How to Define and Add Interposer to Your Die Stack

Product Version SPB16.5
May 24, 2012

Contents

Purpose

Interposers play an important role in your package design. In this document, you will learn how to create an interposer library object and place an instance of that definition into your SiP substrate design.

Audience

This document is intended for Package Designers. If you plan to vertically stack the dies, this document will be useful to understand the various aspects of using Interposers in your designs.

How to Define and Add Interposer to Your Die Stack

Vertical stacking of dies on top of a package substrate is becoming increasingly necessary to reduce the size and cost of the complete package part. Size differences between dies, along with a need to connect two dies together in the same stack without bonding all the way to the substrate, may make it desirable to use an interposer object. Interposers are pieces of a dielectric material with a layer of redistribution metal on top. This metal can be bonded to from dies above it in the stack, and bond wires can connect from the interposer down to a lower die in the stack or directly to the package substrate.

As of the 16.5 version of Cadence SiP tools, interposers have a combination of bond fingers, routing clines, and filled shapes on the routing layer. These conductor elements are all within the symbol's place bound shape, which defines the edges of the dielectric on which the routing exists.

Multi-layer interposers are not supported as of 16.5. Therefore, your interposer symbol cannot contain vias. Since interposers are mechanical symbols and are not associated with a logical component, they cannot include pins.

In this document, you will understand how to create an interposer library object and place an instance of that definition into your SiP substrate design.

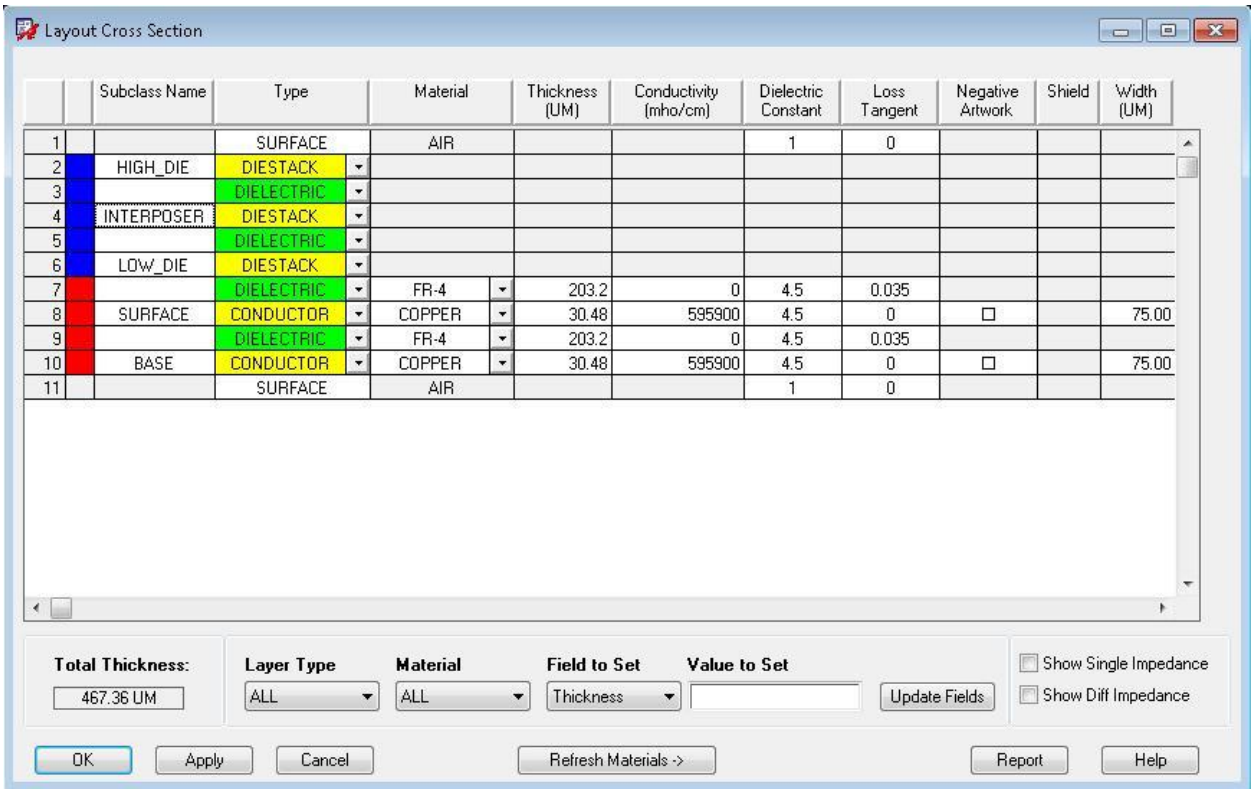
Procedure 1: Configuring the Design Cross-Section for Interposers

Before you can add interposers to the die stacks in your SiP package substrate, you need layers on which they will be placed. Interposers, since they contain routing, are always placed on *DIESTACK* type layers that exist in the die stack region of your design's layer cross-section.

The basic setup of your cross-section is described in further detail here:

Creating the Cross-Section (Layer Stack up) for a New IC Package Substrate

Once the basics are complete, use *Setup* -- *Cross-Section…* (`define lyrstack`) command to add the additional diestack layers for your interposers. These are normally positioned between diestack type layers where dies are placed, as shown in the image below:



Here, you can see the *INTERPOSER* layer, a diestack type layer, has been placed between the two die layers *LOW_DIE* and *HIGH_DIE*. The order of the layers in the cross-section defines the order of the objects which exist in the same die stack. Empty layers are ignored, so if you have two die stacks side-by-side which have different orders of object types, it is okay to leave layers unused in a die stack. Unused layers are ignored by SI extraction, 3D Viewer, Assembly DRC checks, and all other tools which access information about the die stack.

Once you have created the layers for your interposer (s), continue to Procedure 2 below.

Procedure 2: Creating a Library Interposer Symbol Definition

Interposers are package drawing symbols which contain a dielectric outline, thickness, and material. They also have place boundary and assembly documentation shapes like a die component and a part number and reference identifier.
You can follow along with this procedure by watching the video demonstration here:

To create a new library interposer object, start a new package symbol drawing using the *File* -- *New* command. This creates a blank DRA database file for you to begin defining the interposer symbol and routing.

All interposer symbols require a place bound and assembly outline. These outlines provide the extents of the manufactured component as well as the boundary of the dielectric material itself. It is important for the size and outline shape of the interposer to be accurate in order to ensure accurate assembly DRC results and signal integrity extraction.

Add these shapes using the *Add* -- *Frectangle* (`add frect`) command. To ensure that your interposer dimensions are exactly right, it is recommended that you use the `pick` command to select the two corners of the outline. Since `0,0` in the DRA file is always used as the symbol's origin and snap location when the interposer is on your cursor for manipulation in the SiP database, make sure that your outline is offset from this point the way that you want it for future placement.

NOTE: Interposer outlines do not have to be a simple rectangle. To make an interposer with rounded corners, notches, or other distinguishing features for any reason, use the *Shape* menu commands to define the detailed outline of the interposer.
Once you have added the outline on the first layer, copy it to the second layer using the *Edit* -- *Z-Copy Shape* (`zcopy shape`) tool. When you are finished, you should have two copies of the shape on the follow layers:

```
Class: COMPONENT GEOMETRY, Subclass: PLACE_BOUND_TOP
Class: COMPONENT GEOMETRY, Subclass: ASSEMBLY_TOP
```

Always use the *TOP* layers, even if this interposer will be used for bottom-mounted die stacks in your package substrates. When an instance of the interposer is placed, if it is on the bottom side of the substrate, the tool automatically moves these shapes to the corresponding *BOTTOM* layers.

Next, add reference designator placeholder text. This is a text object, usually `U*`, which indicates where the instance's reference designator text should be located. Add this text using the *Add* -- *Text* (`add text`) command When placing instances of this interposer in your design ensure that you use text block and justification that meet your needs.

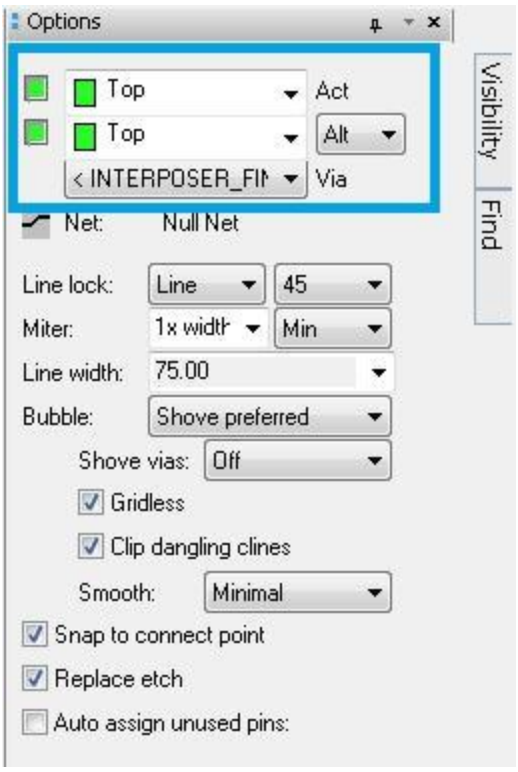
Finally, add the routing and other conductor objects to your design. An interposer can contain any combination of clines, bond fingers, and filled conductor shapes. Create these on the top substrate layer in your interposer symbol definition file, as interposers in 16.5 only support a single layer of routing.

In order to add bond fingers to your substrate, you must first add the necessary padstacks to your constraint via list for the design. If you are unfamiliar with how to do this, follow the link below to a step-by-step description:

Defining a Via List for Interactive Routing in Constraint Manager.

Bond finger padstacks must be single-layer pads and should be defined on the top routing layer in your stack-up.

Use the *Route* -- *Connect* (`add connect`) tool to add clines and vias to your design, creating as many redistribution connections as necessary. To ensure the bond finger padstacks are in the list of via pads; make sure that both your active and alternate layers are set to the top routing layer, as in the image below:



When you have finished adding all of your routing, flag the vias as bond finger objects using the *Edit* -- *Properties* (`property edit`) command again. This time, set the find filter to vias, window select all the vias in your main canvas, and add the *BOND_PAD* property with a value of *TRUE*.

NOTE: Because an interposer does not contain its own logic, all routing can remain on dummy nets. When the interposer is added to a SiP substrate, it automatically uses the nets of the pins which are bonded to or from the interposer fingers.

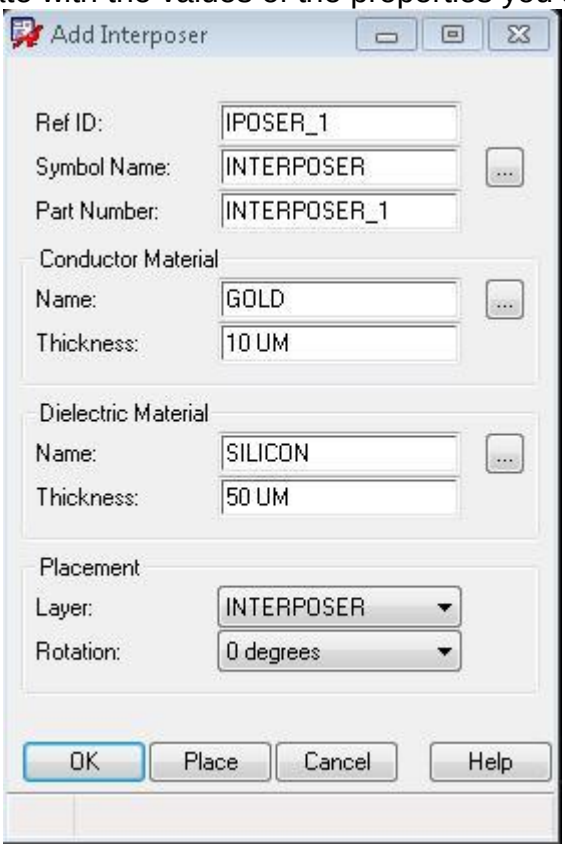
At this point, your interposer definition contains all the mandatory information for it to be used in a SiP substrate. However, it is recommended that you also pre-define the dielectric and conductor layers' material and thickness characteristics, as well as the part number. These are used when adding the interposer to a design to initialize the corresponding fields. This way, your designer need not enter them manually – eliminating the potential for incorrect material specifications that do not correspond with

the manufactured part. Errors like these can cause problems with SI extraction, 3D Viewing, and assembly DRC verification in the future.

To set these properties, use the *Edit* -- *Properties* (`property edit`) tool. Because these properties exist on the symbol definition, they should be created on the design itself, not on the conductor shape you added earlier. To add properties to the design, go to the Find filter tab and, in the *Find by Name* pull-down list, select *Drawing*. Add the *CONDUCTOR_MATERIAL*, *CONDUCTOR_THICKNESS*, *DIELECTRIC_MATERIAL*, *DIELECTRIC_THICKNESS*, and *PART_NUMBER* properties. For the conductor and dielectric material fields, make sure that the name you give has an entry in the `mcmmat.dat` materials file so that SI extraction can obtain electrical characteristic data for the materials.

You have now completed the design of your library interposer object. Use the *File* -- *Save* command to save it in the appropriate directory within your symbol library hierarchy.

Place an instance of this new library object using the *Add* -- *Interposer* (`add interposer`) tool in the SiP Layout editor for your package substrate design. When you specify the symbol name of the part you just created, the fields on the form automatically pre-populate with the values of the properties you set earlier:



Once you select the placement layer and rotation, click the *Place* button. The interposer is attached to your cursor. Click in the main canvas to place an instance of the

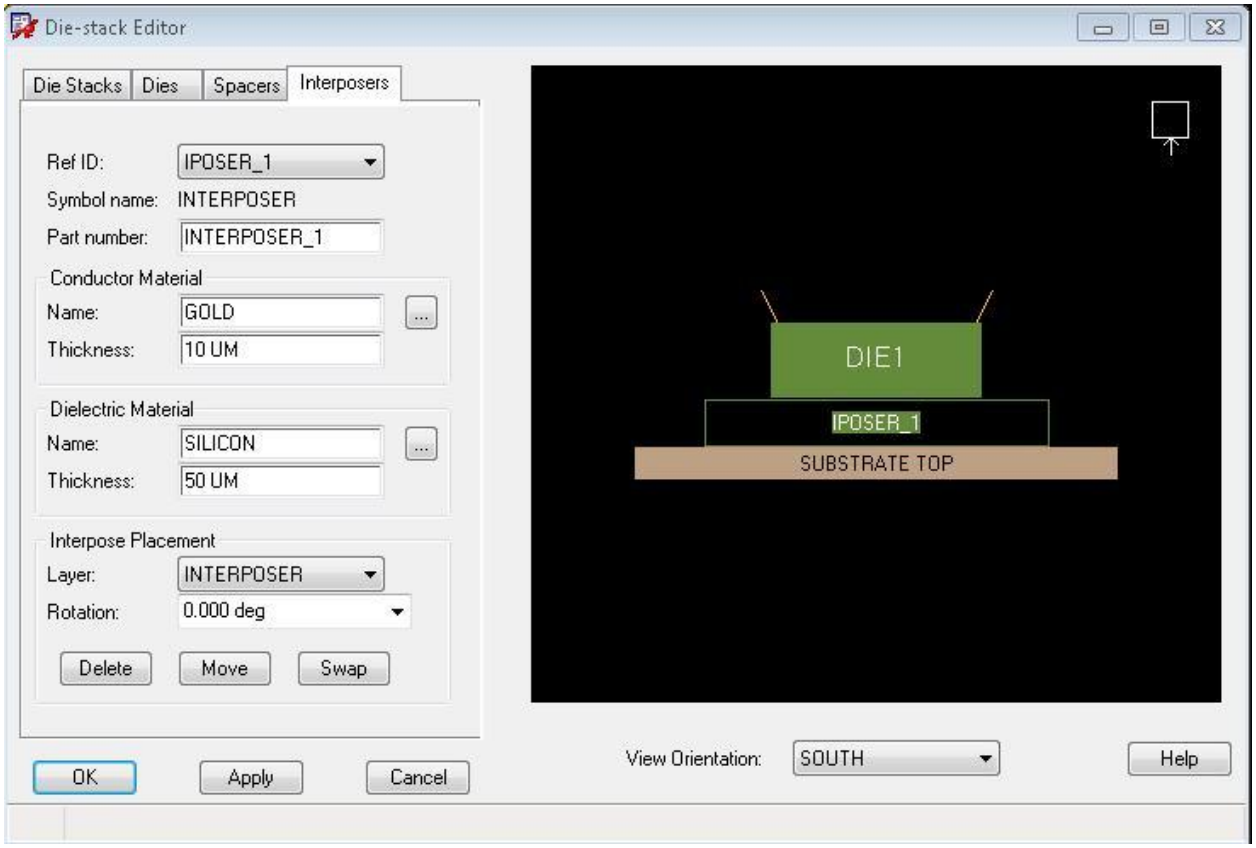
interposer or, for more exact placement, use the `pick` command and form to specify the location. Choose *Snap to…* from the pop-up menu to snap the interposer origin relative to a feature on the die symbol it is being placed above or beneath.

Note: You can place multiple instances of this interposer at the same time by making multiple picks after clicking the *Place* button. Click the *OK* button on the form, or choose Done from the pop-up menu when you are finished adding interposers to your substrate.

Procedure 3: Modifying an Instance of an Interposer in the Die Stack Editor

Once you have added interposers to your design, use the *Edit* -- *Die Stack…* (`diestack editor`) tool to modify the placement or material characteristics of the interposer.

After opening the die stack editor, change to the desired stack on the *Die Stacks* page, then move to the *Interposers* page. Notice that the part number, conductor, and material attributes can be modified uniquely on this instance, as shown below. Changes you make to thicknesses are dynamically updated in the preview image on the right hand side of the form.



To change the vertical order of an interposer in the die stack, change the layer it is placed on. Or, use the *Swap* button to swap the interposer with another die or interposer in the same stack.

When you have finished making changes, click the *OK* button to save the die stack changes to the drawing and return to the main canvas.

Summary

Following the above **procedures**, you can define and add the interposers in your designs.