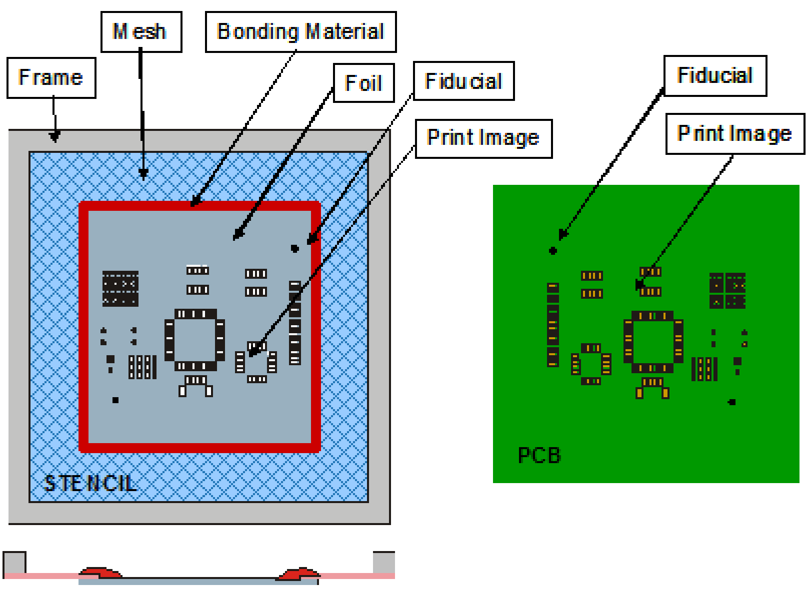
Elek Tasarımı: Karma hassasiyetli PCB'lerde (Üzerinde irili ufaklı ve bacak hatvesi kaba'dan inceye fazla değişen malzemeler olan)

Elek kalınlığı ve bacak hatvesi (aralıklar) orantılı seyretmeli ancak karma PCB olunca büyük malzemeler için yeterince lehim almama veya küçük hatveli malzemelerde fazla lehim alma sonucu kısa devrelerin artması sakıncaları için çözüm olarak;

Eğer tümü için geçerli ortalama bir çözüm aranıyorsa;
1- Ya elek kalınlığı değiştirilmeden üzerindeki delikler %15 ila %30 arasında küçültülecek,
2- Ya elek kalınlığı bir alt düzeye indirilecek,
Ancak yukarıdaki iki yöntem basit ve ucuz olmakla birlikte evdeki bir tek şeker hastası için herkesin şekersiz yemek yemesine benzer ve tasarımı değiştirmeden koşulları zorlama şelindedir. Farklılıklar aşırıysa yeterli olamaz.
İlla da böyle yapılacaksa bazı büyük malzemelerle ilgili deliklerin yözey alanı aynı kalmakla birlikte ızgara şeklinde tasarımı, delik köşelerinin yuvarlatılması, vs..
3- Ya da her malzeme gurubu için farklı kalınlık için ilgili yerleri aşındırmayla inceltilmiş "step-down" bir elek ürettirilecek.

İşin aslı, Ar&Ge ve malzeme satınalma bölümleri işbirliği içinde aynı PCB'ye monte edilecek malzemelerin bu açıdan uyumunu gözeten bir orkestrasyonu tasarım üzerinde yapmasıdır. Başta böyle yapılmış olsada zamanla değişikliklerle de bu durum ortaya çıkabilir.

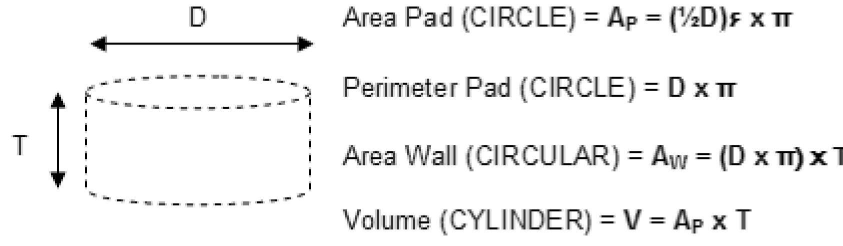
Elek Terminolojisi

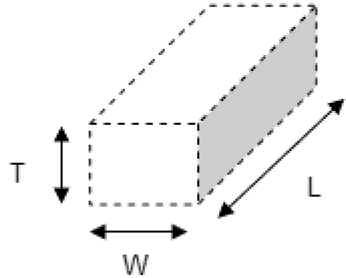


- **Aperture/Delik:** An opening in the stencil that corresponds to a land area on the circuit board to be printed.
- Aperture Design Rules: A set or procedure of rules with recommendations for aperture reductions and modifications in order to optimize stencil design and printing performance.
- **Aperture Reductions and Modifications:** Changes in size or shape of stencil apertures in relation to the corresponding landing pad. These changes are designed to improve paste deposition and yield.
- Aperture Shape: The outline of the opening in the stencil, as viewed from squeegee side. Common shapes include rectangles, squares, rounds, ovals and "home plate."
- **Aperture Size / Delik Ölçüsü:** Refers to the width and length dimensions of the opening in the stencil.
- **Aspect Ratio:** The aspect ratio is the ratio of the aperture opening to the stencil thickness. For chemically etched stencils, this should be greater than 1.5, for laser cut stencils it should be greater than 1.2 and for electroformed stencils which has the best solder paste release characteristics this should be greater than 1.1. Anything less than these recommended ratios will cause the solder paste to stick into the apertures during release as the retaining force of the paste in the aperture will be stronger than the force pulling the paste out of the aperture. See formula.
- **Area Ratio / Alan Oranı:** The relationship between the surface of the aperture and the inside surface of the aperture walls in the stencil. The major difference with Aspect Ratio is that Area Ratio is more suitable to shapes such as circles. Since solder paste has a certain adhesion force, it will stick to the walls of the aperture and to the pad. A ratio of 0.66 pad-to-wall is considered acceptable in our industry (for example a 13.5 mil circle in a 5 mil thick stencil). See formula.
- **CAD/CAM:** Computer Aided Design and Computer Aided Manufacturing is the general terminology for file types used for computerized design and manufacturing.
- **Chem-Etch / Kimyasal Aşındırma:** This is the least sophisticated manufacturing method for making

stencils. It is a photosensitive process that creates a negative photo image on the metal foil. A double sided chemical etching process is used to etch the material away in selected areas resulting in the apertures. The apertures are very smooth but have an hour-glass cross section.

- **Electroform / Elektro-Şekil:** The most sophisticated manufacturing technique for stencils. It is a photosensitive process with a positive image used to make a disposable mandril. Electrolitic plating is used for this additive process where we actually grow the stencil around the apertures. The apertures are very smooth with a very fine resolution.
- **Fiducial / Ölçüt Noktası:** A mark in the artwork that is etched in the stencil along with the apertures. It is used by the machine vision system to align the stencil to the PCB and to verify artwork orientation and location. Fiducials can be fully-etched or half-etched either on the top (squeegee side) or bottom (board side) of the stencil. Sometimes the fiducials are filled with black epoxy to provide contrast between the stencil and the fiducial for visual systems. "Global" fiducials are located outside the aperture footprint while "local" fiducials are placed within the image itself, usually in close proximity to an integrated circuit.
- **File Layer / Katman:** Describes the layer of the PCB that the Gerber information represents. The solder paste layer is a 1:1 reflection of the component lands on the board.
- **Frame Mount /Montaj Çerçevesi:** Stencils are usually secured to an aluminum frame with a tightly-stretched mesh border. Alternatively, a stencil can have mounting holes etched along the perimeter for temporary mounting on a universal sized frame, such as the ALPHA® TETRA™ frame or competing frameless systems. The user requires typically one frame per printer and buys foils only with the stencil image made in the foil. This foil is then tensioned into the re-usable frame.
- **Frame Size /Çerçeve Ölçüsü:** The size of the frame for mounting stencils is determined by the screen printer model. The smaller cast-aluminium frames are usually specified by the inside dimension (ID), e.g., 12 x 17 or 20 x 20 inches. Tubular aluminium frames are usually specified by the outside dimensions (OD), e.g., 29 x 29 inches.
- **Gasketing / Yapışma:** The degree to which a stencil aperture contacts and "seals" against a landing pad. Good gasketing decreases solder "squeeze balls" during printing, that can result in random solder balling and bridging.
- **Gerber Data:** Standardized PCB design language and operating commands which ultimately define the slope and location of apertures in a stencil plate.
- **Land / İletken Alanı:** The conductive area on a PCB to which components or separate circuits are attached. Also referred to as pads.
- **Laser Cut /Laser Kesim:** The most popular manufacturing technique for current stencil technology. The CAD/CAM file is used to directly generate a CNC-file that drives the X-Y driven laser head. The laser will cut the perimeters of the apertures, resulting in tapered apertures. The wall structure is rougher than the other techniques due to the 'melting' effect of the leaser beam.
- **ODB++:** is the most intelligent CAD/CAM data exchange format available today, capturing all CAD/EDA, assembly and PCB fabrication knowledge in one single, unified database. Originally developed by Valor Computerized Systems for use in its own PCB CAD/CAM systems, Genesis 2000, Enterprise 3000 and Trilogy 5000, ODB++ have already become widely accepted as the de facto industry standard, providing unprecedented power to PCB design, fabrication and assembly, with the flexibility to expand as required. In parallel, ODB++ is providing the technological basis for a formal IPC standard
- **Step and Repeat:** Repeating a single Gerber file over two or more areas of the same stencil plate. PCB's are arranged on "break away" panels often referred to as 2 Up, 4 Up, etc. . . . Information on the center-to-center dimensions is essential to prevent mis-registration of panel to stencil.
- **Step Down/Up:** Reduction of stencil thickness in specific areas of a stencil on either the top or bottom side/s. Step UP refers to an area higher/thicker than the majority of the stencil, Step DOWN refers to an area lower/thinner than the majority of the stencil. Stepping is used to control paste deposits on PCB's having both standard pitch and fine pitch components. It is also used for creating an underside cavity to accommodate surface circuitry, via's or other low profile surface components in order to optimize stencil gasketting.
- **Trapezoidal Opening:** The cross-sectional angle of the stencil aperture with the larger opening on the bottom side of the stencil and the smaller opening on the top side. This geometry improves solder paste deposition onto the PCB. The laser-cutting and electroforming processes provide trapezoidal openings, whereas, chemically-etched stencils must be etched from both sides of the stencil with a photoresist on the bottom side of the stencil which has slightly larger apertures than the top side photo-resist (hour glass).
- **KULLANILAN GENEL FORMULLER:**



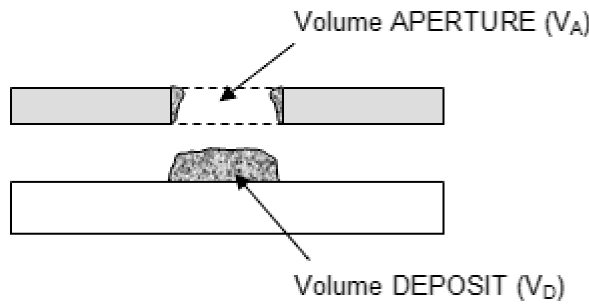


Area Pad (RECTANGLE) = $A_P = W \times L$

Perimeter Pad (RECTANGLE) = $2 \times (W + L)$

Area Wall (RECTANGULAR) = $A_W = 2 \times (W + L) \times T$

Volume (RECTANGULAR/CUBE) = $V = A_P \times T$



Transfer Efficiency (%) = $T_E = V_D/V_A$

Volume APERTURE = V_A = Theoretical Volume Stencil Aperture

Volume DEPOSIT = V_D = Actual or Predicted Volume of Deposit

Aspect Ratio = D_{NA}/T ≥ 1.5

{NARROWEST APERTURE DIMENSION / STENCIL THICKNESS}

Area Ratio = A_P/A_W ≥ 0.66

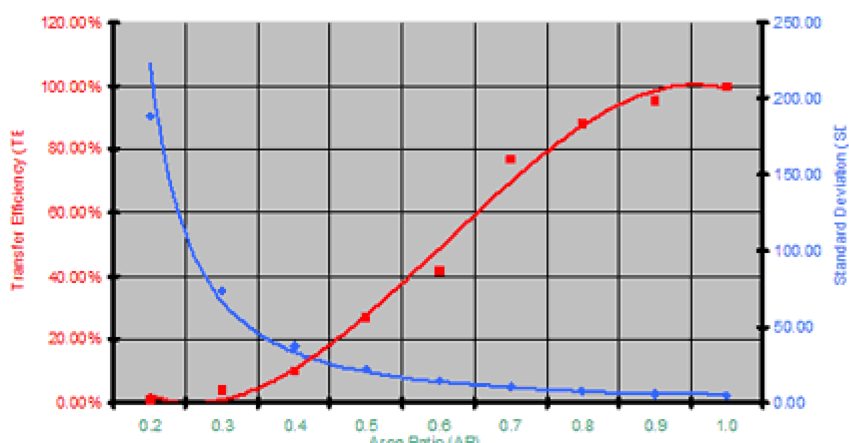
{PAD AREA / WALL AREA}

Stencil Thickness For Mixed Density Boards

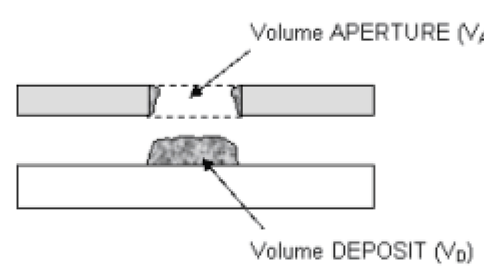
Krem Lehim Serigrafisinde Tasarım İlkeleri:

1. Area Ratio / Alan Oranı

- The relation between the surface of the aperture and the inside surface of the aperture walls in the stencil. The major difference with Aspect Ratio is that Area Ratio is more suitable for shapes such as circles. Since solder paste has a certain adhesion force, it will stick to the walls of the aperture and to the pad.
- A ratio of ≥ 0.66 pad to wall is considered acceptable in our industry (for example a 13.5 mil circle in a 5 mil thick stencil). See formula.
- This factor has the single biggest impact on Transfer Efficiency and Repeatability of the solder paste deposits.



- Transfer Efficiency is the percentage of solder paste of the Theoretical Volume of the aperture that is transferred onto the pad.



Transfer Efficiency (%) = $T_E = V_D/V_A$

Volume APERTURE = V_A = Theoretical Volume Stencil Aperture

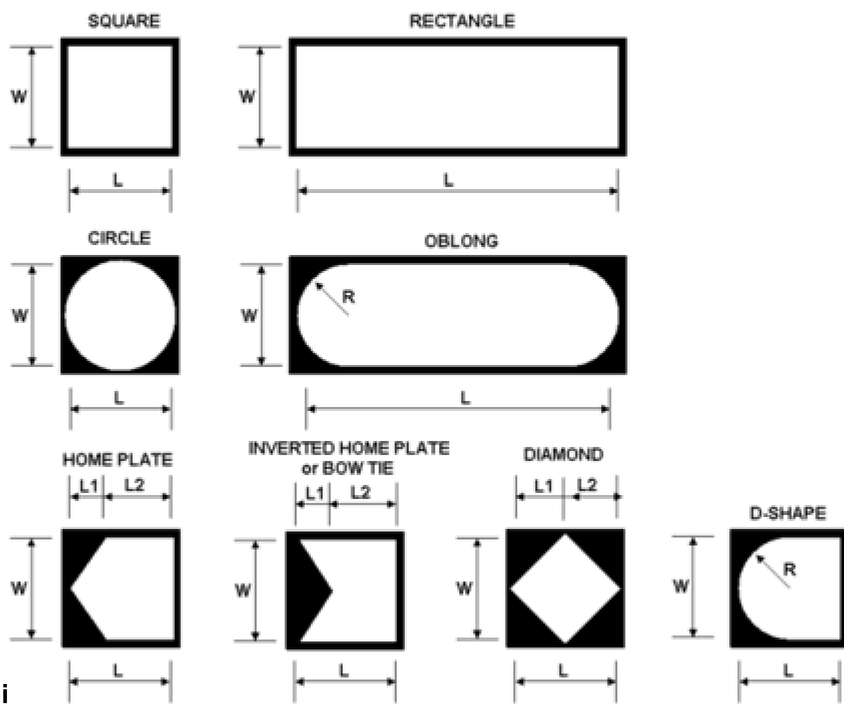
Volume DEPOSIT = V_A = Actual or Predicted Volume of Deposit

- The biggest impact on Area Ratio is stencil thickness:
- Area Ratio = A_P/A_W ≥ 0.66

{PAD AREA / WALL AREA}

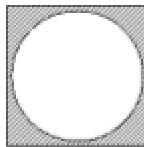
2. Aperture Size / Delik Ölçüsü

- It is recommended that aperture openings are made smaller than the landing pad size. This is called reduction or cropping. The main reasons for this are:
  1. Improved alignment accuracy & repeatability between the PCB pad and the stencil aperture.
  2. Control solder paste volume (prevent bridging & mid ship solder balling).
- Aperture width reductions must be taken equally from each side so that aperture is centered on the pad.
- Aperture lengths can be reduced by similar dimensions to reduce the potential for solder balling.
- For certain components, such as BGA's, CCGA's or other component types that require large volumes of solder, it is recommended to make the stencil apertures larger than the landing pad size. This is called overprinting.



3. Aperture Shape / Delik Şekli

- Different aperture shapes have been found to offer the benefits of better volume control or to prevent defects such as solder balling, bridging, voiding etc. Shapes to consider include:
- Remember a square gives the biggest aperture volume for any given size. The difference in aperture volume is 27% (4/π).



4. Stencil Thickness / Elek Kalınlığı

- Stencil or foil thickness is an important part of stencil design. Optimal paste deposition onto a PCB is impacted by the relationship that exists between the pad size, aperture opening and foil thickness. While the aperture may be appropriately sized for a pad, a stencil that is either too thin or too thick may still cause less than optimal deposition of solder paste.
- This relationship is also known as "aspect." Aspect is the difference in forces that either pull paste from an aperture and on to a pad or cause paste to be held within an aperture. These forces can be quantified and represented as a measurement called the Aspect Ratio. In simple terms, for a paste to be adequately deposited on a pad, the paste surface tension must be stronger that the surface tension of the paste to the aperture wall.
- A broad set of rules has been adopted that help us design stencils with appropriate Aspect Ratios depending on the type of stencil ordered. It is important that the smallest aperture on the board be used for this calculation.

Stencil Type	Ratio of Foil Thickness to Minimum Aperture Width
Chemically Etched / Kimyasal Aşındırma ile	1:1.5
Laser-Cut / Lazer Kesim ile	1:1.3
Electroformed / Elektro-şekledilmiş	1:1.1

		Pad		Thickness Range		Area Ratio Range	
MM	Pitch	W	L	Min	Max	Min	Max
PLCC	1.25	0.65	2.00	0.150	0.250	1.53	0.92
QFP	0.65	0.35	1.50	0.150	0.175	0.83	0.71
QFP	0.50	0.30	1.25	0.125	0.150	0.83	0.69
QFP	0.40	0.25	1.25	0.100	0.125	0.86	0.69
QFP	0.30	0.20	1.00	0.075	0.125	0.86	0.52
0402	N/A	0.50	0.65	0.125	0.150	1.03	0.86
0201	N/A	0.25	0.40	0.075	0.100	0.93	0.69
BGA	1.50	0.75	0.75	0.150	0.200	1.42	1.06
BGA	1.25	0.64	0.64	0.150	0.200	1.23	0.93
BGA	1.00	0.50	0.50	0.115	0.135	1.30	1.11
BGA	0.65	0.33	0.33	0.075	0.100	1.08	0.81
BGA	0.50	0.25	0.25	0.075	0.100	0.83	0.63
MIL	Pitch	W	L	Min	Max	Min	Max
PLCC	50.0	25.6	78.7	5.9	9.8	1.53	0.92
QFP	25.0	13.8	59.1	5.9	6.9	0.83	0.71
QFP	20.0	11.8	49.2	4.9	5.9	0.83	0.69
QFP	16.0	9.8	49.2	3.9	4.9	0.86	0.69
QFP	12.0	7.9	39.4	3.0	4.9	0.86	0.52
0402	N/A	19.7	25.6	4.9	5.9	1.03	0.86
0201	N/A	9.8	15.7	3.0	3.9	0.93	0.69
BGA	60.0	30.0	30.0	5.9	7.9	1.42	1.06
BGA	50.0	25.0	25.0	5.9	7.9	1.23	0.93
BGA	40.0	20.0	20.0	4.5	5.3	1.30	1.11
BGA	25.0	12.5	12.5	3.0	3.9	1.08	0.81
BGA	20.0	10.0	10.0	3.0	3.9	0.83	0.63

Once you start using these design rules, we can convert them into your personal design rules which allow you to have them customized to meet all your printing application requirements.

Yapıştırıcı Serigrafisinde Tasarım İlkeleri:

Stencil Manufacturing Method / Elek Üretim Yöntemi

- Laser cut stainless steel stencils are recommended for printing adhesives.
- For apertures whose minimum span is less than 15 mils (0.375mm), laser cut electro-polished stencils should be used.
- For applications with wider tolerances, chemically etched stencils can be used successfully.

Stencil Thickness / Elek Kalınlığı

- 6 mil (150µm) is the typical stainless steel stencil thickness used. For most applications using the typical range of passive components, a stencil thickness of between 6 – 12 mils (150 – 300µm) is appropriate.

Aperture Design /Delik Tasarımı

- The following table provides recommended aperture shapes and dimensions for the most common components attached using SMD adhesives.
- Cookson recommends the double dot or slot configuration because they give the greatest process window for adequate adhesion and proper placement of the component.
- The pinched slot or oval are modifications of the typical slot and are designed to improve the aperture's air release characteristics.
- Recommended Aperture Sizes, based on an 8 mil (200µm) thick laser cut stencil.
- 1mil=2.54mm

Malzeme Tipi	Tek Nokta Ölçüsü	Yarık Ölçüsü	Çift Nokta Ölçüsü	Sıkılmış Yarık Ölçüsü
0402	12 mil	see Pinched Slots	see Single Dot	12 mil x 18 mil with 15% pinch
0603	22 mil	see Pinched Slots	see Single Dot	20mil x 40 mil with 15% pinch
0805	25 mil	see Pinched Slots	2 x 20mil 50 mil c to c	20 mil x 50 mil with 15% pinch
1206	25 mil	20mil x 60mil	2 x 25 mil .03" 55 mil c to c	20mil x 60mil with 10% pinch
SOT 232	10 mil	see Pinched Slots	2 x 10 mil 30 mil c to c	10 mil x 50 mil with 15% pinch
SOT 23	15 mil	see Pinched Slots	2 x 15 mil 200 mil c to c	15 mil x 100 mil with 15% pinch
SO 14	see dot array	40mil x 300mil	4 x 40 mil 70 mil c to c	40mil x 300mil with 15% pinch
SO 28	see dot array	50mil x 600mil	6 x 50 mil 120 mil c to c	50mil x 600mil with 15% pinch
SOD80	50 mil	40mil x 75mil	2 x 40 mil 35 mil c to c	40mil x 75mil with 15% pinch
Tantalum Cap A **Melf**	30 mil	30mil x 50mil	2 x 25 mil 50 mil c to c	30 mil x 50 mil with 15 % pinch
B	30 mil	30mil x 150mil	2 x 25 mil 50 mil c to c	30 mil x 150 mil with 15 % pinch
C	50 mil	50mil x 115mil	2 x 40 mil 50 mil c to c	50 mil x 115 mil with 15 % pinch
D	60 mil	60mil x 130mil	2 x 50 mil 50 mil c to c	60mil x 130mil with 15% pinch