cadence®

Allegro® Free Physical Viewer

Product Version 23.1
September 2023

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

Contents

1

Getting Started with Allegro Free Physical Viewer

The Cadence® Allegro® X/OrCAD® X FREE Physical Viewer a free downloadable application that can be used by design engineers, manufacturing units, and by customers to review the design files at different phases of design. This application allows you to view and plot databases from Allegro X PCB Editor, OrCAD X PCB Editor, Allegro X Package Designer, and Allegro X PCB SI technology.

It enables you to review the physical design, monitor its progress, assess fabrication and assembly requirements to ensure design specifications are met. Allegro Free Physical Viewer provides all the key features of layout editors required to review a design. You can pan, zoom, highlight, search, assign color to objects and preview the design in 3D using Allegro Free Viewer.

Opening Design in Allegro Free Physical Viewer

To open a layout design in Allegro Free Physical Viewer, follow these steps:

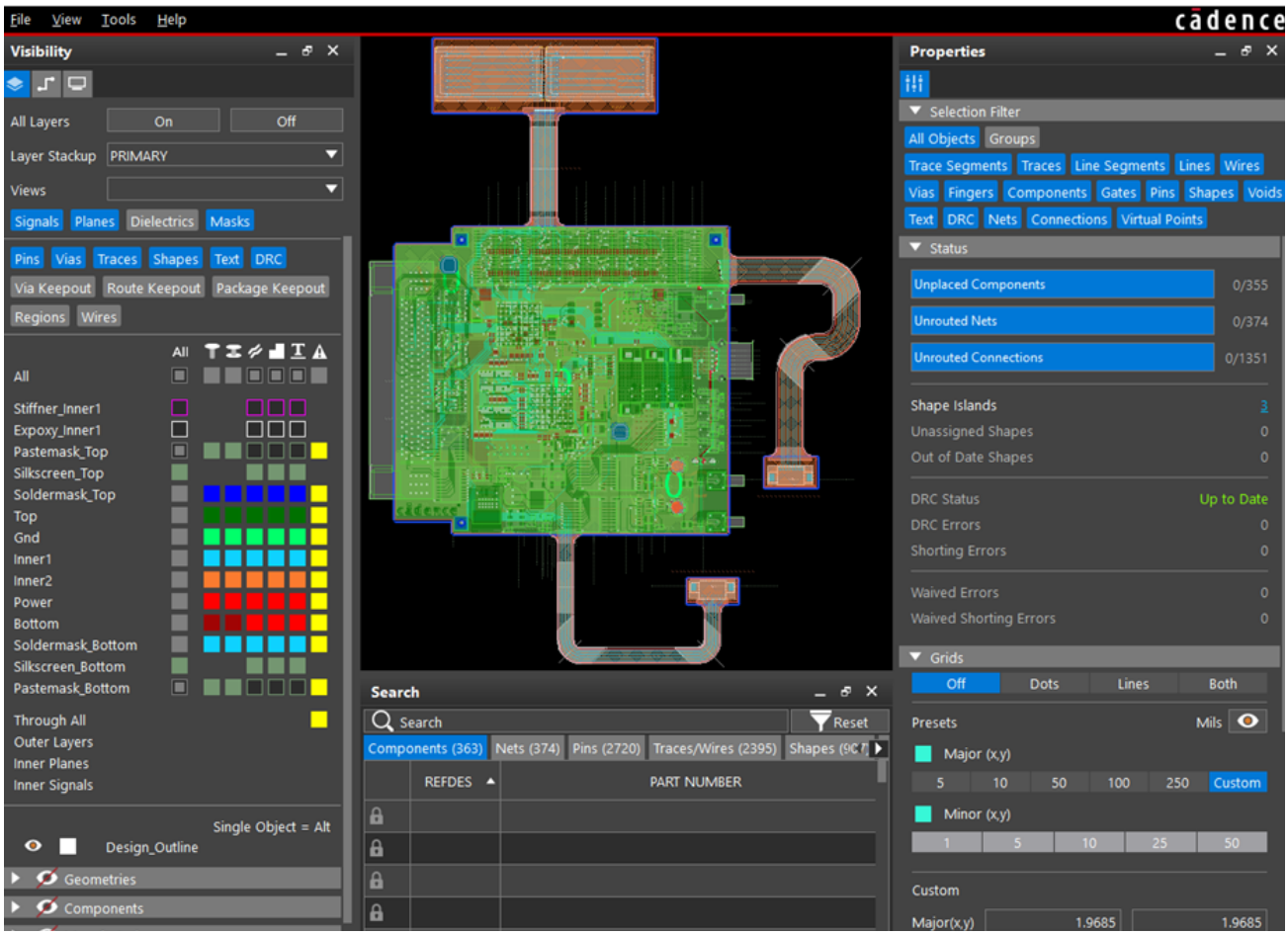
1. Launch the free viewer by selecting *Start – Cadence PCB Viewers 2023 – PCB Editor Viewer 2023*.
2. To launch the free viewer from the command line, type `allegro_free_viewer` in a command window and press `Enter`.

   Allegro Free Physical Viewer window opens with a blank design.
3. Choose *File – Open* and select a file to open in free viewer. The supported designs file types are board (`.brd`), design partition (`.dpf`), module definition (`.mdd`), and symbol drawing (`.dra`).

Tip

You can choose to open default Cadence board design file (`Cadence_Demo.brd`) from the installation hierarchy. This design file is located at *`<installation_hierachy>\share\pcb\examples\board_design`*

The design is always opened in read-only mode and is non-editable. The following image displays default Cadence board design file (`Cadence_Demo.brd`) in Allegro Free Physical Viewer.

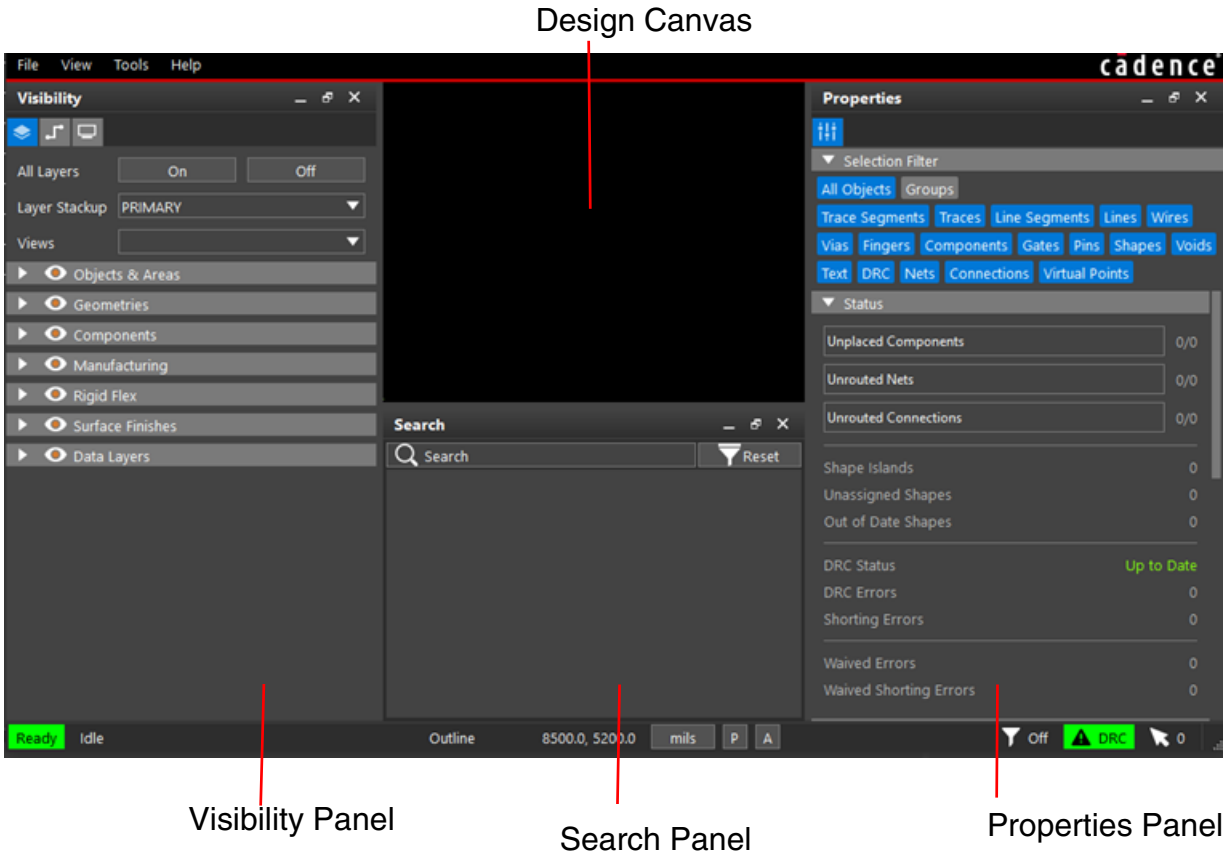


Allegro Free Physical Viewer User Interface

The user interface of Allegro Free Physical Viewer can be divided into three key sections. The left panel is the visibility panel, that has controls for managing the display settings and for modifying the visibility of various design objects such as nets, layers, and objects.

Design canvas is at the center of the application window and displays the physical layout. All your design activities are completed in this section. Just below the design canvas Search window is displayed that lists all the design objects in a spreadsheet format.

The right most panel is the properties panel, that displays the properties and constraint based on the current selection in the canvas.



This section walks you through the main elements of user interface. Understanding the architecture would help you easily interact with the application in a natural and intuitive way.

Visibility Panel

When reviewing a design you can enable or disable the visibility of each design object explicitly using the Visibility panel. It has three tabs that lets you control:

- Layers  : Visibility of layers and assign color to layers and objects
- Nets  : Visibility of nets and assign color to nets
- Display  : Display settings

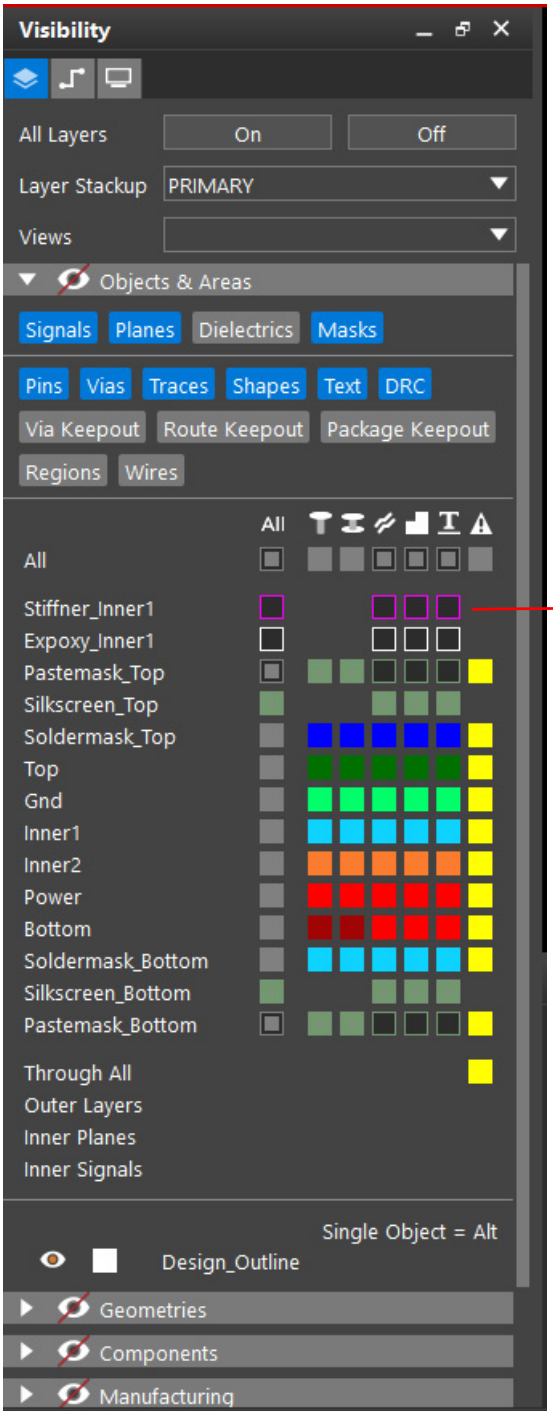
Layers Tab

When you open a design, by default the Layers tab is opened in the Visibility panel. All the layers and objects are visible in the design canvas and the primary stackup is selected. The subsequent collapsible sub sections *Objects & Areas, Geometries*, *Components, Manufacturing, Rigid Flex, Surface Finishes* and *Data Layers* display only those layers and objects that are defined in the primary stackup.

At the top, the following three options are provided that can be selected exclusive of each other:

- *All Layers:* Turns on or off the global visibility
- *Layer Stackup:* Selects stackup other than the primary stackup.
- *Views*: Selects existing view. Lists film generated from artwork.

Depending on which stackup and view is selected the corresponding sub sections dynamically update and displays relevant layers and objects.



Tip

To reset to default view and layer stackup, click the *On* button for *All Layers*.

The toggle eye icon  at the header of sub section turns on or off the visibility of all the items under that sub section. Visibility of any object can be simply turned on or off by enabling or disabling the color box associated to objects for any layer. The color box also indicates the custom color assigned to the object on that layer. Right-click color box to open a color palette and you can choose a different color that gets immediately assigned to the object on that layer.

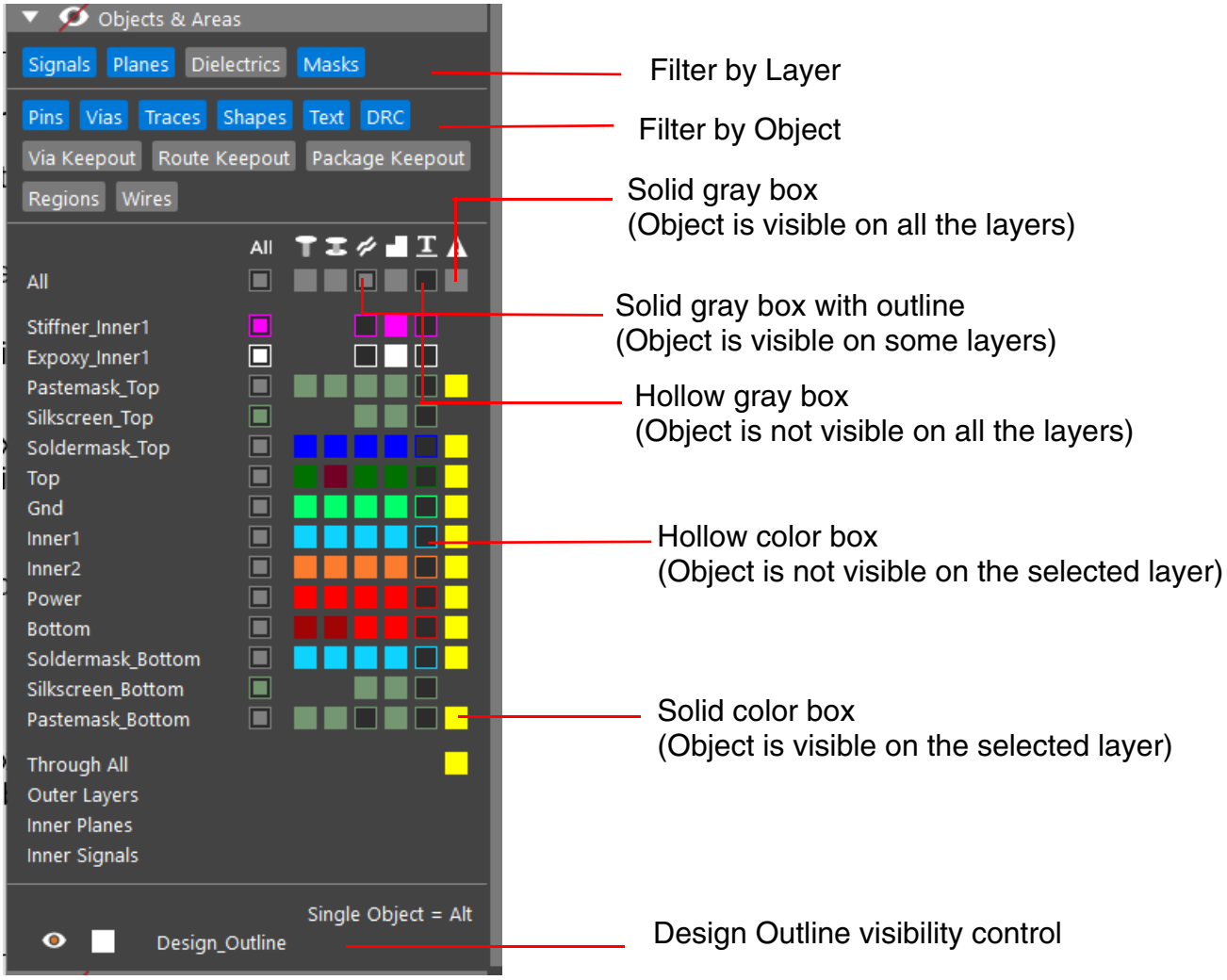
Objects and Areas Sub Section

Objects & Areas sub section lists all the design layers including mask and silkscreen layers. It has filters for both layer types and object types. You can set layer filter for *Signals, Planes, Dielectrics*, and *Masks*. Using color box you can turn on or off individual objects on design layers. You can also assign color and turn on or off visibility of each object per layer.

The color box acts as a toggle and controls the visibility of an object on a layer. To change the color assignment of any object right-click the color box. The following is a simple explanation of color box representation in:

- Solid color box: Indicates that the object is visible on that layer.
- Hollow box with colored outline: Indicates that object is not visible on that layer. The color of outline displays the color assigned to that object on the selected layer.
- Solid gray box: Indicates that object is visible on all layers. Different or the same colors are assigned to object on each layer.
- Hollow box with gray outline: Indicates that object is not visible on all the layers and have different colors assigned to each layer.

- Solid gray box with outline: Indicates that object is visible on some of the layers and have different colors assigned to each layer.



Tip

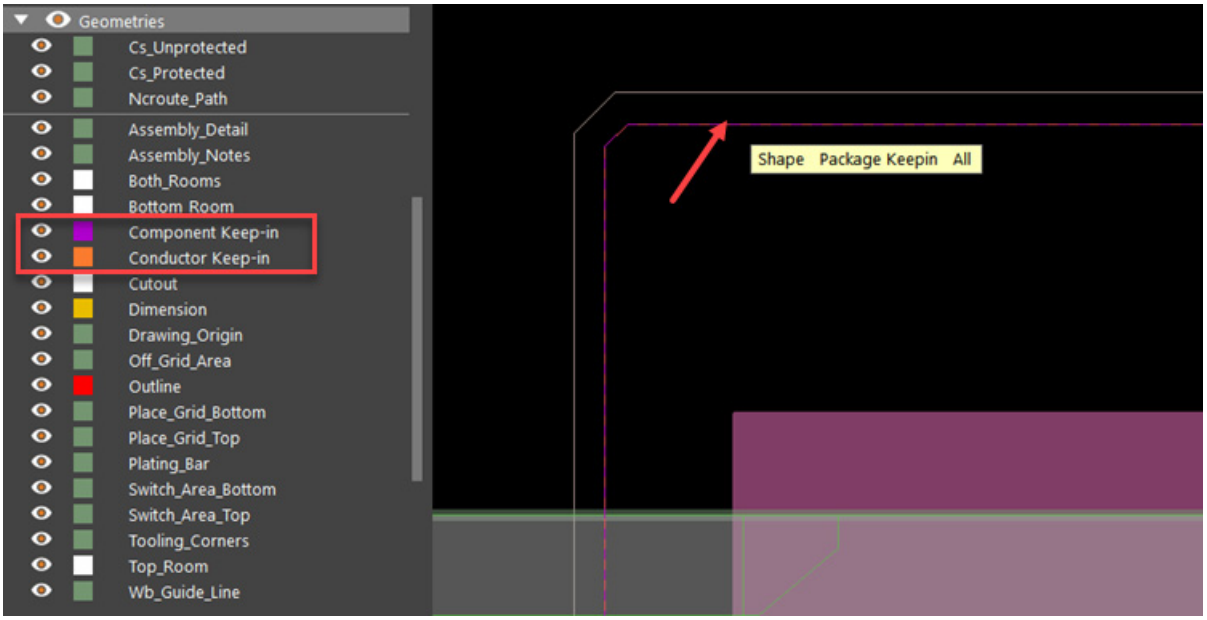
To select a single object, press `ATL` key and click the color box.

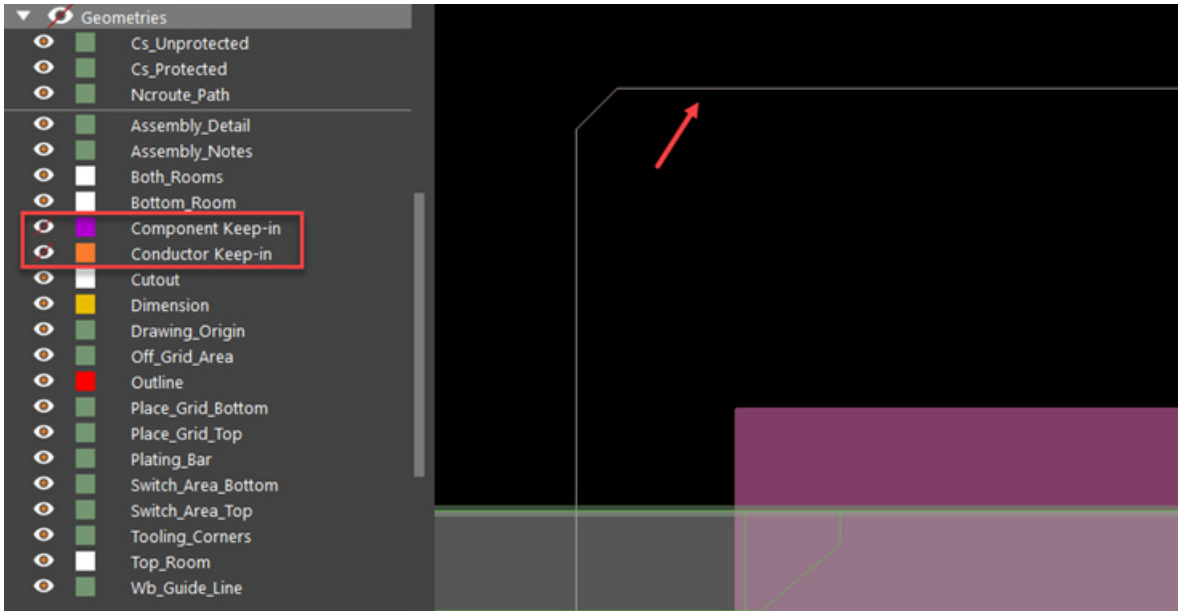
A separate toggle is provided at the bottom of the section to control the visibility of design outline.

Geometries Sub Section

Geometries sub section controls the visibility of board geometry class, such as outline, dimension, and keepin areas in the design canvas. The first part of the section displays user-

defined board geometry layers. You can also assign color at any time using the color box attached with each geometry.

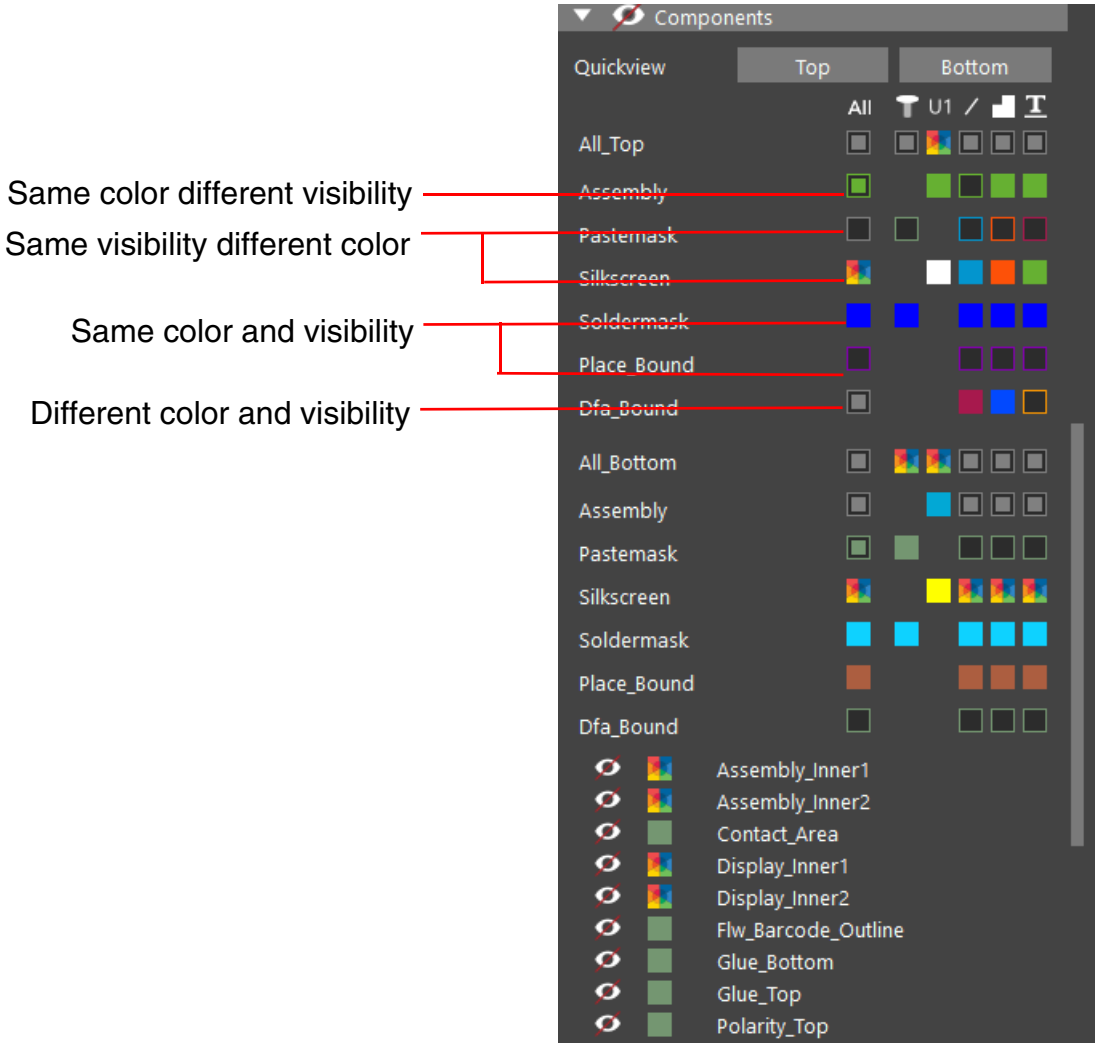




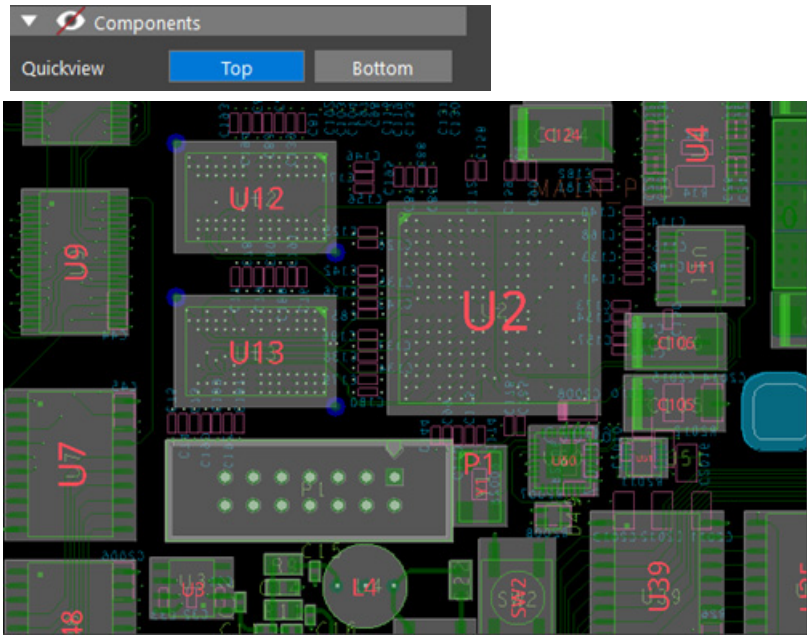
Components Sub Section

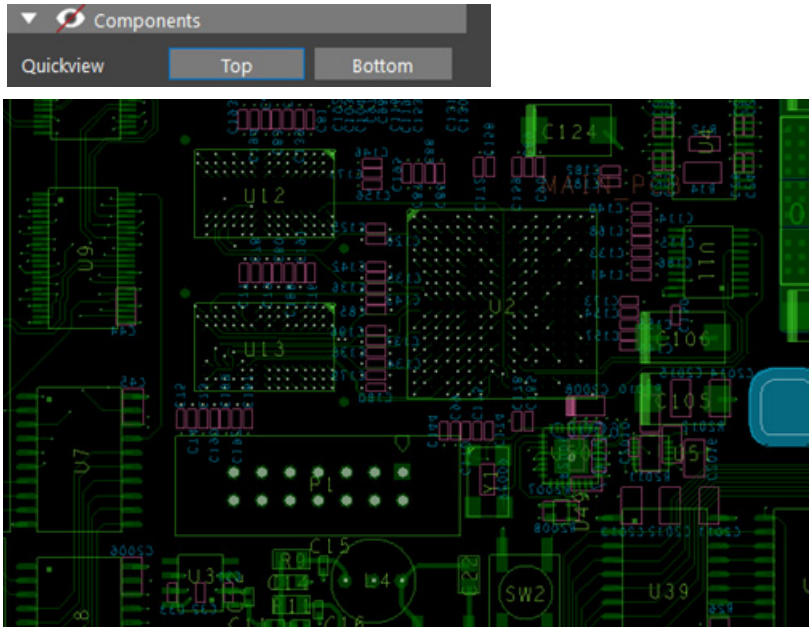
Components sub section controls the visibility of component geometry class/subclass and package geometry class/subclass. This section consists of two tables of layers representing

top and bottom component visibility controls. Layers Assembly, Pastemask, Silkscreen, Soldermask, Place_Bound and Dfa_Bound are defined as rows, and objects Pin, RefDes, Line, Shape and Text are columns.



It also provides a *Quickview* option to control the visibility of components on TOP and BOTTOM layers, respectively. When enabled, this option shows place bound shape of components, its pins, and dynamically-rendered reference designators.

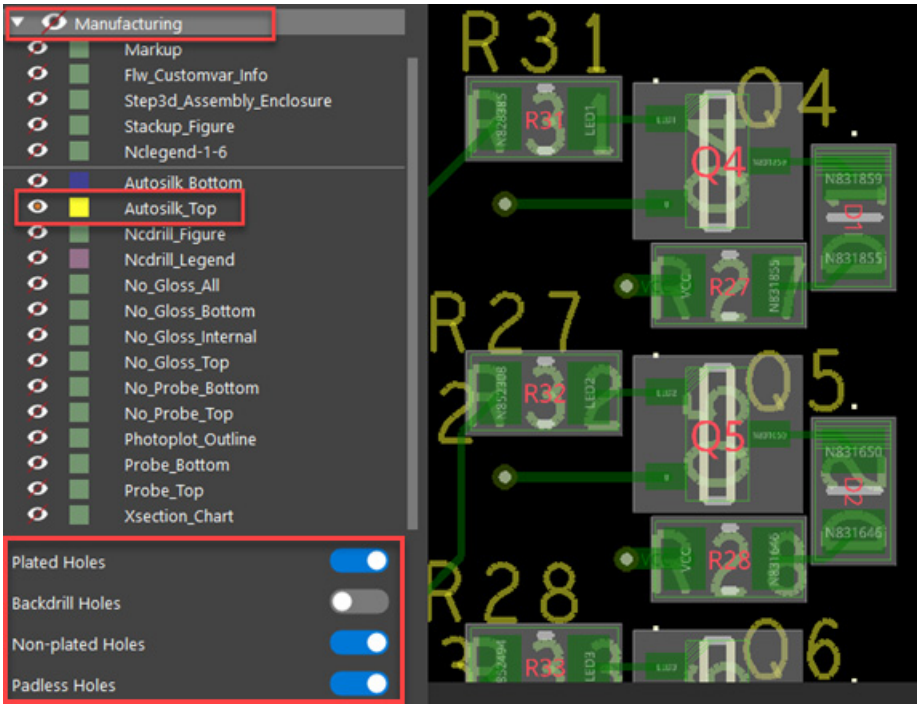


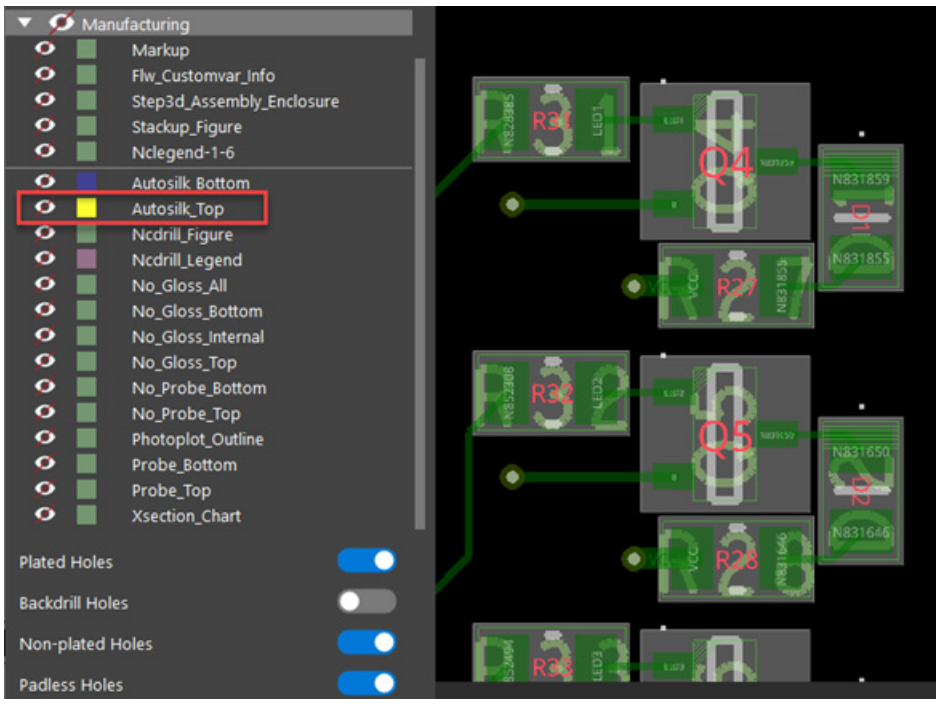


Manufacturing Sub Section

Manufacturing sub section controls the visibility of layers of manufacturing class, such as autosilk, ncdrill, and so on. The display of holes can also be controlled in the sub section. The

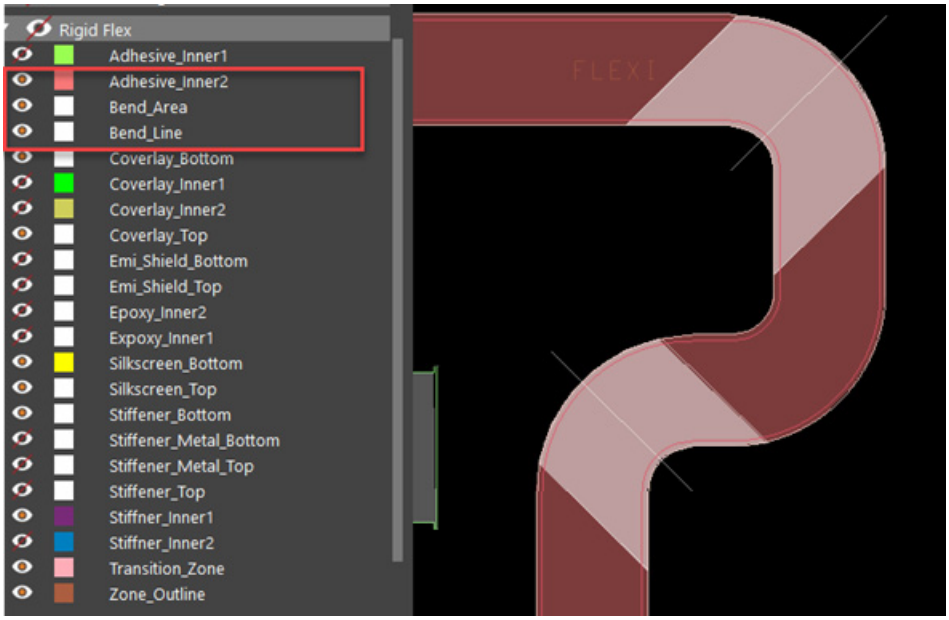
first part of the section displays user-defined manufacturing layers. You can also assign color at any time using the color box attached with each layer.

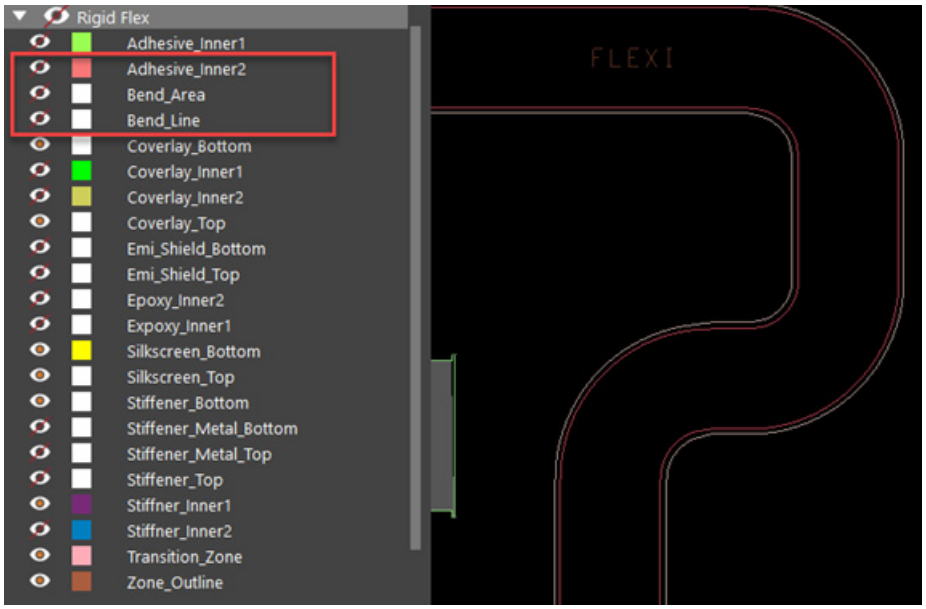




Rigid Flex Sub Section

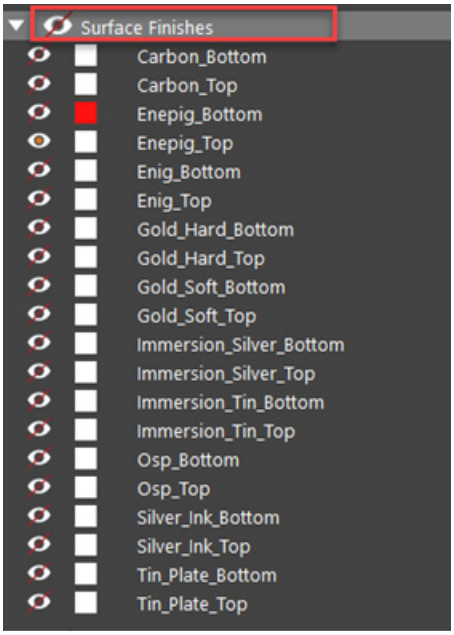
Rigid Flex sub section controls the visibility of all rigid flex layers, such as bend area, bend line, coverlay, and so on. You can also assign color at any time using the color box attached with each rigid flex layer.





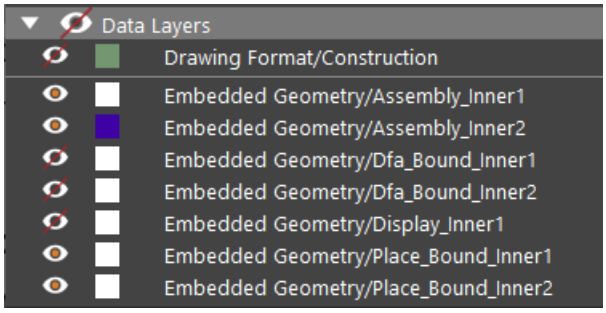
Surface Finish Sub Section

Surface Finish sub section controls the visibility of all layers added to surface finish class. The surface finish layers are added to define finished materials used on a design such as tin plating, gold, Organic Solderabilty Preservative (OSP), and so on.You can assign color at any time using the color box attached with each surface finish layer.



Data Layers Sub Section

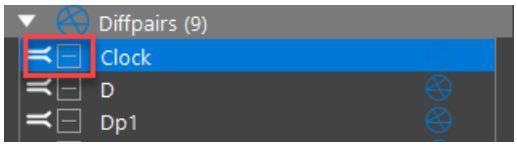
Data Layers sub section lists other classes and subclasses. It also displays layers for Drawing Format class and user-defined subclass. You can control the visibility and color settings of user-defined layers in this sub section.

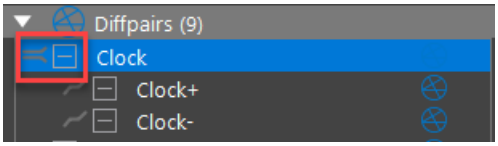


Nets Tab

The *Nets* tab lists nets by net group, match group, net class, bus, differential pair, XNet, and net in different collapsible sub sections. Members of each group are listed in hierarchical order all the way down to nets. In this tab, you can turn on or off the visibility and assign color to individual nets or group of nets.

Each group of net object can be identified by unique icon. Clicking the icon expand and collapse the group element to its immediate members. The white color of icon indicates that a group is expanded and shows all its members nets. When collapsed the icon turns gray.



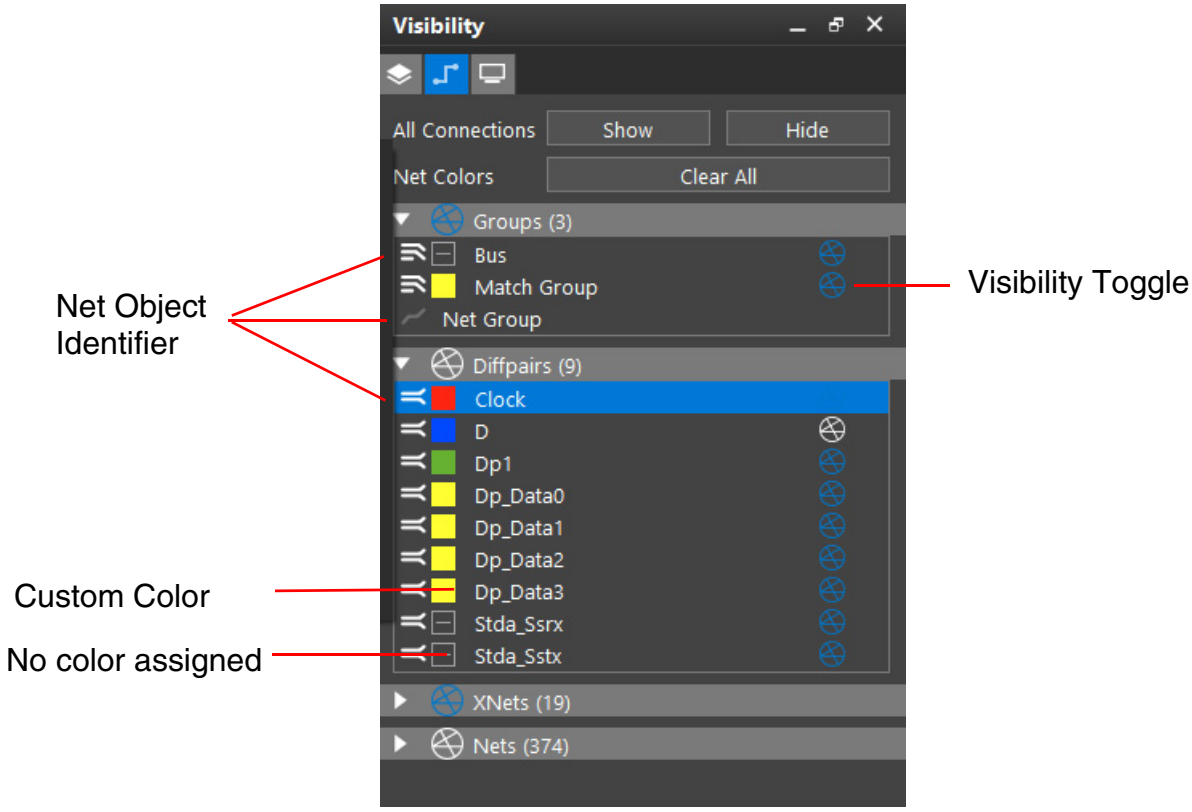


The icon  indicates that visibility state is enabled and the net connection is visible in the design canvas. The icon color changes to white when the visibility is turned off. Clicking on individual  icon toggles the connection visibility on or off of that net object. Any change in the visibility of a parent net is applied to its member nets. If the same parent net and its members are also in other *Nets* sub sections, those sections also update. The visibility of groups of nets can be controlled by toggling the header  icon.

Double-click a net object highlights that net object and zooms to its location in the design canvas.

The color box indicates the custom color state of the net object. When no color is assigned a hollow gray box with a dash inside  is displayed. Assigning color to a parents net applies to that net and all its member nets. Similarly, assigning color to a member net also updates

the color of parent net. If the same parent or member net is part of other sub sections, then the color assignment is reflected in all the sub sections of *Nets* tab.



The connection visibility and color assignment can be controlled in a design through two global controls:

- *All Connections*: Turns on or off the connection visibility (display of rats) of all the net objects in all the sub sections.
- *Net Colors*: Resets all colors and patterns assigned to net objects for custom color and highlight.

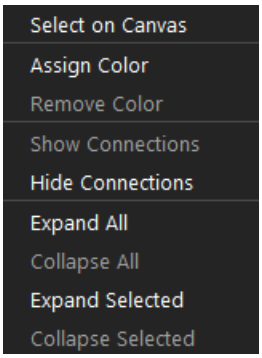
Nets Sub Sections

Nets are categorized under four sub-sections:

Groups		Lists net groups, net classes, match groups, and buses. Clicking on icon toggles to expanding or collapsing immediate members below the group.

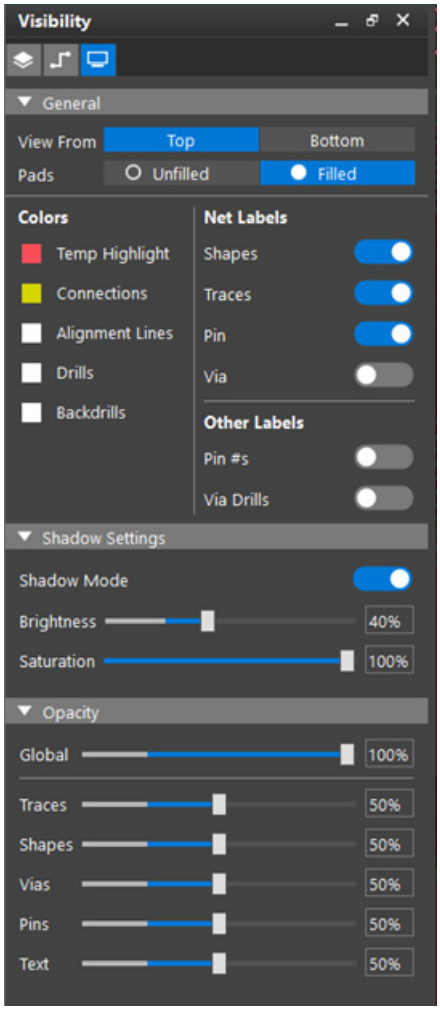
Diff Pairs		Lists differential pairs. Can be a member of *Groups*. Clicking on icon toggles to expanding or collapsing immediate member nets.
XNets		Lists XNets. Can be a member of *Groups* and *Diff Pairs.* Clicking on icon toggles to expanding or collapsing immediate members.
Nets		Lists nets. Can be a member of *Groups, Diff Pairs,* and *XNets.*

The color and visibility controls are also available as right-click options. The following options are accessible on an individual net object.



Display Tab

The *Display* tab provides settings to control the display of objects in the design canvas. You can choose display options from three sections: *General, Shadow Settings* and *Opacity*.



General Sub section

This section controls the on screen design display. You can set the following:

View From	Flips the design along the Y-axis on the drawing canvas by selection *Top* and *Bottom.* OpenGL must be enabled for this option to work.
Pads	Controls display of pads. By default pads are filled in the design.

Colors	Assigns custom color to connections (rats), drill holes, backdrill hole, alignment guides, and for temporary highlight.
Net Labels	Displays net names on shapes, traces, pins, and vias. OpenGL must be enabled for this option to work
Other Labels	Displays via label on top of each blind/buried via, indicating the from-to subclasses spanned by the via. OpenGL must be enabled for this option to work.

Shadow Settings Sub Section

Enable shadow mode and adjust brightness and saturation of non-selected objects in the shadow mode. When enabled, selected objects are not affected at all. Whereas non-selected objects are dimmed and the degree of dimming can be controlled by the brightness and saturation sliding bars. The location of slider handle is also represented as percentage value that can be edited by double-clicking the field.

Opacity Sub Section

Sets opacity for objects. You can assign varying degrees of transparency to traces, shapes, vias, pins, and text in the entire design. Sliding the bar completely to the left causes previously filled geometry, such as shapes, to display with less intensity.

The global value is always equal to the highest percentage value set for any object. The percentage value represent the equivalent of the slider handle location. You can double-click the value and enter a new value for opacity.

Note: OpenGL must be enabled for controlling opacity of the design.

Properties Panel

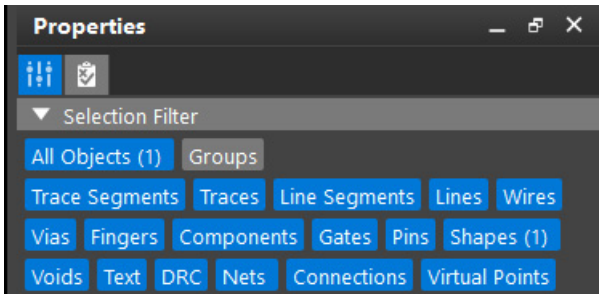
The *Properties* panel provides access to general information, attributes, and constraints. This panel contains two tabs:

- Properties 
- Constraints 

Depending on number of objects selected in canvas these tabs display information in sub sections. In both the tabs, *Selection Filter* sub section is always visible.

Selection Filter Sub Section

To select design objects for viewing in canvas you can use *Selection Filter*. You can select multiple object filters at a time. Only objects enabled in the selection filter can be selected in the canvas. The selected filters are seen in blue color and displays count for each of the objects selected. The *All Objects* option enables and disables all the objects from the filter and provides a summary count of all the objects selected in the canvas. By default, *Groups* filter is disabled.



The *Selection Filter* is available with both properties and constraint tabs of *Properties* panel. The filter is always visible even when no object is selected in the canvas.

Note: The *Selection Filter* is the same as *Find filter* in layout editors with a slight difference in the object filter names. In *Selection Filter*, connections corresponds to ratsnest, traces are clines, gates are functions, wires are bondwires, virtual points are Rat Ts, and components are symbols.

Properties Tab When No Object is Selected

When no object is selected the only *Properties* tab is visible and the *Constraint* tab is hidden. The *Properties* tab provides database-centric information in two four sections: *Status, Grids, Parameters,* and *Attributes*.

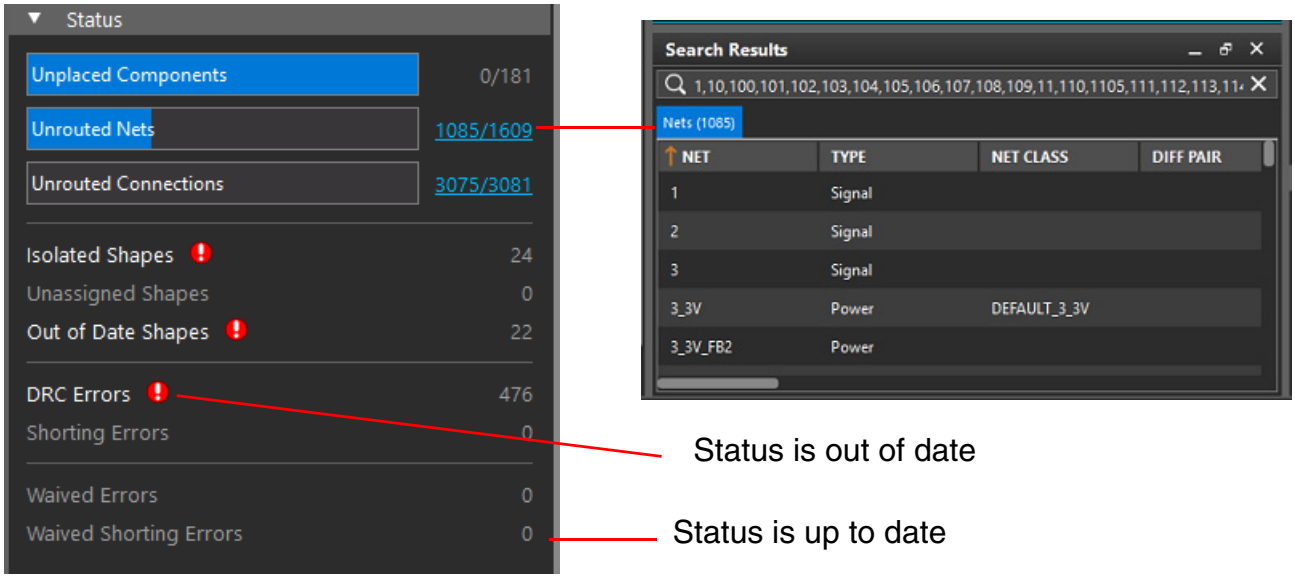
Status Sub Section

Displays design status information of the active design. This sub section provides the details on unplaced components, unrouted nets and connections, shapes, and DRCs.

- Components and nets: A a progress bar indicates the percentage and number indicates value. Clicking hyperlink value produces search results for components, pins, or nets in the *Search* panel.

- Shapes: Displays the number of shapes on nets without connections (shape islands), the number of copper shapes unassigned to a net and the number of non-smooth dynamic shapes.
- DRC: Indicates whether DRC markers are up-to-date. This section also displays the count of waived DRC errors, net short errors, and waived net short errors that exist in the design.

A white color text and  icon indicates that shapes or DRCs are out of date and require update. Dim text and `0` value indicates status is up-to-date.

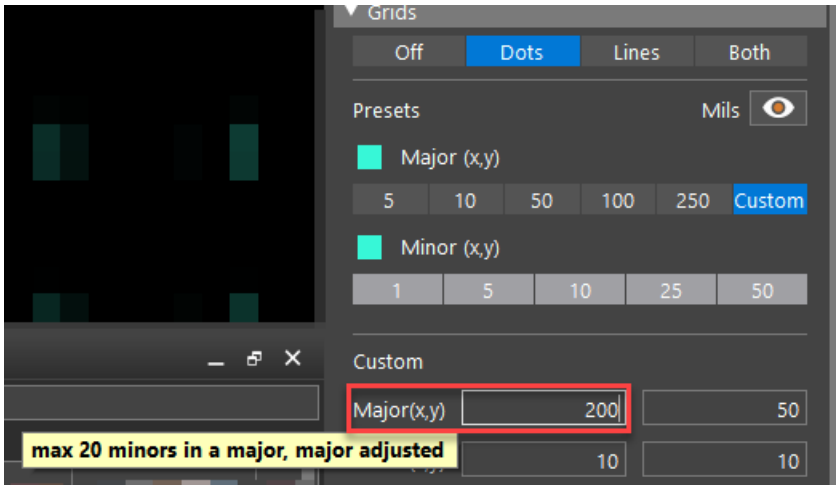


Grids Sub Section

This section provides the ability to control grid settings. The options are available to enable or disable the display of grids, type, color, and define grid values.

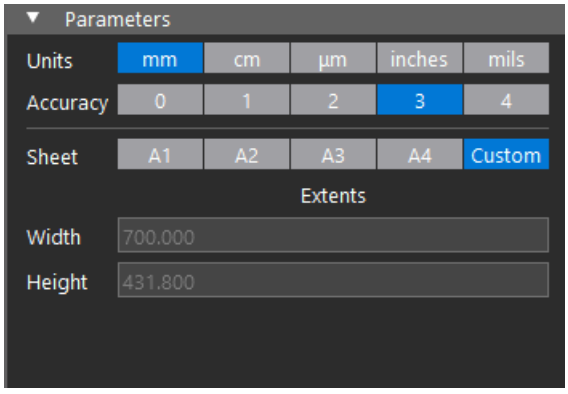
- *Dots*, *Lines*, or *Both* enable the display of grids.
- The current grid unit is displayed, which can be changed using a toggle button.
- The color box displays color assigned to the grid. Clicking the box opens a color palette, and you can choose a different color that gets immediately applied to the grid.
- The grid values can be chosen from preset values for Major and Minor grids or can be customized. Selecting a Major preset value automatically updates the Minor preset value.
- To configure the grid with user-defined values, click on *Custom*. In the *Custom* section, specify *Minor(x,y)* and *Major(x,y)* grid values. The *Major* value can adjust a maximum

of 20 *Minors*. Entering a *Major* value greater than twenty times of *Minor* is considered invalid. A tool tip is displayed, and the *Major* value is auto-set to its maximum, which is twenty times the *Minor* value.



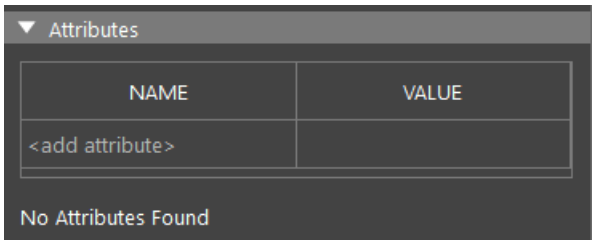
Parameters Sub Section

This section provides information on database units, accuracy, drawing size and extents. The current design values are indicated in blue color and rest of the fields are not highlighted.



Attributes Sub Section

This section displays properties attached to selected objects. This section is blank when nothing is selected in the design canvas.



Properties Tab When One or More Objects are Selected

Design objects when selected in the canvas displays information in the *Arrange, Location*, *General,* and *Attributes* sub sections of the *Properties* tab. When one or more objects are selected *Constraint* tab is also displayed. Depending on the type and number of selected objects, the information which is common to all the selected objects is displayed in the sub sections of the *Properties* tab.

Arrange Sub Section

The *Arrange* sub section display controls for editing objects including rotate, align, flip, mirror. This section remains inactive and non-editable for any selected object.

Location Sub Section

This section is always visible when one or more objects are selected in the design canvas. It displays the layer of the selected object, its X and Y coordinates, and the angle of rotation. The location information is locked and is non-editable. When more than one objects are selected, the common values are displayed and different values are indicated with [...]

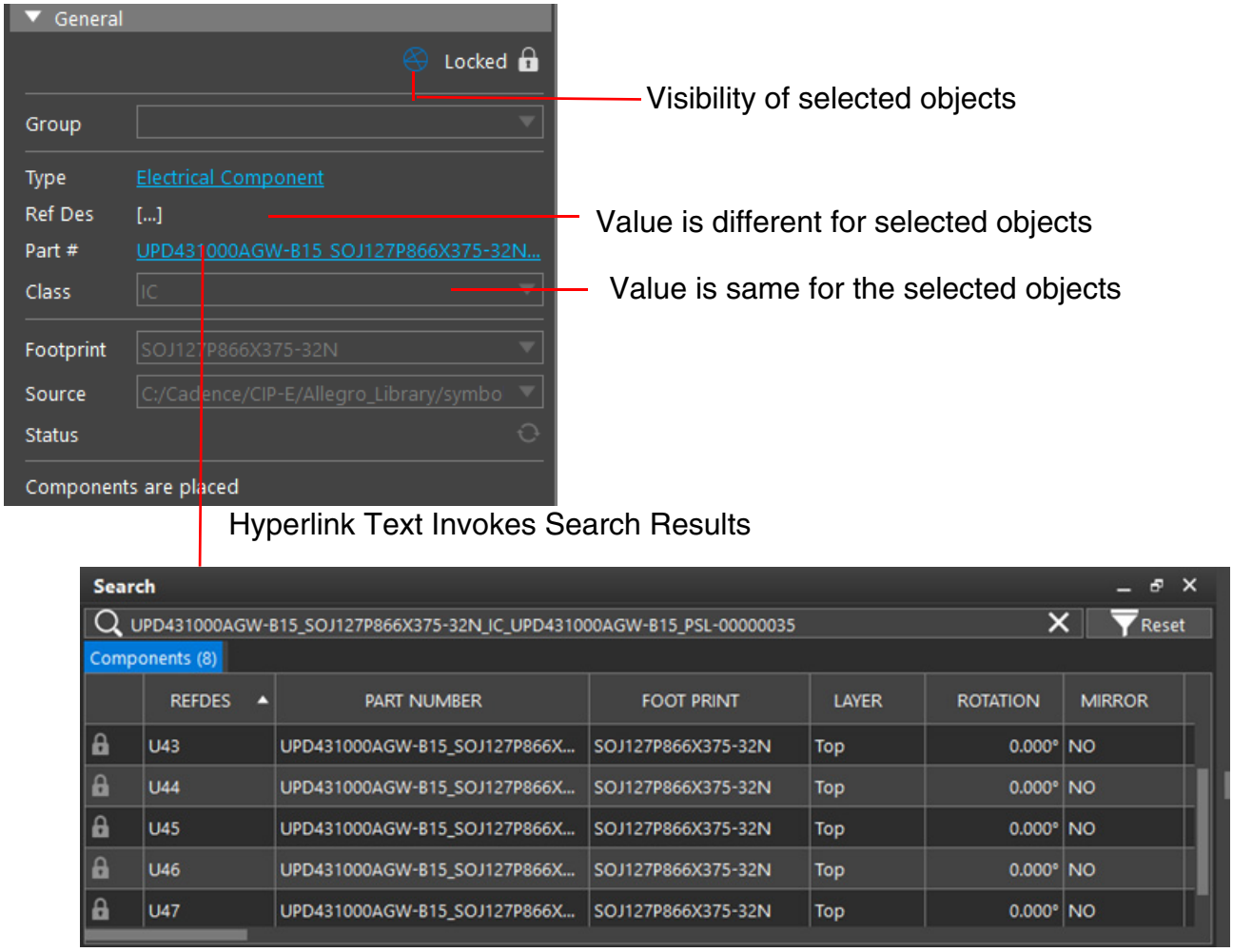
General Sub Section

General sub section is always visible when one or more objects are selected in the design canvas and displays information on the selected objects.

Depending on the object type this section dynamically updates the information.

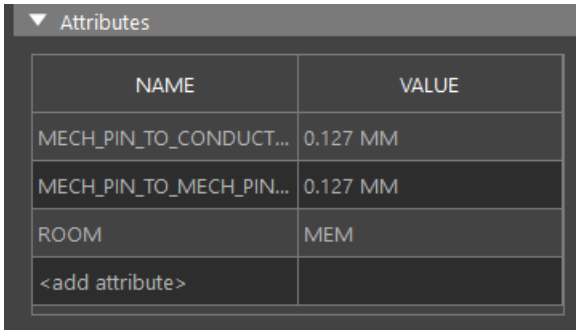
- When more than one objects of the same type are selected, the common values are displayed and different values are indicated with [...].
- When more than one objects of different types are selected, the common fields are displayed and different values are indicated with [...]. When no field is common across all the selected objects the *General* sub section remains blank.

Clicking a hyperlink value act as a search input and show relevant information in the *Search* window.



Attributes Sub Section

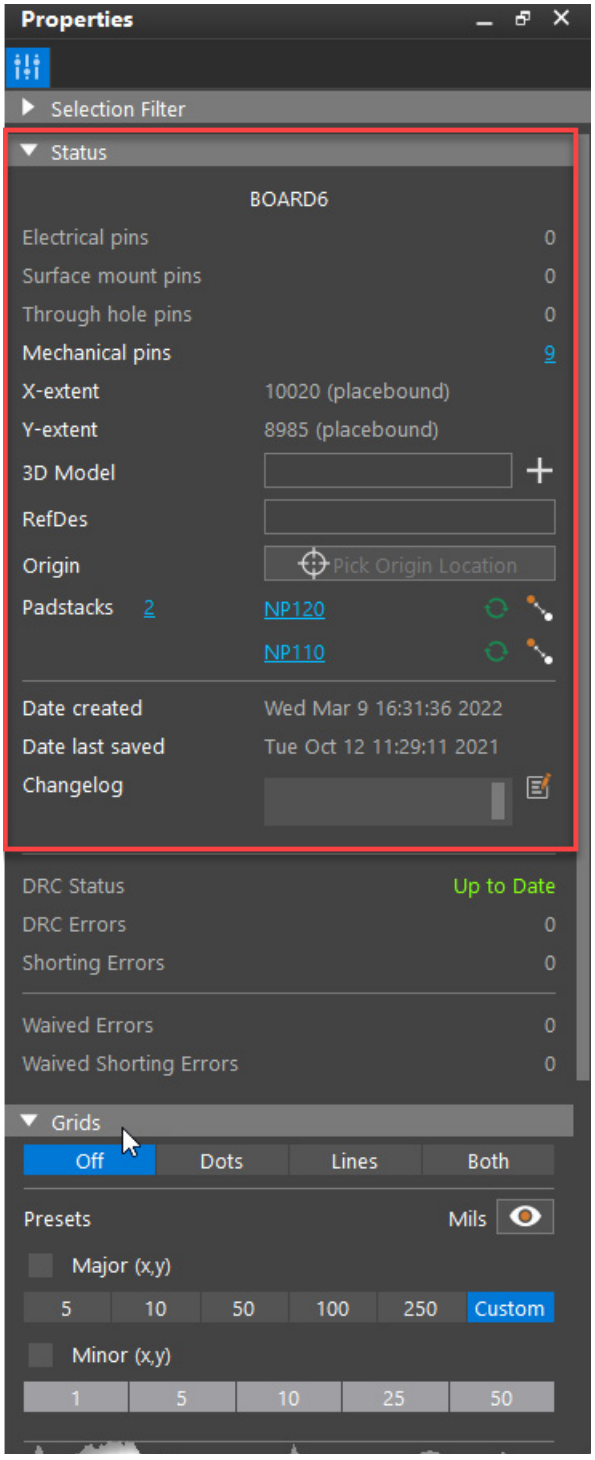
Displays properties attached to selected objects. This section displays both the system-defined and user-defined attached to selected objects. When multiple objects are selected only the common properties are displayed.



A blank row is displayed when no properties are defined no common properties are found for the selected objects.

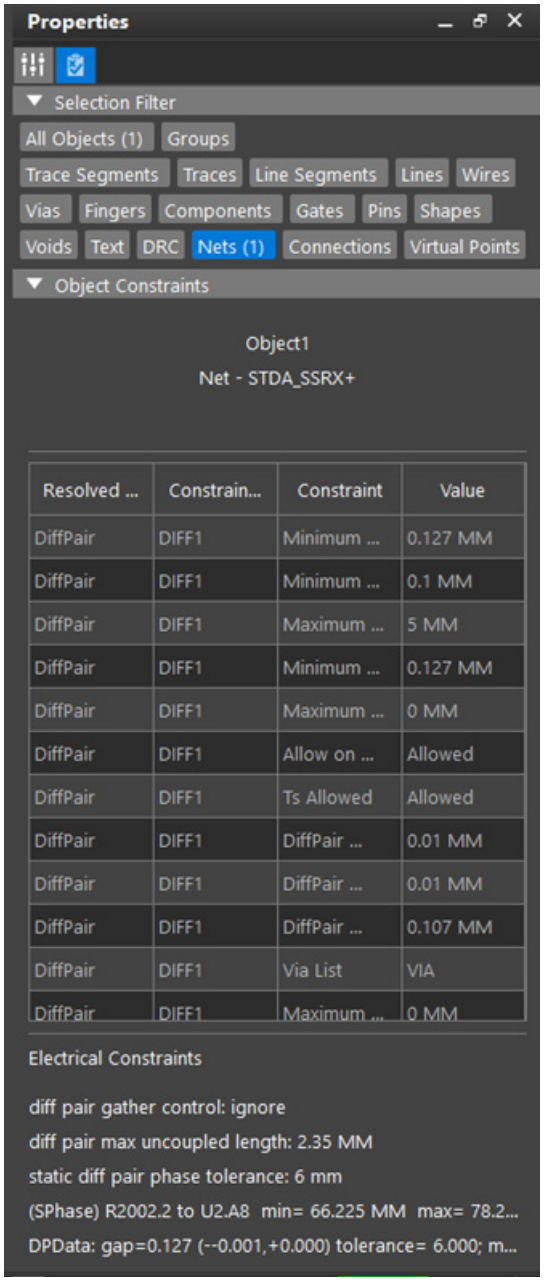
Properties Tab For a Symbol

When a symbol (`.dra`) is opened, the *Status* section of the *Properties* tab provides symbol information that includes pin information, padstack data, 3D model assigned to it, and so on.



Constraints Tab

When an object that has constraints assigned is selected in the design canvas the *Constraints* tab becomes visible. The *Selection Filter* is also available with this tab and lets you filter the object types.



The *Constraints* tab is not displayed, if in the design canvas:

- More than 2 objects are selected

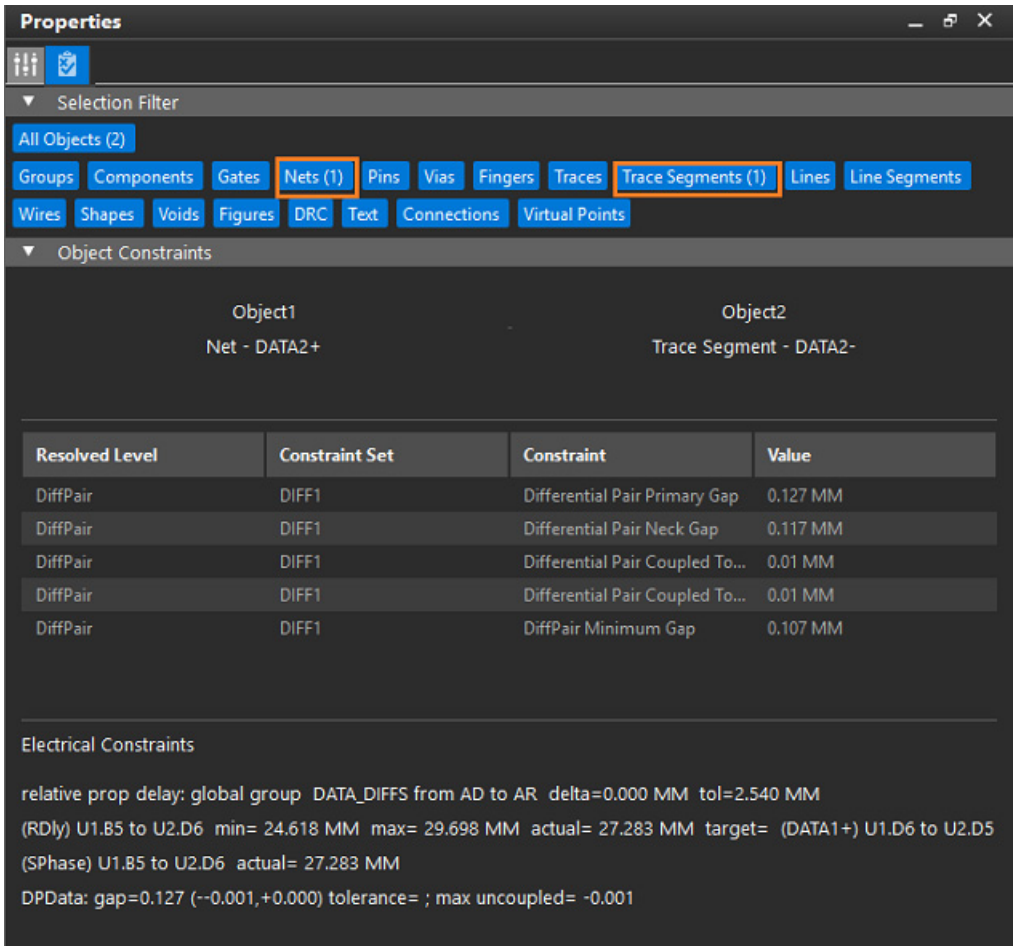
- Selected object does not have any constraint applied to it

Object Constraints Sub Section

This sub section displays constraint information of the selected objects. The following can be displayed:

- Object Name and Type
- Resolved Level
  - Physical: When single object is selected
  - Spacing: When two objects are selected
- Electrical Constraint: When at least one net is selected
- Value: When trace segment is selected

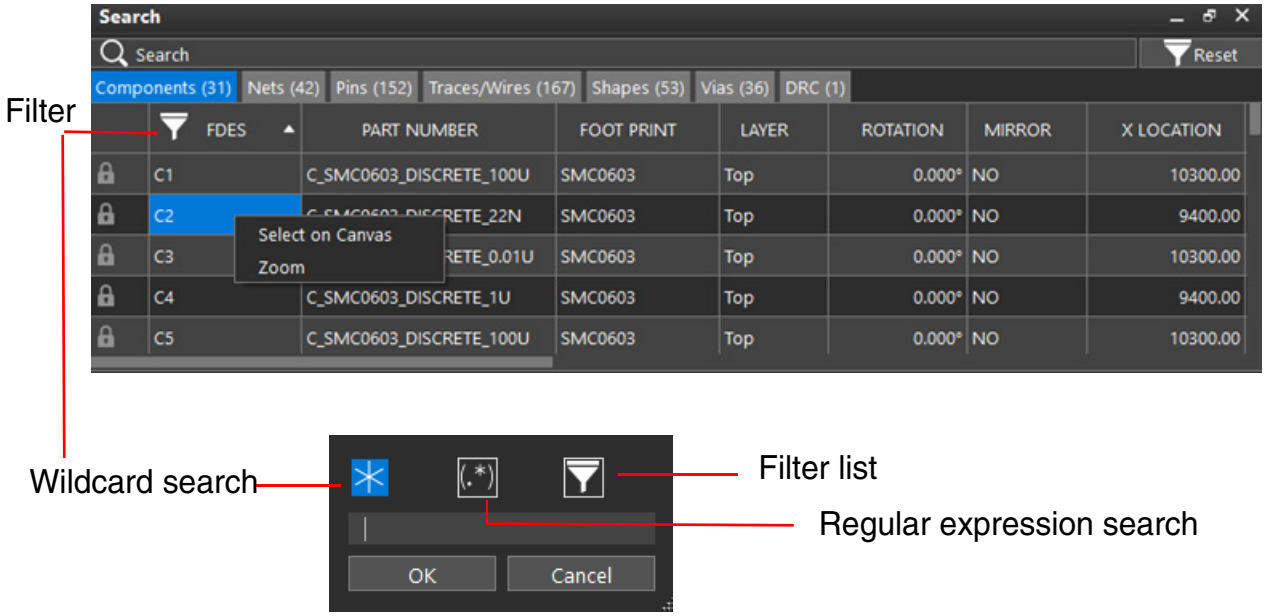
- Air Gap: When two objects are selected



Search Panel

The *Search* panel lets you locate an object in the design. When a design is opened first time many object tabs are displayed in the *Search* panel. The active tab is displayed in blue color. Each tab shows the total count of that object type and a list of objects in a spreadsheet format.

The columns of spreadsheet represents properties of that object type. You can resize, rearrange and filter information in each column.

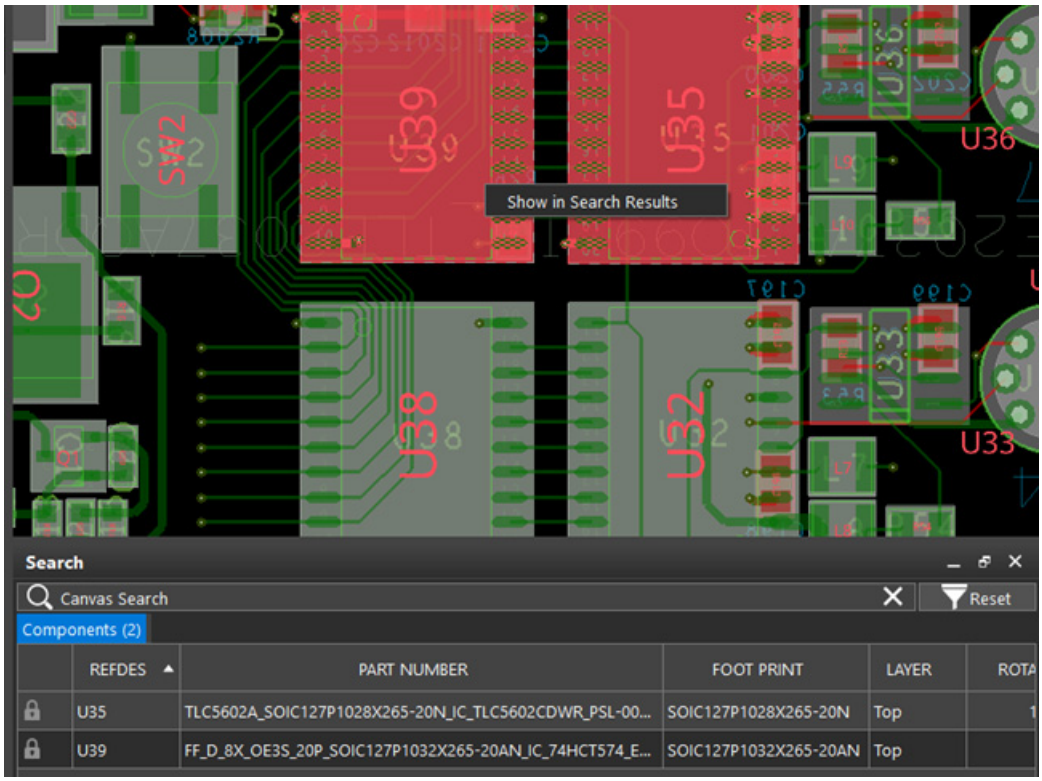


A search bar is displayed just above the spreadsheet to filter the results table. When you search a text or number string only the rows that matches the search string are displayed. Clicking a cell, selects the complete row. You can bring the focus to search bar at any time using keyboard shortcuts `CTRL+F` or by pressing `F3` key.

Double-click an object selects the corresponding object in the design canvas. To select multiple objects in the canvas, you can use right-click option *Select on Canvas*. Click anywhere in the blank space in the design canvas deselects the objects.

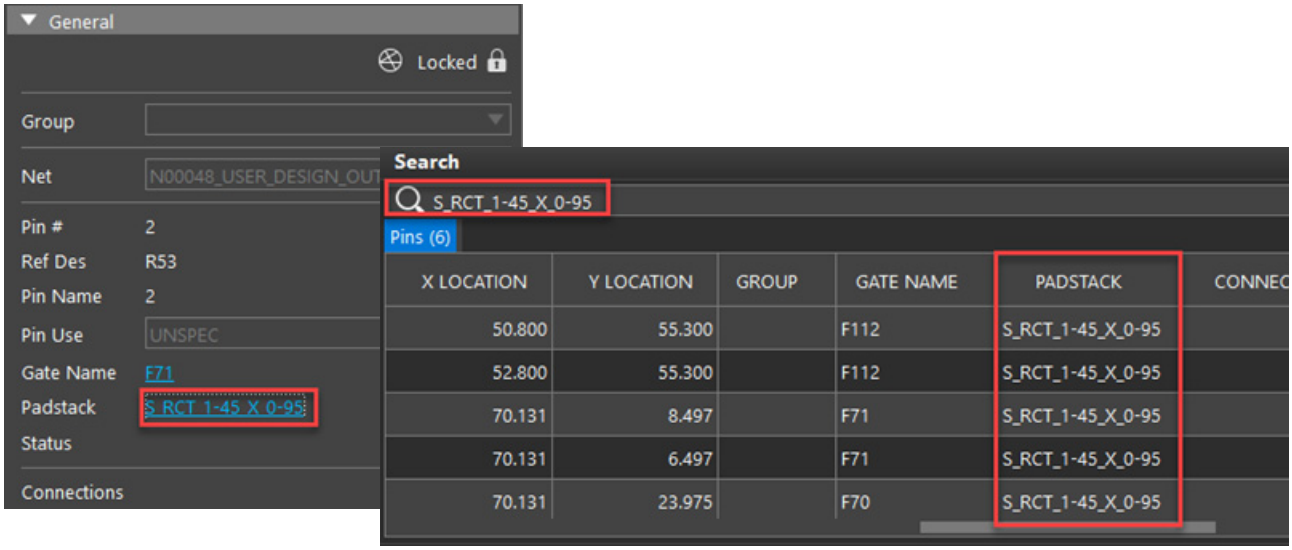
Filtering Search From Canvas

When you right-click on selected objects in the design canvas and choose *Show in Search*, the spreadsheet populates the information of the selected objects and a custom search string *Canvas Search* is displayed in the *Search* panel.



Filtering Search From Properties Panel

Clicking a hyperlink value in the *Properties* panel filters the *Search* for that value. All the objects tabs are filtered and only those are displayed that matches with the filter value.



2

Working With Allegro Free Viewer

When reviewing a design, some of the common tasks performed are described in this section.

- Viewing Objects From Search Results
- Selecting Objects From Canvas Using Selection Filter
- Viewing Net Objects in a Design
- Assigning Color to Net Objects
- Viewing Object Properties In Datatips
- Viewing Dynamic Etch Shape Parameters
- Viewing Pin and Via Parameters
- Viewing Constraint Resolution Between Two Objects
- Viewing Design in 3D Canvas
- Measuring Distance Between Objects
- Generating Prints
- Capturing Canvas Image
- Recording and Replaying Script

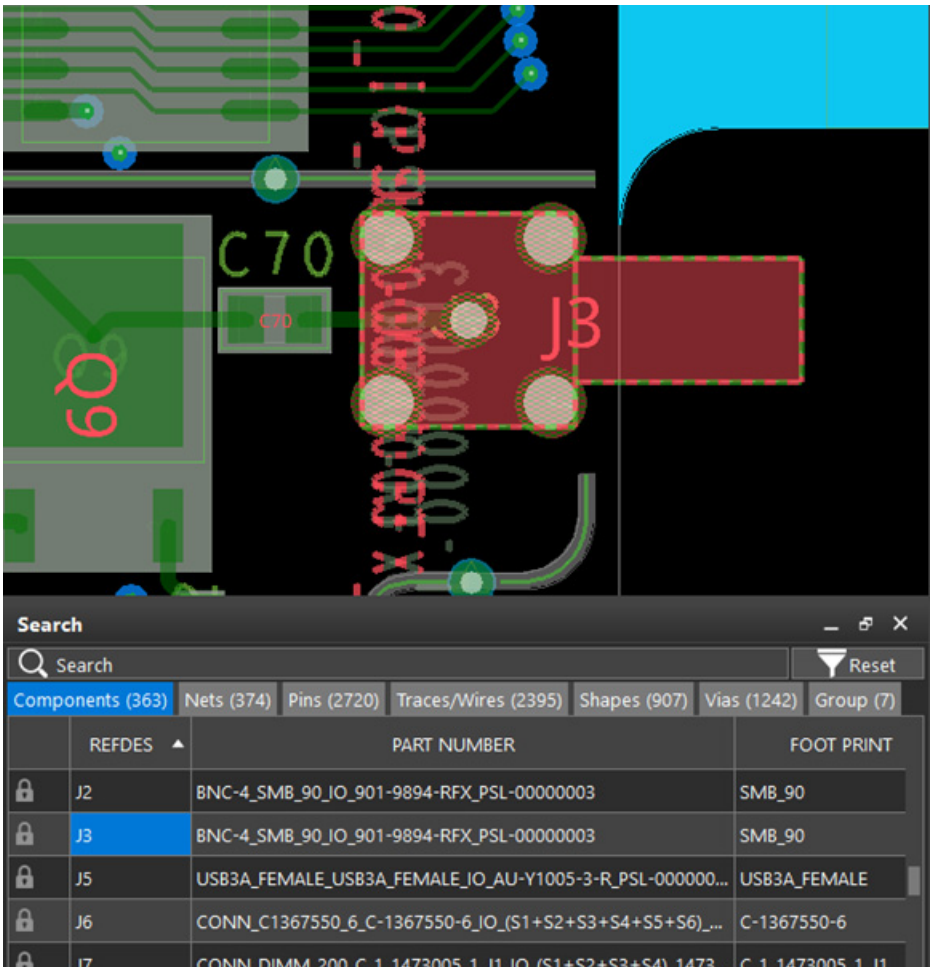
Viewing Objects From Search Results

The *Search* panel, by default, displays all the design objects in a spreadsheet format. Different types of objects are displayed in different tabs of that spreadsheet. To choose any object do the following:

1. In *Search* panel, open an object tab.
2. Double-click an object row.

   The object is highlighted and zoomed to its location in the design canvas.

3. To select multiple objects, press `CTRL` key and select object rows.



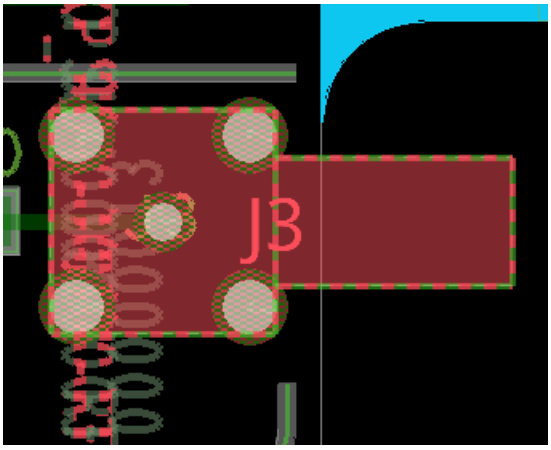
Selecting Objects From Canvas Using Selection Filter

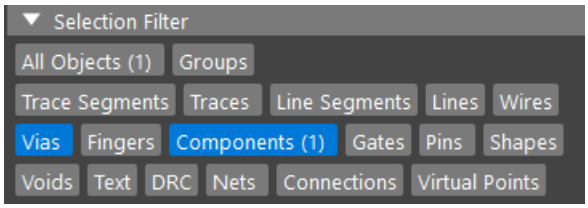
You can choose any object in the design canvas and view its information in the *Properties* panel.

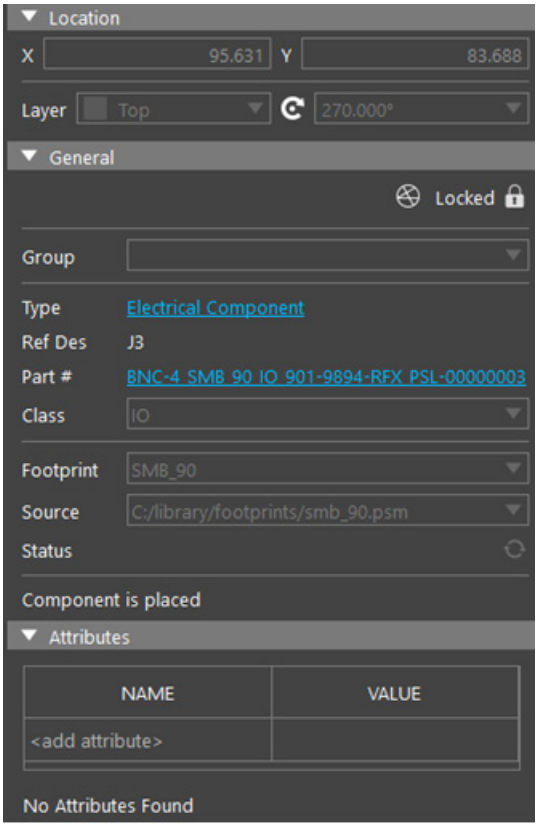
1. Adjust the *Selection Filter* to choose the desired objects.
2. In the design canvas, hover the mouse over an object you want to select.

   A datatip displays the object name and attributes.
3. Click to select the object in the canvas.

The object highlights in the canvas. The *Selection Filter* updates the count of selected object type and *Properties* panel displays the location and other attributes of the selected object.







4. To select multiple objects in the canvas, press `CTRL` key and click to select more objects.

   The count of selected objects updates in the *Selection Filter* and *Properties* panel shows only the properties that are common to all the selected objects.

Viewing Net Objects in a Design

All the nets in a design are listed in the *Nets* tab of the *Visibility* panel. To locate and view a specific net in the design do the following:

1. In the *Visibility* panel, open *Nets* tab.

   Different sub sections of the tab list all the net objects.

2. Expand the *Diff Pairs* sub section.

3. Double-click a diff pair name to view its member nets.

The following have been updated in the user interface:

- Both the member nets of diff pair are selected in the design canvas.
- The *Selection Filter* displays count of Nets.
- The *Properties* tab displays the location and general properties of nets.
- The *Constraints* tab of *Properties* panel displays the constraint resolution status.

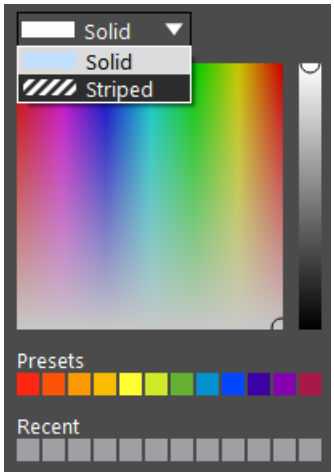
4. To choose another net object double-click net object name.

Assigning Color to Net Objects

For nets, the color box with default color assignment is displayed in gray color. To assign a color to net, perform the following steps:

1. In the *Visibility* panel open *Nets* tab.
2. Expand the *Nets* sub sections to view all nets.
3. Click the color box of the net you want to assign a color or right-click and choose *Assign Color* option.

   A color palette opens to choose a different color and pattern to the net.

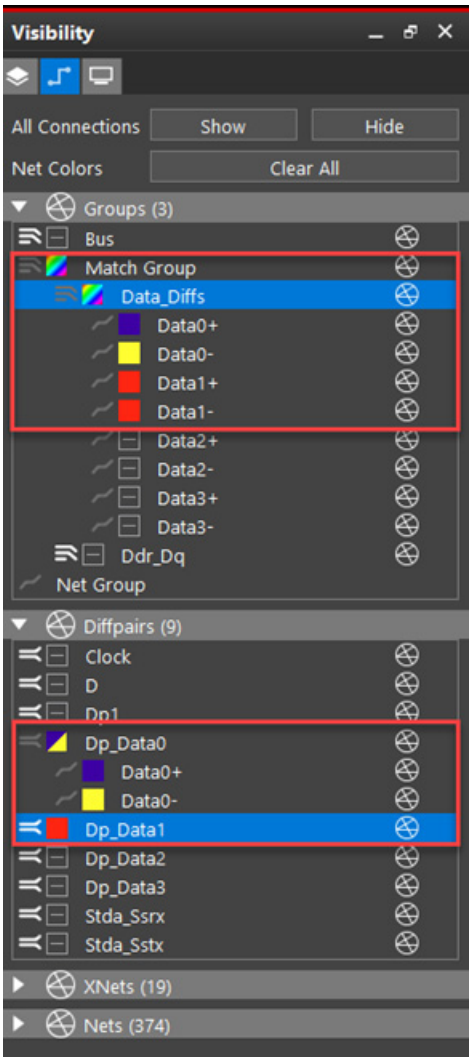


4. Pick a new color or choose from *Presets*.

   The following changes occurred:

   - The color box starts showing the new color for the net.
   - New assigned color is reflected in all the sub sections for that net.

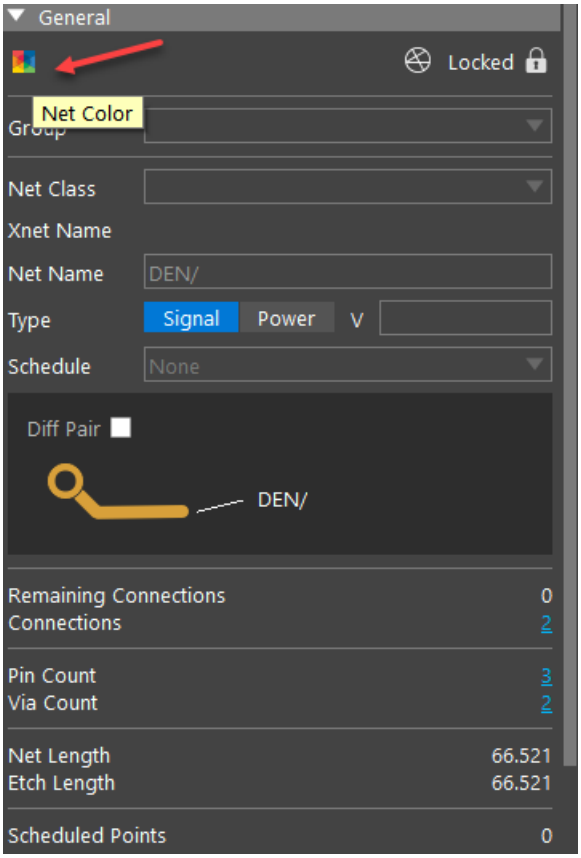
- The color of the parent net also changes.



Note: A rainbow is displayed in the color box of parent net when member nets have different color assigned.

5. To remove the color form the net, right-click the net name and choose *Remove Color*.
6. To clear color assignments on nets, click *Clear All* button for *Net Colors* option.

Note: You can also modify the color of net in the *Properties* panel.



Viewing Object Properties In Datatips

Datatips identifies design element and display information next to the cursor. You can specify the datatip format to display the required information. Do the following to customize the datatips for different object types:

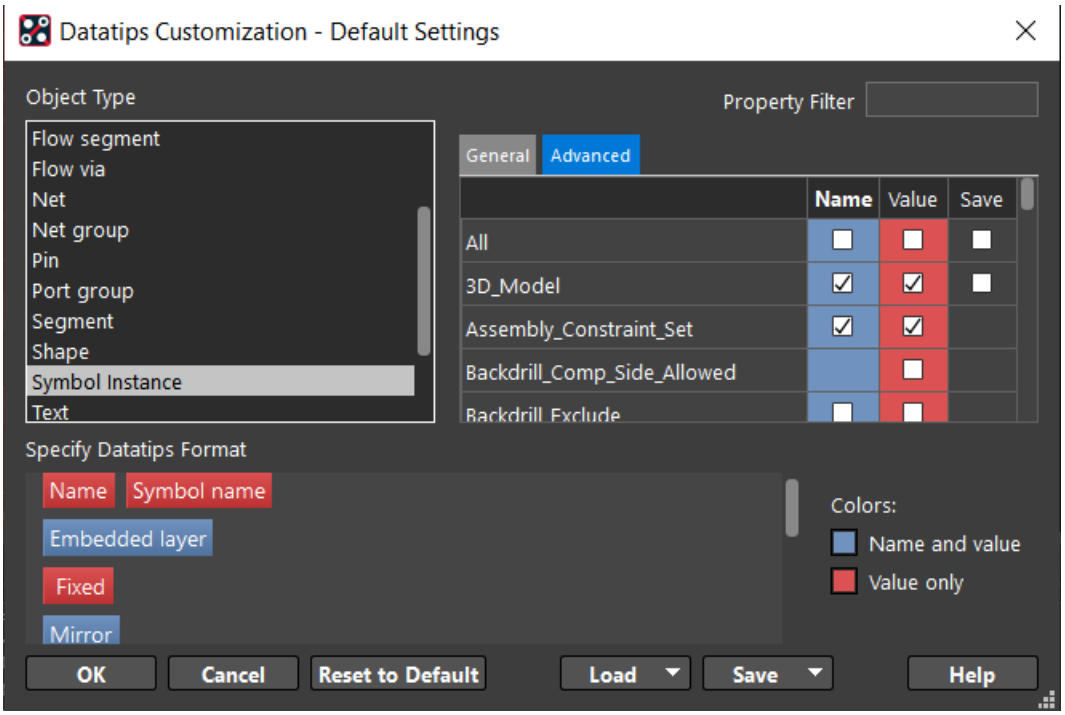
1. Choose *Tools – Datatip Customization* or type `custom datatips` in the command window.

   The *Datatips Customization* dialog box opens.

2. Choose an element from the *Object type* list.

3. In the *General* and *Advanced* tabs, enable the name and value check boxes for the information you want to display in datatips.

4. Click *OK* to close the dialog.

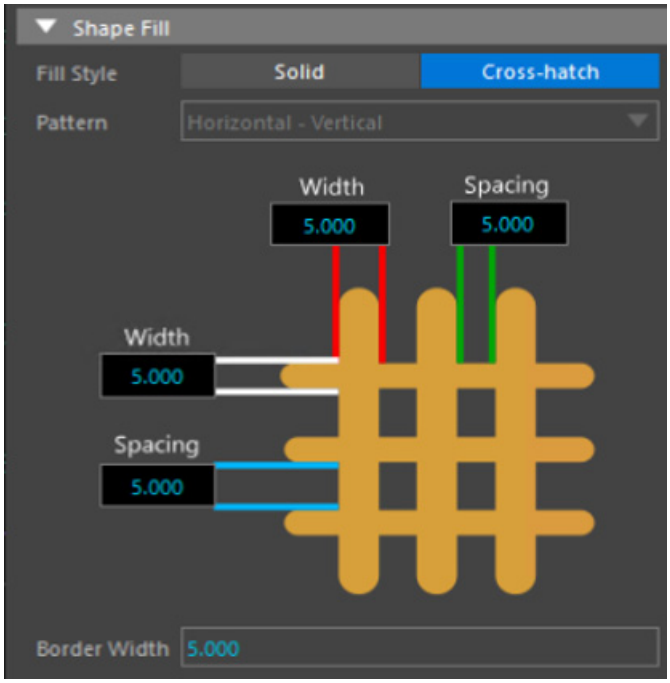


Viewing Dynamic Etch Shape Parameters

A dynamic etch shape when selected in the design canvas displays the following parameters in the *Properties* panel:

- General information of the shape
- Shape fill parameters
- Void control parameters
- Thermal relief parameters

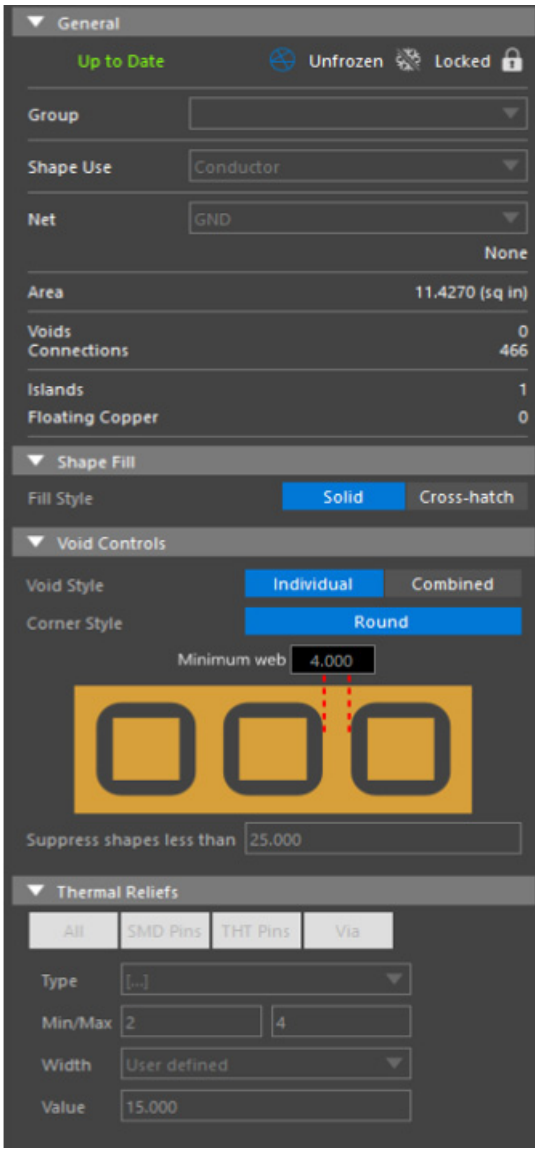
In addition to solid fill, a dynamic etch shape can also have cross-hatch fill as illustrated in the following image:



To view parameters of a dynamic shape etch instance, do the following:

1. In the *Selection Filter* of the *Properties* tab, click *All Objects* to disable all the filters.
2. Click *Shapes* to select shape objects.
3. Select a dynamic etch shape in the design canvas.

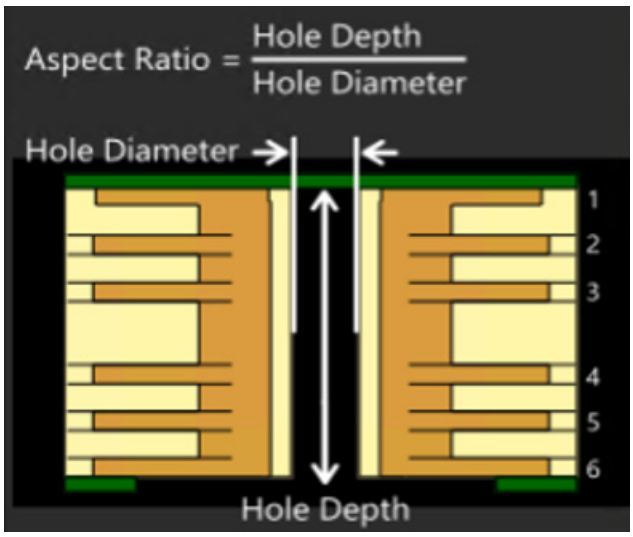
The *Properties* panel displays parameters of the selected shape.



Viewing Pin and Via Parameters

When selecting a pin or via in a design canvas, additional parameters are displayed in the *Thermal Reliefs* and *Padstack* section of the *Properties* panel.

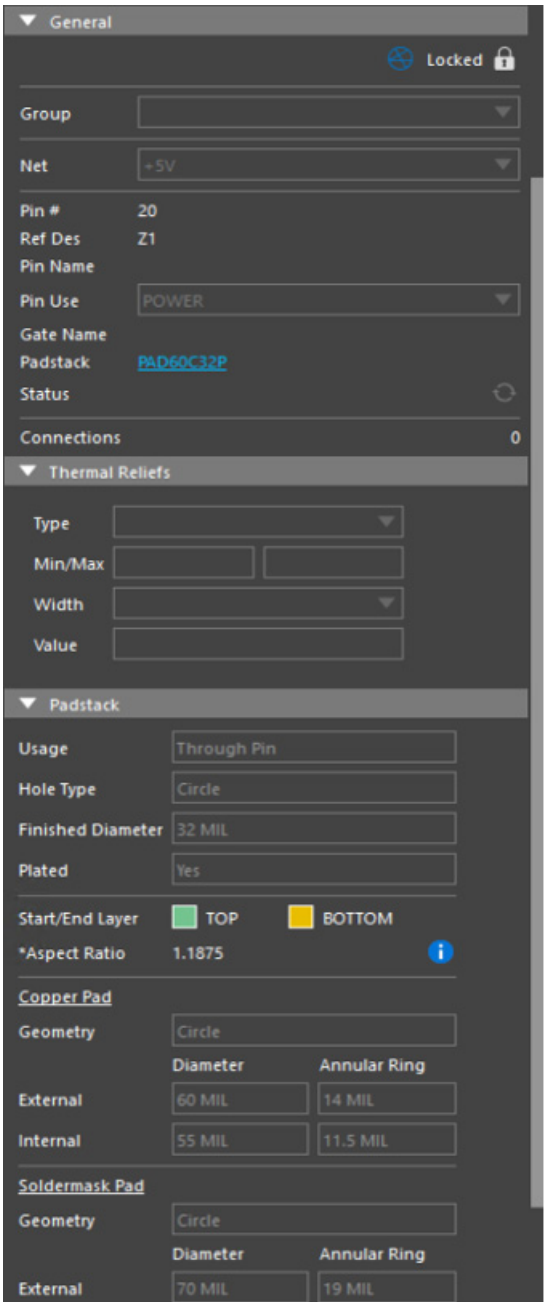
Hover the cursor over the icon, displays the following:



To view properties of a pin or via, do the following:

1. In the *Selection Filter* of the *Properties* tab, click *All Objects* to disable all the filters.
2. Enable either *Pins* or *Vias* or both.
3. Select pins or vias in the design canvas.

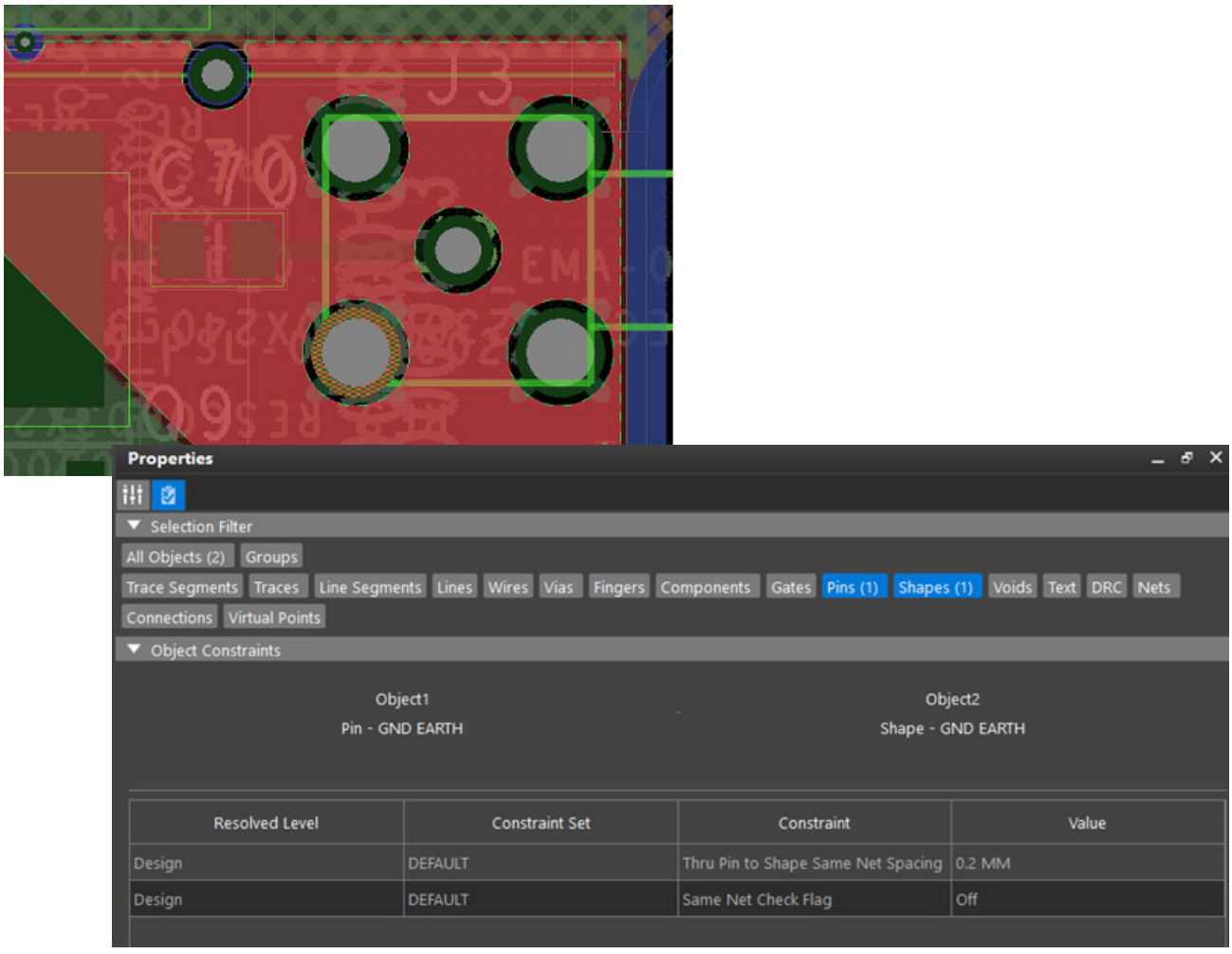
The *Properties* panel displays parameters of the selected pins or vias.



Viewing Constraint Resolution Between Two Objects

When two objects are selected in the design canvas you can view the constraint resolution in the *Constraint* tab of the *Properties* panel.

1. In the *Selection Filter* of the *Properties* tab, click *All Objects* to disable all the filters.
2. Enable object filter for the objects that you want to check the constraint resolution.
3. Hover over the objects in the design canvas and click to select an object.
4. Press `CTRL` key and click to select one more object of the same type.
5. Open *Constraint* tab in the *Properties* panel.
6. The *Object Constraints* section displays the resolved physical or spacing constraints, constraint set, constraint name and value in a tabular format.



Viewing Design in 3D Canvas

To view the design as a manufactured output you can launch 3D Canvas and analyze the design. You can review all or selective objects in the 3D Canvas.

Perform the following to open and view design in 3D Canvas:

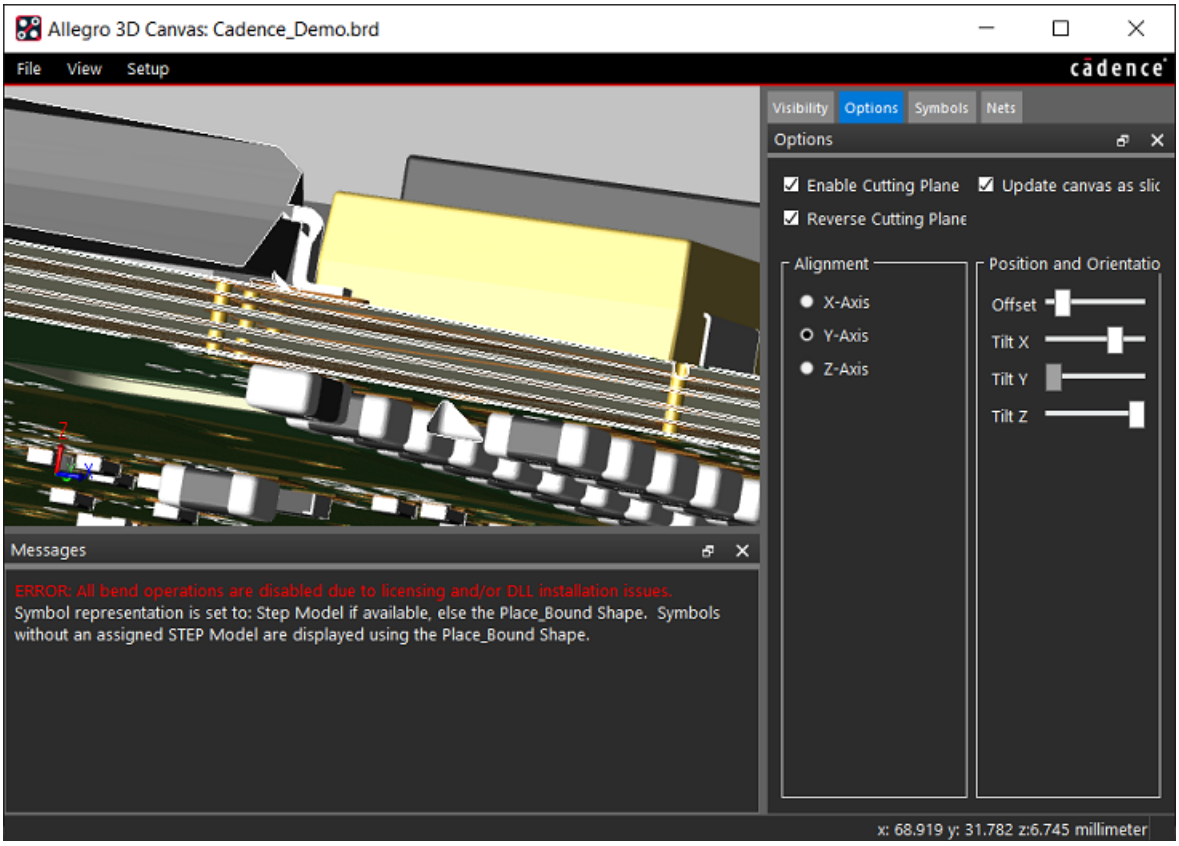
1. Choose *View – 3D Canvas.*
2. In *3D Canvas Filter*, choose objects and layers and click *OK*.

   A progress meter is displayed and designed is launched in the 3D Canvas window.
3. To view internal details of the design in the 3D Canvas, right-click and choose *Cutting Plane*. In the *Options* tab,

   a. Enable cutting plane options.

   b. Choose the alignment axis.

   c. Drag the tilt sliders to change the angle of the cutting plane along the non-aligned axis.



4. To change the appearance of objects and layers in 3D Canvas, choose *Setup – Preferences.*
5. Choose *File – Close* to exit 3D Canvas.

Measuring Distance Between Objects

When reviewing design, you can measure distance between different objects by using keyboard and mouse controls. Do the following to measure distance:

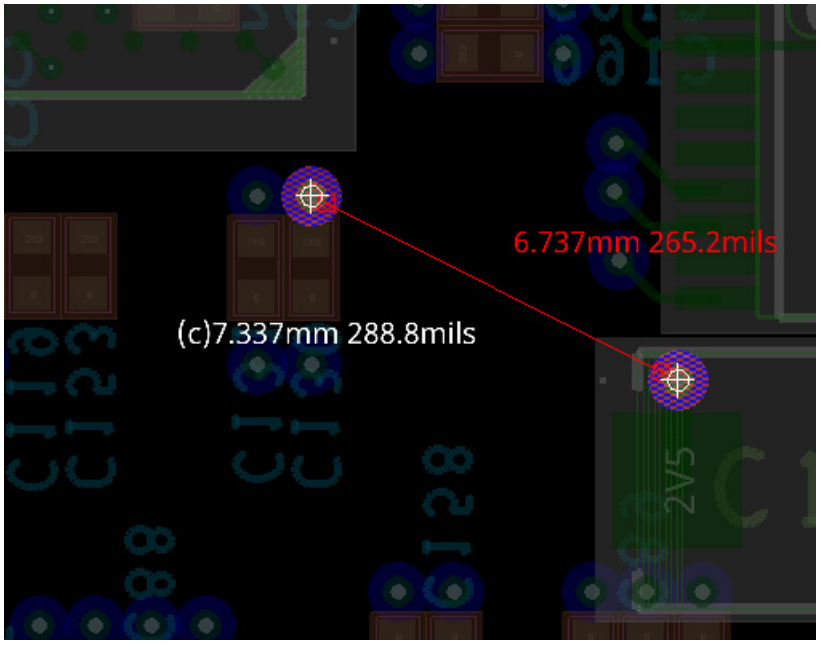
1. Adjust *Selection Filter* to enable the specific object filters.
2. Press and hold `Alt` key.

   Shadow mode is temporarily enabled.
3. Click to select an object that you want to measure from.

   The object highlights in the canvas.
4. Hover the mouse over object you want to measure to.

   The object is temporary highlighted. A rectangle with two cross lines connecting two opposite corners indicates the point on an object where the air gap is measured. The air gap distance between the selected objects is represented by a red dynamic straight line drawn between the centers of the two markers. The air gap measurement displayed in red includes both primary and secondary units.

   If the selected object is a pin or via, the center-to-center spacing between them is also displayed in white color, indicated with (c).



5. To choose a different object, hover over another object.

The first object remains the same and the second object changes. The dynamic line also changes and display the air gap between objects.

6. Repeat the above step to choose different objects to dynamically see measured distance values between first object and new objects.
7. To reset the selection of the first object, click on another object.

   The object highlights and defined as first object.
8. Hover over another object to display measure.

Tip

To measure distance between segments of a shape, enable *LineSegments* in the Selection Filter.

9. To exit measure mode, release the `Alt` key.

   Shadow mode also disables.

Generating Prints

To directly print the design from free viewer, you can use print commands. You can preview the design and setup printing parameters as you do in any other application. Modified parameters are retained in the `.ini` file and applied to every database you open until you change the parameters.

Note: Print commands are available only on Windows platform.

Perform following steps to print a design:

1. Adjust the visibility of the design layers and objects.
2. Zoom to the location which you want to print.
3. Choose *File – Print Setup.*

   The *Plot Setup* dialog box opens.
4. Set printing parameters as required and click *OK* to save the settings.
5. Choose *File – Print*.

   The Print Review window opens.
6. Expand the Print Review to preview the active design and select the printer icon.
7. Choose a printer name. If you want to direct output to a file, enable *Print to file* check box in the Print window.
8. If necessary, set additional printing options and click *Print*.

Capturing Canvas Image

Saving image of design canvas is important when you want to share your screen with others or want to reference it later. To do this task you can capture an image of design canvas either from a menu option or from the command line. You can save this image in any of the four supported image formats (`*.bmp, *.jpg, *.png,` and `*.tif`).

To capture and save the image do the following:

1. Zoom to the location of the design in the canvas which you want to capture.
2. Choose *Tools – Capture Canvas Image* or type `capture image` in the command window.

   The *Capture Image* file browser displays.
3. Browse a directory location to save the image.
4. Specify the name of the image file.
5. Choose an image file format and click *Save*.

   The image is by default is saved in the current design directory in Bitmap (`.bmp`) format.

Recording and Replaying Script

1. Choose *File – Script.*

   The *Scripting* dialog box displays.

2. In the *Name* text box, enter a name for the script.

3. To create a macro, enable the *Macro Record Mode* check box.

4. Click *Record*.

   The *Scripting* dialog box disappears.

5. Perform the tasks that you want the script to run.

   The name of the file and the *Rec* status appears in the Status window.

6. Choose *File – Script* again and click *Stop* in the *Scripting* dialog box.

   A script file(`.scr`) is saved in the current design directory.

7. To replay a script, do the following:

   a. In the *Scripting* dialog box, click *Browse*.

   b. Choose a script file and click *Open*.

   c. Click *Replay*.

   The selected script start running on the design.

3

Menus and Commands

The user interface of Allegro Free viewer is simple and consists of logically organized menus that are required to review a design. This section walks you through the menu options and associated commands of free viewer.

File Menu Commands

The File menu includes standard options to open, close and print a design. You can access the following commands:

- open
- plot
- plot setup
- plot preview
- script
- exit

open

Opens an existing design file in the current directory. A file browser lets you search for the specified design file if you do not provide a file name. You can also open a file from the most recently used files using the *File – Recent Designs* menu.

Menu Path

File – Open

File – Recent Designs

Shortcut Keys

Ctrl+O

Icon

Syntax

`open [<design to open>]`

If you do not provide the `<design to open>` argument, a browser window opens in the current directory.

Examples

```
open master.brd
```

The `master.brd` file opens in the current directory.

```
open \boards\master.brd
```

The `master.brd` file, located in the `boards` directory in the current directory, opens.

Open File Browser Dialog Box

The Open dialog box is a standard file browser. You can type the name of the file you want in the *File name* field, or choose the file from the list. Selecting a directory in the list by double clicking on it pushes into the directory.

The *Look in* drop-down allows you to navigate higher in the directory hierarchy. You can drag and drop folders in the *Look in* column to create links to directories. To delete those link a *Remove* option is available on the right-click menu.

The *File of type* field, by default, provides the typical extension of the file required. The drop-down provides access to additional extensions. You can override the extensions by typing a name with one or more wildcard characters in the *File name* field. The preview button displays a simple graphic of the database, the image of which depends on the type of database you are viewing.

plot

The `plot` command on Windows runs the standard *Windows Print* dialog box.

Menu Path

File – Print

Shortcut Keys

Ctrl+P

plot setup

Lets you set parameters for plotting a design.

Menu Path

File – Print Setup

Plot Setup Dialog Box

The parameters set in the *Plot Setup* dialog box are retained in the `.ini` file. Therefore, they remain in effect for every database you open until you change the parameters.

General Tab

Plot scaling	*Fit to page*: Indicates the plot file is to be scaled to fit the entire plotted page. *Scaling factor*: Indicates the scale of the finished plot. *Default line weight*: Converts any zero width line to a width proportional to the setting. Aids in displaying very thin lines on high-resolution output.
Plot orientation	*Auto center*: Centers the design on the plot page. This control automatically invokes when you choose the Fit to page setting. *Mirror*: Flips the design end-for-end about the Y axis. Useful for viewing top and/or bottom layers.
Plot method	*Color*: Directs the output to print in color. Color is determined by the method specific to the platform you are using. On Unix, color is read from a user-supplied stipples file (`allegro_plot_param.stipples`); if a stipples file is not found, plotter color defaults are used. On Windows, color selection is determined by setting in the Color and Visibility dialog box. *Black and white*: Directs the output to print in black and white.
Plot contents	*Screen contents*: Prints/plots the contents of what is currently displayed in the design area of the user interface. *Sheet contents*: Prints/plots the extents of all currently-visible graphics within the design (not the drawing extents).
IPF setup	*Vectorize text:* Specifies that text output to the IPF file is broken into line vectors. *Width*: specifies the width for simulating the text characters. The width used is established with the environment variable PLOT_VECTEXT_WIDTH. The default is 0. When *Vectorize text* is enabled, and a negative value is entered in the *Width* field, any other width setting of 0 or greater causes photoplot widths to be ignored, and all text is uniformly stroked with the same specified width. The *Vectorize text* and *Width* settings apply as specified when the `create plot` command executes.
OK	Saves the settings and closes the dialog box.
Cancel	Closes the dialog box without saving the settings.

Windows Tab

Only appears on the Windows platform. The `.ini` file retains all settings between sessions.

Non-vectorized Text Control	
Non-Vectorized Text	Choose to generate plot files with true font text, which lets you generate PDF-format plot output with researchable text.
Font	Specifies a font to use; defaults to Courier.
Font Height	Enter a percentage scaling factor for the character height to closely match font text with that of the normal vectorized text display/plot.
Font Width	Enter a percentage scaling factor for the character width to closely match font text with that of the normal vectorized text display/plot.
View Available Fonts	Click to review the available text fonts for the plot device.
Margin Control	
Margin Width	Specify the desired margin width in user units. The default equates to 0.25 inches, or 0.0 if the `noplotmargins` environment variable is set.

plot preview

Lets you preview a plot as it will look when printed. This command opens Print Preview window to preview the active design as it will plot based on the setup parameters in the *Plot Setu*p dialog box.

Menu Path

File – Print

script

This command records a series of actions. It creates a text file containing the commands that you execute and adds a `.scr` extension to the file name. Using the interactive version of the `script` command that displays the Scripting dialog box, you can also replay the script.

A macro is a script that lets you automate a series of point selections and replay them, starting at another coordinate. When you replay a macro, Allegro Free viewer prompts you for

a starting point (origin). The macro places the point selections you recorded relative to this starting point. This is useful in performing operations that you need to repeat on a board/ design drawing, such as repeating complex geometric operations.

The current settings in your design are recorded in the script or macro. To display the script with different settings, you must change them as part of the script.

Menu Path

File – Script

Scripting Dialog Box

Script File	
Name	Specifies the name of the file in which you record your actions. Allegro PCB Editor adds the `.scr` extension to the file name
Browse	Displays the script file data browser that lets you choose a script file to replay
Library	Displays the script file data browser that lets you choose a script file to replay. Opens to your script path location
Generate	Displays a file browser from which you can choose a `.jrl` file to convert into a script without having to leave the current environment. To process the journal file and reconstruct the appropriate script outside of Allegro PCB Editor, run: `j2script <source_jrl_file> <target_allegro_script>`
Macro record mode	Specifies whether or not you record as a macro. When replaying, a macro requires a starting point
Record/Replay	
Record	Starts recording your actions
Stop	Stops recording your actions or replaying a script
Replay	Starts replaying a macro or script
Cancel	Closes the dialog box
Help	Displays the Help window

exit

Exits the active layout, closes the application, and returns to the host operating system. Any changes in the design are discarded.

Menu Path

File – Exit

View Menu Commands

The View menu includes standard options to zoom and window settings. You can access the following commands:

- zoom commands
- redraw
- 3d
- show waived drcs
- blank waived drcs
- showhide window commands
- reset dockwindows

zoom commands

Command Name	Menu Name	Icon	Shortcut Key	Description
`zoom points`	*Zoom By Points*			The `zoom points` command lets you define an area of your layout to zoom in on (magnify).
`zoom fit`	*Zoom Fit*		Numeric key (0)	The `zoom fit` command fits your entire layout in the design window.
`zoom in`	*Zoom In*		Numeric key (+)	The `zoom in` command magnifies your view by a factor of two. You can continue to zoom in on a design by repeating this command. Use dynamic zooming by way of the middle mouse button. A full view of the design, excluding legends and borders, is displayed in the Design window.
`zoom out`	*Zoom Out*		Numeric key (-)	The `zoom out` command halves the magnification of your layout. You can continue to zoom out on a design by repeating this command. Use dynamic zooming by way of the middle mouse button.
`zoom world`	*Zoom World*			The `zoom world` command reduces the magnification of your design so you can view your entire drawing.
`zoom center`	*Zoom Center*			The `zoom center` command moves the indicated point in the drawing into the center of the window display.
`zoom previous`	*Zoom Previous*			The `zoom previous` command lets you to zoom back from the current window extents to the prior view.

redraw

Refreshes the work area.

Menu Path

View – Refresh

Shortcut Keys

F5

Icon

3d

Launches Allegro 3D Canvas, which lets you visualize and analyze a three-dimensional model of a design as a manufactured output. You can also view mechanical objects such as shields, fans, heat sinks and housings.

Menu Path

View – 3D Canvas

Shortcut Keys

Numeric Key 3

Icon

show waived drcs

The `show waived drcs` command lets you display all waived DRC error markers on the board. The command does not waive DRC errors. It rather displays the error markers that exist in the design but are invisible.

This command is the opposite of the `blank waived drcs` command.

Menu Path

View – Waive DRCs – Show

blank waived drcs

The `blank waived drcs` command lets you suppress waived DRC error markers from displaying on the board. The command disappears DRC error markers from the board.

This command is the opposite of the `show waived drcs` command.

Menu Path

View – Waive DRCs – Hide

showhide window commands

These commands toggles the visibility of Allegro Free Viewer windows and panels.

A check mark to the left of a window name menu option in the *View* menu indicates that the window is visible. Clicking a menu option with a check mark, hides the window. When you click the menu option again, the window is displayed at the same position with the same size as earlier. You can dock or undock the window by clicking and dragging it anywhere within or outside the design window. You can also control the visibility of a window by choosing to fold, float, or close the window using the controls at the top-right corner of the window.

Note: To show all window panels in their original positions, use *View – Reset UI to Default* (`reset dockwindows` command).

Command Name	Menu Name	Description
`showhide text`	*Command*	Toggles the visibility of the *Command* window.
`showhide enhfind`	*Search*	Toggles the visibility of the *Search* window.
`showhide enhproperties`	Properties	Toggles the visibility of the *Properties* panel.
`showhide enhvis`	Visibility	Toggles the visibility of the *Visibility* panel.

Syntax

```
showhide enhfind [show] [hide]
```

showhide_find	Displays or hides the panel, depending on its current state.
show	Displays the panel if it is hidden. If it is already visible, no action occurs.
hide	Hides the panel if it is visible. If it is already hidden, no action occurs.

Menu Path

View – Command

View – Search

View – Properties

View – Visibility

reset dockwindows

Restores the *Command, Search, Properties,* and *Visibility* window panels to display in their original positions.

Menu Path

View – Reset UI to Default

Syntax

```
reset_dockwindows
```

Tools Menu Commands

The Tools menu provides options to view design cross-section data, customizing datatips along with capture screen command.

- capture image
- xsection
- custom datatips

capture image

The capture image command captures the screen shots of the selected part of the design canvas and save in Jpeg (`.jpeg`) format. The other formats for saving an image are Bitmap file (`*.bmp`), TIFFT (`*.tif`) and Windows Device Independent Bitmap(`*.dib`).

When you capture an image a file browser is opened to save the image at a desired location. By default, the command is saved in the working directory.

Menu Path

File – Capture Canvas Image

xsection

Displays the layout Cross-section report, which provides information about each layer defined in your layout.

Menu Path

Tools – Cross-section Report

Icon

custom datatips

Lets you customize a context-sensitive datatip that identifies an element when the cursor hovers over it. The datatip configuration file `custdatatips.cdt` contains default datatip information, which loads in the local pcbenv directory when the tool launches. Otherwise, only element names display in datatips.

Menu Path

Tools – Datatip Customization

Datatips Customization Dialog Box

Object type	Choose to customize datatips for clines, nets, symbol instances, pins, vias, or DRCs.
General tab	Lists information to display in a datatip for the element chosen in *Object Type*. Click to Check the *Name* box to the right of the information to include it in the datatip; the *Value* box gets checked automatically, indicating its inclusion in the datatip as well. Select the *Value* box to only include the alphanumeric character string associated with the information in the datatip, which displays as $<*value*>, such as $COMMENT for instance, in *Specify DataTips Format*. Choose *All* to display all information available for the chosen element in the datatip.

Advanced tab	Displays all properties applicable to the chosen *Object Type* and available for inclusion in the datatip. Select the *Name* box to the right of the information to include it in the datatip; the *Value* box gets checked automatically, indicating its inclusion in the datatip as well. Select the *Value* box to only include the alphanumeric character string associated with the information in the data tip, which displays as $<*value*>, such as $COMMENT for instance, in *Specify DataTips Format*. Select the *Save* box, which only appears next to the user-defined attributes, to include these properties in the CDT file on saving it. Select *All* to check the check boxes in the column to display all information available for the chosen element in the data tip. **Note:** For Net objects, *Path length* and *Manhattan length* are included in the Advanced tab.
Property filter	Enter whole words or character strings to locate a subset of the properties available for the chosen element. To specify a character string, use the asterisk (*) as a wildcard character. Displays only when the you choose the *Advanced tab*.
Apply filter	Choose to display a subset of the available properties using the string entered in the *Property filter.*

Specify DataTips Format	Customize the order in which to display datatip information using the following keys: Up arrow: Appends the selected datatip entry to the line above it. Down arrow: Appends the selected datatip entry to the line below it. Left arrow: Transposes the selected datatip entry with that immediately to the left of it (if the first entry in a line is selected, nothing happens). Right arrow: Transposes the selected datatip entry with that immediately to the right of it (if the last entry in a line is selected, nothing happens). ENTER: Inserts a line break, and moves all the data tips immediately after the selected space onto the next line when you choose a space between data tip entries on the same line. BACKSPACE and DELETE: Removes a line break and places all data tips immediately after the selected space on the same line when you choose a space between data tip entries on adjacent lines (line break).
Colors	Click to open Select Color dialog for name and value
Load default CDT file	Loads settings from the default `custdatatips.cdt` file.
Save default CDT file	Saves modifications to the default settings in the `custdatatips.cdt` file.
Load Custom CDT File	Imports your customized datatip settings from an external `.cdt` file and applies them to the current design. A file browser appears with the filter set to `*.cdt` and a list of all `.cdt` files available in the current local working directory. You can manually browse to other directories to open a `.cdt` file. For instance, you may create a file with settings that suit a particular design's requirements; each time you open that design, import settings from that `.cdt` file.

Save Custom CDT File	Exports the current design's customized datatip settings to an external `.cdt` file stored in your local working directory. A file browser appears with the filter set to `*.cdt` and a list of all `.cdt` files in the current local working directory. You can manually browse to other directories to specify an alternate save location.
OK	Saves settings to the *.cdt* file currently loaded and closes the dialog box.
Cancel	Closes the dialog box without saving any changes.
Reset to defaults	Removes all datatips customization and restores original settings.

4

Troubleshooting

What is the minimum system requirement to install Allegro Free Physical Viewer?

The supported operating systems are:

- Windows 10 (64-bit) Professional, including Dark Theme mode
- Windows 11 (64-bit) Professional and Enterprise
- Windows Server 2016 (All Service Packs)
- Windows Server 2019

Cadence® OrCAD® and Allegro® products do not support Windows 10 Starter and Home Basic. In addition, Windows Server support does not include support for Windows Remote Desktop. Windows RT and Tablets/Phones, including Windows 10 Phone, are also not supported.

Recommended hardware is

- 16 GB RAM
- 50 GB free disk space
- A dedicated graphics card (purchased within the last two years) installed with graphics card drivers updated

Note: Do not install Allegro Free Physical Viewer over any existing Cadence installation. Allegro Free Physical Viewer should not be installed in the same directory with any other Cadence product.

Allegro Free Physical Viewer aborts the installation if Cadence PCB 22.1 is already installed on the system. This includes installing into the same hierarchy as the previous versions of Allegro Free Physical Viewer. You may be required to reboot the computer once the installation has finished.

Note: Allegro Free Physical Viewer does not support silent installation.

What are the different versions of Allegro Free Physical Viewer available?

The following are the different versions of Allegro Free Physical Viewer available for download are:

- Allegro/OrCAD/SIP/MCM Free Physical Viewer 22.1 reads the designs for versions 17.4 and 17.2.
- Allegro/OrCAD/SIP/MCM Free Physical Viewer 17.4 reads the designs for versions 16.x and 17.2.
- Allegro/OrCAD/SIP/MCM Free Physical Viewer 16.6 reads the designs for versions 15.x and 16.6.
- Allegro/OrCAD/SIP/MCM Free Physical Viewer 15.x reads the designs for versions 14.x, 15.0, 15.1, 15.2, 15.5, 15.5.1, and 15.7.

You can download these versions from Allegro Downloads.

Is there any standalone Allegro Free Physical Viewer available for Linux or UNIX OS?

The Linux version of Allegro Free Physical Viewer gets installed with any of the SPB installation. The name of the executable file is `allegro_free_viewer` and is located at *<installation_directory>/tools/bin* folder.

A standalone version of Allegro Free Physical Viewer for Linux or UNIX platforms is not available.

Is it possible to cross-probe between Allegro Design Entry HDL to Allegro Free Physical Viewer?

It is not possible to crossprobe from Allegro Design Entry HDL to Allegro Free Physical Viewer as well as Allegro Design Entry CIS to Allegro Free Physical Viewer. Since Allegro Free Physical Viewer is considered a shareware program, it does not offer all functionalities of the actual product.

What is the difference between Allegro Free Physical Viewer and Allegro Physical Viewer Plus?

Allegro Physical Viewer Plus is a database viewer for Allegro PCB Editor, Allegro PCB SI, and Allegro Package Designer databases (`.brd` and `.mcm`). It is intended for use by the

manufacturing or design engineer to review designs and send comments and markups back to the design owner. This is facilitated through a markup layer that lets the reviewer add comments and markups to the design and email either the entire database or just the markup layer back to the design owner. Only the markup layer can be changed. The design data cannot be modified.

Allegro Physical Viewer Plus also includes the capability to view database constraints through the Constraint Manager system. However, none of the design constraints can be modified.

This product is available on the Windows, UNIX, and Linux operating systems. The Allegro Physical Viewer Plus features include:

- Pan, zoom, colors
- Ratsnest on/off
- Highlight & dehighlight
- Editing on a markup layer
- Show element
- Measure
- Find
- Reports
- Printing and plotting

Allegro Physical Viewer Plus is a licensed product and the Cadence Sales Representative can help you find more details about the Sales Packages included in this product. Allegro Free Physical Viewer on the other hand can be downloaded free from the following location:

https://www.cadence.com/en_US/home/tools/allegro-downloads-start.html

After you install Allegro Free Physical Viewer, refer to the `readme.rtf` file to know the features it supports.

Is there any Allegro Free Physical Viewer available when I install OrCAD and Allegro (SPB) products? If yes, how can I access it?

Yes, Allegro Free Physical Viewer gets installed when installing full-fledged products from SPB installation.

Once the SPB installation is finished, you can:

1. Navigate to installation directory
   <installation_directory>:\Cadence\SPB_22.X\tools\bin
2. Look for the `allegro_free_viewer` executable

You can either double-click on the executable or invoke it through the command prompt.

Why Allegro Free Physical Viewers 2022 aborts while installing?

If you have Cadence OrCAD and Allegro release 22.1 products installed, Cadence PCB Viewers are already installed in the system and Cadence PCB Viewer 2022 installation will not proceed.

To access:

- Allegro Capture Viewer, use
  <installation_directory>\tools\bin\capture.exe -viewer
- Allegro PCB Editor viewer, use
  <installation_directory>\tools\bin\allegro_free_viewer.exe
- Allegro Physical Viewer Plus, use
  <installation_directory>\tools\bin\allegro.exe -viewer
- APD+/SiP Viewer, use
  <installation_directory>\tools\bin\allegro_free_viewer.exe -sip

How to enable OpenGL in Allegro Free Physical Viewer?

To enable OpenGL in Allegro Free Physical Viewer, set an environment variable *viewer_useoglgraphics* and restart the application.

By default, OpenGL on Windows is disabled for Allegro Free Physical Viewer.

Can SKILL be run on Allegro Free Physical Viewer?

Allegro Free Physical Viewer cannot use any customization and therefore, the SKILL programs cannot be used with this product.

How to use the Alt key for selecting a single layer in the Visibility panel on Linux machines?

Open Windows Preferences dialog using System – Preferences – Windows menu and set the *Movement Key* to *Super.* You can use both Alt and Control keys for single layer and layer pair selection, respectively.

Error Messages and Solution

Error: This application has failed to start because its side-by-side configuration is incorrect.

This error comes when the system does not have the Microsoft Visual C++ redistributable installed. To resolve this error, install required Microsoft Visual C++ redistributable.

Cadence provides the required versions of Microsoft Visual C++ with the SPB installation. The files are located at *<installation_directoty>/tools/mbase* folder.

This location contains two directories:

- vn2005: Microsoft 2005 Redistributable Packages for 32bit and 64bit:
  - Required for SPB 17.0 or earlier releases (for example, 16.x and 15.x).
  - The 64bit package is only required for certain Signal Integrity binaries starting with the 16.5 release
- vn2012: Microsoft 2012 Redistributable Packages for 32bit and 64bit
  - Both packages are required for SPB release 17.0

Installing Cadence software via the Cadence-provided installer does not require these packages. If Cadence SPB software aborts on the startup, you will need to install these packages.

`vcredist_x86.exe` is a Microsoft Redistributable Package, which installs runtime components required SPB 16.2 onward.

`vcredist_x64.exe` is a Microsoft Redistributable Package, which installs runtime components required to run 64bit executables.

With SPB 17.4-2019 installation, redistributable files are available for Microsoft 2005 and 2012.

“ERROR (SPMHOD36): Cannot open drawing” reported when opening designs using Allegro/Sip FREE Physical Viewer

This error comes when you try to open a board database (`.brd`) using APD+ Free Physical Viewer or if you try to open a SiP / MCM database using Allegro Free Physical Viewer.

The correct product options you need to use to open the database without any error message:

- Use Allegro Free Physical Viewer (*Cadence PCB Viewers 2022 – PCB Editor Viewer 2022*) to open and view a `*.brd` file.
- Use APD+ Free Physical Viewer (*Cadence PCB Viewers 2022 – APD+ Viewer 2022*) to open and view a `.sip` or `.mcm` file.

ERROR 1152: Error extracting to the temporary location

This error usually occurs when there is insufficient disk space where the files are being extracted or the installer is corrupted. Check the following conditions in sequence to resolve this error:

1. Ensure enough disk space is available
2. Verify that TEMP folder has write permission
3. Ensure no spaces in the TEMP / TMP variable values, for example, C:/TEMP or C:/TMP.

Try download and install Allegro Free Physical Viewer again.