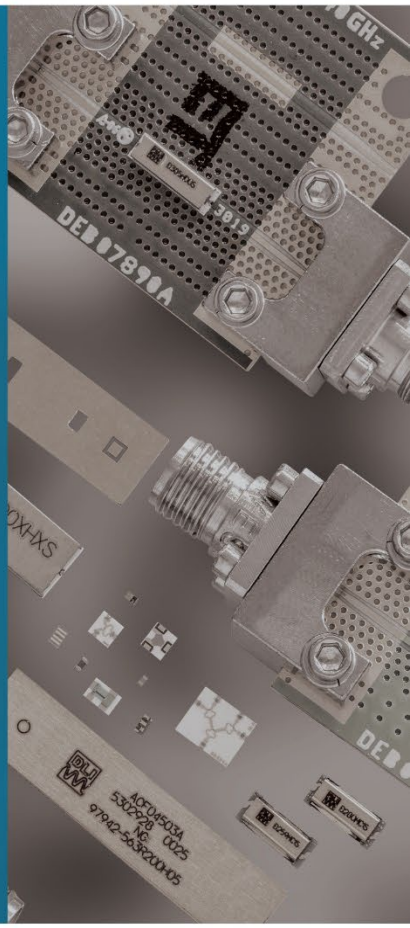
Microwave Products Guide

This guide is intended to provide additional general information and recommendations about DLI brand Microwave Products, including catalog (COTS) parts as well as custom designed part numbers.



For ordering information, please refer to the Microwave Products Catalog or the Knowles website.

For information specific to a particular part number, please reference the datasheet available on our website.

Table of Contents

General Information for All DLI Brand Devices

Recommendations for Specific Device Types

Intro to DLI Brand Devices

The Microwave Products Catalog provides a listing of DLI brand devices, available as COTS parts, to serve the RF and Microwave market including:

- RF Filters (Microstrip Technology)
  - Bandpass
  - Lowpass
  - Highpass
- Cavity Filters
- Power Dividers
- Couplers
- Gain Equalizers

These devices utilize thin film technology. DLI's specialized ceramic materials provide a substrate with enhanced properties in comparison to traditional ceramics such as alumina. DLI uses precise manufacturing techniques to apply thin metal layers to create a pattern which enables DLI brand devices to achieve the RF performance specified in the datasheet.

Knowles also offers many custom devices from the DLI brand based on similar technology to the devices listed in the Microwave Products Catalog, which also benefit from the best practices summarized in this document.

There are two options for the configuration of these devices: one for chip and wire applications requiring wirebond (WB) connections; and another, surface mount devices (SMD), requiring soldered connections in a printed circuit board assembly line.

This guide includes general information pertinent to all DLI brand devices (packaging and FAQs), as well as detailed recommendations specific to certain device types and/or configurations.

DLI Brand Devices: Storage and Handling FAQs

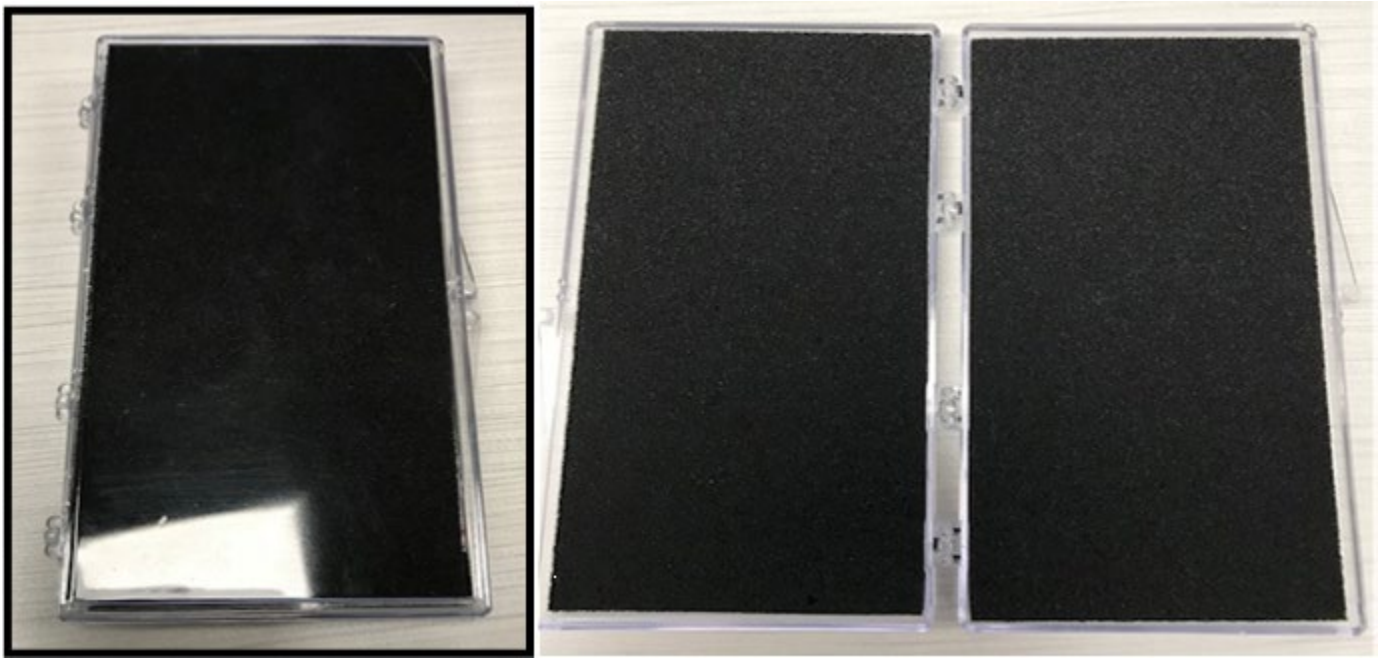
- Materials Declarations, Certifications, Conflict Minerals Statements, RoHS and REACH Statements available online here
- All DLI brand devices are Moisture Sensitivity Level (MSL): 1
  - Per IPC/JEDEC J-STD-020D
- All DLI brand devices fall under Group 1 – Fungus Inert Materials
  - Per MIL-STD-820; Paragraph 508.8
- All DLI brand devices are not ESD sensitive (passive devices)
  - Note: DLI packaging is ESD sensitive to make sure it is safe for ESD sensitive manufacturing environments, even if the parts themselves are not sensitive.
- Tape and Reel or Bulk Packaged Storage Temperature: up to 40˚C (cool, dry storage is preferred)
- Part Storage (Pre-Assembly): -55 to 125˚C
- Post-assembly Storage and Operating Temperature Range: -55 to 125˚C
- No concern for outgassing, all materials compliant as follows:
  - Ceramic
  - Metallization
  - Cover
  - Solder
  - Solder Dam

DLI Brand Devices: Bulk Packaging

Bulk Packaging Information

- Unless otherwise specified, parts are delivered in bulk packaging shown in the picture below. Parts are placed between protective ESD safe foam layers in a plastic box measuring 5.25" x 3.25“
- Over-packaging bag will also be ESD safe

Figure: Example of bulk packaging closed and open



DLI Brand Devices: Outside Packaging

Your order will arrive in a cardboard box that is chosen to fit the contents, as seen here:

When you open the box, you'll see the order paperwork and the contents wrapped in bubble packaging for protection from movement during shipping.

DLI Brand Devices: Tape and Reel Packaging

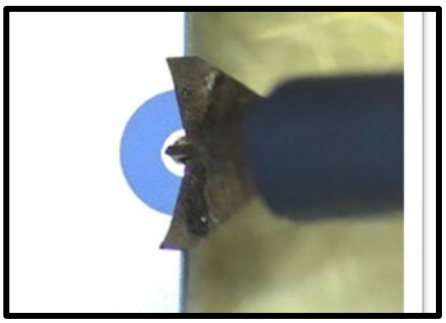
Each reel will be individually packaged in an anti-static bag and wrapped with bubble wrap. Once the bubble wrap is removed, the markings will be visible from the outside of the bag. See pics of the front and back below.

Simply remove the sticker keeping the bag closed, and the tape and reel parts are ready to use.

Contact DLIengineering@Knowles.com if you require TnR specs for a specific part number

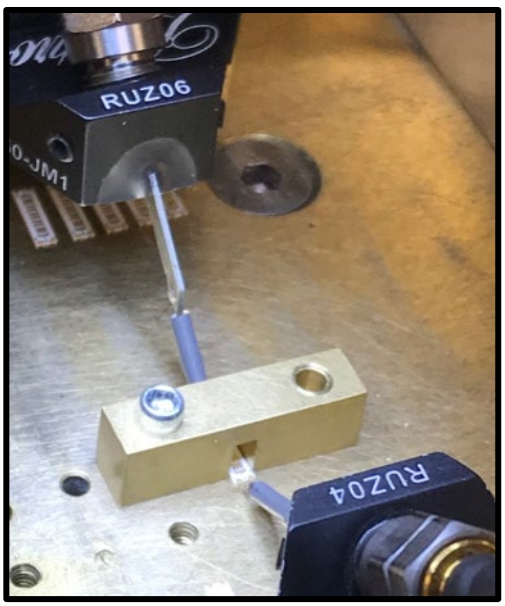
DLI Brand Devices: RF Testing and Shielding

- Knowles has precision measurement capability up to 67GHz, utilizing multiple vector network analyzers
- Filters are probed with coplanar RF probes (ground signal ground (GSG)) with 450 or 750 micron pitch, depending on operating frequency and configuration, on the bottom side of the filter
- When using coplanar RF probes, it is critical that all 3 points of the probe are equally in contact with the device for an accurate reading
- When devices do not include integrated shielding, custom fixtures are employed to assure accurate measurements without outside noise
- Shielding and customer housing dimensions are critical to account for during the design phase for continuity between filter modeling, measured data and actual use environments (contact DLIengineering@knowles.com if you are not sure your shielding solution aligns with the device design)



Top: Coplanar RF probe example

Bottom: Example of test fixture

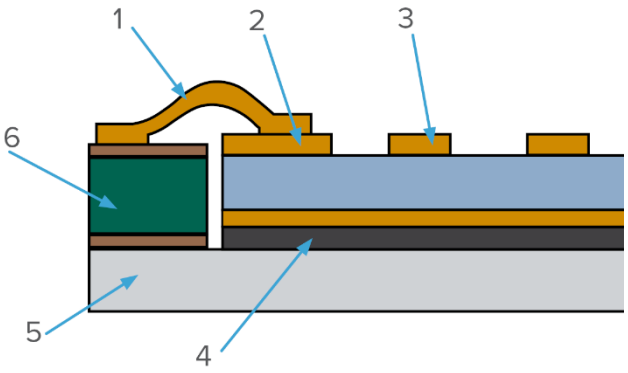


DLI Brand Devices: Available Configurations

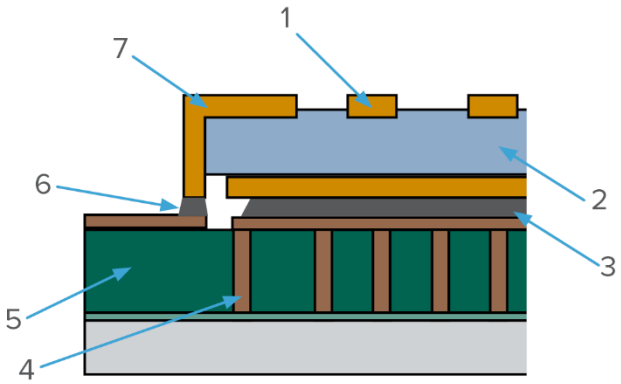
Optimal performance relies on having optimal assembly, especially with increasing frequencies. DLI offers devices with the following configurations, to fit applications focused on the highest rejection (WB), or optimized for volume manufacturing (SMD). See the general descriptions as follows:

- Chip and Wire or Wirebond (WB): For detailed recommendations reference
  - Typically mounted to board with conductive epoxy for grounding (not solderable finish unless specified)
  - IO connections are wire- or ribbon-bonded
  - Customer will design necessary RF shielding or channelization into application
  - Board and housing materials should be chosen with the aim to minimize CTE mismatch with the ceramic filter
  - Termination finish: Gold
  - Filter PN designates W for last character
- Surface mount (SMD): Detailed recommendations in this guide
  - Typically mounted during reflow soldering with solder paste
  - IO connections and ground pad are soldered using solder paste
  - Most SMD devices include integrated shielding if needed (unless specified)
  - Termination finish: ENIG
  - Filter PN designates S for last character

Wirebond Termination



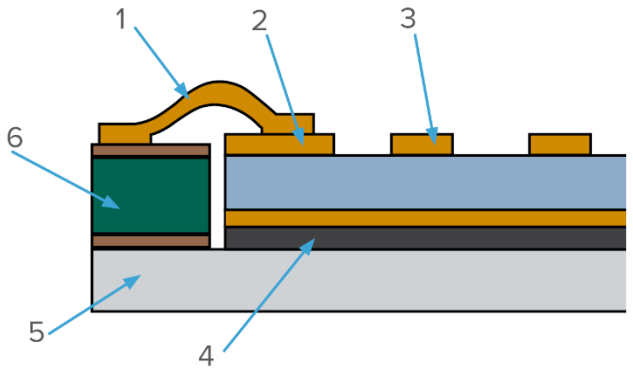
SMD Termination



DLI Brand Devices: Available Configurations

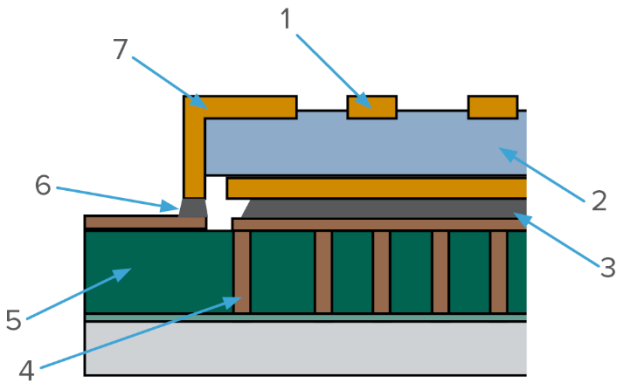
- Wirebond (WB) Callouts
  1. Wire/ribbon signal bond
  2. Signal pad on part (Au termination)
  3. RF pattern
  4. Conductive epoxy mount (ground pad)
  5. Housing floor
  6. PCB with cavity for part

Wirebond Termination

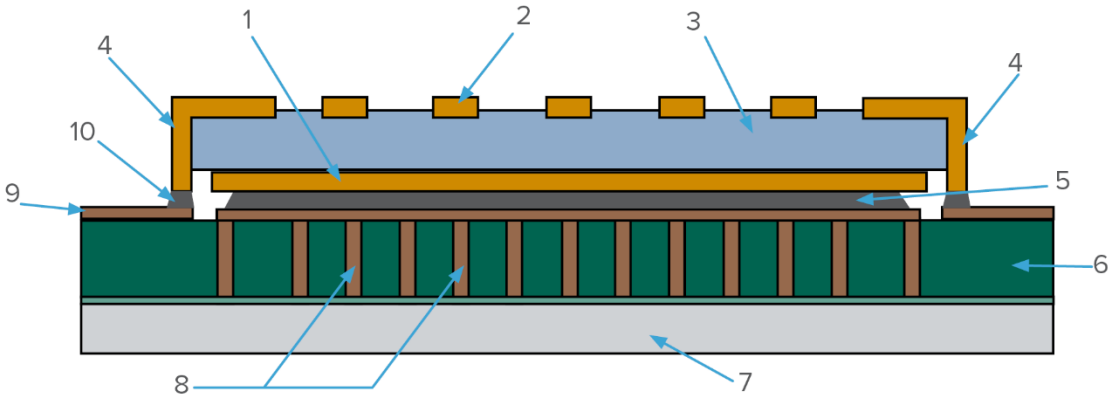


- Surface mount (SMD) Callouts *(details next slide)*
  1. RF pattern (non-solderable)
  2. Ceramic substrate
  3. Solder attachment (ground pad)
  4. PCB vias in pad
  5. Printed circuit board (PCB)
  6. Solder attachment (IO)
  7. IO castellation (ENIG)

SMD Termination



DLI Brand Surface Mount Devices: Definitions

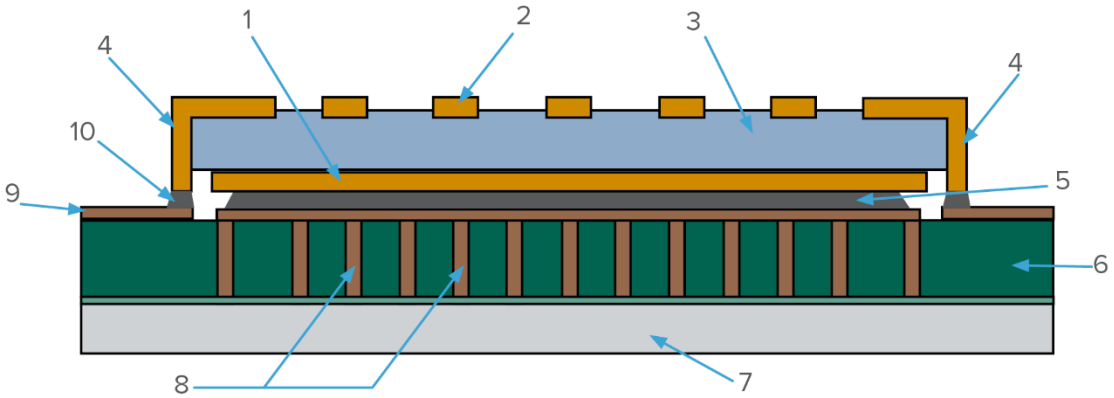


This diagram is a cross section through a surface mounted device to illustrate the necessary connections and configuration.

1. Ground plane on bottom side of device
   - Needs to be soldered to ground plane on PCB
2. RF pattern on top side of device (not a solderable surface)
3. Ceramic substrate
4. IOs extend from RF pattern traces on top side to castellations for connection to PCB signal pads
5. Soldered ground connection
6. Printed circuit board (PCB)
7. Housing for PCB
8. PCB vias in pad for grounding
9. Traces on PCB to connect with IOs
10. Soldered IO connection

Note: diagram excludes RF shielding that is present for some devices

DLI Brand Surface Mount Devices: Critical Factors for PCB Design

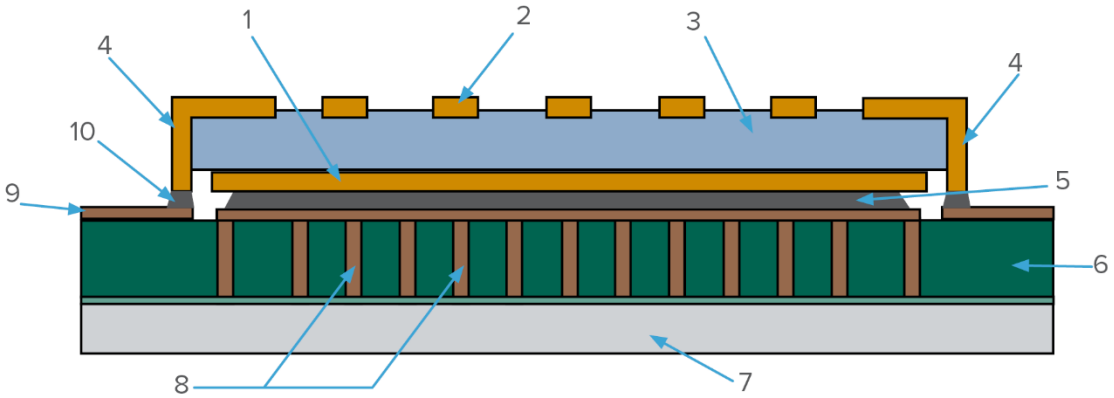


This diagram is a cross section through a surface mounted device to illustrate the necessary connections and configuration.

When designing the PCB landing pattern, note these critical factors:

- All devices in the Microwave Products Catalog have a recommended PCB layout, available in the design files on Knowles' website
- Traces will connect to the IOs (Image Ref. 3) and ground plane will connect to the bottom of the part (1)
  - Note: some devices may have more than 2 IOs
- Solder mask pattern shall not extend under the body of the device (3-5 mils clearance preferred from perimeter of device)
- PCB board materials should be targeted to minimize CTE mismatch with the ceramic substrate material
  - Typically DLI brand devices were designed based on a 10 mil thick board (top layer) with Rogers R4350B material
- 50Ω trace dimensions are application specific
- PCB stack-up should be rigid
  - Ceramic substrates are prone to cracking resulting from board flex resulting from warpage during assembly or mechanical stresses created by fixturing/screws
- To ensure adequate grounding, land pattern will need sufficient plated vias (preferably filled and plated)
- Stencil designs should be aligned with the bottom side of the part to avoid excessive solder (see examples in device specific sections)

DLI Brand Surface Mount Devices: Critical Factors for Assembly



This diagram is a cross section through a surface mounted device to illustrate the necessary connections and configuration.

When assembling these devices, keep in mind these critical factors for successful assembly:

- Robust solder connections on IOs and ground plane (callouts 5 and 10)
- Performance is designed considering 2-3 mils finished solder standoff between the board and part once assembled. Use this goal to calculate the necessary solder apertures considering your desired stencil thickness and aperture design.
- Solder is not intended to fill the I/O castellations (as you would expect for a via). Solder wetting will form a fillet at the bottom; around 70% of the height of the castellation is acceptable. Additionally, solder flow to the top of the castellation is best avoided as excess solder on top of the ceramic substrate can compromise performance.
- Best practice is to use a reflow profile as recommended by your solder material supplier. This should fall within the profile window found on pg. 14
- If using a hot plate for reflow, do not place the board directly onto the hot plate. Using a stainless steel or alumina slab under the board will slow the rate of heat transfer.
- Careful handling is critical. Plastic tipped tweezers or a vacuum pick-up tool are recommended. For parts without integrated shielding, where there is an exposed pattern on top, extra care should be taken not to scratch the metal pattern.

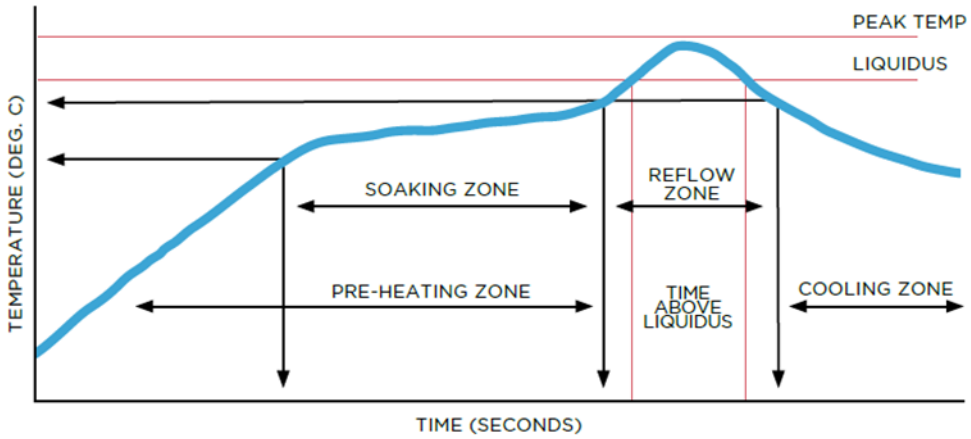
DLI brand Surface Mount Devices: Solder Paste Choice and Reflow Profile

- Typically solder paste is stencil printed for the assembly process, dispensed solder paste is also acceptable
- The table below lists reflow recommendations for the most common alloys for PCB assembly

Alloy Composition	Liquidus (˚C)	Typical Peak Temp (˚C)	Typical TAL (sec)	Typical Reflow Cycles
Eutectic Tin-Lead				
63Sn/37Pb	183	210 (max 230)	40-60 (max 90)	2
Near-Eutectic Tin-Lead				
62Sn/36Pb/2Ag	181	210 (max 230)	40-60 (max 90)	2
62.6Sn/37Pb/0.4Ag	182	210 (max 230)	40-60 (max 90)	2
60Sn/40Pb	191	220 (max 240)	40-60 (max 90)	2
Near-Eutectic Pb-free				
95.5Sn/3.8Ag/0.7Cu	220	240 (max 250)	30-60 (max 100)	2
96.5Sn/3Ag/0.5Cu	220	240 (max 250)	30-60 (max 100)	2
95.5Sn/4Ag/0.5Cu	225	245 (max 255)	30-60 (max 100)	2
98.5Sn/1Ag/0.5Cu	227	245 (max 255)	30-60 (max 100)	2

- Reflow profiling should be conducted based on the chosen solder alloy for assembly, taking into account the thermal mass of the full printed circuit board assembly, according to industry standard best practices
- Reflow profiling should be conducted with a thermocouple nearby to ensure parts aren't exposed to excessive temperatures
- Any exposure to temperatures above 260˚C (outside of any recommended profile range) can impact RF performance, part integrity and shielding
- DLI brand devices are designed to be robust enough to withstand the multiple reflow cycles that may be required for the PCB assembly process
- Rework is not recommended for DLI brand devices, whether to add solder or to remove a device from one board and assemble on another
- Touch up or rework with a soldering iron is not recommended because contact with the ceramic substrate risks cracking resulting from the thermal shock
- Hot air rework in combination with base plate heating will mitigate concerns from soldering irons. It is critical to heat the entire part/solder joint, if rework is necessary.

DLI Brand Surface Mount Devices: Reflow Profile (Continued)

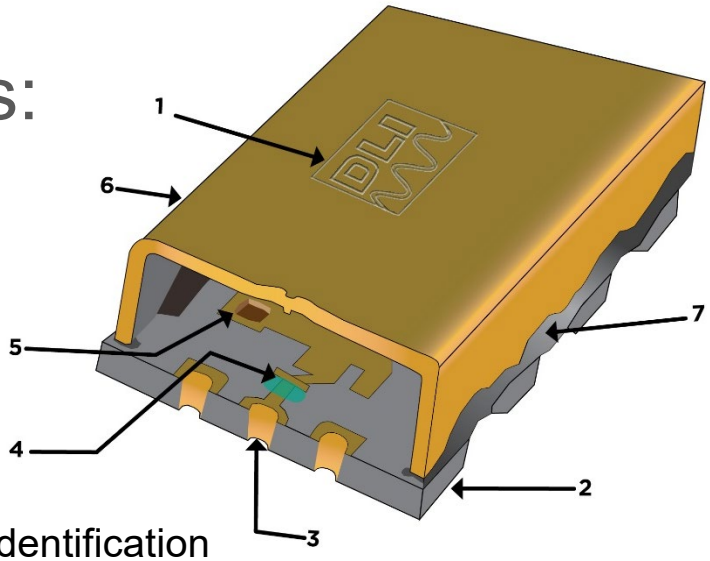


The diagram above identifies the critical parts of a reflow profile to consider for best assembly outcomes

- Reflow profiling is a powerful tool to dial in repeatable soldering and optimize to minimize voiding for the best RF performance
- The table below outlines recommendations and limits for each section
- Acceptable results can be achieved without reaching the maximum process parameters listed here. These values are chosen to demonstrate the widest possible process window without increased risk of compromising device integrity and performance

	Slope/Heating Rate (˚C/sec)	Max. Duration (minutes)
Pre-heat Zone: flux activation and outgasing in preparation for soldering	<2.5 (max. tolerable)	5
Soak Zone: only as needed to equilibrate temperatures across the board before reflow zone	0.5-1	2
Reflow Zone: critical to melt solder and achieve target time above liquidus	<2	2
Cooling Zone: controlled cooling to achieve robust solder joints, not too fast to avoid thermal shocking components	2-6	3

DLI Brand Filters: Anatomy and Definitions



1. DLI logo and laser etched identification
2. Ceramic substrate
3. IO castellation *(second IO not visible in this picture at other end of part)*
   - Solder is meant to form a fillet at the bottom of the castellation, around 70% is acceptable
   - Excessive solder flow onto the topside of the ceramic substrate risks degradation of RF performance
4. Solder dam deposit
   - May vary from this example in color or size/shape
   - Included in many filter designs to mitigate the risk of solder flow on the top of the ceramic substrate
5. Interior plated vias
6. Integrated shield
   - Gold-plated 430SS Stainless Steel 3-sided Cover
7. High melting point solder for cover attachment
   - RoHS Compliant solder per exemption 7a.
   - Composition: 92.5% Pb, 5.0% Sn, 2.5% Ag (Indalloy 151)
   - Melting Point: 287-296˚C
   - During the cover attachment process, the solder forms an irregular pattern on the side of the cover and bottom side of the part, this is normal and does not interfere with the subsequent surface mount soldering process

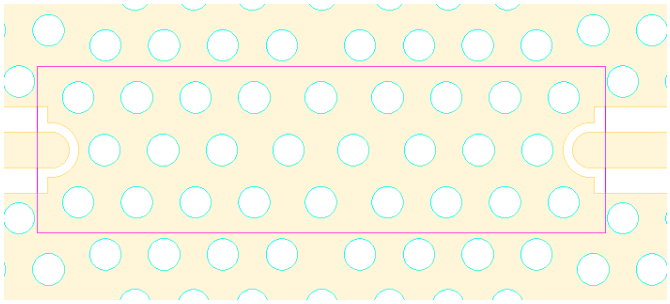
DLI Brand Filters: Stencil Design Recommendations

All devices in the Microwave Products Catalog have a recommended PCB layout, available in the design files on Knowles' website

Stencil designs should be based on the PCB layout following these steps:

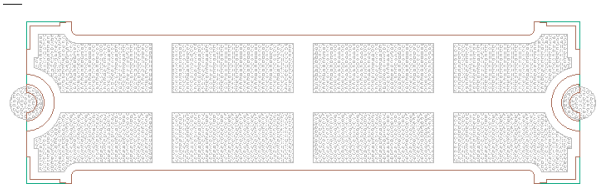
Outline of bottom of part: Orange outlines the ground pad and IOs

PCB layout per page 11 with vias in green and limits of solder mask in pink

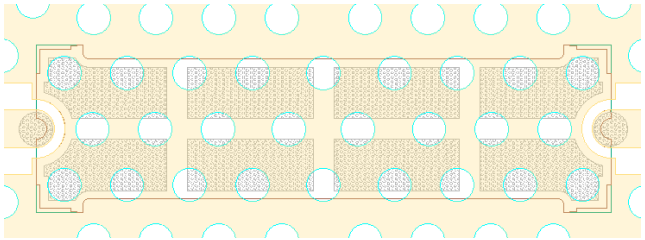


1. Make a circular deposit at each IO with a diameter matching the annular ring diameter on the bottom of the part (Figure 1)
2. Window pane the ground plane targeting 70-80% area coverage with deposits of a similar size. Maintain at least 10 mils distance between the IO deposit and all ground plane deposits

Bottom of part with grey solder areas



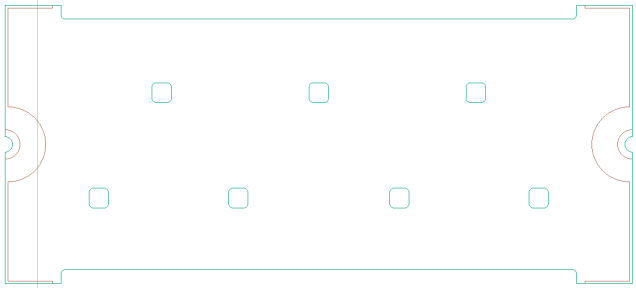
PCB layout with overlaid part including grey solder areas



DLI Brand Filters: Stencil Design Recommendations – Internal Vias

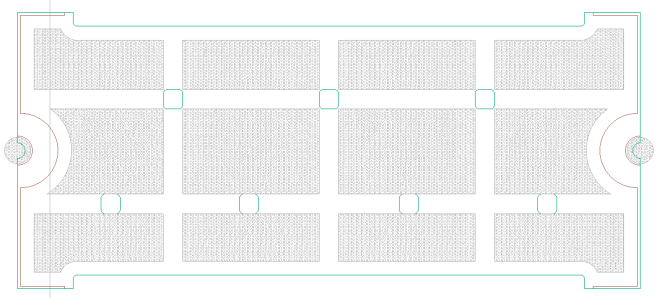
For internal vias, it may benefit FPY to avoid printing solder directly beneath internal vias. The following example proposes design approaches to achieve this.

Outline of bottom of part: Orange outlines the ground pad and IOs

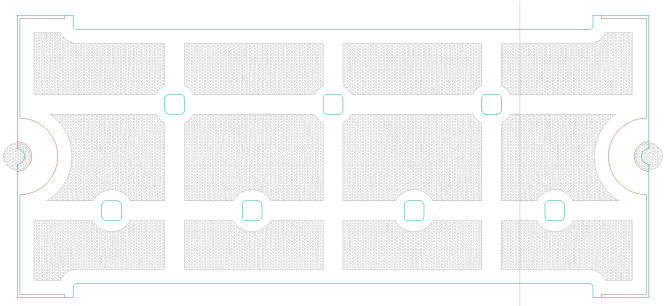


1. Make a circular deposit at each IO with a diameter matching the annular ring diameter on the bottom of the part (Figure 1)
2. Window pane the ground plane targeting 70-80% area coverage with deposits of a similar size. Maintain at least 10 mils distance between the IO deposit and all ground plane deposits
3. Align the window pane areas to via location to minimize solder volume available to flow up the vias

Design option for Thin Stencils (4 mil or less)
Bottom of part with grey solder areas



Design option for Thicker Stencils
Bottom of part with grey solder areas



DLI Brand Filters: Stencil Design Recommendations

Notes for Best Practices:

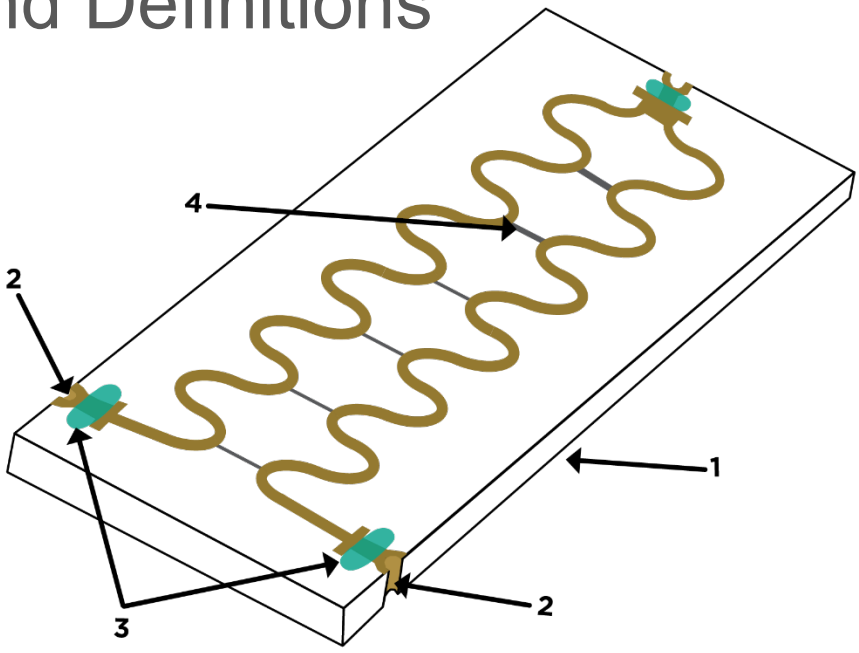
- If a design has internal vias (can be noted in the bottom view drawing), align the panes of the ground plane stencil design to avoid deposits in the same position as the vias
- Solder mask pattern on the PCB shall not extend under the body of the device (2-3 mils clearance preferred from perimeter of device)
- IO deposits should follow the area ratio rule of >0.66 to avoid variation in solder deposition (see chart below where acceptable sizes are in green and **typical sizes** are bold)

$$\text{Area Ratio} = \frac{\text{Area of the Aperture}}{\text{Area of the Walls}} > 0.66$$

Area Ratio Decision Chart

	Circle Aperture Diameter (mil)									
Stencil Thickness (mil)	5	6	7	8	9	10	**11**	**12**	**13**	**14**
5	0.25	0.30	0.35	0.40	0.45	0.50	0.55	0.60	0.65	**0.70**
4	0.31	0.38	0.44	0.50	0.56	0.63	**0.69**	**0.75**	**0.81**	**0.88**
3	0.42	0.50	0.58	**0.67**	**0.75**	**0.83**	**0.92**	**1.00**	**1.08**	**1.17**

DLI Brand Power Dividers: Anatomy and Definitions



DLI brand Power Dividers can be used to divide or combine a signal. Critical areas are as follows

1. Ceramic substrate
2. IO castellation
   - Solder is meant to form a fillet at the bottom of the castellation, around 70% is acceptable
   - Excessive solder flow onto the topside of the ceramic substrate risks degradation of RF performance
3. Solder dam deposit
   - May vary from this example in color or size/shape
   - Included in many device designs to mitigate the risk of solder flow on the top of the ceramic substrate
4. Resistors (thin film TaN)
   - Avoid scratching these areas during pick and place

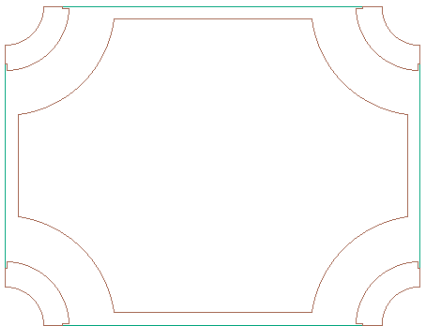
DLI Brand Power Dividers and Couplers: Stencil Design Example

All devices in the Microwave Products Catalog have a recommended PCB layout, available in the design files on Knowles' website

Stencil designs should be based on the PCB layout following these steps:

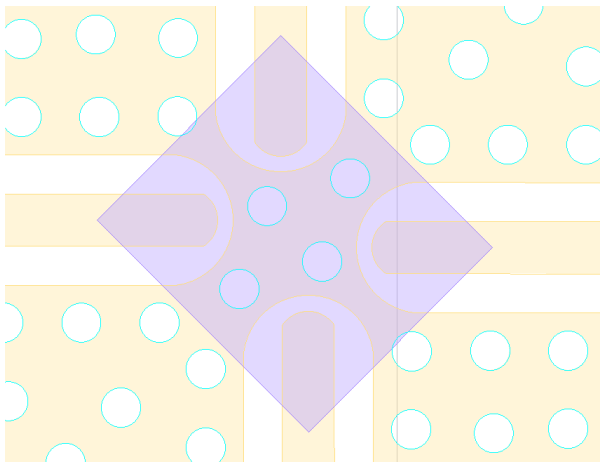
1. Make a circular deposit at each IO with a diameter matching the annular ring diameter on the bottom of the part (Figure 1)

Outline of bottom of part: Orange outlines the ground pad and IOs

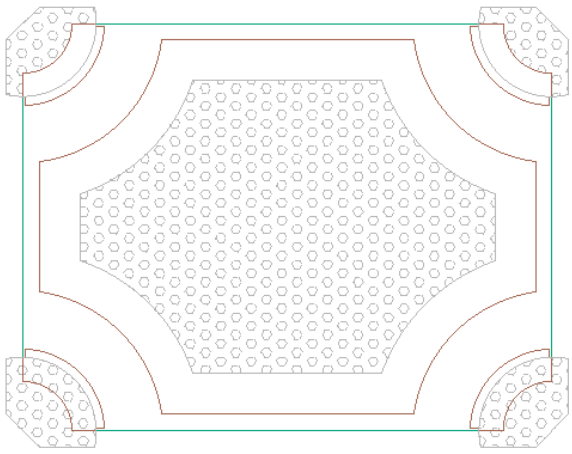


2. Window pane the ground plane targeting 70-80% area coverage with deposits of a similar size. Maintain at least 10 mils distance between the IO deposit and all ground plane deposits

PCB layout per page 11 with vias in green and limits of solder mask keep-out area in purple (note 45˚ part rotation for alignment)



Bottom of part with grey solder areas



DLI Brand Power Dividers and Couplers: Stencil Design

Notes for Best Practices:

- Solder mask pattern on the PCB shall not extend under the body of the device (2-3 mils clearance preferred from perimeter of device)
- IO deposits should follow the area ratio rule of >0.66 to avoid variation in solder deposition (see chart below where acceptable sizes are in green and **typical sizes** are bold)

$$\text{Area Ratio} = \frac{\text{Area of the Aperture}}{\text{Area of the Walls}} > 0.66$$

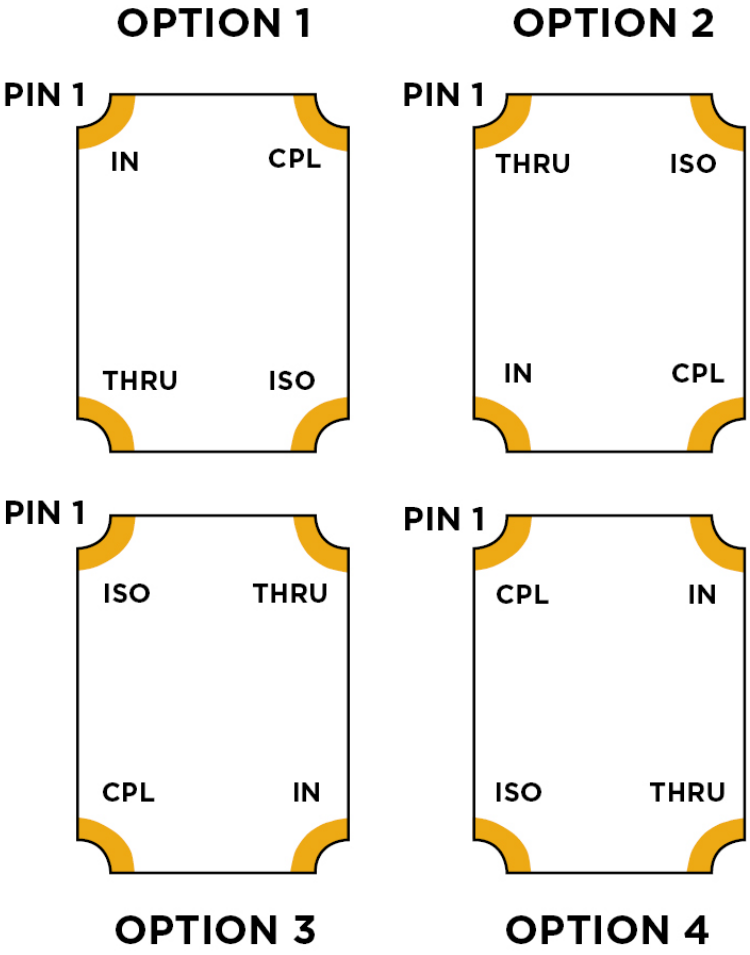
Area Ratio Decision Chart

Circle Aperture Diameter (mil)

Stencil Thickness (mil)	5	6	7	8	9	10	**11**	**12**	**13**	**14**
5	0.25	0.30	0.35	0.40	0.45	0.50	0.55	0.60	0.65	**0.70**
4	0.31	0.38	0.44	0.50	0.56	0.63	**0.69**	**0.75**	**0.81**	**0.88**
3	0.42	0.50	0.58	**0.67**	**0.75**	**0.83**	**0.92**	**1.00**	**1.08**	**1.17**

DLI Brand Directional Couplers: Symmetry and Port Configurations

- Symmetrical and Asymmetrical Directional Couplers that do not incorporate an integrated termination can be used in one of the four configurations below
- Symmetrical couplers have a 90˚ phase relationship between the through and coupled paths
- Asymmetrical or Levy style couplers do not have a known phase relationship between the ports
- For wideband couplers if phase relationship is not needed than Asymmetrical couplers are usually preferred due to their smaller size



For technical inquiries contact
DLIengineering@knowles.com

RoHS Compliance Statement

Knowles is a leading supplier to the electronic components market and is fully committed to offering products supporting Restriction of Hazardous Substances (RoHS) directives. All our dielectric formulations are RoHS compliant along with a broad range of capacitors with RoHS compliant terminations. Knowles complies with the requirements of the individual customer and will maintain product offerings that meet industry demands.

Quality and Environmental Policy

Knowles' reputation for quality and environmental responsibility is based on a commitment not only to meet customer requirements, but to exceed their expectations. The entire organization, beginning with top management, strives to achieve excellence in designing, manufacturing and delivering High Q capacitors and proprietary thin film components for niche high-frequency applications, while maintaining safe and healthy working conditions.

Furthermore, Knowles is committed to achieving these goals in an environmentally responsible manner through a commitment to comply with environmental regulations and pollution prevention initiatives. Knowles strives to continually improve the effectiveness of its quality and environmental management system through the establishment and monitoring of objectives and targets.

Microwave products proudly made in the USA.

knowles

For more information, visit us online at
www.knowles.com