cadence®

Wire Bond Tutorial

Product Version 23.1
September 2023

Cadence Design Systems, Inc. (Cadence), 2655 Seely Ave., San Jose, CA 95134, USA.

Contents

Preface

This tutorial provides a combination of designs and procedures that will help you understand the wire bond design software. It illustrates a variety of techniques available for incorporating a wire bond methodology into an existing packaging design flow.

You can:

- View a basic flow for a wire bond design highlighting the wire bonding features.

  You can access the designs and files for each module from `<install_directory>/docs/wb_tut/examples`.

- Build a simple design to show the features and flow.

Tools Used and Related Software Requirements

Allegro X Advanced Package Designer (APD) Release 23.1, Windows or Linux

Note: 3D Canvas is available with the *Allegro X Advanced Package Designer* or *Allegro Package Designer L* licenses.

Topics Included in This Tutorial

- Module 1: Creating BGA and Die
  - BGA Creation/Import BGA Text In
  - Die Flag Import from DXF/compose shape Command
  - Die Creation/Import Die Text-in
  - Power and Ground Ring Creation
  - Best Fit Path for Existing Bond Fingers
- Module 2: Creating and Editing Guide Paths and Wire Bonds
  - Wire Bond Guide Creation
  - Advanced Filtering and Wire Bond Settings

- Wire Bond Process And Flow
- Wire Bond Push, Shove, and Move
- Wire Bond Merge
- Wire Bond Tack Point Move
- Wire Bond Redistribute
- Wire Bond Space Evenly
- Wire Bond Center

- Module 3: Die Shrink and ECO
  - Die Shrink and the ECO Process
- Module 4: Viewing 3D Models Using 3D Canvas
  - Cadence 3D Design Viewer Extraction
  - Multiple Profiles in Cadence 3D Design Viewer with DRC

Module 1: Creating BGA and Die

This module covers:

- BGA Creation/Import BGA Text In
- Die Flag Import from DXF/compose shape Command
- Die Creation/Import Die Text-in
- Power and Ground Ring Creation
- Best Fit Path for Existing Bond Fingers

Note: You can access the designs and files for this module from `<install_directory>/docs/wb_tut/examples/Module_1`.

BGA Creation/Import BGA Text In

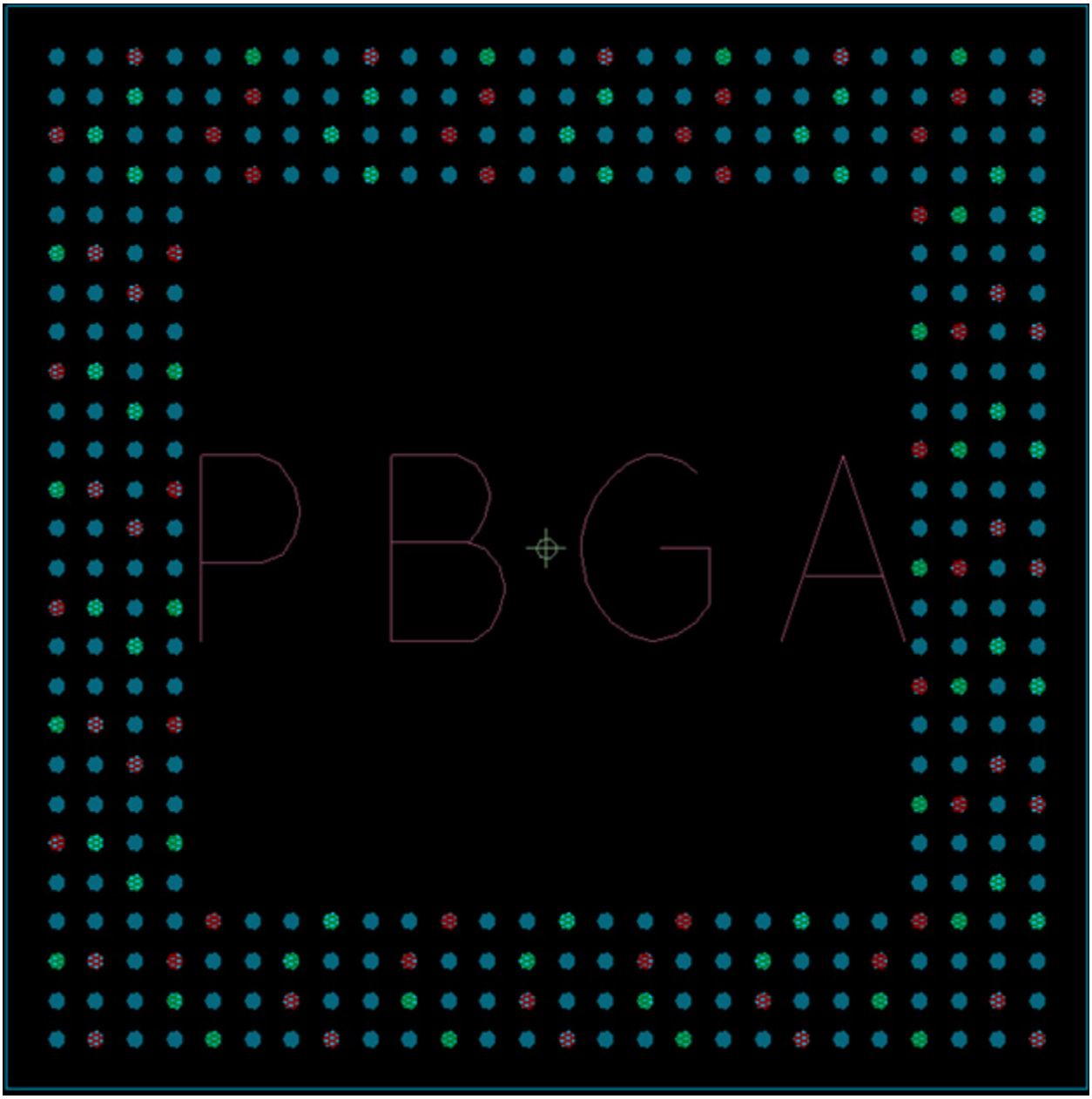
You can import an existing Ball Grid Array (BGA) using the text-in wizard. You can also easily create your own in real time or choose an off-the-shelf package from other vendors.

1. Start *Allegro X Advanced Package Designer (APD)*.

   Use either the *Allegro X Advanced Package Designer* or *Allegro Package Designer L* licenses.

2. Open the *wirebond_start* database from the *Module_1* directory. The database is currently empty.
3. Choose *Add – Standard Package – BGA Text-In Wizard* from the menu.
4. Choose the *PBGA_data.txt* file from the *Module_1* directory, and click *Open*.
5. Click *Next* to accept the defaults in the Step 2 screen.
6. Similarly, click *Next* in the Step 3, Step 3A, and Step 4 screens.
7. Click *Finish* to complete the BGA Text-in.

   Your design should look like the one shown.



Die Flag Import from DXF/compose shape Command

A die flag is a large metal shape underneath the die that adheres and grounds the wire bond die to the package substrate. Options to create die flags include:

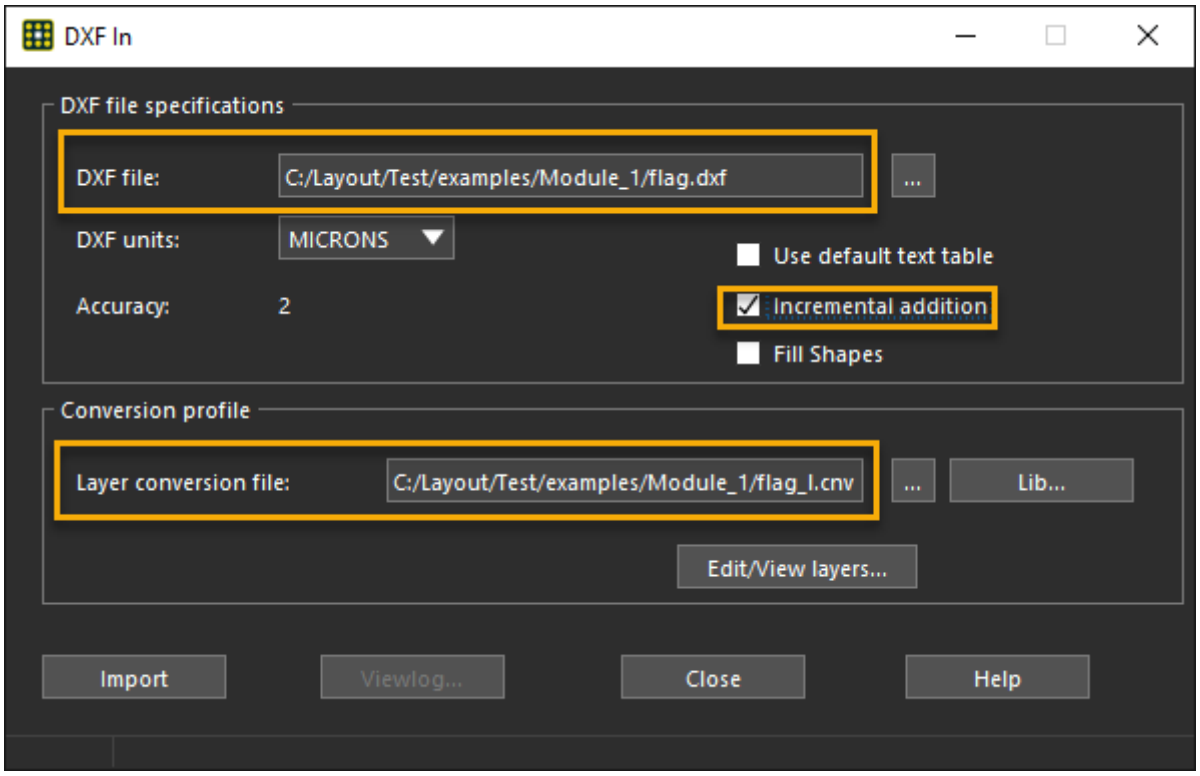
- Automatic creation with the ring generator (Refer to Power and Ground Ring Creation .)
- DXF (AutoCad) import

  In this tutorial, a DXF example is used.
- *File – Import* menu options

  Review the menu options to find other import methods for a die flag.
- In Wire Bond Edit application mode, choose *Add Flag* from the pop-up menu for a die

To import a die flag:

1. Choose *File – Import – DXF* from the menu.
2. In the DXF In dialog box, specify the *flag.dxf* file in the *DXF file* field by browsing to the *Module 1* directory and opening the file.
3. Set *Incremental addition*.



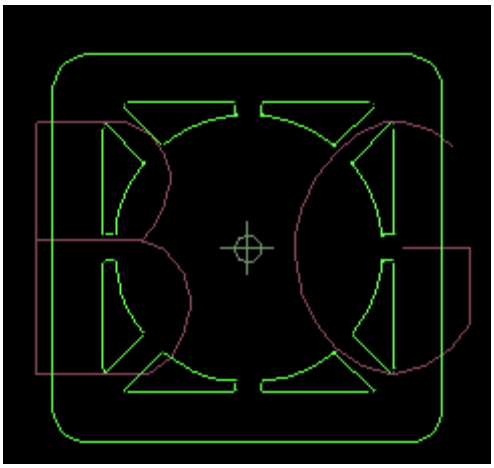
Note: On setting *Incremental addition*, a message box appears informing you that the DXF file accuracy is greater than the design accuracy. Click *Yes* to allow DXF data to be truncated.

The *flag_l.cnv* layer conversion file name appears in the *Layer conversion file* field. You can browse to and open the layer conversion file, flag_l.cnv if it is not set automatically.

4. Click *Import*.

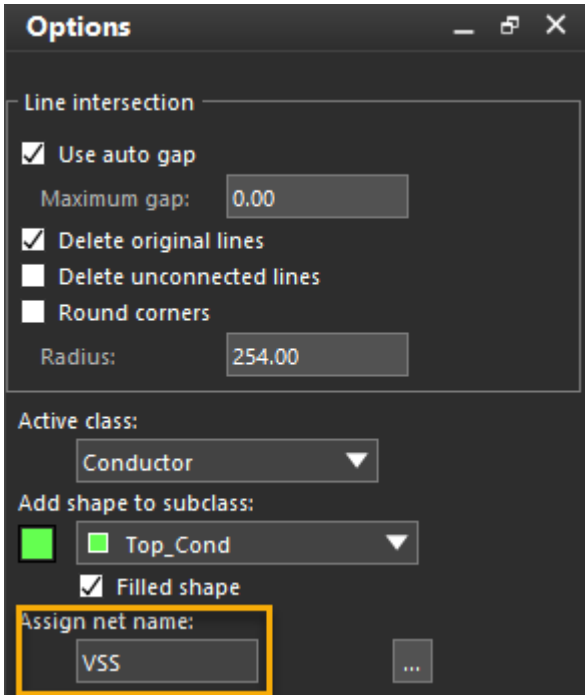
   The imported DXF shape metal is on the top layer metal.

5. Close the DXF In dialog box.



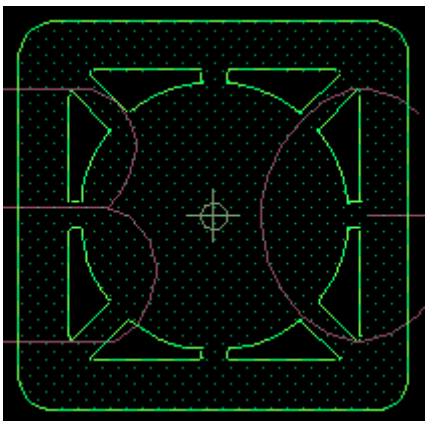
Now that the die flax has been imported, fill in the shape by doing the following steps:

1. Choose *Shape – Compose Shape* from the menu.
2. In the *Options* pane ensure the subclass for the shape is *Top_Cond* and enter or browse to the net name (*VSS*) to set the *Assign net name* field.



3. Window around the die flag.
4. Right-click and choose *Done*.

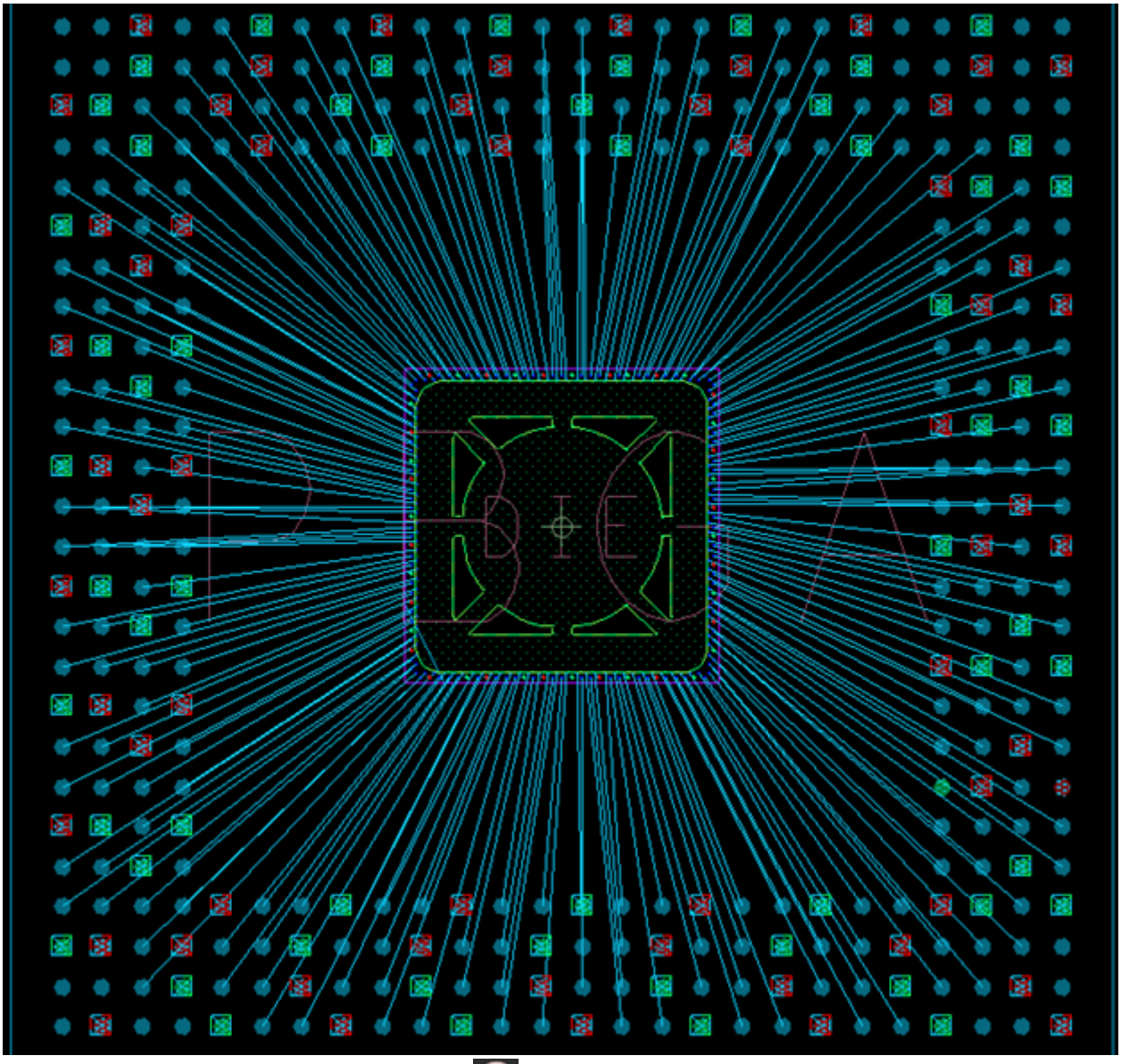
The result depends on how you define the shape in the file, but the final result for this example appears as shown.



Die Creation/Import Die Text-in

You can import die data in many ways. This example uses a common spreadsheet format.

1. Choose *Add – Standard Die – Die Text-In Wizard* from the menu.
2. Choose the *DIE_data.txt* file and click *Open* to start the import process.
3. Click *Next* in the Step 2, Step3, Step3A, and Step4 screens.
4. Click Finish in the Step 5 screen.
5. After the die has been imported into the design, click the *Zoom Fit* toolbar icon ( ) for a better view.
6. Click the *Rats All* toolbar icon ( ) to display the net assignments.



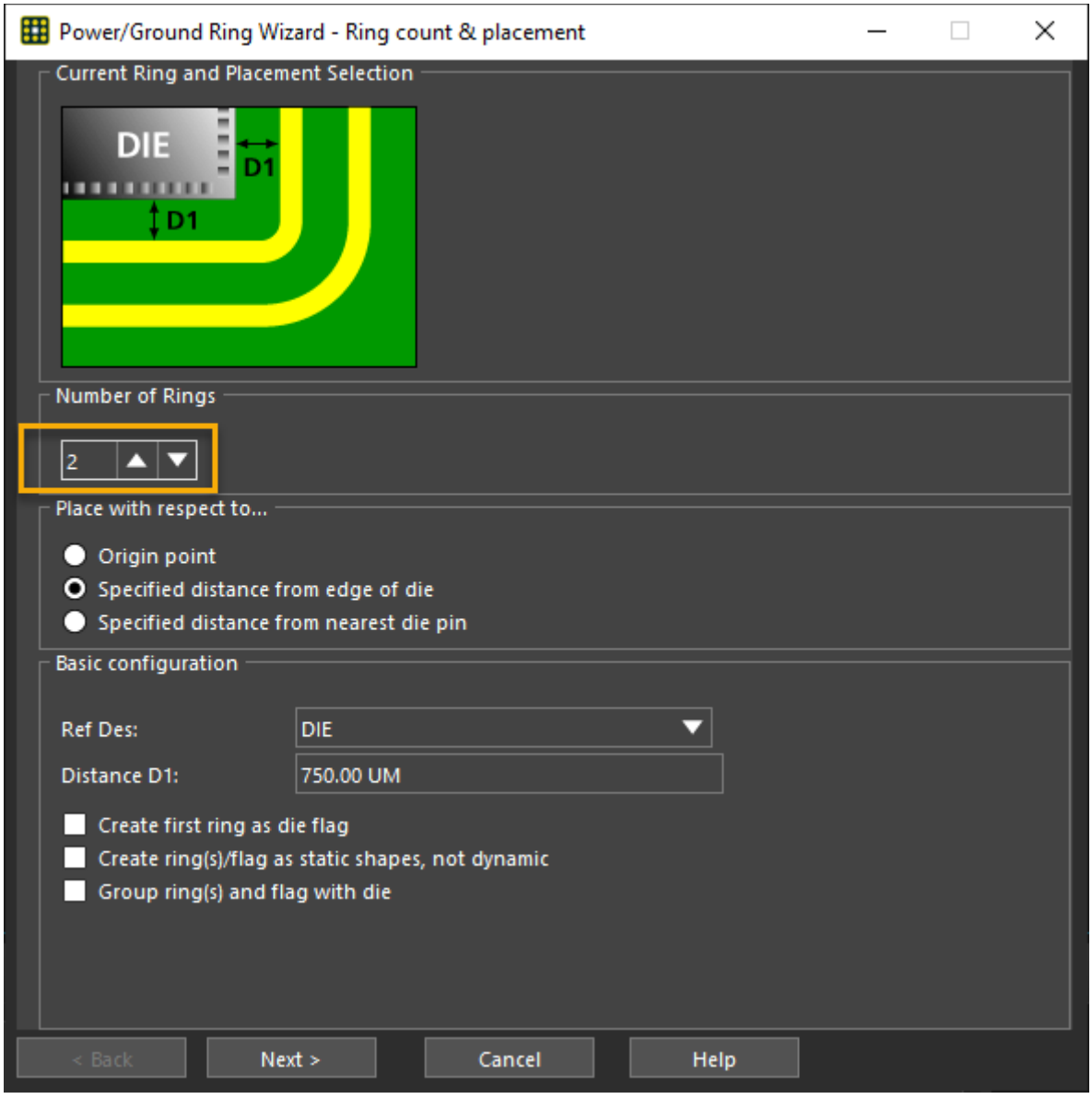
7. Click the *Unrats All* toolbar icon () to stop displaying ratsnests.

Power and Ground Ring Creation

Power and ground rings are exposed rings of metal surrounding a die that supply power and ground to the die.

Note: Before you begin power and ground ring creation, make sure that the ratsnest lines are off so that you can view the design more easily.

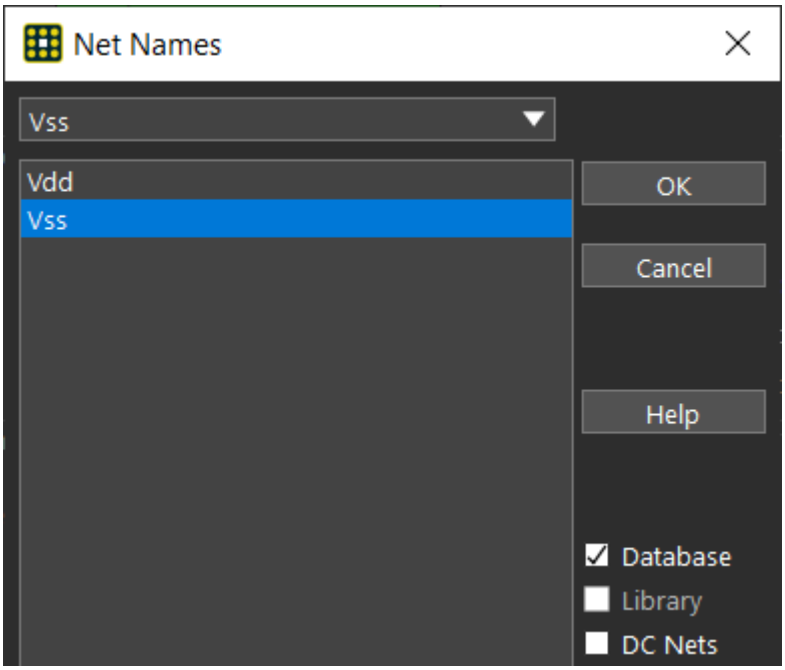
1. Choose *Route – Power/Ground Ring Generator* from the menu to start the ring generation process.
2. When the Power/Ground Ring Wizard dialog box appears, be sure that the number of rings is set to `2`, accept the other defaults, and click *Next*.



Note: You can set *Create first ring as die flag* to create a basic die flag.

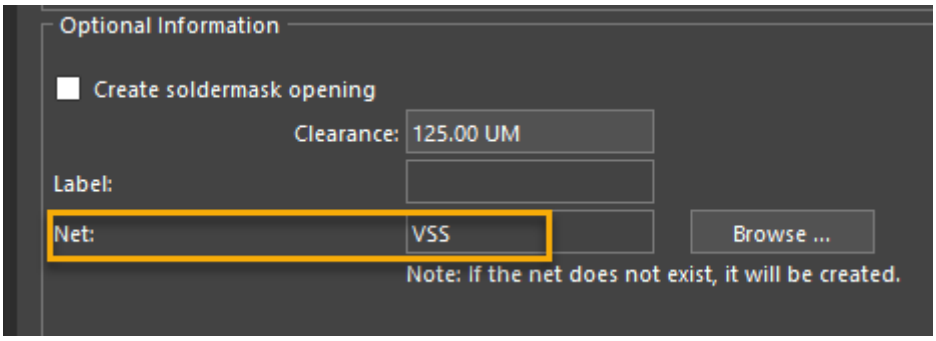
3. Define Ring 1 and the net associated with it.

a. Browse and choose *VSS* in the Net Names dialog box.



b. Click *OK*.

c. Specify the Label as VSS.



d. Click *Next*.

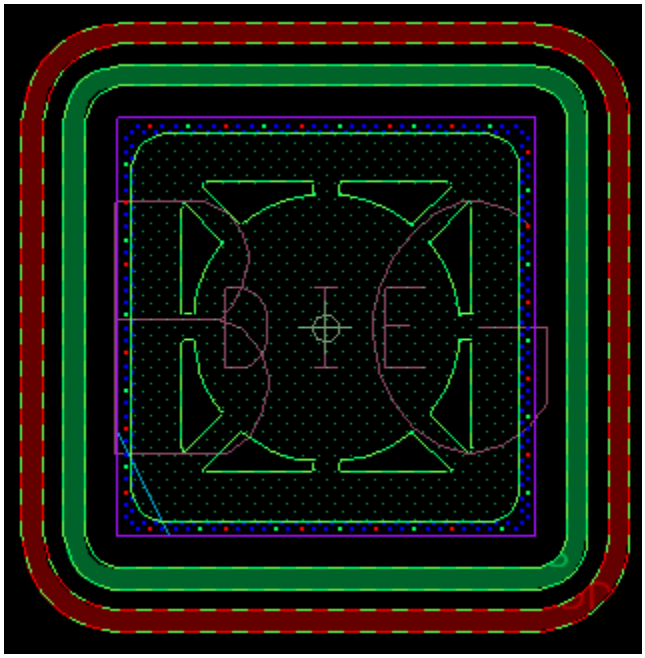
The first ring should appear in your design. It is associated with the proper net, in this case, VSS.

4. For the second ring, choose VDD as the Net and specify the Label as VDD.

5. Click *Next*.

6. Click *Finish* in the Result Verification screen to complete the process.

   The completed rings appear as shown.

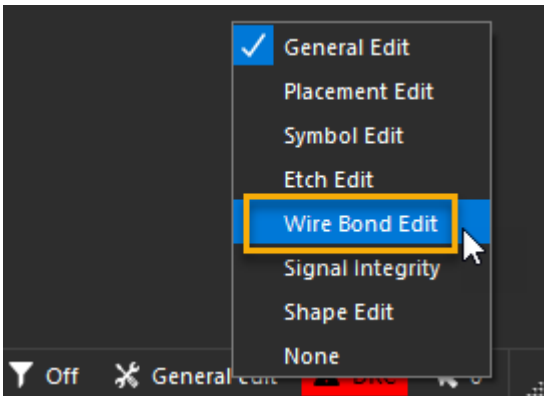


Best Fit Path for Existing Bond Fingers

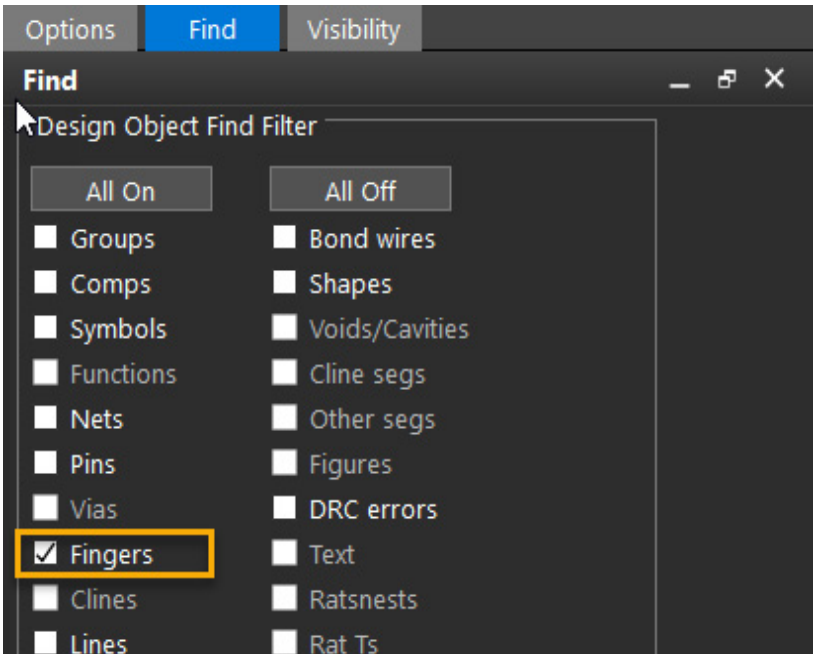
If you have a design with bond fingers in the Free Placement style, and you want to snap them to a guide path, or if a guide path was accidentally removed from the design, you may want to recompute a guide path for the affected fingers.

To create the best fit path:

1. Open the *best_fit_path* database.
2. Set up the Wire Bond Edit application mode by choosing the application mode from the Task bar.

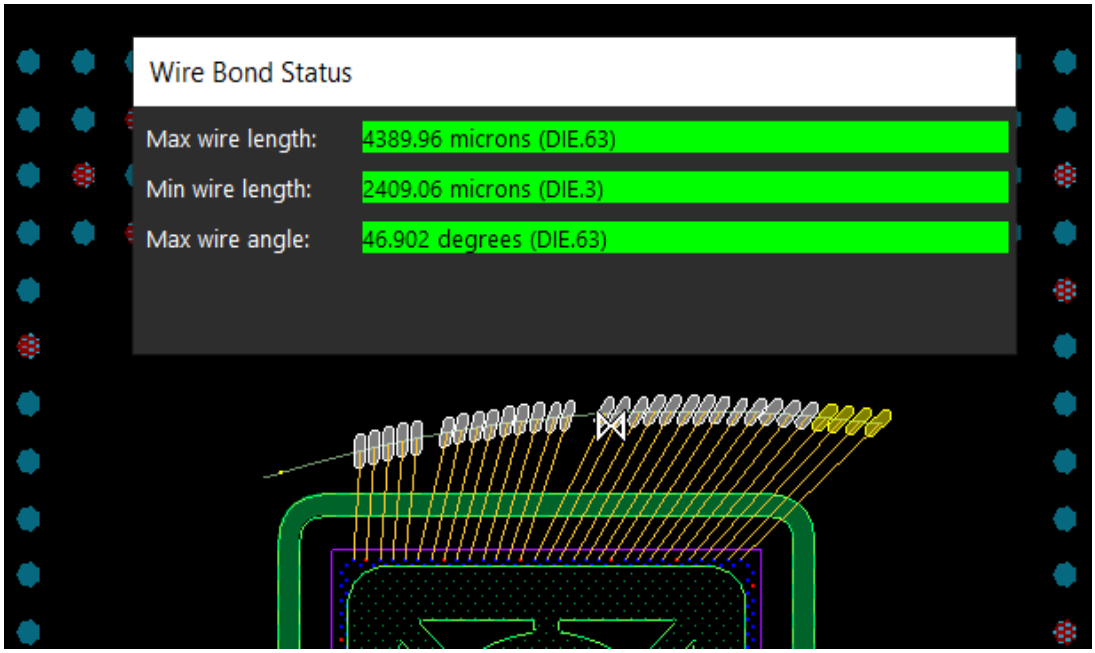


3. In the Find pane, set only *Fingers* (Click *All Off* and then set *Fingers*).



4. Window around the bond fingers for which you are creating a path, and choose *Create Best Fit Path* from the pop-up menu.

5. Select the fingers again, and choose *Move* from the pop-up menu.



The bond fingers snap to the guide path and follow it during future push and shove operations.

Module 2: Creating and Editing Guide Paths and Wire Bonds

This module covers:

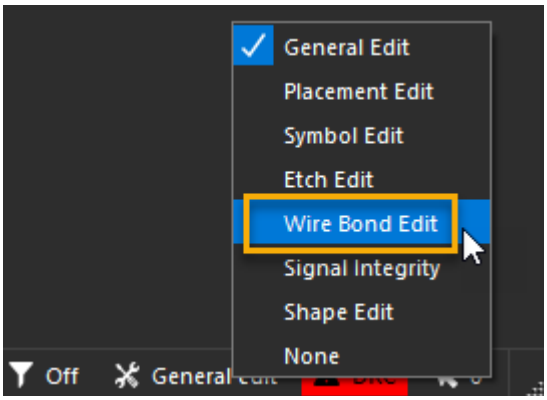
- Wire Bond Guide Creation
- Advanced Filtering and Wire Bond Settings
- Wire Bond Process And Flow
- Wire Bond Push, Shove, and Move
- Wire Bond Merge
- Wire Bond Tack Point Move
- Wire Bond Redistribute
- Wire Bond Space Evenly
- Wire Bond Center

Note: You can access the designs and files for this module from `<install_directory>/docs/wb_tut/examples/Module_2`.

Wire Bond Guide Creation

1. Start APD.
2. Open the *ready4guides* database from Module_2.

3. Set up the *Wire Bond Edit* application mode by choosing the application mode from the Task bar.



   The active application mode is shown on the right of the task bar.

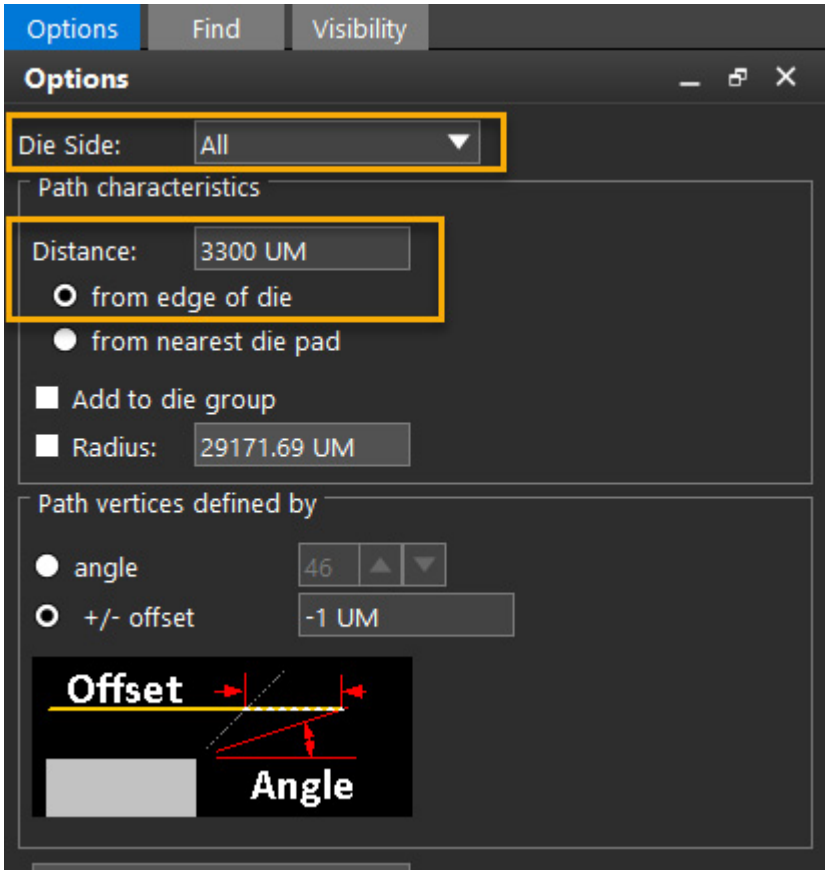
4. Make sure that *Comps* is set in the *Find* pane. You must also set *Lines* if there are existing guides that you are going to edit.

5. Click to select the die and then choose *Add Guide Path* from the pop-up menu.

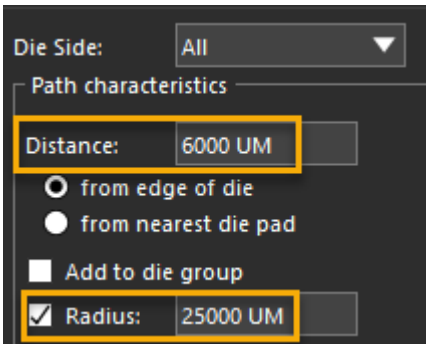
6. In the *Options* pane, choose *All* for *Die Side*.

   You can select one or all of the die sides. For this example, choose *All* as it saves time and there will be additional guides for wire bond trade-offs (possibilities for considering other solutions).

7. Type `3300 UM` in the *Distance* field and ensure *from edge of die* is set.

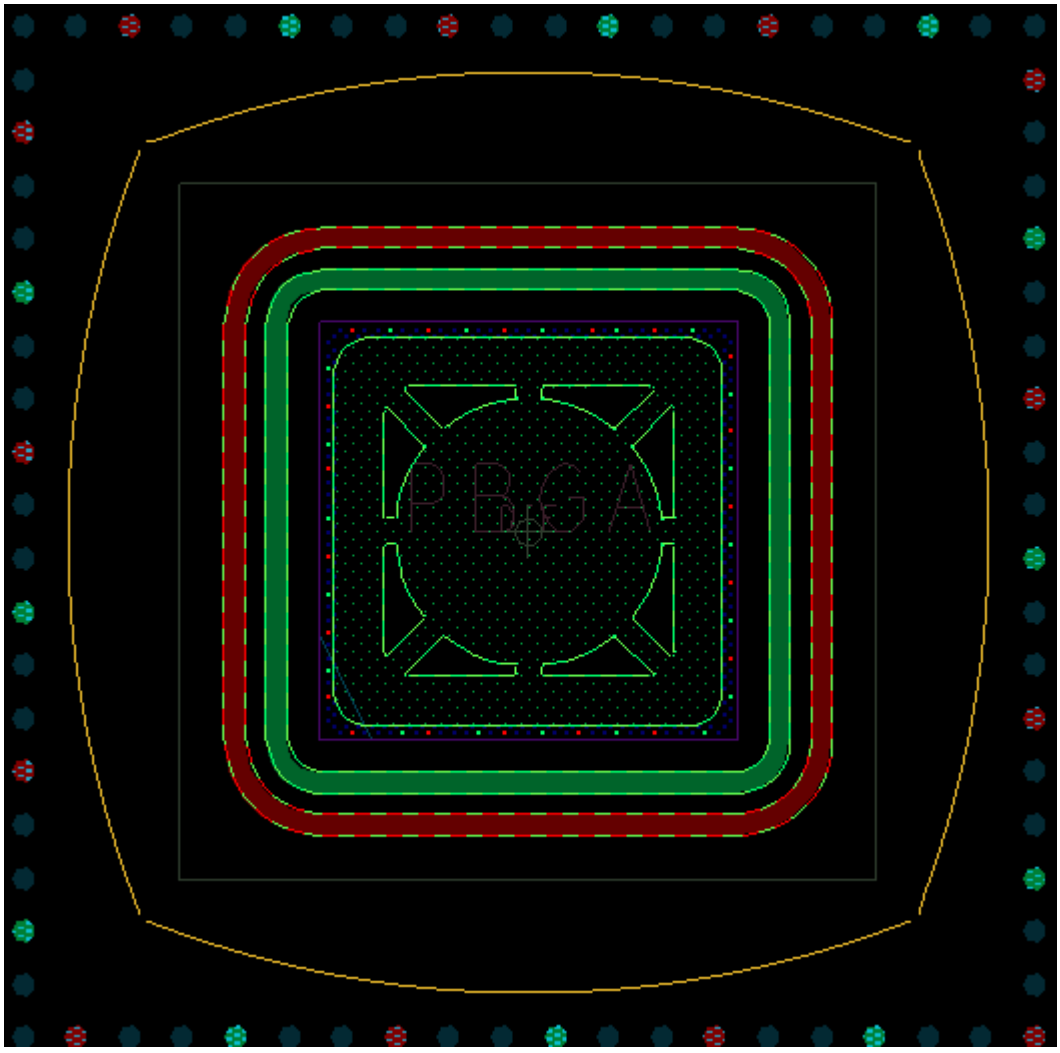


8. Click *Create guide paths*.
9. Specify at a distance of `6000 UM` and set a *Radius* of `25000 UM` for the final guide path.



10. Click *Create guide path(s)* and then click *Done adding guides*.

    You can easily create and edit any type of path. Options are available depending on whether you have bond fingers attached to the path.



11. Set *Lines* in the Find pane, then select the guide that you want to edit.

The guide is highlighted.

12. Choose *Edit* from the pop-up menu, and select the center point on the line.

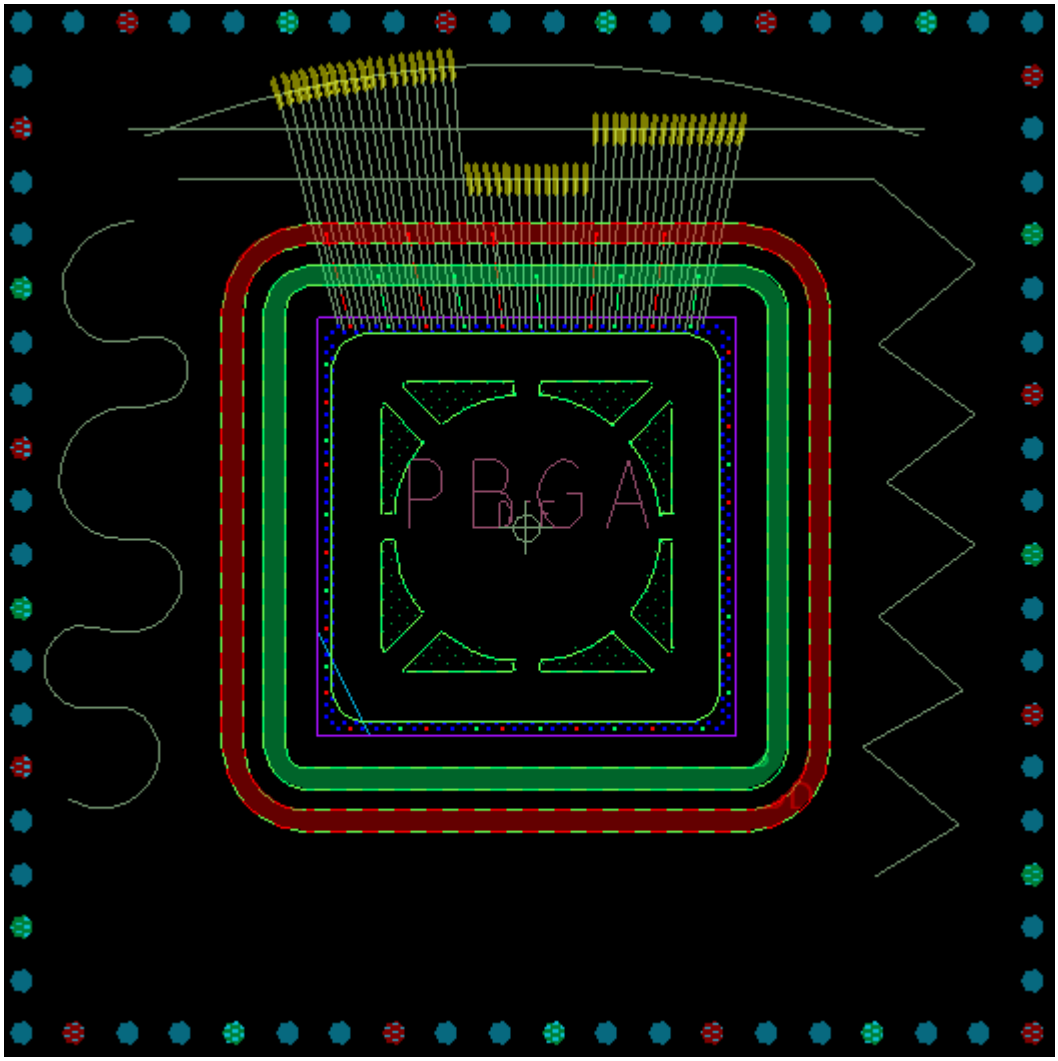
The options in the pop-up menu are: *Edit* (change the shape of the line), *Move* (move the line keep the shape), and *Copy* (copy the line).

This example shows the flexibility of the wire bond guide paths.You can edit, move and copy wire bond guides when you are in the Wire Bond Edit application mode. You can add a line to the *Wb_Guide_Line* subclass and shape it as you want.

You can also use the *Edit – Vertex* menu command on an existing line.

13. Open the *ready4guide_edit* database from the *Module_2* directory

You do not need to save the previous database. Notice the sawtooth shape in the design, which shows the flexibility of the tool. You can add guides of any type or shape.



Advanced Filtering and Wire Bond Settings

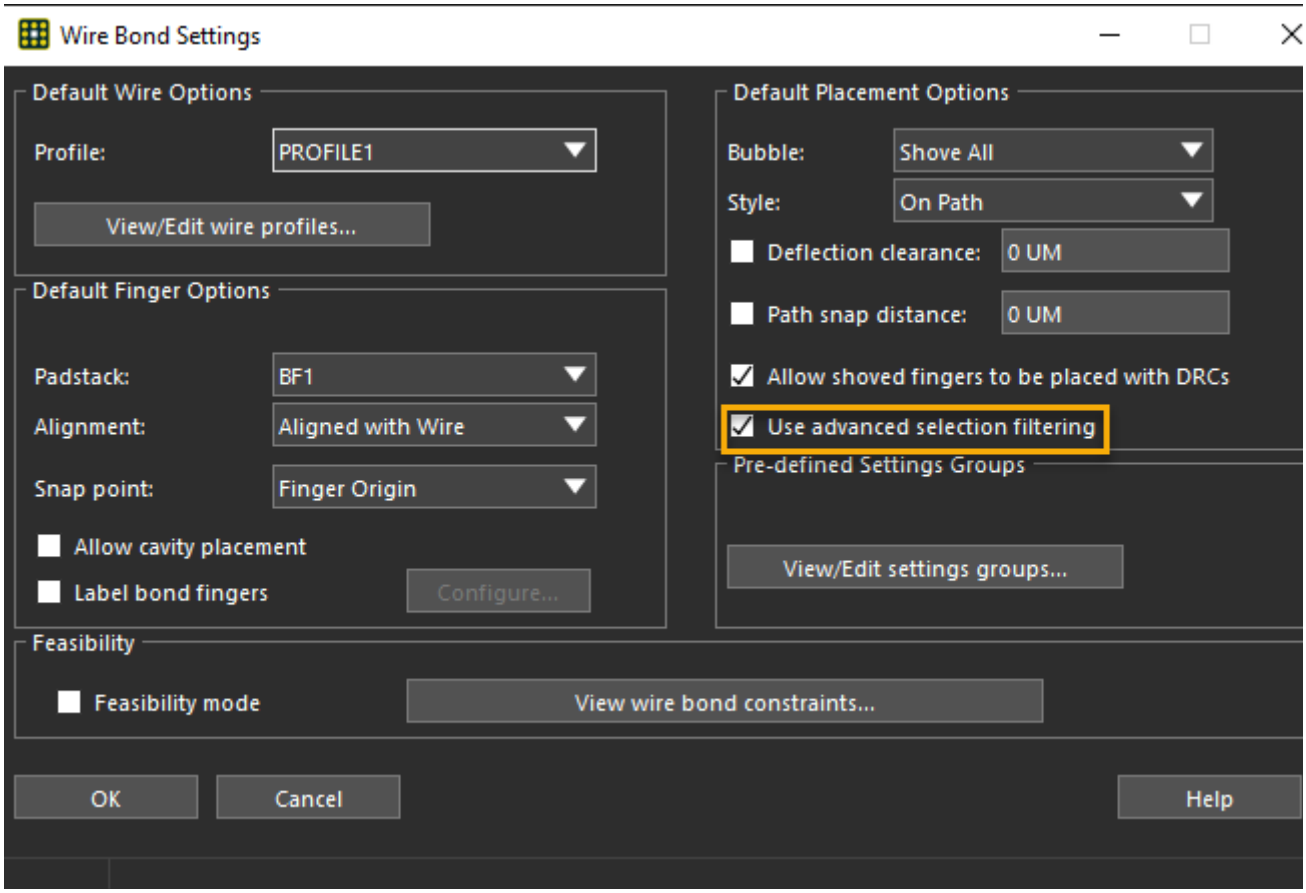
The Advanced Selection Filtering option lets you filter nets or pins, or both, when you are adding wire bonds using the `wirebond select` and `wirebond add` commands:

To perform advanced filtering during a wire bond operation:

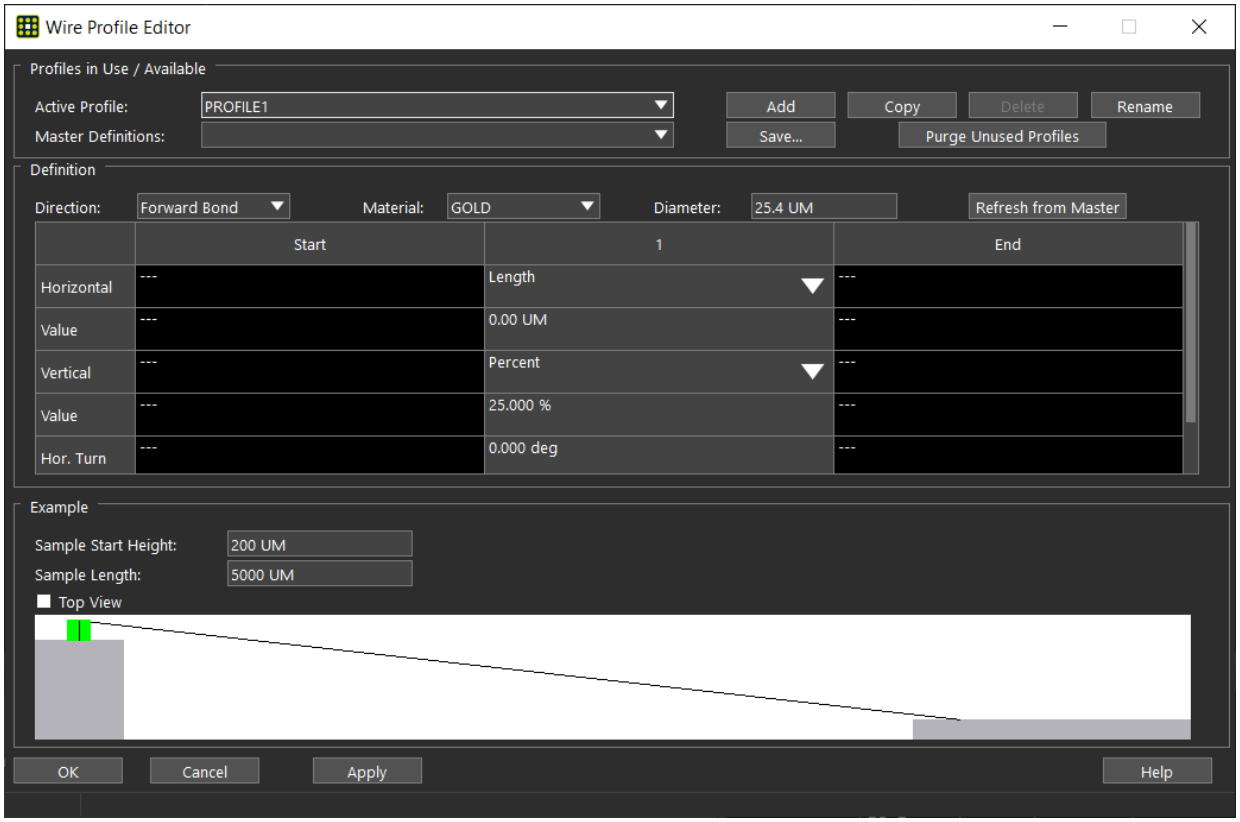
1. Open the *advanced_filtering* database. You do not have to save the previous database.

2. Select the die and from the pop-up menu choose *Quick Utilities – Wirebond Global Settings*.

The Wire Bond Settings dialog box that appears allows you to specify global wire bond settings such as wire profiles, define bond finger options, and update bond finger labels and DRC markers.



3. Click *View/Edit wire profiles* to open the Wire Profile Editor.



The Wire Profile Editor displays the active wire profile, which is *PROFILE1* in this example. PROFILE1 is the default profile. You can select the active profile from a list of those currently in the design or in the Master Definitions file. Once you select a profile, the screen refreshes to show data in that profile. You can also edit, add, and copy profiles, or delete profiles that are not being used in the design.

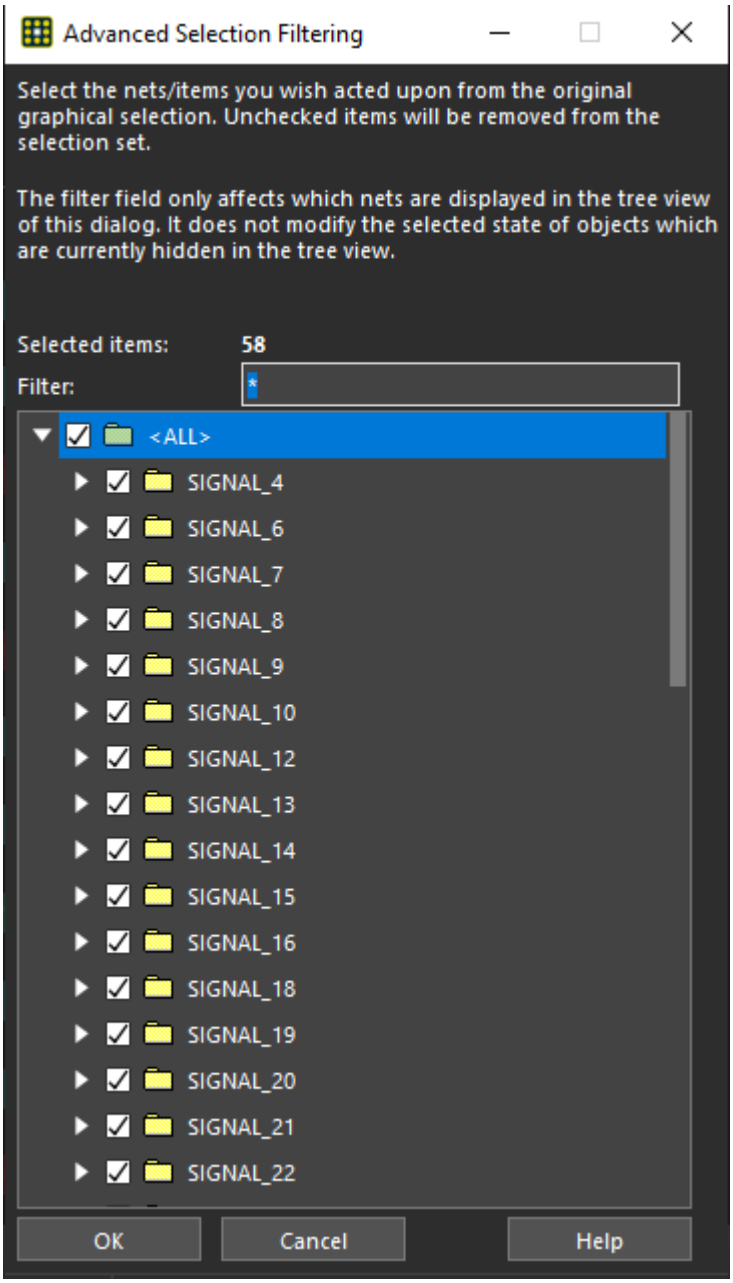
4. Click *Apply*, and then *OK* to close the dialog box.
5. Ensure the *Use advanced selection filtering* is set in the Wire Bond Settings dialog box, and click *OK*.

   Note: This setting is persistent in the database. Once you turn on this setting, it is on until you turn it off.

6. Set only *Pins* in the Find pane.
7. Select the pins at the top of the die, and choose *Add Wire Bond* from the pop-up menu.

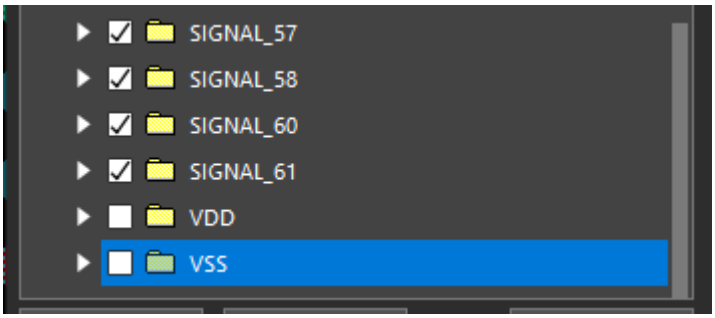
   The *Advanced Selection Filtering* dialog box appears. The tree view displays top-level items (the nets) that you can click on to see the pins associated with them.

Checking the box next to an item includes it in the filtered set. Unchecking the box removes it from the set. You can use *All* to toggle the visibility for all items to the same state in one click. By default, all items originally selected in the design are checked.



Ensure that any nets that you would like to remove from your initial selection are not set.

8. For this example, scroll down to the bottom of the list, and ensure that *VDD* and *VSS* are not set.



   The package tool ignores these pins when adding wire bonds.

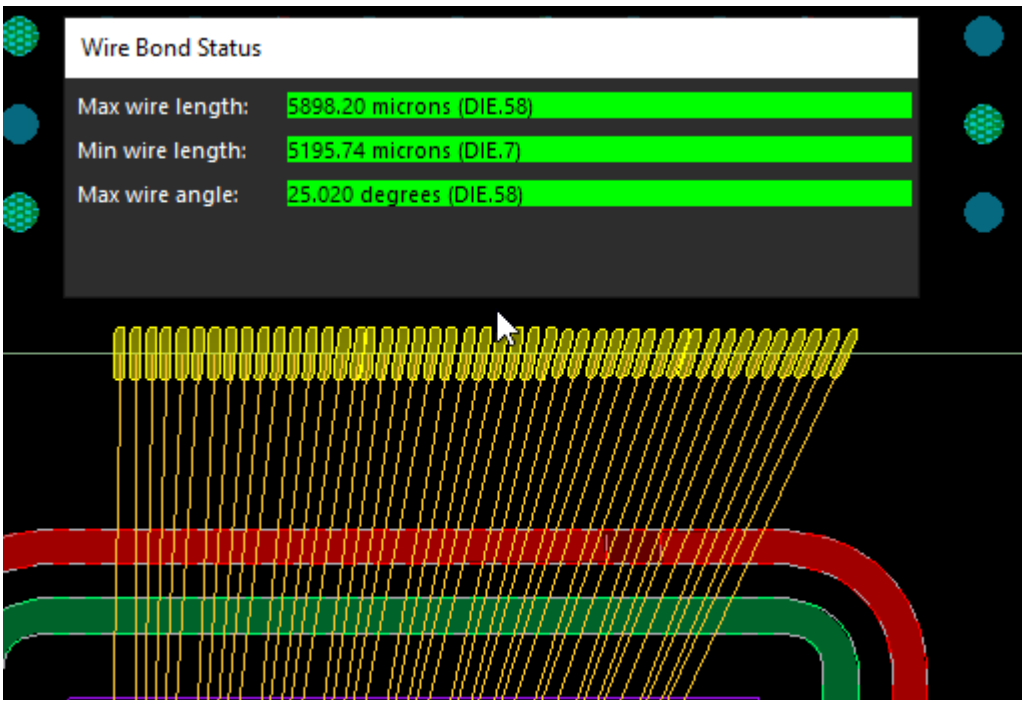
   By default, the *Filter* field displays an asterisk (*) which means that the list displays all the selected nets.

   Look at the canvas, and you will see the highlighting update. This helps you to visualize what will be wire bonded.

9. To manage your list, modify the *Filter* field by typing in `SIG*` and then pressing the `Tab` key.

   Note: This field is case-sensitive.

10. Now, remove selection from *SIGNAL_60*, *SIGNAL_61*, *SIGNAL_62*, and *SIGNAL_63*, then click *OK*.



The wire bonds appear. The tool ignored the pins that you excluded in the Advanced Filtering Selection dialog box. You can also exclude pins and nets from wire bond add operations by attaching the `No_Wirebond` property.

11. Click to place the wire bonds.

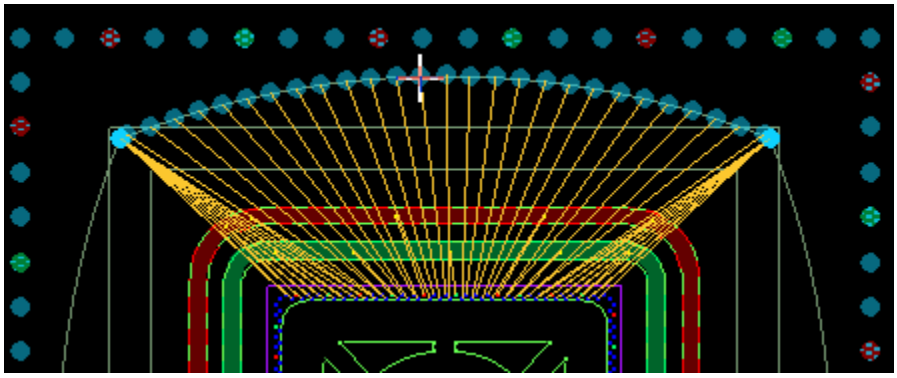
Wire Bond Process And Flow

This section reviews the wire bond process and flow.

1. Open the *ready4WB* database from the *Module_2* directory.
2. Make sure *Pins* is selected in the Find pane.
3. Select the die pins that you want to wire bond.
4. Choose *Add Wire Bond* from the pop-up menu.

   The default mode enables that the bond fingers automatically attach to your cursor.

   The wire bonds are now attached to your cursor.



5. Check the *Options* panel and command window for options and status.

   In the Options pane, you can change the bond finger, profile (the 3D curvature model for a set of bond wires), and layer, as well as the Pattern style and Bubble type (push and shove types).

6. Select the pull-down menus in the *Options* panel to see the other options. For additional information, see the user documentation.

7. Watch the Wire Bond Status window as you move from guide to guide.

   If you have a very large designs (high pin count), DRC errors can slow down the performance.

8. Right-click and choose *Settings* to open the Wire Bond Settings window.

   You can now change DRC settings in the Constraint Manager. The constraints are editable only if you set *Feasibility mode*.

   Note: Constraints with the feasibility option might not meet requirements for the finished design. Use this only for feasibility.

   Set *Label bond fingers* and configure to update the bond finger text labels so that you never have to worry about the text getting out-of-date with the BOND_PAD property values.

   Note: You can also refer to variable settings in User Preferences (enved command) that affect the placement engine.

   Setting the *WIREBOND_IGNORE_WIRE_PROFILES* environment variable causes the tool to treat all wires as part of the same profile for placement only.

   By default, during wire bond placement, the tool attempts to satisfy wire-to-wire, finger-to-wire, and finger-to-finger constraint values whether or not these values are currently enabled for online DRC checks. By setting the

WIREBOND_IGNORE_DISABLED_CNS environment variable the constraint is not used to guide placement if it is not enabled for DRC.

Wire Bond Push, Shove, and Move

In APD, many processes require pads to be shapes.

1. Open the *ready4push_shove* database.

   Click *Zoom Fit* to be able to see the design properly.

   A few bond wires (used to establish electrical connectivity between die or from die to package substrate) are added to the original design on different guides to show how easily the tool handles it.

2. Make sure that *Fingers* is set in the Find pane.

3. Select the bond fingers that you want to edit, and choose *Move* from the pop-up menu.

4. Check the command window for informational messages.

   There is a built-in hierarchy in the select command's object selection. You do not need to change your *Find* panel often. Below is the selection hierarchy:

   - Groups
   - Components
   - Pins
   - Fingers
   - Bond wires
   - Guide Paths (Lines)
   - Shapes

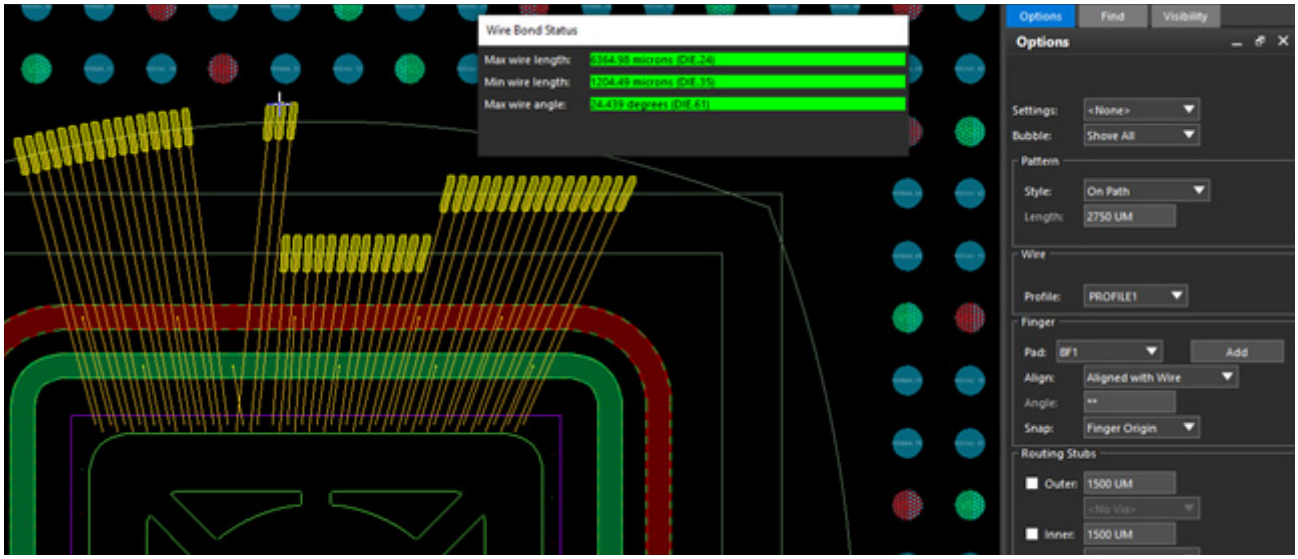
   This means that if you have both fingers and guide paths checked in the *Find* panel, when you window and select objects of both types, the tool automatically performs the task for the higher-priority finger objects, and disregards the guide paths. It saves you from modifying the Find Filter window pane in most cases unless you need to turn groups or components on or off.

5. Check the *Options* panel for the type of move (Bubble). In this case, the tool pushes and shoves all wires and bond fingers on all guides.

6. Select a bond finger, move it to another guide then push it around. This mode limits the number of DRC errors that are generated.

   With the Bubble option changed to *Shove Path*, wires and fingers on the same guide move, but the other guide paths remain unchanged. Of course, this causes DRC errors if the profile and groups are the same as they are in this simple example.

7. Try some of the other Bubble options and see how it affects the results.



You can also move the wire bond guides just as you moved the wire bonds previously.

You may also want to set up default actions for the *Route – Wire Bond – Select* command at some point. As an example, since most of the time when you pick bond fingers, you want to move them, you can right-click and choose the *Move* option once. Then, while moving the fingers, you can right-click and choose the *Set Default Action* option.

The next time you select fingers, the tool automatically assumes that you want to move them and puts them on your cursor without your having to display the mouse pop-up. Then, if needed, you can override that default operation on a per-use command (so you could right-click and choose *Delete* instead of the default action, *Move*).

8. Set *Lines* in the *Find* panel, select the guide that you want to move, and choose *Move* from the pop-up menu.

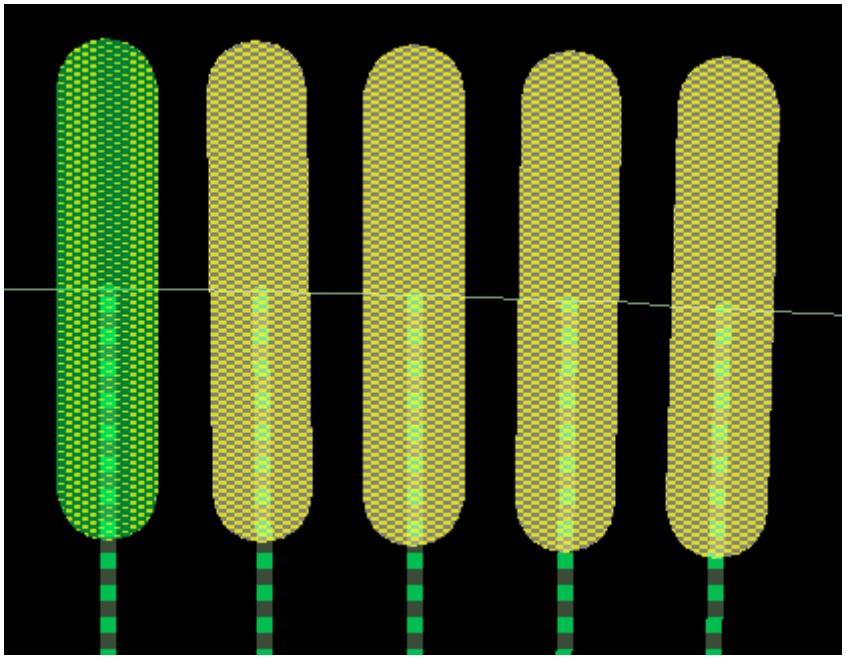
Wire Bond Merge

1. Open the *merge_pads* database from the *Module_2* directory. You do not need to save the previous database.

   A separate design is used for this example because the bond fingers must be on the same net (VSS in this case) to run merge pads.

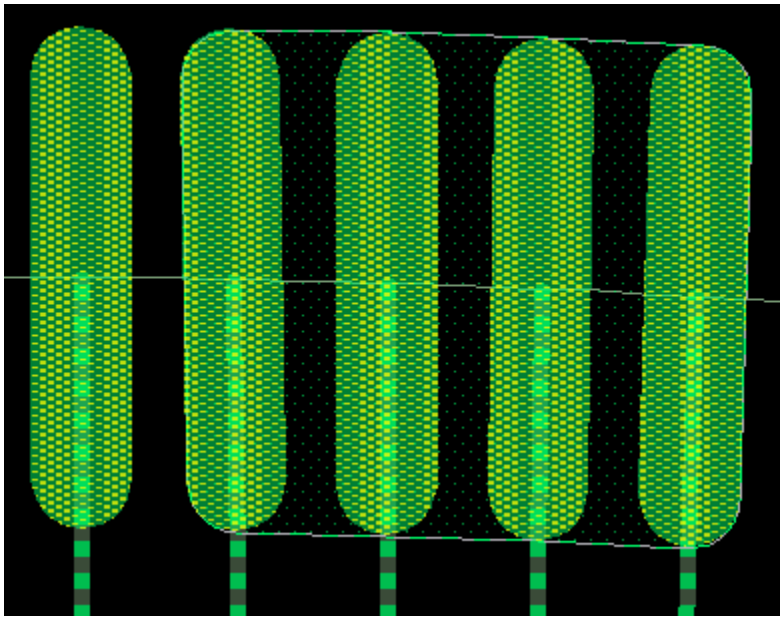
2. Make sure that Fingers is selected in the *Find* panel.

3. From the five fingers in the middle, select the right-most four fingers.



4. Choose *Create Merge Finger Shape* from the pop-up menu.

   Your design should show that all the bond fingers are connected.



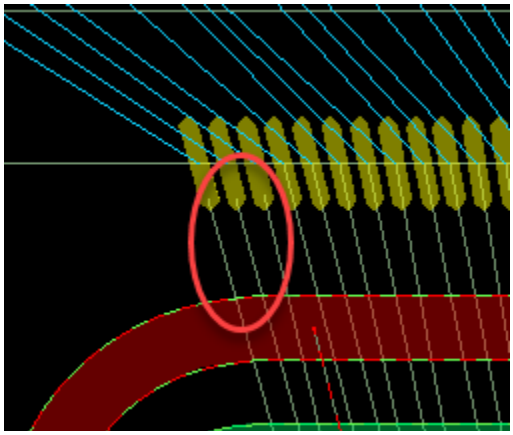
You can push and shove the merged pads as well. The tool maintains the relative spacing between the connected fingers. Or if you want to push them together for a smaller pad, set the Same Net constraint (set the *Bondfinger to Bondfinger* constraint in the *Same Net Spacing* domain in Constraint Manager to `0 UM`). You will have to push them together.

Wire Bond Tack Point Move

With the Wire Bond Tack Point Move feature, you can move one or many wires along the finger axis.

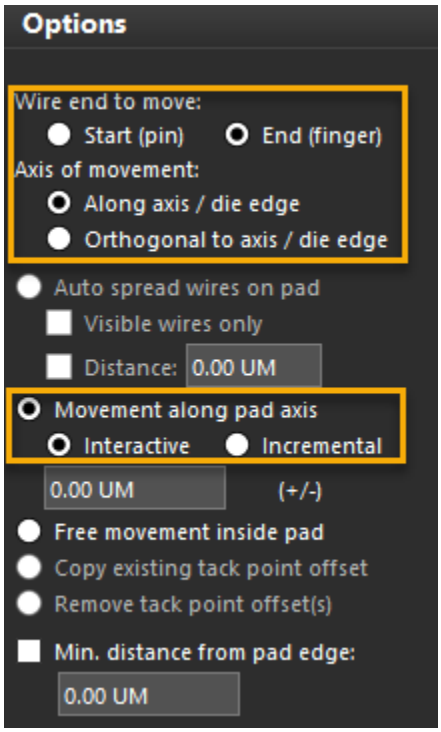
1. Open the ready4tackptmv database from the Module_2 directory. You do not need to save the previous database.

   You will move the wire bonds in the red circle.



2. Set *Bond wires* in the *Find* panel.
3. Window around the bond wires to select them.
4. From the pop-up menu, choose *Move Tack Point*.

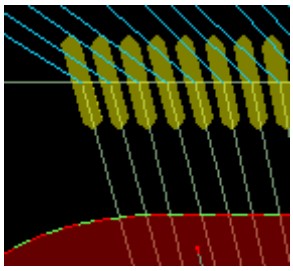
   Observe the *Options* panel.



Tack point is the contact point of the bond wire to the bond finger or the bond wire to the die pad. We will change the *End (finger)* along the axis/ die edge. The movement along pad axis will be interactive, that is, we will use mouse-clicks to place the wire end.

The wire bond moves with the cursor.

5. Click to place the wire ends as shown.



Perform this task as a final step in the design process, just as you would do with the bond finger soldermask generation.

Wire Bond Redistribute

You can take bond fingers from one guide path and distribute them evenly with two or more guide paths.

1. Open the *ready4redistribute* database from the *Module_2* directory.

   You do not need to save the previous database.

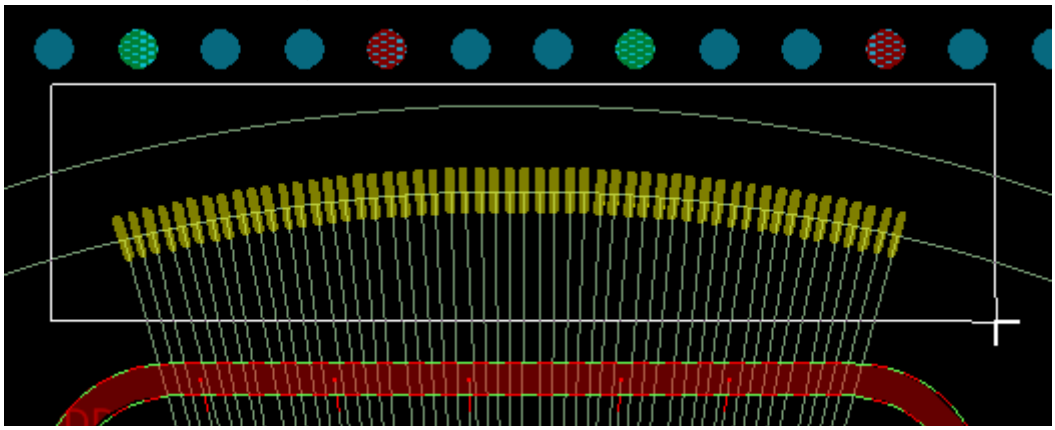
2. Set *Lines* in the *Find* panel.

3. Click on the lower guide path with all the bond fingers on it (becomes highlighted), and choose *Redistribute Fingers* from the pop-up menu.

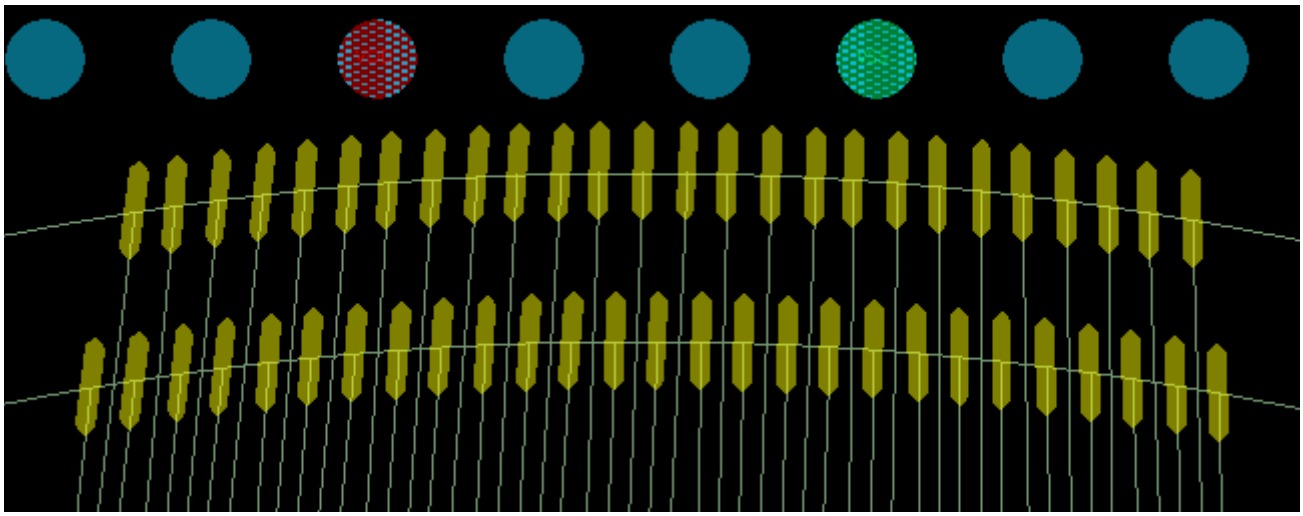
   You can explore the options in the pull-down menus of the *Options* panel.

   Remember to look in the command window for more information.

4. Window around both the guide paths.



The bond fingers are distributed between the two guide paths.



Wire Bond Space Evenly

1. Open the *ready4space_even_center* database from the *Module_2* directory.

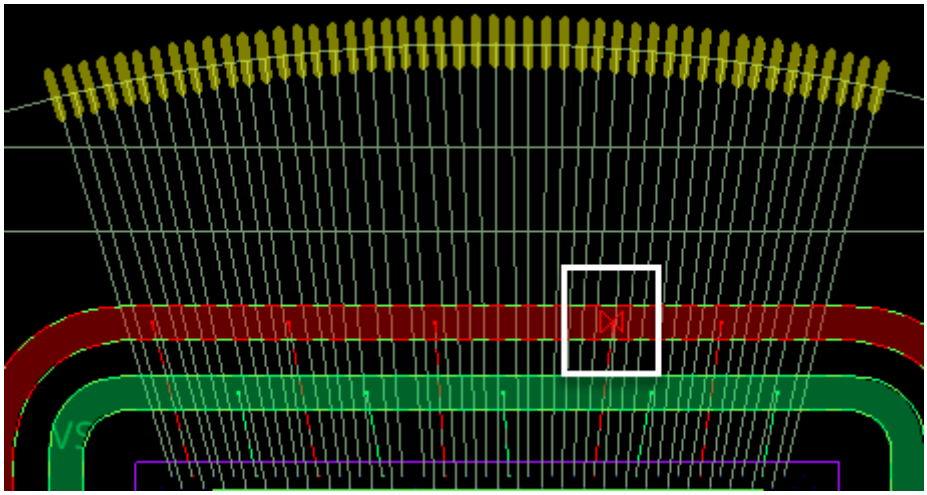
   You do not need to save the previous database.

   The bond fingers are not evenly distributed.

2. Set *Fingers* or *Bond wires* in the *Find* panel.
3. Window around the entire group of bond fingers.
4. Choose *Space Evenly* from the pop-up menu.

The bond wires and fingers are spaced evenly.

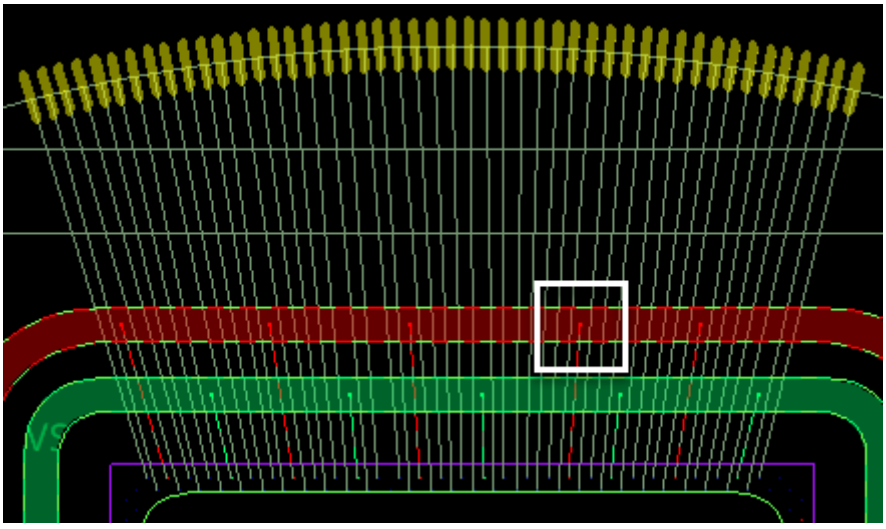
There is a DRC error. You can clean it up with the Wire Bond Center feature.



Wire Bond Center

1. Set *Bond wires* in the Find pane.
2. Select the wire bonds.
3. Choose *Center Wires* from the pop-up menu.

   The wires are centered and spaced evenly.



Note: You can set the environmental variable *wirebond_center_ring_bonds* to run a post-process step to attempt to center the power and ground wires between adjacent wires on both sides. Although this improves manufacturability, it might result in a slight performance decrease during interactive wire bonding.

4. Exit APD.

Module 3: Die Shrink and ECO

This module covers:

- Die Shrink and the ECO Process

Note: You can access the designs and files for this module from `<install_directory>/docs/wb_tut/examples/Module_3`.

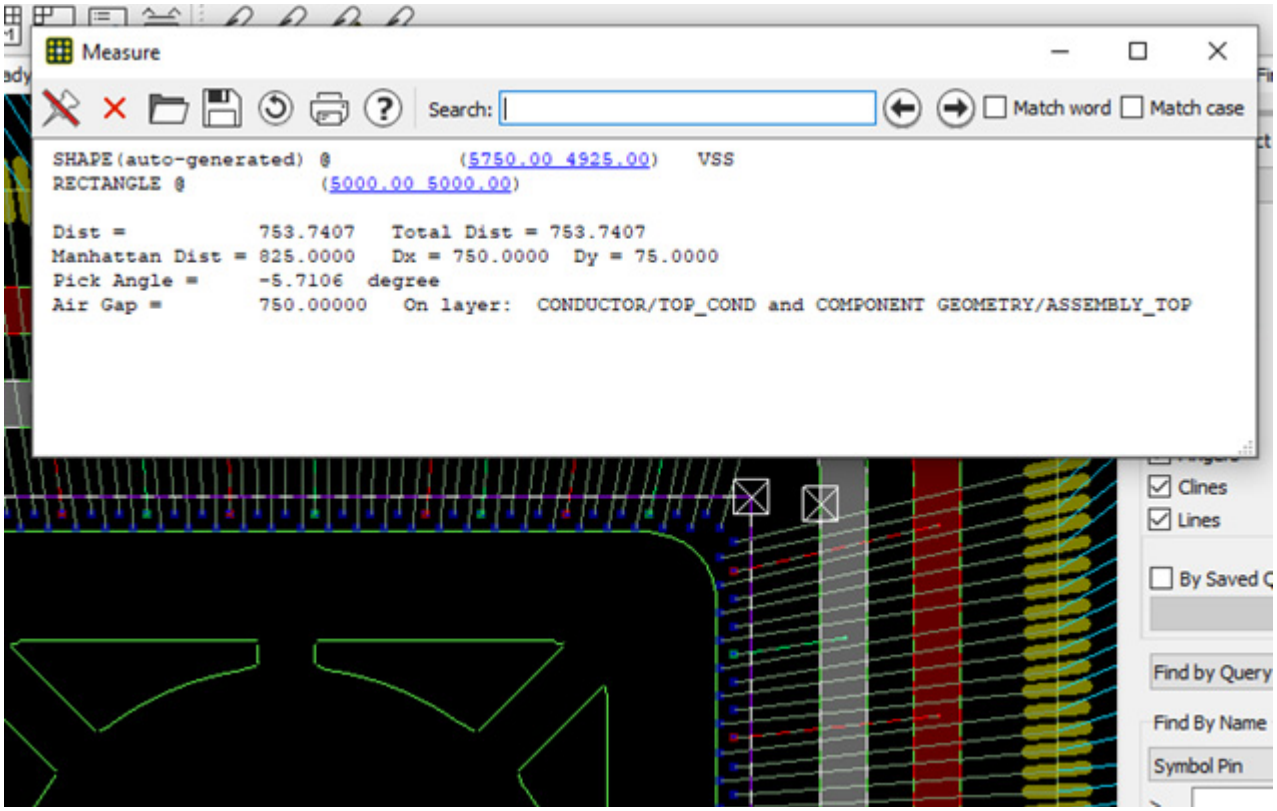
Die Shrink and the ECO Process

After you have the die in the package, you may want to do an ECO or shrink the die. This process is easily supported by APD.

1. From *Module_3* open the *ready4ECO* database.

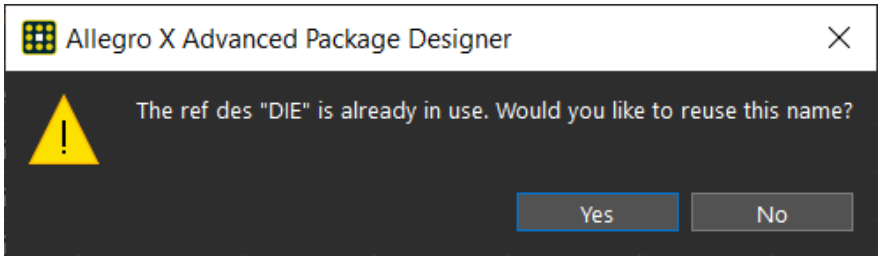
   You will simulate an ECO where the die has been shrunk by `20%` within the text file. Also notice that you can do a "shrink" in real time using all the die import forms (see Figure 4-22)

2. Click Show Measure () to measure the distance from the die edge to the ring to show the change in die size.



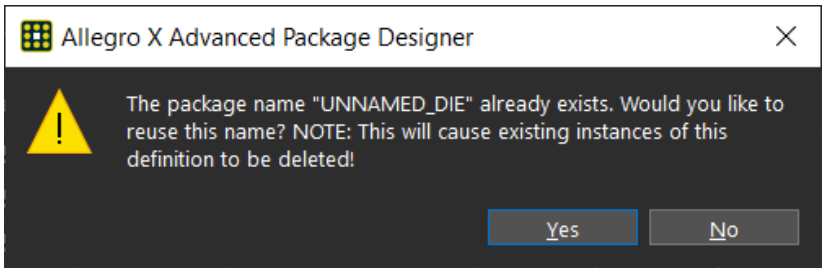
3. To replace the current die with a new die, choose *Add – Standard Die – Die Text-In Wizard* from the menu.
4. Choose the *DIE_data_shrink.txt* file representing the replacement die, and click *Open*.
5. Accept the defaults on the Step 2 screen, and click *Next*.
6. Accept the defaults on the Step 3 screen, and click *Next*.

   A prompt appears stating that the reference designator DIE is already in use.

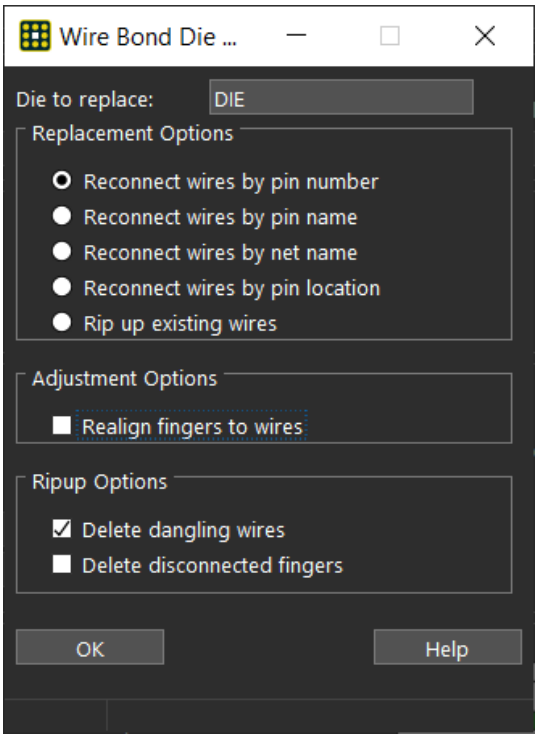


7. Click *Yes*.

A prompt appears stating that reusing an existing package name will delete instances of this definition.



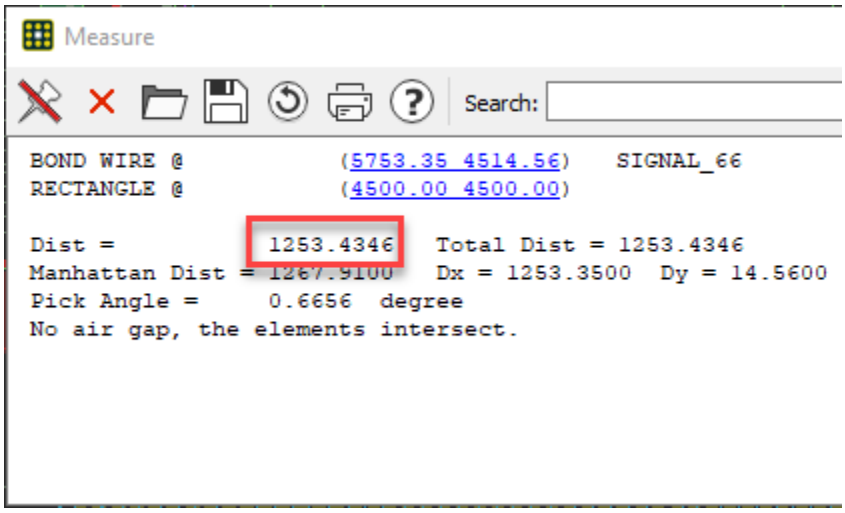
8. Click *Yes* to confirm the deletion of "UNNAMED_DIE".
9. Accept all the defaults in the Step 4 screen and click *Next*.
10. In the *Wire Bond Die Replace* dialog that appears, unset *Realign fingers to wires*.



You can reconnect wires according to pin numbers, net names, or pin location. You an also select rip up existing wires to delete all wires and then connect the wires manually. You can also realign wires, and delete dangling wires and disconnected fingers.

11. Click *OK*.

12. Click *Finish* to complete the import of the new die (`20%` manufacturing shrink) into the design.



You have read in a new die that has been shrunk, but you can also do this in real time by specifying, say another 5% in Screen2 (Step 5).

Observe that the smaller die is automatically reconnected to the wire bonds. There are a few DRC errors that need to be cleaned up, but it is easy to move them and clean up.

Module 4: Using the Cadence 3D Design Viewer

This module covers:

- Cadence 3D Design Viewer Extraction
- Multiple Profiles in Cadence 3D Design Viewer with DRC

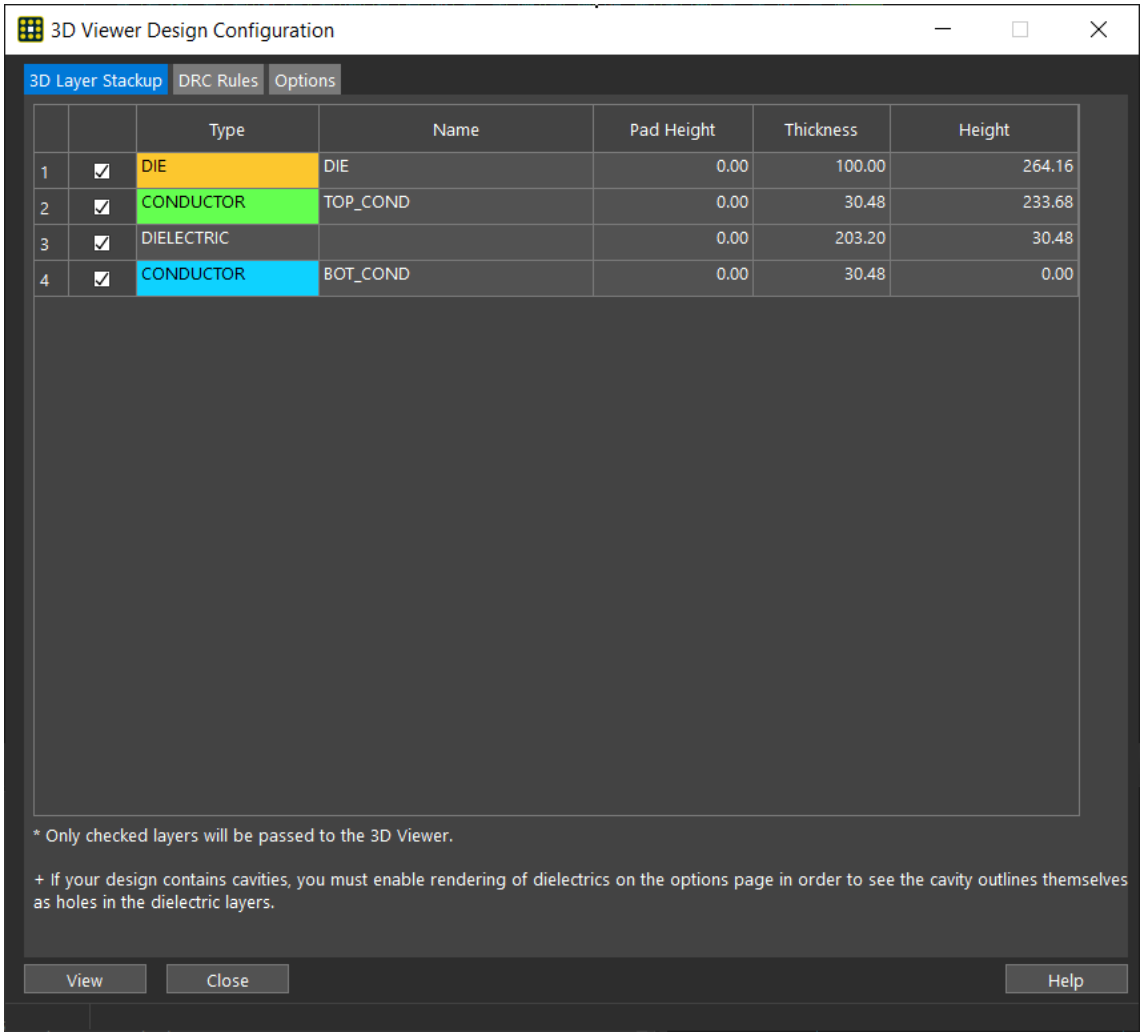
Note: You can access the designs and files for this module from `<install_directory>/docs/wb_tut/examples/Module_4`.

Cadence 3D Design Viewer Extraction

Note: Although available by default for the *Allegro X Advanced Package Designer* license, you must select the *3D Viewer* option if you are use the *Allegro Package Designer L* license.

Note: You might have to register the Microsoft Flex Grid control to be able to work with Cadence 3D Design Viewer. To register the control, open command prompt (choose *Start – Run* and enter `cmd`) and run the command `regsrv32 <installation>\tools\3DViewer\msflxgrd.ocx`.

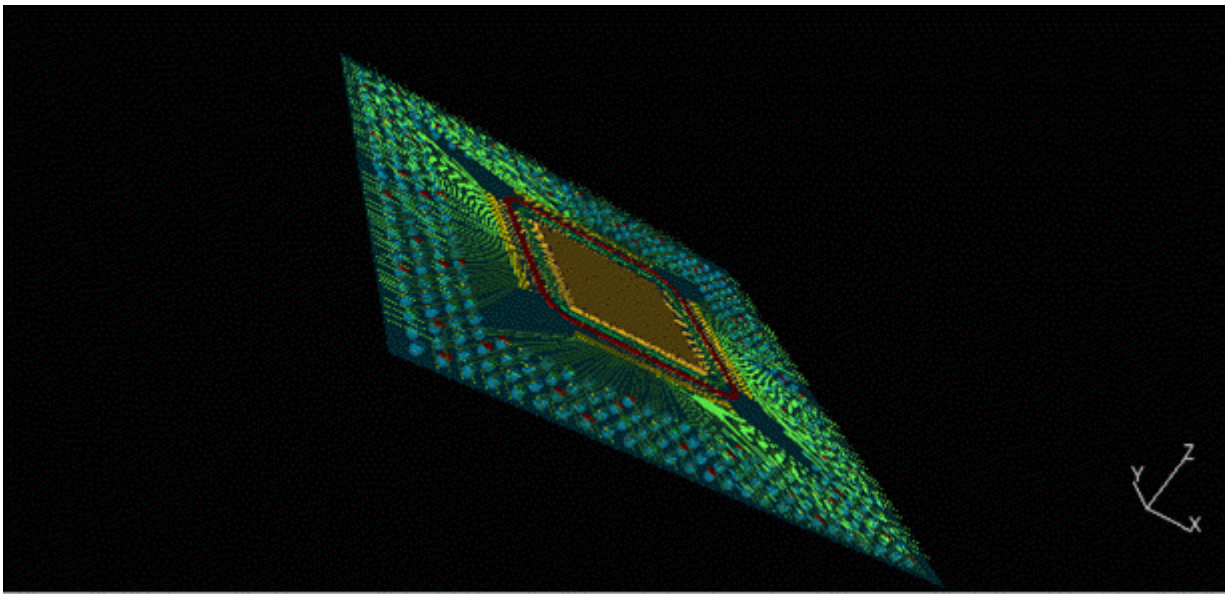
1. Open the *ready4viewer* database from *Module_4*.
2. For a view of the package in 3D, choose *View -– 3D Model*.



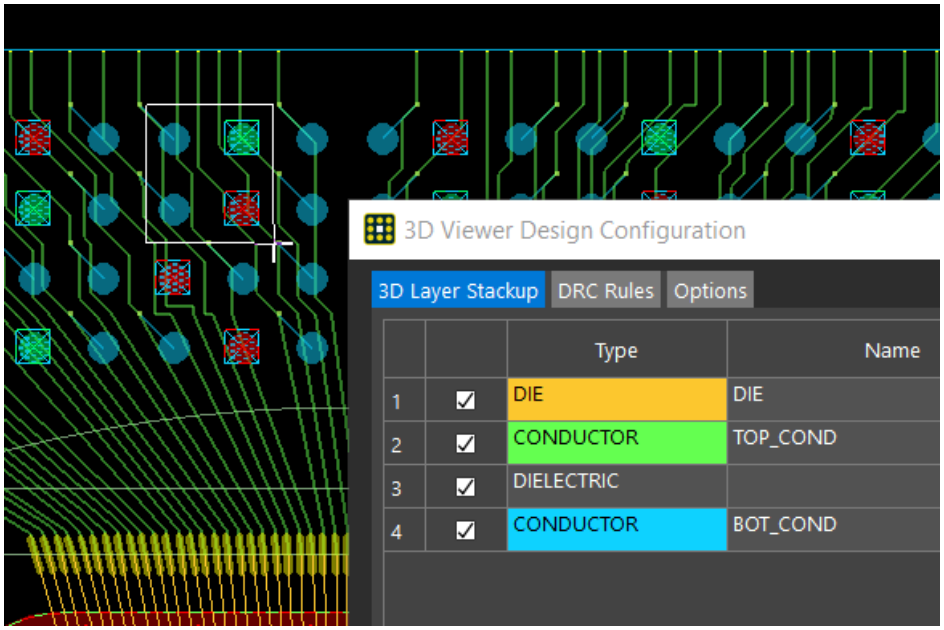
In this dialog, you can choose the layers in the stackup.

3. Click *View* in the 3D Viewer Design Configuration window.

   Once in the viewer, click and hold to rotate and right-click to zoom in and out. Use the default, which is the entire package (Figure 5-24).

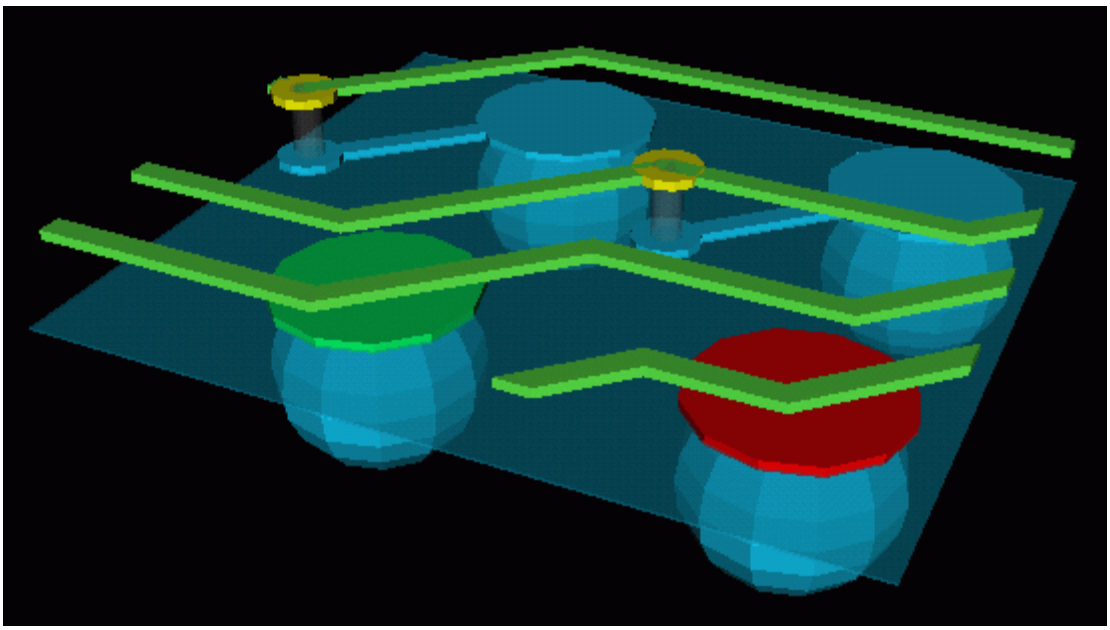


4. To view only selected objects, choose *View – 3D Model* and then select an area by windowing around the objects in the design.



You can select to view a window, design, or net from the *Trim to* list in the *Options* panel of the 3D Viewer Design Configuration window. For a net, you can also specify a clearance area. By default, selection is set to *Window*.

5. Click *View* in the 3D Viewer Design Configuration dialog box to view the selected area.



Multiple Profiles in Cadence 3D Design Viewer with DRC

1. To see an example of multiple profiles in the Cadence 3D Design Viewer, open the *ready4viewer_multi_profile* database from Module_4.

   You do not have to save the previous database.

2. Choose *View – 3D Model* from the menu.

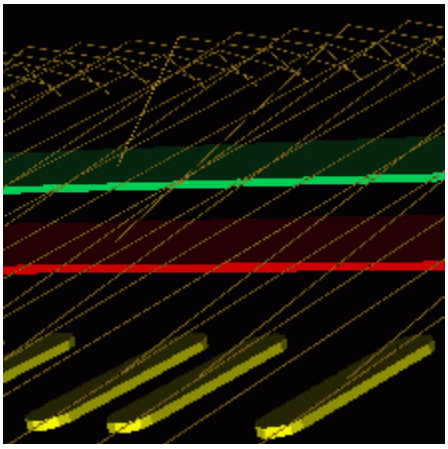
3. Click *View*.

4. In 3D Design Viewer, choose *View – Layers*.

   Profile 1 and Profile 2 have been modified using the *Wire Profile Editor* in APD so you can easily see the difference.

   Profiles are a must in a stacked-die design. The electrical tools (loop group height) also take this into account when building models for simulation. You will extract the profiles and the *TOP_COND* layers to view them in the 3D Design Viewer.

5. Uncheck the *All* column of the *All LAYERS* row. Then, click under *All* for the rows for PROFILE1, PROFILE2, and TOP_COND to select them.



6. Click under *All* for the ALL LAYERS row to display all the layers. Click *Close*.
7. To add DRC rules to your design, from the *Cadence 3D Design Viewer* window, choose *DRC – Rules*.
8. In the DRC Rules dialog box, click *Add Rule*.

   You can use the defaults or make your own rules. Define the marker type, size and color.
9. Click *Check Rules* to run DRC or re-run it after you make changes.

   These DRC rules are outside the APD constraint system.