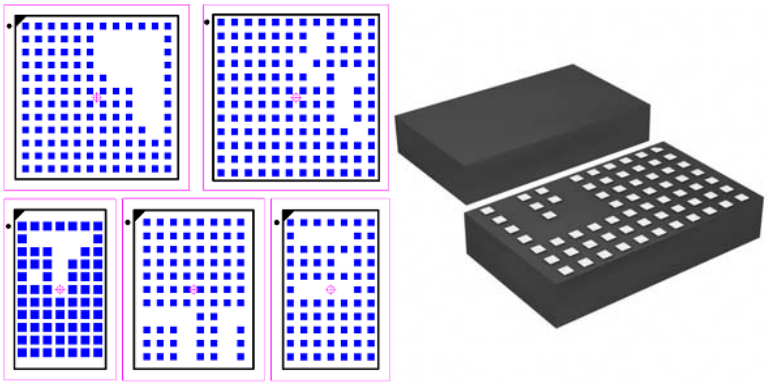

FIG 2


FIG 3


FIG 4


FIG 5

Line Grid Array



Device		Comments	Stencil Thick.
LGA 50 MIL PITCH	29 Mil with square round radius 3 mil, (SHOULD NOT EXCEED SOLDER MASK)		5 Mil
LGA 40 MIL PITCH	23 Mil with square round radius 3 mil, (SHOULD NOT EXCEED SOLDER MASK)		5 Mil
LGA 31 MIL PITCH	17 to 19 Mil square round radius 2 mil, (SHOULD NOT EXCEED SOLDER MASK)		5 Mil

Note: Lower thickness is not recommended due to low standoff of the component. And thicker metal is not recommended due to too much flux to burn off and out gassing issues. Try to see what is recommended by the part manufacture.

Chip components

Table 1

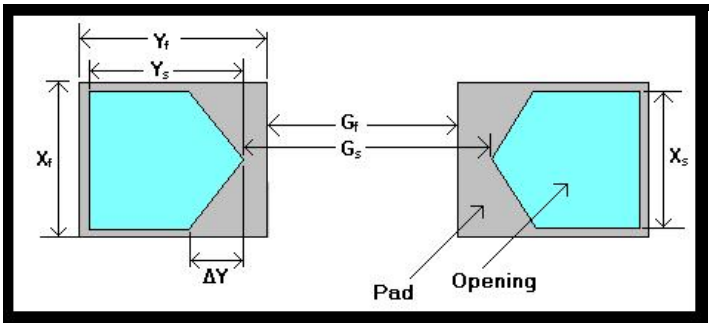


Table 2

Device	Aperture Length (Ratio to pad)	Aperture Width (Ratio to pad)	Comments	Stencil Thick.
1005	1:1 OR 8 mil min pas size	1:1 OR 8 mil min pad size	0.2 mm gap between pads.	3,4
0201	0.5 mil all around reduction	0.5 mil all around reduction	0.28 mm gap between pads. 1/5 home plates for NO clean process. (Table 1)	4
0402	1 mil all around reduction	1 mil all around reduction	Min gap of 15 mil between pads, ¼ home plates recommended for no clean process.	5
0603, 0805	5-10% reduction	5-10% reduction	1/3 home plate for NO clean leaded. 30/40/30 inverse home plates for NO clean lead free. (Table 2)	5,6
1205, 1210	5% reduction	5% reduction	¼ Home plates for NO clean leaded, 30/40/30 inverse home places for No clean lead free (Table 2)	5,6
Tantalum capacitors,	1:1	1:1	Overprint the pads by 5-10% if possible	5,6
MELF C-Shape Pad	1:1	1:1	U – shape with 30/40/30. This will avoid component being roll over.	5,6

We recommend to do either home plate or radius inverted home plates to avoid mid chip solder balls or too much paste between the pads. Depends on the process and the foil thickness a 5%- 10% reduction is also recommended from the pad layout. Make sure the paste file is overlaid against the copper to see if there is any discrepancy between paste and copper layers. Some board shop tends to reduce the pads from copper and create the paste layers.