Invisipin® Assembly Guide

Introduction

The R&D Interconnect Solutions Invisipin® is a solderable conductive elastomer interconnect. Invisipin® consists of a solderable metal flange and a conductive elastomer. This structure allows infinitely configurable interconnects with excellent electrical and mechanical properties. The Invisipin® must be installed using appropriate consideration of flange and elastomer mechanical characteristics.

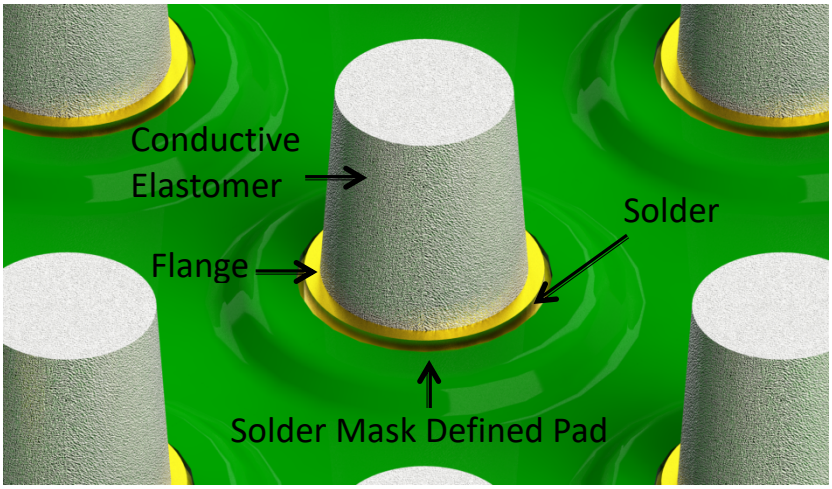
This document provides general recommendations and guidelines for PCB design, solder assembly, and the Invisipin® mechanical over-compression stop application. Guidelines are provided in order to maximize Invisipin® performance and to ensure a highly reliable solder connection of the Invisipin® to the PCB.

1.0 PCB Design Recommendations

The Invisipin® requires a high quality solder connection for optimal electrical, mechanical, and thermal performance. Recommendations in this document assume that design and layout guidelines are followed.

1.1 PCB Pad Design

1.1.1 Solder Mask Defined Pads



Solder Mask Defined Pads

R&D Interconnect Solutions®
1660 East Race Street
Allentown, PA 18109
www.rdis.com

1.1.1A PCB Solder Mask Defined Pad Diameter - The PCB solder mask defined pad diameter should be 38µm (1.5mils) larger than the Invisipin® flange.

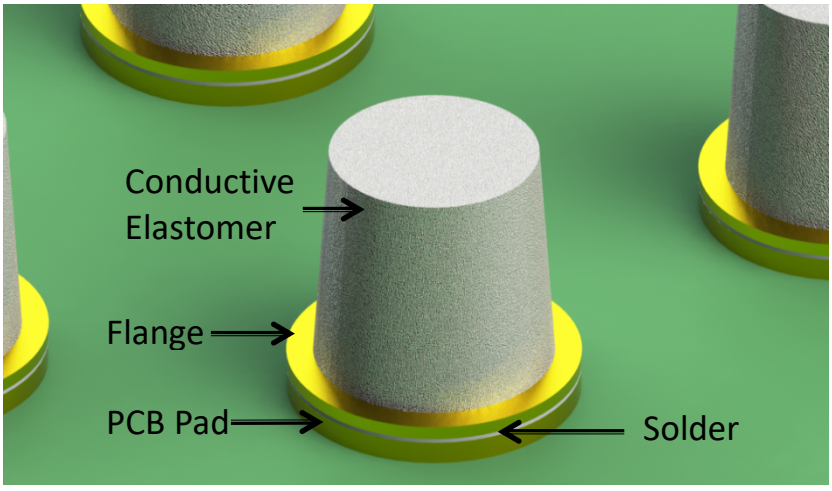
Note: The solder mask opening needs to be slightly larger than the Invisipin® flange so the flange forms a good solder joint and doesn't sit on the edge of the solder mask.

Invisipin® Flange Diameter (mm[mils])	Solder Mask Opening Diameter (mm[mils])
0.279[11.0]	0.318[12.5]
0.381[15.0]	0.419[16.5]
0.457[18.0]	0.495[19.5]
0.559[22.0]	0.597[23.5]
0.686[27.0]	0.724[28.5]

1.1.1B PCB Pad Vias – Filled vias are recommended if via locations are within the solder mask defined pad opening.

Note: Un-filled vias in solderable areas affect finished solder volume. Invisipin® solder joints need to have a controlled solder volume.

1.1.2 Non-Solder Mask Defined Pads



Non-Solder Mask Defined Pads

1.1.2A PCB Pad Diameter - The PCB pad should have the same diameter as the Invisipin® flange.

Note: If the flange is larger than the pad or the pad is larger than the flange, the pin will not meet alignment specifications. Lateral motion can occur during re-flow causing the pin to be off center with respect to the pad.

1.1.2B PCB Pad Vias – Filled vias are recommended and offset vias are *not* recommended for non-solder mask defined pads (NSMD.)

Note: Un-filled vias and offset vias in solderable areas affect finished solder volume. Invisipin® solder joints need to have a controlled solder volume.

Note: Off-set vias in solderable areas are not recommended because they allow lateral motion of the Invisipin® during reflow. This causes positional error.

1.1.3 PCB Pad Plating - The following PCB metal plating processes are recommended.

- ENIG (Electroless Nickel Immersion Gold)
- ENEPIG (Electroless Nickel Electroless Palladium Immersion Gold)
- OSP (Organic Solderability Preservative)
- Hot Air Solder Leveling (HASL) finish, the surface finish should be maintained within a 25µm (1mil) range.
- Tin plated Cu

Note: Invisipin® gold thickness ranges from 0.102µm (4µin) to 0.254µm (10µin). Please take this into consideration when specifying PCB pad plating to avoid gold embrittlement.

1.2 PCB Board Surface Metal Features – If using NSMD pads, the PCB layout should not have any traces or other features within the over-compression stop extents.

Note: Having a planar surface when mounting the over-compression stop creates a uniform Invisipin® compression stroke which maximizes performance. If extra traces/metal features are required, they must be accounted for in the mechanical design.

1.3 PCB Board Surface Solder Mask/ Silk-Screen – If using NSMD pads, the PCB layout should not have any solder mask or silk-screen within the over-compression stop extents.

2.0 Solder Paste Deposition, Pin Placement, and Reflow Recommendations

When installed in accordance with this assembly guide the Invisipin® will be perpendicular to the PCB, correctly positioned, and have a reliable solder joint. Having the Invisipin® perpendicular to the PCB maximizes electrical performance and this is achieved by having consistent and correct solder volume

2.1.1 Solder Paste Stencil

The solder paste stencil should be designed in accordance with IPC-7525 Stencil Design Guidelines. When calculating the final solder thickness, solder paste is about 50% solder by volume.

The recommended finished solder joint thickness for Invisipin® is 19-25µm (0.75-1mils.)
This finished solder thickness generally cannot be achieved by direct 1:1 print with the stencil aperture equal to the pad diameter. Printing Type 3 or Type 4 solder using stencils thinner than 76µm (3mils) leads to inconsistent solder volume.

> *Recommendation:* The stencil should be made from stainless steel and have a thickness ≥ 0.003". Adjust stencil hole diameters to achieve target finished solder thickness. Refer to IPC 7527 for guidelines.

Example Solder Paste Stencil Hole Diameters:

- Stencil Thickness = 76µm (3mils)
- Target Finished Solder Thickness = 20.3µm (0.8mils)

Invisipin® Flange Diameter (mm[mils])	Solder Stencil Hole Diameter (mm[mils])
0.279[11.0]	0.203[8.0]
0.381[15.0]	0.279[11.0]
0.457[18.0]	0.333[13.1]
0.559[22.0]	0.409[16.1]
0.686[27.0]	0.500[19.7]

2.1.2 Solder Paste

Various types and grades of solder paste can be used for surface mounting. The Invisipin® has not been tested for solvent or saponified cleaning.

> *Recommendation:* Any Type 3-4 solder paste that is either water-soluble or no clean is acceptable.

2.2 Placement

The Invisipin® is able to be placed onto the PCB using many industry standard component pick-and-place systems. Pick and place equipment must have adequate positional placement accuracy to ensure that the pin is centered on the PCB pad.

Recommendation: A component pick-and-place system with both a top-and-bottom vision system should be used.

Recommendation: The Invisipin® should be placed and released into the solder paste. Forces on the top of the pin should not exceed the maximum recommendation for any given pin type.

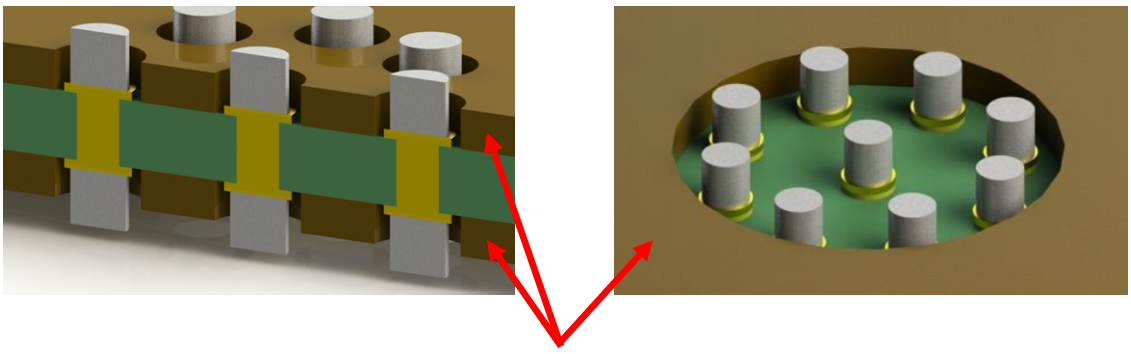
2.3 Solder Reflow

The Invisipin® may be surface mount soldered using standard IR or convection SMT reflow process. Invisipin® is qualified for reflow cycles at 245°C peak reflow temperature.

3.0 Over-Compression Stop Design Recommendations

An important part of an Invisipin® application is the use of an over-compression stop. This layer usually consists of a dielectric spacer. It is used to keep the compression stroke within acceptable limits. Several common over-compression stop configurations are shown below.

Isolated Pin Configuration **Air Dielectric Configuration**



3.1 Over-Compression Stop Material

The over-compression stop should be made of a non-compressible material that has suitable physical properties for the application. Coefficient of thermal expansion, thermal stability, electrical characteristics, and thickness tolerance are several examples of properties to consider

Note: Kapton and Cirlex are two recommended materials.

3.2 Over-Compression Stop Thickness

Over-compression stop thickness should be based on the overall interface design. Pad height, solder mask, routing, annular rings, etc. should be factored in when determining thickness. Both the Invisipin® and mating PCB interfaces need to be considered.

> *Note:* A properly designed over-compression stop keeps the Invisipin® compression stroke within acceptable limits. Limits are specified in pin specific datasheets and the Invisipin® catalog sheet.

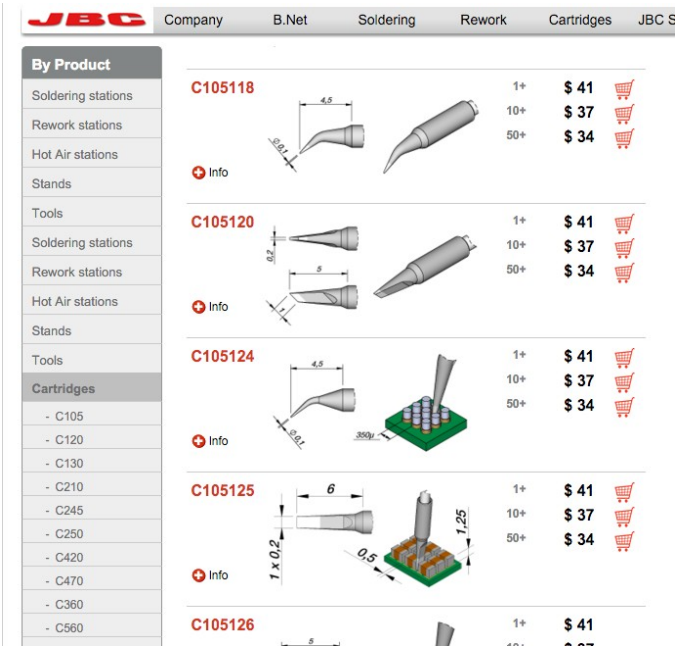
3. 3 Over-Compression Stop Pin Clearance Distance

Clearance holes and cut-outs should be sized so the Invisipin® can expand properly when compressed. The conductive elastomer column expands outward during the compression stroke. There needs to be clearance around the pin to allow for this expansion.

> *Note:* The over-compression stop should be designed such that the volume of the stop cylinder is ≥ 120% of the volume of the board pad, solder, pin structure, and any component of the mating interface that may enter the cylinder during periods of full compression.

4.0 Where to Purchase Specialized Solder Tips

JBC Tools offers specialized solder tips for Invisipin®:
Website Link: http://www.jbctools.com/catalegfitxa.php?idpro=15#C105



C105126 Cartridge Conical Ø 0,1

C105 cartridges are the **smallest** of the JBC range.

They are appropriate for soldering and repairing **very small-sized** components at **high precision**, even under a microscope.

C105 works with **NT105** Nano Soldering Iron and **NP105** Nano Tweezers.

Supplied with 5 units in the **DC-A** dispenser case.

JBC Long Life Tips offer instant heat up, excellent heat transfer and great durability.

They last **5 times longer** than other brands thanks to the Exclusive Heating System and the Sleep & Hibernation features.