Cadence 3D Design Viewer User Guide

Product Version 23.1
September 2023

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

Contents

Preface

This preface discusses the following:

Important

3D Canvas is an unsupported prototype as of release 17.4-2019. Allegro 3D Canvas, accessible from View – 3D Model, is recommended instead. Allegro 3D Canvas is available with *Allegro Package Designer+* and *Allegro Package Designer L*.

About This Guide

The Purpose of this Guide

This user guide provides detailed information about how to use Cadence 3D Design Viewer. It also explains how 3D Viewer works with your product. This guide contains suggested workflow descriptions, procedures, and command descriptions.

The Target Audience of this Guide

This user guide is intended to assist users who are working with 3D Viewer for the first time, or who work with 3D Viewer infrequently and need to refresh their knowledge of the tool.

This guide may also prove useful to experienced users who want to update their knowledge about the latest features and functionality provided by 3D Viewer.

How to Use This Guide

The purpose of this guide is to:

- Explain recommended workflows for using 3D Viewer.
- Provide detailed procedures that you should follow in order to use 3D Viewer effectively.
- Describe the 3D Viewer environment and command set.

Chapter Descriptions

This guide consists of five chapters and two appendices:

- Chapter 1

  Explains what 3D Viewer is. It also explains the recommended main workflow and secondary workflows that you should follow when working with 3D Viewer.

- Chapter 2

  Explains the user interface, how to use the mouse, and how to work with layers.

- Chapter 3

  Explains how to load and save data, and how to view and analyze 3D models.

- Chapter 4

  Explains how to work with the optional Design Rule Checking feature.

- Chapter 5

  Explains how 3D Viewer interfaces with other Allegro Package Designer+ (APD+) and how to exchange data between tools.

- Appendix 1 - 3Di Interchange Format v1.7

  This appendix provides a detailed description of the 3Di Interchange file format.

- Appendix 2 - Troubleshooting Guide

  This appendix provides troubleshooting tips for common problems, and technical support resources.

- Glossary

  Provides definitions for technical terms and acronyms found throughout this user guide.

Typographic and Syntax Conventions

This list describes the syntax conventions used in this guide.

`literal (LITERAL)`	Nonitalic or (UPPERCASE) words indicate key words that you must enter literally. These keywords represent command (function, routine) or option names.
`argument`	Words in italics indicate user-defined arguments for which you must substitute a value.
|	Vertical bars (OR-bars) separate possible choices for a single argument. They take precedence over any other character. For example, `command` argument | argument
[ ]	Brackets denote optional arguments. When used with OR-bars, they enclose a list of choices. You can choose one argument from the list.
{ }	Braces are used with OR-bars and enclose a list of choices. You must choose one argument from the list.
...	Three dots (...) indicate that you can repeat the previous argument. If they are used with brackets, you can specify zero or more arguments. If they are used without brackets, you must specify at least one argument, but you can specify more. *`argument`*`...: specify at least one argument, but more are possible` `[`*`argument`*`]...: you can specify zero or more arguments`
,...	A comma and three dots together indicate that if you specify more than one argument, you must separate those arguments by commas.
`Courier font`	Indicates command line examples.

1

Workflow Descriptions

This chapter discusses the following:

Important

3D Canvas is an unsupported prototype as of release 17.4-2019. Allegro 3D Canvas, accessible from View – 3D Model, is recommended instead. Allegro 3D Canvas is available with *Allegro Package Designer+* and *Allegro Package Designer L*.

About Cadence 3D Design Viewer

Cadence 3D Design Viewer is a simple but useful three-dimensional viewer that enables IC package designers to visualize and analyze their designs in three dimensions. 3D Viewer is an option that is integrated with your layout tool, enabling design engineers to generate, modify, and exchange design views quickly and easily.

Important

3D Viewer uses Visualization ToolKit© (VTK), a software system that generates 3D computer graphics. VTK requires the use of graphics cards which support OpenGL. Your graphics system should support OpenGL and have at least 64MB of dedicated (not shared) video memory.

3D Viewer allows you to control the orientation of the 3D model, specify color assignments, and control which objects to display. In addition to visualization, 3D Viewer allows you to modify or create new wire profiles, parameters, and groups.

3D Viewer provides additional features that include Markup (an easy way to make comments directly on the screen display) and 3D DRC (design rule checking in three dimensions).

- The Markup feature allows you to annotate the displayed image and export industry-standard (`.jpg`) images with notes and diagrams.
- The 3D DRC engine checks spacing rules in full three dimensions for wires, metal, and component bodies. Errors in 3D are highlighted by small 3D markers. You can click on a marker to display the nature of the error and highlight the geometries that triggered the error. 3D DRC is an extra-cost option.

The input to 3D Viewer is the 3Di file format. (The 3Di file is extracted automatically from your layout tool.) This format is a simple ASCII description of basic geometries such as polygons, wires, pads, and color information. Each geometry is tagged with attributes that add "intelligence" to the 3Di database. These attributes enable 3D Viewer to perform specific functions, such as highlighting complete nets. When you display a 3Di file, 3D Viewer first generates three-dimensional facets from the more general model description. It then loads the data and allows you to pan, zoom, and orbit the 3D model (see Loading Data).

For a detailed specification of the 3Di file format, see Appendix 1 - 3Di Interchange Format v1.7.

3D Viewer Use Model with APD+

To be able to use 3D Viewer from Allegro Package Designer+ (available with L and XL only), you must select the *3D Viewer* option. This option is available in Cadence Product Choices when you start APD+ or when you change editor (*File – Change Editor*).

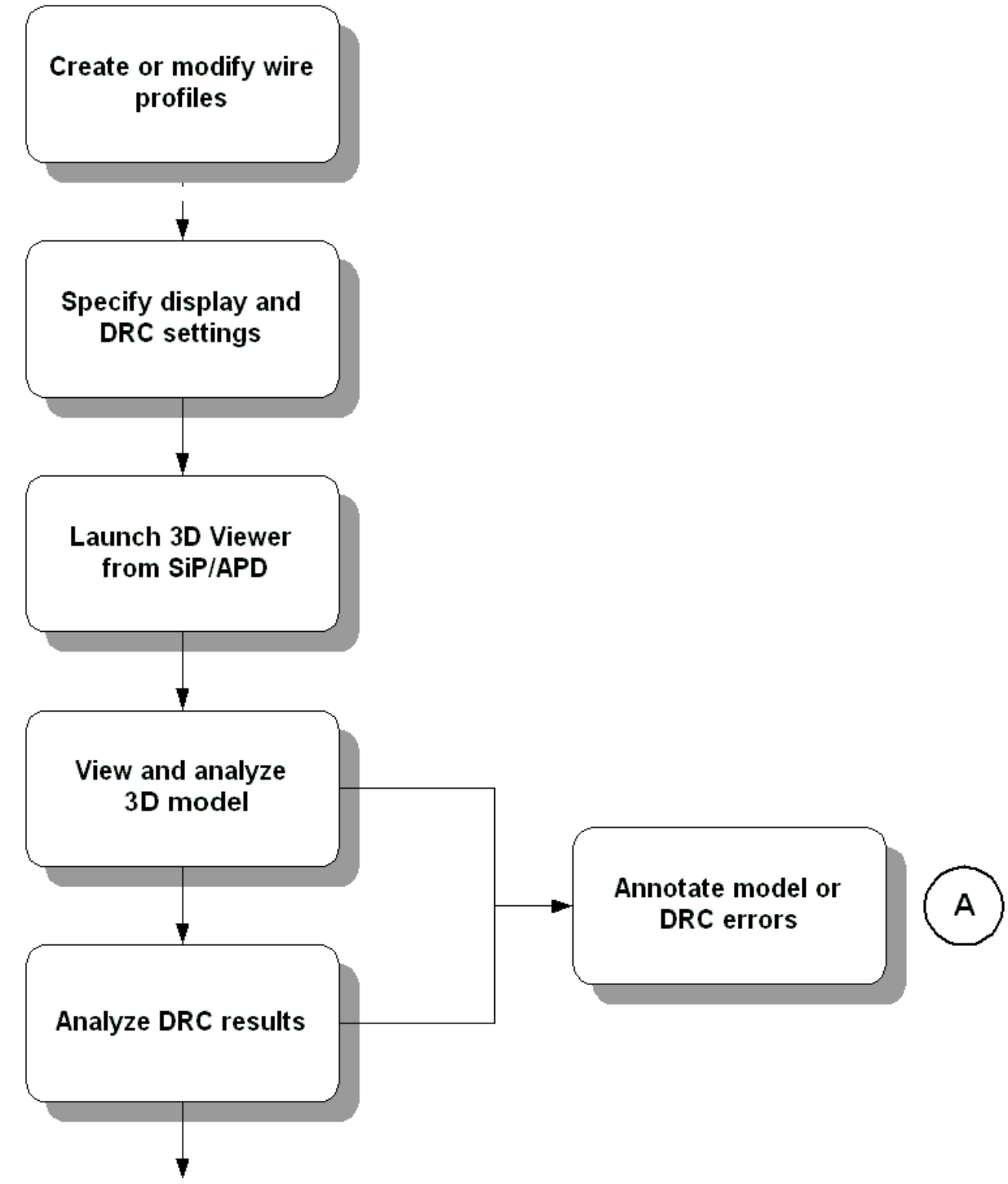
The *3D Viewer* option is not available with Allegro Package Designer+ license because APD+ includes the appropriate viewer license already. This is consistent with all other product options and menu entries.

- You can release 3D Viewer Option license without exiting the tool by performing a change editor.
- When you choose *File – Change Editor* in APD+ and pick a different tool, if the 3D Viewer window is open, it will be closed.

Main Workflow

The following diagram shows the main recommended workflow that you should follow when using 3D Viewer. By following this workflow, you achieve the best results in viewing and analyzing 3D models.

.

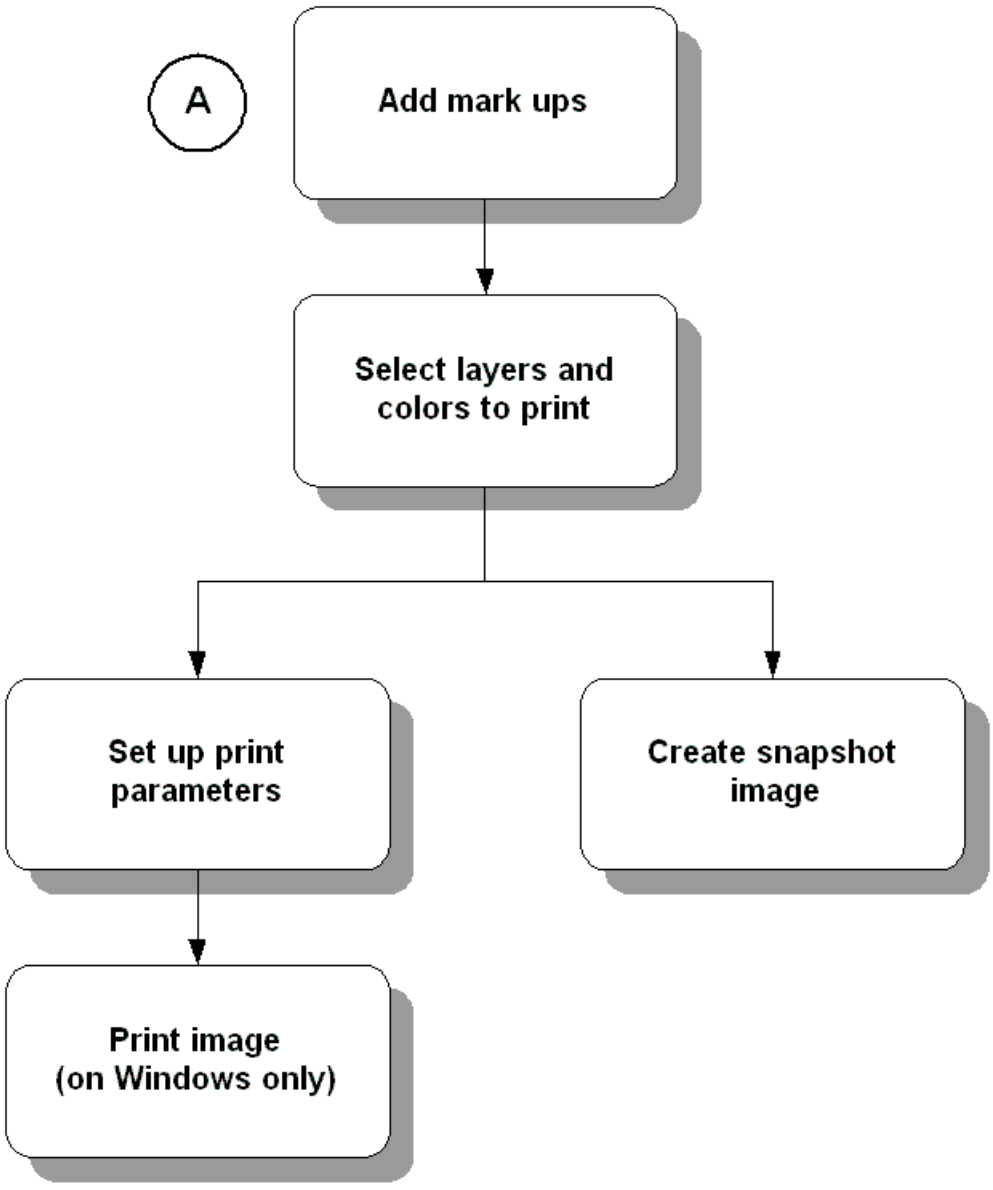


Secondary Workflow

The diagram below shows the additional, secondary workflow that you may want to follow when annotating the 3D model or the 3D DRC errors.

Note: Printing is only supported on Windows platforms.

Printing Annotations



2

Understanding the 3D Viewer Environment

This chapter discusses the following:

Understanding the APD+ Interface to 3D Viewer

Important

3D Canvas is an unsupported prototype as of release 17.4-2019. Allegro 3D Canvas, accessible from View – 3D Model, is recommended instead. Allegro 3D Canvas is available with *Allegro Package Designer+* and *Allegro Package Designer L*.

In your layout tool, when you choose *3D Model* from the *View* menu, the 3D Viewer Design Configuration dialog box appears (see Figure 2-1). Because the viewer uses a lot of memory, choose only the layers, objects, or area that you want shown in the viewer by checking the appropriate boxes in the 3D Viewer Design Configuration dialog box. This action can give you better performance. It also makes it easier for you to navigate through the viewer, since the tool suppresses the secondary objects in which you are not interested from the view.

Tip

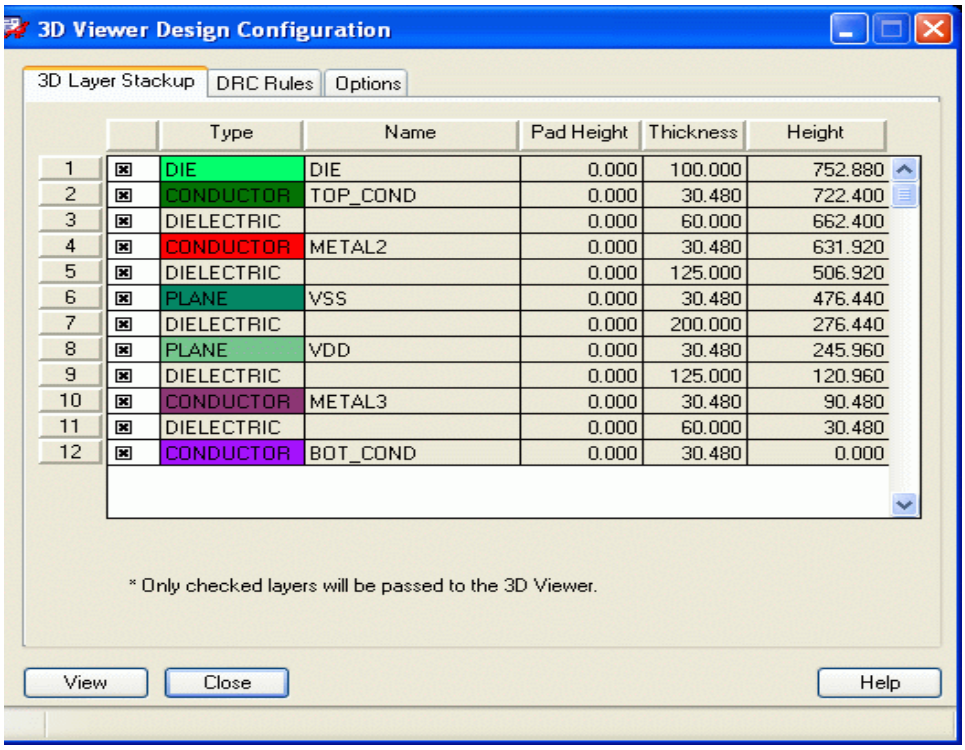
You can launch 3D Viewer directly without displaying the configuration dialog box by selecting items in the design and then choosing *3D Model* from the *View* menu. 3D Viewer will show only the selected items.

Note: Select the 3D Viewer option (available only with L and XL licenses) in Cadence Product Choice or when changing editor (*File – Change Editor*) to be able to use 3D Viewer with APD+. The option is not available with the *Allegro Package Designer+* license because it contains the license already. For more information, refer to 3D Viewer Use Model with APD+ on page 10.

This interface to 3D Viewer contains two default tabs (*3D Layer Stackup* (see Figure 2-2) and *Options (*see Figure 2-4), and a third tab (*DRC Rules* (see Figure 2-3)) that is available in some tools for an additional cost (see 3D Design Rule Checks). Use this dialog box to set up the viewing and checking parameters that control how 3D Viewer displays the 3D model.

To be able to use 3D Viewer from APD+ (available with L and XL only), you must select he 3D Viewer option in Cadence Product Choices or change Editor (File – Change Editor). For more information, see 3D Viewer Use Model with APD+ on page 10.

Figure 2-1 3D Viewer Design Configuration



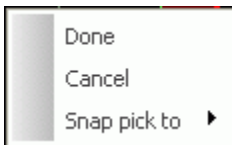
The following three buttons are available regardless of the tab that is currently active.

This button...	Does this...
View	Generates the 3D Viewer file based on all current options and the current design file in the layout tool.

Close	Closes the 3D Viewer Design Configuration dialog box and returns to the layout tool.
Help	Displays context-sensitive Help information.

Right-click Controls

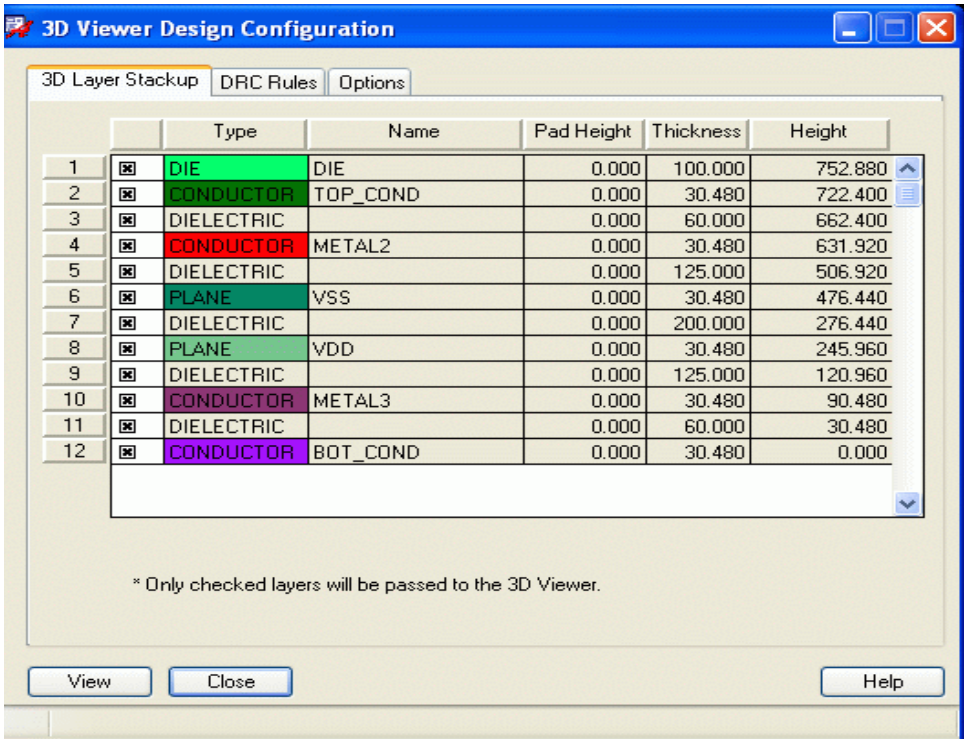
When you click the right mouse button in the editing window of your layout tool while the 3D Viewer Design Configuration dialog box is active, a pop-up menu appears with the following command options.



Right-click Commands

This command...	Does this...
Done	Closes the dialog box after launching 3D Viewer. This functions the same way the *View* command works.
Cancel	Closes the dialog box without launching 3D Viewer. This works the same way the *Close* command works.
Snap pick to	Selects region or object depending on the option chosen. Displays a menu to select options from as shown:

Figure 2-2 3D Layer Stackup Tab



3D Layer Stackup

This group box contains five columns of information for each named layer in the design: *Type*, *Name*, *Pad Height*, *Thickness*, and *Height* from the bottom layer of the design. For default layers in the design, you cannot edit these fields.

Note: Die-stack layers are not listed, since wires are modeled based on wire profiles. In APD+, you add the dies manually. In APD+, the die information appears automatically, populated from the Die-stack Editor. Wire bond profiles are not listed and are based on the configurations set up in the wire profile editor and saved in the database.

The first column of check boxes allows you to specify whether to have the 3D Viewer display and process particular layers. If you enable the check box for a layer, that layer is passed to 3D Viewer.

For die type layers, you may enter the height of the pads (or depth of passivation openings for wirebond dies) in APD+. This allows 3D Viewer to model these components more accurately.

Substrate layer types display with their backgrounds colored the same as the conductor traces on that layer. (The color is derived from the color/visibility parameters in your layout tool.)

You can add spacer and die ordering information in APD+ to obtain an accurate 3D representation of the design. By right-clicking on a row number for a conductor layer, you can perform one of the following commands to add, delete, or reorder the layer stackup.

Layer Stackup Commands

This command...	Does this...
Add	Adds a new layer in the design. Choose either *SPACER* or *DIE* as the layer type. If you choose *SPACER*, you can enter values for *Name* and *Thickness*. If you choose *DIE*, you must choose from one of the die reference designators in the *Name* column, and then you can enter a *Thickness* value.
Delete	Deletes the layer on the selected row. *Delete* applies only to die and spacer layers that you have previously added to the stackup using the *Add* command. You cannot delete layers on the regular package substrate layers.
Up	Moves a die or spacer layer up one in the ordering. *Up* only applies to die and spacer layers that you have previously added to the stackup using the *Add* command. You cannot move layers on the regular package substrate layers.
Down	Moves a die or spacer layer down one in the ordering. *Down* applies only to die and spacer layers that you have previously added to the stackup using the *Add* command. You cannot move layers on the regular package substrate layers.

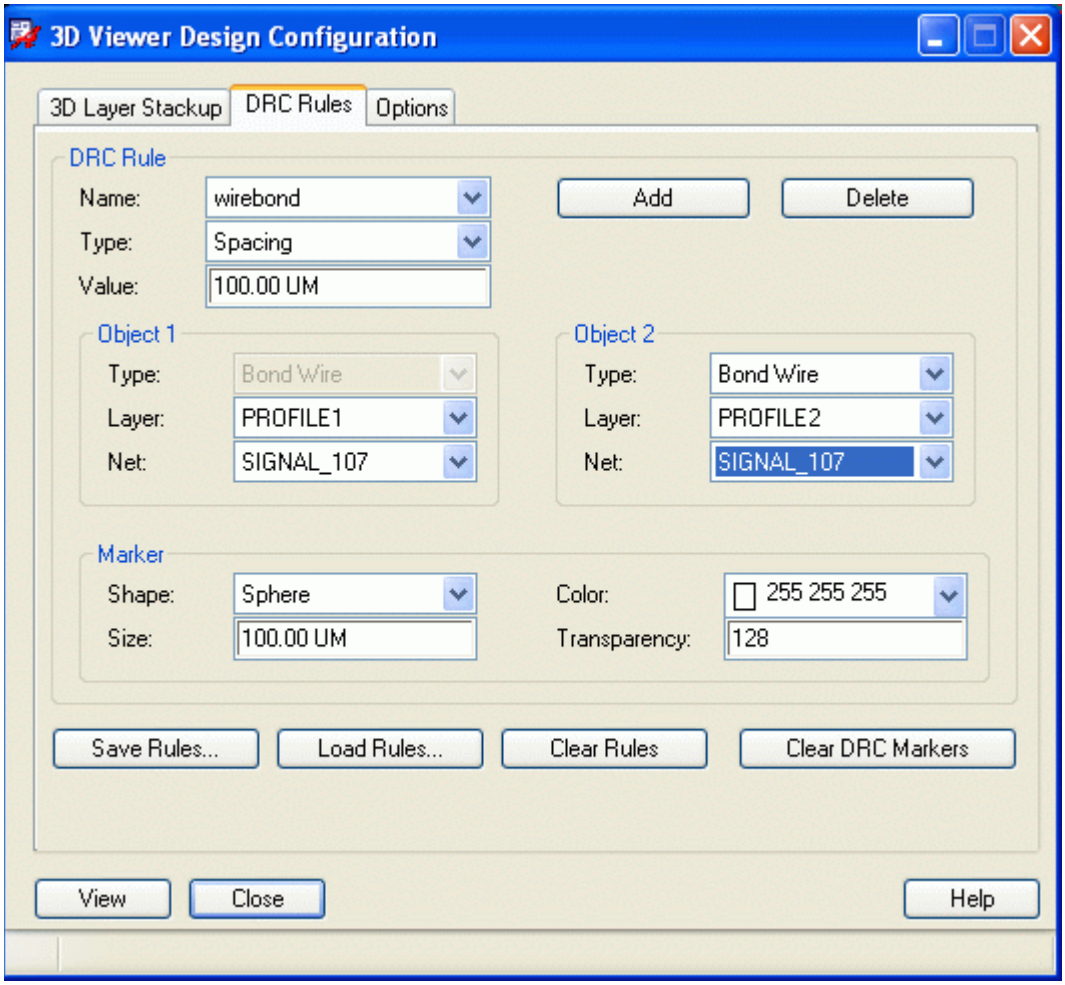
Figure 2-3 DRC Rules Tab

Important

The *DRC Rules* tab is an optional tab and appears only if your license supports the 3D DRC feature.

Tip

See Setting up 3D DRC Parameters for more details on defining DRC Rules.

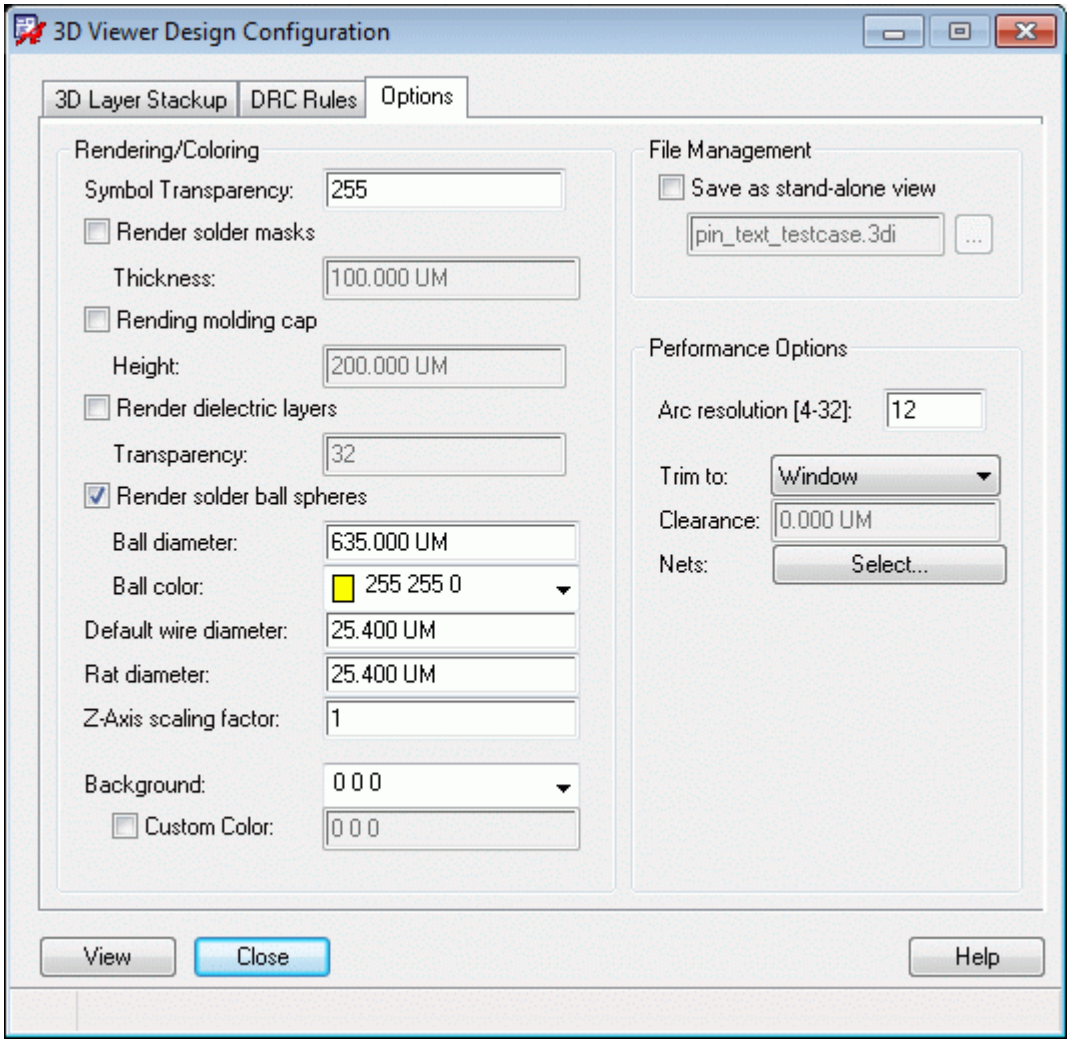


This command...	Does this...
Name	Specifies the name of the DRC rule to modify. You can enter any name. The drop-down list shows all of the rules that are currently defined. Since the tool now supports backannotation of DRCs to the main `mcm` design, this name is used in the DRC markers that are created. Be sure that you specify a meaningful name.

This command...	Does this...
Add	Adds a new DRC constraint to the current rule deck. The default rule name that is assigned automatically is “n”, where n is the number of rules in the design plus 1. For example, the first rule is named “1”, the second is named “2”.
Delete	Deletes the current rule from the rule deck.
Type	Specifies the type of rule to define. Only one type of rule is supported: `Spacing`.
Value	Specifies the value of the rule. For spacing rules, this is the clearance required between the two objects specified by the Object filters. The default value is 100 microns.
Object Type	Specifies the type of object to compare. The default object is *Bond Wire*. You can choose the following types from the drop-down list: *Bond Wire*, *Bond Finger*, *Shape*, *Cline*, and *Symbol*.
Object Layer	Specifies the layer on which to compute the object’s extents. The default is *<ANY>*, meaning that all layers with geometries on them are used. You can choose any layer from the drop-down list. The drop-down list also contains an entry for each bond wire profile since these are pseudo-layers representing the heights of the wires.
Object Net	Specifies the net to filter, if you apply net filtering for this rule. You can choose any net from the drop-down list.
DRC Marker Shape	Specifies the shape (*Sphere*, *Cube*) for the DRC markers. The default is *Sphere*.
DRC Marker Size	Specifies the size for the DRC markers. The default value is the value of DRC spacing (*Value* field above) for the active DRC rule. It is recommended that you keep the *DRC Marker Size* in the 3D Viewer window the same size as the spacing value. This way, when you look at a violation in the display, you can see the extent of the violation by how much the sphere shape overlaps the two items that are too close together. If the two objects barely touch the sphere, you are close to meeting the required value. If they overlap significantly, the two objects are too close together.

This command...	Does this...
DRC Marker Color	Specifies the color for the DRC markers. The default is *255 255 255* (White). (White provides maximum contrast against a black background.) You can choose a predefined color from the drop-down list.
DRC Marker Transparency	Specifies the transparency of the DRC markers. This value ranges from *0* (invisible) to *255* (completely opaque). The default value is *128*.
Save Rules	Saves the current set of rules for reuse in this or other designs. This saves to an XML format file (see Understanding the 3D DRC Rules File), with the default file name `<your_design_name>.xml`.
Load Rules	Loads an existing rule deck from an XML file stored on disk. These rules are loaded on top of your existing rules, allowing for hierarchical rule sets.
Clear Rules	Removes all rules currently specified in the rule deck. Use this command prior to reading in a completely new set of rules from a rules file.
Clear DRC Markers	Clears all the DRC markers in the design.

Figure 2-4 Options Tab



This command...	Does this...
Rendering/coloring	
Symbol transparency	Specifies the transparency value for symbol outlines. This value ranges from *0* (invisible) to *255* (completely opaque). The default value is 255.

This command...	Does this...
Render solder masks	Controls whether the tool displays the solder mask opening.
Thickness	Specifies the thickness of the solder mask.
Rending molding cap	Controls whether the tool displays the package molding cap.
Height	Specifies the height of the molding cap above the solder mask layer, or the top conductor layer if soldermask layer is not displayed.
Render dielectric layers	Controls whether the tool displays the dielectric layers. **Note:** You must select this option to see the cavity outlines as holes in the substrate.
Transparency	Specifies the transparency of the displayed dielectric layers.
Render solder ball spheres	Controls whether the solder balls of the package are rendered. If so, then the specified ball diameter is used. The default setting is enabled.
Ball diameter	Specifies the diameter to use for the BGA balls. You can specify this only if you have enabled *Render solder ball spheres*. The default value is the diameter of the padstack representing the balls. (Note that balls are not actually spherical.)
Ball color	Specifies the color of the solder balls. This defaults to the color specified for pins on the bottom conductor layer.
Default wire diameter	Specifies the diameter for rendering bond wires that have a width of `0` in the database. This defaults to the line width constraint on the top diestack layer. If no non-zero bonding wire diameter is recorded in the database, this value defaults to `1.0 mil`.
Rat diameter	Specifies the diameter of the rat line. Rats are drawn in the standard rats color on a separate layer.

This command...	Does this...
Z-Axis scaling factor	Specifies the Z-axis multiplier used to increase the displayed thickness of layers in the design. The default value is `1` (no scaling). **Note:** This is for display purposes only and does not effect DRC measurements.
Background	Specifies the color assignment for the background of the 3D Viewer display. The default setting is `0 0 0` (Black). You can choose a predefined color from the drop-down list, which shows the defined colors for the active drawing, plus black and white.
Custom Color	If enabled, this specifies a custom RGB color value to use for the background instead of the predefined colors in the *Background* drop-down list. The default value is *0 0 0* (Black).
File Management	
Save as stand-alone view	When enabled, you can save the view of the design in 3D Viewer under a custom file name. By default, the file name for the `.3di` file matches the file name of the current design: `<design_name>.3di`. If you enable this option, you can specify distinct file names for multiple static views of the design. You can enter an absolute or a relative file name for the file. If the file name already exists, you are warned and asked whether to overwrite the existing file. Use the browse button (...) to locate folders and directories.
Performance Options	
Arc resolution	Specifies the resolution (number of vertices) for modeling an arc segment. This value can range from `4-32` points. The default value is `12`.
Trim to	Controls whether the tool generates a cross-section type view into a slice of the design, based on the selected design, window, or net.

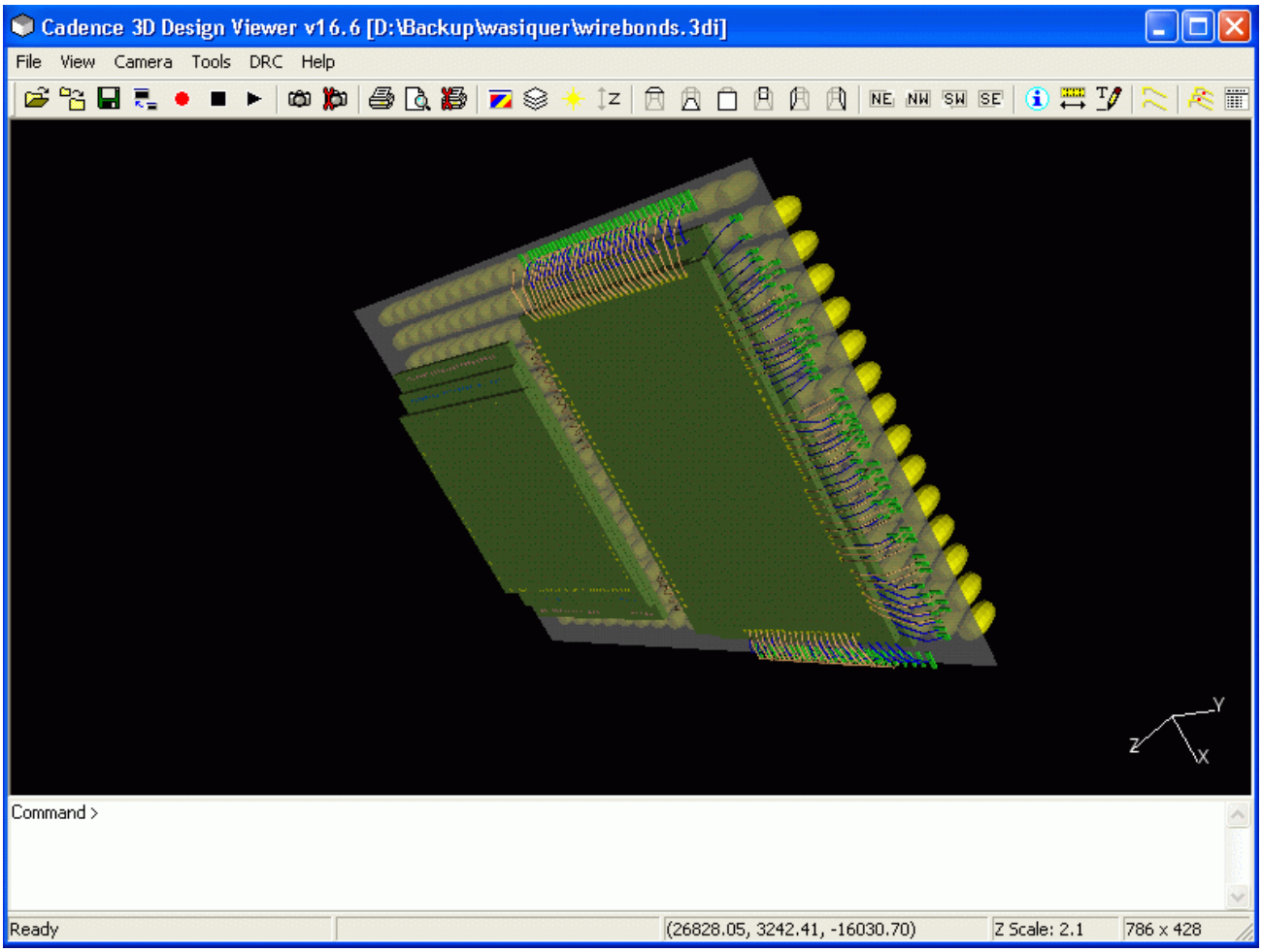
This command...	Does this...
Nets	Select to view only nets from a selected window region. Available when you select *Window* in *Trim to*. After windowing around a region, click *Select* and then select the nets within that region. When you click *Select*, only *Nets* is selected in the Find filter.
Clearance	Specifies the 3-dimensional distance that is to be displayed in all direction from a selected net. Available only when *Net* is selected in Trim to.

Understanding the User Interface

The 3D Viewer environment consists of menus, toolbars, a View window, a status bar, and a command window (shown in Figure 2-5).

Note: You can also enter commands in the command window.

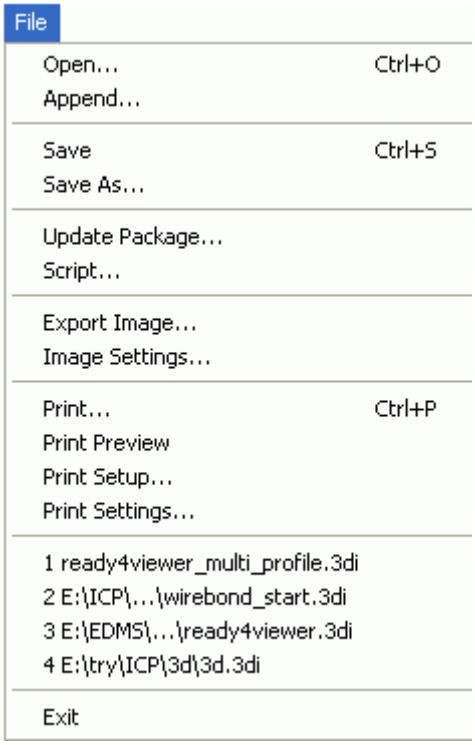
Figure 2-5 3D Viewer Window



Menus

The following sections describe the command menus in the 3D Viewer window.

File Menu

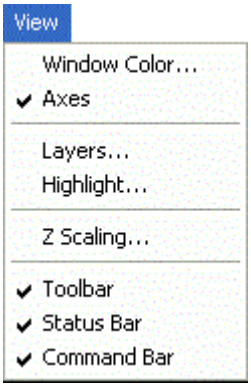


File Menu Commands

This command...	Does this...
Open	Opens a `.3di` file.
Append	Lets you append 3Di files on top of the 3Di file shown in the 3D Viewer Window. The appended files represent other pieces of the design. For example, in a package-on-package design, you might append the top package on top of the bottom package. Then you could run DRC checks between the wires of the die on the bottom package with the bottom of the top package. Or you might place more accurate models of the discrete components on top of the package. See Appending a File on page 46 for additional information.
Save	Saves the current design in 3Di format.

Save As	Saves the current design in 3Di format under a different name that you specify.
Update Package	Saves the changes to the base design that you made in the 3D Viewer to your layout tool. It does not save the appended files.
Script	Lets you record and play back application operations which originate from the menus or the command bar.
Export Image	Exports the current image in the View window as a graphic file (`.jpg`, `.tif`, `.bmp`).
Image Settings	Lets you specify the image size and scaling factor for export.
Print	Prints the current image in the View window.
Print Preview	Previews the current image to show how it will print.
Print Setup	Allows you to set up the printer parameters.
Print Settings	Lets you define the print quality (dpi) and enable or disable printing of the background window color.
Exit	Exits the program.

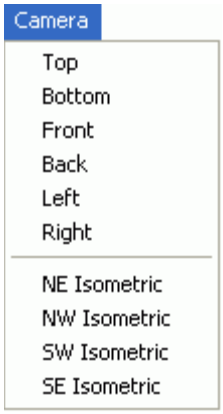
View Menu



View Menu Commands

This command...	Does this...
Window Color	Lets you change the background color of the View window.
Axes	Toggles between displaying or not displaying the orientation marker (3D axes).
Layers	Lets you choose which layers you want to display. You can also change the colors of the layers. **Note:** If you change the color, all objects on the layer are assigned the same color even if different objects on the same layer originally had different color assignments.
Highlight	Lets you specify the color of highlighted objects and the dimming factor for non-highlighted objects.
Z Scaling	Lets you specify the Z-axis scaling factor for the display.
Toolbar	Lets you choose which of the toolbars that you want to display.
Status Bar	Toggles between displaying or not displaying the status bar.
Command Bar	Displays the command window so that you can enter a command instead of using the menu bar at the top of the window.

Camera Menu



Camera Menu Commands

This command...	Does this...
Top	Sets the display to view from the top.
Bottom	Sets the display to view from the bottom.
Front	Sets the display to view from the front.
Back	Sets the display to view from the back.
Left	Sets the display to view from the left.
Right	Sets the display to view from the right.
NE Isometric	Sets the display to view from a northeast-oriented isometric angle.
NW Isometric	Sets the display to view from a northwest-oriented isometric angle.
SW Isometric	Sets the display to view from a southwest-oriented isometric angle.
SE Isometric	Sets the display to view from a southeast-oriented isometric angle.

Tools Menu

Tools Menu Commands

This command...	Does this...
Info	Provides detailed information about a particular object that you select in the design. The selected object is highlighted and the rest of the design is dimmed.
Distance	Calculates the distance between two objects that you select in the design and displays the measurement in the current design units.
Mark Up	Lets you set up text font, size, color, and symbol types for mark ups.
Wire Profiles	Displays the Wire Profiles dialog box that contains complete information about the wire profiles. You can edit the various parameters to modify the wire profiles. Any changes that you make are reflected in the View window once you apply them.
Frame Rate Test	Executes a pre-programmed performance test and reports scene rendering and frame rate information. This is useful for evaluating graphic card performance.

DRC Menu

DRC Menu Commands

This command...	Does this...
Rules	Lets you set up DRC rules and rerun the 3D DRC check process. With this command, you can also modify the values of rules that were configured when the viewer was first launched.
Report	Generates and displays a report file containing any DRC errors in the design.

Help Menu

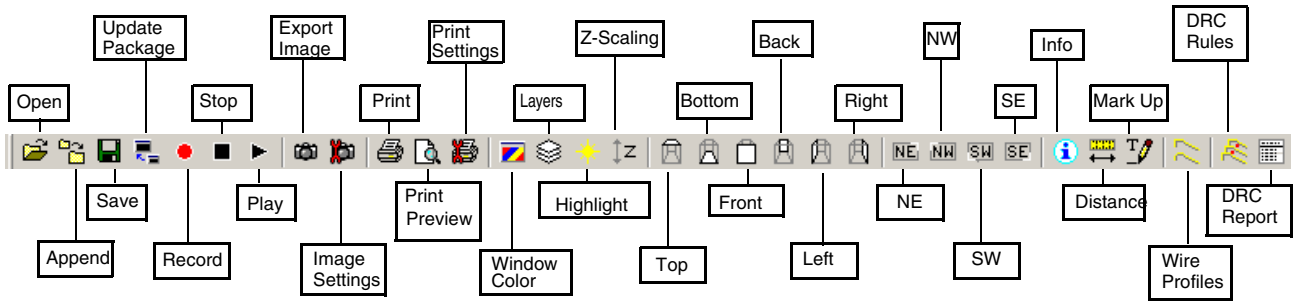
Help Menu Commands

This command...	Does this...
Cadence 3D Design Viewer Help	Provides access to the online Help information about 3D Viewer.
About Cadence 3D Design Viewer	Provides product version information about 3D Viewer.

Toolbar

The following diagram identifies the buttons on the toolbar in the 3D Viewer user interface. The toolbar buttons execute the same commands that are found in the corresponding menus

.



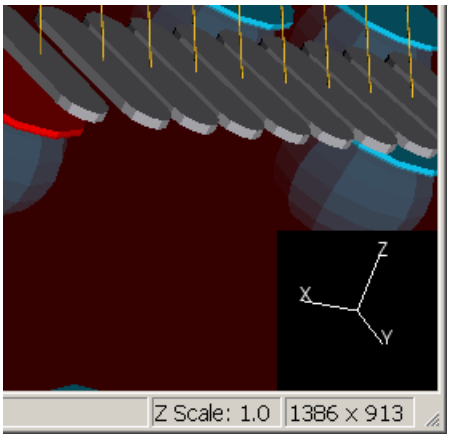
Tip

If you click a preset camera view button once (for example, *Top*), the display changes to that view without changing the field of view. If you click the same view button a second time, the display zooms out to the extents of the design.

Using the Mouse

With a three-button mouse, you can easily control the viewing perspective of a model in the View window. You can interactively manipulate (rotate, pan, zoom) the viewpoint, or camera, of the model. The magnitude of the mouse motion is proportional to the camera motion. For example, small left button motions cause small changes in the rotation of the camera around its focal point. With a little practice, these motions become instinctive.

An orientation marker appears in the lower right corner of the View window. This orientation marker indicates the current spatial orientation of the model in space, showing the X, Y and Z axis lines. The orientation marker adjusts automatically as you manipulate the viewpoint.



Basic Mouse Button Controls

Figure 2-6 describes how to use the mouse buttons to control the viewing perspective.

Figure 2-6 Basic Mouse Button Controls for a Three-Button Mouse

Left Button (Orbit)

Hold the left button down. As you move the cursor, the design rotates or orbits around its center.

Ctrl + Left Button (Spin)

Hold down the `Ctrl` key while pressing the left mouse button. As you move the cursor, the design spins around its center. Unlike orbiting, spinning restricts the rotation of the object to a given plane.

Middle Button (Pan)

Hold the middle button down. As you move the cursor, the object pans left, right, up, or down according to the way the cursor moves.

Right Button (Zoom)

Hold the right button down. If you move the cursor up, you zoom in on the object. If you move the cursor down, you zoom out.

Special Mouse Button Controls

If you have a two-button mouse instead of a three-button mouse, you can still take advantage of the viewing controls, as described in Figure 2-7.

Figure 2-7 Basic Mouse Button Controls for a Two-Button Mouse

Ctrl + Left Button (Spin)

Hold down the `Ctrl` key while pressing the left mouse button. As you move the cursor, the design spins around its center. Unlike orbiting, spinning restricts the rotation of the object to a given plane.

Shift + Left Button (Pan)

Hold down the `Shift` key while pressing the left mouse button. As you move the cursor, the object pans left, right, up, or down according to the way the cursor moves.

Ctrl + Shift + Left Button (Zoom)

Hold down the `Ctrl` and `Shift` keys simultaneously while pressing the left mouse button. If you move the cursor up, you zoom in on the object. If you move the cursor down, you zoom out.

Working with Layers

In the Layer dialog box, you can turn the display of layers on and off individually. You can also change the color and transparency of layers.

You can assign a color and transparency to each layer, which greatly improves the ability to isolate the objects that you want to see against a complex background. For example, it is common practice to set the die body to a low transparency (50%, or alpha 128). This way you can see objects that would normally be hidden behind it (bond fingers and wire paths), but at the same time still see enough of the die body to know which wires belong to this die body.

Note: By default, the BGA outline is used as the substrate outline. If a design does not have a BGA, the substrate outline is based on the shapes on the outline layer in the design. As a result, you can customize the outline, if needed, and will see the outline in all situations.

The following are the default settings:

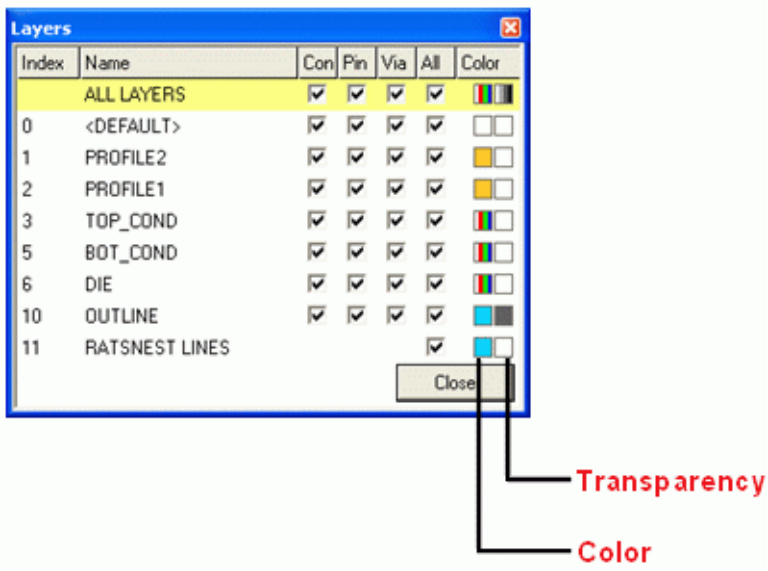
- Die bodies are Blue and 50% transparent.
- Discretes are Green and 50% transparent.
- BGA boundaries are Red and 50% transparent.

Changing the Display of Layers

To change the display of layers:

1. From the *View* menu in the 3D Viewer window, choose *Layers*.

   The Layers dialog box appears.



2. Check the boxes under the appropriate columns for each layer that you want to display, or check the *All Layers* box to display all layers in each column.

   The display changes according to the settings that you choose.

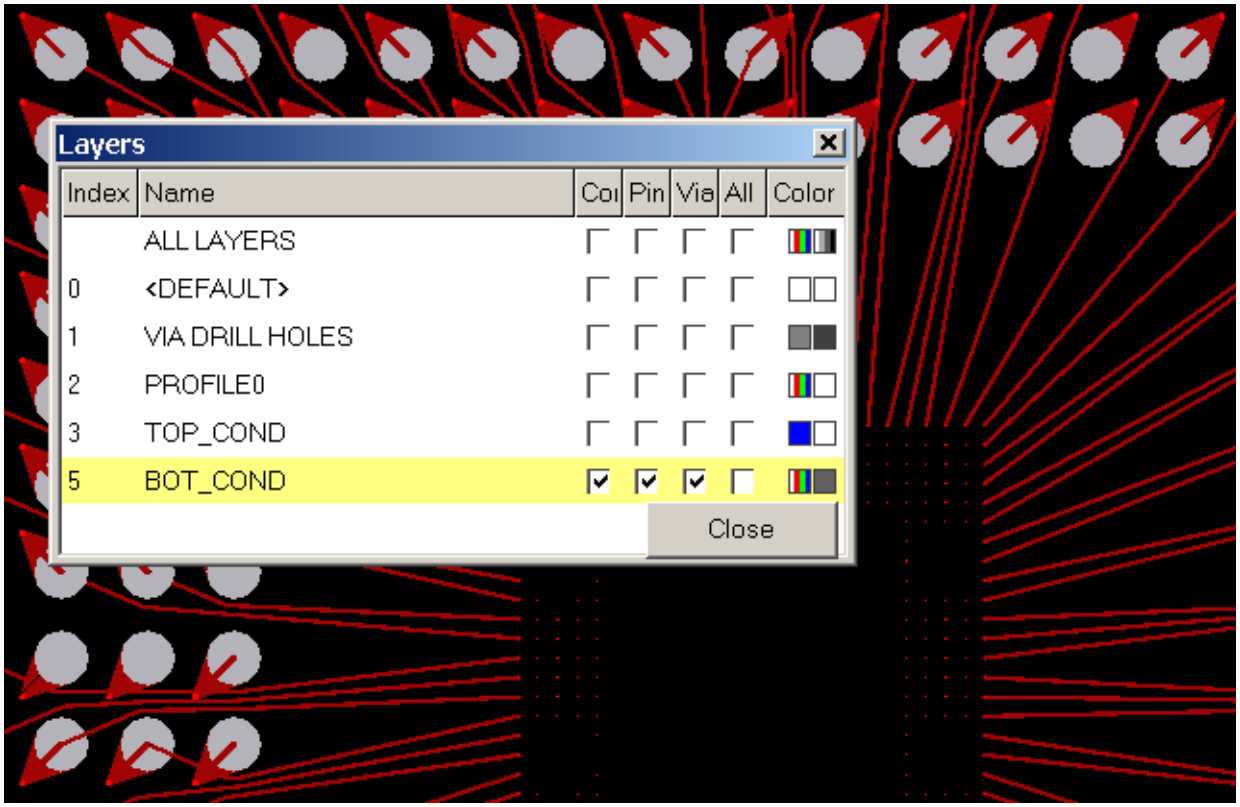
   Note: *Layer 0* contains all objects that are not assigned to a specific layer in the database, such as drill holes.

3. Click the color boxes under the *Color* column for a particular layer to change the color assignments for that layer. The first (left-hand) box indicates the color assignment. The second (right-hand) box indicates the transparency assignment.

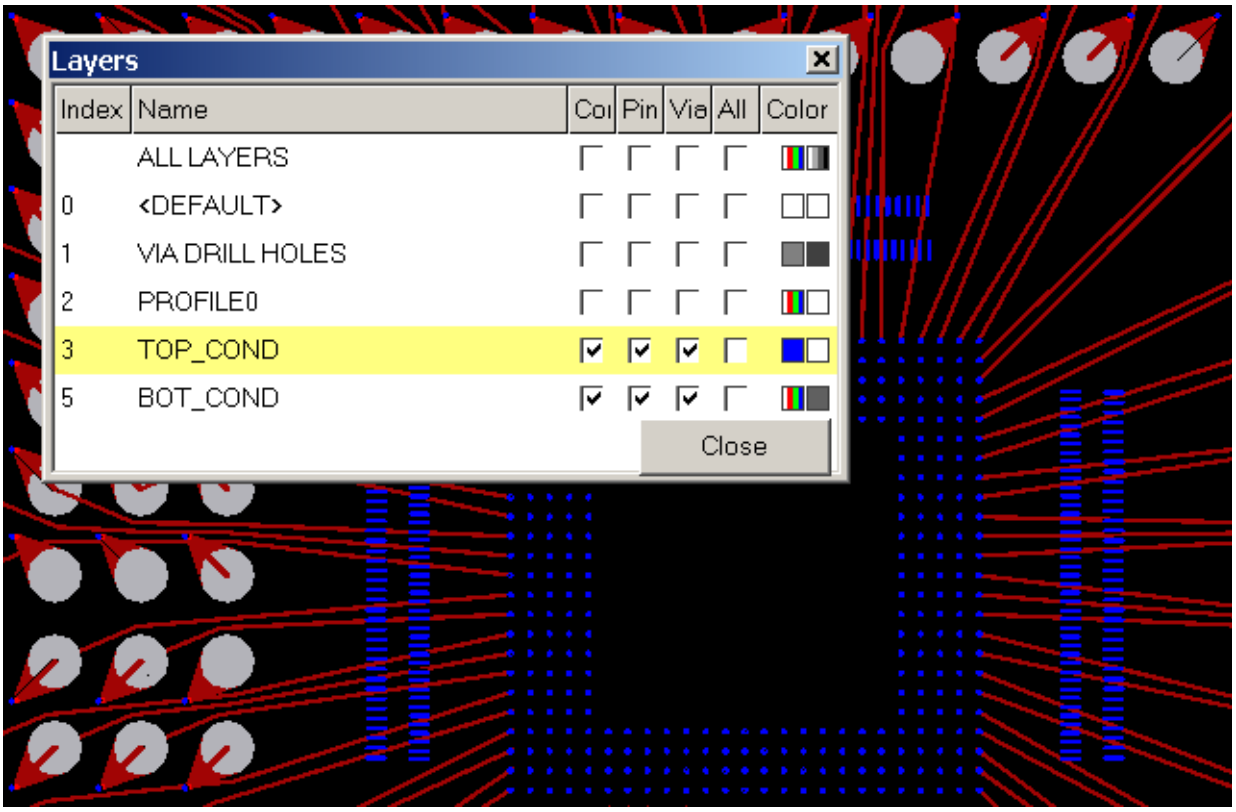
Caution

The color boxes are striped if different colors are assigned to different objects in the database that all reside on a common layer. If you change the color assignments in 3D Viewer so that all objects on a given layer have the same color, the color box will be solid. This overrides any permanent highlighting that was carried over from the layout view. This may not always reveal the detail that you need to see in the 3D model, so use caution when redefining layer colors.

In the following example, only the *Bottom* layer is visible, which displays the routes and pads on the bottom layer of the model.



In the next example, both the top and bottom layers are visible. This displays the conductor patterns for both the top and bottom layers of the model.



3

Working with 3D Viewer

This chapter discusses the following:

Loading Data

Cadence 3D Design Viewer accepts design data generated from IC package designs that are created using APD+. The design data is in the 3D Interchange (3Di) format and contains geometries, tables of properties and design rule checks. 3Di is a simple description of polygons in the X, Y plane that have height and thickness attributes. Artwork Conversion Software, Inc. developed this format as part of the effort to model IC packages and substrates in three dimensions. (For detailed specifications on the 3Di format, see Appendix 1 - 3Di Interchange Format v1.7.)

Important

3D Viewer requires the use of graphics cards that support OpenGL. Your graphics system should support OpenGL and have at least 64MB of dedicated (not shared) video memory.

Opening a Data File

When 3D Viewer loads a file, it uses the design data to generate sophisticated three-dimensional models, which are then displayed for inspection and analysis. The application that creates the 3Di file can attach as many or as few properties to each geometry as desired. The only property that 3D Viewer must have in order to create a reasonable display is a color property.

Typically, you launch 3D Viewer from APD+ to view 3D model information generated from the design database in your layout tool.

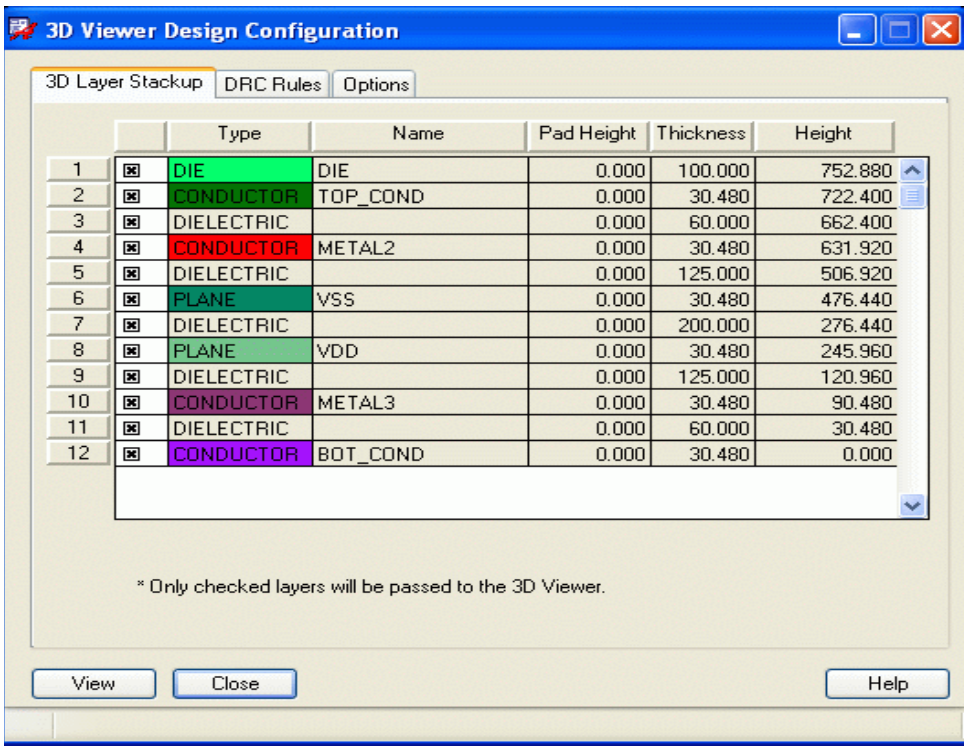
Note: Depending on the complexity of the design rule checks that you set up, there may be a long delay before 3D Viewer scans the data and displays the 3D model.

Launching the 3D Viewer from Your Layout Tool

To launch 3D Viewer from your layout tool:

1. From the *View* menu in APD+, choose *3D Model* or type `view 3d` in the layout tool command line and press `Enter`.

   The 3D Viewer Design Configuration dialog box appears.

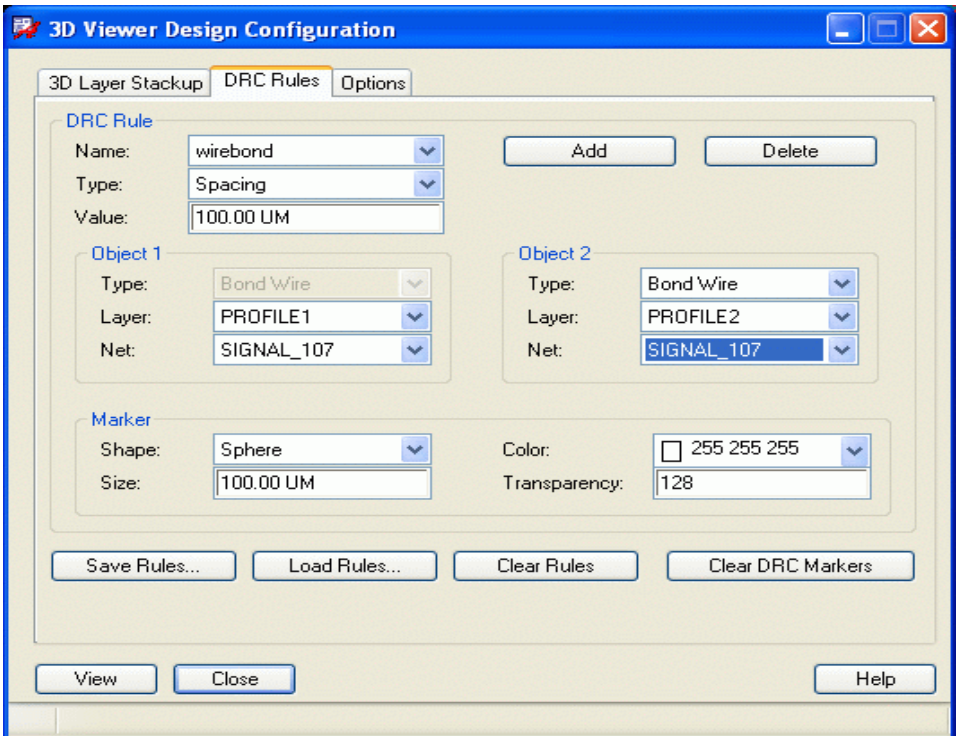


2. In the *3D Layer Stackup* tab, choose the layers that you want to display in 3D Viewer by checking the boxes next to the layer names. Edit the spreadsheet cells, or right-click on the row number of an existing layer to add a new layer.

Tip

See 3D Layer Stackup Tab for more details about working with the *3D Layer Stackup* tab.

3. Click the *DRC Rules* tab.



4. Under the *DRC Rule* group box, choose the rule that you want to apply from the *Name* drop-down list.

5. Click *Add* to define a new rule. Enter a name for the new rule and specify the DRC parameters in the *Object 1*, *Object 2* and *Marker* group boxes.

6. Perform any of the following other options in the *DRC Rules* tab:

   - ❑ Choose a rule and click *Delete* to remove a rule from the list.
   - ❑ Click *Clear Rules* to delete all the rules from the list.
   - ❑ Click *Save Rules* to save the set of rules that you have just defined.

The design rule checks are saved as `<designfilename>drc.xml` in the current working directory for your design. You can enter a different file name or choose to save to a different directory.

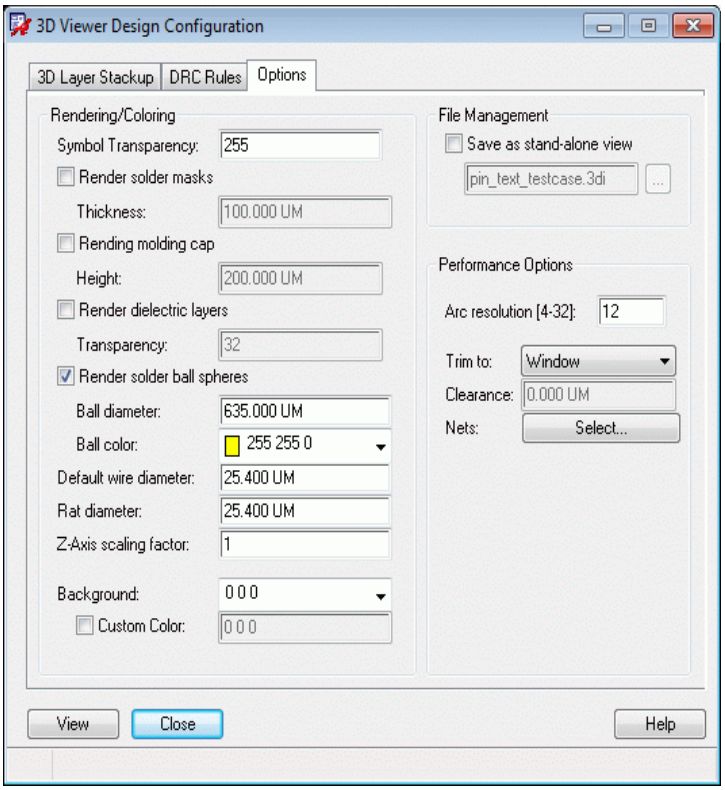
- ❑ Click *Load Rules* to load a set of rules that you defined and saved previously.

  You are prompted for the file name and location of the rules file that you want to load.

Tip

For more detailed information about defining rules with the *DRC Rules* tab, see DRC Rules Tab and Setting up 3D DRC Parameters.

7. Click the *Options* tab.



8. Under the *Rendering/Coloring* group box:

a. Check *Render solder masks* to display the solder mask opening. Enter a value in the *Thickness* box to specify the thickness of the solder mask.

b. Check *Rending molding cap* to display package molding cap. Enter a value in the *Height* box to specify the height of the molding cap.

Note: To create a detailed molding cap outline, create a `PACKAGE_CAP` layer on the SUBSTRATE GEOMETRY class and define molding cap outlines on that layer.

a. Check *Render dielectric layers* to display the dielectric layers. Enter a value in the *Transparency* box to specify the transparency of the displayed layer.

Note: You must select this option to see the cavity outlines as holes in the substrate.

9. Under the *File Management* group box, enable *Save as stand-alone view* to save the 3D design file (`.3di`) for future reuse.

Note: When specifying values for *Arc resolution*, consider the fact that the number of faces effects the DRC calculations, impacts database size, and memory in the viewer. A low number of segments executes faster but a high segment count gives a more accurate check.

Tip

See Options Tab for more details about working with the *Options* tab.

10. Click *View*. (If you do not want to launch 3D Viewer, click *Close* to dismiss the dialog box.)

3D Viewer processes the 3Di file and the design rule checks, then displays the 3D model.

Note: For large designs with complex rules, there may be a delay of several minutes before 3D Viewer responds with a “busy” indicator to show that it is processing the design.

Starting 3D Viewer in Standalone Mode

To start 3D Viewer in standalone mode without first launching your layout tool:

➤ Type `3dvu` at the command prompt.

This file is typically installed in the directory:
`C:\Cadence\SPB_16.6\tools\3dviewer`.

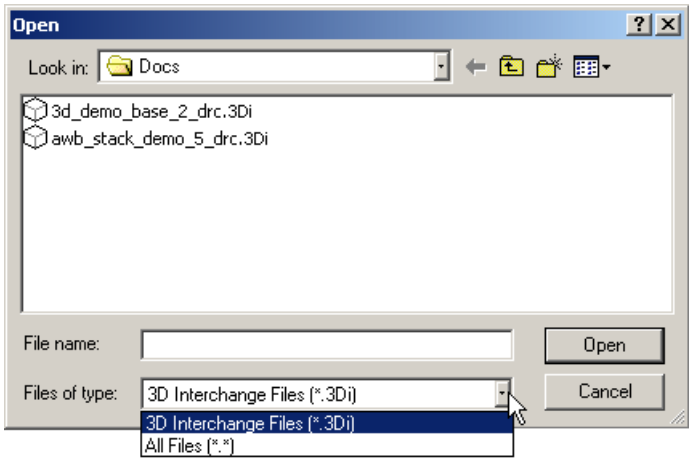
A batch file in the main install directory finds and configures the `3dvu.exe` file and necessary library paths.

Note: If you launch 3D Viewer this way, you are not able to back-annotate changes to the original `mcm` database.

To open a file from within 3D Viewer:

1. From the *File* menu, choose *Open*.

The Open dialog box appears.



2. In the *Files of type* drop-down list, choose the type of file (`*.3di`) that you want to open. (The default setting is `*.3di`.)
3. In the directory listing, click on the file name to choose the specific file that you want to open. (Use the *Look in* list to locate a different directory.)
4. Click *Open*.

   The selected file is loaded and the three-dimensional data appears as a 3D model in the View window.

Appending a File

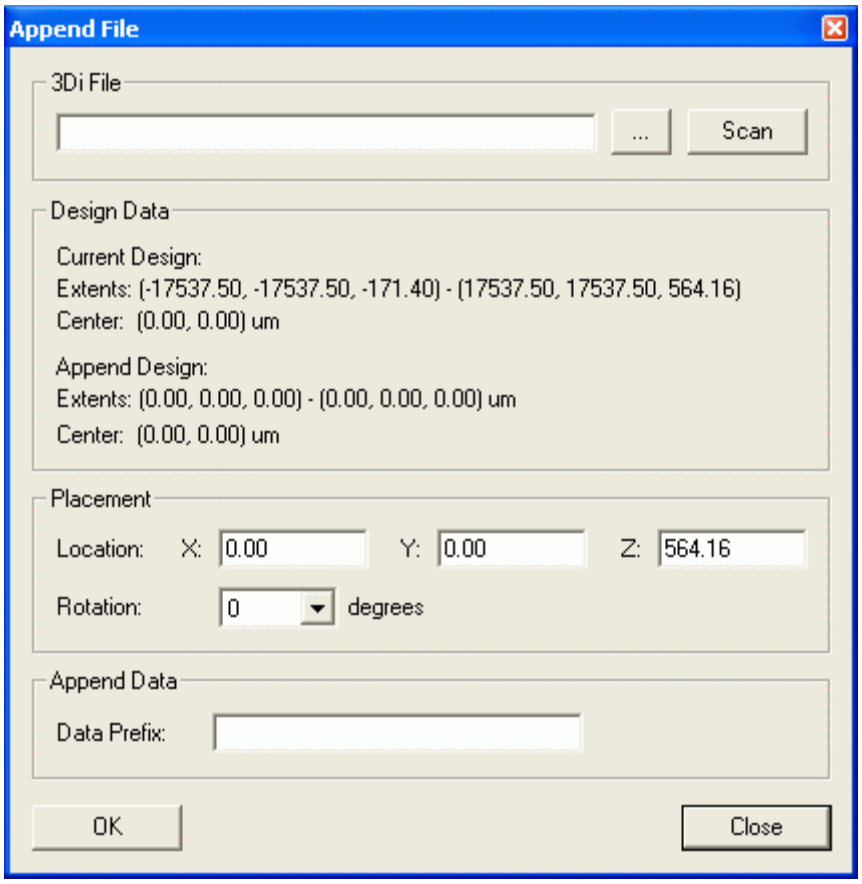
To append a file from within the 3D Viewer:

1. From the *File* menu in the 3D Viewer window, choose *Append*.

   The Append dialog box appears.

   It displays the extents and center data for the current design. The center is two-dimensional. The tool considers the reference point for the existing data (excluding bond

wires) to be at the maximum Z height of all extrusion, sphere, and barrel data (excluding DRCs).



2. Type the name of the file that you want to import in the *3Di* field or browse to locate the file.
3. Click the *Scan* button to update the extents and center data in the *Append Design* area.

   After you click *Scan* and move your cursor into the 3D Viewer window, a rectangle representing the data outline appears. This rectangle resizes if you zoom, pan, and change location or rotation values. It helps you determine where the tool should place the data.

   The default location represents centering the appended file on top of the existing data in the X, Y plane with the bottom of the appended file residing on the top of the existing data. Notice that the center is two-dimensional but the reference point is the bottom of the append data.
4. Type values in the *Location* fields to offset the data from the default location.

5. Use the drop-down list to specify a value in the *Rotation* field.
6. Type a prefix in the *Data Prefix* field to make the appended file a unique name.
7. Click *OK* to append the data.

   No changes are made to the original 3Di file on disk.

Viewing and Manipulating the 3D Model

You can change the display of a 3D model in the View window by assigning different colors to the layers of the design, and by turning on and off the display of different layers. You can also manipulate a 3D model in a variety of ways to view the model from different angles and perspectives. In addition, 3D Viewer allows you to control how the wire models are defined and displayed.

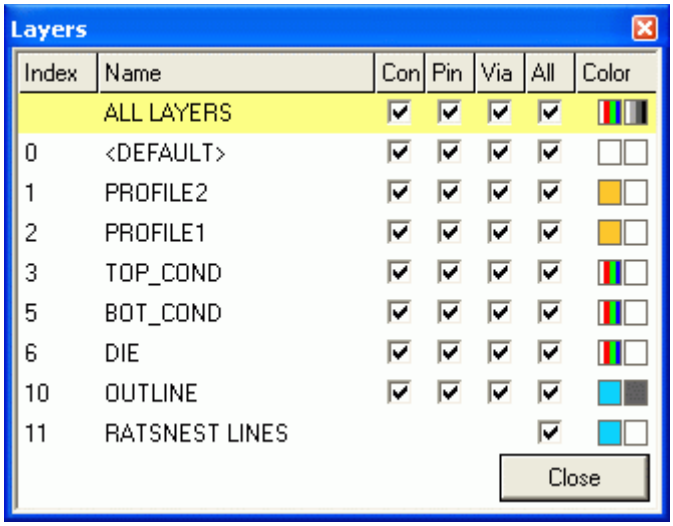
Choosing Layers and Color Assignments

You can choose which layers to display and colors to assign to those layers in the Layers dialog box.

To choose the layers for display:

1. From the *View* menu in the 3D Viewer window, choose *Layers*.

   The Layers dialog box appears.



2. Check the boxes for the layers and objects that you want to display. (To display all layers at once, check the box for *All Layers*.)

   The image in the View window changes to display the layers that you have selected.

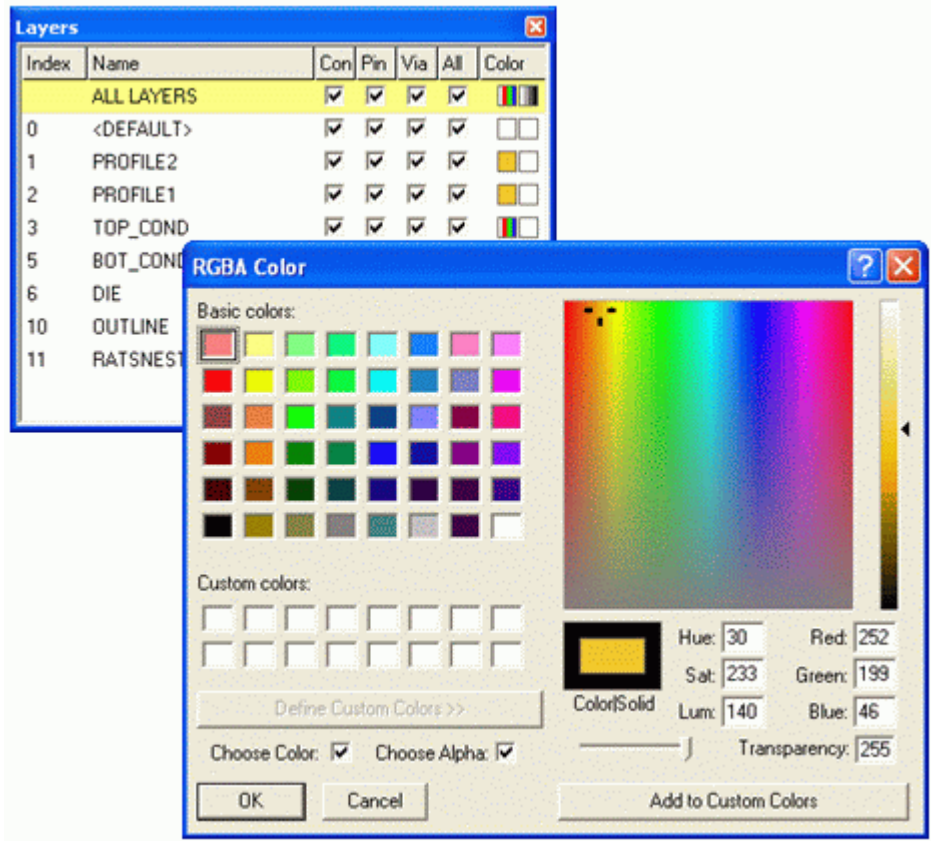
To assign colors to the layers:

1. From the *View* menu in the 3D Viewer window, choose *Layers*.

   The Layers dialog box appears.

2. Under the *Color* column, click the color box for the layer that you want to define.

   The RGBA Color dialog box appears.



3. From the color palette, choose the new color that you want to assign to this layer. Use the slider bar to adjust the transparency setting.

   The *Color*|*Solid* box displays the color choice and transparency level.


Caution

The color boxes are striped if different colors are assigned to different objects in the database that all reside on a common layer. If you change the color assignments in 3D Viewer so that all objects on a given layer have the same color, the color box will be solid. This may not always reveal the detail that you need to see in the 3D model. *When you change the color assignment to a single color, the original assignments are lost and you cannot reset the objects to different colors again.*

4. Click *OK*.

   The image in the View window changes to display the new color for the selected layer.

5. Click *Close* to dismiss the dialog box.


Tip

To change the background color of the View window from the default (black) setting, choose *Window Color* from the *View* menu and choose the desired background color from the color palette.

Setting the Z Scale Factor

The Z scale factor determines the vertical elevation of the displayed 3D model. (The default setting is 1; the range is from 1.0 to 10.0.) The higher the Z scale factor, the “taller” the image will be. Most packages are very large in X,Y compared to Z. (For example, a 30x30 mm package may be only 1 mm thick.) Because of this, you may have difficulty visualizing the Z-axis for dense designs such as flip chips.You can adjust the Z scale factor to distort the display in the vertical axis, and thereby facilitate the analysis of the design.

Note: The Z scale factor has no effect on 3D DRC calculations. It is purely a visual scaling. The 3D DRC engine reverts the Z scale to 1.0.

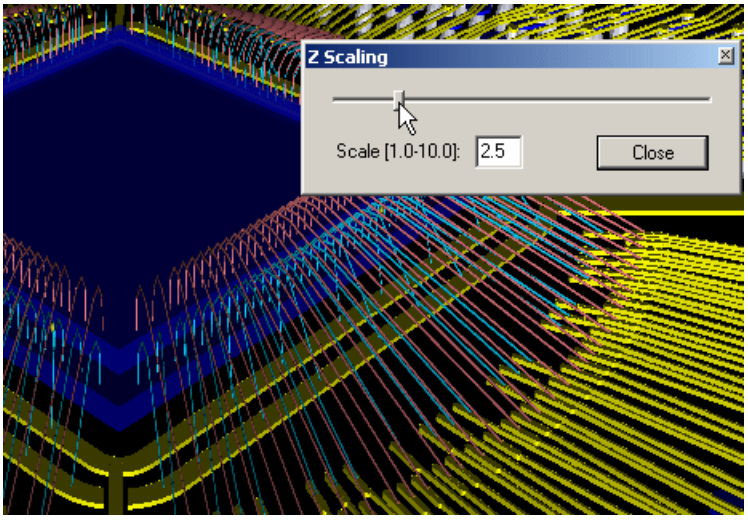
To set the z scale:

1. From the *View* menu in the 3D Viewer window, choose *Z scaling*.

   The Z Scaling dialog box appears.

2. Click on the slider bar and slide it to the right to increase the Z scale factor.

The image in the View window changes to display the new vertical distortion.



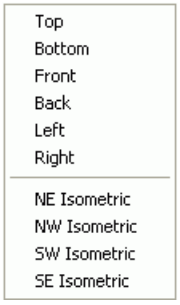
3. Click *Close* to dismiss the dialog box.

Manipulating the Model

You can manipulate the 3D model that is displayed in the View window to view it from different angles or perspectives.

To change the view of a model:

➤ From the *Camera* menu, choose the desired viewing angle or perspective from the list. (You can also click the corresponding *Camera* toolbar buttons to select these options.)



The image in the View window changes to reflect your selected viewing angle or perspective.

Tip

You can use the mouse buttons to pan and zoom through the design, or orbit the model. (For more details, see Using the Mouse.)

Selecting and Highlighting Objects

You can select specific objects in the design to obtain detailed information about those objects. You can also highlight specific objects so that you can analyze the design more effectively.

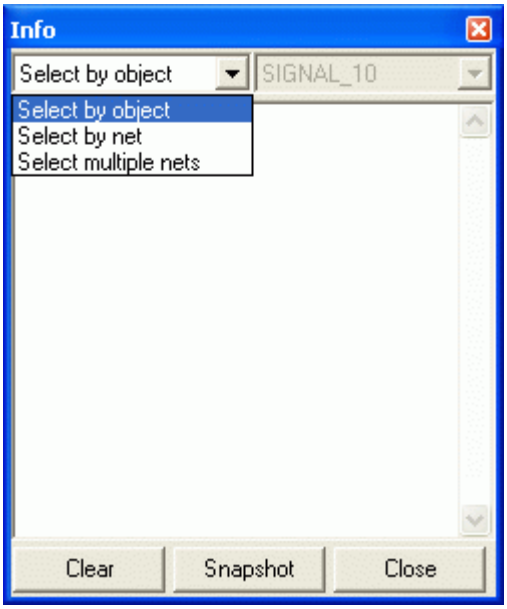
Selecting a Design Object

You can select specific design objects so you can analyze the design by individual elements.

To select an object or net:

1. In the 3D Design Viewer window, choose *Tools – Info*.

   The Info dialog box appears.



Important

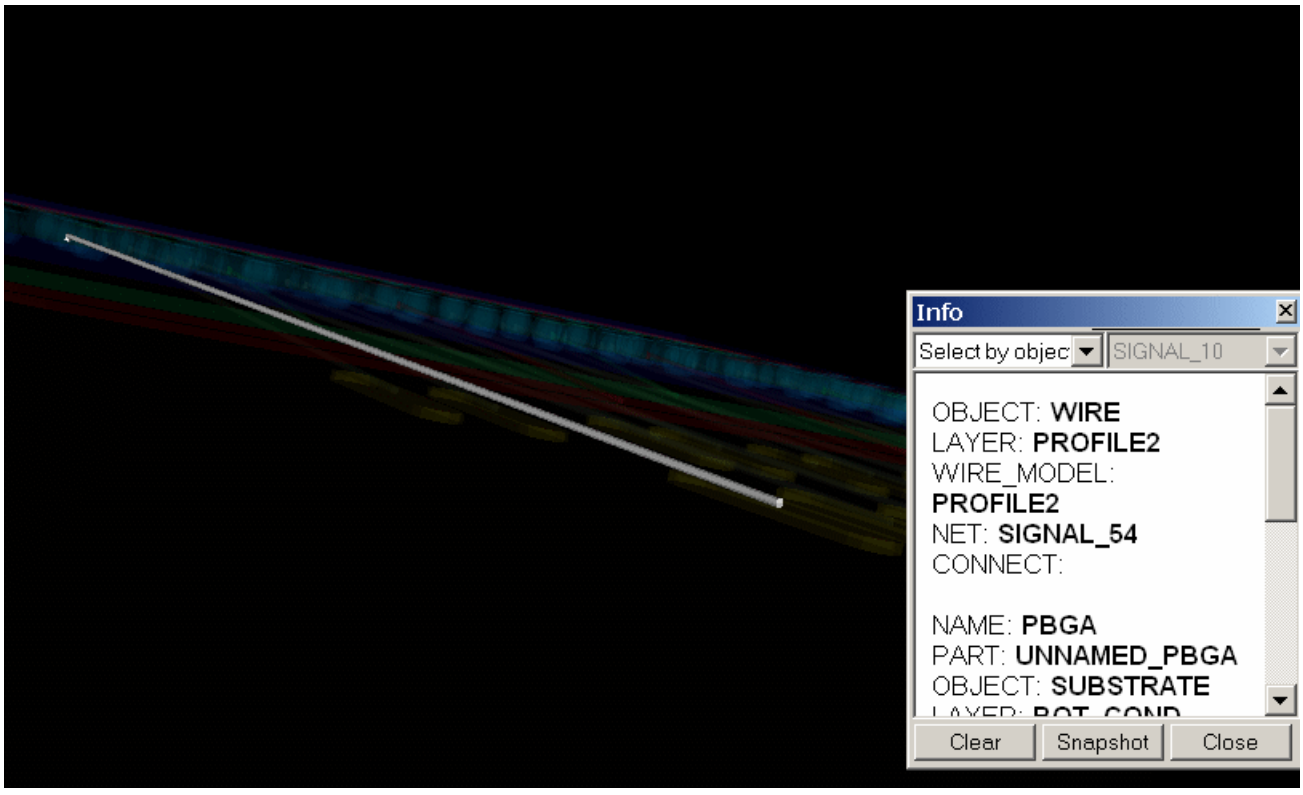
You must first activate the Info dialog box to be able to select an object. (For more information about using the Info dialog box, see Obtaining Information About Objects.)

2. In the drop-down list, choose *Select by object*. (You can also click *Select by net* to identify and highlight an entire net, or click *Select multiple nets* to identify and highlight more than one nets).
3. Click on the specific object (or net) in the design that you want to select.

Tip

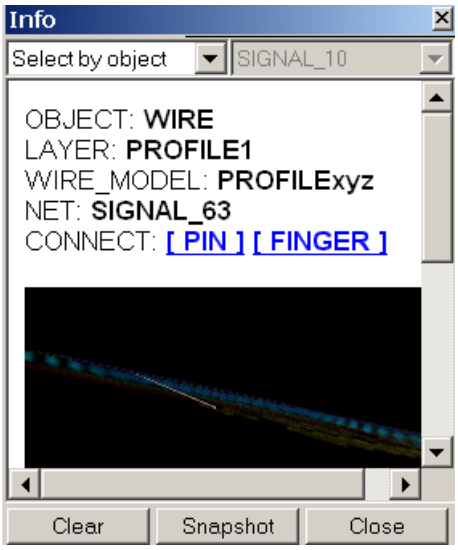
Use the pan, zoom, and orbit functions of your three-button mouse to locate the object you want to select (see Using the Mouse).

The selected object is highlighted and the Info dialog box displays detailed information about that object.



4. Click *Clear* to erase the highlighting of the selected object, clear the information in the Info dialog box, and restore the display to normal.

5. Click *Snapshot* to insert a snapshot image of the current display into the Info dialog box.



6. Click *Save* to save the information contained in the Info dialog box (including any snapshots) as an HTML file on your hard disk. (See Obtaining Information About Objects for more details about using the Info dialog box for documenting your analysis.)

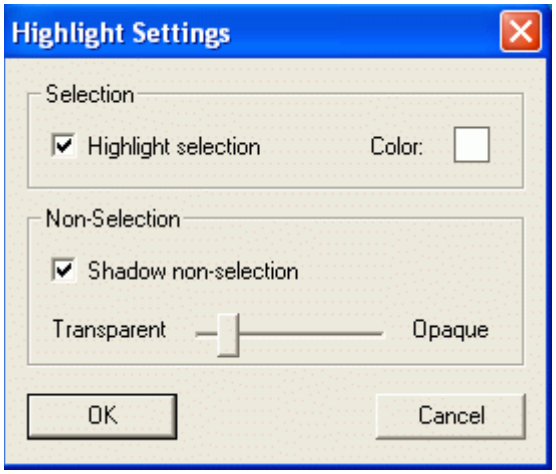
Highlighting a Design Object

When you select an object in the design, that object is highlighted. You can specify the color for highlighting objects, and set the dimming factor for non-highlighted objects. You can also disable the highlighting feature.

To specify highlighting parameters:

1. From the *View* menu in the 3D Viewer window, choose *Highlight*.

   The Highlight Settings dialog box appears.



2. Under the *Selection* group box, check or clear the *Highlight selection* box.
   - When this is enabled, whatever you select is highlighted with the color you specify.
   - When this is disabled, selected objects are not highlighted; they appear normally.
3. Under the *Selection* group box, click *Color* to specify a different color to assign to highlighted objects.
4. Under the *Non-Selection* group box, check or clear the *Shadow non-selection* box.
   - When this is enabled, the non-selected objects dim and become less visible.
   - When this is disabled, the non-selected objects appear normally.
5. Under the *Non-Selection* group box, adjust the *Transparent/Opaque* slider bar to specify how much visibility that you want to apply to shadowed objects.
   - With the slider bar towards *Transparent*, shadowed objects become less visible.
   - With the slider bar towards *Opaque*, shadowed objects become more visible.
6. Click *OK* to dismiss the dialog box and save the new settings.

Analyzing the 3D Model

3D Viewer provides several tools for analyzing the 3D model and obtaining information about the design. You can:

- Define wire models.

- Obtain detailed information about design objects, and annotate or mark up the rendered image so that you can document the design.
- Perform design rule checks on the 3D model.

Creating, Modifying, and Deleting Wire Profiles

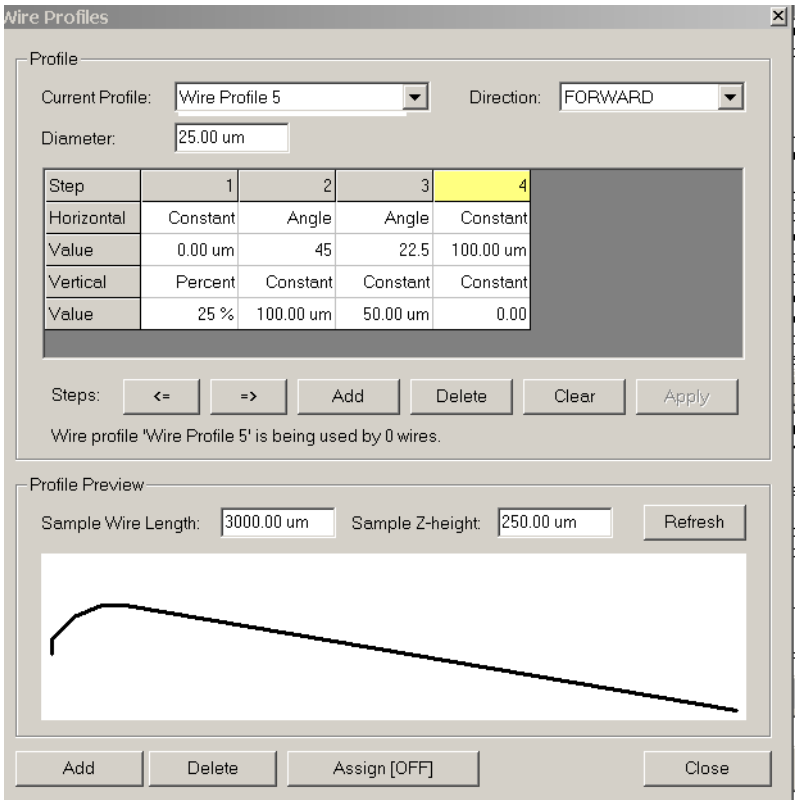
The 3D Viewer supports dynamic wire profile definitions. You can:

- Modify the properties of the wire profiles used in the design.
- Add new wire profile definitions
- Delete existing wire profiles.

If you launch this tool from APD+ and not standalone, for any wire profile changes you make, you can back-annotate them to the active design where they are stored in the database for next time.

To access the Wire Profiles dialog box (Figure 3-1), choose *Tools – Wire Profiles* from the menu bar.

Figure 3-1 Wire Profiles Dialog Box



How the Wire Profiles Dialog Box Works

The Wire Profiles dialog box displays one wire profile (*Current Profile*) at a time. You can choose from a list of wire profiles. Once you choose a profile, the screen refreshes to show data in that profile. You can also add, clear, and delete profiles.

Wire profiles have an associated direction that you can change. *Forward* implies that the wire runs from the die pad to the bond finger; *Reverse* means that the wire runs from the bond finger to the die pad. For die-to-die wire bonds, the forward direction means that you are moving from the pad that is farther away from the substrate to the pad that is closer to the substrate.

Horizontal and Vertical Components

You can create a wire profile by defining a series of steps that specify how the tool should advance from one point on the wire to the next. Each step consists of a horizontal and a vertical component. For each component, you can choose one of the following options:

- Length – Specifies a fixed value.

  This option supports positive and negative values in any units. The tool converts and displays the units as database units.

- Percent – Provides the same movement but as a percentage of the wire's length.

  This value differs for each wire that uses this profile. Use values between 1 and 100. The tool supports decimals.

- Angle – Specifies a change in the angle of the given amount (in degrees).

  Positive angles move away from the substrate; negative angles move toward the substrate. Angles are measured from the horizontal plane. Use values between -90 and +90. The tool supports decimals.

- Switch – Indicates that the tool should move to the other end of the wire for the next point.

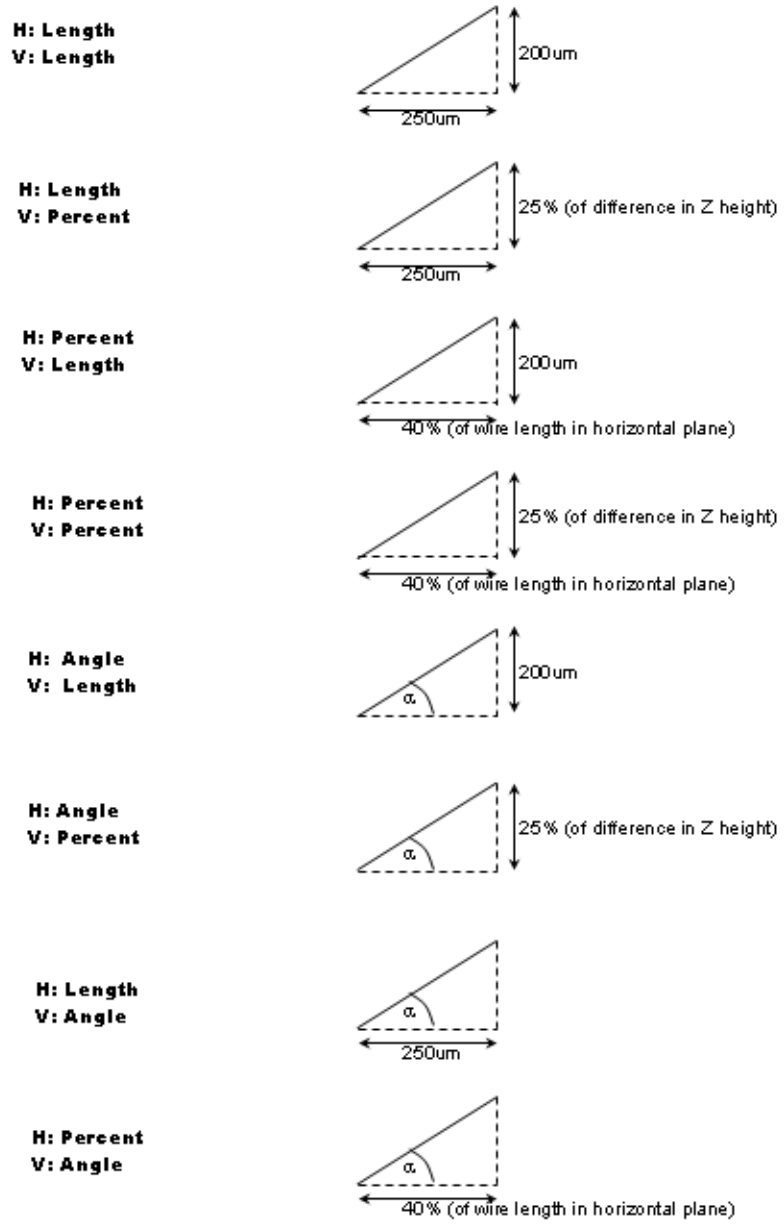
  The tool automatically removes the two value fields and sets the *Horizontal* and *Vertical* components to *Switch*. The remaining steps start from the other end of the wire. The tool proceeds to the segment where it left off.

  You can use only one *Switch* option per wire profile.

As you add each step, you can view a graphical representation of the profile that you are creating. The tool also displays an error message if you enter any illegal specifications, for example, leaving both the horizontal and vertical components at 0 UM.

Figure 3-2 helps to visualize the step components.

Figure 3-2 Horizontal and Vertical Components



Wire Profiles Dialog Box Descriptions

Current Profile	Specifies a list of all profile names currently defined in the design. You can change the name of the profile by typing in the field.
Direction	Specifies whether this profile defines a *Forward Bond* where the wire runs from the die pad to the bond finger or a *Reverse Bond* where the wire runs from the bond finger back to the die pad). The default setting is *Forward Bond*.
Diameter	Specifies the wire diameter. Use a positive integer. The default setting is 25um.
Step Table	Specifies a table of steps that make up the *Current Profile*. These steps define how to advance from one point on the wire to the next. Each step consists of a horizontal and a vertical component, described below. As you change the Step table, the tool automatically refreshes the profile preview, described below.

Horizontal

Provides a menu with these options to specify the movement type in the horizontal plane (X/Y axes along the length of the wire):

- Length – Specifies a fixed distance movement along the horizontal wire from the previous point in the model. The tool supports positive and negative values in any units.
- Percent – Specifies a distance movement along the horizontal wire from the previous point in the model, but specifies the distance as a percentage of the wire's length. Use values between 1 and 100. The tool supports decimals.
- Angle – Specifies a change in angle of the given amount in degrees, with positive moving away from the substrate and negative moving towards it. Use values between -90 and +90. The tool supports decimals.

  The tool controls the actual length moved along the horizontal plane for an angular specification with the angle and the percent/length specification for the vertical movement. For example, with a 45-degree angle and a height change of 50um, the horizontal movement is also be 50 UM (rise = run).
- Switch – Indicates to the tool that it should take the next point from the other end of the wire. For example, if you are moving from the beginning of the wire, the next point starts at the other end of the wire. Choosing *Switch* automatically removes the two value fields and sets both *Horizontal* and *Vertical* components to *Switch*.

Value

Lets you specify the value associated with the *Horizontal* movement type.

Vertical

Provides a menu with these options to specify the vertical movement type:

- Length – Specifies a fixed distance movement vertically from the previous point in the model.
- Percent – Specifies the percentage of the elevation difference between the wire start and end.

  The height of the wire that the tool uses for percentages is the difference in height between the start item and the destination item. Either may be a die pin or a finger (start can be a finger in an interposer). From the height of the first item, the tool subtracts the height of the second item.

  For example, a die on the substrate is 250 UM thick. Therefore, the height of the die pad (Start item) is 250 UM. The finger, which is on the substrate, is at 0 UM. The difference in height is 250 UM.

  Now, if you want to move vertically by 50%, the tool computes as 0.5 * 250 UM = 125 UM.
- Angle – Specifies a change in angle of the given amount, with positive moving away from the substrate and negative moving towards it.
- Switch – Indicates to the tool that it should take the next point from the other end of the wire. For example, if you are moving from the beginning of the wire, the next point starts at the other end of the wire. Choosing *Switch* automatically removes the two *Value* fields and sets both *Vertical* and *Horizontal* components to *Switch*.

Value

Lets you specify the value associated with the vertical movement type.

Note: If you include a step that has two zero values resulting in a zero-length step or a step with two angles defined, but no horizontal or vertical component, the following message appears:

```
These steps are invalid and need to be fixed or removed: 3.
```

Steps

<= and =>

Click on a step and then the arrow to move it forward or backward in the profile.

Add	Click to add a new step. The step appears at the end of the profile. You can use the arrows to move it.
Delete	Lets you delete a step. Click on the step that you want to delete and then click *Delete*.
Clear	Removes all the steps from the *Current Profile.*
Apply	When you change the *Current Profile*, the *Apply* button becomes available. If you try to display a different profile, assign wires, add a new profile, or close the dialog box before clicking *Apply*, a prompt appears. If you click: ■ *Cancel*, the tool takes no action. ■ *No*, the tool removes any changes in the dialog box before proceeding. ■ *Yes*, the tool updates the wire profiles in the database before continuing.
Profile Preview	The tool displays a profile preview based on *Sample Wire Length* and *Sample Z-height* values. This preview updates based on any changes you make in the steps, even before you click *Apply*. You can preview the wire profile before you save any changes.
Sample Wire Length	Specifies the wire length to use for drawing the graphical example. This default setting is 3000 UM or equivalent units.
Sample Z-height	Specifies the starting height or the distance from the top of the substrate (used for the graphical example). By default, this setting is 250 UM. You can use a negative number for the Z-value. Profiles moving in a FORWARD direction move from left to right and those moving in REVERSE, move from right to left.
Refresh	Click *Refresh* to update the *Profile Preview* when you change the *Sample Wire Length* and *Sample Z-height* values. When you change the Step table, the tool automatically refreshes the preview.
Add	Click this button to add a new wire profile. The name of the profile is *Wire Profile <n>* depending on how many profiles exist. You are prompted to apply or save changes if necessary.

Delete	Click this button to delete a *Current Profile.* If any of the wires in the design use the *Current Profile,* the tool displays a message that you cannot delete the profile.
Assign [OFF]/[On]	Toggles between two states: enabling or disabling the assignment function. Click the button to *On* and then click the wires in the 3D Viewer that you are assigning to the *Current Profile* values. The tool updates the dialog box to reflect the number of wires in use and the view according to any highlighted settings in place. The graphical depiction of the wires in the design changes to reflect the new profile description. However, the DRC calculations are not up-to-date at this point. You must manually refresh them by choosing *DRC – Rules* and clicking the *Check Rules* button.
Close	Closes the dialog box and ends the assignment state if necessary. You are prompted to apply or save changes if necessary.

Adding a Wire Profile

To add a wire profile:

1. Choose *Tools – Wire Profiles* from the menu bar in the 3D Viewer window.
2. At the bottom of the Wire Profiles dialog box, click *Add.*

   The *Wire Profile <n>* name appears in the *Current Profile* text box. You can change the name.
3. Specify the profile direction in the *Direction* field: *Forward Bond* or *Reverse Bond*.
4. Modify the wire diameter, if needed.
5. Modify the values for the *Sample Wire Length* and *Sample Z-height*, if needed.
6. Click *Add* in the *Steps* frame.

   A format for creating a step appears. The *Horizontal* and *Vertical* Components are set to 0.
7. Click on the drop-down menu in the *Horizontal* field to view the options. Then choose an option and specify a value in the *Values* field below it. See Wire Profiles Dialog Box Descriptions on page 60 for descriptions of the options.

8. Click on the drop-down menu in the *Vertical* field to view the options. Then choose your option and specify a value in the *Values* field below it. See Wire Profiles Dialog Box Descriptions on page 60.

   The image changes as you change movement types and values.

9. Repeat Steps 6 through 8 until you complete your profile.
10. Click *Apply* to save the wire profile.
11. When finished, click *Close* to close the dialog box.

Modifying a Wire Profile

To edit a wire profile:

1. In the Wire Profiles dialog box, click the *Current Profile* list and choose the profile that you want to edit.
2. Change values in these fields: *Direction*, *Diameter*, *Sample Wire Length*, or S*ample Z-height,* if needed.
3. If you change the values for *Sample Wire Length* or S*ample Z-height,* click *Refresh* to update the Profile Preview.
4. Click on a step and use the arrows to move the position of a step.
5. Click *Add* to add a new step.

   It appears at the end of the profile. You can use the arrows to move it.

6. Click on a step and click *Delete* to delete it.
7. Click *Clear* to remove all steps.
8. Click *Apply* to save the changes to the wire profile.

Deleting a Wire Profile

To delete a wire profile:

1. In the Wire Profiles dialog box, click the *Current Profile* list and choose the profile that you want to delete.
2. At the bottom of the dialog box, click *Delete.*

   If any of the wires in the design are using the *Current Profile,* the tool displays a message that you cannot delete the profile.

Assigning Wire Profiles

To assign a wire profile to wire bonds:

1. In the Wire Profiles dialog box, click the *Current Profile* list and choose the profile that you want to assign to the wire bonds.
2. Toggle *Assign [OFF]* to [*ON*].
3. Click the wire bonds that you are assigning to the *Current Profile*.

   The wire bonds are highlighted in the design and are assigned to the *Current Profile*. The tool updates the dialog box to reflect the number of wires in use and the view is updated according to any highlighted settings in place.
4. To update the package, choose *File – Update Package* in the Cadence 3D Design Viewer window.
5. In the Update Package dialog box, check the appropriate boxes.
6. For additional information on this dialog box, see Updating the Package on page 97.
7. Click *OK*.

Obtaining Information About Objects

The 3Di database consists primarily of 3D objects and information about each object. Attributes are assigned to each object and the values of these attributes are stored in tables. You can view these object attributes and their values to help you analyze the 3D model.

The following table describes the attributes that are assigned to an object.

Attribute	Description
`ID`	A unique identification number that is automatically assigned to the object. Example: `ID: 113480976`
`OBJECT`	The object type. Example: `OBJECT: WIRE`
`NAME`	The name of the object. Example: `NAME: DIE1`

Attribute	Description
LAYER	The layer where the object resides. Example: `LAYER: WIRE MODEL 0`
LAYERS	All the layers where a pin or via is defined. Example: `LAYERS: WIRE MODEL 0`
NET	The net name associated with the object. Example: `NET: DIE1_152`
WIRE_MODEL	The wire model description associated with the object. Example: `WIRE_MODEL: ARTWORK FORWARD 250.0 400.0 250.0 350.0`
CONNECT	The objects connected to the object. Example: `CONNECT:` **[PIN] [FINGER]**

Displaying Information About an Object

To display information about an object:

1. From the *Tools* menu in the 3D Viewer window, choose *Info*.

   The Info dialog box appears.

2. In the drop-down box, choose *Select by object* or *Select by net*.

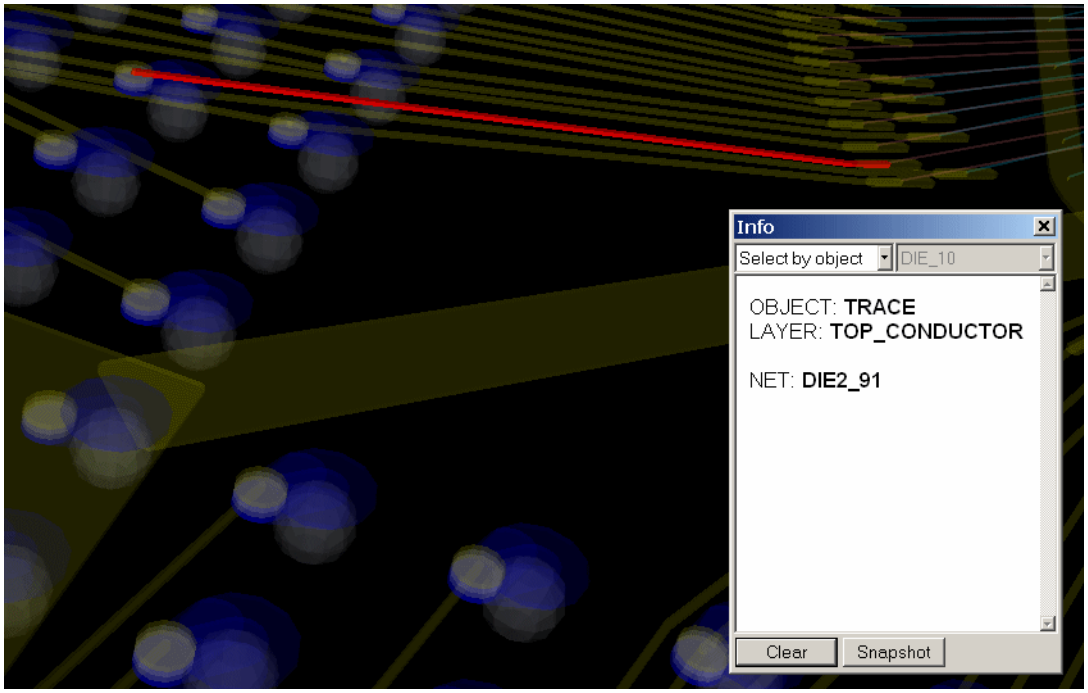
   - *Select by object* lets you select a design object and displays the attribute information associated with that object.
   - *Select by net* lets you select a net in the design and displays only the name of that net. (This mode is most useful for tracing a complex net through a multi-layer design.)

3. Click on the specific object (or net) in the design.

Tip

Use the pan, zoom, and orbit functions of your three-button mouse to locate the object that you want to select (see Using the Mouse). Be careful when using the orbit function (left mouse button) after selecting an item, since a left click is used to select a new object, even when you hold the button down.

The selected object is highlighted and the Info dialog box displays detailed information about that object.

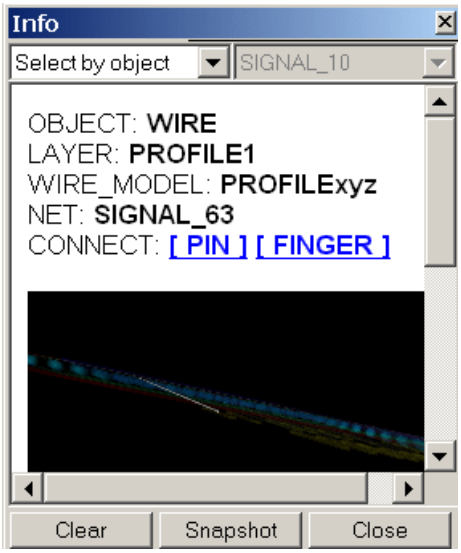


Tip

To make it easier to read the information, you can resize the *Info* dialog box by dragging the sides of the box.

4. Continue to click on other objects to add information about them to the list in the Info dialog box.
5. Click *Clear* to erase the highlighting of the selected object, clear the information in the Info dialog box, and restore the display to normal.

6. Click *Snapshot* to insert a snapshot image of the current display into the Info dialog box.



7. Click *Close* to dismiss the dialog box.

   Note: The information contained in the Info dialog box is saved automatically as `<filename>.html` in the same directory as your design file (`<filename>.3di`). By default, the snapshot images from the *Info* box are saved in a subfolder (`<filename>_images.dir`) of the same directory as your design file (`<filename>.3di`), and are numbered sequentially as `<filename>_1.jpg`, `<filename>_2.jpg`, and so on.

Measuring in 3D

You can measure the 3D distance between the following design objects:

- Wires
- Traces
- Shapes
- Bond fingers
- Substrates (non-circular)

You cannot measure between the following objects:

- Any circular extrusions such as vias or ball pads
- Any spheres (solder balls)

Note: Select a bond wire for one of the two objects that you select for a distance measurement, or you receive a warning message. Any of the other objects that are listed here can serve as the second object for measurement.

To measure the 3d distance between two objects:

1. From the *Tools* menu, choose *Distance*.

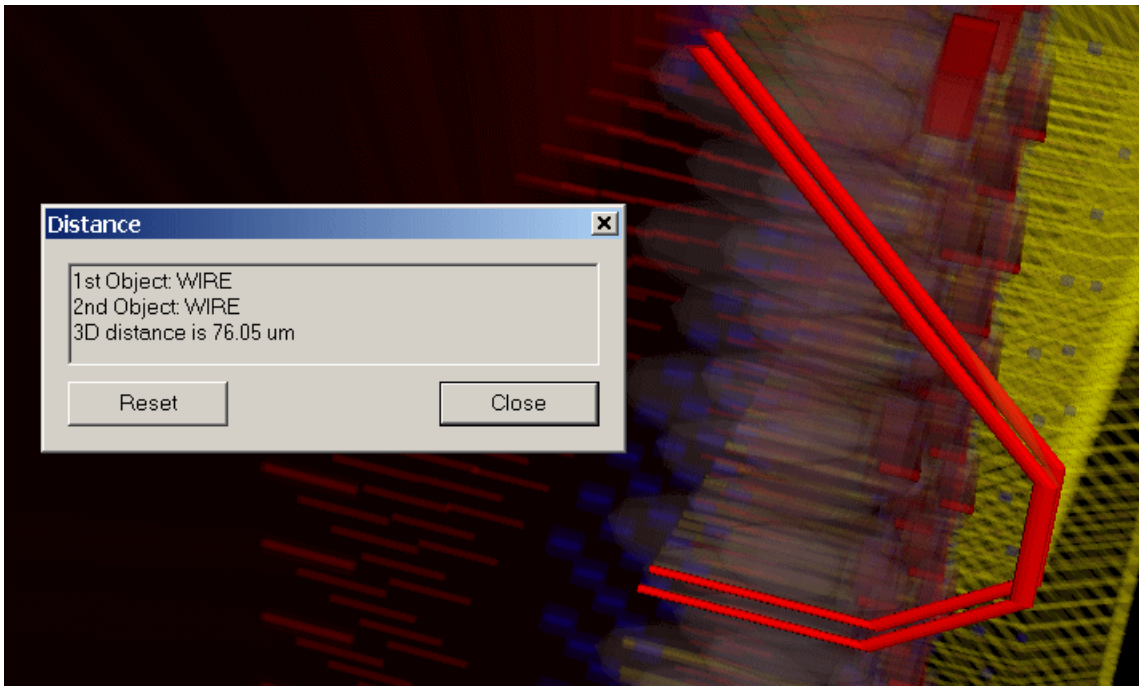
   The Distance dialog box appears.

2. Click on the object from which you want to start measuring.

   The object is highlighted.

3. Click a second time on the object to which you want to measure.

   The second object is highlighted. The selected objects are identified and the exact 3D distance (in the current design units) between the two objects is shown in the Distance dialog box.



4. Click on two new objects to repeat the process and measure between those objects.

Tip

Click *Reset* to clear the information in the Distance dialog box before making another measurement.

5. Click *Close* to dismiss the dialog box.

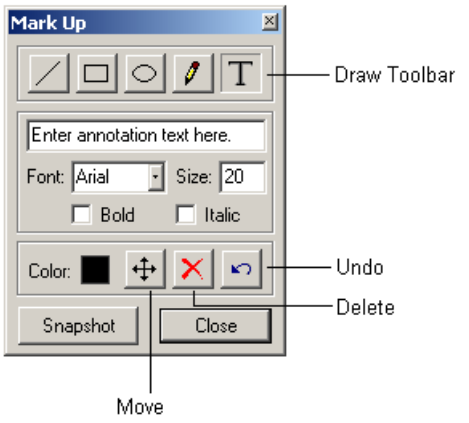
Annotating the Image

You can mark up the displayed image with annotations to help you track design errors or record important notes for documenting design changes.

Caution

Any markups that you add to the display are only temporary. They are deleted when you dismiss the Mark Up dialog box. To be sure that you save the markups for future reference, use the** File — Export **Image command before you dismiss the Mark Up dialog box (see <u>Exporting an Image</u>).

To insert annotations, first choose *Mark Up* from the *Tools* menu. In the Mark Up dialog box, you can choose different markup options to apply to the design.



Tip

Before you start the markup process, be sure to pan and zoom into the desired view of the design so that the image you want to export will be set up properly. Once you start annotating, you cannot change the 3D viewpoint behind the markups.

Adding a Text Note

To add a text note:

1. In the Mark Up dialog box, click the *T* (Text) toolbar button.

2. Click the *Color* box to choose a color that contrasts with the background and layer colors so that the note will be visible.
3. Choose the *Font* and *Size* for the text, and check the *Bold* or *Italic* boxes, if desired.
4. In the text box, type the note that you want to add.
5. Click in the View window where you want the note to be inserted.

   The text note you entered appears in the display.

Marking an Area of the Design

To mark off an area of the design:

1. In the Mark Up dialog box, click either the *Rectangle* or *Oval* toolbar button.
2. Check the *Color* box to choose a color that contrasts with the background and layer colors so that the rectangle or oval will be visible.
3. Click in the View window where you want to insert the rectangle or oval, then drag the cursor to increase the size and surround the area you wish to mark. (A phantom image appears as you size the shape you are drawing.)

Tip

Press and hold the `Ctrl` key while in *Rectangle* mode to draw a square. Press and hold the `Ctrl` key while in *Oval* mode to draw a circle.

4. Click again to complete the shape.

   The shape appears in the display.

Drawing a Leader Line

To draw a leader line:

1. In the Mark Up dialog box, click the / (Line) toolbar button.
2. Check the *Color* box to choose a color that contrasts with the background and layer colors so that the line will be visible. (Red, yellow, or orange stand out well against most other colors.)
3. Click in the View window where you want to start the line, then drag the cursor to where you want to end the line.
4. Click again to complete the line.

The leader line appears in the display.

Sketching a Line

To sketch a line:

1. In the Mark Up dialog box, click the *Pencil* toolbar button.
2. Check the *Color* box to choose a color that contrasts with the background and layer colors so that the line will be visible.
3. Click in the View window where you want to start the line, then drag the cursor to sketch in the line however you want it to look. (A phantom image appears as you sketch.)
4. Click again to complete the sketch.

   The sketched line appears in the display.

Undoing a Markup

To undo a markup:

1. In the Mark Up dialog box, click the *Undo* toolbar button.

   The last markup you inserted in the display disappears.
2. Continue to click the *Undo* toolbar button to remove previous markups.

Deleting a Markup

To delete a markup:

1. In the Mark Up dialog box, click the *Delete* toolbar button.
2. Click on the annotation that you want to delete.

   The annotation disappears from the display.

Moving a Markup

To move a markup:

1. In the Mark Up dialog box, click the *Move* toolbar button.

2. In the View window, click on the markup that you want to move and hold down the left mouse button.
3. Drag the cursor to the new location, then release the button.

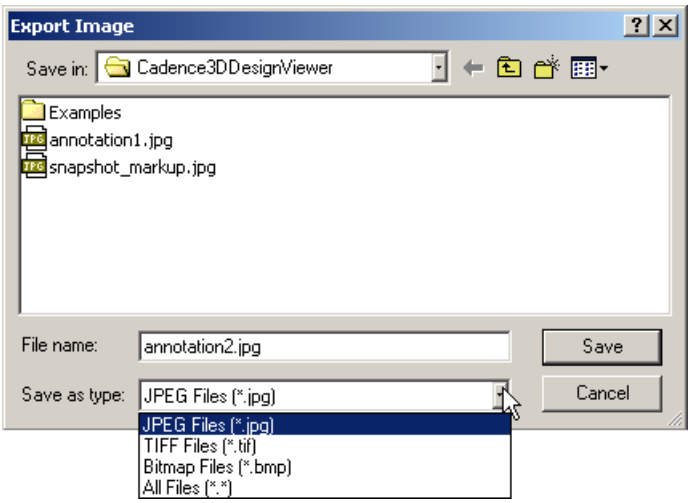
   The markup appears in the new location.

Creating a Snapshot Image of the Markups

To create a snapshot image of the markups:

1. In the Mark Up dialog box, click *Snapshot*.

   The Export Image dialog box appears.



2. Click *Save* to save the image.

   By default, 3D Viewer saves the displayed image as `<filename>.jpg`, in the same directory location as your design file (`<filename>.3di`).

Caution

Each time you click Shapshot again, the old file `<filename>.jpg` ***is overwritten with the new image. To preserve the previous markup image, be sure to rename the old file before making another shapshot.***

Printing an Image

You can print the image that is currently displayed in the View window, including any annotations you have made. You can specify the quality (dpi resolution) of the printed image, and preview the image before printing.

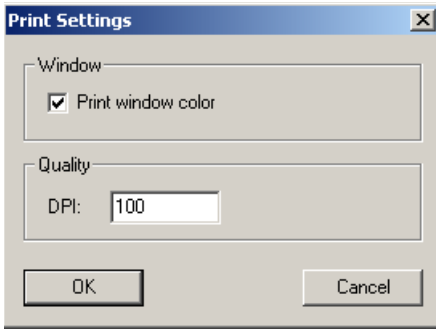
Note: Printing is supported only on Windows.

Before you print an image, be sure to specify the particular layers and colors that you want printed, so that these appear correctly in the View window (see Choosing Layers and Color Assignments).

To print an image:

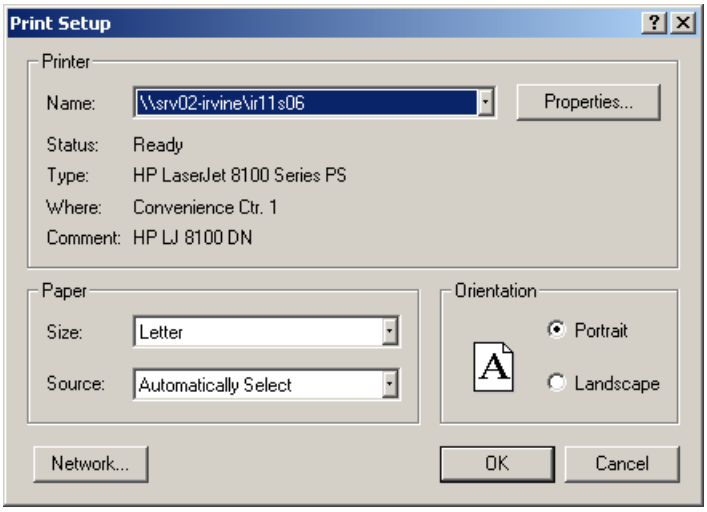
1. From the *File* menu, choose *Print Settings*.

   The Print Settings dialog box appears.



2. Under the *Window* group box, check or clear the *Print window color* box. (The default setting is enabled.)
   - If you enable this option, the background color of the View window prints.
   - If you disable this option, the background of the View window is transparent and does not print.
3. Under the *Quality* group box, in the *DPI* text box, specify the resolution (in dpi) for the printed image. The default value is `100`.
4. Click *OK*.
5. From the *File* menu, choose *Print Setup*.

The Print Setup dialog box appears.



6. Choose the appropriate printer assignment and properties for your computer system.
7. Click *OK*.
8. From the *File* menu, choose *Print Preview*.

   The View window changes to the print preview display. The image you see displayed here is what is printed. You can change layer color assignments by clicking *Color*.
9. Click *Print* to print the image. (To change any settings, click *Close* to return to the standard View window.)

   The Print dialog box appears.
10. Specify the print range and number of copies to be printed.
11. Click *OK*.

    The image is printed.

Exporting an Image

You can export the image that is currently displayed in the View window, including any annotations that you made. When you export the image, you can save it to the hard disk of your computer as either a `.jpg`, `.tif`, or `.bmp` file. By default, the image is saved as a `.jpg` file in the same directory location as the source files (*.3di*) for your design.

You can specify a scale factor (integer) and output a bitmap image that is a much higher resolution than what is currently viewed on screen. The higher resolution output is valuable if you intend to print the image.

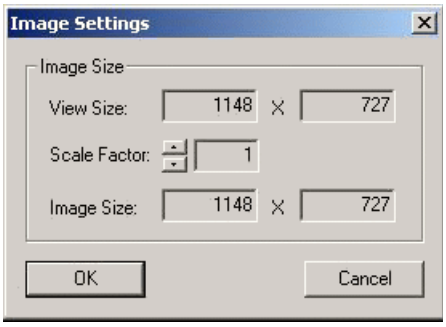
Caution

Before you export an image, be sure to specify the particular layers and colors that you want printed, so that these appear correctly in the View window (see Choosing Layers and Color Assignments).

To export an image:

1. From the *File* menu, choose *Image Settings*.

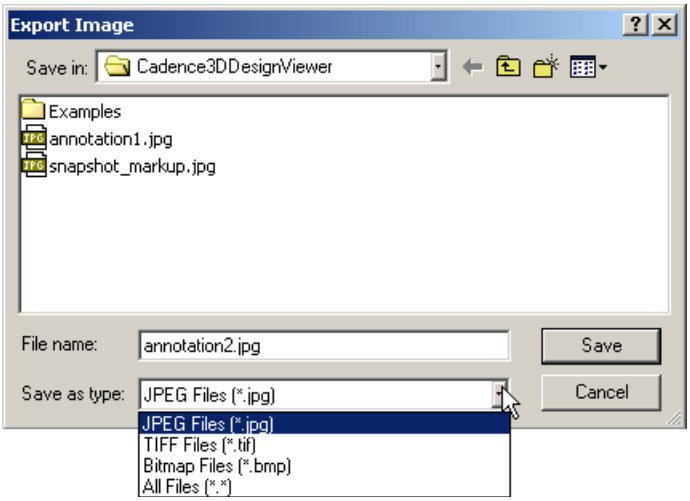
   The Image Settings dialog box appears.



2. Specify the parameters that you want to set for *View Size* (in pixels). (The default *View Size* values show the current size of the image in the View window.)
3. Set the *Scale Factor* between `1` and `10` to magnify the image size. Use the scroll buttons to choose the desired value.

   The *Image Size* values adjust automatically based on the factor you select.
4. Click *OK*.
5. From the *File* menu, choose *Export Image*.

The Export Image dialog box appears.



6. From the *Save in* drop-down list, choose the directory where you want to save the file.
7. From the *Save as type* drop-down list, choose the file format for the file (`.jpg, .tif, .bmp`).
8. In the *File name* text box, enter the name for the file.
9. Click *Save*.

   The image displayed in the View window is saved to your hard disk under the file name that you specified and in the directory you specified.

Saving Data

3D Viewer saves the current design as a 3Di file (`<filename>.3di`). You may want to save when you need to send a 3Di file to another or save changes made that you are not ready to save to the `mcm` design. You also should save your design if you are running the 3D Viewer standalone as you are unable to update the design.

Changes to the following are also saved:

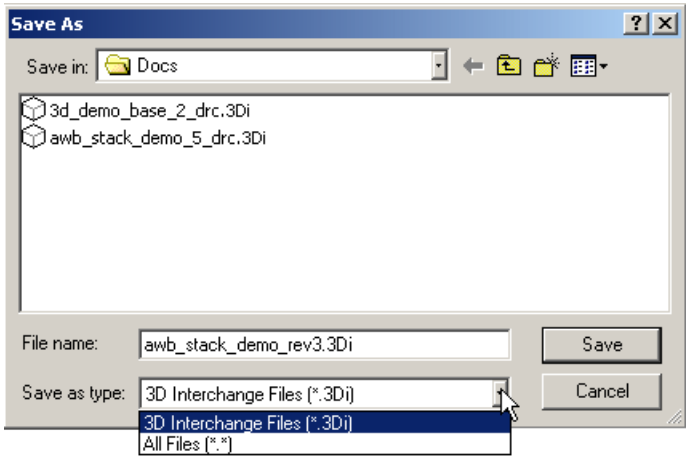
- Wire groups
- Wire group assignments
- Models
- Model parameters

- Layer color and transparency settings
- Design rule checks and 3D DRC error markers

To save a file under a new name:

1. From the *File* menu, choose *Save As*.

   The Save As dialog box appears.



2. In the *File name* text box, enter a new file name for the file.
3. In the *Save as type* drop-down list, choose the file format for the new file.
4. Click *Save*.

   The file is saved under the new file name.

Scripting

To record and play scripts within the View window:

1. Choose *File – Script* from the menu bar.

   The Scripting dialog box appears.

2. Browse to find the directory where you want the script to reside.
3. Type the name in the *Script File* field.
4. Click *Record* to start the recording.
5. Perform the desired actions in the Viewer window.

6. Click *Stop* when you are finished.
7. Click *Play* to play the recording.
8. Click *Close* to dismiss the dialog box.
9. To view the ASCII script, use a text editor.

4

Performing Design Rule Checks

This chapter discusses the following:

3D Design Rule Checks

Important

The 3D Design Rule Checks feature is a special option. You must have a specific license to run this option. If you are not licensed to run it, you will not be able to perform the functions described in this chapter.

The 3D Design Rule Checks feature lets you validate wire bond patterns both visually and through a series of three-dimensional design rule checks (DRCs).The DRC marker has been refined to better show the violation point on each object. You can take advantage of this functionality at one or more of the following design stages:

- After the initial wirebond pattern generation is complete
- After designing a die stack using the die stack editor, to verify the stack and the wirebond pattern
- After package design is completed, but before the final electrical analysis and manufacturing preparation has been done

You may invoke this feature at any time on any part of the design to obtain a detailed 3D model of that area. This can be useful for visualizing a via structure or a net's path through the package, for example. You can perform the following checks:

- Validate that wire-component clearances meet assembly guidelines. For instance, a spacer must be thick enough that the top die, when mounted, does not touch or impact the bond wires connecting to a lower die.
- Ensure that the wirebond pattern can be implemented during bond wiring. Since wires are placed one at a time, there are very specific rules regarding the order in which wires are placed, and the separation that must be maintained between adjacent wires in each axis.
- Verify that test points will be accessible after the package is manufactured.

Applying 3D Design Rule Checks

The recommended use model for applying 3D Design Rule Checks is described below. (See Setting up 3D DRC Parameters for more detailed procedures.)

1. Reach a state in your design flow using APD+ where you want to see the geometries in a 3D perspective from the main top-down, 2D view.
2. In your layout tool, from the *View* menu, choose *3D Model*, or enter `view 3d` in the command line, or click the *View 3D* toolbar button.
3. In the *Options* tab of the 3D Viewer dialog box, set up the graphical viewer options that you want to use for this design. (Normally, you do not have to adjust these settings since most of the values are derived from your current design.)
4. In the *3D Layer Stackup* tab, define the stackup ordering and height of any dies, spacers, or other layers in your design. Note that the values listed for thickness and height are the actual values, and are not scaled by any scaling factor that you may have set in your view options.
5. Set up any wire profile / loop height group models to be used with this design. There is one line listed for each wire profile, so that each can be custom-configured to meet your specifications.
6. In the *DRC Rules* tab, you can clear the existing set of rules, define rules, or load rules from an external file. To save your changes for later reuse, you may save these to a file.
7. Select the objects in the main layout tool design window that you want to display in 3D Viewer. If you do not select anything, 3D Viewer generates a model of the complete design.
8. In the 3D Viewer dialog box, click *View* to process the design and display the three-dimensional view of the objects that you have selected in 3D Viewer.

If a 3D Viewer window is currently open for the active APD+ session, it is closed if you open a new one.

9. Use 3D Viewer to view, check, query, or generate reports on the three-dimensional view of the design.
10. You may leave 3D Viewer open for reference as you continue to modify your design in your layout tool.

    Note: Any changes you make in your layout tool are *NOT* dynamically reflected in 3D Viewer.
11. When finished working with the three-dimensional view, close 3D Viewer and return to your layout tool.

Setting up 3D DRC Parameters

You define rules for 3D Design Rule Checks prior to loading the design data from your layout tool. In your layout tool, click the *View* menu and then choose *3D Model* to access the 3D Viewer dialog box, where you can set up the DRC parameters.

Tip

You can write script files in your layout tool to set up design rule checks and launch 3D Viewer. You can then modify these scripts and reuse them for other designs. For more details about DRC scripts, see Using Script Files to Launch 3D Viewer.

You can also use and modify 3D design rule checks that may have been created by the CAD library group at your company and stored as XML files on disk.

The 3D design rule checks are logically constructed:

- Define a group of entities for the first data input
- Define a group of entities for the second data input
- Define a rule or parameter to test

Example 1

All wires in group 1	first data set
All wires in group 2	second data set
min spacing = 50 UM	rule and parameter

Example 2

All wires	first data set
substrate on layer DIE1	second data set
min spacing = 100 UM	rule and parameter

Note: Certain design rule checks require additional exceptions. Consider the case where you want to assure that a wire does not cross into a die body substrate. Each section of the wire must be checked independently. However, if the wire that you are testing originates from the die under consideration, then one of the wire endpoints will be touching the die body and the first vertex may also be closer than the minimum spacing, without creating a violation. Therefore, the design rule has to understand this and not flag the endpoint as a violation.

For example, to apply a different design rule between wires in profile group 1 than between those in profile group 2, make two distinct design rules. In both rules, specify `wire` as the object *Type* for the two items. In the first rule, pick wire profile 1 for both *Objects* and in the second rule, pick wire profile 2. That way, each is checked against a different spacing value.

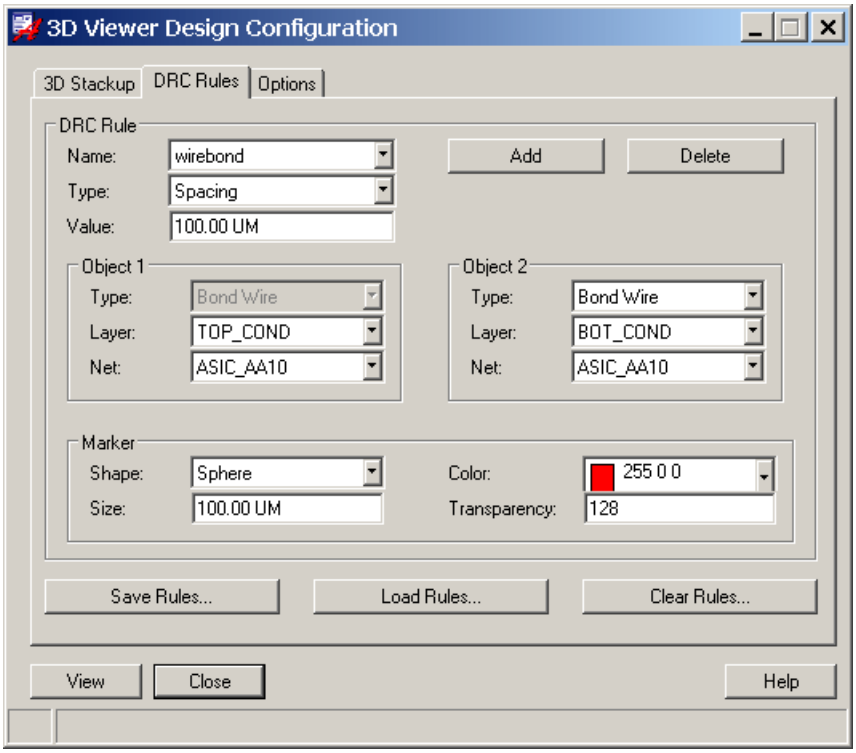
Creating a New Design Rule

To create a new design rule:

1. In your layout tool, from the *View* menu, choose *3D Model*.

   The 3D Viewer Design Configuration dialog box appears.

2. Click the *DRC Rules* tab.



3. Under the *DRC Rule* group box, click *Add*.

   A dialog box appears.

4. In the *Name for new DRC Rule* text box, type a name for the rule, or accept the default number that is automatically assigned. (Each new rule is automatically assigned a sequential number by default, starting with `1`.)

5. Click *OK*.

   The new rule name appears in the *Name* drop-down list. Default values appear automatically for the other parameters in the *DRC Rules* tab.

6. From the *Type* drop-down list, choose the type of design rule check that you want to perform.

Note: Currently, *Spacing* check is the only type of design rule check that is supported.

7. In the *Value* text box, enter the minimum spacing value (in microns).
8. Under the *Object 1* group box, use the drop-down lists to choose the *Type* of object, the *Layer,* and the *Net* to which the design rule check should apply. The choices vary depending on the design database with which you are working.
9. Repeat Step 8 for the *Object 2* group box to apply the DRC to a second group of objects.
10. Under the *Marker* group box, specify the *Shape*, *Size* (in microns), *Color* (RGB), and *Transparency* that you want to use for the DRC markers.

Deleting Design Rules

To delete all design rules:

➤ Click *Clear Rules*.

All the rules in the *Name* drop-down list box under the *DRC Rule* group box are deleted.

Saving and Reusing 3D DRC Rules

You can export the 3D DRC definitions as an external file and reuse them across multiple designs. Only one type of rule is supported: `Spacing`.

To save a design rule:

➤ In the *DRC Rules* tab of the 3D Viewer Design Configuration dialog box, click *Save Rules*.

Your current rule configuration is always saved within your current design.

To load a design rule:

➤ In the *DRC Rules* tab of the 3D Viewer Design Configuration dialog box, click *Load Rules*.

Understanding the 3D DRC Rules File

The 3D DRC rules file is an XML file. It is generated and saved under the default file name `view_3d_rules.xml` when you click *Save Rules*. The XML file consists of sections that start with the header `<spacing_rule>`. Each section defines an individual spacing rule. Each rule definition accepts exactly two objects for which the air gap on a specific layer is

measured and compared to the specified constraint value. For instance, you may want bond fingers on the top substrate layer to be spaced a minimum of 50 microns apart. This defines a spacing constraint.

The following is a sample of a rules file that defines two “bond wire to bond wire” spacing rules.

```
<?xml version="1.0" encoding="ISO-8859-1" ?>

<!DOCTYPE drc_rule_deck SYSTEM "cdn3ddrc.dtd">

<drc_rule_deck>

  <spacing_rule>

    <name>Wire2Wire_Top</name>

    <value>300 UM</value>

    <object>

        <type>Wire</type>

        <layer>WB1</layer>

        <net>ALL</net>

    </object>

    <object>

        <type>Wire</type>

        <layer>WB1</layer>

        <net>ALL</net>

    </object>

  </spacing_rule>

  <spacing_rule>

    <name>Wire2Wire_Bottom</name>

    <value>500 UM</value>
```

```
    <object>
        <type>Wire</type>
        <layer>WB2</layer>
        <net>ALL</net>
    </object>
    <object>
        <type>Wire</type>
        <layer>WB2</layer>
        <net>ALL</net>
    </object>
  </spacing_rule>
</drc_rule_deck>
```

Viewing and Analyzing 3D DRC Results

You can view the 3D DRC results in two different ways in 3D Viewer:

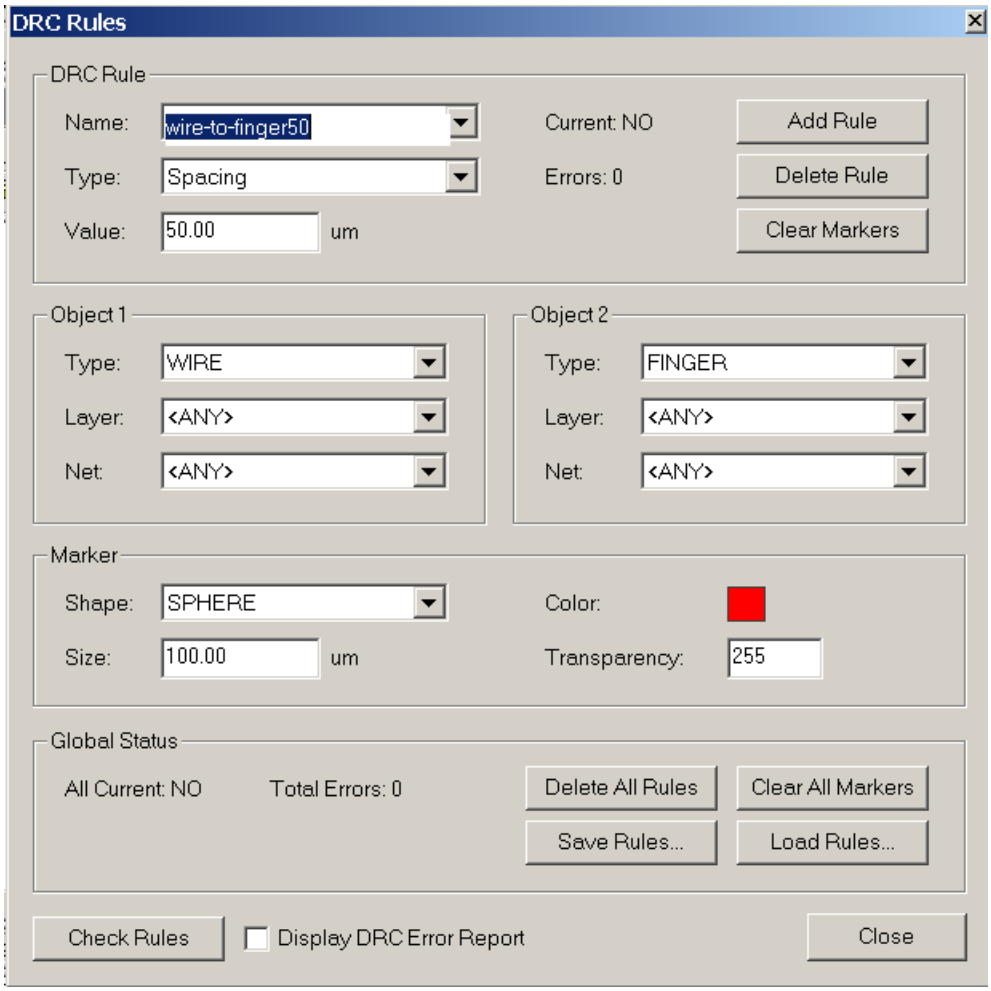
- 3D DRC Error Display: Each DRC error is displayed over the 3D design as a small error sphere or cube. You can control the size, shape, color, and transparency of the error marker that is associated with each rule. Be sure that you follow the recommendation that the *Marker Size* match the spacing value.
- 3D DRC Error Report File: A tabular text file (ASCII format) is generated. This report file summarizes the rules, shows the location of violations, and identifies the types of violations.

Viewing Error Markers in the Design

To view the 3D DRC error markers in the design:

1. From the *DRC* menu, choose *Rules*.

The DRC Rules dialog box appears.



2. Specify the rules that you want to apply, as follows:

This command...	Does this...
Name	Specifies the name of the design rule being modified. You can enter any name. The drop-down list shows all of the rules that are currently defined.
Type	Specifies the type of rule to define. Only one type of rule is supported: `Spacing`.
Value	Specifies the value of the rule. For spacing rules, this is the clearance required between the two objects specified by the Object filters. The default value is 100 microns.

This command...	Does this...
Add Rule	Adds a new DRC constraint to the current rule deck. The default rule name that is assigned automatically is “n”, where n is the number of rules in the design plus 1. For example, the first rule is named “1”, the second is named “2”.
Delete Rule	Deletes the current rule out of the rule deck.
Clear Markers	Removes previous DRC markers for the current rule that is being edited.
Object Type	Specifies the type of object to compare. The default object is *Bond Wire*. You can choose the following types from the drop-down list: *Bond Wire*, *Bond Finger*, *Shape*, *Cline*, and *Symbol*.
Object Layer	Specifies the layer on which to compute the object’s extents. The default is *<ANY>*, meaning that all layers with geometries on them are used. You can choose any layer from the drop-down list. The drop-down list also contains an entry for each bond wire profile since these are pseudo-layers representing the heights of the wires.
Object Net	Specifies the net to filter, if you want to apply net filtering for this rule. You can choose any net from the drop-down list.
DRC Marker Shape	Specifies the shape (*Sphere*, *Cube*) for the DRC markers. The default is *Sphere*.
DRC Marker Size	Specifies the size for the DRC markers. The default value is the same as the spacing value.
DRC Marker Color	Specifies the color for the DRC markers. The default is `255 255 255` (White). (White provides maximum contrast against a black background.) You can choose a predefined color from the drop-down list.
DRC Marker Transparency	Specifies the transparency of the DRC markers. This value ranges from `0` (invisible) to `255` (completely opaque). The default value is `128`.
Delete All Rules	Deletes all the rules currently specified in the rule deck.
Clear All Markers	Removes all previous DRC markers for all the rules.

This command...	Does this...
Save Rules	Saves the current set of rules for reuse in this or other designs. This saves to an XML format file (see Understanding the 3D DRC Rules File), with the default file name `<your_design_name>.xml`.
Load Rules	Loads an existing rule deck from an XML file stored on disk. These rules are loaded on top of your existing rules, allowing for hierarchical rule sets.
Check Rules	Runs the 3D rules check process.
Display DRC Error Report	Generates and displays the DRC error report, if enabled.
Close	Closes the dialog box without executing the check process.

3. Click *Check Rules*.

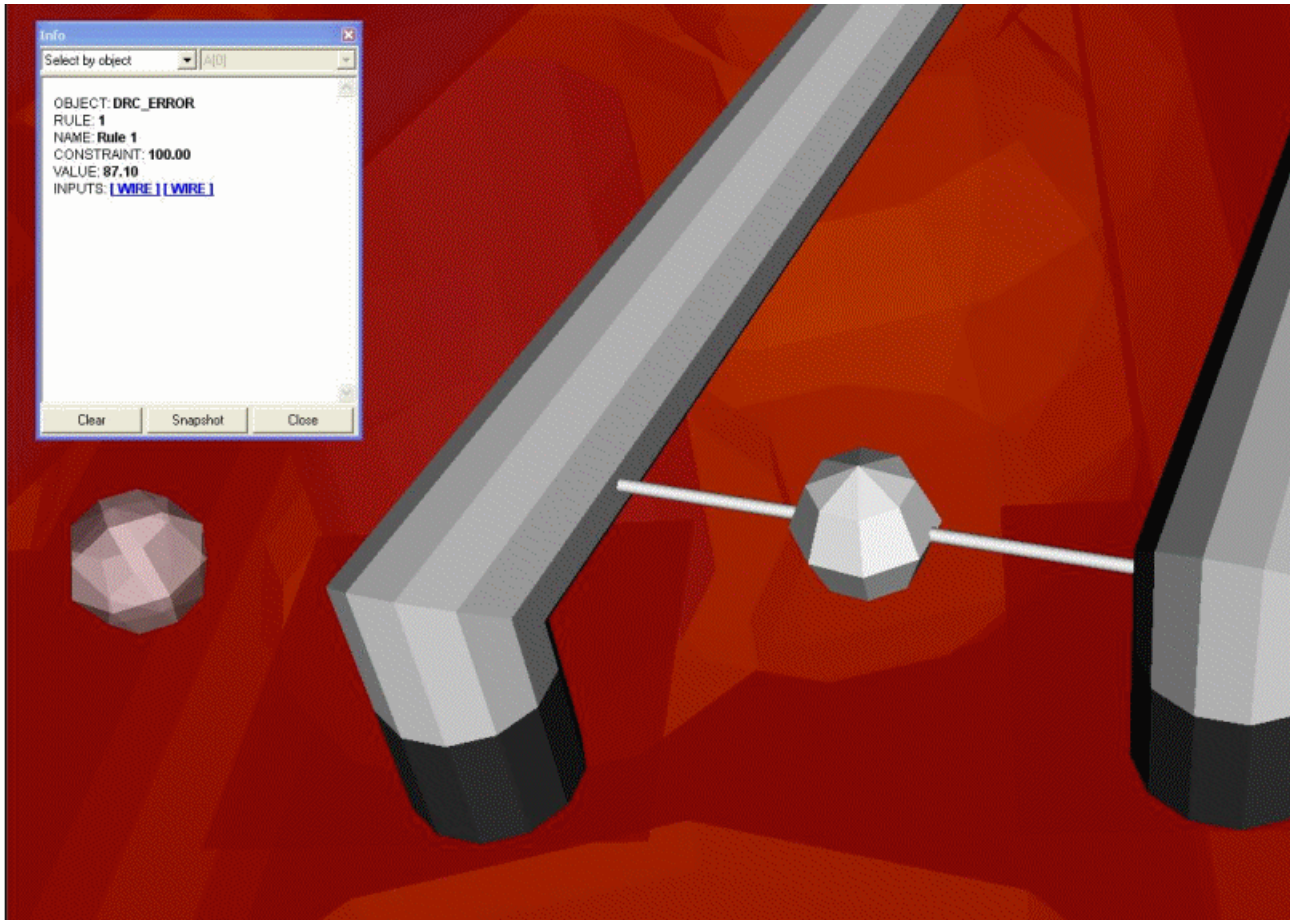
If any violations are found, 3D DRC markers appear on the 3D model.

Note: When you click *Check Rules*, any existing DRC markers are removed and are replaced by the new markers.

4. Use the zoom and pan functions to focus on any DRC markers that appear in the design. You may also want to turn off the display of certain layers, or adjust the Z scale, to make the DRC markers clearly visible.

The following illustration shows an example of a 3D DRC marker. If you click on a DRC Marker in Info mode, a narrow 3D tube appears indicating the two closest points in

violation. As you click another marker, the 3D tube moves to indicate the violation on that marker.



Tip

You can get detailed information about a particular error marker, such as: the rule that is violated, the actual value vs. the limit, and the identity of the two objects that are causing the violation. To do this, first choose *Info* from the *Tools* menu, then click on the error marker you want more information about. The details about that marker appear in the *Info* table.

Displaying the Error Report File

To display the 3d DRC error report file:

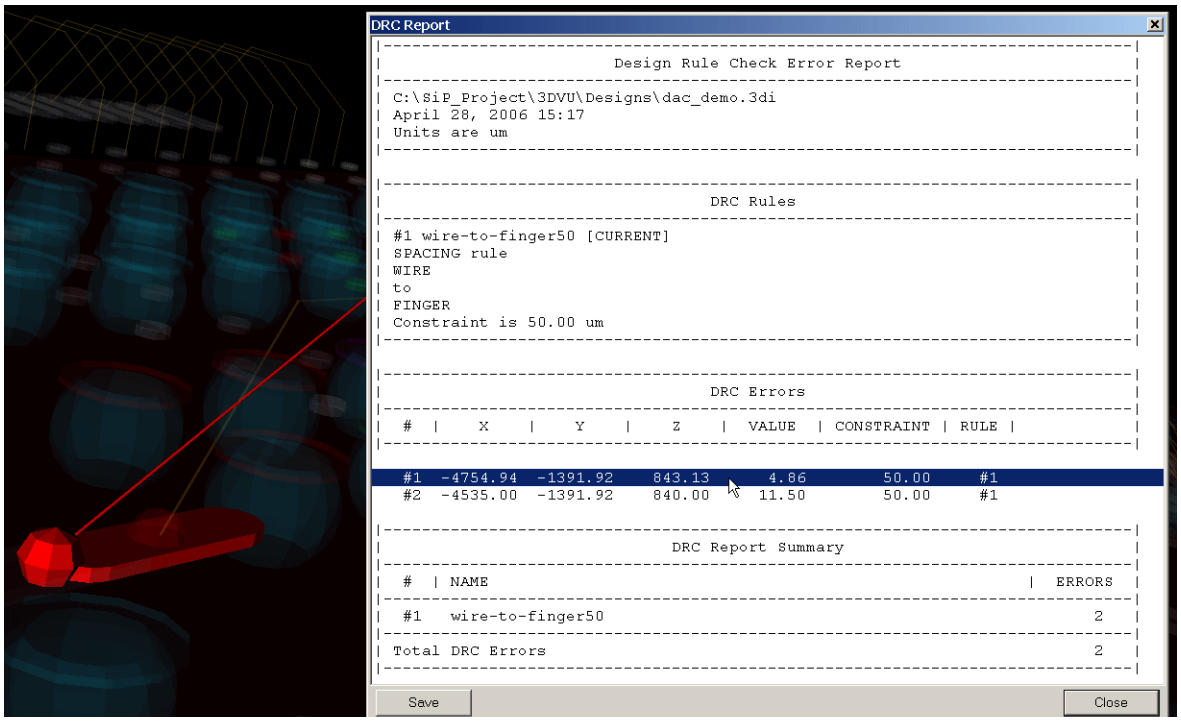
1. From the *DRC* menu, choose *Report*.

The DRC report file appears if one has been generated previously. Otherwise, a blank report file appears.

Note: To generate a DRC report, you must enable the *Display DRC Error Report* option in the *DRC Rules* dialog box when you perform a rules check.

2. Click on a violation in the report file to highlight that violation in the View window.

   The violation is highlighted.



3. Click *Save* in the *DRC Report* window to save the file as a tab-delimited ASCII text file.

   By default, the report file is saved in your current working directory as `<design filename>.txt`. The file lists the X, Y, and Z coordinates of each DRC error, and shows which rule was violated along with the value that generated the error.

Exporting 3D DRC Data

You can export the 3D DRC data from 3D Viewer in the following ways:

- Print or export images of the 3D DRC markers in the same way you print or export other images of the View window. (See Printing an Image and Exporting an Image.)

- Copy and paste information from the 3D DRC report file (ASCII format) into other design project documentation.

5

Interfacing with Your Layout Tool

This chapter discusses the following:

Creating a 3Di File

A 3Di file consists of design, modeling, and DRC data derived from an IC package design database created in APD+. When you launch 3D Viewer from your layout tool (see Opening a Data File), a 3Di file is generated and loaded automatically. The 3Di file (`<filename>.3di`) is automatically saved in the same directory as the layout design file (`<filename>.mcm`), unless otherwise specified.

You can open other 3Di files from within 3D Viewer by using the *File — Open* command.

Using Script Files to Launch 3D Viewer

You can write script files in APD+ that define 3D design rules and launch 3D Viewer. These script files follow the same command syntax and structure as do all other script files that are used with your layout tool. (For more information about script files, access the user documentation from the *Help* menu of your layout tool.)

Updating the Package

After you make changes in the 3D Viewer, you need to update the package in the layout tool.

Note: The command is available only if you launched the 3D viewer from within APD+, and that neither the layout tool nor the viewer has opened a different design.

To update the package:

1. From the menu bar in the Cadence Design Viewer window, choose *File – Update Package*.

   The Update Package dialog box appears.

2. Check the *Wire Profile* box to update the package design with the profile assignments for the individual wires. For example, if you change a wire from Profile 1 to Profile 2 and you check this box, the change is updated in the `mcm` design.

3. Check the *Diameter* box to update the diameter of the individual wires in the `mcm` design.

4. Check the *Wire Profile Definitions* box to update the package with all wire profile definitions, both the modified wire profile definitions and new ones.

5. Check the *DRC Errors* box to update the package with the DRC errors generated in the model.

   All violations are flagged as external DRC markers in the main `mcm` design. You can clear these markers by opening the 3D Viewer in your layout tool, choosing the *DRC* tab, and clicking the *Clear DRC Markers* button.

6. Click *OK* to update the specified objects or attributes in the `mcm` design.

Appendix 1 - 3Di Interchange Format v1.7

Header

3Di <version> <units> <precision>

Example

```
3Di 1.700000 microns 2
```

Comment

<comment>

Example

```
# 3Di file produced by Cadence 3D Design Viewer v1.1.7 [April 6, 2012]
```

TABLE HISTORY

<tool> <version> <date and time>

Example

```
TABLE HISTORY
1 Allegro Package Designer+ vD075.17.4 [Mon Mar 12 20:19:42 2012]
```

Table Prototypes

TABLE <name>

Example

```
TABLE OBJECT
```

DRC Rules

DRC_RULES

RULE <number> <name>

<type> <n_inputs> <parameters>

GEOM

<table> <index>

.

.

GEOM

<table> <index>

.

<marker>

```
DRC_RULES
RULE 1 Wire-Wire Spacing Rule
SPAC 2 25.000000
GEOM
OBJECT 6
GEOM
OBJECT 6
GEOM
COLOR 30
SPHE 0.000000 0.000000 0.000000 50.000000 12
```

Wire Profiles

WIRE_MODELS

WIRE_MODEL <number>

<name> <radius> <points> <direction>

STEP

HOR [X] | ANG <value>

VER [X] | ANG <value>

.

.

STEP

.

SWITCH

.

STEP

.

```
WIRE_MODELS

WIRE_MODEL 1
PROFILE 1 12.500000 4 FORWARD
STEP
HOR 0.000000
VER X 0.250000
STEP
HOR 500.000000
VER X 0.000000

JEDEC_5P 12.700000 5 FORWARD
STEP
ANG 85.000000
VER 250.000000
STEP
HOR X 0.125000
VER -50.000000
SWITCH
STEP
HOR X 0.500000
ANG 5.000000
```

Geometries

GEOMETRIES

GEOM

<table> <index>

<attribute> <value>

.

EXTR <n_children> <z1> <z2>

2DPG <n_points>

<x1> <y1> ... <xn> <yn>

2DCR <x> <y> <radius> [<sides>]

GEOM

<table> <index>

<attribute> <value>

.

WIRE <radius> [<sides>]

3DPL <n_points>

<x1> <y1> <z1> ... <xn> <yn> <zn>

GEOM

<table> <index>

<attribute> <value>

.

SPHE <x> <y> <z> <radius> [<sides>]

GEOM

<table> <index>

<attribute> <value>

.

BARL <x> <y> <z> <radius> <r_lower> <r_upper> [<sides>]

GEOM

<table> <index>

<attribute> <value>

.

PATH <vertices>

<vertex_x_coordinate> <vertex_y_coordinate> <width_at_vertex>

.

```
GEOMETRIES

GEOM
OBJECT 1
ID 91452306
LAYER 5
NET 492
COLOR 2
EXTR 1 0.0 10.0
2DPG 4
500.0 500.0 580.0 500.0 580.0 580.0 500.0 580.0


GEOM
OBJECT 6
ID 91453345
LAYER 5
NET 492
COLOR 2
```

```
WIRE 12.5 16
3DPL 5
-4620.0 1470.0 0.0 -4620.0 1470.0 250.0 -4859.833 1540.57 400.0
-5195.599 1639.368 400.0 -5475.0 1721.58 0.0


GEOM
OBJECT 7
ID 91454293
SPHE 5000.0 4000.0 -250.0 300.0 16

GEOM
OBJECT 7
ID 93006682
BARL 3000.0 8000.0 -250.0 500.0 100.00 100.00 16
```

DRC Markers

DRC_MARKERS

MARK

RULE <rule_number>

CONSTRAINT <rule_parameter>

VALUE <violation_value>

INPUTS <violation_object> ...

3DPT <x> <y> <z> [<in1_x> <in1_y> <in1_z> <in2_x> <in2_y> <in2_z>]

```
DRC_MARKERS
MARK
RULE 1
CONSTRAINT 50.0
VALUE 48.2395
INPUTS ID:91233200 ID:91279772
```

```
3DPT 2847.778014 -974.189991 398.196888
```

Tables

TABLE <name>

<index> <value>

```
TABLE OBJECT
1 PIN
2 BOND FINGER
3 VIA
4 METAL
5 TRACE
6 WIRE
7 SOLDER BALL
8 SUBSTRATE
9 DRC MARKER

TABLE LAYER
1 BOTTOM_CONDUCTOR
2 VDD
3 VSS
4 TOP_CONDUCTOR
5 WIREBOND2
6 WIREBOND1

TABLE NET
1 DIE1_10
2 DIE1_100
3 VSS
4 VDD

TABLE COLOR
1 255 255 255
```

```
2 14 210 255
3 255 121 203
```

Appendix 2 - Troubleshooting Guide

This appendix discusses the following:

Troubleshooting Common Problems

The following list contains descriptions of common problems that you may encounter while using Cadence 3D Design Viewer, along with suggestions for resolving those problems.

Missing Bond Finger Attributes

The bond fingers do not have bond finger attributes. If you use the Wire Bonding module, you should not encounter this problem. This situation may result if you are placing the wires manually as clines.

Result: The bond fingers may not be exported to 3D Viewer. Or, if they are exported to 3D Viewer, they may not be properly identified as bond fingers.

Solution: Make sure that all bond fingers have the correct attributes. Use *Edit — Property* to add the attributes, then regenerate the 3Di file.

Unassigned Wire Height Group for Wire Bonds

Wire bonds are not assigned a proper Wire Height Group. The WHG attribute (property name: `WIREBOND_PROFILE_NAME`) is new and most users have no need to use it or know what it is. Therefore this is the most common problem one sees.

Result: Only one wire model can be assigned to all of the wires. The 3D view will not be very accurate.

Solution: Use *Edit Property* to attach the Wire Height Group property to all wires. Then, make sure that the attribute value (the group that the wires belong to) is appropriate.

Multiple Designs in the Same MCM File

Multiple variants of the same component exist in the same MCM file.

Result: In the 3D View, you will see too many die bodies and wires.

Solution: Do not put multiple variants of the same component in a single MCM file. If you do, make sure that you know which wire layers and die symbols go together and deselect them. You can turn off the display of them in the *Visible* column of the Layers dialog box.

Unknown Stackup Order in APD+

You do not know the height or stackup order of the die when you are in APD+.

Result: The 3D results make no sense. The die bodies intersect each other and the wires.

Solution: You must keep track of each die thickness and the vertical order in which the die are stacked up. This information is essential to building a correct 3D model.

Note: Refer to the documentation on the Die-stack Editor for making any die ordering changes.

Bad Wire Bond Profile Parameters

Configure all wire profiles using the *Route – Wire Bond – Settings* command and choosing the *Wire Profile Editor* prior to invoking the 3D viewer. Because the dialog box provides a preview panel, you should not have any issues regarding scrambled wire profiles.

Slow Display Performance

The 3D display is very slow.

Result: Panning, zooming and orbiting are very sluggish.

Solution: You may have too many objects. If you are examining wires, you do not need to export all substrate layers; only the top metal is required. Re-export the data using only the required elements.

You may have set the display resolution too fine. The circle setting may be too fine. For example, a value of 16 (16 sides per circle) is very fine but will create thousands more faces than a value of 8. Specify the resolution when you generate the export file. You cannot specify the resolution in 3D Viewer.

You may have a slow graphics card or CPU. A high quality graphics card and a high speed CPU will greatly improve the display time. Most graphics cards released in the last two years are fast and have good OpenGL support. Some graphics chips that are integrated on the motherboard do not offer the best performance.

Important

VTK requires the use of graphics cards which support OpenGL. The recommended graphics system is AGP4X with full support of OpenGL and at least 64 MB of dedicated (not shared) video memory.

Getting Additional Help

If you cannot resolve a problem, the following resources should provide you with additional help.

Cadence Online Support

Cadence Online Support online customer support gives you answers to your technical questions. Find the latest in Interim Software Rollups (HotFixes), case and product change release (PCR) information, technical documentation, solutions, software updates and more.

Note: To register on Cadence Online Support you will need your email address and your host-ID or serial number.

To access Cadence Online Support, go to:

http://support.cadence.com/

Customer Support

Cadence Customer Support is available online. There are specific email addresses, phone and FAX numbers for different regions of the world. The web site provides contact details for your particular area.

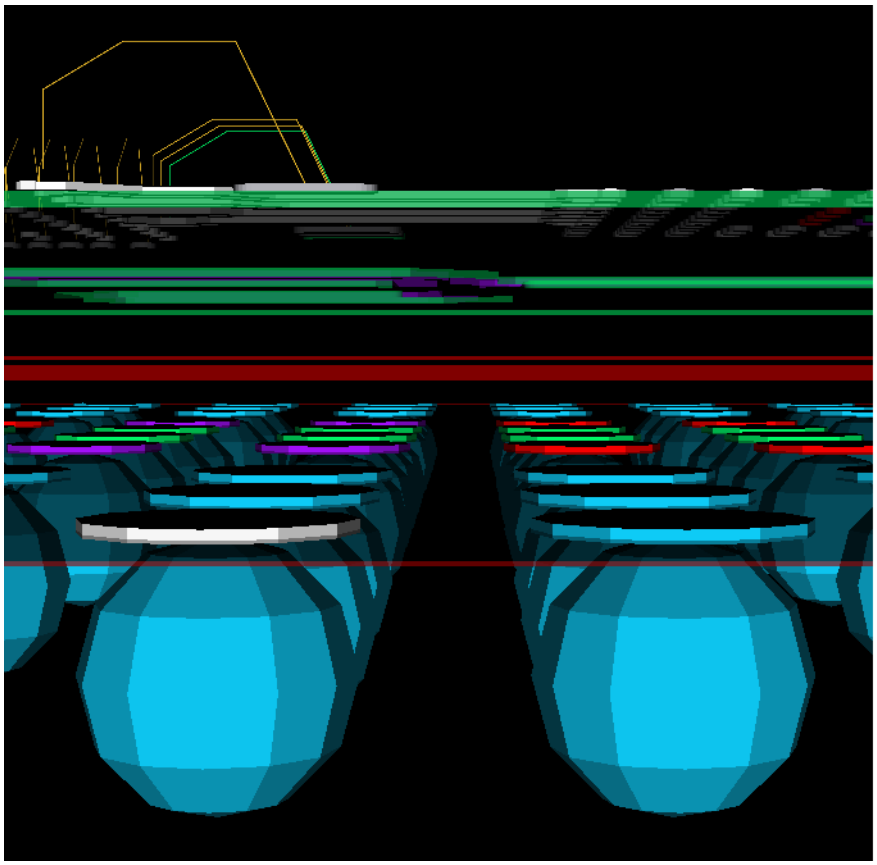
To contact cadence customer support, go to:

http://support.cadence.com

Glossary

This chapter provides definitions for technical terms and acronyms that are used throughout this user guide. The following illustration explains certain terms as they are applied to a sample IC package design.

Cross-section of an IC Package Design



Numerical

3Di	The file format extracted from an APD+ Layout database and passed to 3D Viewer. The 3Di format is a 2 ½ dimension database – except for wires, most objects are defined in the XY plane and assigned an extrusion thickness and Z position. This file also includes the wire models, model parameters and optional DRC rules.

D

Die body	The silicon body of the die.
DRC	Design Rule Check. Design rule checking in 3D Viewer is applied to either wire-wire or wire-extrusion, where extrusion is any other 3Di database object (pin, via, symbol, trace, and so on).

O

OpenGL	A graphics system/interface used by VTK to draw to the computer display screen. Most graphics cards for PCs have good support for OpenGL; high-end UNIX workstations also support OpenGL in dedicated hardware. Low-end UNIX workstations and PCs with embedded graphics often use the main CPU to implement OpenGL solely in software, which results in slow graphics response.

S

Symbol	The physical boundary of the IC package (BGA), die body, or discrete component body.

V

VTK	Visualization ToolKit. This is open source software used to display 3D models. 3D Viewer uses VTK as its primary 3D display engine with all 3D computations being performed using proprietary Cadence algorithms.

W

Wire Group	A group of wires that share a common Wire Model and parameters for that model. In your layout tool, each wire can be assigned a wire group attribute.
Wire Model	A parameterized model used to define a wire's path through the Z (vertical) plane. There are hardwired models (currently 4 and 5 point models) and a newer user-defined model which is a series of piece-wise linear formulas.