



OrCAD X Presto User Guide

Product Version 23.1
September 2023

© 2023 Cadence Design Systems, Inc.
Printed in the United States of America.

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

Open SystemC, Open SystemC Initiative, OSCI, SystemC, and SystemC Initiative are trademarks or registered trademarks of Open SystemC Initiative, Inc. in the United States and other countries and are used with permission.

Trademarks: Trademarks and service marks of Cadence Design Systems, Inc. (Cadence) contained in this document are attributed to Cadence with the appropriate symbol. For queries regarding Cadence's trademarks, contact the corporate legal department at the address shown above or call 800.862.4522.

All other trademarks are the property of their respective holders.

Restricted Permission: This publication is protected by copyright law and international treaties and contains trade secrets and proprietary information owned by Cadence. Unauthorized reproduction or distribution of this publication, or any portion of it, may result in civil and criminal penalties. Except as specified in this permission statement, this publication may not be copied, reproduced, modified, published, uploaded, posted, transmitted, or distributed in any way, without prior written permission from Cadence. Unless otherwise agreed to by Cadence in writing, this statement grants Cadence customers permission to print one (1) hard copy of this publication subject to the following conditions:

1. The publication may be used only in accordance with a written agreement between Cadence and its customer.
2. The publication may not be modified in any way.
3. Any authorized copy of the publication or portion thereof must include all original copyright, trademark, and other proprietary notices and this permission statement.
4. The information contained in this document cannot be used in the development of like products or software, whether for internal or external use, and shall not be used for the benefit of any other party, whether or not for consideration.

Disclaimer: Information in this publication is subject to change without notice and does not represent a commitment on the part of Cadence. Except as may be explicitly set forth in such agreement, Cadence does not make, and expressly disclaims, any representations or warranties as to the completeness, accuracy or usefulness of the information contained in this document. Cadence does not warrant that use of such information will not infringe any third party rights, nor does Cadence assume any liability for damages or costs of any kind that may result from use of such information. Cadence is committed to using respectful language in our code and communications. We are also active in the removal and replacement of inappropriate language from existing content. This product documentation may however contain material that is no longer considered appropriate but still reflects long-standing industry terminology. Such content will be addressed at a time when the related software can be updated without end-user impact.

Restricted Rights: Use, duplication, or disclosure by the Government is subject to restrictions as set forth in FAR52.227-14 and DFAR252.227-7013 et seq. or its successor.

Contents

1		8
OrCAD X Presto		8
Key Features		9
Documentation Links		9
2		10
Creating a PCB Layout in OrCAD X Presto: Basic Design Tasks		10
Creating a New Board Design		10
Related Topics		13
Configuring Component Footprint Libraries		14
Setting Up Layers Stack-Up		16
Placing Components		20
Routing the Board		21
Documenting PCB Design		23
Exporting Design Data		24
3		28
Authoring and Editing Footprints in OrCAD X Presto		28
Editing Footprints		28
Related Topics		31
Adding Pins to a Footprint in OrCAD X Presto		32
Adding Pins by Searching Padstacks		33
Related Topics		34
Adding Pins with the Padstack Editor		35
Related Topics		36
Adding Pins with the Padstack Wizard		37
Related Topics		41
Using the Add Pin Widget		42
4		46
Setting Up the OrCAD X Presto Design Environment		46
Setting Up Grids		46
Setting Up Backup Preferences		48
Controlling Visibility of Design Objects		49

Controlling Visibility of Individual Objects	52
Customizing Colors of Design Objects	59
Controlling Visibility in 3D	60
Customizing Keyboard Shortcuts in OrCAD X Presto	65
5	69
Editing Design Elements in OrCAD X Presto	69
Copying Objects	69
Measuring the Distance between Objects in OrCAD X Presto	72
Swapping Components, Diff Pairs, and Pins in OrCAD X Presto	75
Swapping Pins in OrCAD X Presto	77
Swapping Diff Pairs in OrCAD X Presto	83
Swapping Components in OrCAD X Presto	87
Creating Custom Design Outline in OrCAD X Presto	91
6	99
Managing Shapes in OrCAD X Presto	99
Creating and Adding Shapes in OrCAD X Presto	99
Drawing Rectangles and Circles	100
Editing Shape Edges in OrCAD X Presto	101
7	104
Creating Zones and Bends for Rigid-Flex Design in OrCAD X Presto	104
Related Topics	109
Adding Zone Stackups in OrCAD X Presto	109
Creating and Editing Bends in OrCAD X Presto	112
Creating and Editing Zones in OrCAD X Presto	116
8	120
Routing Nets in OrCAD X Presto	120
Resolving Timing Requirements with Delay Tuning	120
Splitting Stacked Vias	124
Using Structures for Quick Routing in OrCAD X Presto	127
Related Topics	127
Creating Via Structures in OrCAD X Presto	127
Prerequisite	127
Steps	127
Placing Via Structures in OrCAD X Presto	131
Prerequisite	131

Steps	131
Related Topics	132
Editing Via Structures in OrCAD X Presto	133
Prerequisite	133
Steps	133
Related Topics	134
9	136
Documenting PCB with LiveDoc in OrCAD X Presto	136
What is LiveDoc?	137
Key Features of LiveDoc	137
Related Topics	138
Launching LiveDoc	139
Related Topics	144
LiveDoc User Interface	145
LiveDoc Canvas	145
Panels	147
Toolbars	151
Related Topics	154
LiveDoc Preferences	154
Related Topics	156
Using LiveDoc Templates	157
Creating Custom Templates	157
Related Topics	158
LiveDoc Workflow	159
Related Topics	160
Adding Page in LiveDoc	161
Related Topics	162
Defining Page Title Block	163
Related Topics	165
PCB Views in LiveDoc	166
Related Topics	167
Placing PCB Views in LiveDoc	168
Related Topics	168
Placing a PCB View in LiveDoc Canvas	170
Placing Stackup Charts and Tables in LiveDoc Canvas	173
Placing Drill Charts in LiveDoc Canvas	178

Viewing and Sharing PCB Views in LiveDoc	180
Changing Display of PCB Views	180
Customizing PCB Views	181
Exporting LiveDoc Data to PDF	183
Related Topics	185
Defining Dimensions in LiveDoc	186
Adding Linear Dimensions	186
Adding Angular Dimensions	192
Adding Ordinate Dimensions	193
Adding Leader-Oriented Dimensions	194
Related Topics	198
Defining Annotations in LiveDoc	199
Related Topics	202
Managing LiveDoc	203
Moving LiveDoc Tab to a New Workspace	203
Closing LiveDoc	203
Reopening LiveDoc	203
Deleting LiveDoc	204
Related Topics	204
10	206
Reviewing and Analyzing Designs in OrCAD X Presto	206
Analyzing Design in OrCAD X Presto 3D Environment	206
Generating 3D Exports in OrCAD X Presto	206
Overview of the OrCAD X Presto 3D Environment	209
Component Shape Visibility	209
Footprint Boundaries	209
3D Models	210
Reviewing Designs Using Comments and Markups in OrCAD X Presto	213
Adding Comments and Markups in OrCAD X Presto	213
Editing Comments and Markups in OrCAD X Presto	218
Viewing and Navigating Comments and Markups in OrCAD X Presto	221
Generating Reports in OrCAD X Presto	227
Dangling Traces, Vias and Antenna Report	230
Related Topics	230
Missing Tapers Report	231
Missing Teardrops Report	232

Shape Islands Report	233
Unassigned Shapes Report	235
11	237
OrCAD X Presto Frequently Asked Questions	237
Why am I not able to place a PCB view in the LiveDoc canvas?	237
Why do I see incorrect PCB views after editing the board design outline?	238
How can I find unassigned shapes?	239

OrCAD X Presto

OrCAD X is an innovative design platform that caters to the needs of individuals and small to midsize businesses. It focuses on providing a cohesive and comprehensive solution for all design requirements. The addition of the X in this new product platform signifies its ability to extend its capabilities to the cloud, enabling you to leverage additional services like X AI.

OrCAD X Presto—a new layout environment within the OrCAD X platform—offers a cutting-edge solution for layout design. The interoperability between OrCAD X Presto and Allegro layout editor ensures compatibility and easy transition of layout designs. OrCAD X Presto can be used in cloud-connected and unconnected modes, enabling you to work flexibly, based on your preferences and requirements.

The user interface of OrCAD X Presto is designed to be intuitive, capable, efficient, and accessible. It offers compact menus and a streamlined workflow, enabling you to create layout designs quickly and efficiently. Advanced features and functionalities are readily available, empowering designers to deliver high-quality designs efficiently. The informative and interactive panels provide valuable insights, accelerate search and navigation, and effectively manage properties and visibility, ensuring a productive and seamless user experience. OrCAD X Presto eliminates the need for modal dialog boxes. Instead, layout toolbars and floating menus reduce distractions and ensure the design space is unblocked and accessible, enhancing productivity and allowing you to focus on your work.

Additionally, OrCAD X Presto includes an integrated 3D viewer that seamlessly switches between 2D and 3D views. This feature enables designers to perform fast and accurate 3D analysis, supporting 3D clearance Design Rule Checks (DRCs). This integration enhances visualization capabilities, enabling you to identify and address potential manufacturing issues quickly and efficiently.

OrCAD X Presto is a powerful, user-friendly, and feature-rich layout environment that facilitates rapid design iterations, reduces time-to-market, and ensures high-quality layouts.

Key Features

The following are the key features of OrCAD X Presto:

- [LiveDoc](#)
- [Design Markup](#)
- [3D X](#)
- X AI – The next-generation design solution that leverages generative AI algorithms and off-site, scalable computational resources to synthesize PCB designs. This cloud-based service brings to the PCB design world a level of automation seen only in the digital IC space. It removes time-to-market bottlenecks by parallelizing tasks and freeing critical design resources. It accelerates placement, power planes generation, as well as critical net routing, providing early insights into feasibility, form and fit, and exploring a wide solution space to find optimal and cost-effective solutions, all from within the Allegro X platform.

Documentation Links

- [OrCAD X Presto User Guide](#)
- [Quick Reference Guide to OrCAD X Presto Interface and Commands](#)
- [OrCAD X Capture with OrCAD X Presto Tutorial](#)
- [OrCAD X Presto Known Problems and Solutions](#)

Creating a PCB Layout in OrCAD X Presto: Basic Design Tasks

The OrCAD X Presto design environment enhances the productivity of layout designers and accelerates the layout design process. Its intuitive and efficient features accelerate the overall PCB design turnaround time and reduce time to market.

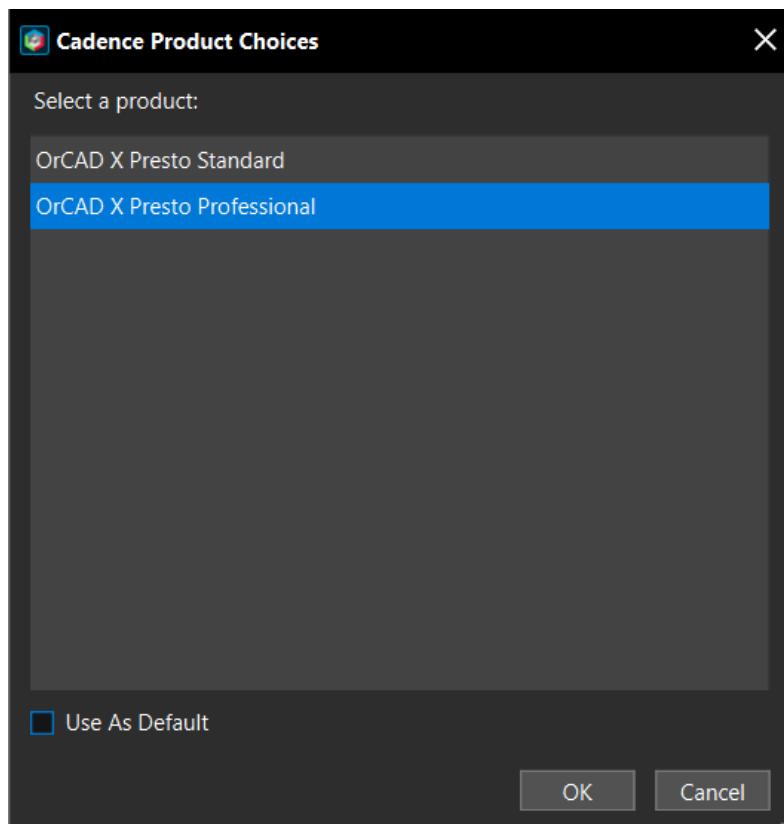
To create a PCB layout in OrCAD X Presto, you need to complete the following design tasks:

- [Creating a New Board Design](#)
- [Configuring Component Footprint Libraries](#)
- [Setting Up Layers Stack-Up](#)
- [Placing Components](#)
- [Routing the Board](#)
- [Documenting PCB Design](#)
- [Exporting Design Data](#)

Creating a New Board Design

Layout design begins with creating a new board design. To do so, follow these steps:

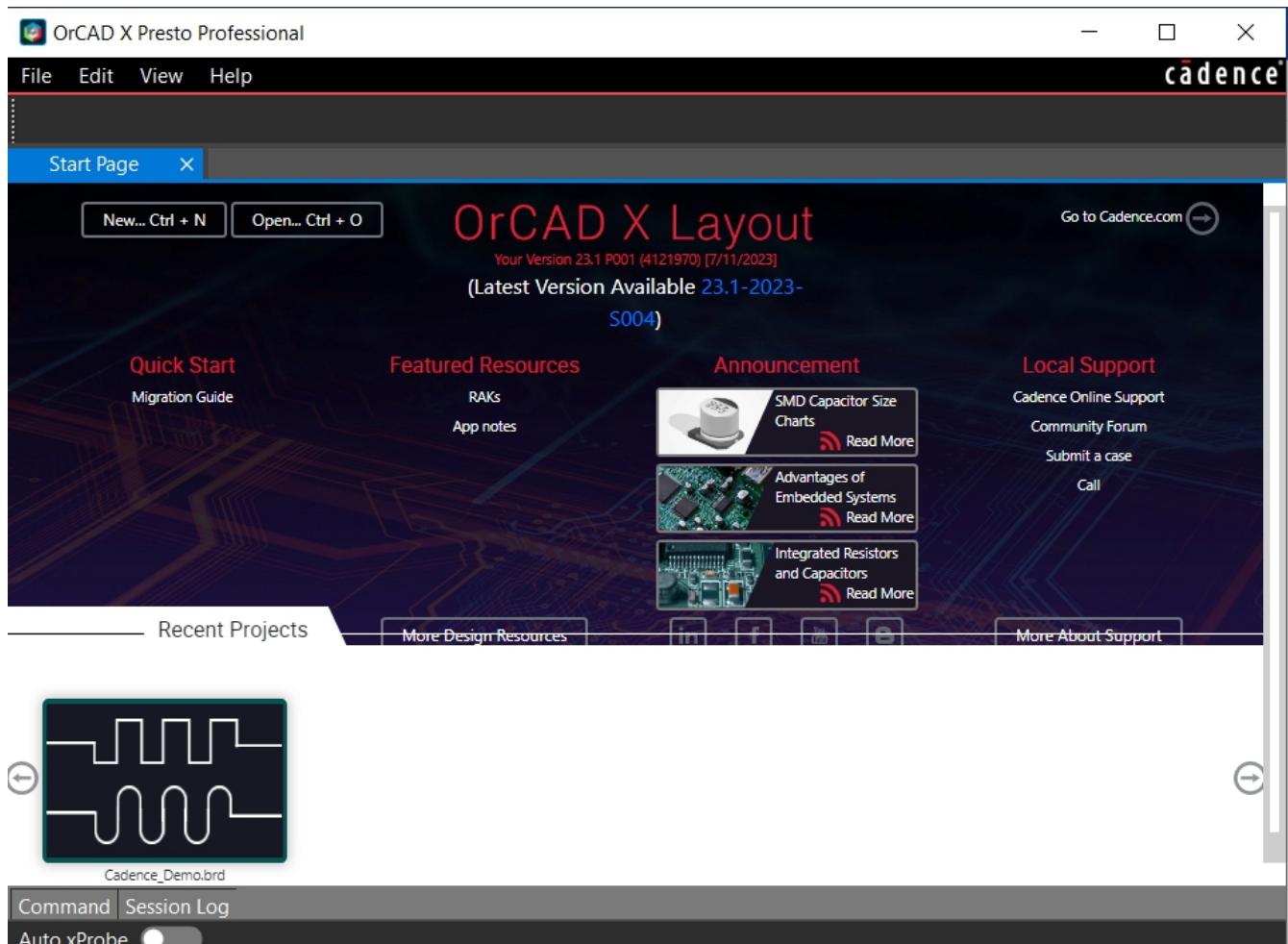
1. Choose *Start – Cadence PCB 2023 – OrCAD X Presto 2023* on Windows.
The *Cadence Product Choices* dialog box is displayed.



2. Choose a product license and click *OK*.
OrCAD X Presto is launched, and the *Start Page* is displayed.

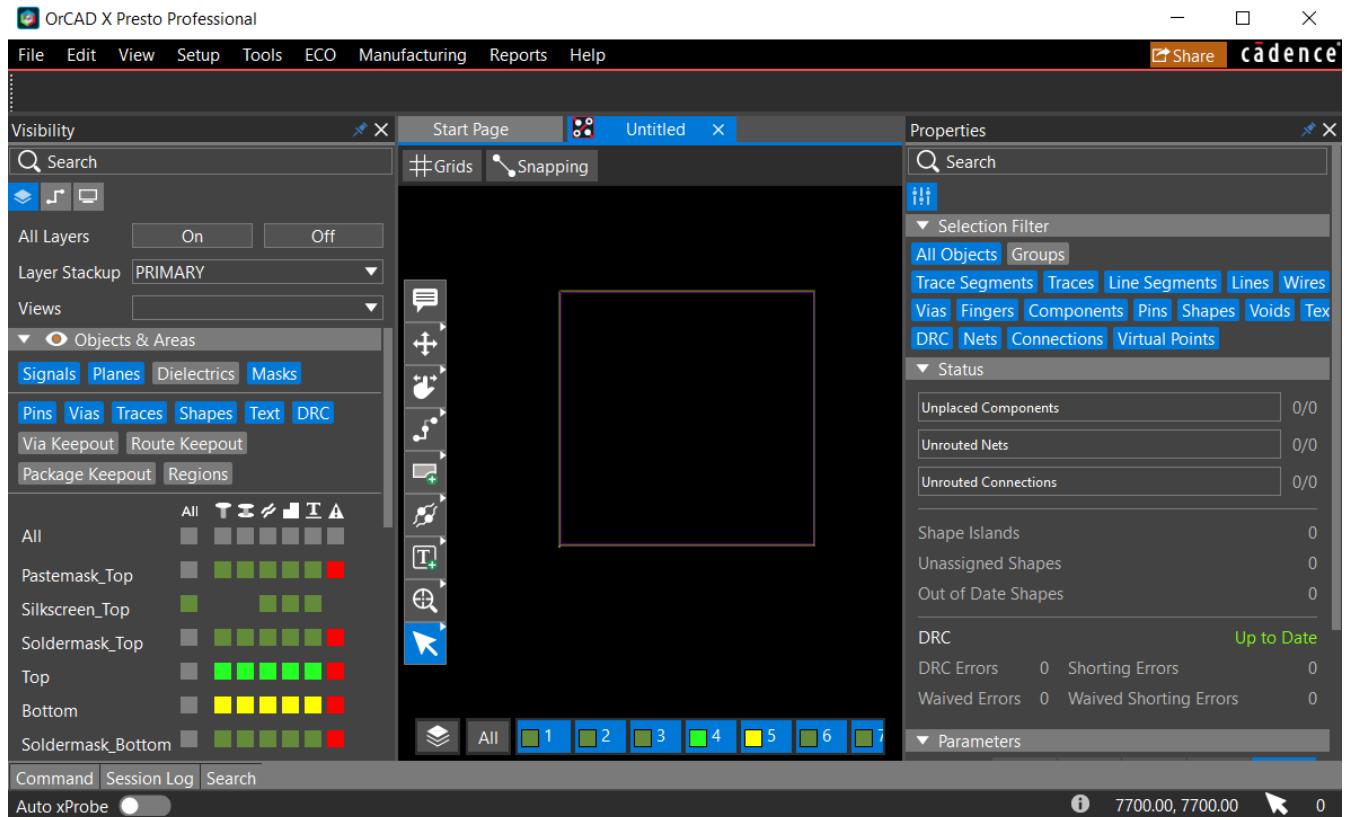
OrCAD X Presto User Guide

Creating a PCB Layout in OrCAD X Presto: Basic Design Tasks--Creating a New Board Design

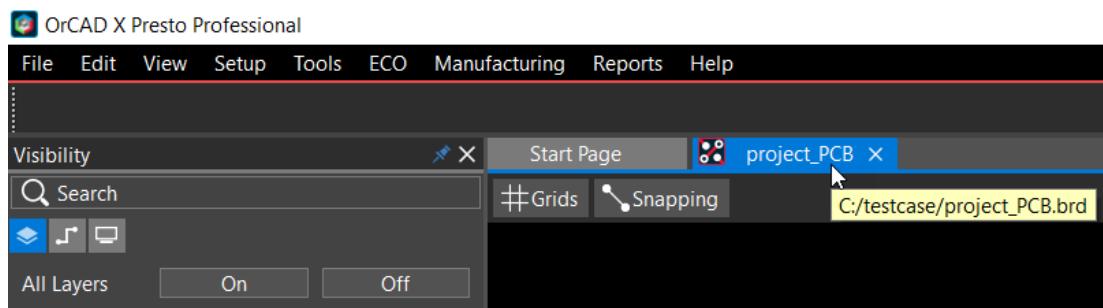


3. Choose *File – New – Board* to create a new board design (.brd) file.

A blank board (`Untitled.brd`) with a default rectangular design outline opens in a new tab.



4. Choose *File – Save As* to specify the name and path of the board design.
The design name shows in the tab. A tooltip displays the path when you hover the cursor over the tab.



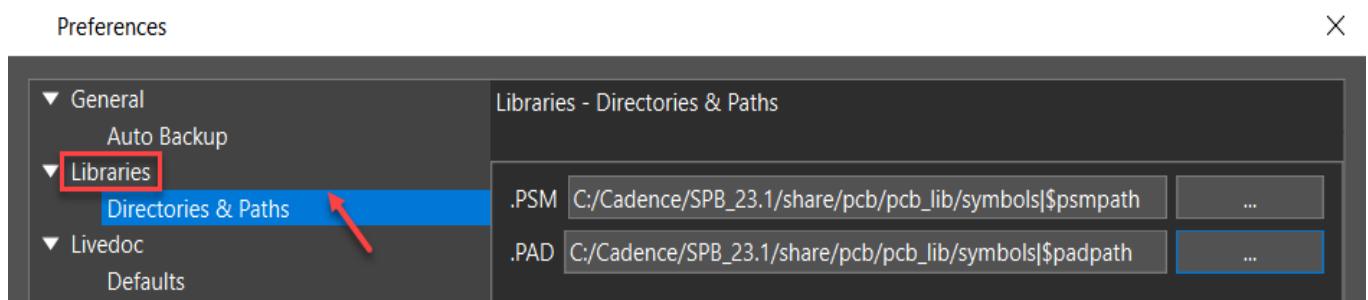
Related Topics

- Quick Reference Guide to PCB Editor X Interface and Commands

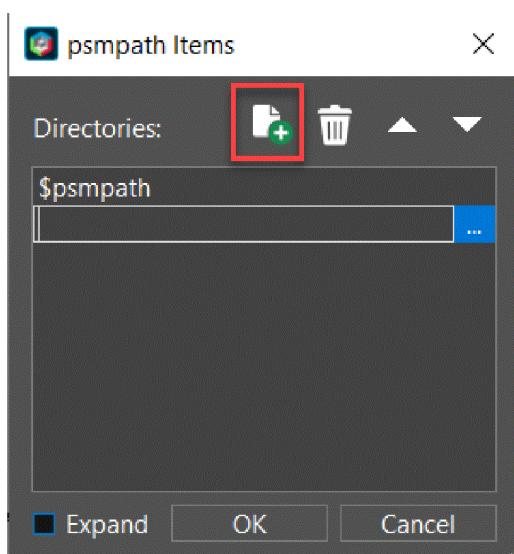
Configuring Component Footprint Libraries

After creating a blank layout (.brd) in OrCAD X Presto, you need to define the path of footprint libraries. A footprint library is a collection of graphic symbols representing packages (*.sm), mechanical elements, drawing formats (*.dra), and custom pads and padstacks (*.pad). To specify the path of component footprint libraries, do the following:

1. Choose *Edit – Preferences* to open the *Preferences* dialog box.
2. In the left pane of the dialog box, select the *Directories & Paths* folder under *Libraries*.



3. To define the path to package symbols (footprints), click the browse button for the *.PSM* field. The *psmpath Items* dialog box is displayed.
4. To add a library path, click the icon.

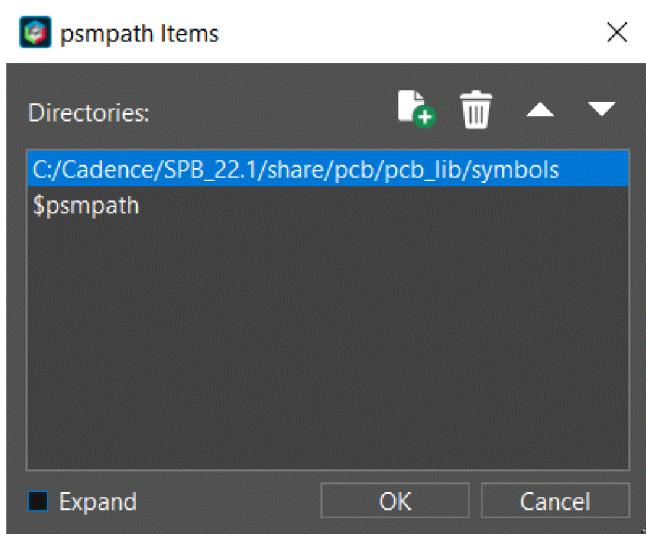


An empty item is added to the list.

5. Click the browse button next to the newly added item.

The *Select Directory* dialog box opens.

6. Navigate to the location of the footprints library directory and click *OK*.
7. Click the up arrow button to set the path to the footprint library before the default library.



⚠ If there are common components in both the libraries, the library that appears first in the list is honored.

8. Click *OK* to close the *psmpath Items* dialog box.
9. Repeat steps 3–8 for the `.PAD` field to specify the path of the padstack library.
10. Click *OK* to close the *Preferences* dialog box.

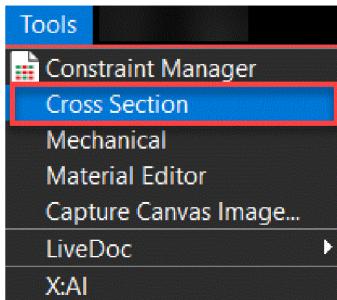
Setting Up Layers Stack-Up

When creating the layer stack-up, it is necessary to define the various characteristics of PCB layers, including their material, layer type, name, thickness, dielectric constant, and electrical conductivity. While routing, you might need to insert additional routing layers because the design is too dense to complete or delete extra layers because of an ECO.

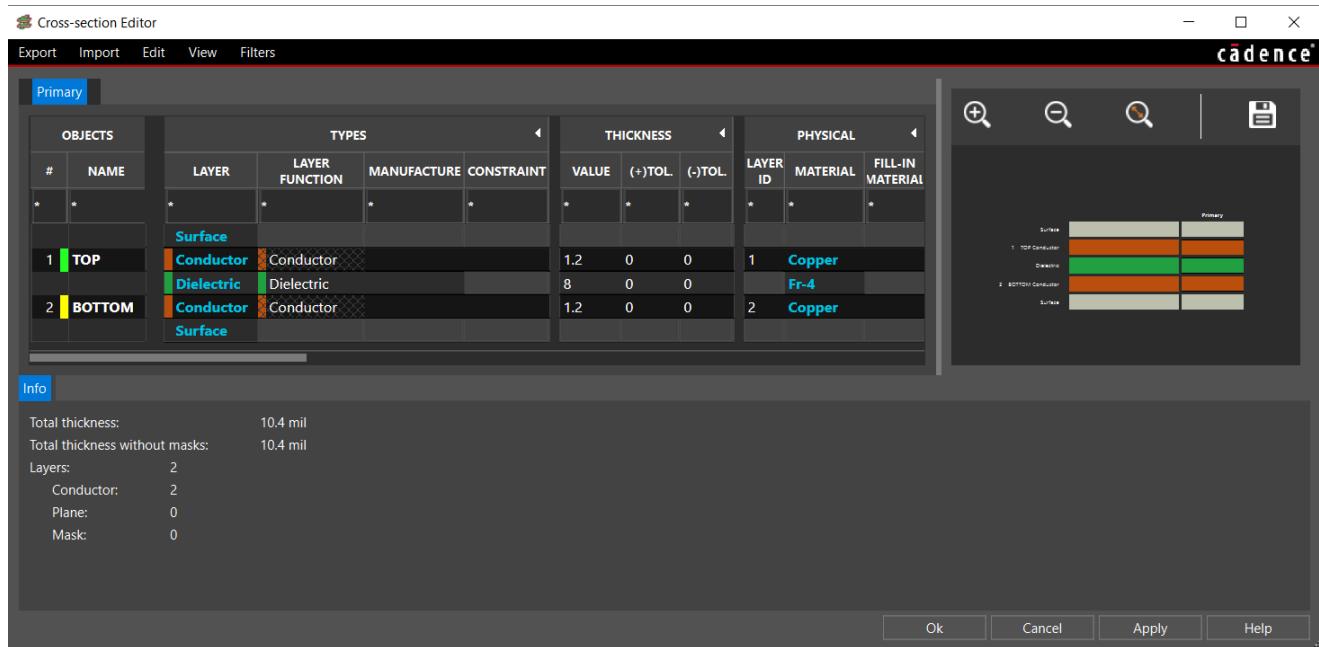
OrCAD X Presto provides a spreadsheet-based utility, Cross-section Editor to specify an accurate layer stack-up. You can view and modify the board layer setup in the Cross-section Editor window by adding or deleting layers.

Add or modify layers as follows:

1. Choose *Tools – Cross Section*.

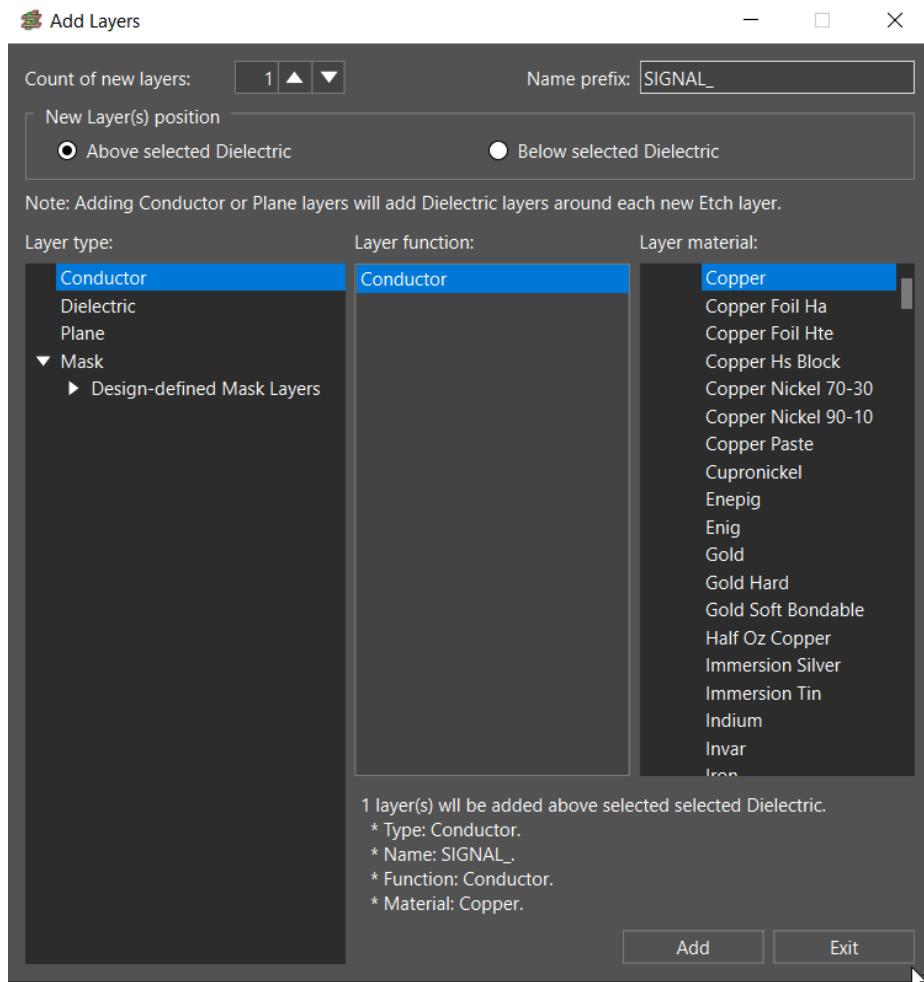


The Cross-section Editor window opens. The spreadsheet rows represent the primary layer material (stack-up), and the columns represent the various properties of each layer (conductor and dielectric). The rows are displayed in the physical order, starting from the **TOP** layer and ending at the **BOTTOM** layer.



- To add a new layer, right-click a layer in the *Primary* tab and choose *Add Layer Above* or *Add Layer Below*.

The *Add Layers* dialog box opens.



3. Specify the number of layers to be added, prefix for layer name, position of the new layers, layer type, layer function, and layer material, and click *Add* in the *Add Layers* dialog box. New layers are added above or below the selected layer.
4. Click *Exit* to close the *Add Layers* dialog box.
5. Optionally, choose *Add Layer Pair Above* or *Add Layer Pair Below* to add a pair of layers.
6. To change a layer definition, do the following steps in the *Object* column:
 - a. Select the *Layer Name* cell and specify a name for the layer.
 - b. In the *Layer Function* column, specify the layer type to define the purpose that layer serves in the layout design. For example, *Signal*.
 - c. In the *Material* column, choose a material type from the list to define the electrical characteristics of the layer provided by the manufacturing partner. For example, *Copper*.
7. In the *Thickness* column of the layer, enter a number representing the thickness for that layer.

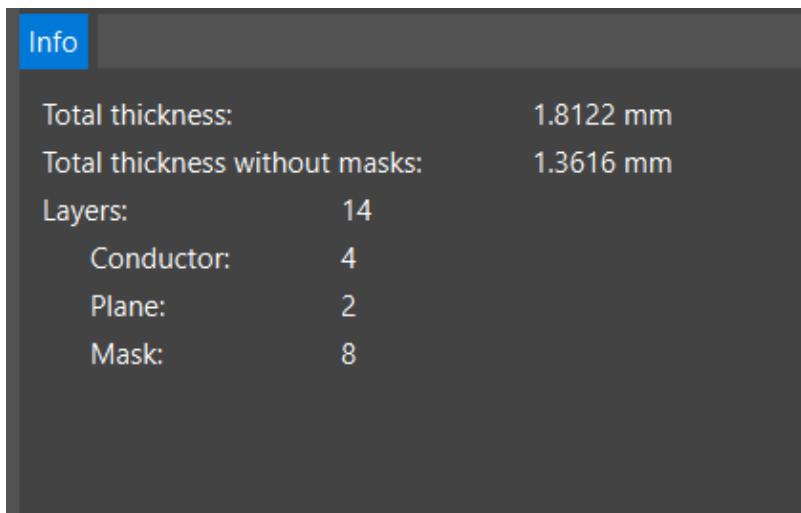
The thicknesses of an individual layer can be derived from the manufacturing data supplied by the manufacturing partner.

8. In the *Conductivity* column, enter the value of the electrical conductivity of the layer as provided with the manufacturing specifications.

 The default settings for the Cross-section Editor window are stored and maintained in the `materials.dat` file that exists in the following location:

`<installation_directory>\latest\share\pcb\text`

9. Repeat steps 2–6 to add more layers to the board.
10. Select the layer, right-click, and choose *Remove Layer* delete a layer from the cross-section.
11. View the cross-sectional summary of a board design in the *Info* tab at the bottom of the Cross-section Editor window. This *Info* tab displays the total number of layers of each type and their thickness.

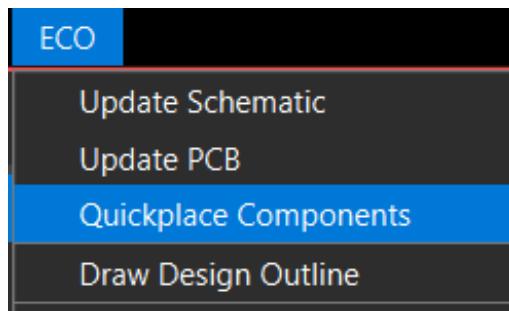


12. Close the Cross-section Editor window when you have finished setting up layers.

Placing Components

To place components in a design in OrCAD X Presto, do the following:

1. Choose *ECO – Quickplace Components* to place all the components in the design canvas in a single step.



All components are placed along the longest edge of the design outline shape. Rats are displayed between pins of the same net and help you place connected components close to each other.

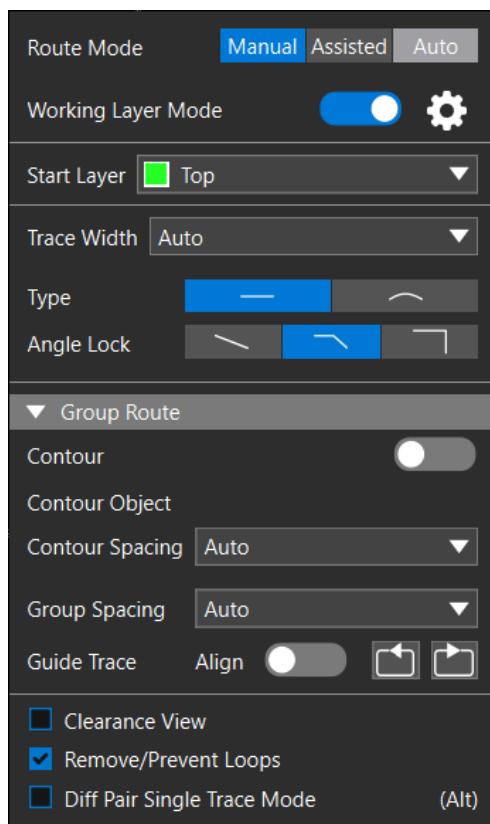
2. Click the *Move* icon  from the functional toolbar to move symbols inside the design outline. The selected symbol attaches to the cursor.
3. Optionally, in the *Arrange* pane of the *Properties* panel, click the *Rotate Left* or *Rotate Right* button to rotate the symbol by 90 degrees either clockwise or counterclockwise.
4. Check the *Status* pane for *Unplaced Components* in the *Properties* panel to verify that all the components are placed.

Routing the Board

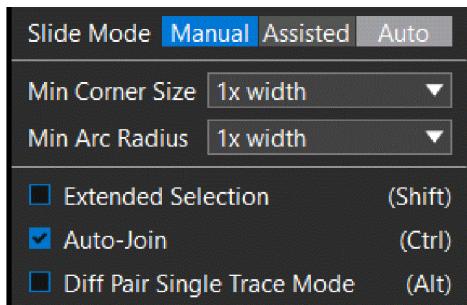
OrCAD X Presto provides various options to route connections on a board. Route a net as follows:

1. Click the *Add Connect*  icon from the functional toolbar.

The route widget opens, displaying options for routing single connect lines (traces) and differential pairs.



2. Set *Route Mode* to *Manual*.
3. Set *Trace Width* to *Constraint*.
The default width of the connect line is defined in Constraint Manager.
4. Choose a net to route and move the cursor to start routing.
5. To route the trace on a different layer, double-click to insert a via.
6. To adjust the route, click the *Slide*  icon from the functional toolbar.
The slide widget displays options to slide the traces.



7. Set the *Slide Mode* to *Manual*.
8. Route all of the nets in the design.
9. Click the zoom fit icon  from the functional toolbar to view the complete design.

Documenting PCB Design

Once the layout design is completed and checked for errors, the documentation phase begins, which includes the creation of fabrication and assembly drawings, generation of drill data, artwork, and so on. OrCAD X Presto offers *LiveDoc*, which automates the documentation process using predefined templates to quickly capture the manufacturing and assembly requirements.

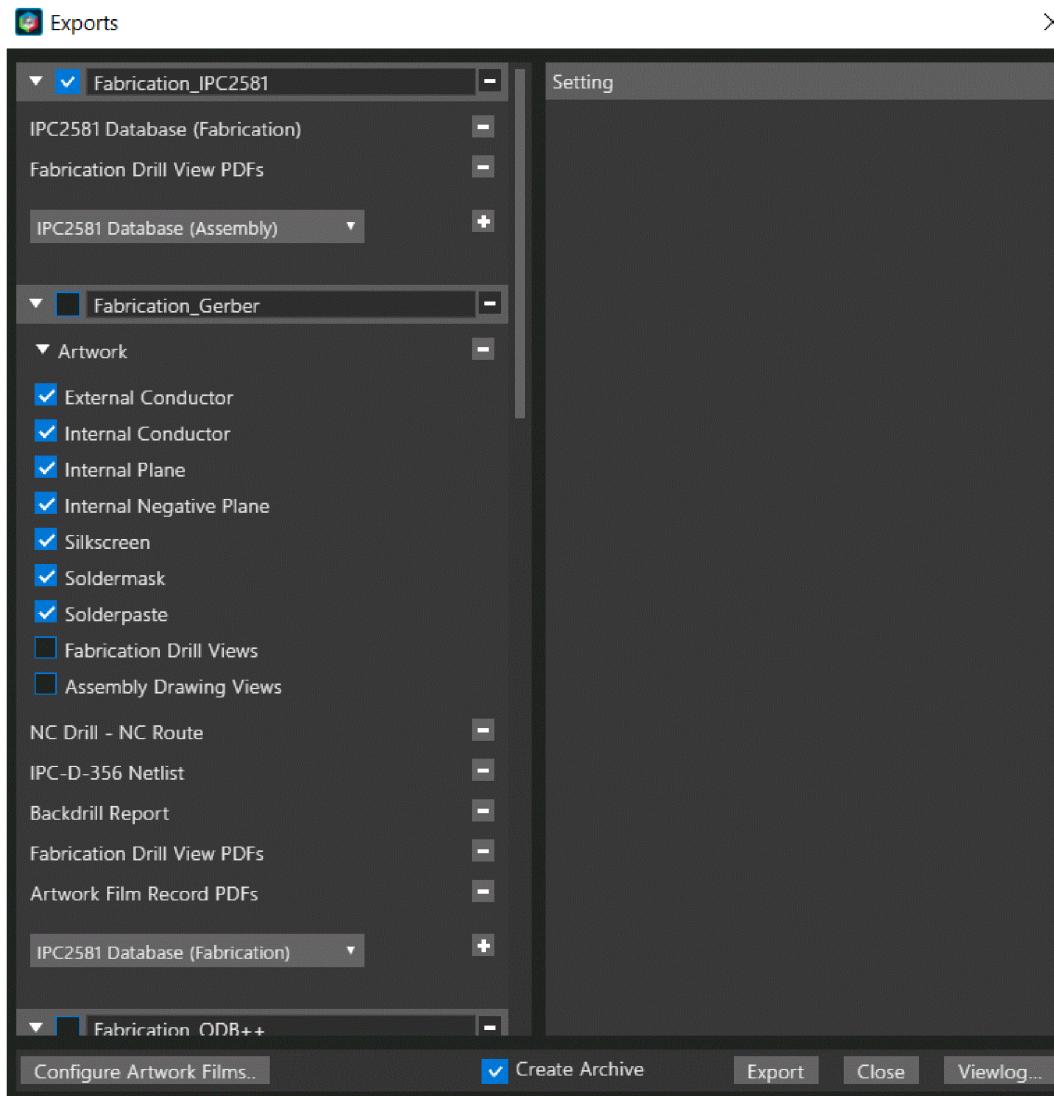
Exporting Design Data

The *Exports* dialog box generates manufacturing output (fabrication and assembly data). You can select multiple manufacturing output formats and generate all the data as an archived file in a single step.

To generate output files, follow these steps:

1. Choose *Manufacturing – Export to Manufacturing*.

The *Exports* dialog box opens. Default manufacturing output formats are listed on the left, while settings are displayed on the right.

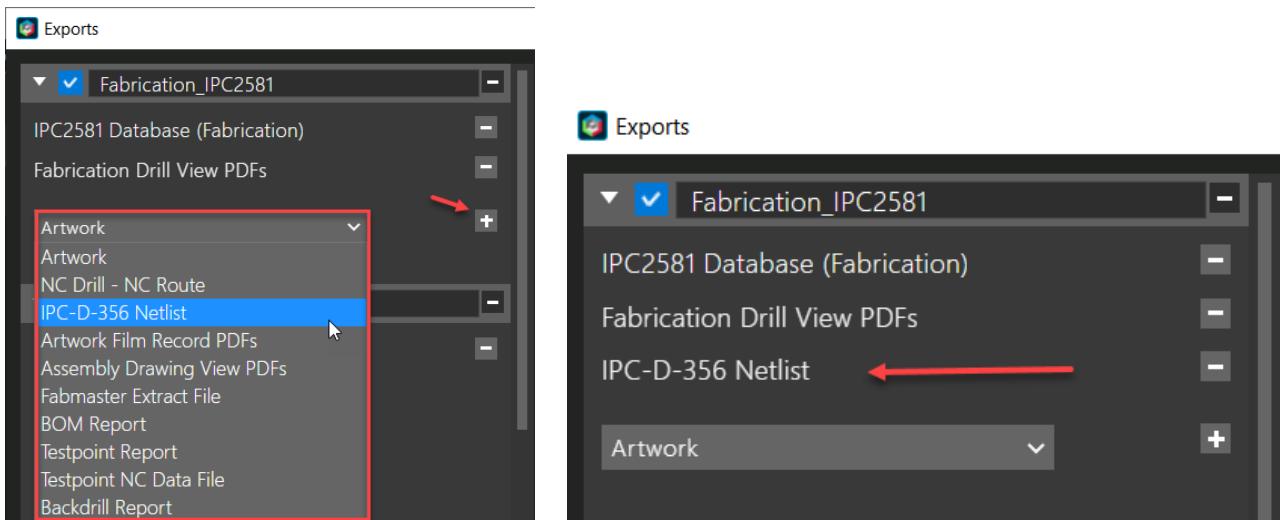


2. To select a manufacturing output, enable the check box before it. By default,

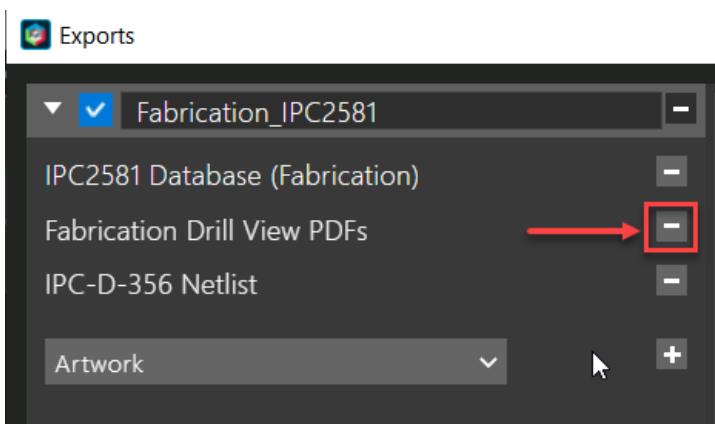
Fabrication_IPC2581 is selected.

 An installation of ODB++ Inside the software package is required to generate output files for *Fabrication_ODB++*.

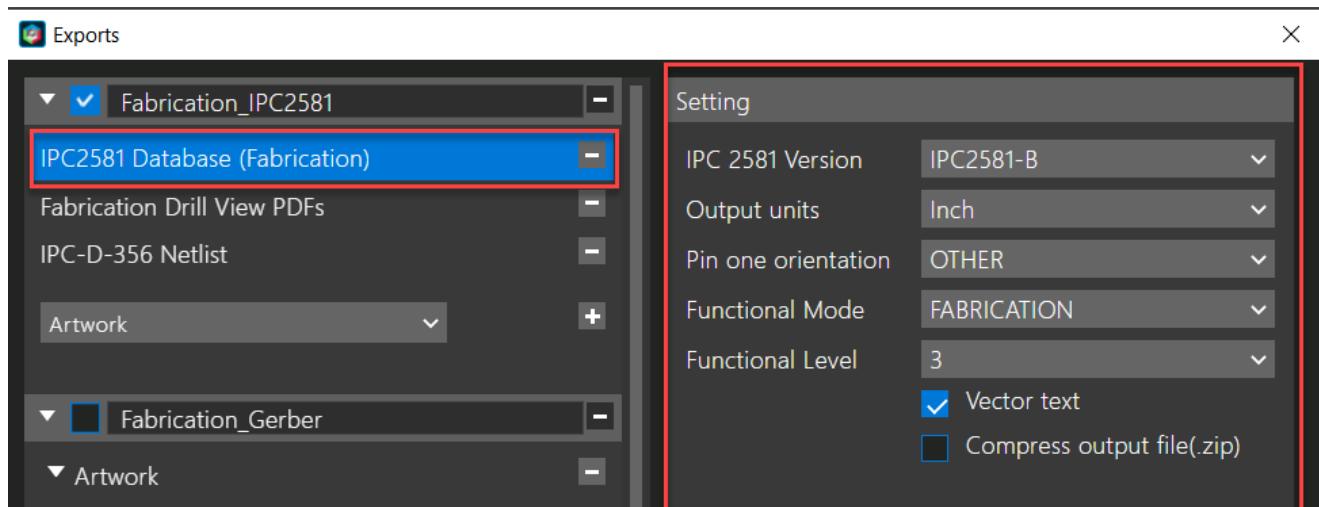
3. To generate a manufacturing data format, choose it from the list and click the  icon.



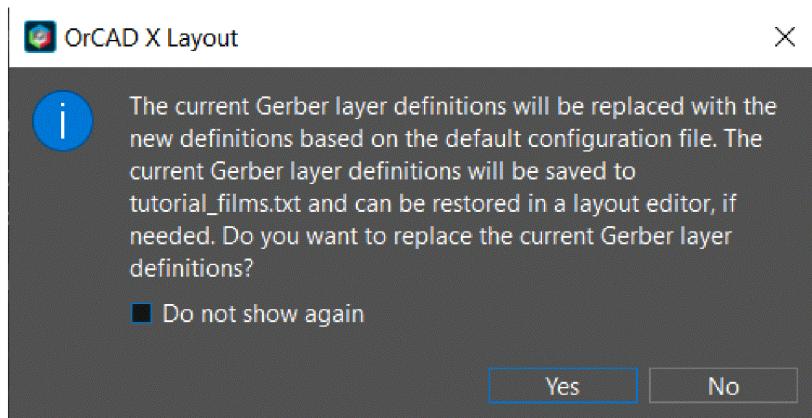
4. Click  to remove a manufacturing data format you do not want to generate.



5. To modify the parameters of a manufacturing data format, choose it in the left section and change the values in the *Setting* section displayed on the right.



6. Ensure the *Create Archive* check box is selected.
7. Click *Export* to generate all the selected manufacturing outputs as archived files.
8. Click *Yes* in the following message prompt.

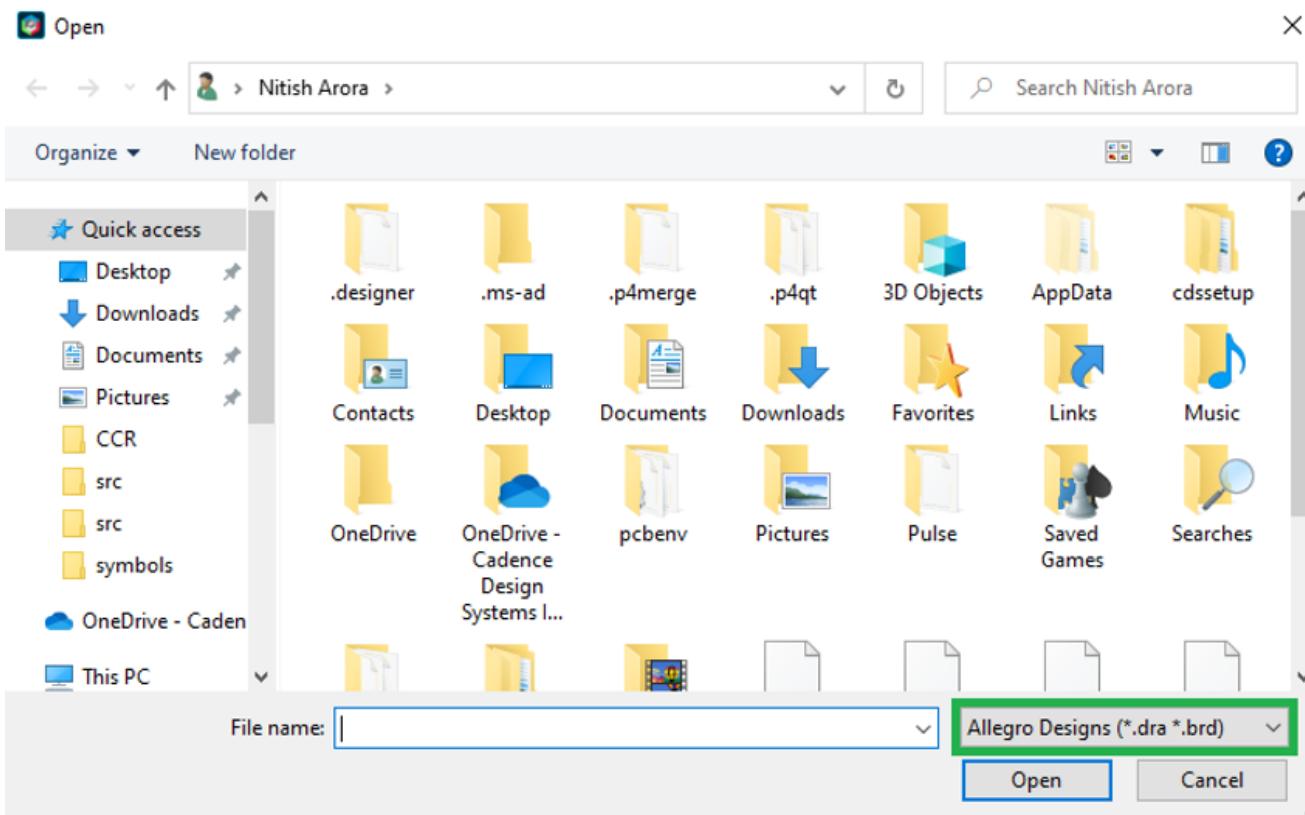


A progress bar displays the status of the export process.

9. Click *Viewlog* to open the log file to review the errors and warnings.

Authoring and Editing Footprints in OrCAD X Presto

Launch OrCAD X Presto, and open an existing PCB symbol .dra file.

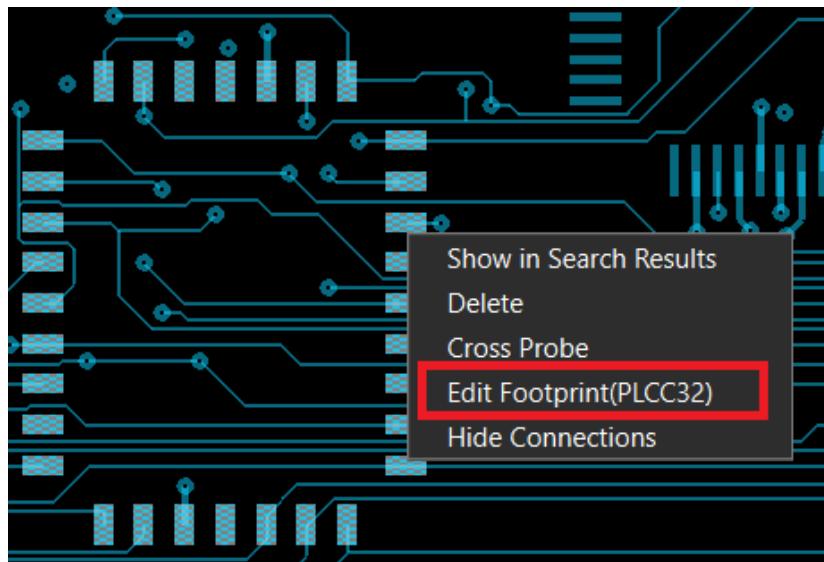


You can switch between opening .brd and .dra files in same session.

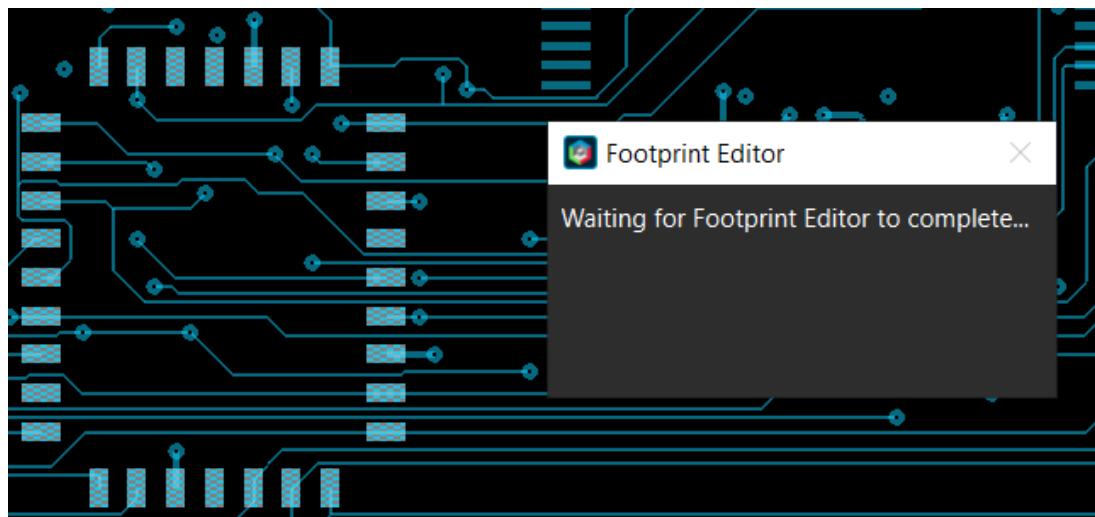
Editing Footprints

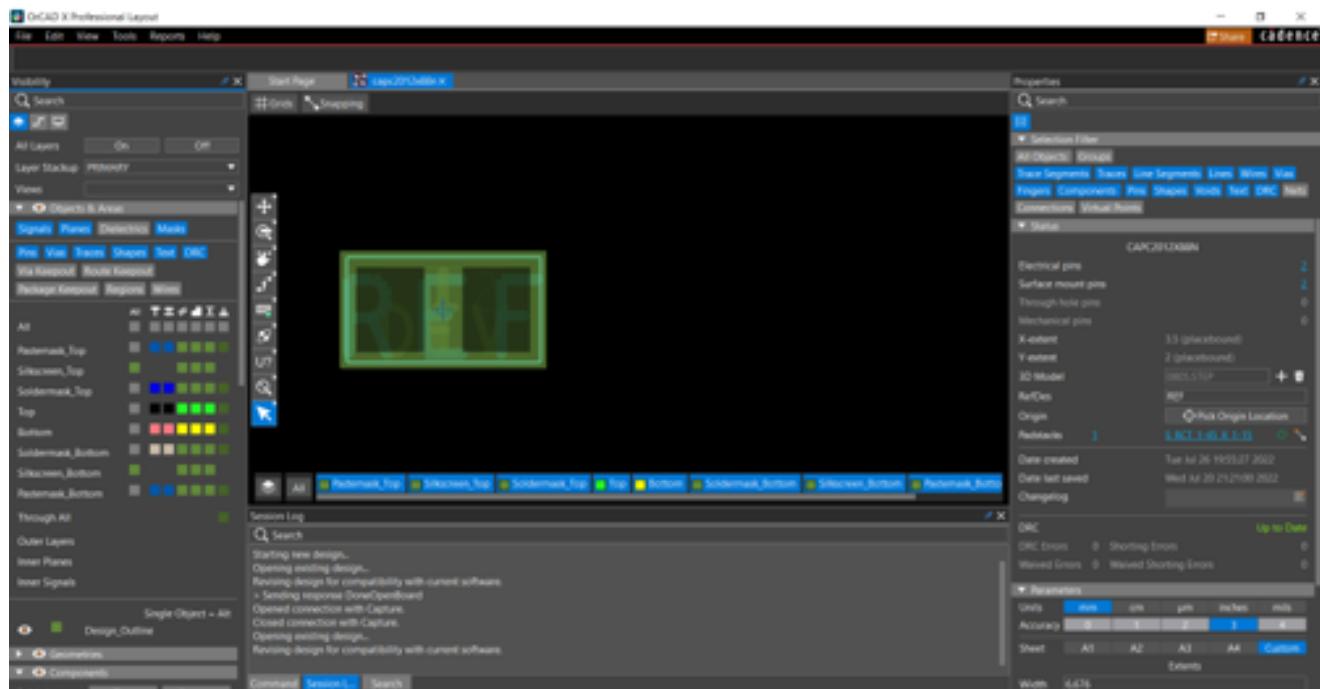
Open your layout design in OrCAD X, and do the following:

1. Right-click on a symbol and choose *Edit Footprint*.

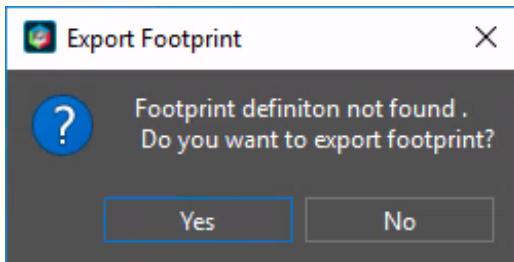


The symbol opens in another instance of OrCAD X—that is, the Footprint Editor (which uses the same license as the parent application, so you do not need an additional license).

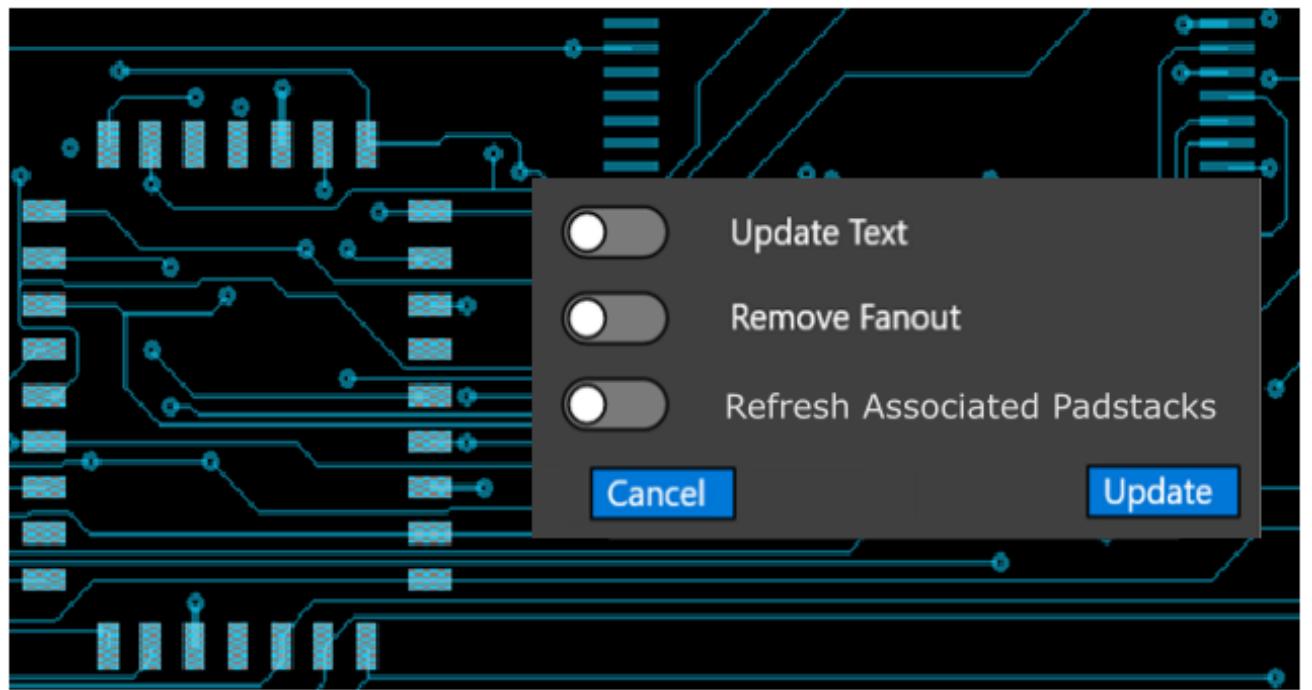




⚠ If you have not defined the footprint, you see the following dialog box:



2. Modify as necessary, and click Save to exit the Footprint Editor.
3. Set the on/off sliders as necessary in the following form and Update to save.



Related Topics

- [Adding Pins to a Footprint in OrCAD X Presto](#)
- [Editing Pin Locations](#)

Adding Pins to a Footprint in OrCAD X Presto

In OrCAD X Presto, you can add a pin to a PCB symbol using one of the following three methods:

1. Add a pin by [searching for padstacks](#).
2. Build or add a pad using the [Padstack Wizard](#) (available in OrCAD X Professional only).
3. Build or add a pad using the [Padstack Editor](#).

Related Topics

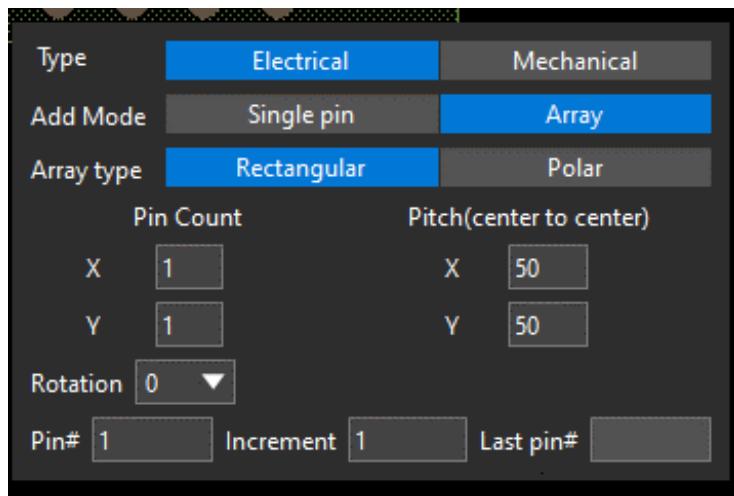
- [Using the Add Pin Widget](#)

Adding Pins by Searching Padstacks

When you choose *Place pin by search*, a search panel opens at the bottom of the UI that shows a list of all pads in the \$padpath and current working directory.

Single-click on a pad from the list in the search panel to show the pad information in the Properties panel.

Double-click on a pad from the list to choose the padstack, which is attached to the cursor. The *Add pin* widget opens, as follows, and the command add pin is active for you to place your pin on the canvas.

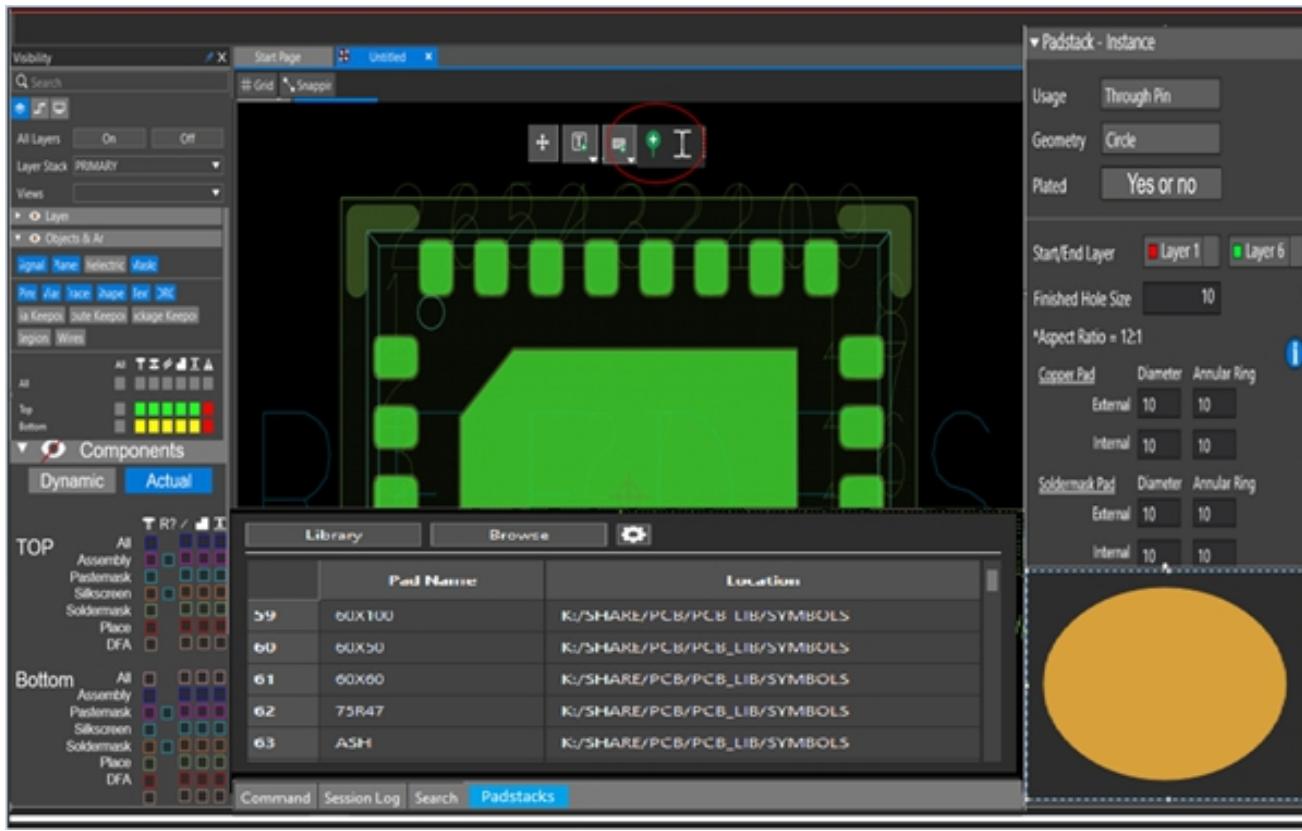


In the search widget, there is a library table and *Browse* button.

- Library shows the list of pads in their library and the current working directory.
- Browse opens a file browser and enables you to navigate and choose a .PAD from any location. When you browse to choose a PAD, the .PAD file is copied into your working directory and the selected pad is attached to your cursor for placement.
- The *Settings* button opens the user preferences dialog for you to modify the padpath.

When choosing a thru hole padstack, the property panel shows the attributes of the thru hole pad and a TOP view of the padstack.

When you choose a Surface mount pin, the property panel appears similar to the following:

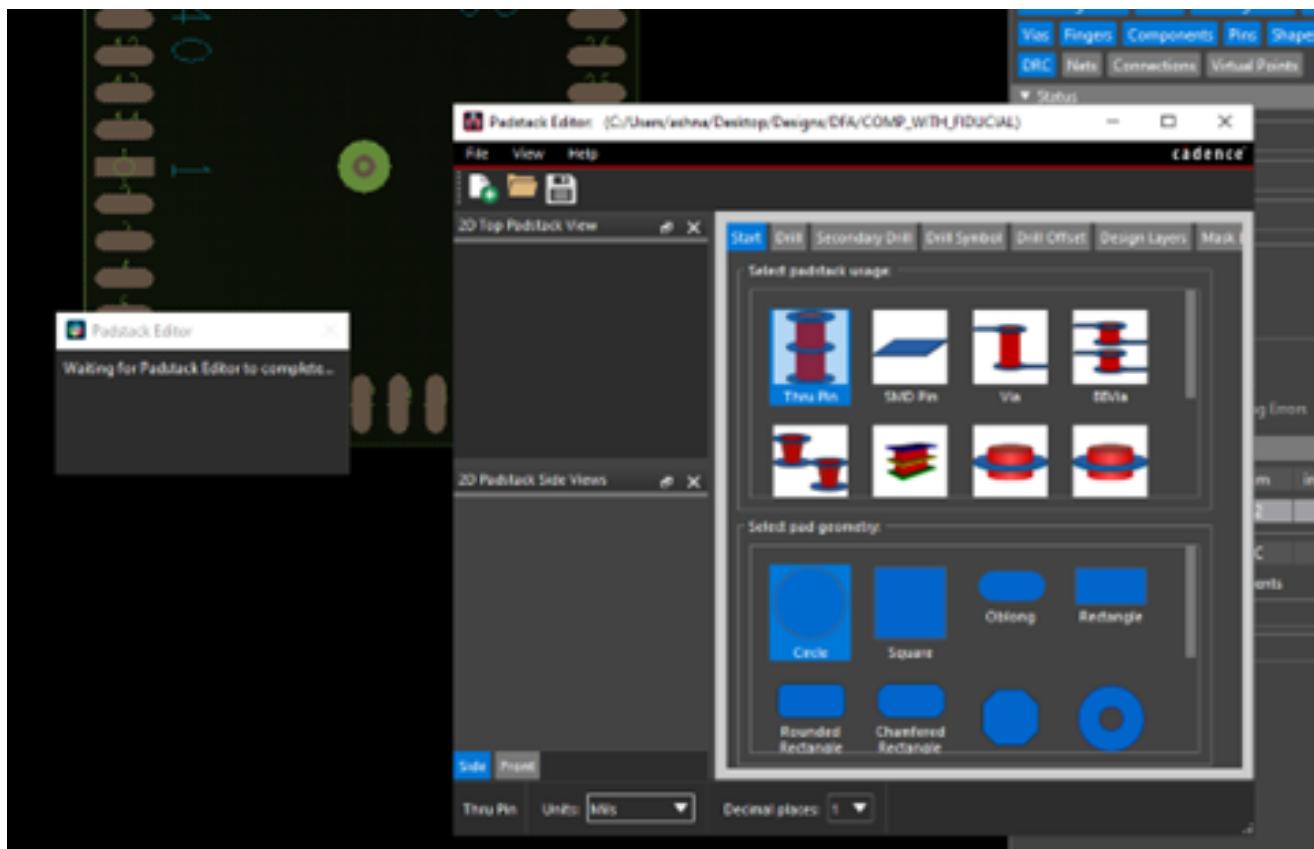


Related Topics

- [Adding Pins with the Padstack Wizard](#)
- [Using the Add Pin Widget](#)

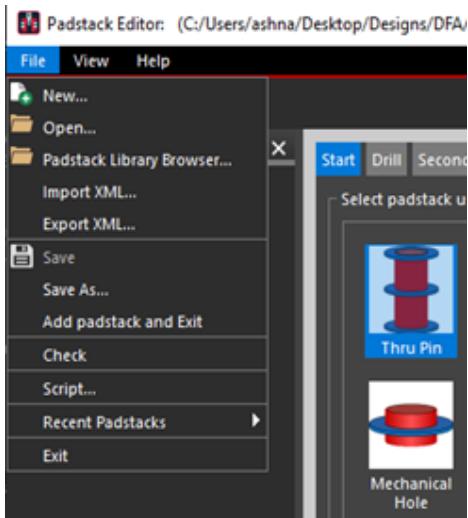
Adding Pins with the Padstack Editor

To launch the padstack editor, choose the third option from the floating *Add Pin* toolbar. You can create a new padstack or open an existing padstack and make the modifications.



The menu option *Add Padstack and Exit* is available only when launched from the Footprint editor. This option prompts you for the padstack path and then exits.

On exit, the padstack editor closes and the *Add pin* command is available. Use the add pin widget shown below to modify pin details and place the pad on the canvas.



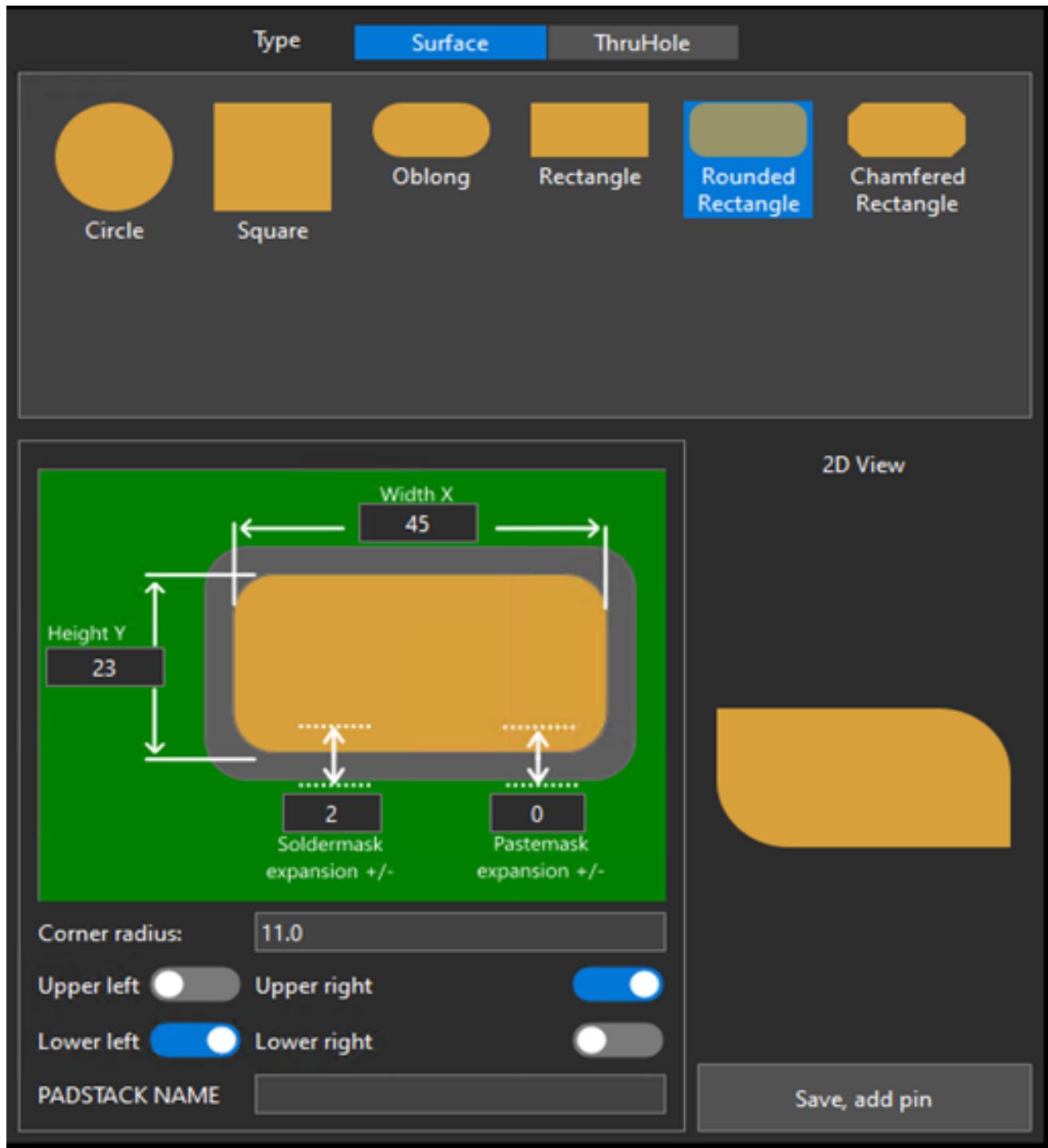
Right-click on an existing padstack for the *Modify padstack* to open the padstack editor.

Related Topics

- [Adding Pins by Searching Padstacks](#)
- [Adding Pins with the Padstack Wizard](#)
- [Using the Add Pin Widget](#)

Adding Pins with the Padstack Wizard

The wizard utility can guide you through required inputs to add or create a new padstack, as follows:



You can enter padstack parameters like width, height, diameter and other relevant details, based on your choice of six available pad shapes, as seen above.

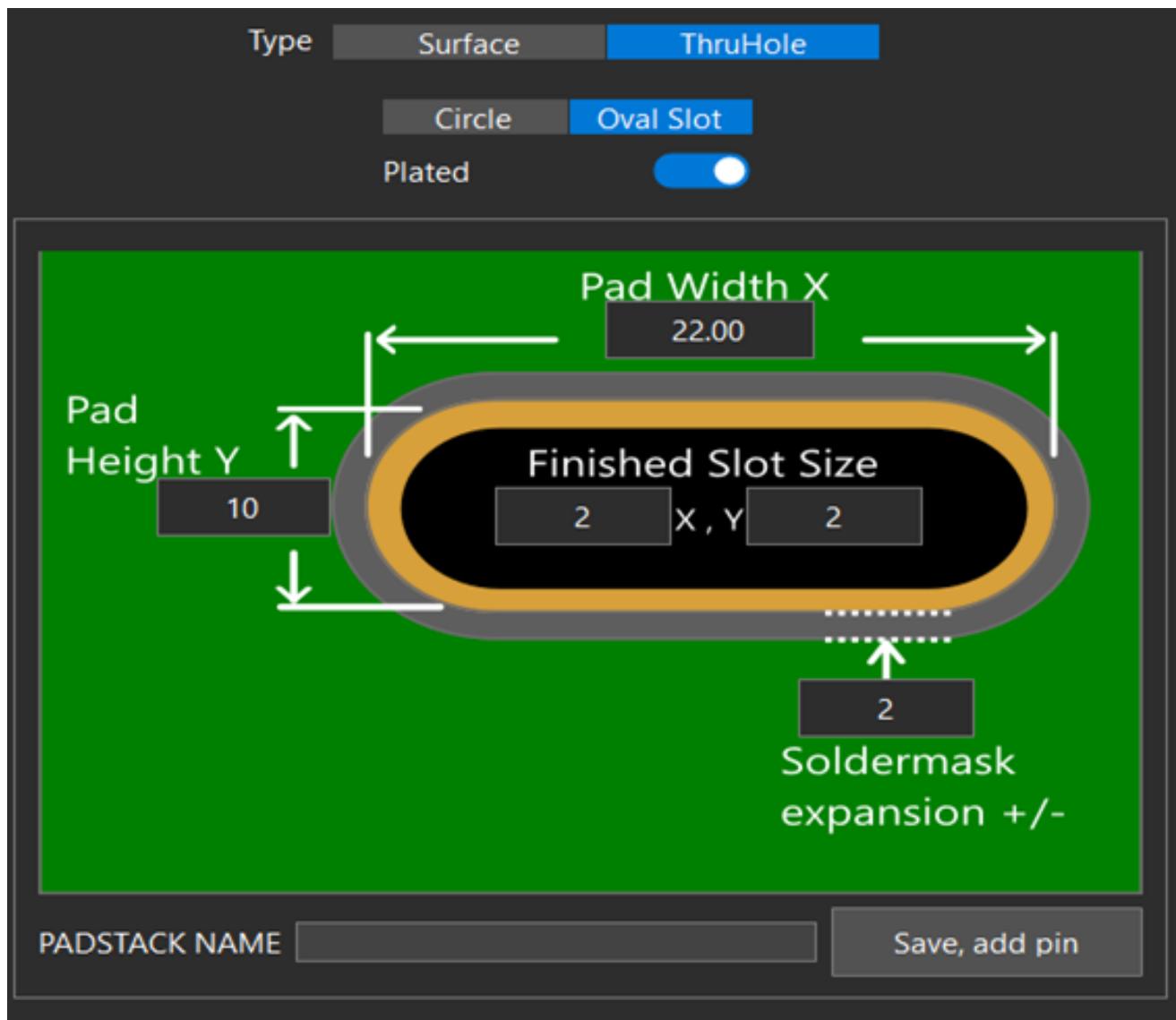
Soldermask and pastemask expansion creates a padstack on the soldermask top and pastemask top. The shape of the padstack created remains the same as the shape selected in the UI, and the dimensions depend on the soldermask/pastemask expansion specified in the UI. For example, if the width of the design padstack specified is 45, then the width of the padstack created on soldermask top will be $45 +/\text{-}$ soldermask expansion. A similar calculation is applicable for pastemask.

The Wizard shows a 2D view of the padstack, which is dynamically created based on the shapes and dimensions you provide.

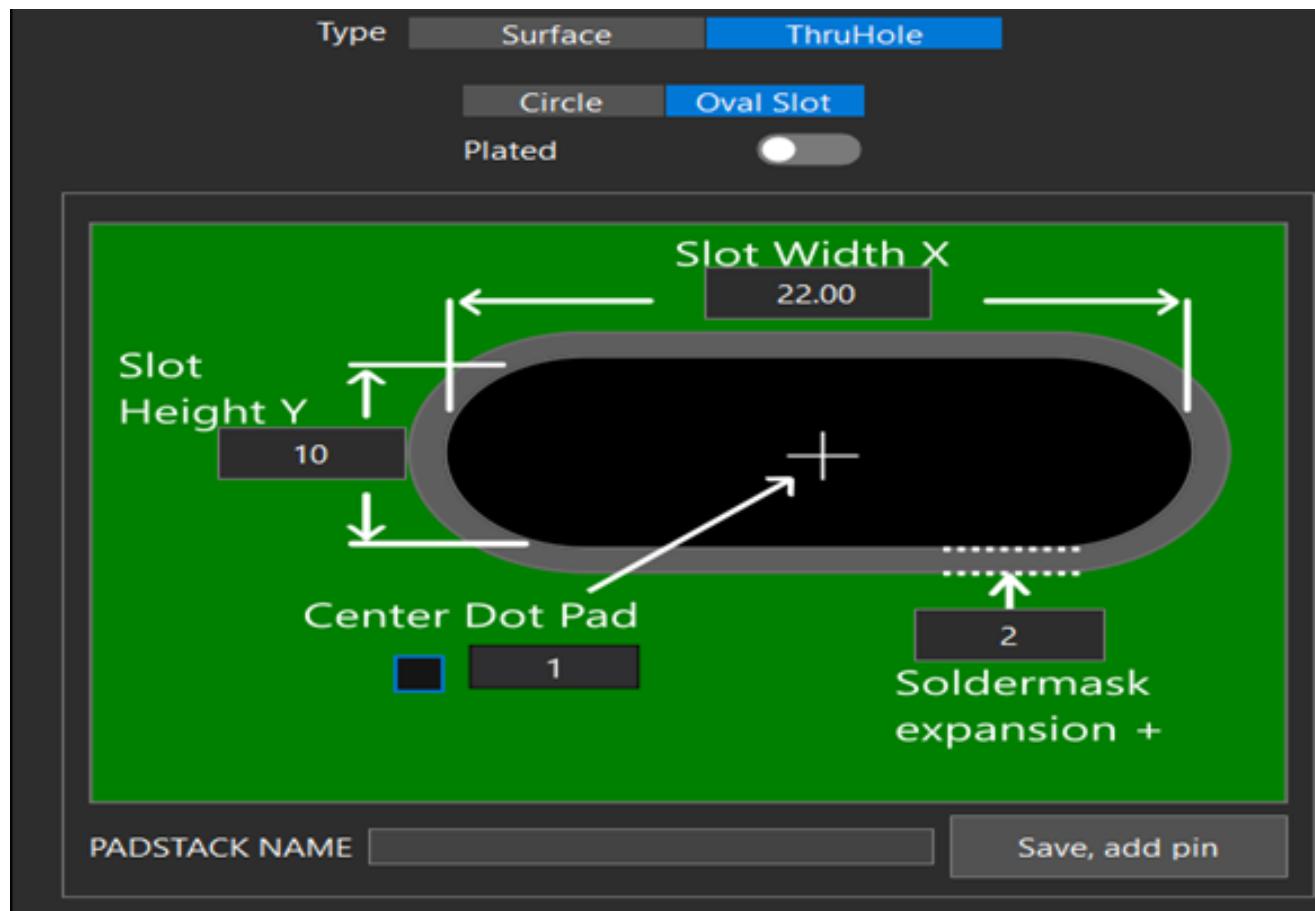
When you click the Save button, the padstack is validated for any errors. Any errors are displayed in the session log. Once all errors are cleared, the padstack—with name `test.pad`—is created on the disk (in the first path picked from the `$padpath` user setting). If no path exists, it skips to the next path mentioned in the `$padpath`. If there are no errors, the add pin command is active for entering pin details.

For the case of thru holes, there will be two tabs, Circle and Oval slot, as follows.

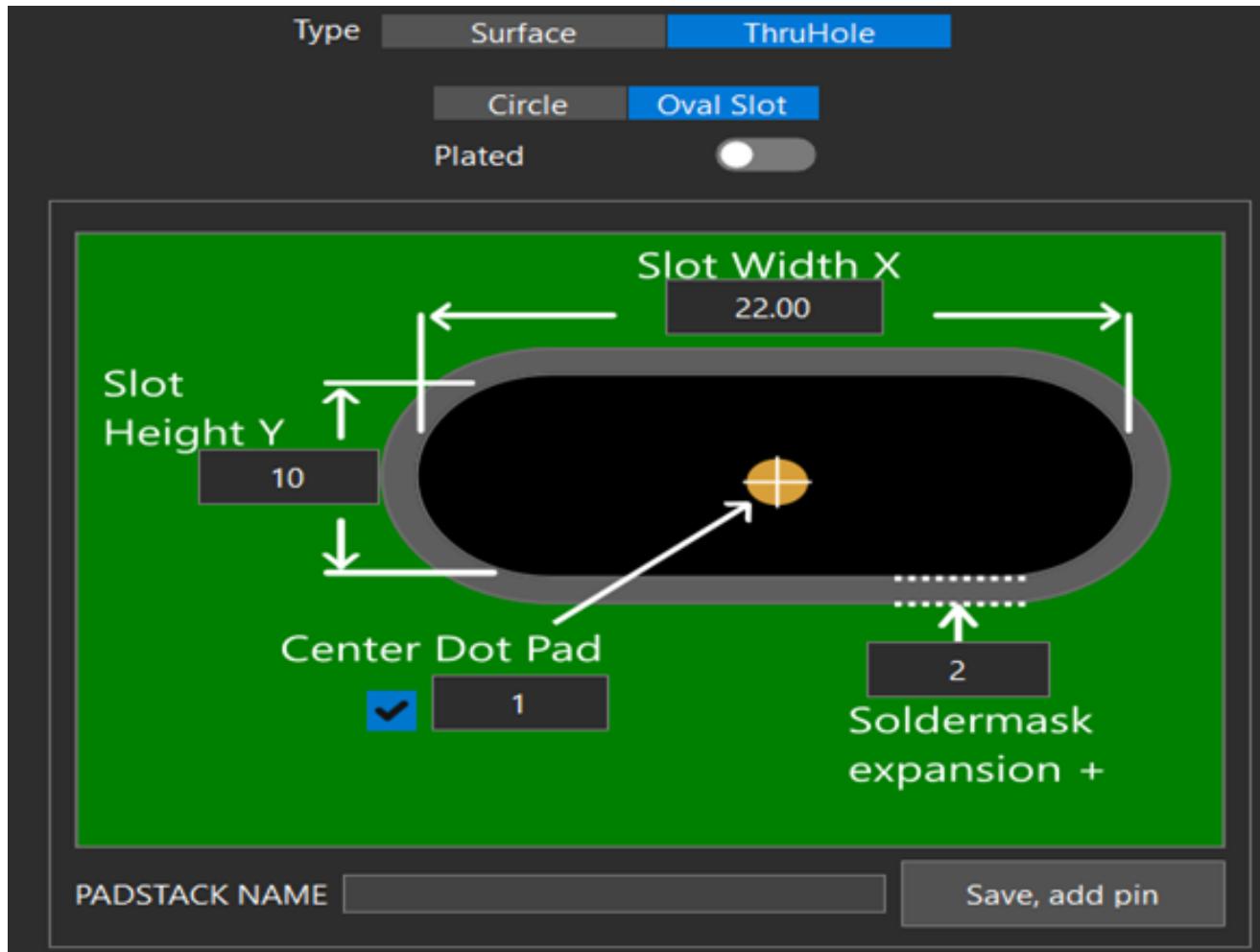
Plated, Oval Slot



Non-Plated, Oval Slot, Center Off



Non-Plated, Oval Slot, Center On

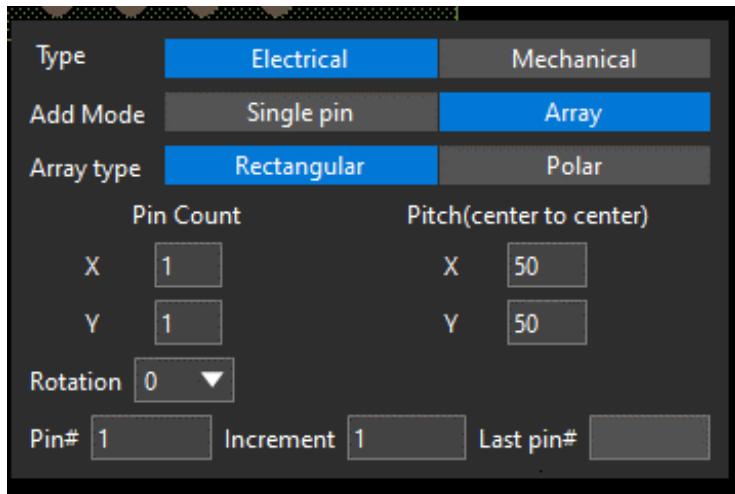


Related Topics

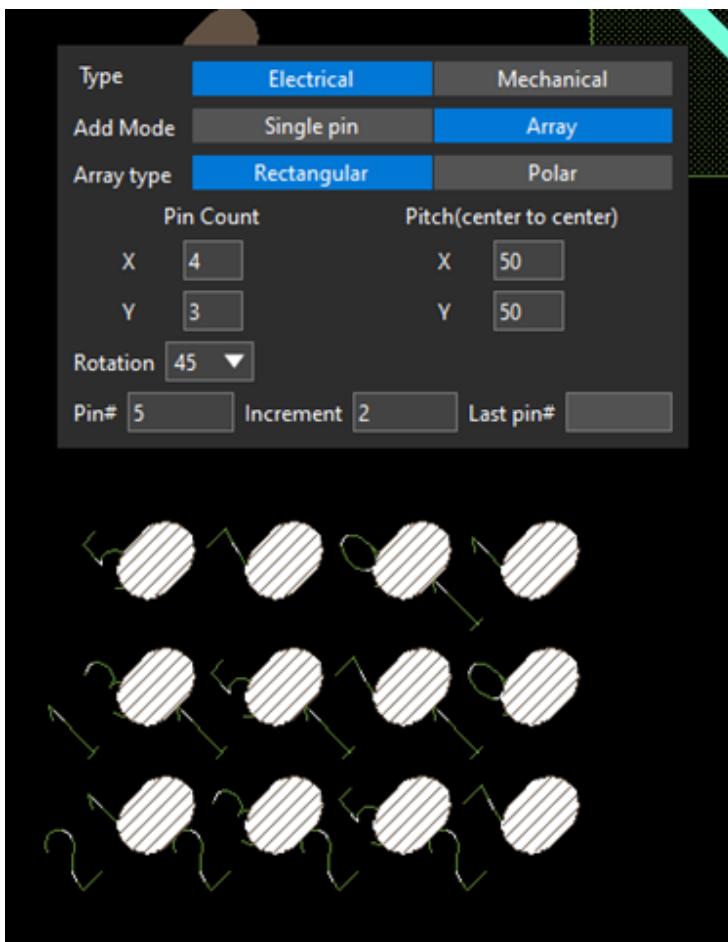
- [Adding Pins by Searching Padstacks](#)
- [Adding Pins with the Padstack Editor](#)
- [Using the Add Pin Widget](#)

Using the Add Pin Widget

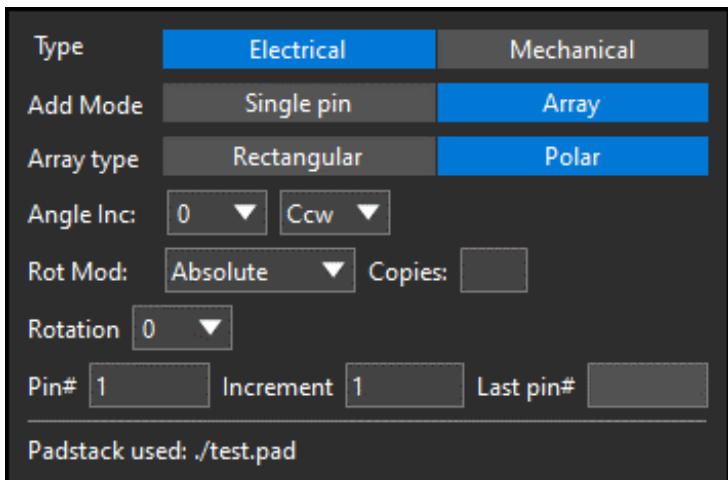
Once you create a padstack, using any of the three methods ([by searching padstacks](#), [with the Padstack Wizard](#), or [with the padstack editor](#)), the add pin command becomes active and you can specify pin parameters, as follows:



You can enter the details above, after which the pad will get attached to the cursor, and you can place the pin on the canvas. For example, if you enter the following details, then based on the inputs provided below, the pin placement is as follows:



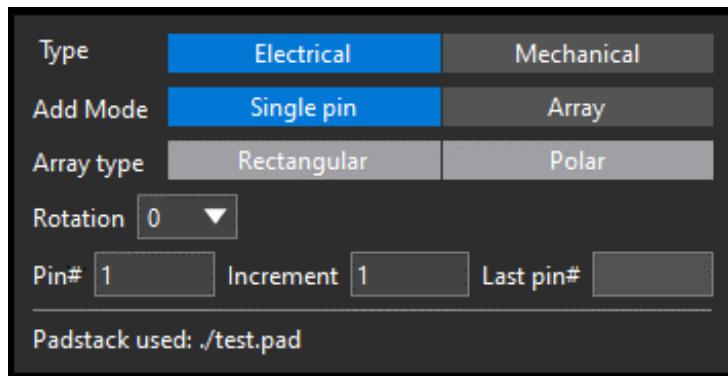
If the array type is *Polar*, you see the following available parameters:



If your chosen add mode is *Single pin*, the *Rectangular* and *Polar* array types are unavailable, as shown below:

OrCAD X Presto User Guide

Authoring and Editing Footprints in OrCAD X Presto--Adding Pins to a Footprint in OrCAD X Presto



OrCAD X Presto User Guide

Authoring and Editing Footprints in OrCAD X Presto--Adding Pins to a Footprint in OrCAD X Presto

Setting Up the OrCAD X Presto Design Environment

When setting up the OrCAD X Presto layout design environment, specify grid size, adjust the color display and visibility of layers and objects required.

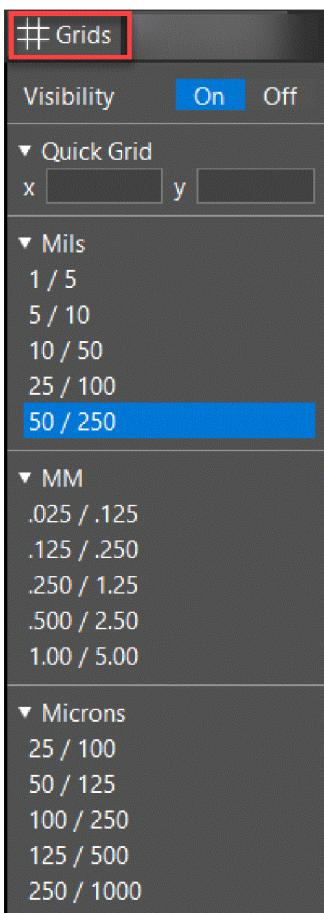
- [Setting Up Grids](#)
- [Setting Up Backup Preferences](#)
- [Controlling Visibility of Design Objects](#)
- [Controlling Visibility of Individual Objects](#)
- [Customizing Colors of Design Objects](#)
- [Controlling Visibility in 3D](#)
- [Customizing Keyboard Shortcuts in OrCAD X Presto](#)

Setting Up Grids

Grids help control component placement by establishing a regular spacing pattern between components in horizontal and vertical directions. Routing operations also required grids to place connections at the grid points.

To set up grids in OrCAD X Presto, do the following:

1. Click the  icon from the top-left corner of the design canvas to view the *Grids* widget.



The *Grids* widget provides options for selecting the *x* and *y* values for conductor and non-conductor grids in a design. You can also customize the grid by entering the grid values. The widget displays grid values in **Mils**, **MM**, and **Microns**. The default increment setting for non-conductor layers is *x*=100, *y*=100. For conductor layers, it is *x*=25, *y*=25.

2. Click the *On* button in the *Visibility* section to enable grids.
3. Select any predefined grid value.
The modified grid values are applied globally and used for subsequent routing or placement operations. However, this change does not affect placed components and existing connections and vias.
4. Alternatively, specify the *x* and *y* grid points in the *Quick Grid* section. You can choose the non-uniform spacing between lines by entering different *x* and *y* values.

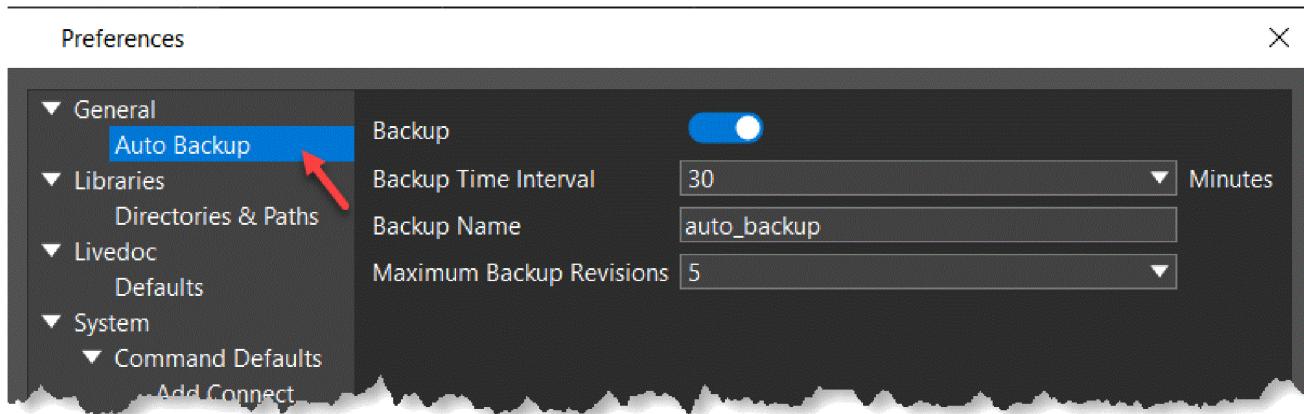
-  You can either set a global grid for all components or separate grids for different types of components. Similarly, you can use a separate grid for each conductor layer in a design.

Setting Up Backup Preferences

OrCAD X Presto keeps design files safe by saving them automatically at regular intervals. The auto backup option is enabled by default and saves multiple revisions periodically in the working design directory.

To modify auto backup preferences, do the following:

1. Choose *Edit – Preferences*.
The *Preferences* dialog box opens.
2. Select the *Auto Backup* page in the *Preferences* dialog box under the *General* category.



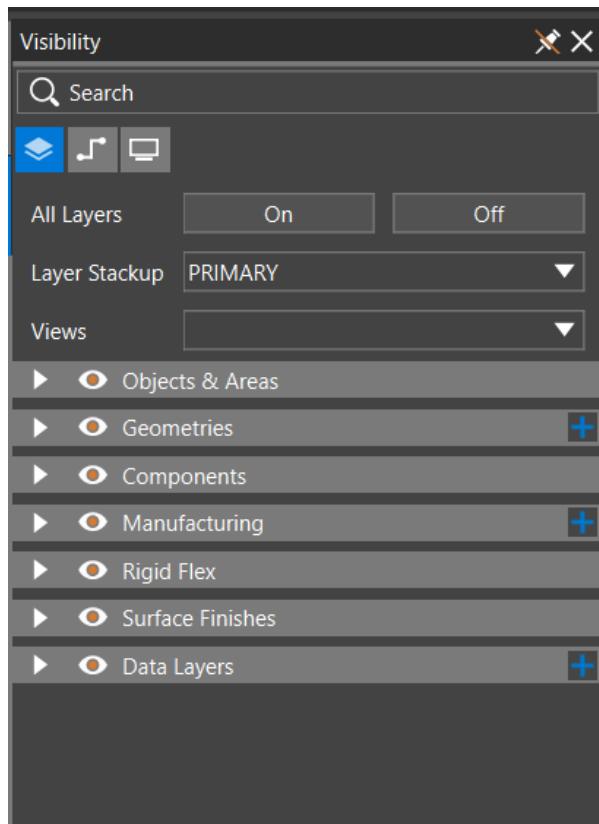
3. Specify the *Backup Time Interval* between 5 minutes to 360 minutes or choose a value from the list.
The auto backup is performed once the specified time is reached.
4. Modify the *Backup Name* or use the default name.
5. Specify the *Maximum Backup Revisions* between 1 and 10 or choose a value from the list.
This value determines the number of backups stored.
If the backup revision is set to 3, three versions *auto_backup.brd*, *auto_backup.brd,1*, and *auto_backup.brd,2* are available in the working directory at a time. Other versions are overwritten.
6. Click *OK* to apply the changes.
The configured settings are saved in the home directory.

Controlling Visibility of Design Objects

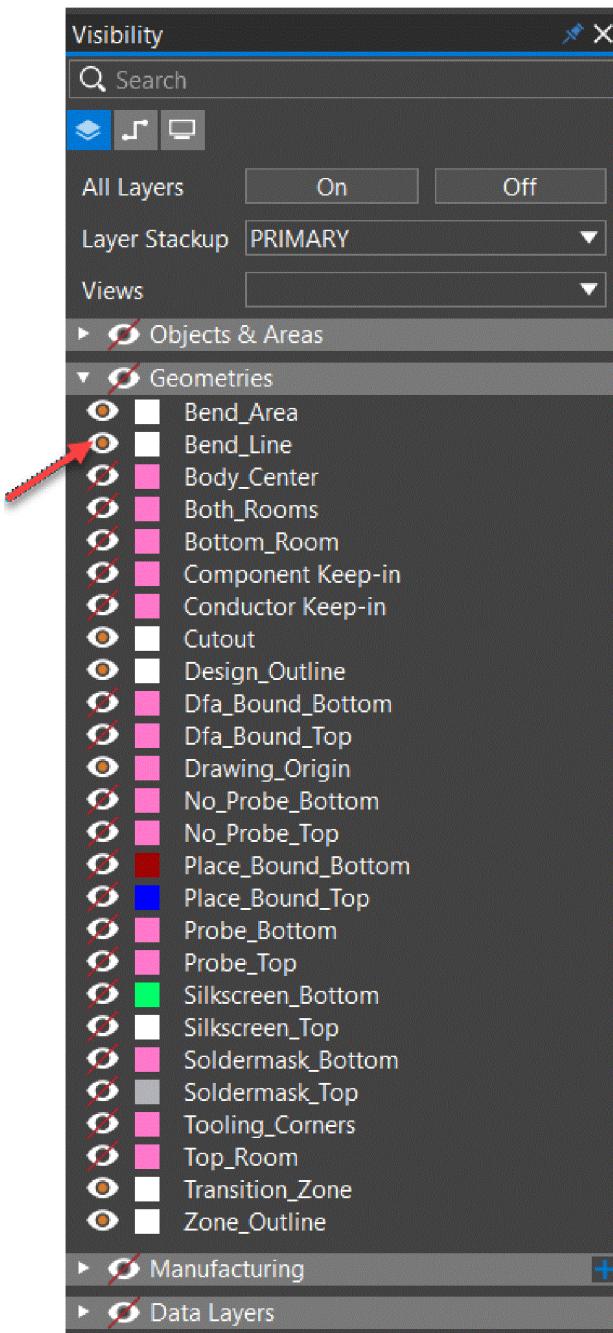
The visibility of all design objects and layers can be modified in the *Visibility* panel to customize the design environment that suits your viewing requirements both in 2D and 3D. You can modify the visibility of individual layers, objects, and geometries, opacity and transparency, display labels for nets, pins, vias, color for highlighting, drills, and connections of the complete design.

To modify the visibility of design objects, do the following:

1. Open the *Layers* tab in the *Visibility* panel. This tab controls the displays of all the layer types, objects, and components for the selected stack-up.

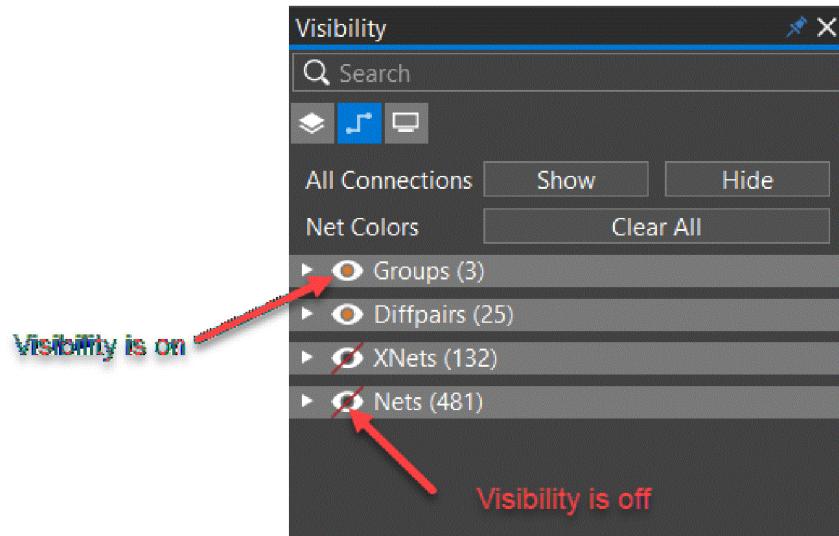


2. To toggle the visibility of objects of the same type, click the header-level visibility icon of the collapsible sub-sections, such as *Objects & Areas*, *Geometries*, *Manufacturing*, or *Data Layers*.
3. To toggle the visibility of individual objects, expand the group and click the visibility icon  associated with that object.



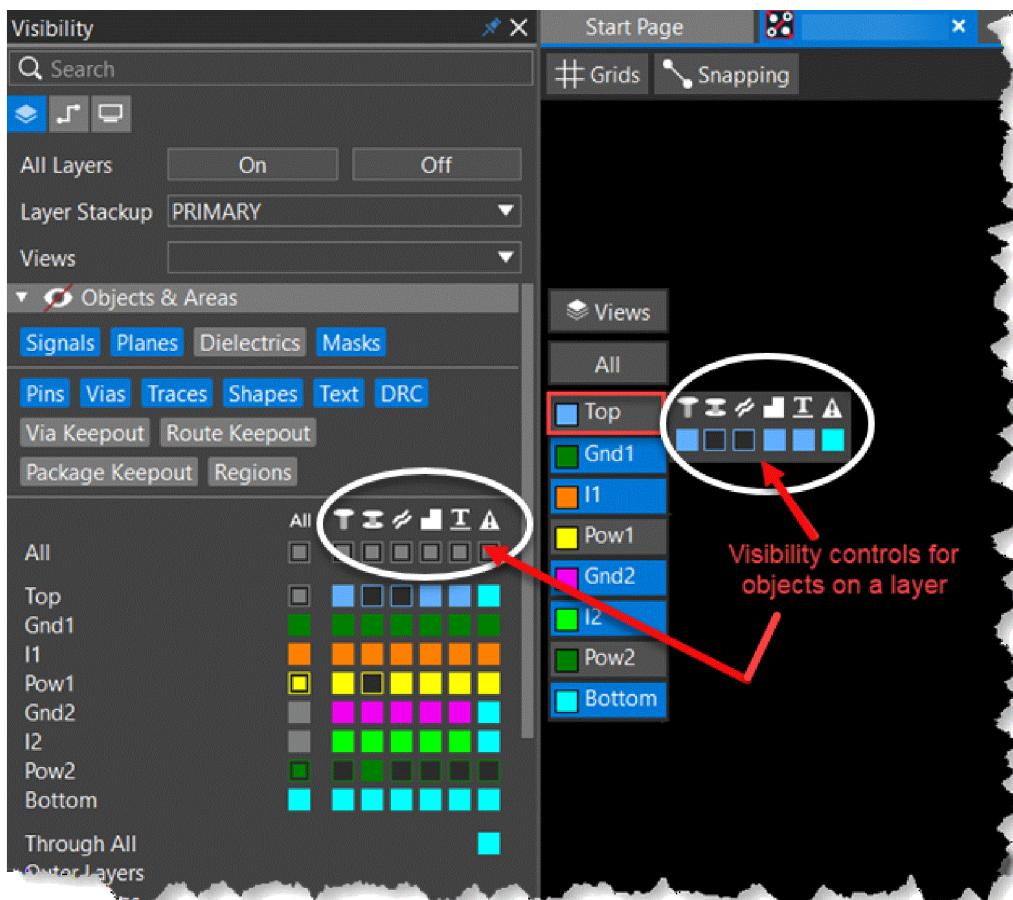
4. Similarly, open the *Nets* tab to modify the visibility of any group of nets or a single net by clicking the visibility icon.

The following image displays visibility state of different types of a net in a design:



Controlling Visibility of Individual Objects

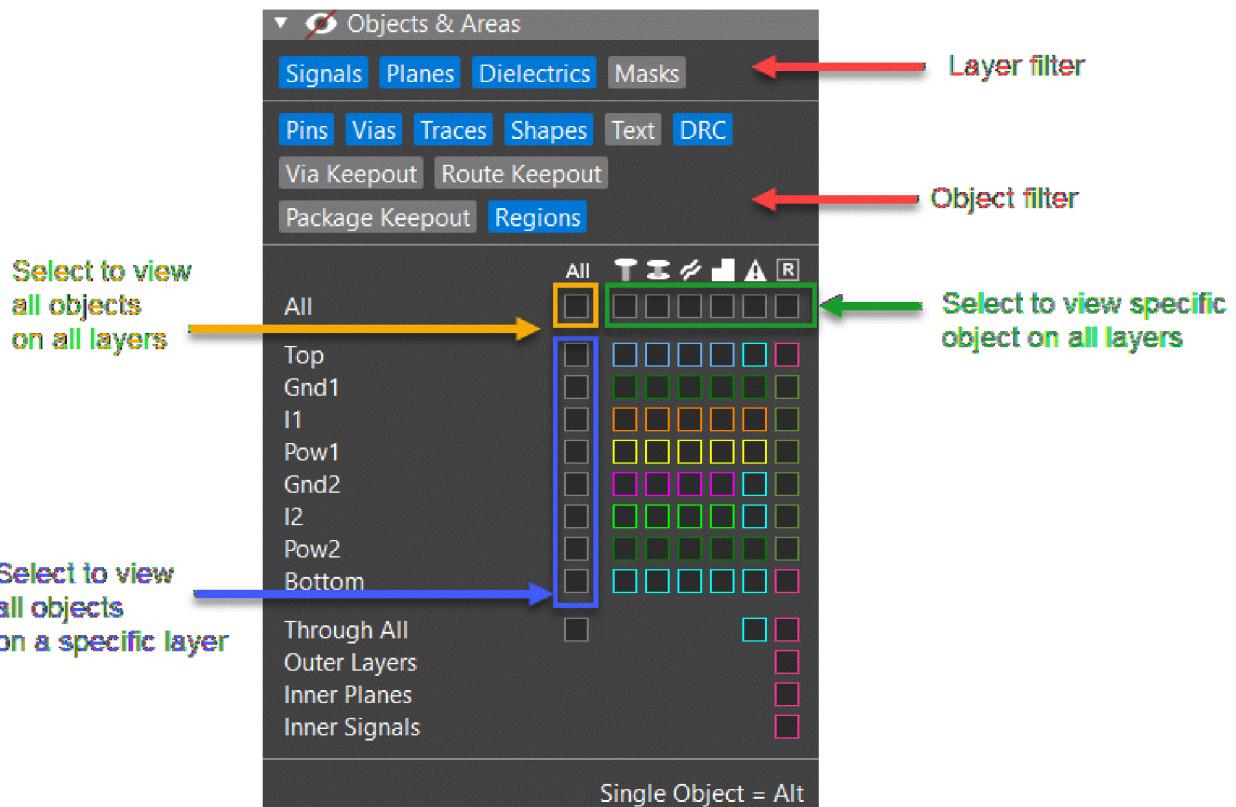
The *Visibility* panel and the floating layer toolbar provide controls to enable or disable the visibility of any object for any layer.



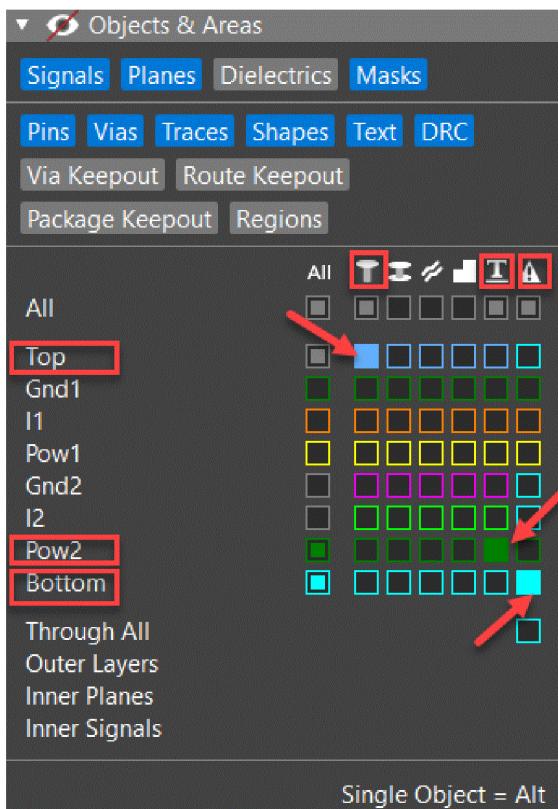
From Visibility Panel

To modify the visibility of any object on any layer, do the following:

1. Expand the *Objects & Areas* group.
This number of layers and objects displayed under this group depends on the layers and object types enabled in the filter section.
2. Click the colored toggle box to change the visibility of an object on a layer.



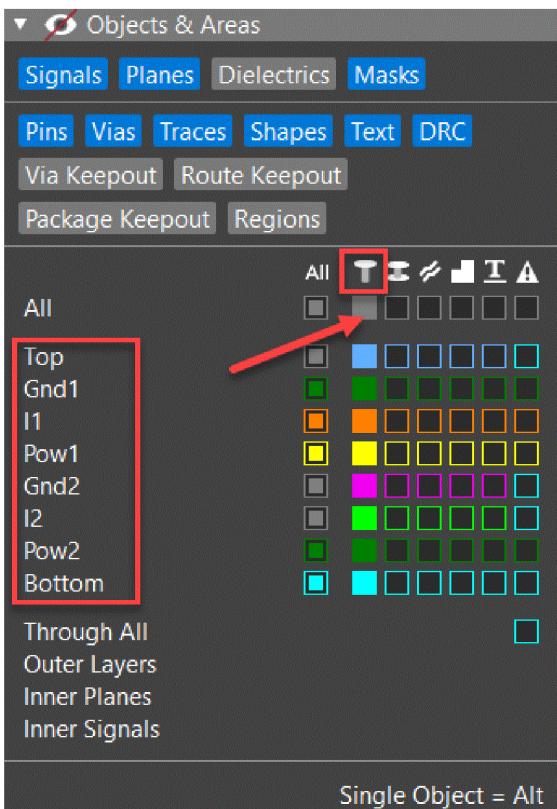
3. To toggle the visibility of an individual object on a single layer, click the hollow box with a colored outline under the object type column and the layer row in the grid view. For example, to turn on the visibility of *Pins* on the *TOP* layer, text on the *POW2* layer and DRCs on the *BOTTOM* layer, select the hollow color boxes as illustrated in the following snapshot.



Setting visibility of Pins on TOP layer

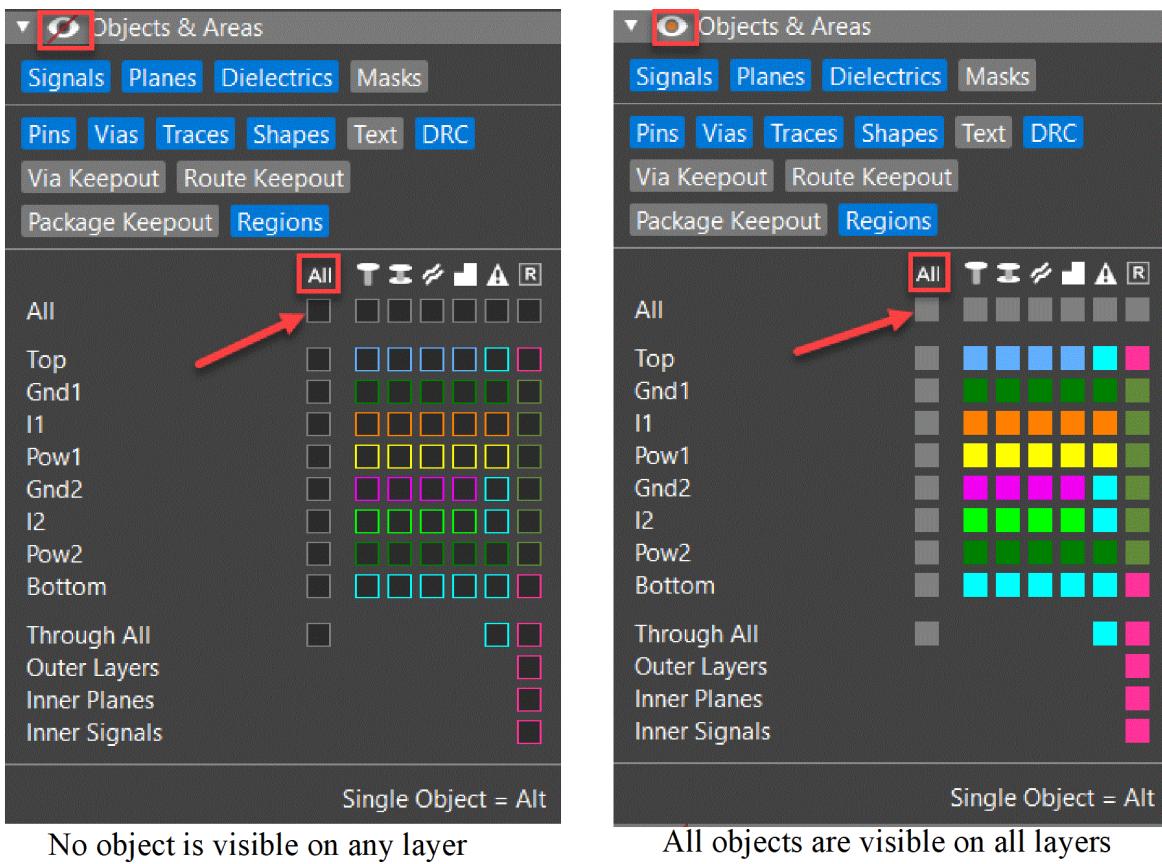
The solid color box indicates that the pins are visible on the **TOP** layer, the text is visible on the **POW2** layer and the DRCs are visible on the **BOTTOM** layer.

4. To toggle the visibility of an individual object on all the layers, click the hollow box with a gray outline under the object type icon.
For example, select the hollow box with a gray outline to enable the visibility of *Pins* on all layers, as illustrated in the following image:



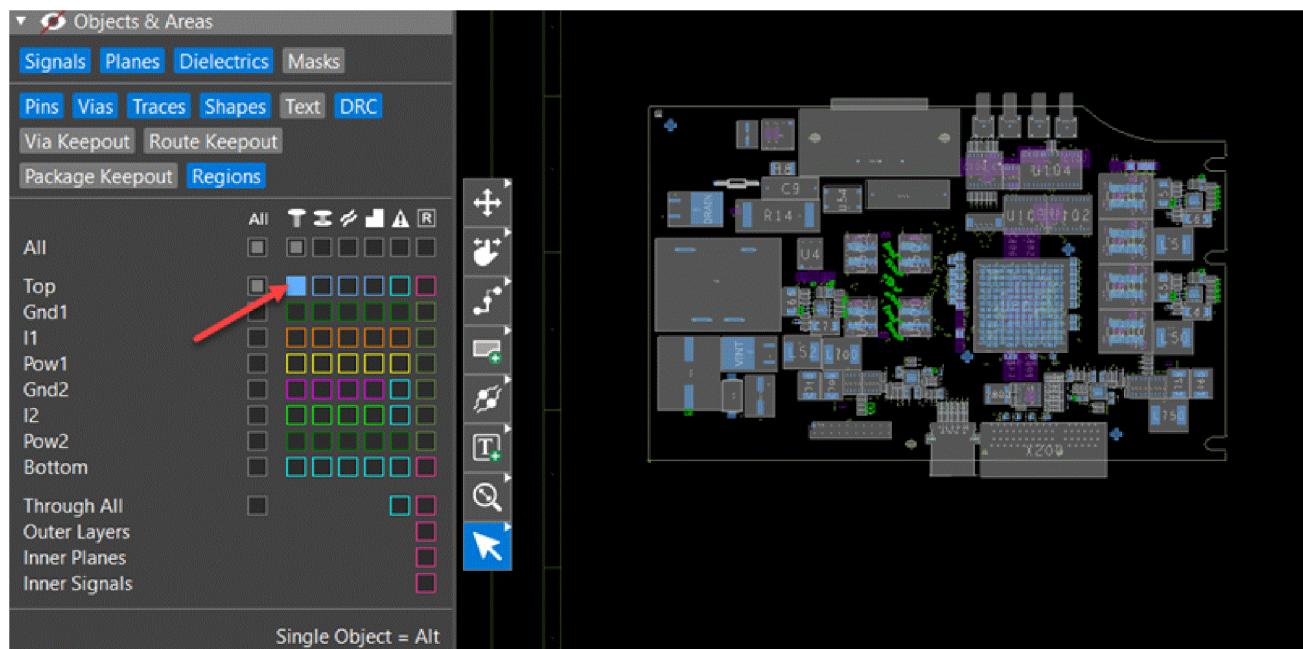
The solid gray box under the *Pins* object type icon indicates that pins are visible on all layers. Solid color boxes of different or the same color are displayed under the *Pins* object type column, indicating that the pins are visible for all layers.

5. To toggle the visibility of all the objects on all layers, click the hollow box with a gray outline under *All* in the grid view.



The header visibility icon toggles. The solid gray boxes under object type icons indicate that the object is visible on all layers. Similarly, the solid gray boxes in front of the layers show that all the objects are visible on that layer.

- To change the visibility of a single object at a time, press and hold the `Alt` key and click the color box in the grid view. In the following image, only pins on the `TOP` layer are displayed in the design. The visibility is turned off for the pins on the remaining layers. The rest of other objects are also not displayed at all.



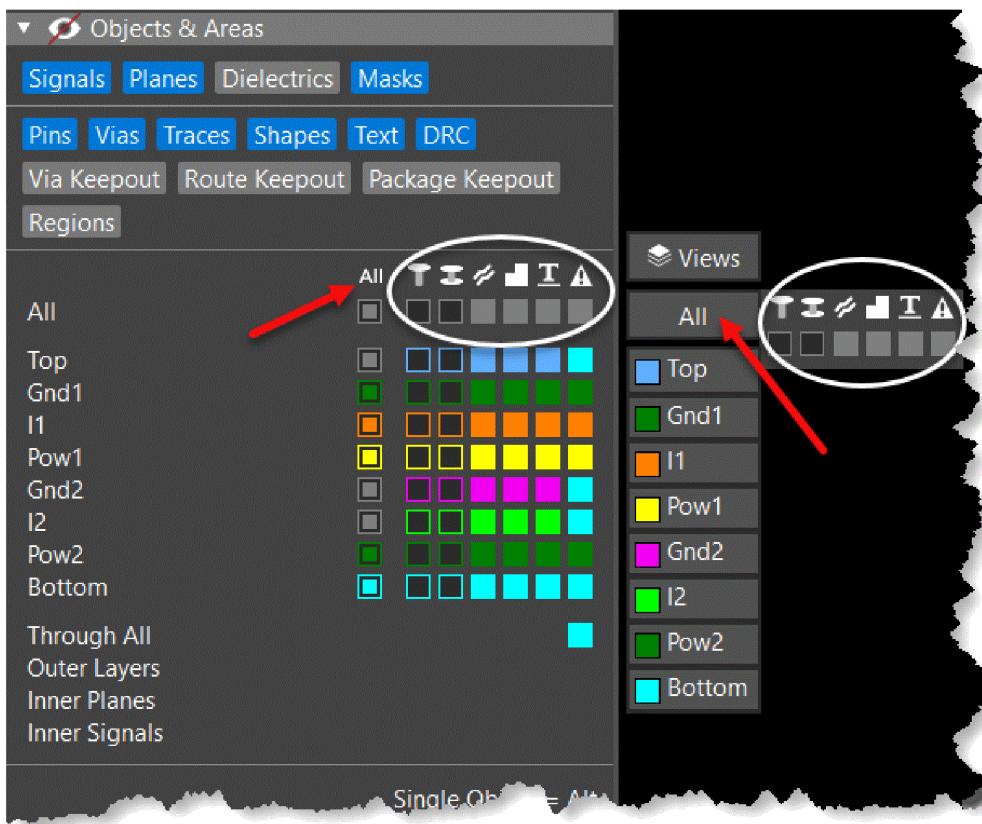
From Layer Toolbar

To modify the visibility of an object from the layer toolbar, do the following:

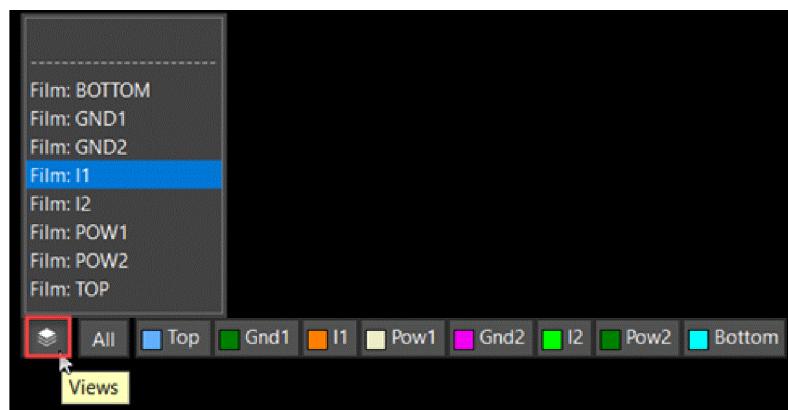
1. Right-click a layer to view the shortcut menu.
2. Click the colored toggle box to change the visibility state of individual objects on that layer.



3. Alternatively, click the *All* button to change the visibility of specific objects for each layer.



4. Optionally, click the *Views* icon to select an existing view.
A list of films generated from the artwork is displayed.

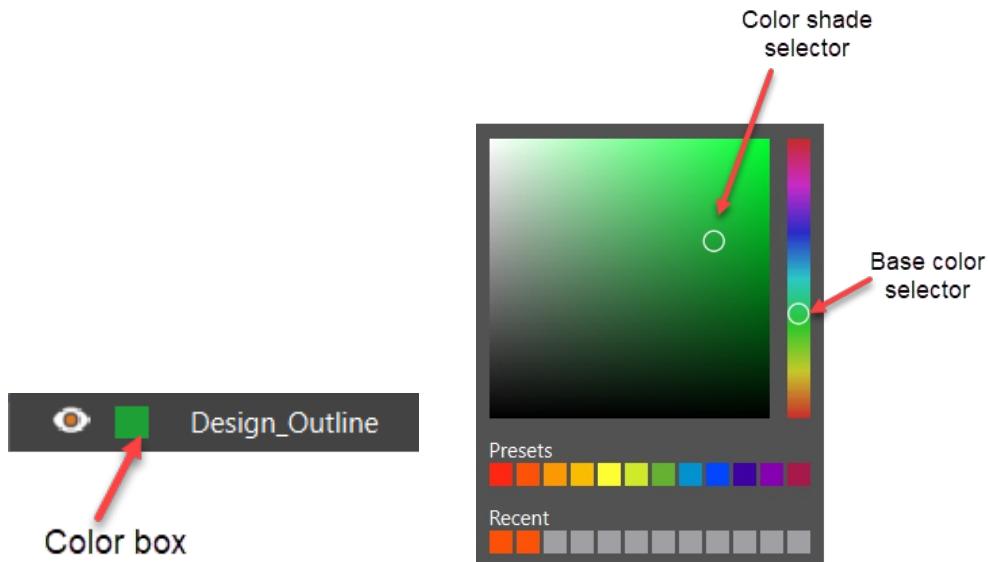


Depending on the selected view, the visibility of objects is dynamically updated for the objects on each layer.

Customizing Colors of Design Objects

The color settings for any design object, whether a layer, net, component, or design geometry, can be changed from the *Visibility* panel. Each object in the *Visibility* panel has a small color button that displays its default color. To modify the color of any object, do the following:

1. Click the color box to open the color selector dialog box.
2. Choose a different color either from the *Presets* section or use the slider of the vertical color bar to first select a base color and then move the cursor in the color palette to choose the brighter or darker shade.



The new color is displayed in the color box of that object and is also applied to it in the design canvas.

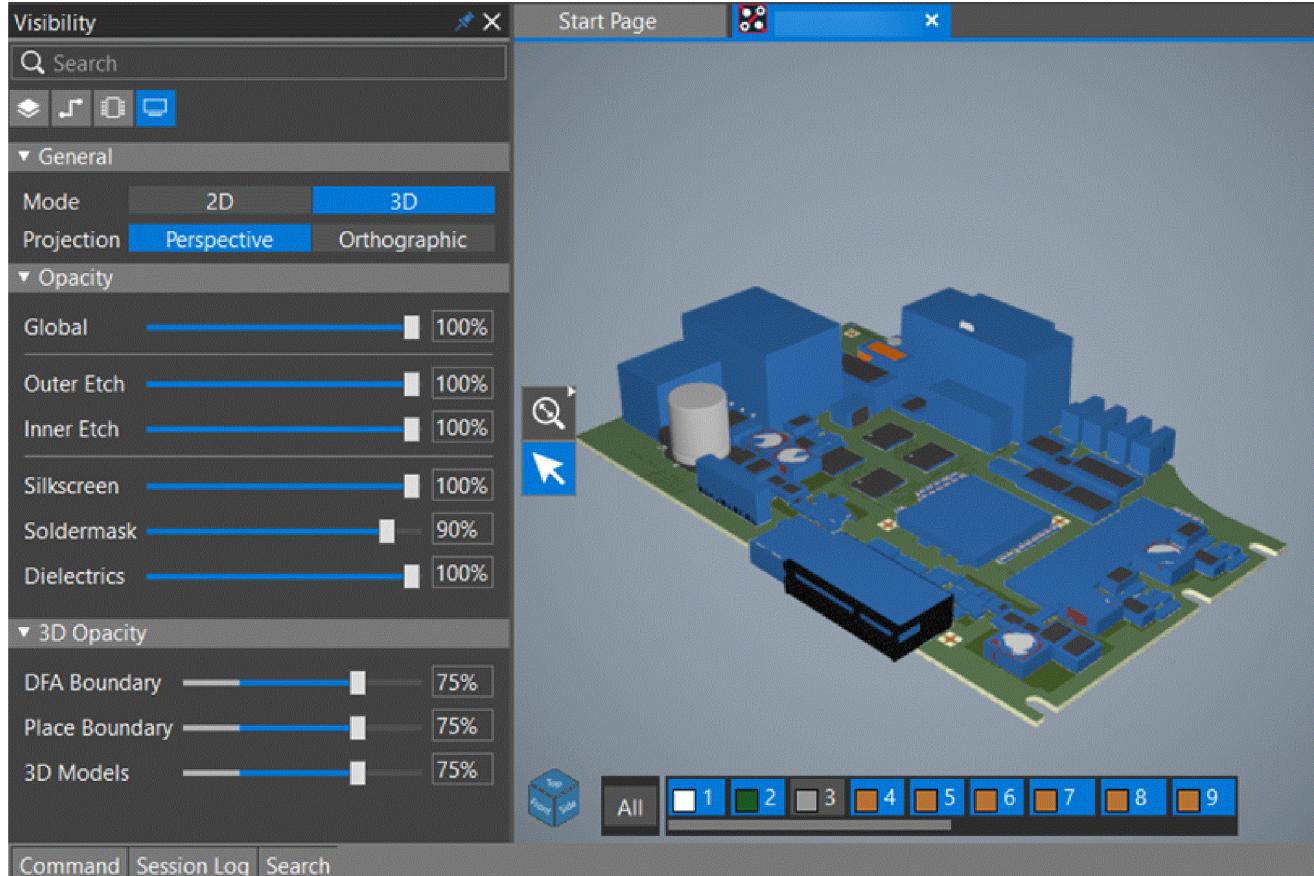
3. To apply

Controlling Visibility in 3D

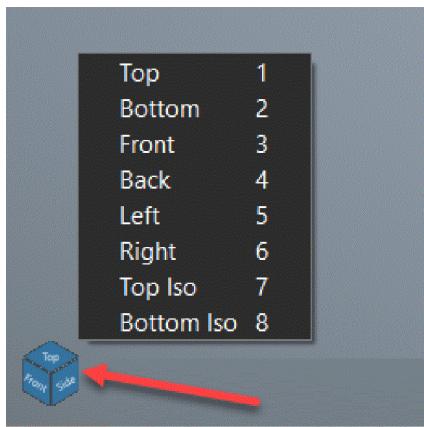
To set up visibility of objects and layers in 3D, do the following:

1. Open the *General* tab of the *Visibility* panel and choose *3D* mode.

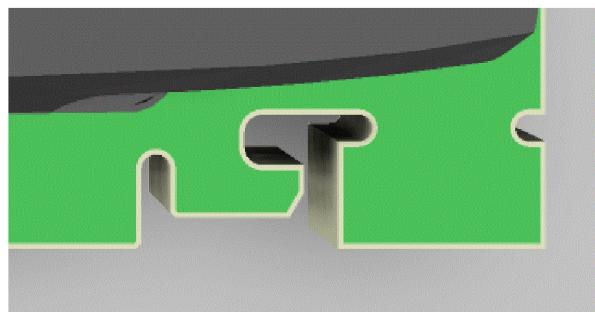
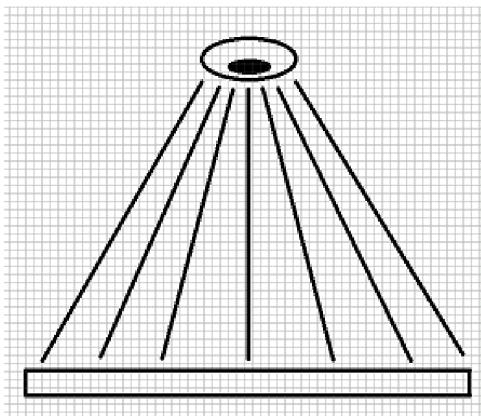
The *Visibility* panel displays settings for viewing a design in 3D and the canvas switches to 3D. Components placed on the TOP and BOTTOM layers along with mechanical components are displayed in 3D.



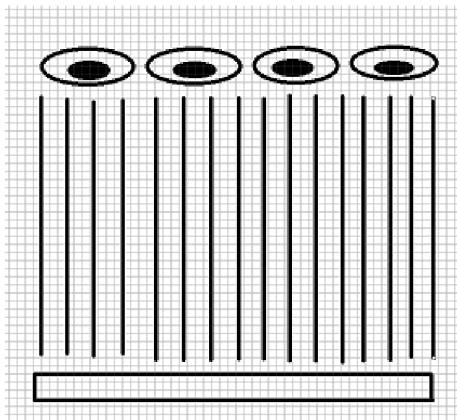
2. Use the Camera icon , placed at the bottom-right of the design canvas, to view 3D graphical data in different perspectives.



3. Select the *Perspective* projection type to view the design as if you are standing above the center of the assembled board. Objects that are far away appear smaller than those are near. This option is selected by default.



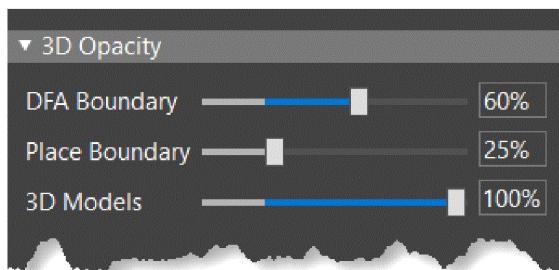
Alternatively, select the *Orthographic* projection type to view the design as if you are standing directly over each object of the board. In this view, all of the objects appear the same size.



4. In the layer toolbar, click any layer to turn on or off its visibility in the 3D design canvas.



5. Adjust the slider or specify a value to set the transparency of 3D objects in the *3D Opacity* section.

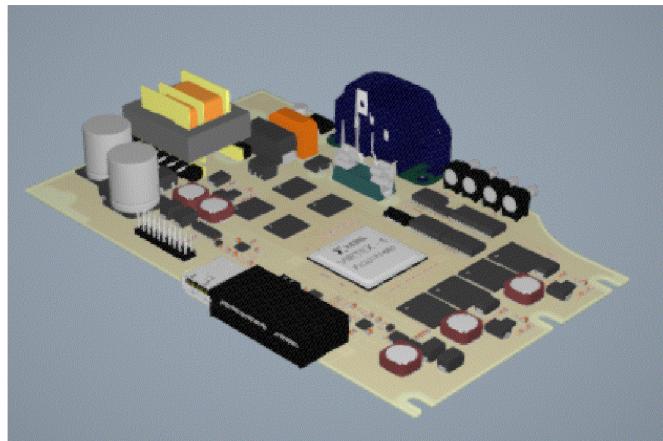


You can set the opacity value between transparent (25%) and opaque (100%).

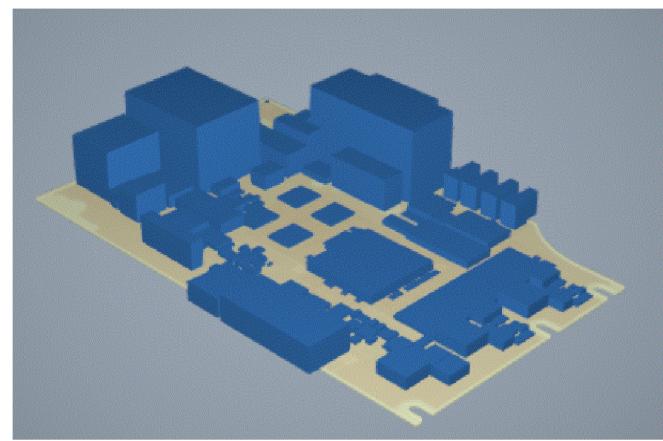
6. To choose components for displaying in 3D, do the following:

- a. Open the *Components* tab that is added to the *Visibility* panel in the 3D mode.
- b. Select *3D Bodies* as *Model*, *Place*, or *DFA*.

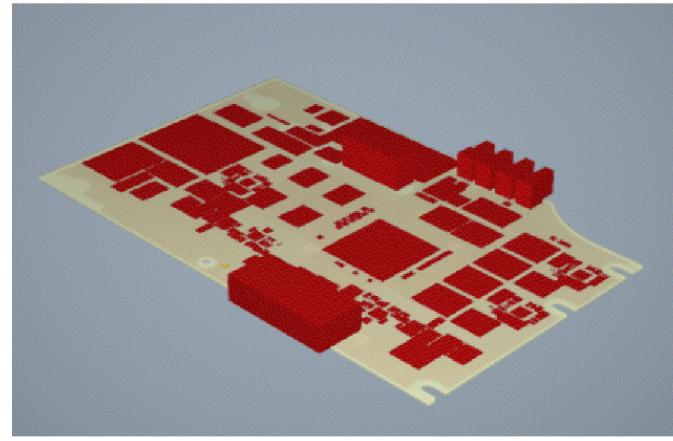
The following image illustrates how a design look when different types of 3D bodies are selected:



Displaying only STEP model

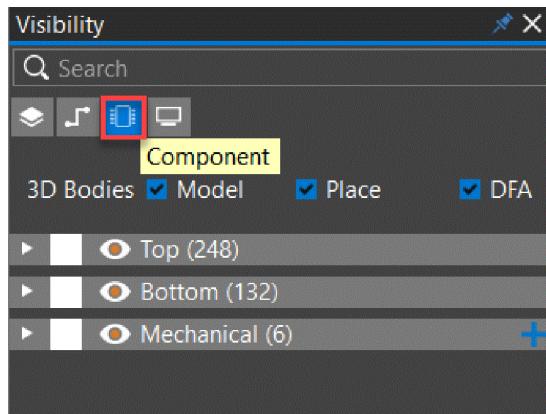


Displaying only place bound shape

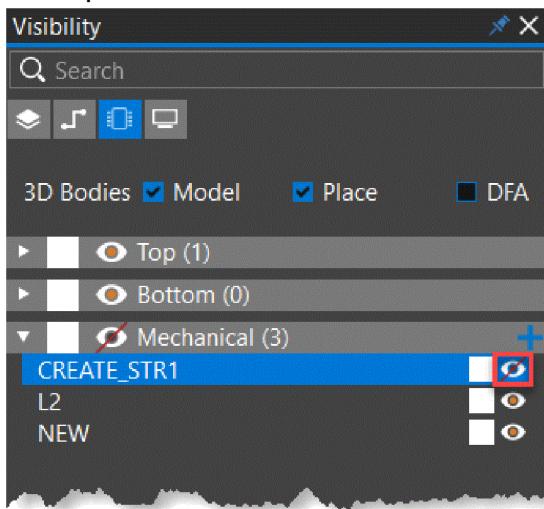


Displaying only DFA bound shape

- c. Expand the groups *Top*, *Bottom*, and *Mechanical* to view a list of components placed on these layers.



- d. Select a component from the list and click the visibility icon to toggle its visibility.

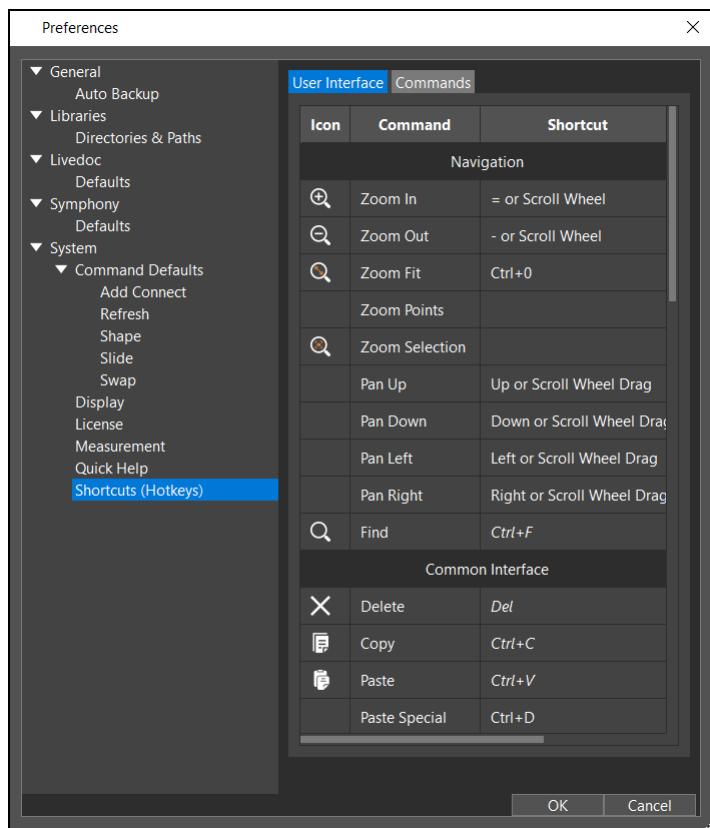


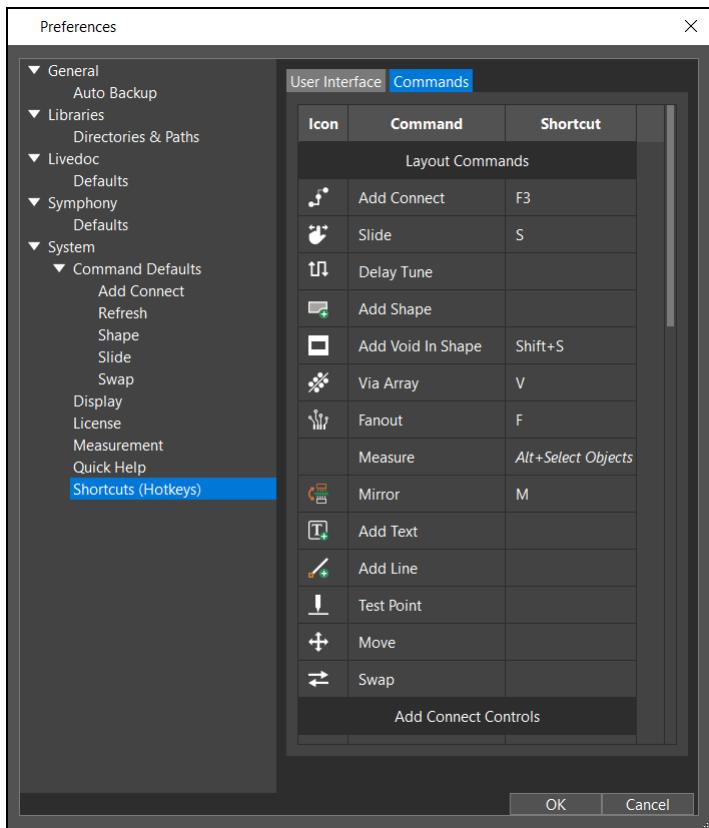
- e. Repeat the above step to modify the visibility of individual components.
Alternatively, use the header-level visibility icon to disable the visibility of all the components under that group.

Customizing Keyboard Shortcuts in OrCAD X Presto

Shortcut keys make navigation and execution of frequently-used commands easy. OrCAD X Presto provides quick access to interface-based commands and navigation controls using standard shortcut keys. You can view the currently assigned shortcut keys in the *Preferences* dialog box. Additionally, you can assign or remove keyboard shortcuts to any command or display control in OrCAD X Presto using the following steps:

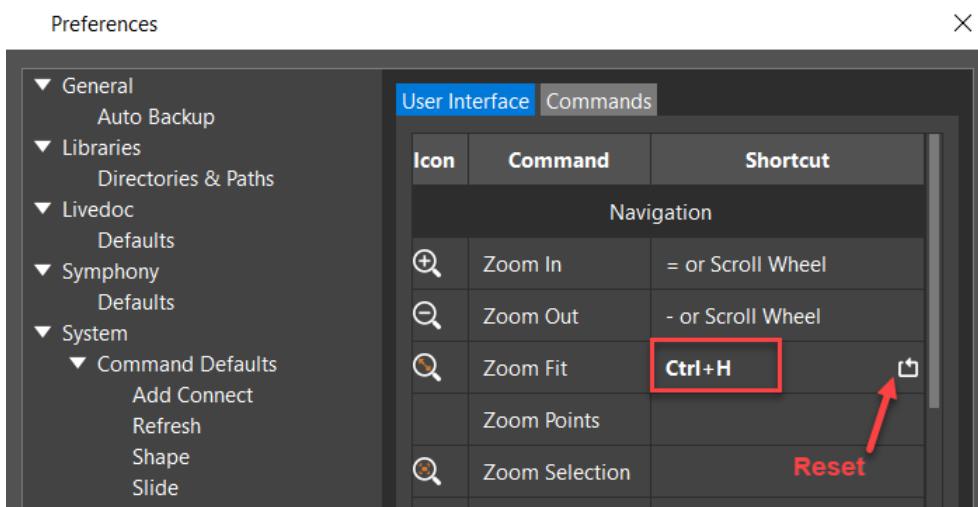
1. Choose *Edit – Preferences*.
The *Preferences* dialog box opens.
2. Open the *Shortcuts (Hotkeys)* page to view the default shortcut keys assignment.





A table displays the icon, name, and default shortcut key assignments.

1. Double-click the shortcut cell in the table to change or add a new shortcut key.
The selection is highlighted in blue.
2. Press the key or the combination of keys that you want to assign.
The new key is displayed in bold to indicate it as an overridden value. A *Reset* button also starts displaying.



1. Press the Reset button to restore the default shortcut key.
2. Double-click the shortcut key and press the Delete button to remove the shortcut key.
3. Click *OK* in the *Preferences* dialog box.

OrCAD X Presto User Guide

Setting Up the OrCAD X Presto Design Environment--Customizing Keyboard Shortcuts in OrCAD X
Presto

Editing Design Elements in OrCAD X Presto

- [Copying Objects](#)
- [Measuring the Distance between Objects in OrCAD X Presto](#)
- [Swapping Components, Diff Pairs, and Pins in OrCAD X Presto](#)
- [Creating Custom Design Outline in OrCAD X Presto](#)

Copying Objects

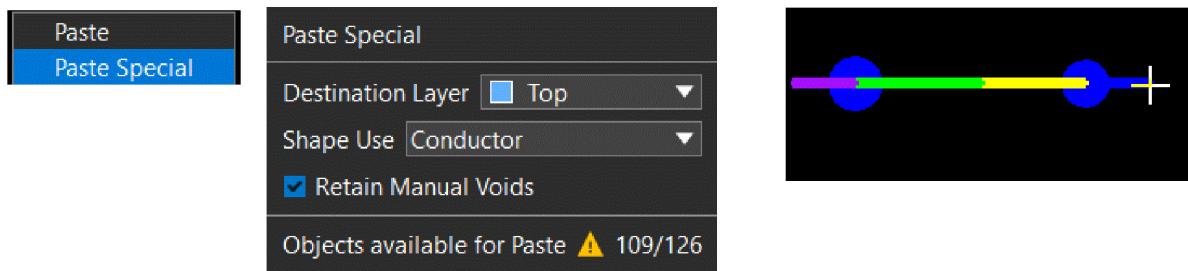
OrCAD X Presto provides the *Paste Special* command to copy design objects across the layers of the PCB design. Specifying a destination layer when copying traces and shapes speeds up the design time by leveraging the same shape and etch patterns on additional layers.

The Paste Special command also provides the capability to specify a different shape when pasting the copied shape object. It helps create constraint regions or keepout areas based on existing shape geometry. You can also exclude existing manual voids in the copied shapes during the paste operation if these voids are not needed on destination layers.

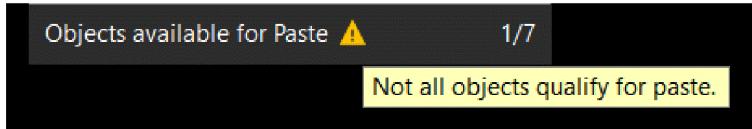
To paste objects using the paste special command:

1. Set the *Selection Filter* and copy objects in the design canvas.
2. Choose *Edit – Paste Special* or right-click and choose *Paste Special*.

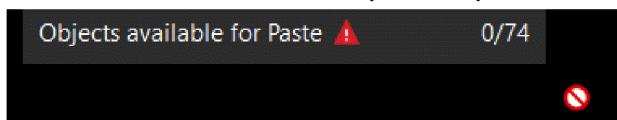
The Paste Special widget opens and displays relevant fields only when valid objects are copied and saved in the paste buffer. Valid objects include shapes (Etch, Keepout, Constraint Region, Keepin), traces, trace segments, and lines. A pre-paste view of the objects is attached to the cursor that reflect the current settings in the Paste Special widget.



The bottom part of the widget displays the status of the command that includes the number of objects available for paste, warning or error symbol, and a visual cue on the cursor. The status changes dynamically based on the objects copied and widget settings. Hovering the cursor over the warning or error symbol displays more information on the tool tip text. A warning is indicated by a yellow triangle and displayed when only one object qualifies for paste from the copy buffer.

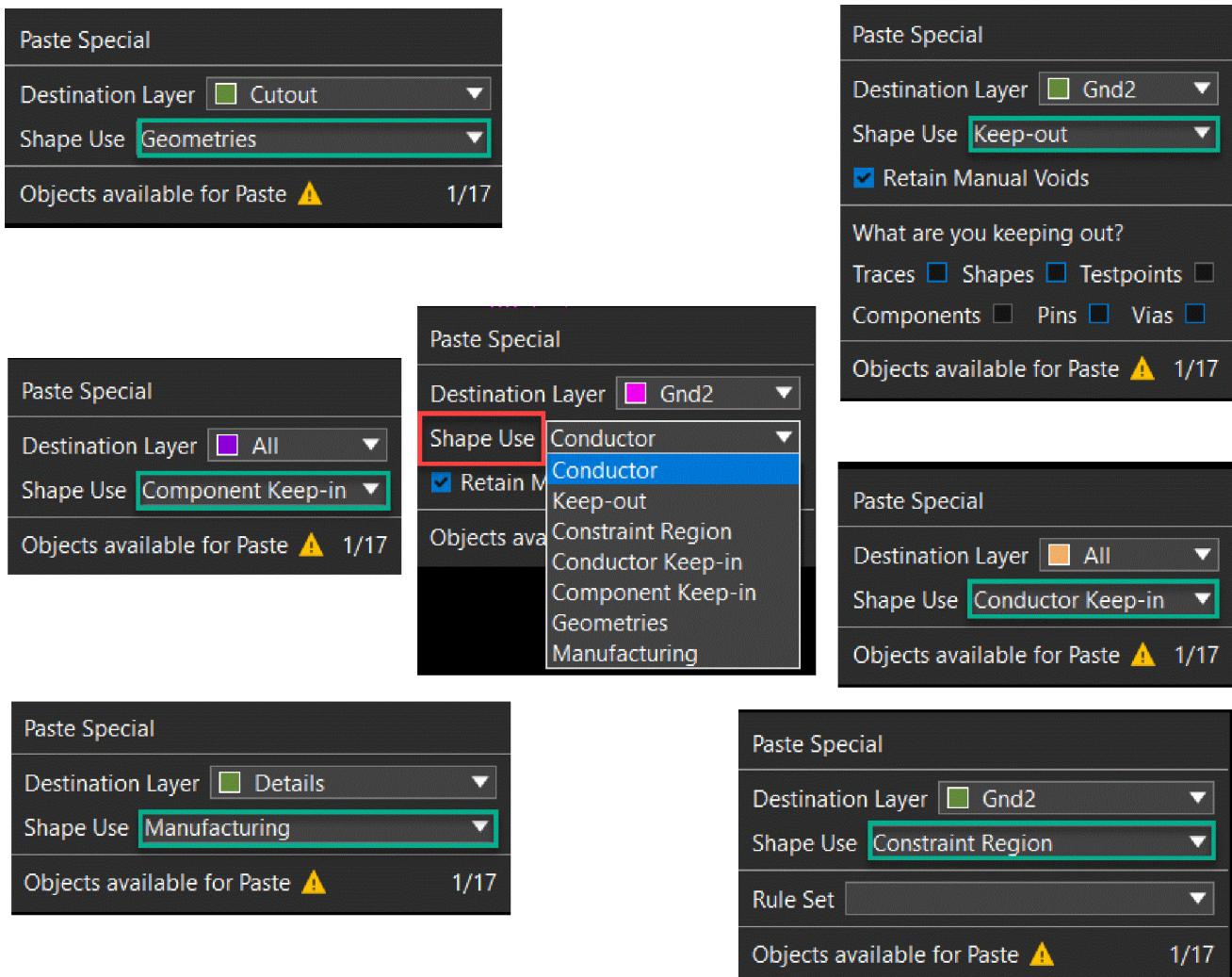


An error is displayed as a red triangle when none of the objects being copied are available for paste. In case of the error, the cursor changes to red circle with slash (the universal no sign) as a visual cue to indicate that paste operation cannot be performed.



3. Choose a design layer from the *Destination Layer* setting as required.

The *Destination Layer* field lists all design layers. The default value of this field changes in context with the copied objects and the *Shape Use* settings as follows:



4. Modify the *Shape Use* setting to specify the shape of the pasted shape objects. This option is available only when a shape is copied. If multiple shapes exist in the copy buffer and none of them is valid for the shape type set as the *Shape Use* setting, no paste operation is performed.
5. If a shape is selected with voids, choose to exclude or retain manual voids using the *Retain Manual Voids* check box.
6. To paste the copied objects, enable the destination objects in the *Selection Filter*.
7. To select a destination, click the design canvas or window select multiple objects.

Measuring the Distance between Objects in OrCAD X Presto

Measuring the distance between objects displays the air gap between selection points. If pins or vias are selected, the center-to-center spacing is also measured. The measurement results can be customized to display the data with dual units and accuracy in a user-defined color. To measure the distance between two objects in OrCAD X Presto, do the following:..

1. Press and hold the `Alt` key.

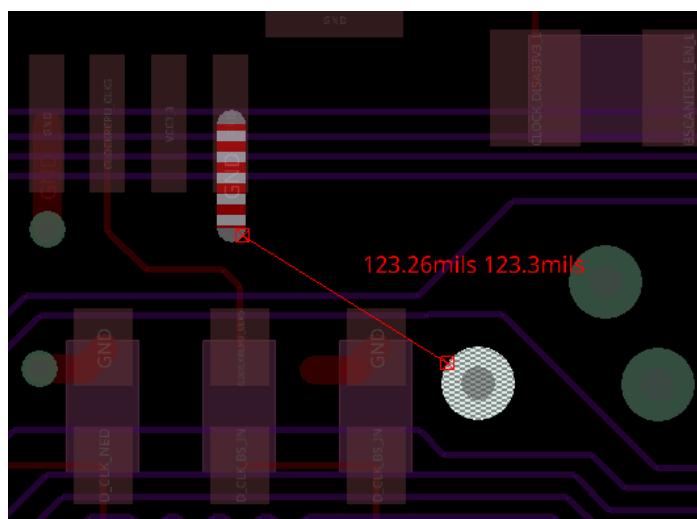
Shadow mode is temporarily enabled, and the measurement mode is active.

2. Click to choose the first object in the design canvas.

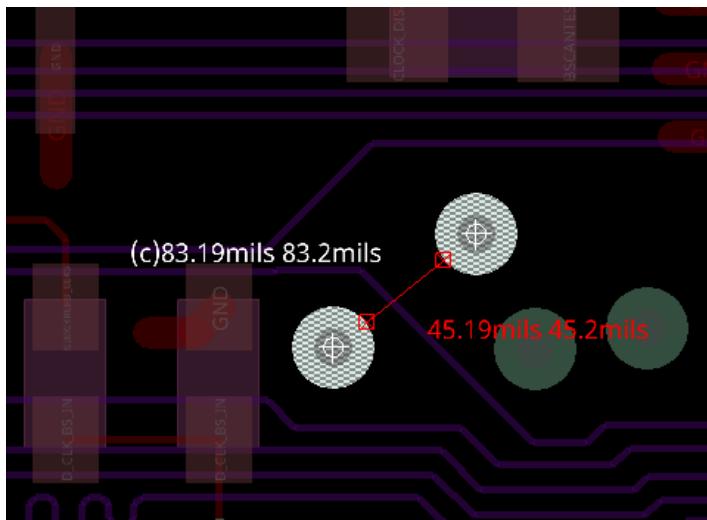
The selected object highlights in the canvas.

3. Hover the mouse over the second object you want to measure.

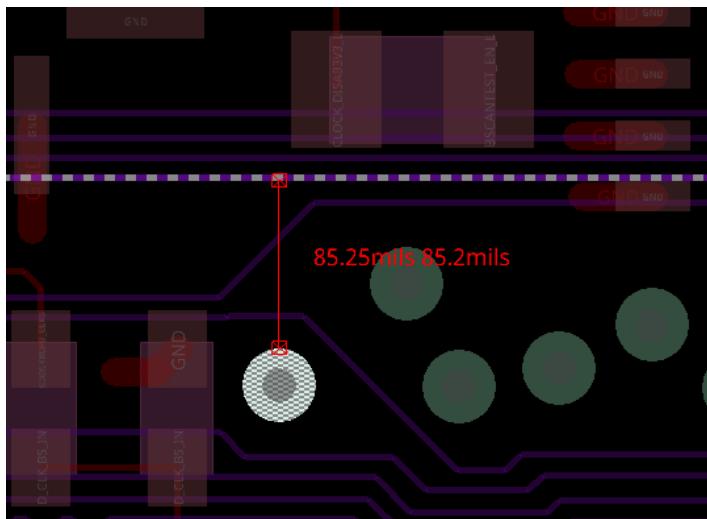
The second object is temporarily highlighted. A rectangle with two cross lines connecting two opposite corners indicates the point on an object where the air gap is measured. The air gap distance between the selected objects is represented by a dynamic straight red line drawn between the centers of the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



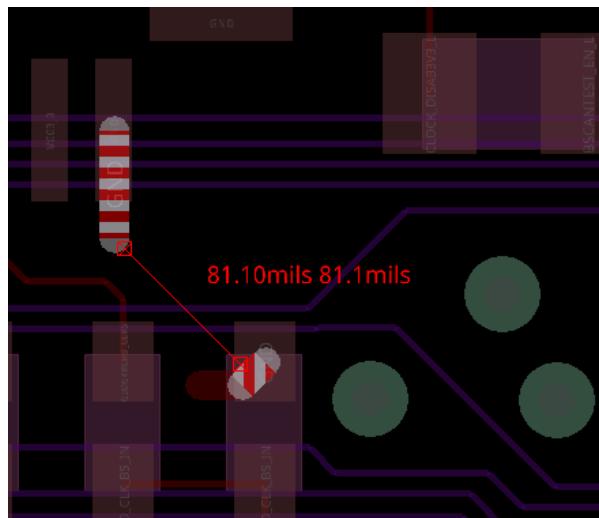
If the selected object is a pin or via, the center-to-center spacing between them is also displayed in white color, indicated with (c).



4. Hover over another object to choose a different object,
The first object remains the same, and the second object changes. The dynamic line also
changes and displays the air gap between objects.

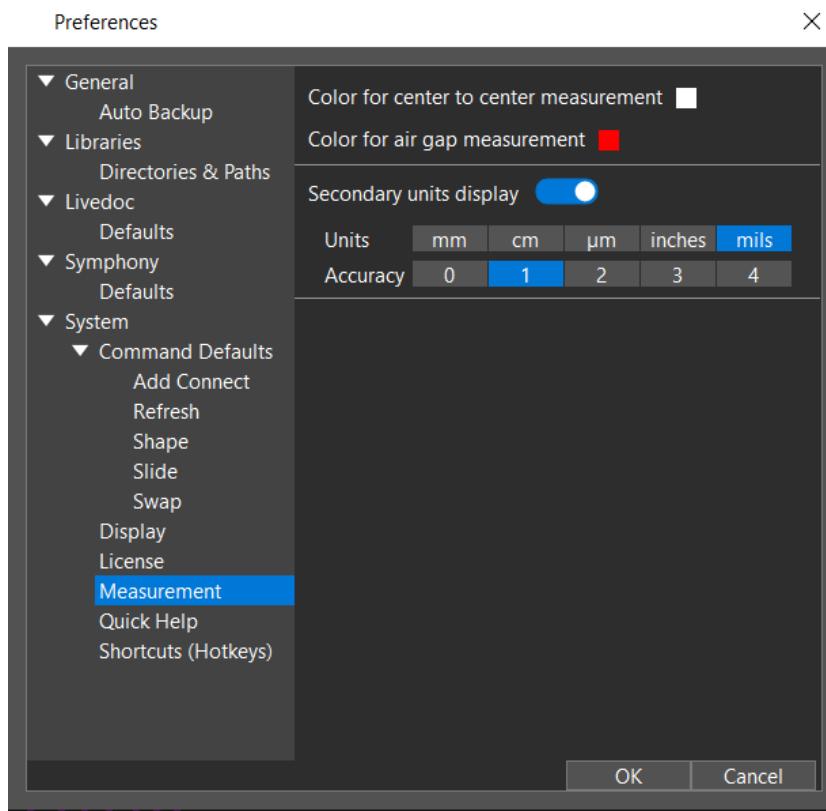


5. Repeat the previous step to choose different objects to dynamically see measured distance
values between the first and new objects.
6. Click another object to reset the selection of the first object,
The selected object is highlighted and defined as the first object.



7. Choose *Edit – Preferences* to modify the measurement settings.

The *Preferences* dialog box opens.



8. To exit measure mode, release the *Alt* key.

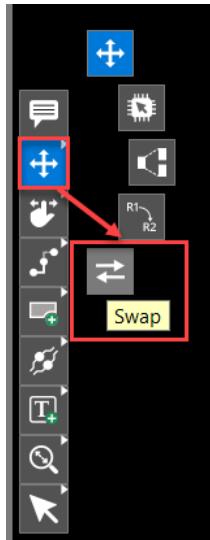
Shadow mode is also exited.

Swapping Components, Diff Pairs, and Pins in OrCAD X Presto

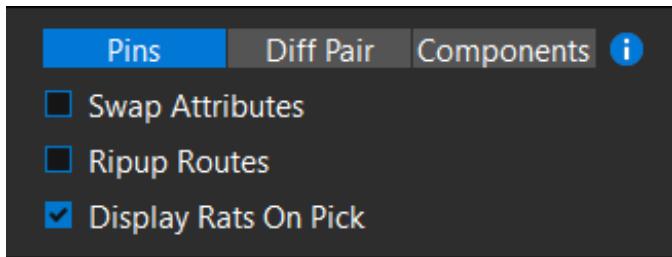
Components, pins, and diff pairs can be swapped to reduce trace lengths and uncross flight lines to improve routing and placement, particularly in complicated PCB designs with high pin count components, such as DDRs, FPGAs, and microcontrollers.

To enable the swap commands in OrCAD X Presto do the following:

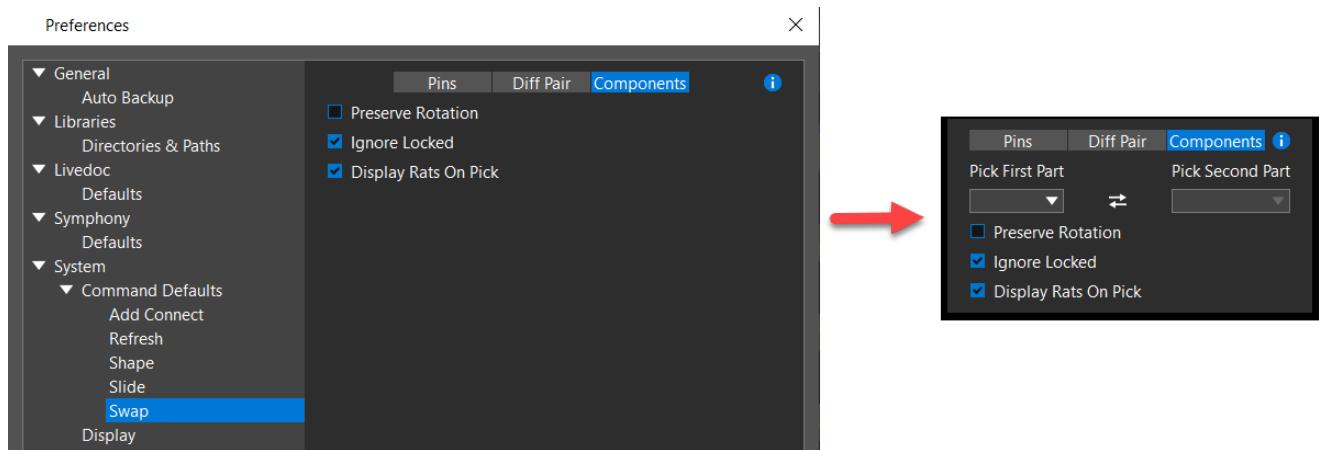
1. Right-click the *Move* icon in the floating command toolbar and select the *Swap* icon.



A floating command window opens with three tabs: *Pins*, *Diff Pair*, and *Components*.



2. Modify the default settings of the swap floating command window using the *Preferences* dialog box.



3. Optionally, assign a shortcut key or hotkey to access the swap commands quickly. You can assign shortcuts in the *Preferences* dialog box.

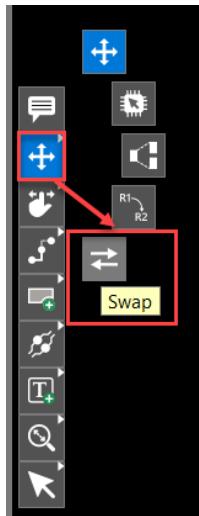
Related Topics

- [Customizing Keyboard Shortcuts in OrCAD X Presto](#)
- [Swapping Pins in OrCAD X Presto](#)
- [Swapping Diff Pairs in OrCAD X Presto](#)
- [Swapping Components in OrCAD X Presto](#)

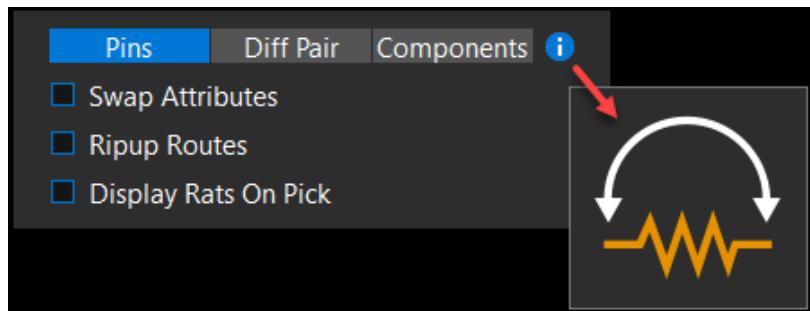
Swapping Pins in OrCAD X Presto

You can swap two pins if they have the same PIN USE values. The property `LAST_PIN_SWAP` is attached to pins and tracks all swapped pins. The value of the property is the name of the pin with which it was last swapped. To swap pins in OrCAD X Presto, do the following,

1. Right-click the *Move* icon from the floating command toolbar and select the *Swap* icon.



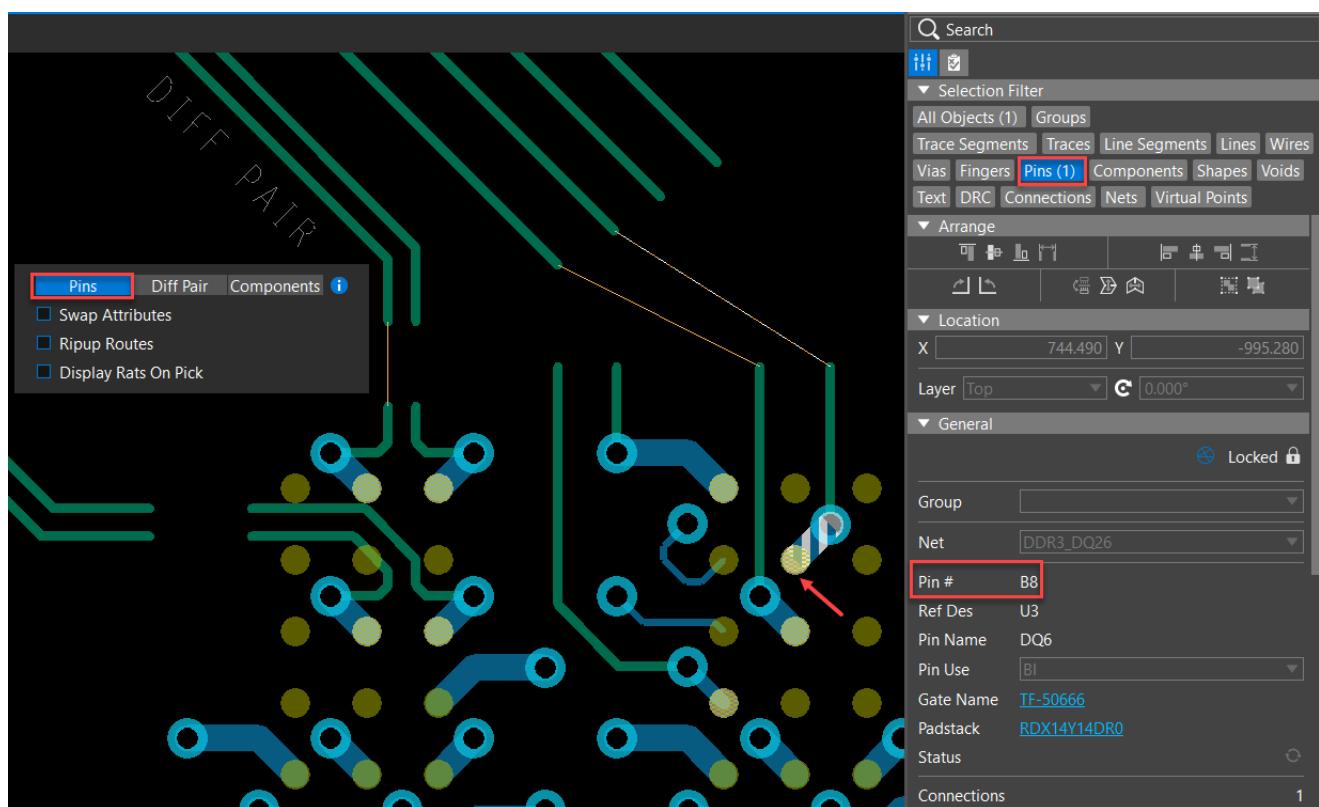
2. Choose *Pins* in the floating command window and hover the cursor over the icon to display the guiding image.



3. Enable *Pins* in the *Selection Filter*.

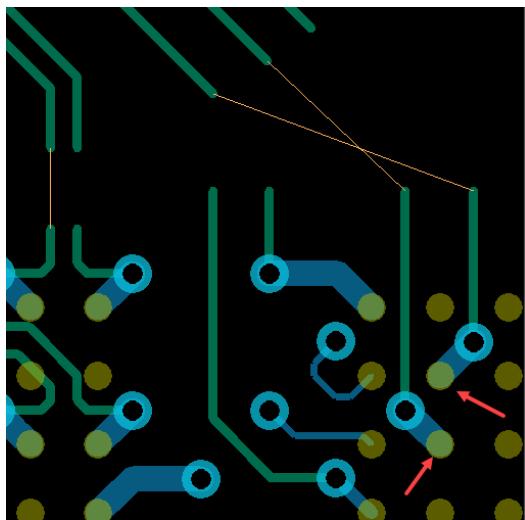
4. Click to choose the first pin to swap.

The chosen pin, flight lines, and all pins available to swap become highlighted.



5. Choose the second pin to swap from the highlighted pins.

The swapping is completed as displayed in the Session Log window.

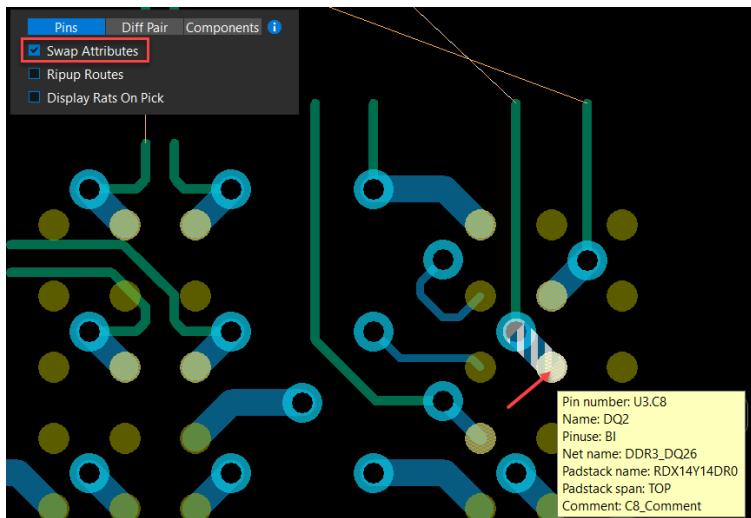


If you choose a pin that is not highlighted, the Session Log window prompt explains why the pin is not swappable.

6. Enable the *Swap Attributes* check box and select the pins to swap.

Pins are swapped with their attributes, which you can specify in the *Attributes* section of the *Properties* panel. The following image illustrates the pin swap with their attributes:

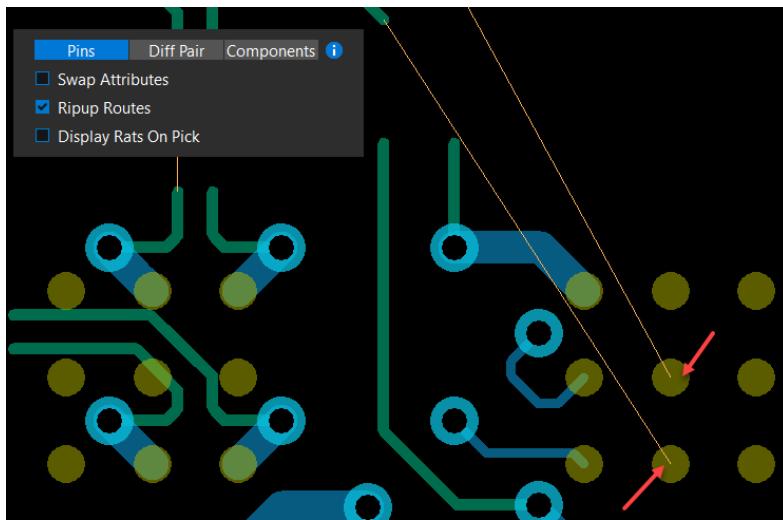




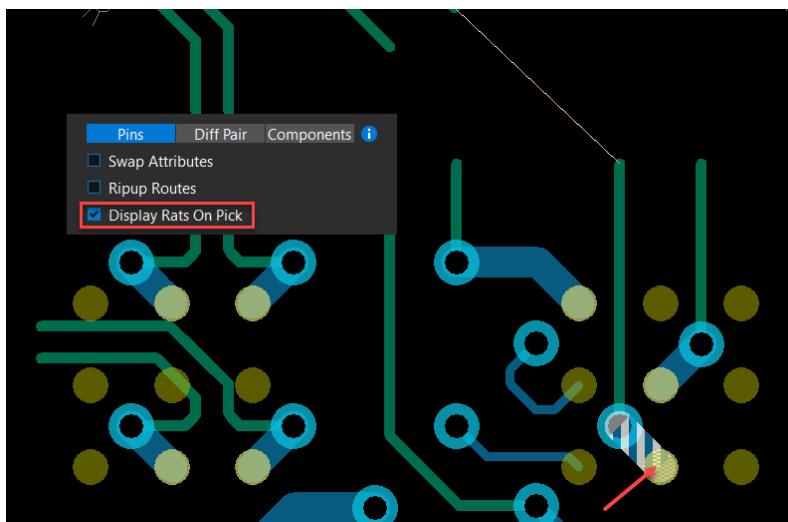
7. Enable the *Ripup Routes* check box and select the pins to swap.

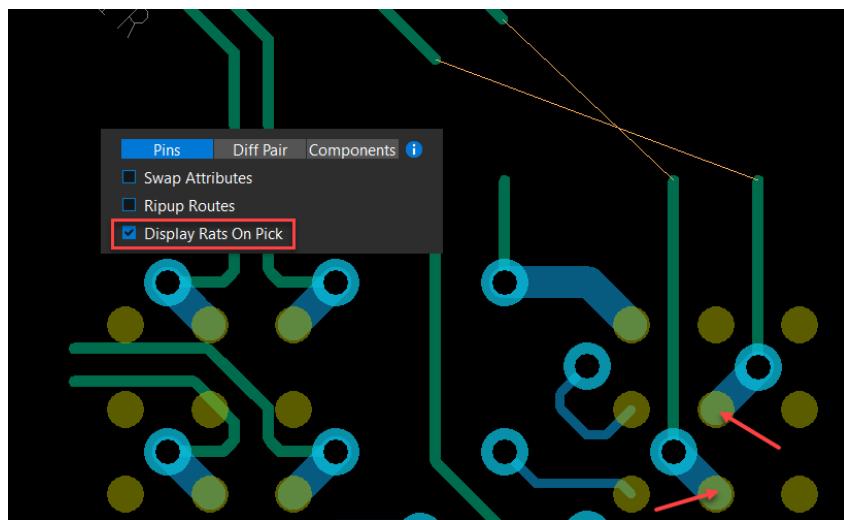
Pins are swapped with their etch elements removed. The following image displays the removal of associated etch and vias after pin swapping:





8. Enable *Display Rats on Pick* check box and select the pins to swap.
Rats are displayed when the pins are for swapping, as follows:





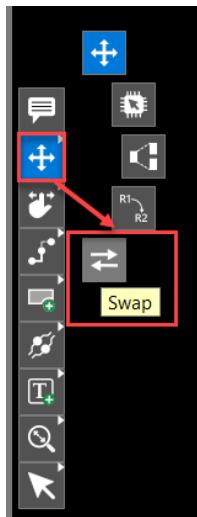
9. Select pins to remain in the pin-swapping command or press *Escape* to exit the command.

Swapping Diff Pairs in OrCAD X Presto

Diff pair swapping has two modes: *Diff Pair Swap* and *Polarity Swap*. The *Diff Pair Swap* mode swaps two pins at one end of a diff pair with two pins at one end of another diff pair. The *Swap Polarity* mode swaps the negative and positive pins at one end of a single diff pair. The property `LAST_PIN_SWAP` is attached to pins and tracks all swapped pins. The value of the property is the name of the pin with which it was last swapped.

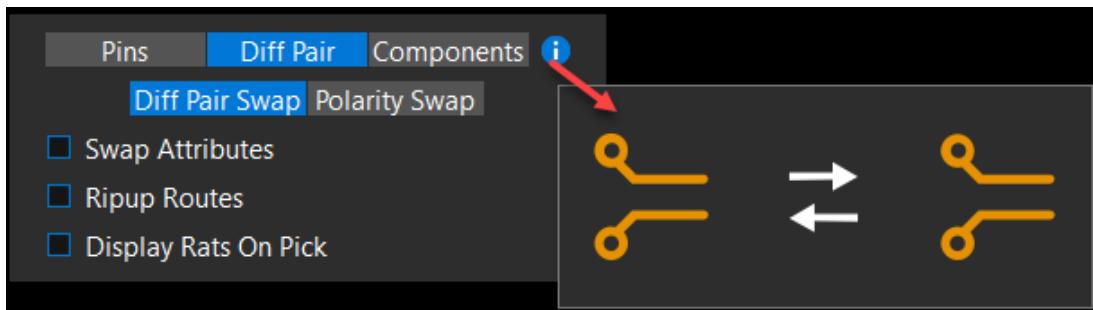
To swap diff pairs in OrCAD X Presto, do the following:

1. Right-click the *Move* icon from the floating command toolbar and select the *Swap* icon.



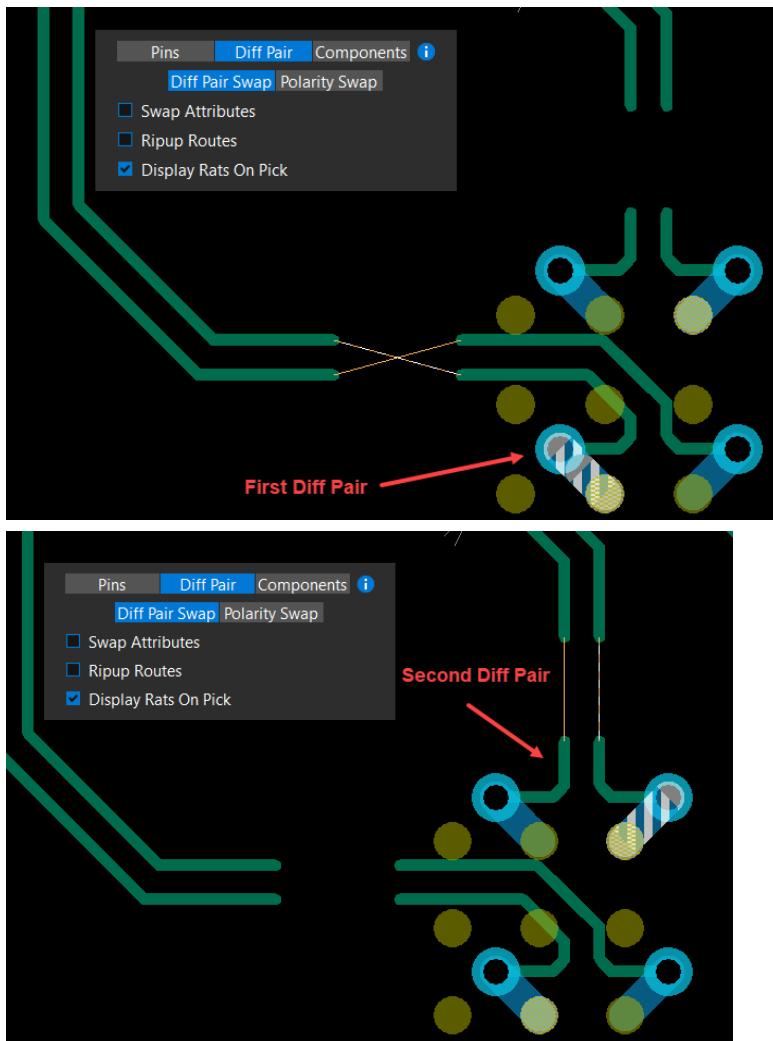
The *Swap* icon becomes active in the toolbar, and a floating command window opens with three tabs: *Pins*, *Diff Pair*, and *Components*.

2. Choose *Diff Pair Swap* in the swap command window and hover the cursor over the icon to display the guiding image.

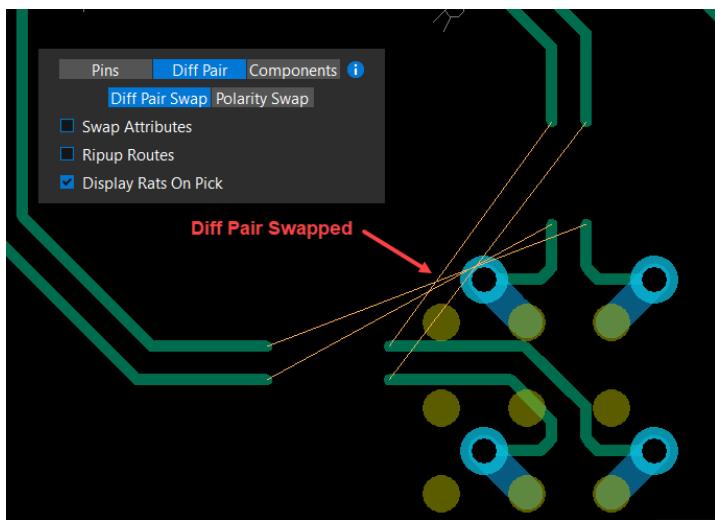


3. Enable the *Display Rats On Pick* check box. Use this option if rats nests display is turned off in the design.

4. Select *Pins* in the *Selection Filter*.
5. Choose a pin of a diff pair net to indicate the first pin pair to swap.
The chosen pin, flight line, and eligible pin of other diff pair nets become highlighted.

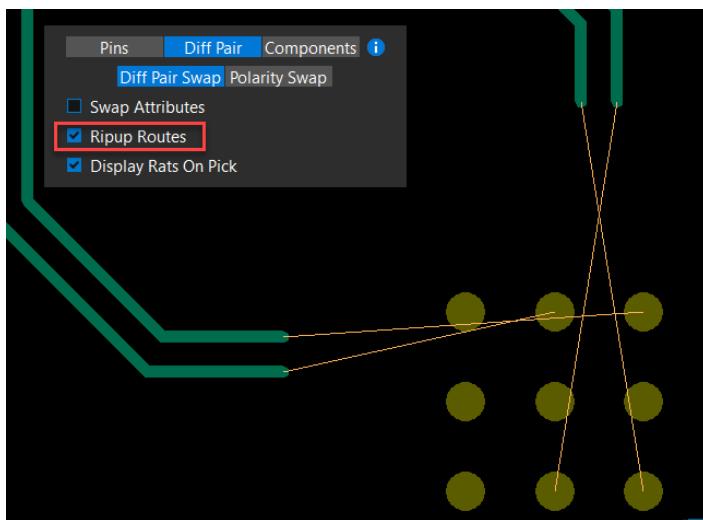


6. Choose a pin of another diff pair net from the highlighted pins to swap.
The pins of the diff pair nets are swapped, as follows:

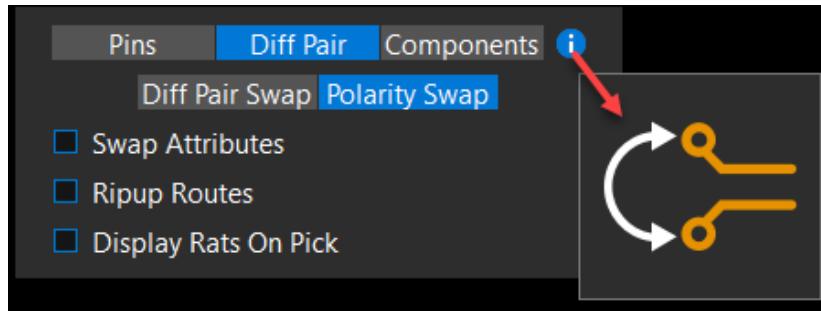


7. Enable the *Ripup Routes* checkbox and select diff pair pins to swap.

The pins of the diff pair nets are swapped with their etch elements removed, as follows:

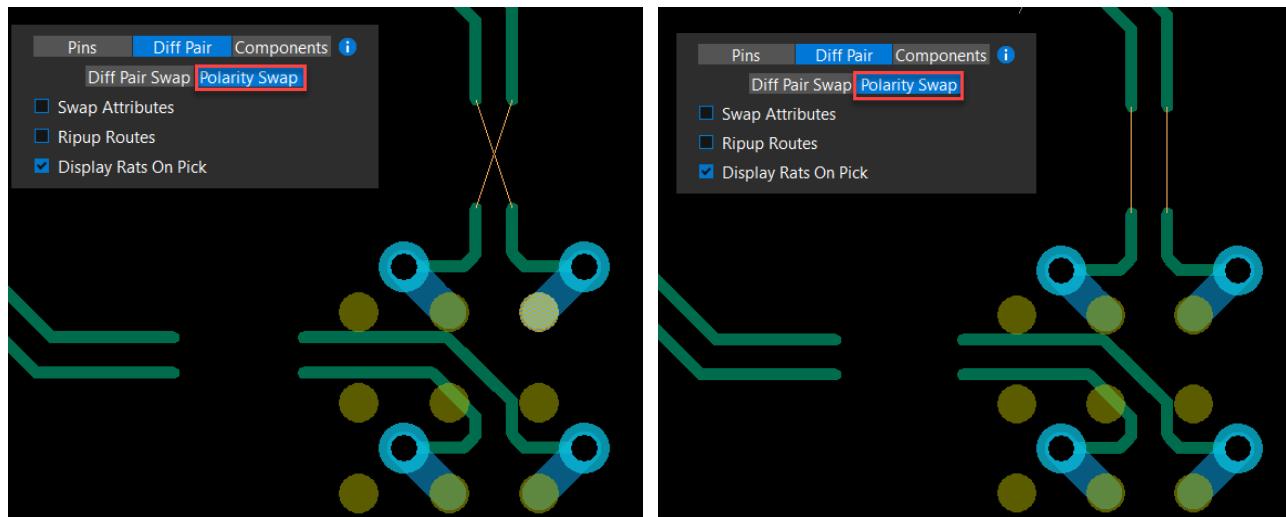


8. Choose *Polarity Swap* in the floating command window and hover the cursor over the icon to display the guiding image.



9. Enable the *Display Rats On Pick* check box if the rats nests are not visible in the design.
10. Choose a pin of a diff pair to swap pin polarity.

The polarity of the pin pair is swapped, as follows:



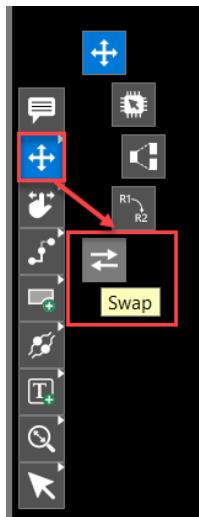
11. Select pins to remain in the pin-swapping command or press *Escape* to exit the command.

Swapping Components in OrCAD X Presto

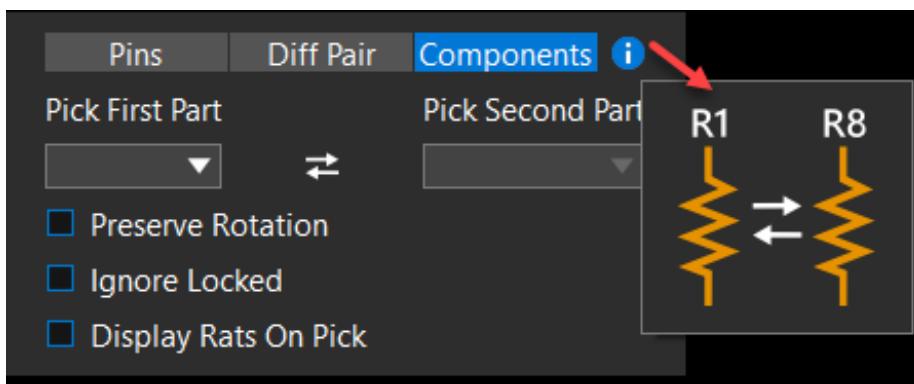
You can swap components in OrCAD X Presto by choosing them in the design canvas or the combo box of the swap command floating command window. The locked components cannot be selected for swapping and should be unlocked manually.

To swap components:

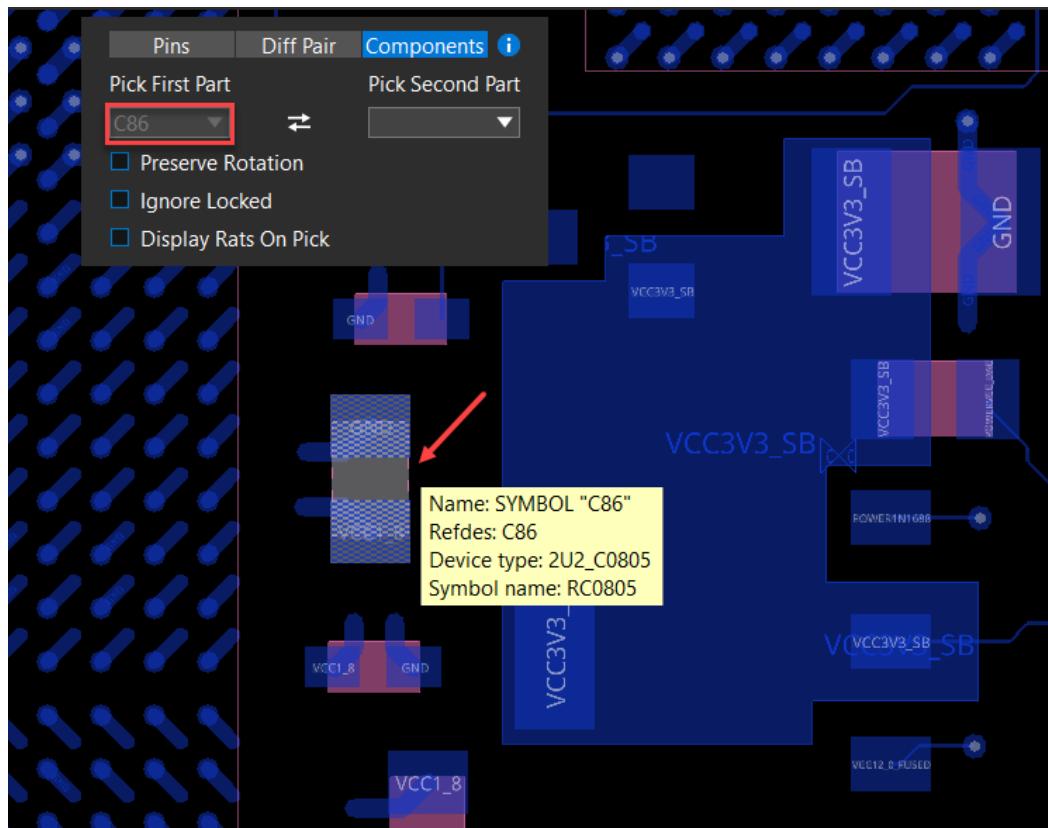
1. Right-click the *Move* icon from the floating command toolbar and select the *Swap* icon.



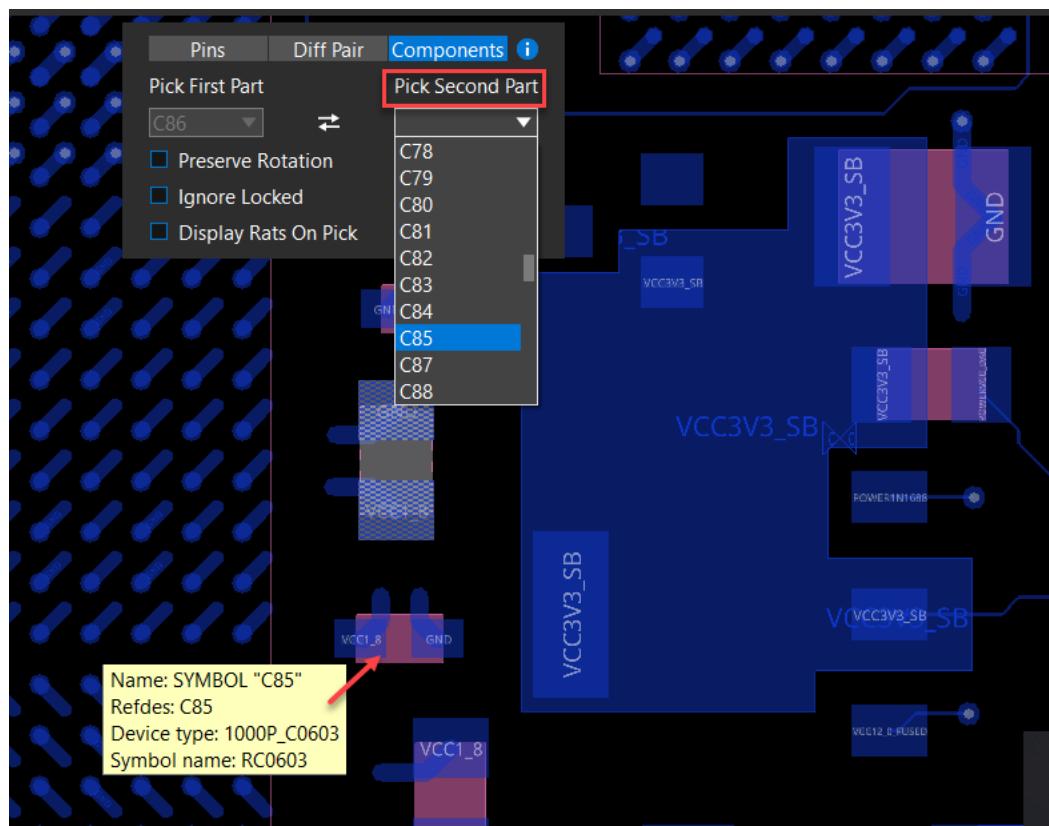
2. Choose *Components* in the floating command window and hover the cursor over the icon to display the guiding image.



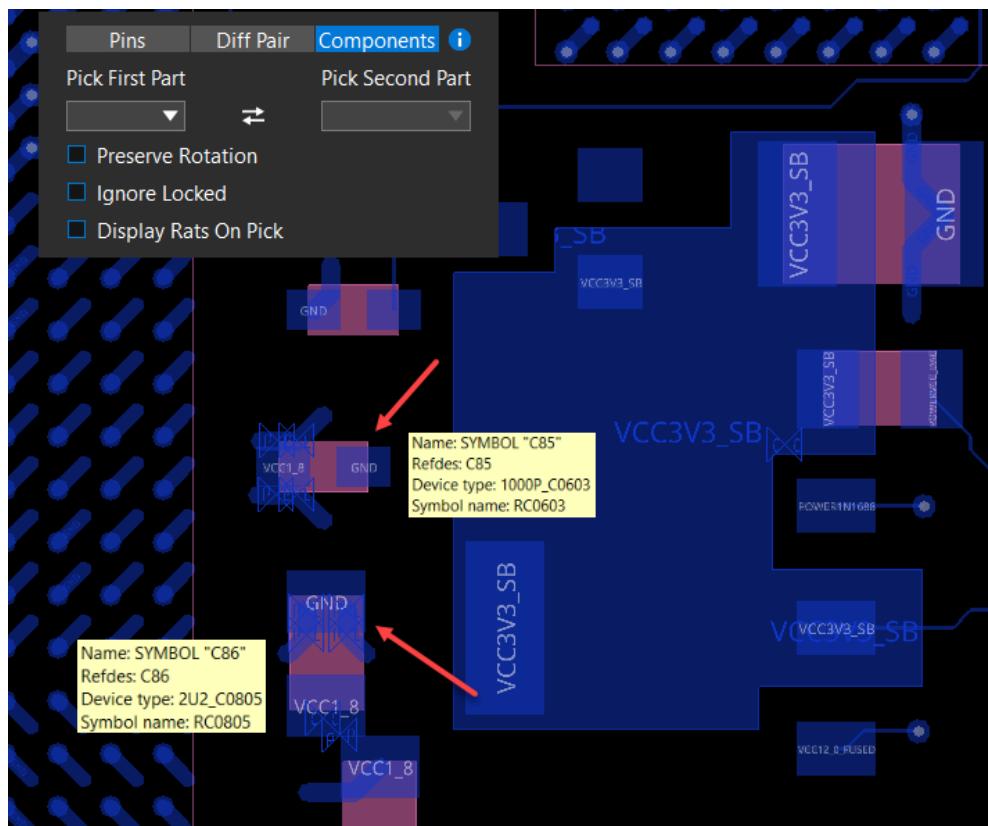
3. Select the first component in the design canvas or in the *Pick First Part* pull-down list. This list displays reference designators of unlocked components.
The component is highlighted, and its reference designator is displayed in the *Pick First Part* combo box.



4. Click the second component in the design canvas or select the reference designator in the *Pick Second Part* pull-down list.



The components are swapped, and DRCs are displayed for any violation resulting from the component swapping.



5. Enable the *Ignore Locked* check box to select locked components for swapping.



You can lock and unlock a component by clicking the in the *Property* panel.

6. Enable *Preserve Rotation* check box to retain the orientation of the swapped components.
7. Select components to continue swapping or press *Escape* to exit the command.

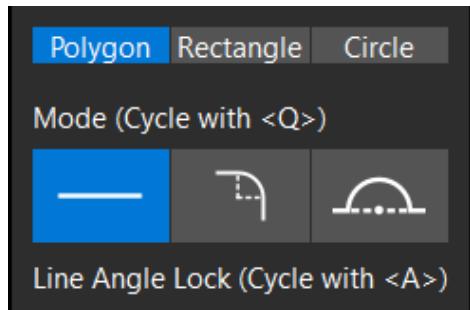
Creating Custom Design Outline in OrCAD X Presto

The custom design outline can be interactively created in rectangular, square, circular, or polygon shapes in OrCAD X Presto. When creating a design outline by drawing shapes, arcs, and lines, the specific measurements are dynamically shown to assist the drawing creation. The new design outline replaces the existing design outline in the canvas. Additionally, conductor and component keep-in shapes are re-generated based on the settings in the *Property* panel.

To modify the default design outline, do the following:

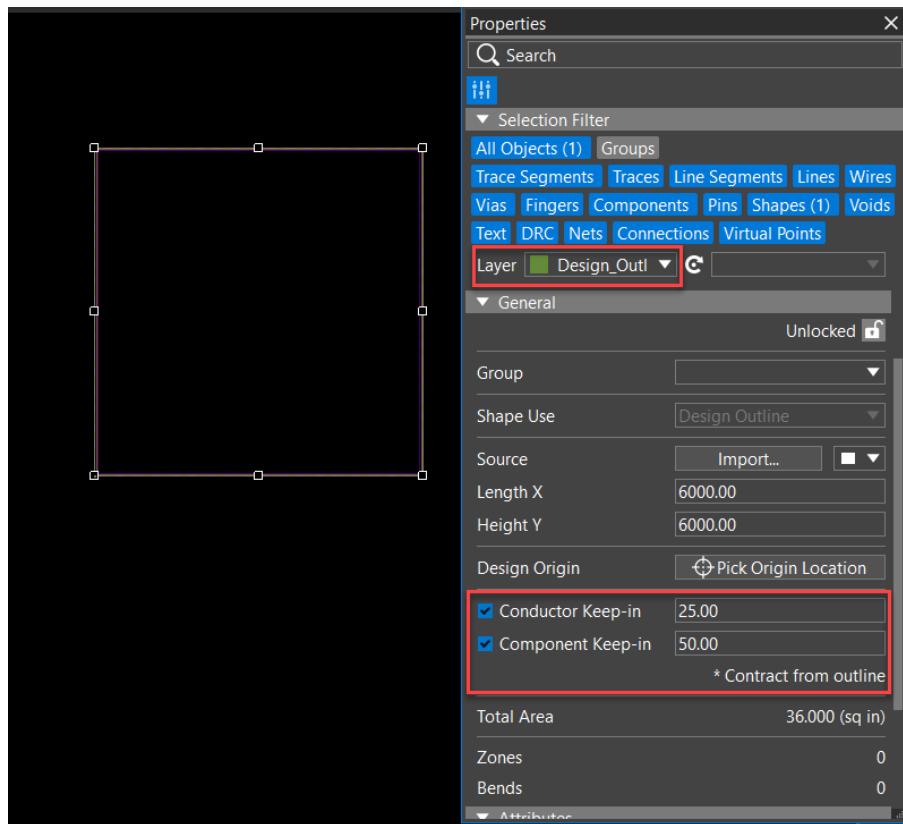
1. Choose *ECO – Draw Design Outline* from the main menu.

A floating command window opens, displaying design outline drawing parameters. The default is a *Polygon*.



2. Click to select the existing design outline.

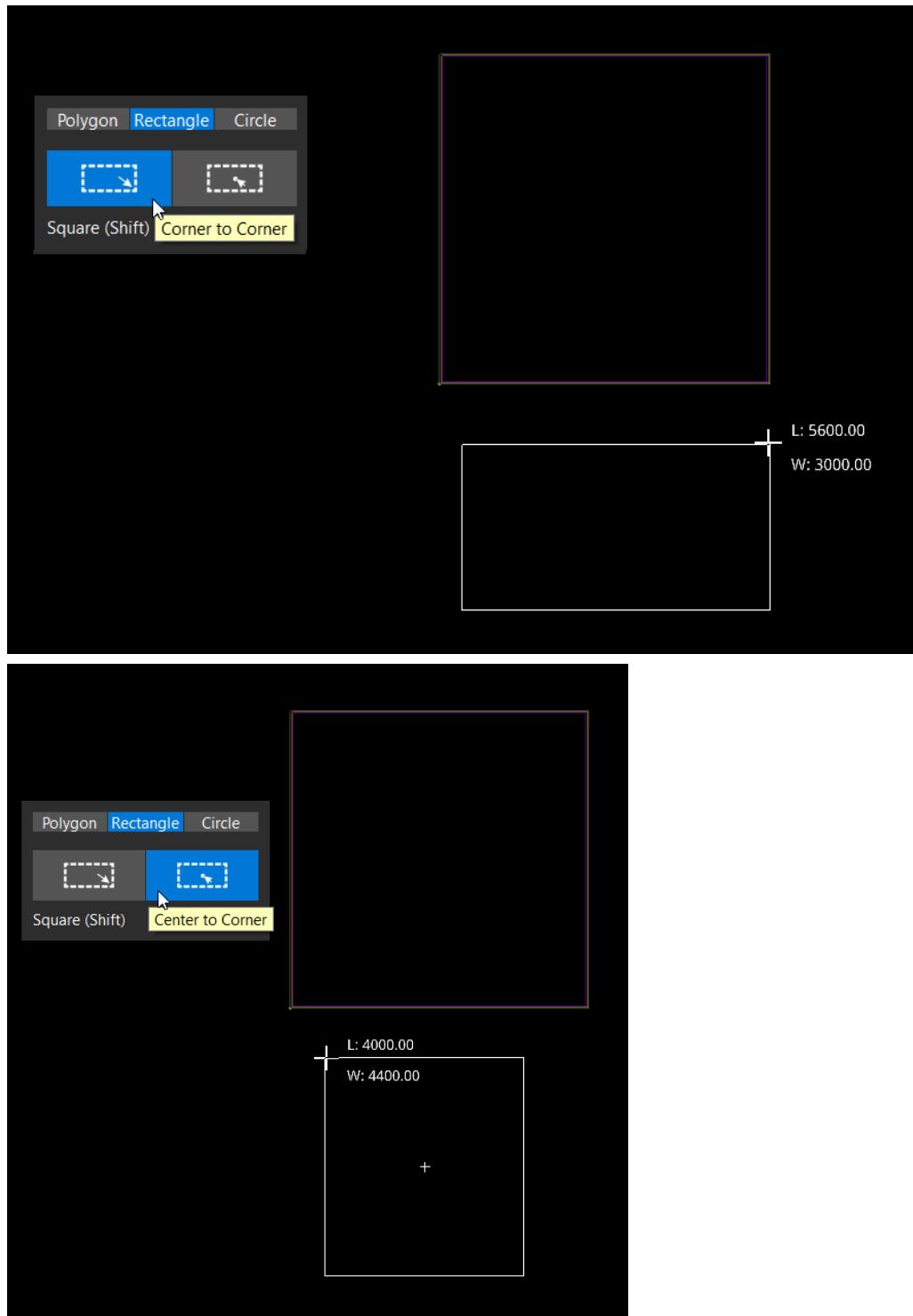
The design outline shape is highlighted, and the *Property* panel displays the drawing details.



3. Review the *Property* panel settings for *Conductor Keep-in* and *Component Keep-in*. Modify the settings as required.
4. Draw a new rectangular or square design outline as follows:
 - a. Select the *Corner to Corner* or *Center to Corner* mode in the floating command window. Press `Shift` while selecting the mode to create a square outline shape.
 - b. Locate the coordinates by moving the cursor where you want to start the outline and click on the design canvas.
The selected point is used as the first corner or center of the rectangle, depending on the mode enabled in the floating command window.
 - c. Move the cursor in the desired direction, following the dimension values displayed with the cursor movement.
The second corner is attached to the cursor location and dynamically displays the dimensions as you move the cursor.
- d. Press `Enter` or click on the design canvas to complete the rectangle design outline

Press the `Backspace` key to undo the last action.

shape.

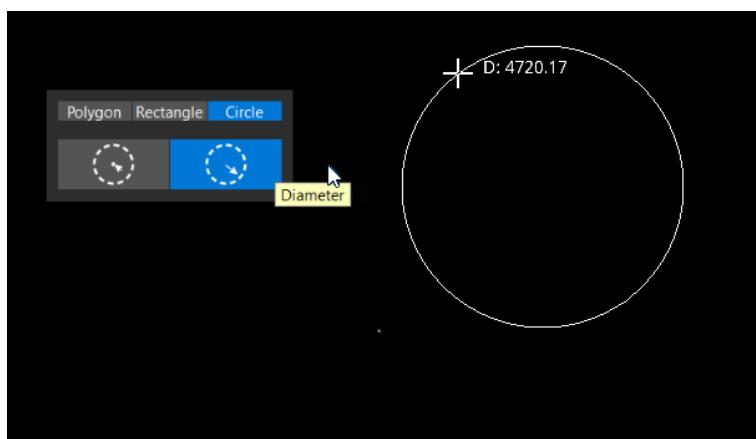
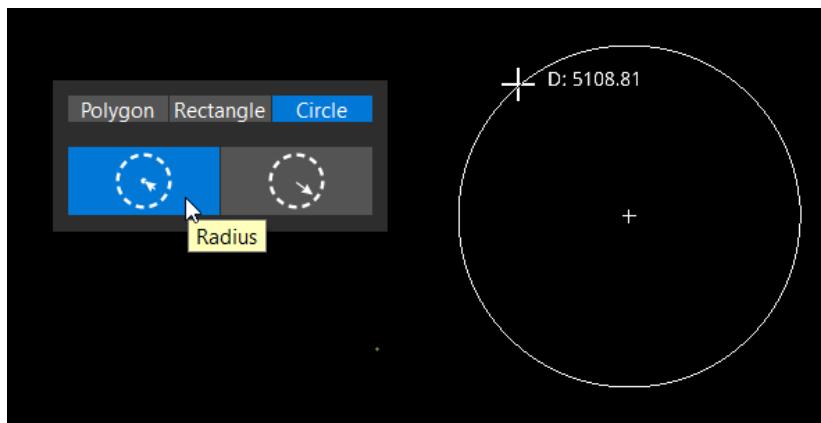


The new rectangular design outline replaces the existing design outline. The keep-in

shapes are regenerated as per the settings in the *Property* panel.

5. Draw a circular outline using the following steps:

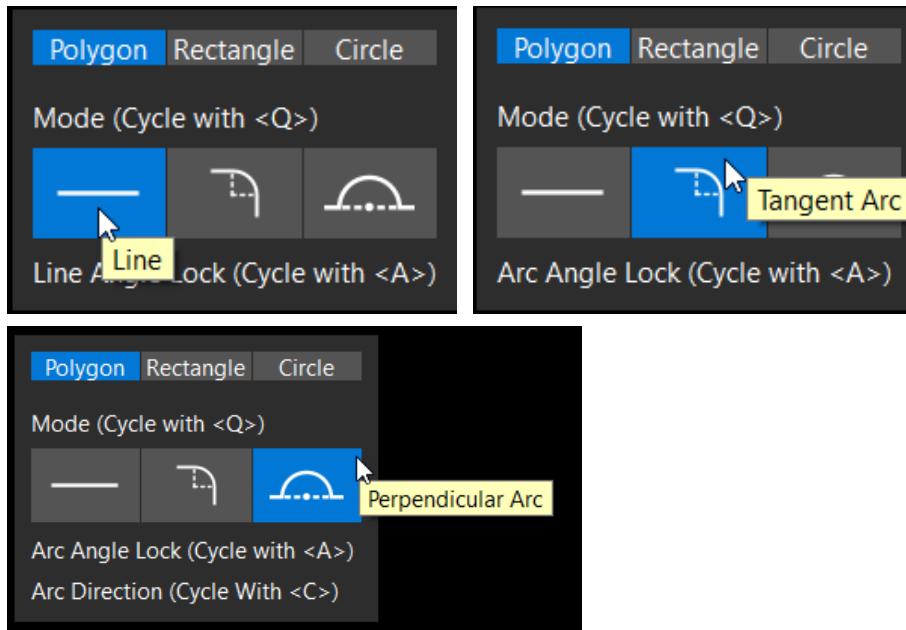
- a. Select the *Radius* or *Diameter* mode in the floating command window.
 - b. Locate the coordinates by moving the cursor where you want to start the outline and click on the design canvas.
- The selected point is used as the center or a point on the circumference of the circle, depending on the mode enabled in the command window.
- c. Move the cursor to draw a circle with the required radius or diameter.
The circle dimensions are dynamically displayed as you move the cursor.
 - d. Press `Enter` or click on the design canvas to complete the circular design outline shape.



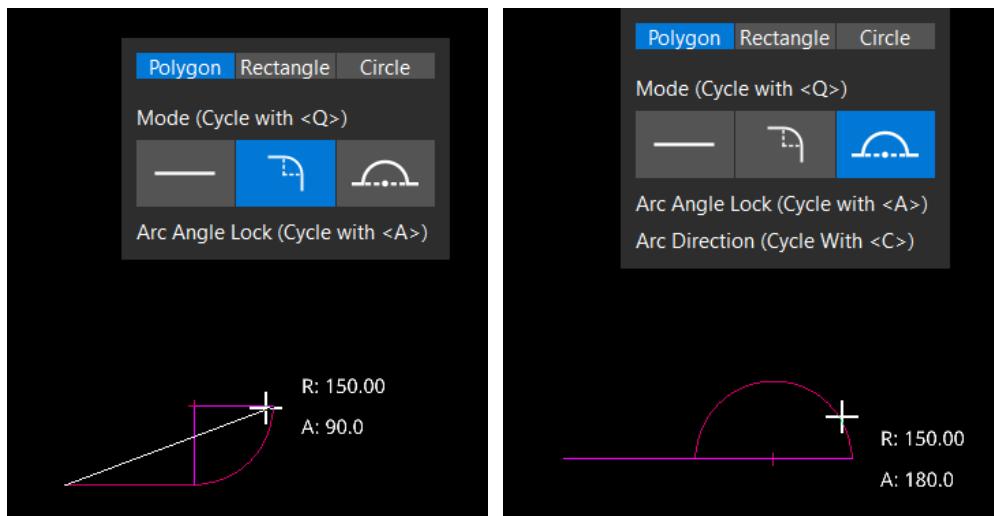
The new circular design outline shape replaces the existing design outline. The keep-in shapes are regenerated as per the settings in the *Property* panel.

6. Draw a polygon shape design outline by performing the following steps:

- a. Select *Line*, *Tangent Arc*, or *Perpendicular Arc* modes in the floating command window.



The center point of a *Tangent Arc* is orthogonal to the line segment it is connected, and mouse movements draw a 45-degree arc. The center point of the *Perpendicular Arc* aligns with the line segment it is connected to, and mouse movements draw a 180-degree half-circle. The arc can be drawn on either side of the line segment.



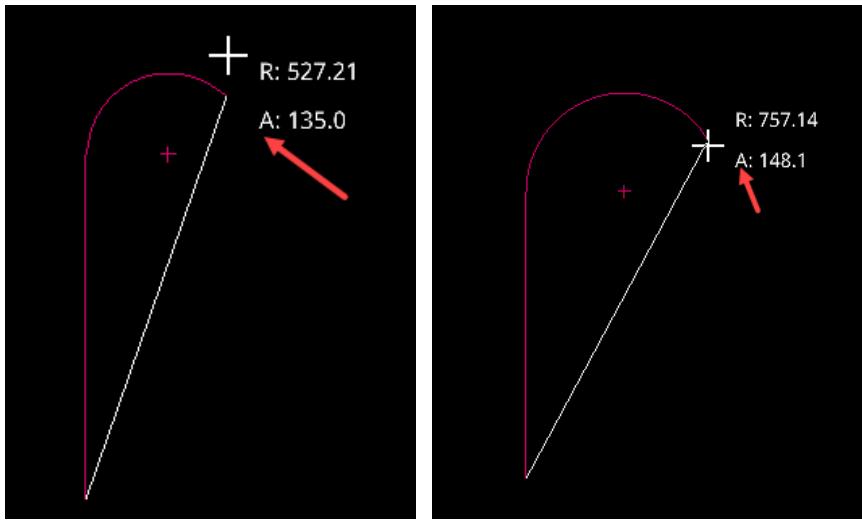
- b. Locate the coordinates by moving the cursor where you want to start the outline and click on the design canvas.

The selected point is used as the starting point.

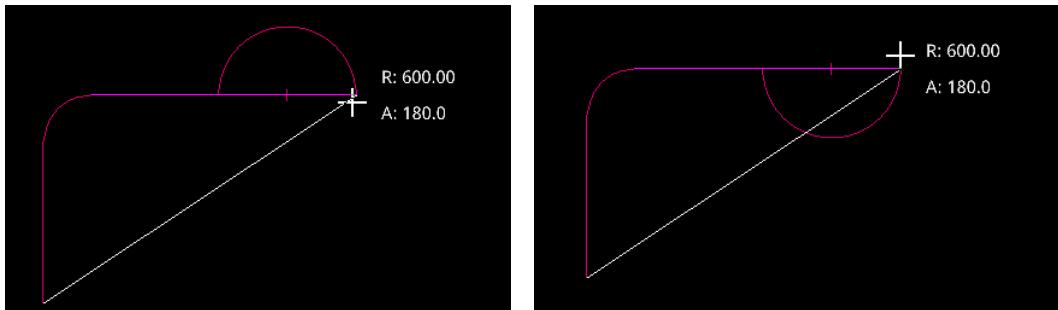
- c. Move the cursor to draw a line, arc, or half circle.

The dimensions are dynamically displayed as you move the cursor.

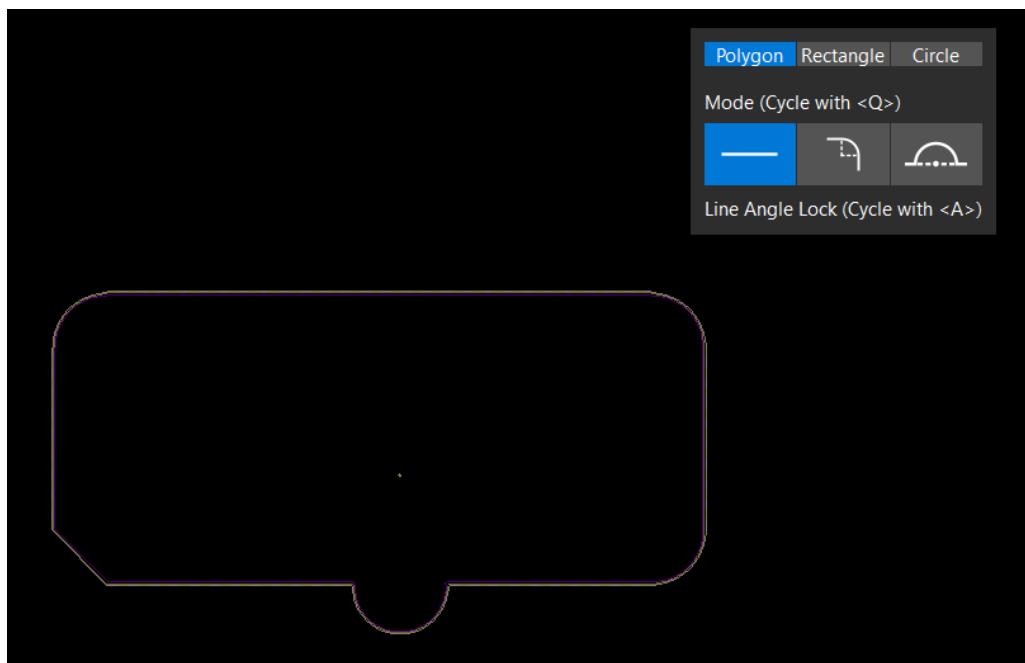
- d. Optionally, press the **A** key to toggle the angle lock values for a line or arc from 90 and 45 degree to any angle.



- e. Optionally, press the **C** key to change the arc direction for half circle.



- f. Keep drawing to complete the polygon shape design outline.



The new polygon shape design outline replaces the existing design outline. The keep-in shapes are regenerated as per the settings in the *Property* panel.

7. Press the `Escape` key to exit command.

Managing Shapes in OrCAD X Presto

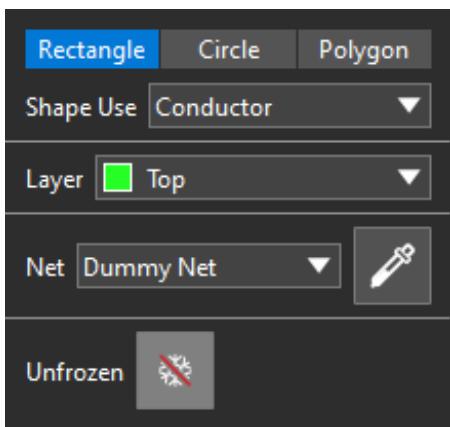
- [Creating and Adding Shapes in OrCAD X Presto](#)
- [Editing Shape Edges in OrCAD X Presto](#)

Creating and Adding Shapes in OrCAD X Presto

Choose the *Add Shape* icon from the toolbar.



Choose the shape (*Rectangle*, *Circle*, *Polygon*) you would like to draw, as shown in the following *Add Shape* dialog:



Drawing Rectangles and Circles

Drawing circles and basic rectangles is fairly straightforward.

Click and hold the left mouse button to begin drawing the selected shape. The point at which the cursor is place when you first click the mouse button will become a corner of your rectangle, or a point on the edge of circle, respectively.

As you move the cursor with the mouse button depressed, the shape will get larger or smaller, according to your movements. Let go of the mouse button to finalize the size of your shape.

Related Topics

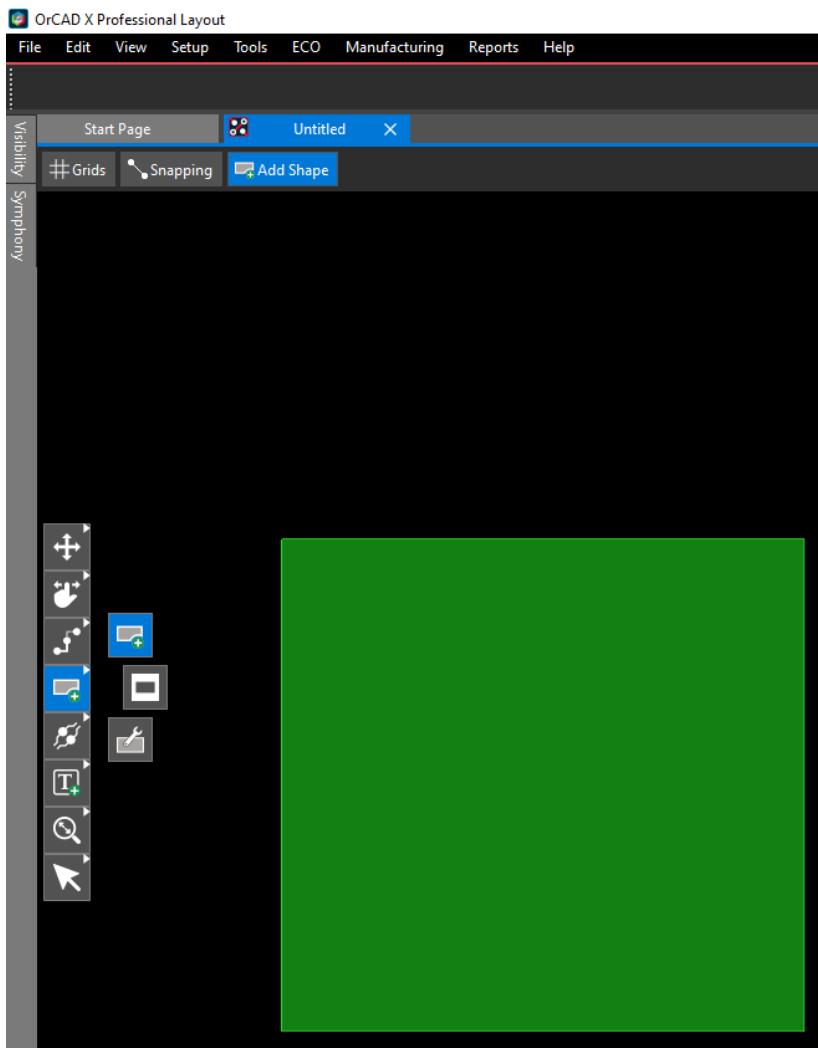
- [Authoring and Editing Footprints in OrCAD X Presto](#)
- [Creating and Editing Zones in OrCAD X Presto](#)
- [Creating and Editing Bends in OrCAD X Presto](#)
- [Editing Shape Edges in OrCAD X Presto](#)
- [Drawing Polygons](#)

Editing Shape Edges in OrCAD X Presto

Sometimes, it is easier to create the rough polygon of a shape with corners and then apply trimming to achieve desired chamfers or rounded arcs. In OrCAD X Presto, you can choose entire existing shapes or individual corners of those shapes, and trim them into either rounded arcs or chamfers.

To trim corners, first open the *Shape Utilities* dialog, as follows:

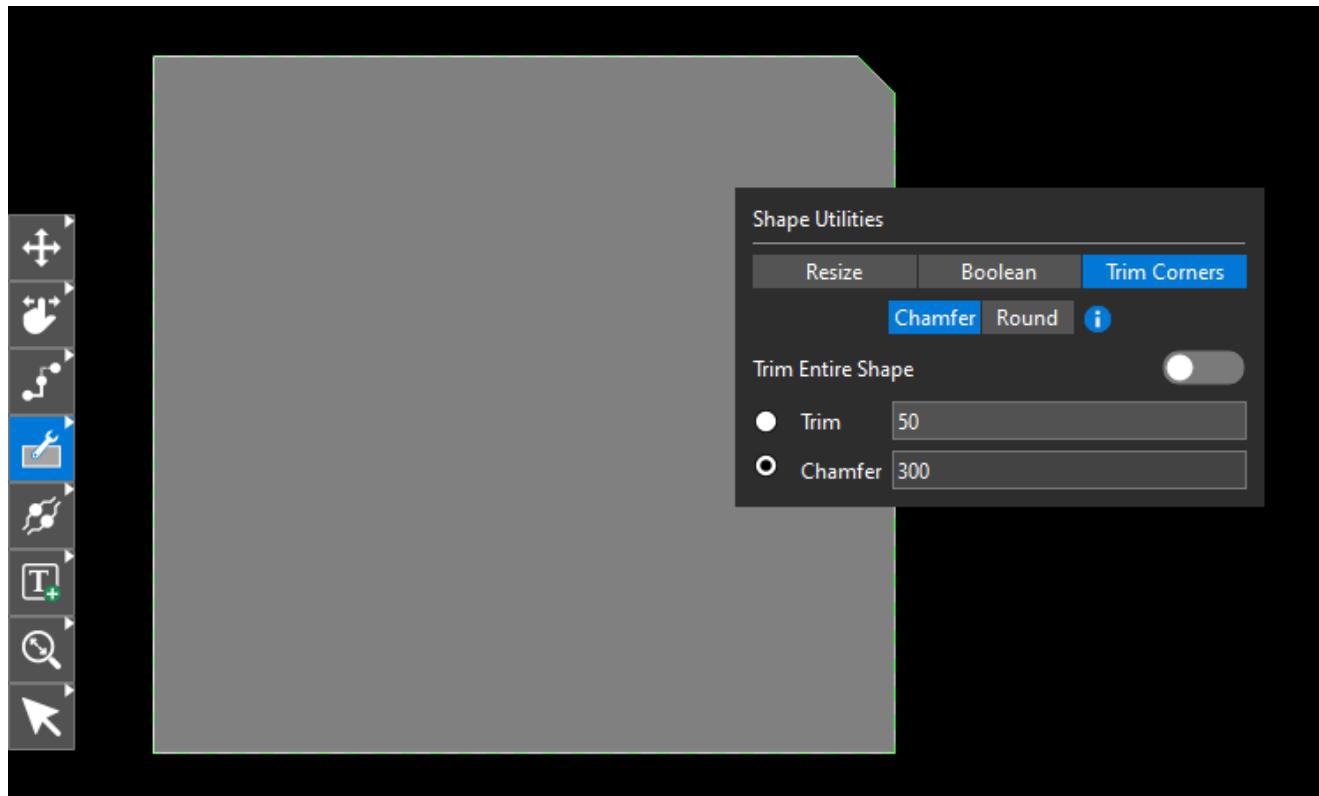
1. Right-click over the *Add Shape* icon in the toolbar.
Additional icons will appear, as shown below.
2. Click the bottom-most icon for *Shape Utilities*.



To edit corners of a shape individually:

1. In the *Shape Utilities* menu, choose *Trim Corners*.

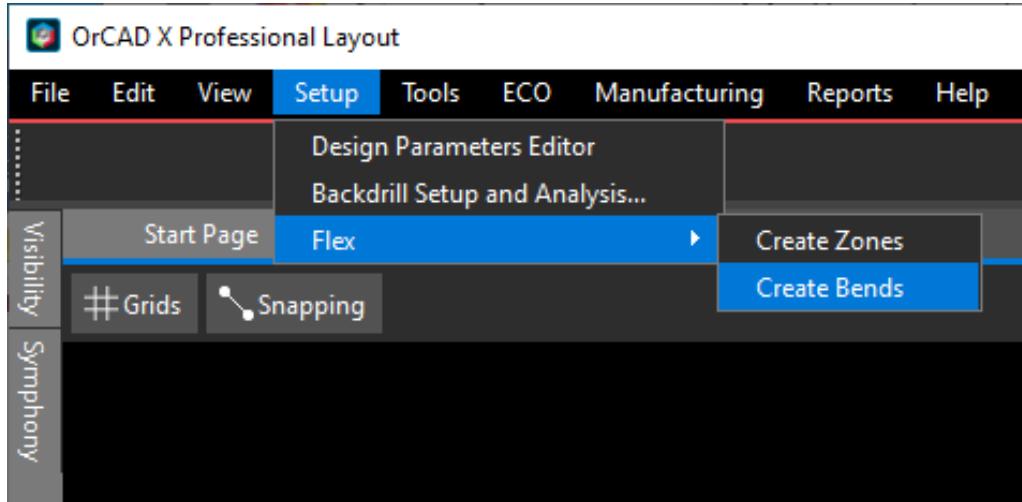
2. Toggle OFF *Trim Entire Shape*.



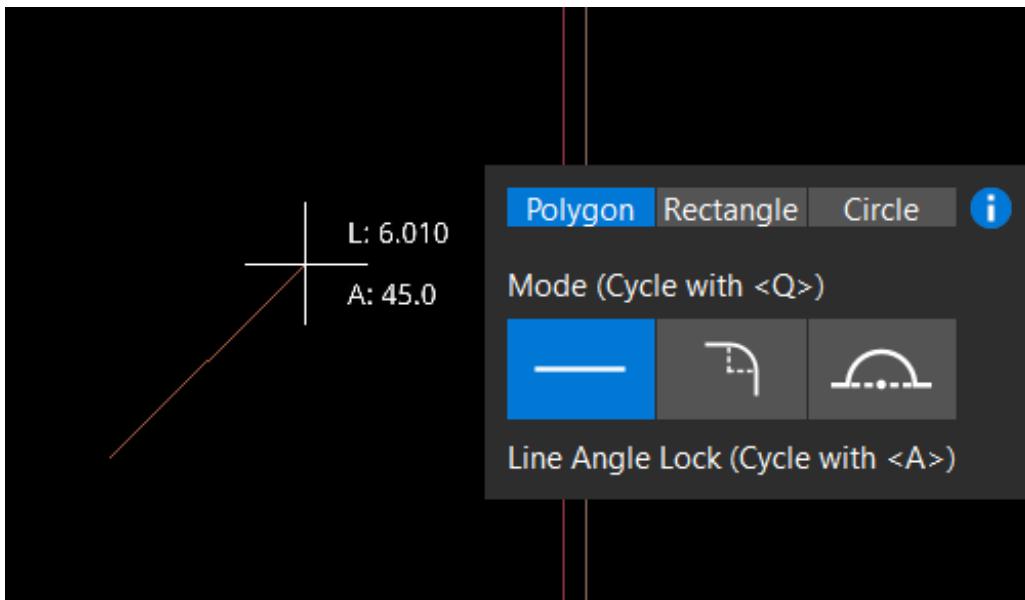
3. Choose either *Chamfer* or *Round*.
4. Set the desired chamfer or radius value.
5. Click on the corner of the shape you are trimming.

Creating Zones and Bends for Rigid-Flex Design in OrCAD X Presto

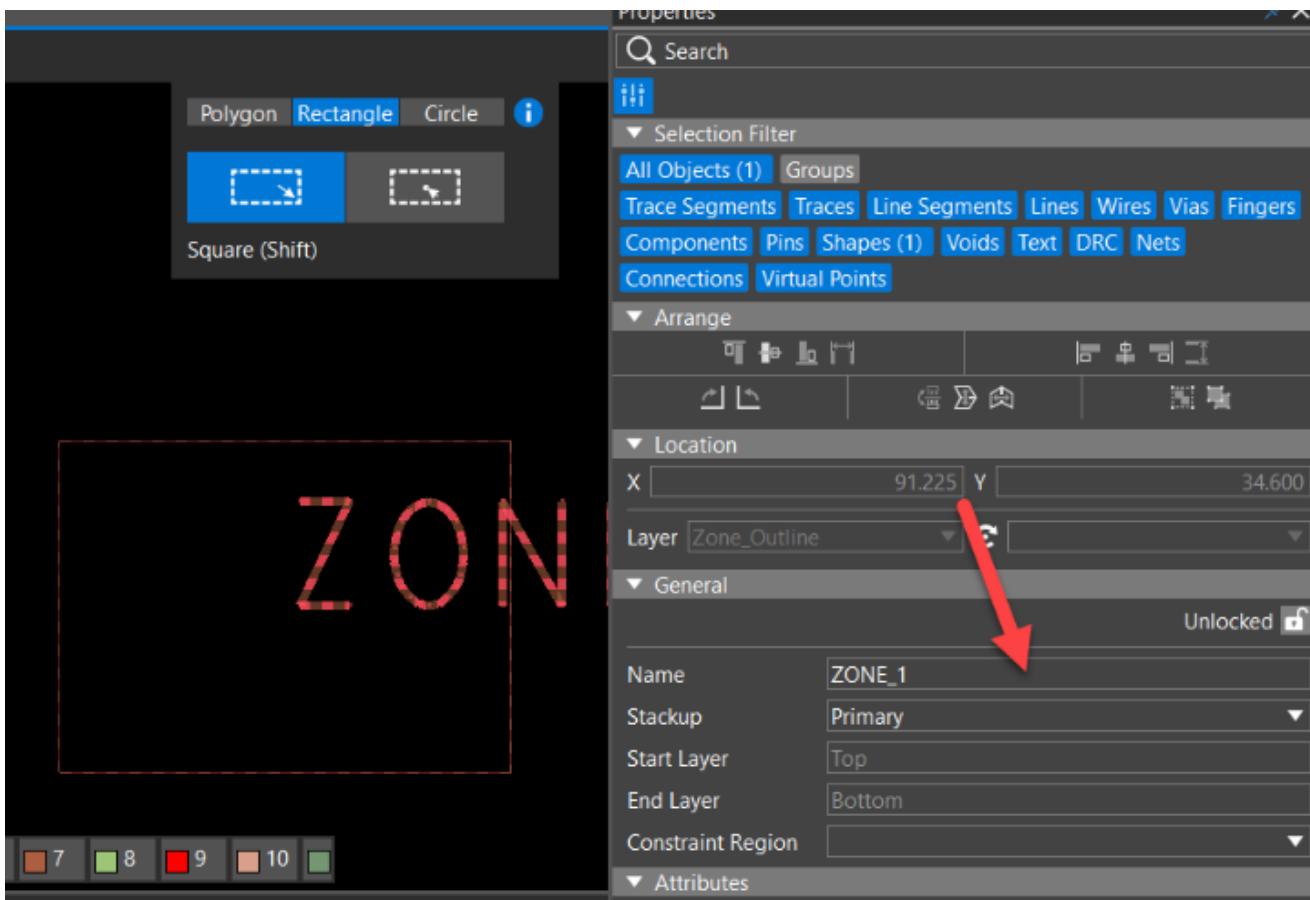
You can create both zones and bends from the *Setup – Flex* menu, as follows.



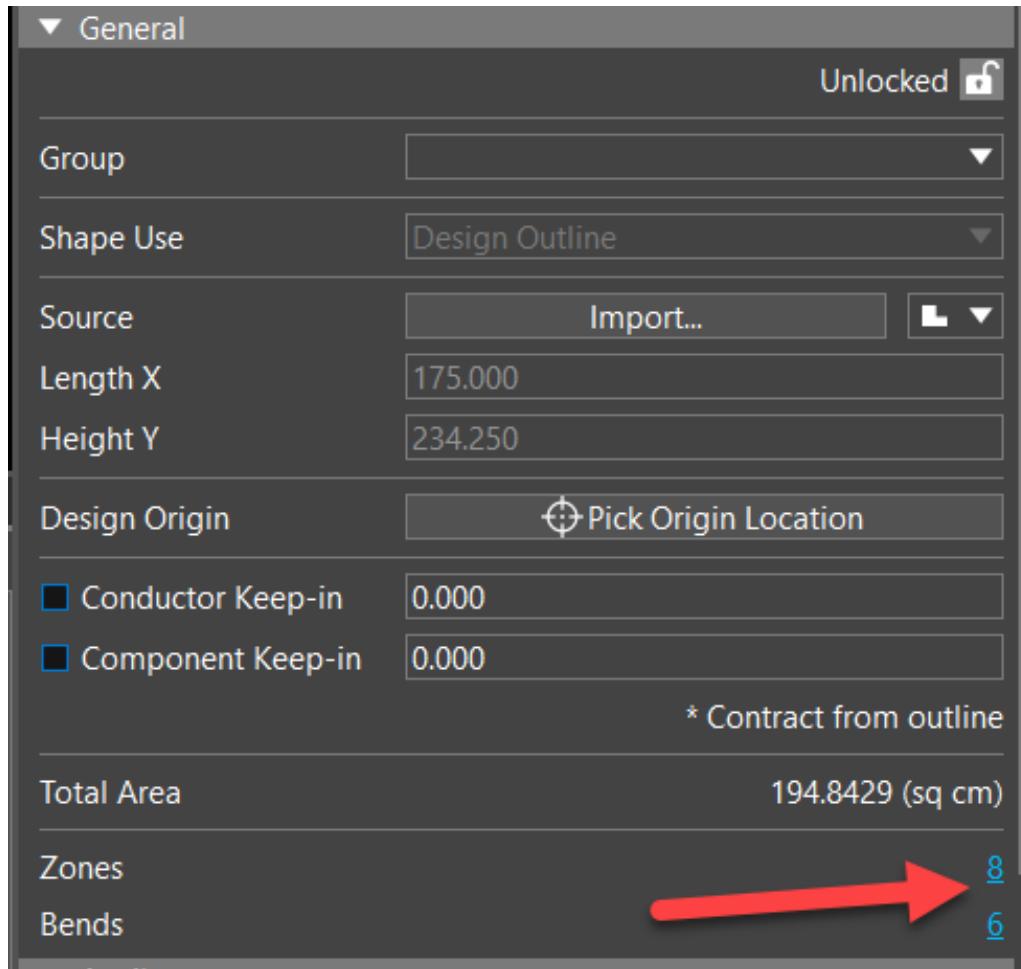
When creating a zone, the draw shape utility enables you to seamlessly switch between line, arc, and tangent arc, while showing length and angle.



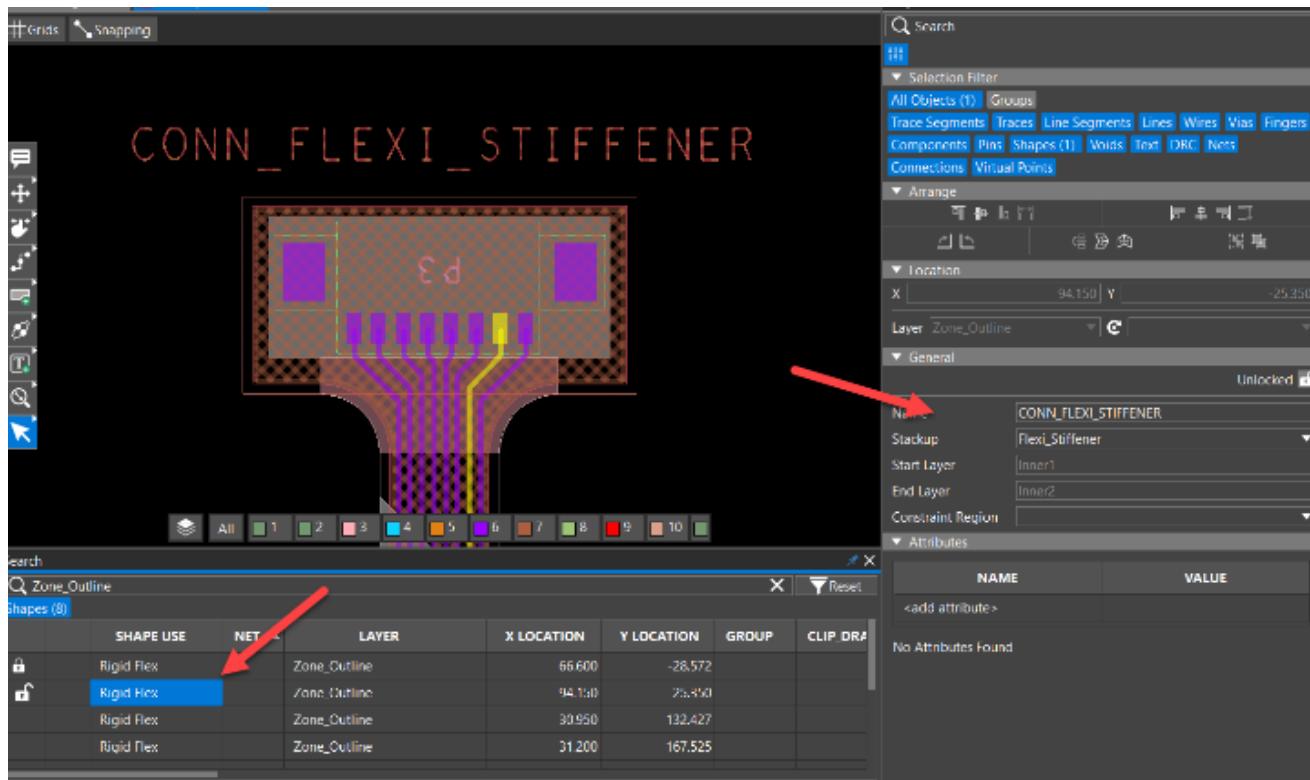
Once you draw a zone or a bend line, its properties appear in the *Properties* panel.



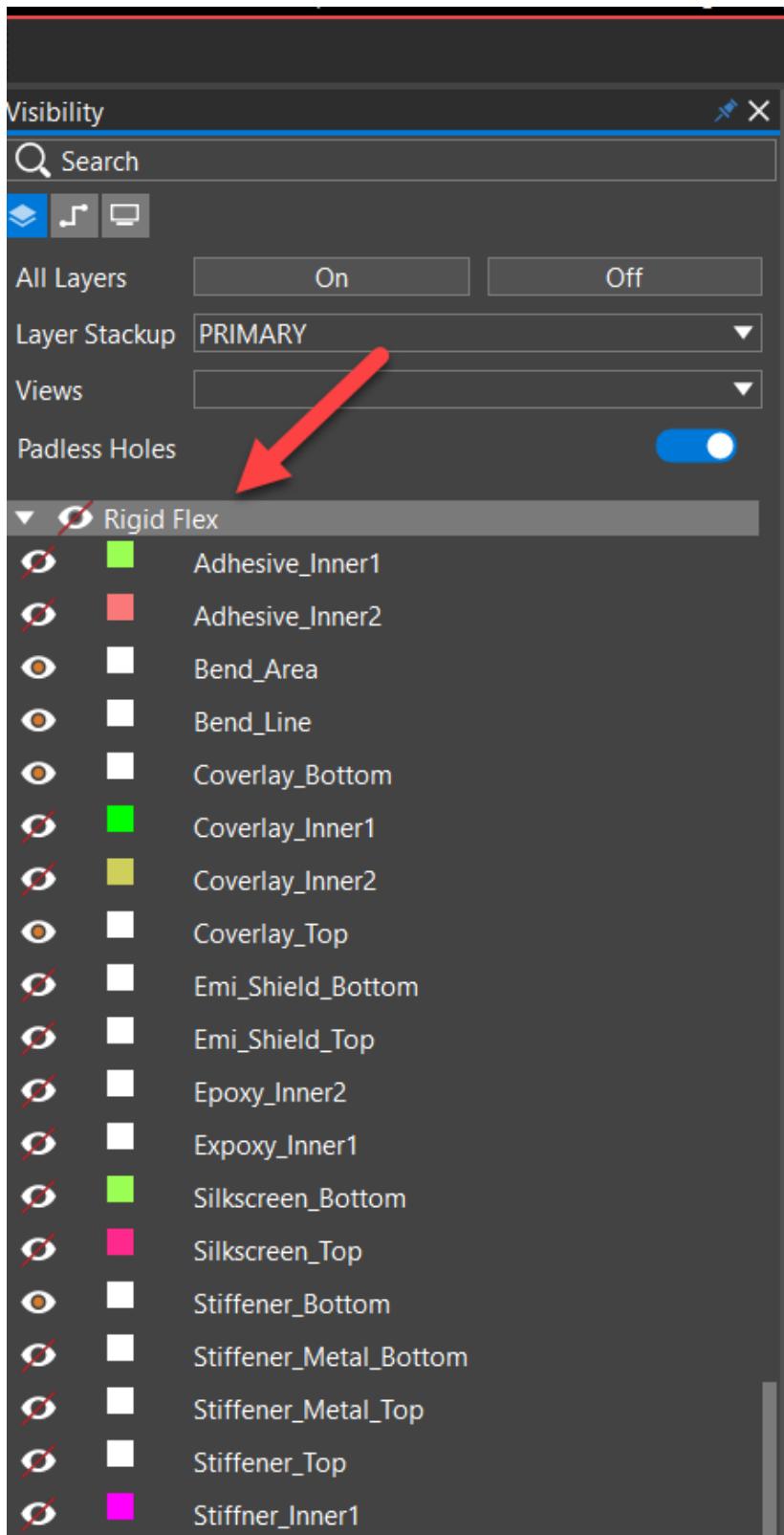
You can choose the design outline to see links showing the number of zones and bends.



Clicking the blue *Zones* and *Bends* links gives you the list of all in the search table, from which you can navigate and change properties of any zone or bend.



You can access and control visibility of flex data in the *Visibility* panel, as follows.

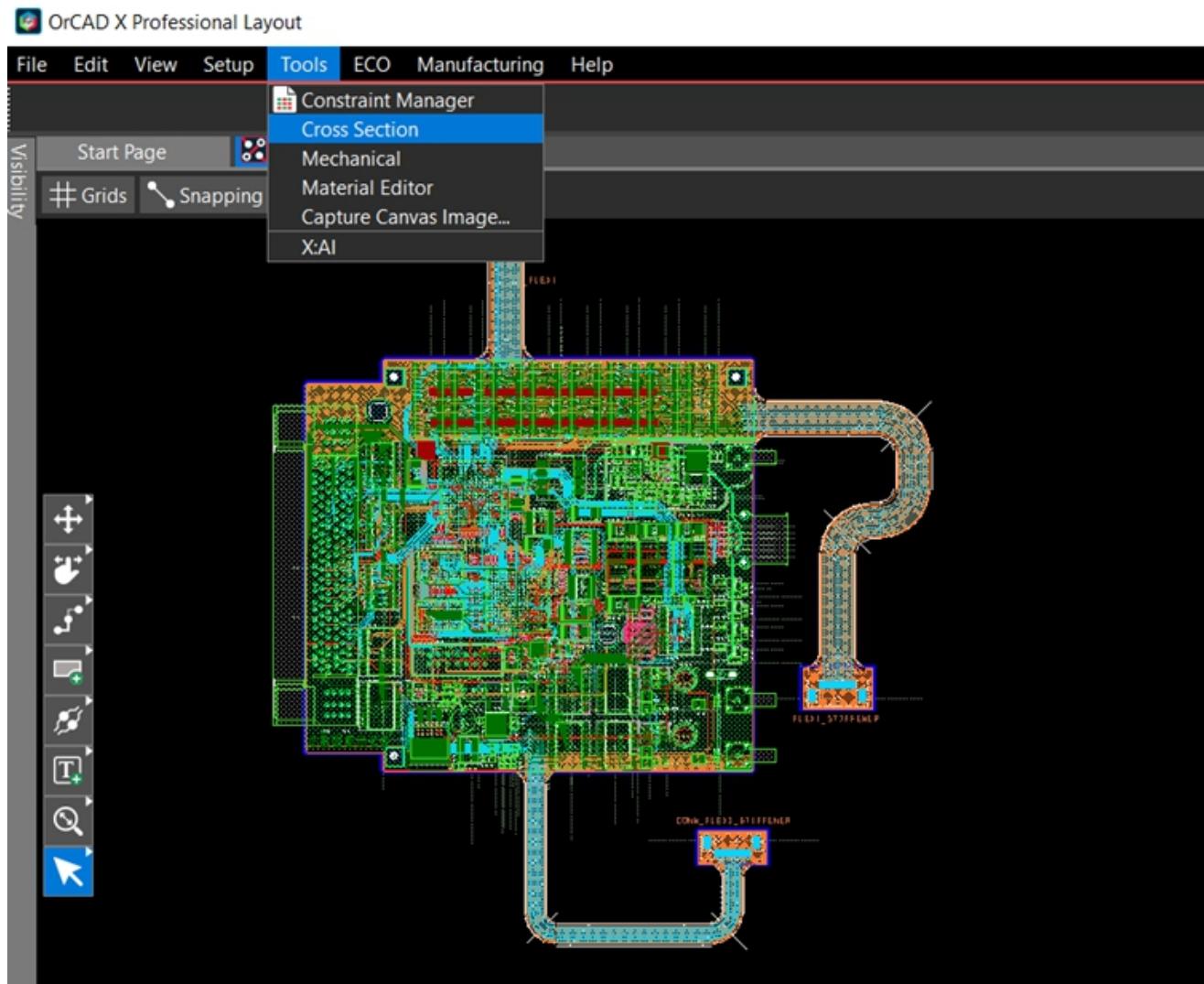


Related Topics

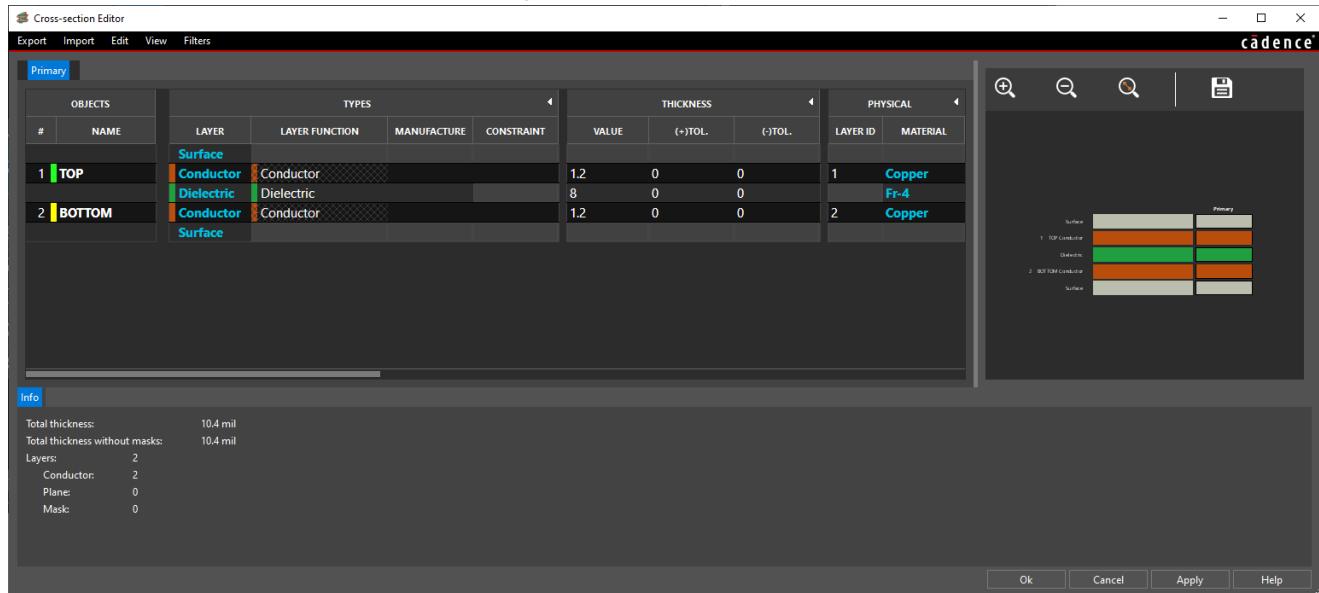
- [Adding Zone Stackups in OrCAD X Presto](#)
- [Creating and Editing Bends in OrCAD X Presto](#)
- [Creating and Editing Zones in OrCAD X Presto](#)

Adding Zone Stackups in OrCAD X Presto

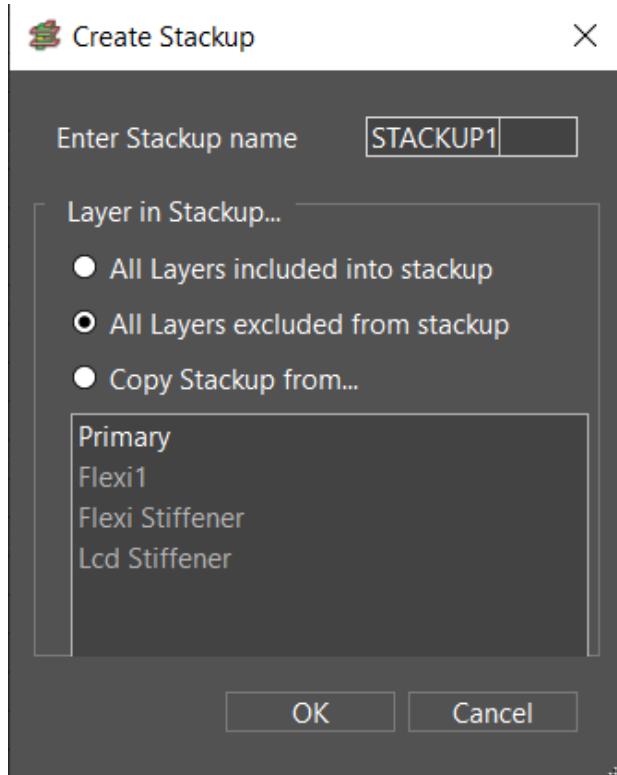
1. To launch the Cross-Section Editor, choose *Tools – Cross Section* from the top menu, as follows:



The Cross-Section Editor window opens, as follows:



- Choose *View – Multi Stackups mode*, then *Edit – Add Stackup*. The *Create Stackup* window opens, as follows:



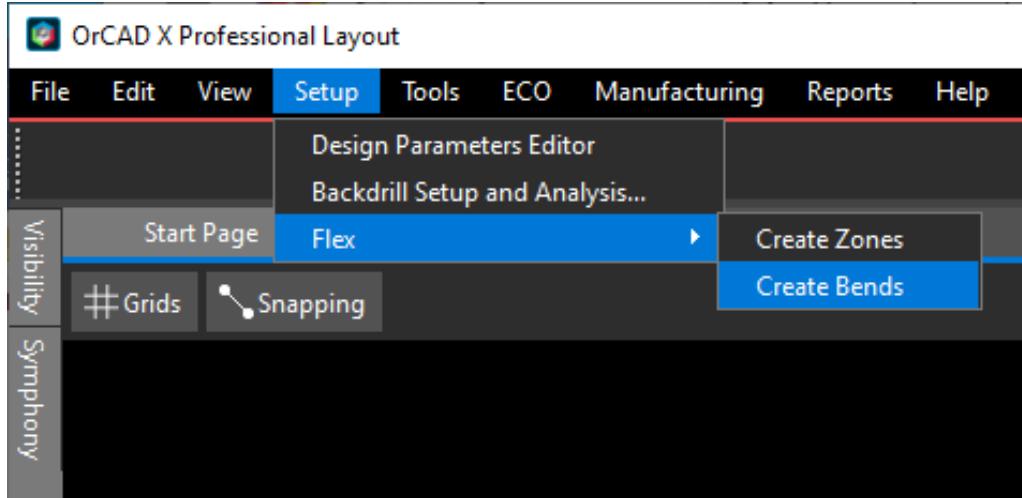
For each stackup tab, the cross-section table contains the following data—Objects, Types, Thickness, Physical, Embedded, and Signal Integrity—as follows:

OBJECTS		TYPES		THICKNESS		PHYSICAL		EMBEDDED		SIGNAL INTEGRITY		
#	NAME	LAYER	LAYER FUNCTION		VALUE	LAYER ID	MATERIAL		EMBEDDED STATUS	CONDUCTIVITY	Dielectric Constant	SI IGNORE
1 TOP	STIFFNER_INNER1	Mask	Dielectric Prepreg	0.2032			Polyimide			0	4.3	<input type="checkbox"/>
	EXPOXY_INNER1	Mask	Dielectric Adhesive	0.025			Adhesive Epoxy			0	4.6	<input type="checkbox"/>
	PASTEMASK_TOP	Mask	Solder Paste	0.0762			Solder Paste Sac			76900	1	<input type="checkbox"/>
	SILKSCREEN_TOP	Mask	Silkscreen	0.015			Epoxy Ink Uv C...			0	4.4	<input type="checkbox"/>
	SOLDERMASK_TOP	Mask	Solder Mask	0.02			Soldermask			0	3.7	<input type="checkbox"/>
	Conductor	Conductor		0.035	1	1oz Copper		Not embedded	580000	4.5	<input type="checkbox"/>	<input type="checkbox"/>
	Dielectric	Dielectric		0.2		Fr-4			0	4.5	<input type="checkbox"/>	<input type="checkbox"/>
	Plane	Plane		0.035	2	1oz Copper		Not embedded	580000	4.5	<input type="checkbox"/>	<input type="checkbox"/>
	Dielectric	Dielectric		0.2		Fr-4			0	4.5	<input type="checkbox"/>	<input type="checkbox"/>
	Dielectric	Dielectric		0.0508		Fr-4 No Flow Pr...			0	4	<input type="checkbox"/>	<input type="checkbox"/>
2 GND	Conductor	Conductor		0.035	3	1oz Copper		Not embedded	580000	4.5	<input type="checkbox"/>	<input type="checkbox"/>
	Dielectric	Dielectric		0.25		Polyimide			0	4.3	<input type="checkbox"/>	<input type="checkbox"/>
	Conductor	Conductor		0.035	4	1oz Copper		Not embedded	580000	4.5	<input type="checkbox"/>	<input type="checkbox"/>
	Dielectric	Dielectric		0.0508		Fr-4 No Flow Pr...			0	4	<input type="checkbox"/>	<input type="checkbox"/>
	Dielectric	Dielectric		0.2		Fr-4			0	4.5	<input type="checkbox"/>	<input type="checkbox"/>
3 INNER1	Plane	Plane		0.035	5	1oz Copper		Not embedded	580000	4.5	<input type="checkbox"/>	<input type="checkbox"/>
	Dielectric	Dielectric		0.2		Fr-4			0	4.5	<input type="checkbox"/>	<input type="checkbox"/>
	Conductor	Conductor		0.035		1oz Copper		Not embedded	580000	4.5	<input type="checkbox"/>	<input type="checkbox"/>
	Dielectric	Dielectric		0.0508		Fr-4 No Flow Pr...			0	4	<input type="checkbox"/>	<input type="checkbox"/>
4 INNER2	Conductor	Conductor		0.035		1oz Copper		Not embedded	580000	4.5	<input type="checkbox"/>	<input type="checkbox"/>
	Dielectric	Dielectric		0.2		Polyimide			0	4.3	<input type="checkbox"/>	<input type="checkbox"/>
	Plane	Plane		0.035		1oz Copper		Not embedded	580000	4.5	<input type="checkbox"/>	<input type="checkbox"/>
	Dielectric	Dielectric		0.2		Fr-4			0	4.5	<input type="checkbox"/>	<input type="checkbox"/>
5 POWER	Conductor	Conductor		0.035		1oz Copper		Not embedded	580000	4.5	<input type="checkbox"/>	<input type="checkbox"/>
	Dielectric	Dielectric		0.2		Polyimide			0	4.5	<input type="checkbox"/>	<input type="checkbox"/>
	Plane	Plane		0.035		1oz Copper		Not embedded	580000	4.5	<input type="checkbox"/>	<input type="checkbox"/>
	Dielectric	Dielectric		0.2		Fr-4			0	4.5	<input type="checkbox"/>	<input type="checkbox"/>
6 BOTTOM	Conductor	Conductor		0.035		1oz Copper		Not embedded	580000	4.5	<input type="checkbox"/>	<input type="checkbox"/>
	SOLDERMASK_BOTTOM	Mask	Solder Mask	0.02		Soldermask			0	3.7	<input type="checkbox"/>	<input type="checkbox"/>
	SILKSCREEN_BOTTOM	Mask	Silkscreen	0.015		Epoxy Ink Uv C...			0	4.4	<input type="checkbox"/>	<input type="checkbox"/>
	PASTEMASK_BOTTOM	Mask	Solder Paste	0.0762		Solder Paste Sac			76900	1	<input type="checkbox"/>	<input type="checkbox"/>
	Surface									1		

You can expand any column to view all sub-columns by double-clicking on the column heading.

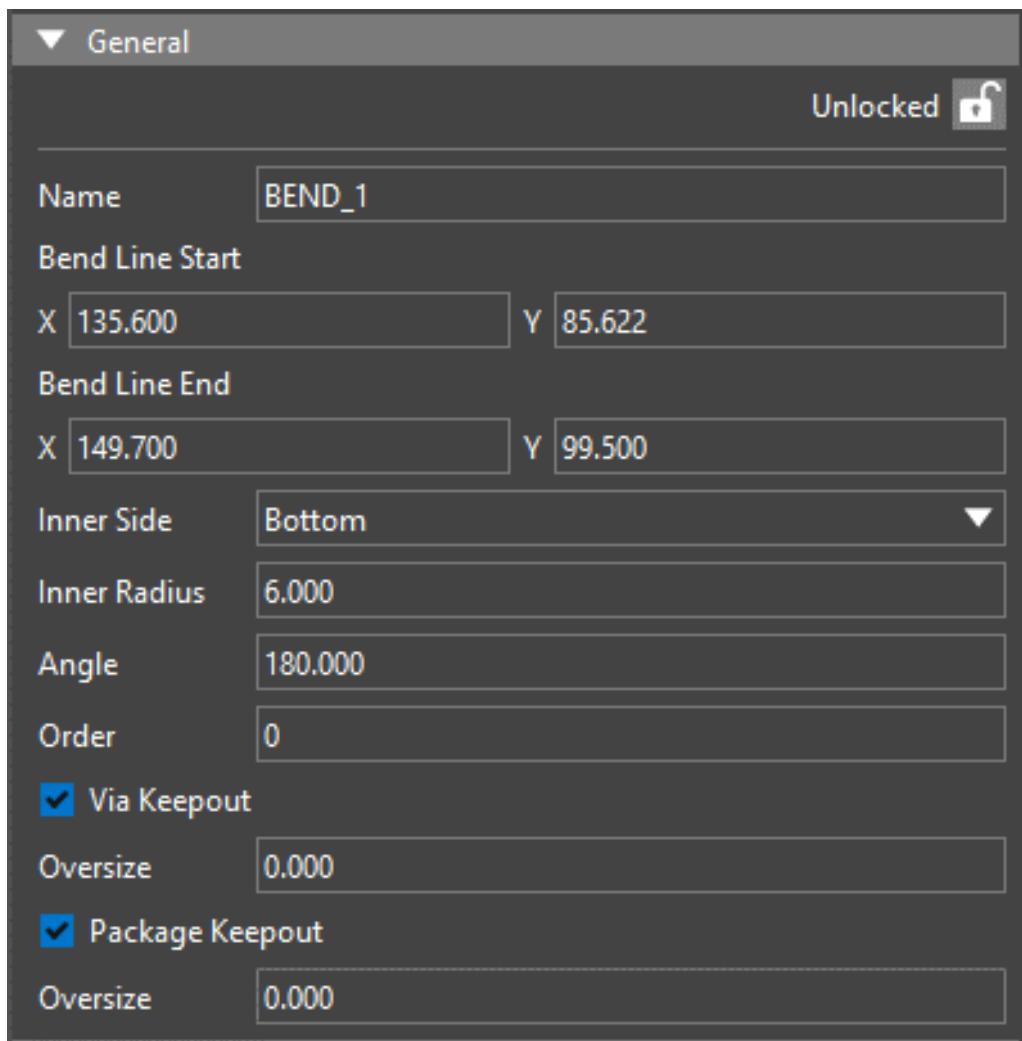
Creating and Editing Bends in OrCAD X Presto

To create bends, from the top menu, choose *Setup – Flex – Create Bends*, as follows:



You can start drawing a line segment to specify the bend line.

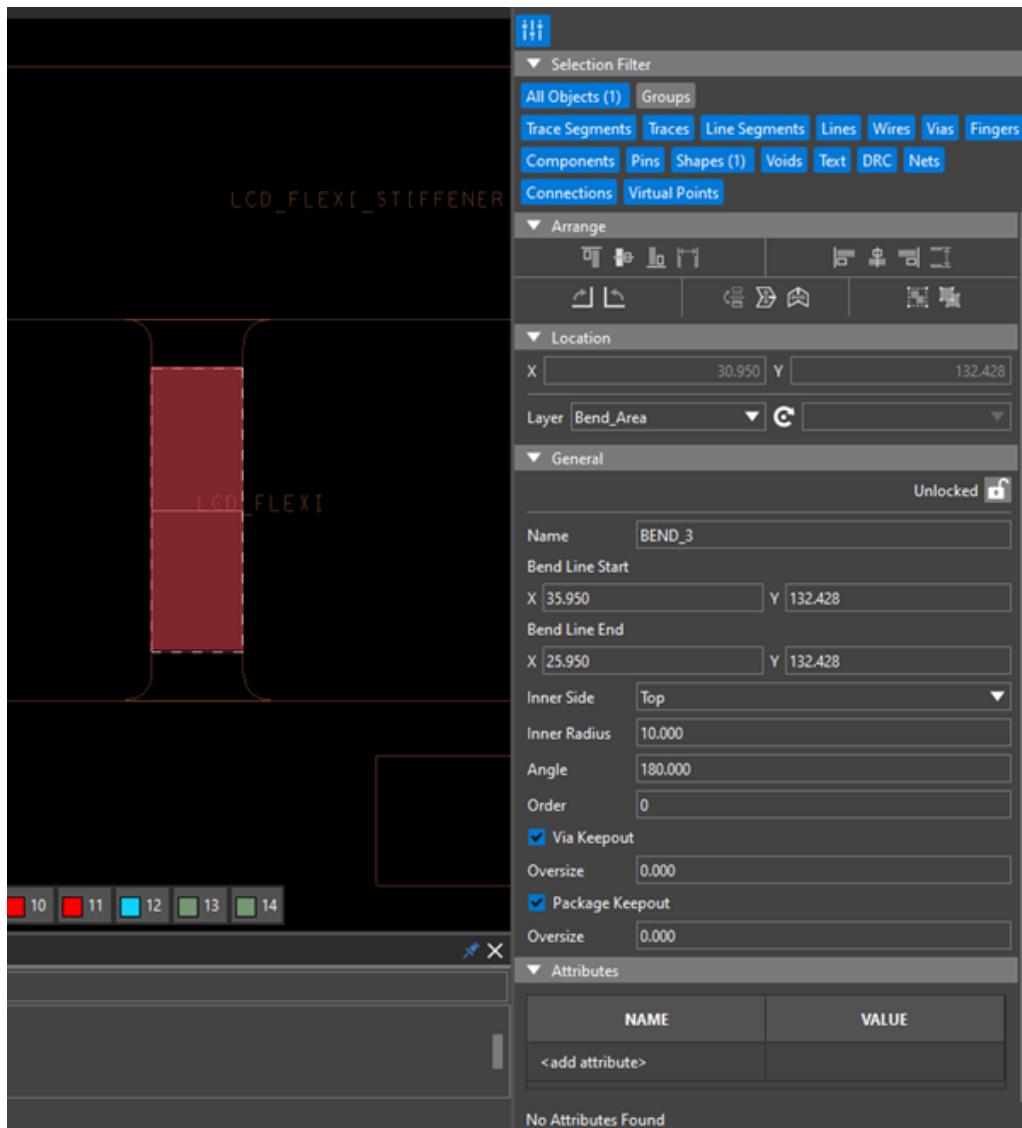
After creating a bend, its shape in the Rigid Flex/Bend Area layer is selected and the General properties pane looks similar to the following.



By default, bends are created with the `BEND_<number>` naming convention, beginning with `BEND_1`, but you can rename as you like.

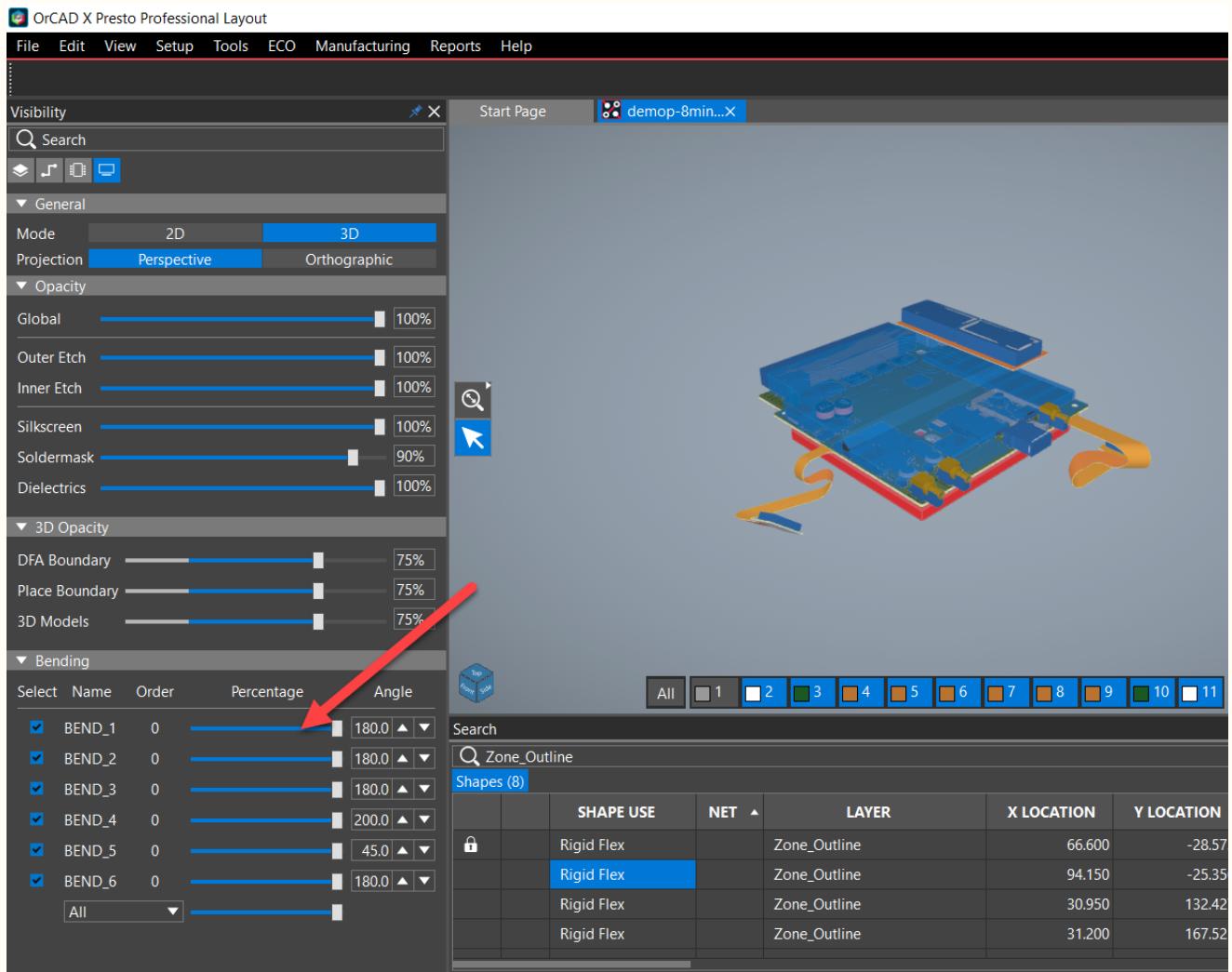
Additionally, bends are unlocked by default, but you can manually unlock or lock them at any time.

You can choose a bend by its bend area shape, bend line, or keepouts. You can edit only the bend properties and not the object properties from the property pane, as follows. When you edit the selected bend, if any of the new values makes it invalid, then the attributes reverts to the previous valid values.



You can delete a bend by choosing its bend area or bend line, and either pressing the `delete` key or right-clicking and choosing the *Delete* option. When a bend is deleted, all of its associated objects including bend line, bend area, and keepouts are deleted as well.

⚠ The 3D canvas supports bending as well.

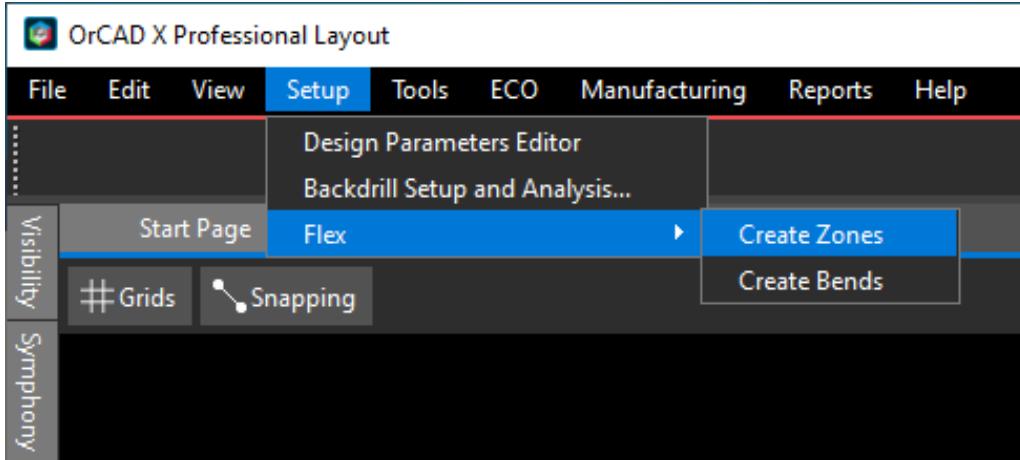


Related Topics

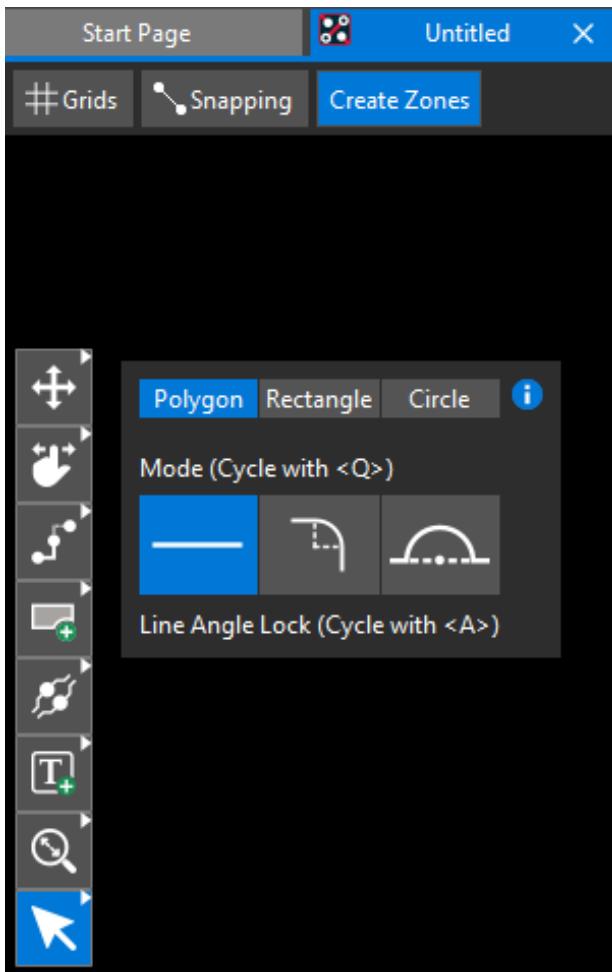
- [Creating and Editing Zones in OrCAD X Presto](#)

Creating and Editing Zones in OrCAD X Presto

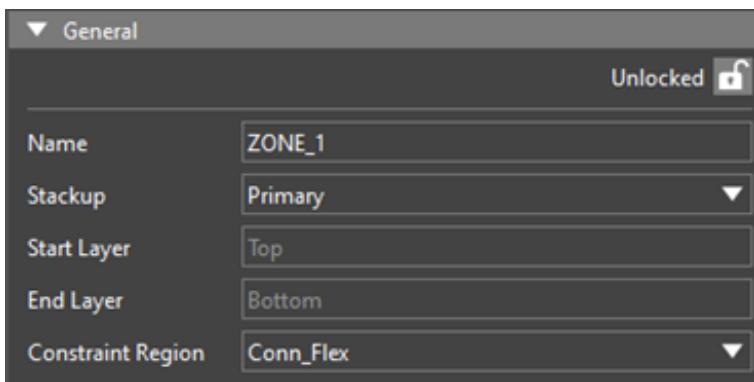
To create zones, from the top menu, choose *Setup – Flex – Create Zones*, as follows:



The tool opens the Create Zones dialog box, from which you can create polygonal, rectangular, or circular zones, based on your selection:



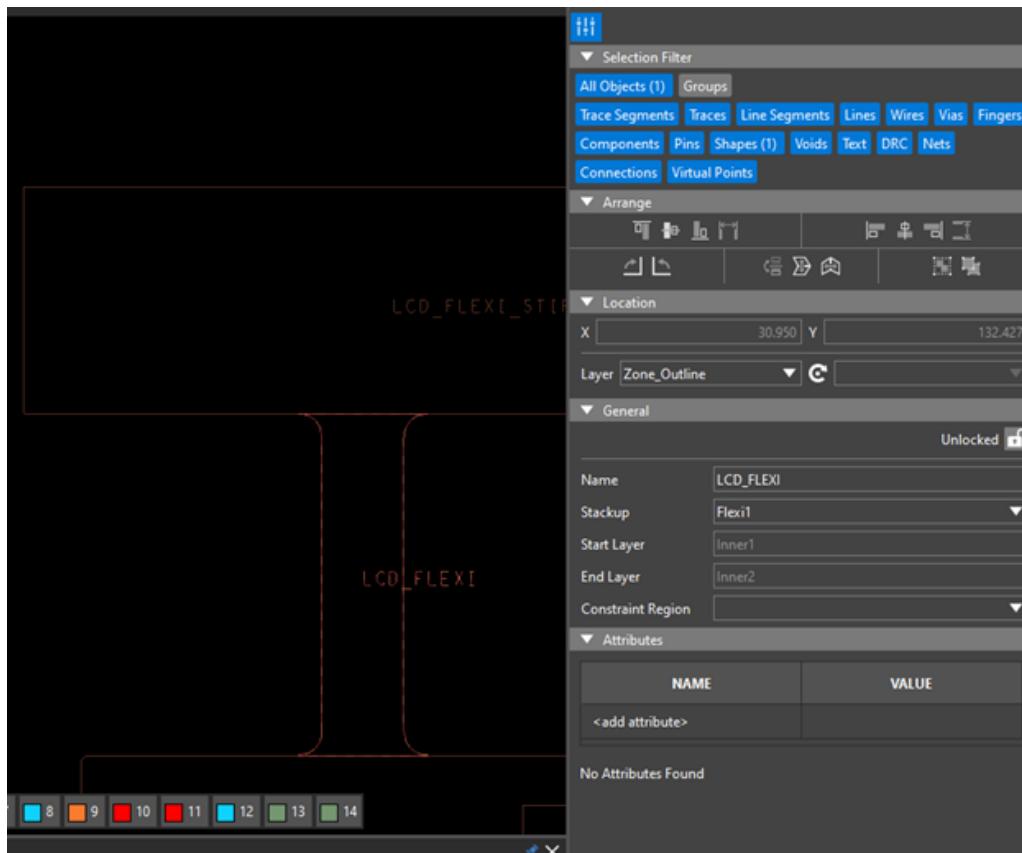
When the zone is created, its boundary shape is selected and the *General* properties pane looks similar to the following.



By default, zones are created with the `zone_<number>` naming convention, beginning with `zone_1`, but you can rename as you like.

Additionally, zones are unlocked by default, but you can manually unlock or lock them at any time.

When you choose a zone using its boundary shape, constraint region, or keepouts, the property pane shows its info and is unlocked by default, as follows. You cannot edit or delete the zone outline, constraint region, and keepouts. The constraint region and keepouts are generated automatically when selected from the dropdown in the Properties pane.



You can delete a zone by choosing its boundary shape, unlocking the zone if it is already locked, and then pressing the `delete` key or right-clicking and choosing the *Delete* option, which deletes the entire zone, including the outline, constraint region, and keepout shapes.

Related Topics

- [Creating and Editing Bends in OrCAD X Presto](#)

OrCAD X Presto User Guide

Creating Zones and Bends for Rigid-Flex Design in OrCAD X Presto--Creating and Editing Zones in
OrCAD X Presto

Routing Nets in OrCAD X Presto

OrCAD X Presto offers several routing features for interactive routing.

- [Resolving Timing Requirements with Delay Tuning](#)
- [Splitting Stacked Vias](#)
- [Using Structures for Quick Routing in OrCAD X Presto](#)

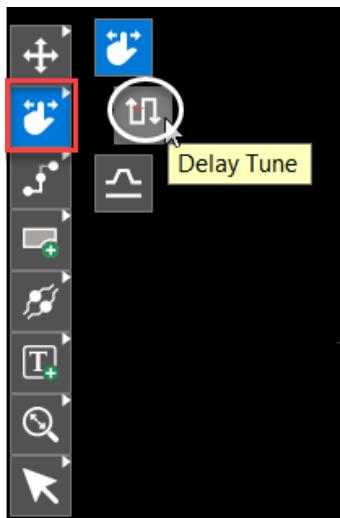
Resolving Timing Requirements with Delay Tuning

Meeting timing requirements is difficult and time-consuming in densely-populated boards. The *Delay Tune* command provides specialized tune bump patterns to match the signal lengths to resolve the timing requirements in a design.

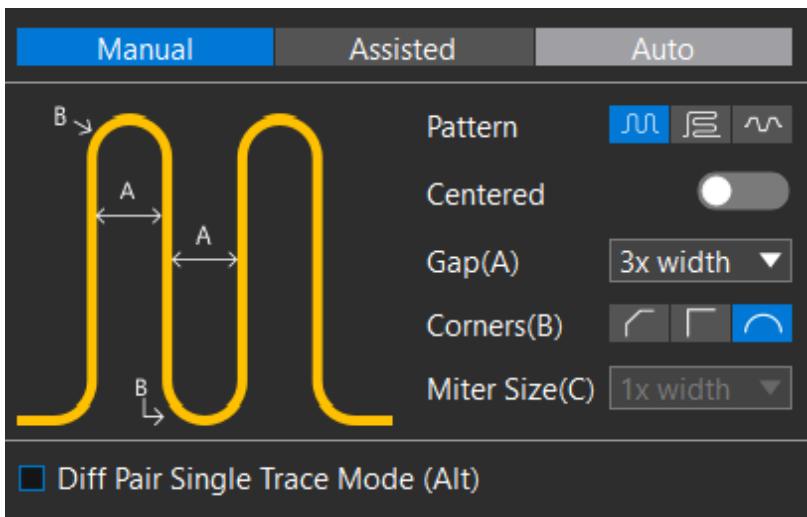
 The delay tune command is available only with the OrCAD X Presto Professional Layout license.

To add delay tune to a trace, do the following:

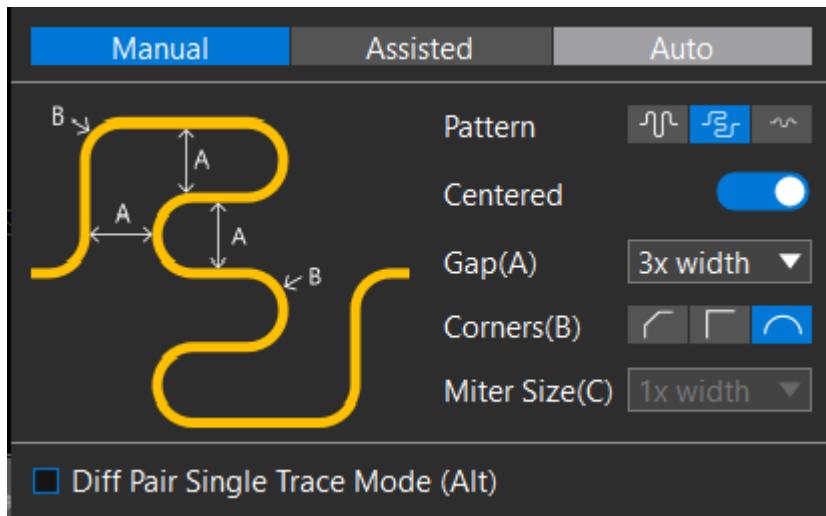
1. From the floating command toolbar, right-click the *Slide* icon and click the *Delay Tune* icon.



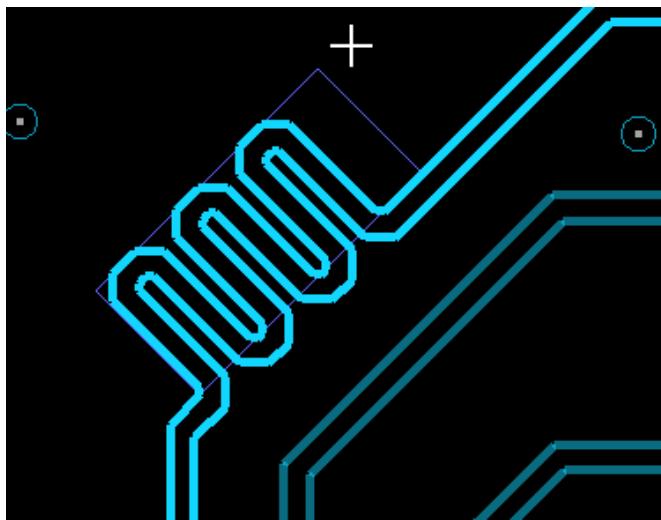
The Delay Tune widget opens with the default *Manual* mode, which displays the pattern details on the left and the bump parameter settings on the right.



2. In the Delay Tune widget, set up the tune parameters as follows:
 - a. Click to choose a bump pattern icon from Accordion, Trombone, and Sawtooth. The selected pattern is highlighted in blue.
 - b. Optionally, click to change the slider for generating centered bump patterns. The centered tune bump patterns have pattern entry and exit points at the center of the pattern.

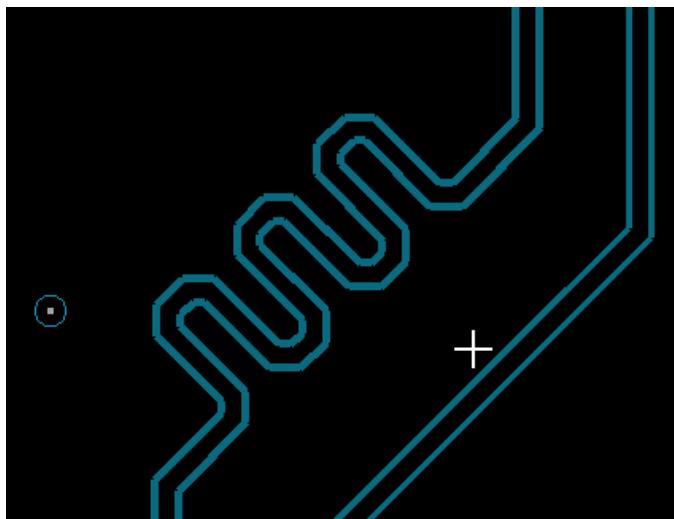


- c. Choose the tune gap parameter value from the *Gap(A)* list. It controls the inside spacing of the bump pattern.
 - d. Click to select a tune corner icon for the Accordion and Trombone patterns. The three icons represent 45 degree, 90 degree, and Full Arc. This parameter is grayed out for the Sawtooth pattern.
 - e. If you set a 45 degree corner for the chamfered Accordion and Trombone patterns, choose a value from the *Miter Size(C)* list. This parameter is grayed out for other patterns and corner settings.
 - f. Optionally, enable *Diff Pair Single Trace Mode* to tune the select segment of a differential pair trace.
3. Click to choose a trace segment in the design canvas.
4. Position the cursor to define the length, height, and direction of the pattern. A rectangle highlights the area to be used to generate the tuning bumps.



5. Click again to define the rectangular area for creating the bump pattern.

The selected tune pattern is added to the trace and the *Properties* panel displays the global status of the design.



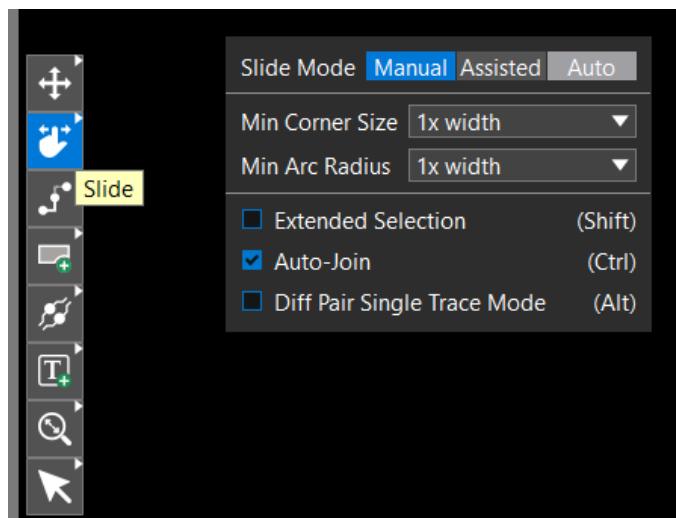
The *Properties* panel displays the location and general attributes of the selected trace segment along with the *Heads Up Display* that helps achieve the desired length. This real-time meter turns green when the desired length is met.

Splitting Stacked Vias

Stacked vias are used in place of through-hole vias by placing multiple vias on top of one another to reduce space and parasitic capacitance. A via stack is an arrangement of two or more same-net vias directly connected at the same location, with adjacent vias in the stack sharing one common layer. Sliding a via stack enables splitting the stack into a sub-stack or a single via.

To split stacked vias, do the following:

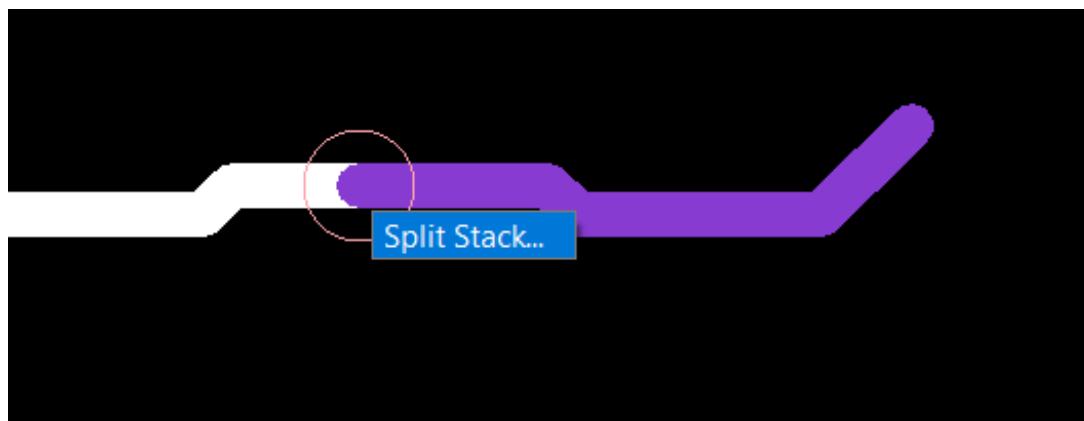
1. Click the *Slide* icon from the floating command toolbar.



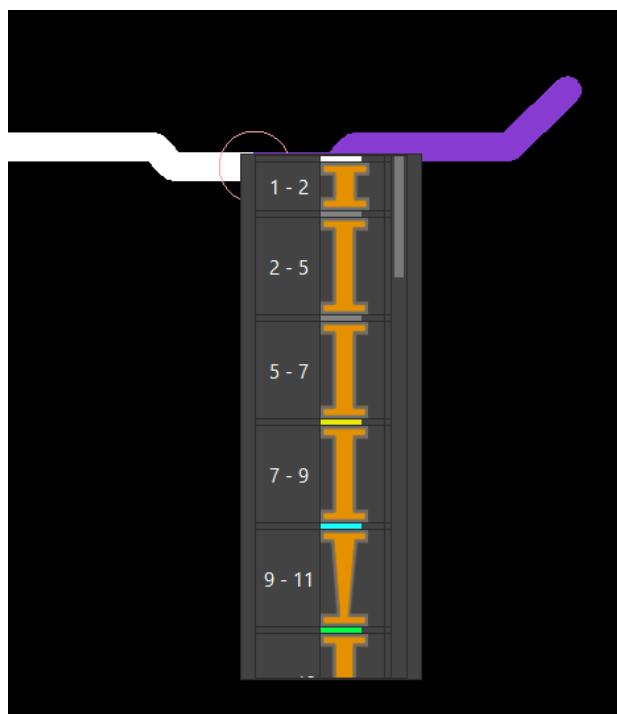
2. Select stacked vias. Ensure that vias are enabled in the Selection Filter.

Stacked vias slide as a single object that includes a group of vias and their connections, regardless of the via selected in the stack.

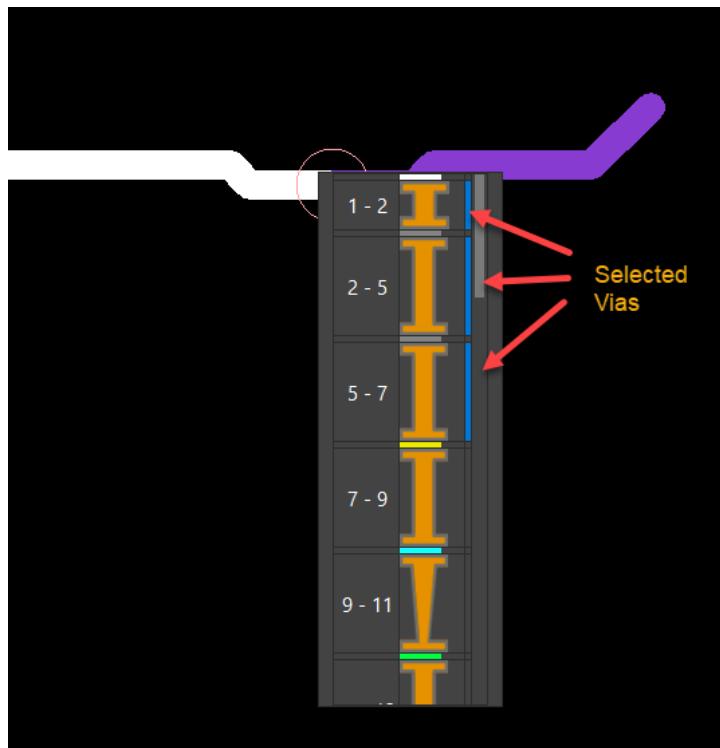
3. Right-click and choose *Split Stack*.



Via selection window is displayed showing the stacked vias and the layer numbers.



4. In the via selection window, click to the right edge to choose a single via. To choose a range of vias, hold the left mouse button and click the vias.



The selection of vias is indicated in blue. The selected via or via stack is separated from the master via stack and can be slid independently.

⚠ When sliding stacked vias outside the pin pad, a cline is added to prevent orphan vias or vias that disconnect from the net.

Using Structures for Quick Routing in OrCAD X Presto

In OrCAD X Presto, structures are reusable design element created by combining vias, clines, static shape, and route keepouts. Structures can be placed as a single entity in the same or different designs, and can be managed quickly and simply.

-  The structure command is available only with the OrCAD X Presto Professional Layout license.

Related Topics

- [Creating Via Structures in OrCAD X Presto](#)
- [Placing Via Structures in OrCAD X Presto](#)
- [Editing Via Structures in OrCAD X Presto](#)

Creating Via Structures in OrCAD X Presto

Via structures are reusable design elements that combine vias, clines, static shapes, and route keepout shapes. Via structures are placed as symbols in the same or different designs and can be managed quickly and simply.

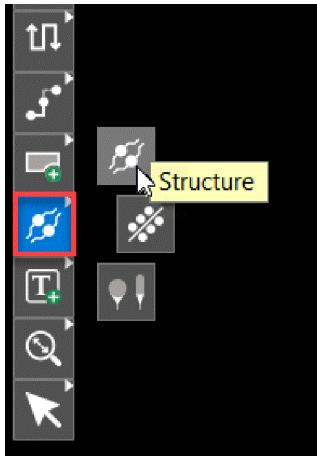
Prerequisite

- The command to create via structure is available only with the OrCAD X Professional Layout license.
- Vias used for creating the structure symbol should be placed in the design or exist in the design library.

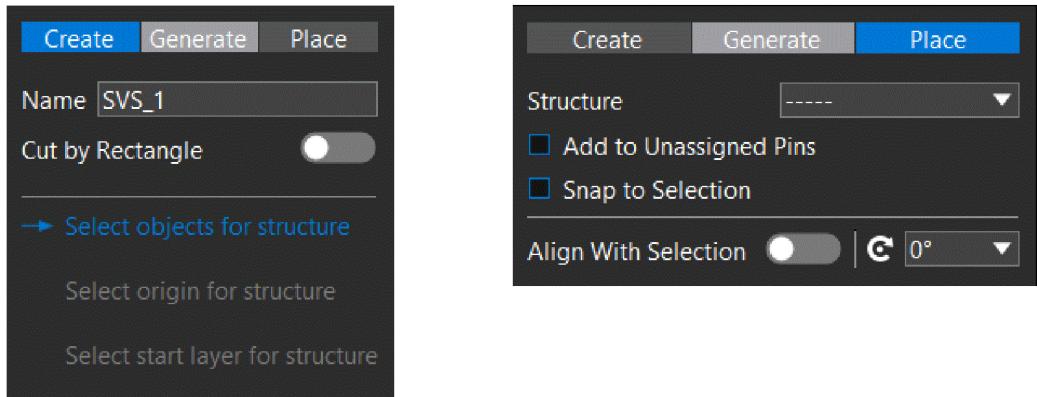
Steps

To create via structures, do the following:

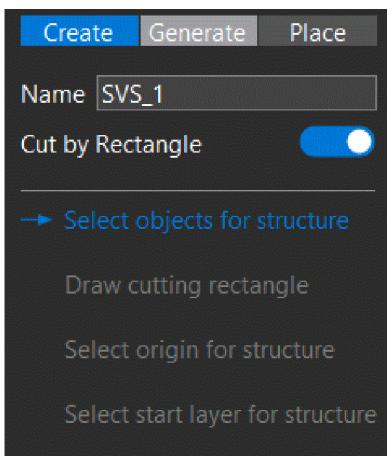
1. From the floating command toolbar, right-click the *Structure* icon and select the *Structure* icon again from the displayed icons.



A widget opens with three functional tabs: *Create*, *Generate*, and *Place*. Guiding text is provided to help with the structure creation and placement. Blue text indicates the next step, green text indicates the action is completed, and gray text shows the step is incomplete.



2. In the widget, specify a name for the structure or use the default name.
3. Toggle the *Cut by Rectangle* option to include clines and line segments enclosed within a rectangle to select the objects to be combined as a structure.
A new step, *Draw cutting rectangle*, is added to the workflow.



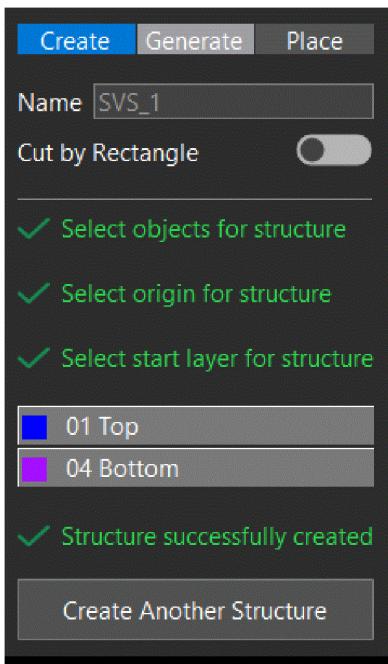
4. Ensure that the visibility of clines and vias is set on all layers.
Only visible objects are included in a structure.
5. Enable the object types in the Selection Filter.
6. Choose the elements of the structure by selecting vias and clines on the same net. To select multiple objects, click the items and then press and hold down the mouse button while moving the mouse.

 Ensure that at least one via or cline is selected.

If the selected objects are connected to a pin, the command automatically determines the origin and start layer and creates the structure.

7. Select *Cut by Rectangle* if you enabled the *Cut by Rectangle* option. Draw a rectangle by selecting the start and end points with mouse clicks.
8. If the selected objects are not connected to a pin, do the following:
 - a. Click the center of the via or dangling cline end point to choose the origin of the structure.
 - b. Select the start layer for the structure in the widget.

The structure created is saved in the design database, and the widget displays a new option, *Create Another Structure*, at the bottom.



Related Topics

- [Editing Via Structures in OrCAD X Presto](#)
- [Placing Via Structures in OrCAD X Presto](#)

Placing Via Structures in OrCAD X Presto

Via structures provides a quick way of adding and replacing the required pieces of etch in a design.

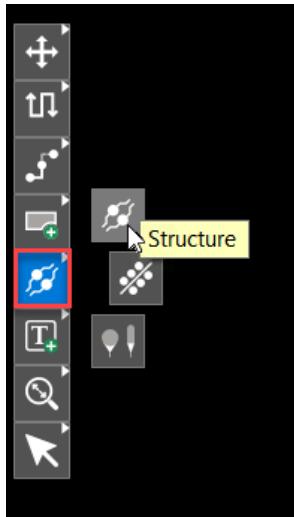
Prerequisite

The command to create via structure is available only with the OrCAD X Presto Professional Layout license.

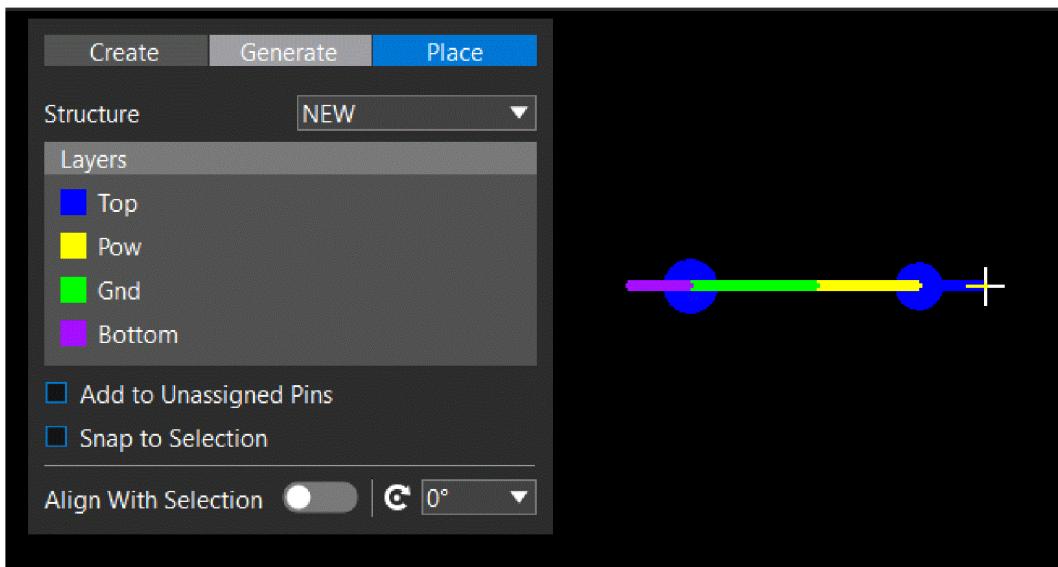
Steps

To place a via structure in a design, do the following:

1. From the floating command toolbar, right-click the *Structure* icon and select the *Structure* icon again from the displayed icons.



2. Click the *Place* tab in the Structure widget.
 3. Select a structure from the *Structure* list. The list displays all the structures saved with the design database.
- The selected structure attaches to the cursor at the structure's origin point. The widget starts displaying names and colors of the layers the structure exists on from the *Visibility* panel.



4. Optionally, to allow the structure to be attached to the elements that are not assigned to a net, enable the *Add to Unassigned Pins* check box.
5. Specify an angle to rotate the structure or toggle the *Align With Selection* option to align the structure with a pin.
6. Click to choose a pin to attach the structure. To attach a structure to multiple pins, press and hold down the mouse button while moving the mouse to create a window.
7. Keep selecting pins in the design canvas to place the structure.
8. Optionally, to snap the structure only to pins, vias, or clines, enable the *Snap to Selection* check box. By default, this option is disabled and you can place the structure anywhere in the design.

Related Topics

- [Creating Via Structures](#)
- [Editing Via Structures](#)

Editing Via Structures in OrCAD X Presto

Via structures are placed as symbols in a layout and can be modified during the course of the design cycle. A via structure can be modified in multiple ways.

Prerequisite

The command to create via structure is available only with the OrCAD X Presto Professional Layout license.

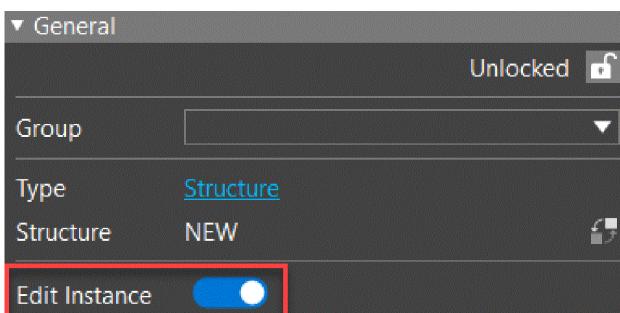
Steps

The following methods can be used to edit via structures in a design.

Editing Via Structure Instance

To edit a specific instance of a via structure, follow these steps:

1. Select the symbol from the *Search* panel or in the design canvas.
2. In the *Properties* panel, toggle the *Edit Instance* option.
This action unlocks the structure symbol and enables it for editing.



3. Modify the structure definition by adding, deleting, or sliding the trace segments.

Redefining a Via Structure

You can update the definition of all the placed instances to match the selected via structure using the *Redefine* command, as follows:

1. Select a structure symbol.

2. To modify the structure definition, toggle the *Edit Instance* option in the *Properties* panel.
3. Apply interactive commands to any of the objects that belong to a structure. For example, slide, move, rotate, mirror, and delete.
4. Right-click a structure symbol and choose *Redefine*.
5. Click *Yes* in the confirmation dialog box to refresh all instances to match the new definition.

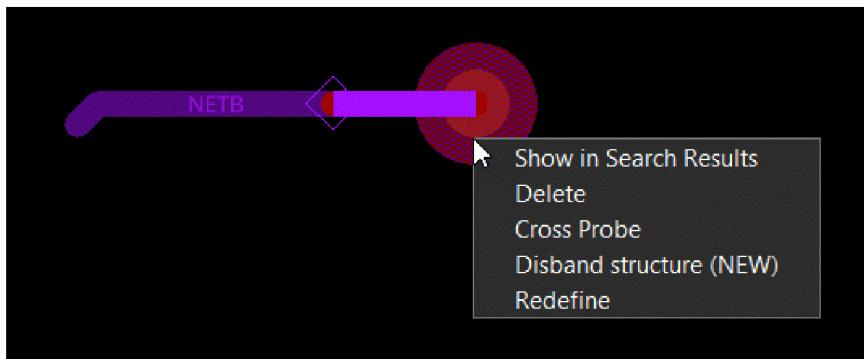
Disbanding a Via Structure

A via structure can be broken into its individual objects.

- To disband a structure, right-click a structure symbol and choose *Disband structure*.
The structure is converted into individual clines, vias, and shapes.

Deleting a Via Structure

- To delete a via structure, right-click and choose *Delete*.



Related Topics

- [Creating Via Structures](#)
- [Placing Via Structures](#)

OrCAD X Presto User Guide

Routing Nets in OrCAD X Presto--Using Structures for Quick Routing in OrCAD X Presto

Documenting PCB with LiveDoc in OrCAD X Presto

A wide variety of documents are used to capture the requirements for fabrication assembly and testing of a PCB, including fabrication drawing, dimensioning, drill data, stack-up data, fabrication notes, assembly drawing, BOM drawing, artwork, and so on.

 These documents follow multiple industry standards but are prone to human error during creation and interpretation.

- [What is LiveDoc?](#)
- [Launching LiveDoc](#)
- [LiveDoc User Interface](#)
- [LiveDoc Preferences](#)
- [Using LiveDoc Templates](#)
- [LiveDoc Workflow](#)
- [Adding Page in LiveDoc](#)
- [Defining Page Title Block](#)
- [PCB Views in LiveDoc](#)
- [Placing PCB Views in LiveDoc](#)
- [Viewing and Sharing PCB Views in LiveDoc](#)
- [Defining Dimensions in LiveDoc](#)

- [Defining Annotations in LiveDoc](#)
- [Managing LiveDoc](#)

What is LiveDoc?

LiveDoc is a PCB documentation-authoring tool, integrated with OrCAD X Presto, that automates the documentation creation process to produce complex PCB documentation in a fraction of the time it takes using the traditional methods. As a result, it reduces the effort of manually creating and maintaining multiple documents and eliminates the scope of human error.

Utilizing the board design data, LiveDoc creates linked PCB views, drawing details, document notes, drill charts, and other crucial documentation details. The result is documentation that more accurately articulates instructions for the successful fabrication, assembly, and inspection of PCBs.

The LiveDoc documentation environment enables designers to convey manufacturing, assembly, and testing requirements to fabrication and assembly partners quickly, simply, and automatically. The unified environment ensures that the information is up-to-date, accurate, and ready to share.

LiveDoc is a Windows-based application.

Key Features of LiveDoc

LiveDoc offers all drafting and dimensioning functionalities to automate the creation of fabrication drawings, assembly drawings, and manufacturing files. Some of the key features of LiveDoc include:

- Derives the data from the active board design file to create drawings. Design changes made in OrCAD X Presto are automatically propagated to all the affected drawing elements, reducing time spent on updating the documentation due to the design changes. LiveDoc reads and writes data to and from the primary board file and saves it with the board database.
- Provides tight integration with OrCAD X Presto that enables simple click-through passing of design data to expedite the drawing creation process.
- It is an interactive documentation editor driven by *Place*, *Properties*, and *Visibility* panels to create manufacturing drawings.
- Displays PCB views from a default template with predefined grid settings. The PCB views are

scaled-down replicas of the same information in the primary board file.

- Automatically generates component assembly views, drill pattern views, details, and charts. It also generates drawings from customized templates. These templates are created based on the requirements, which can also be exported to other designs. Any changes to the board file are dynamically propagated to the LiveDoc views, ensuring consistency.
- A single file can be exported in a PDF format for sharing and reviewing using a web browser.
- Quickly document PCB requirements with dimensioning, standard fabrication, and assembly notes.

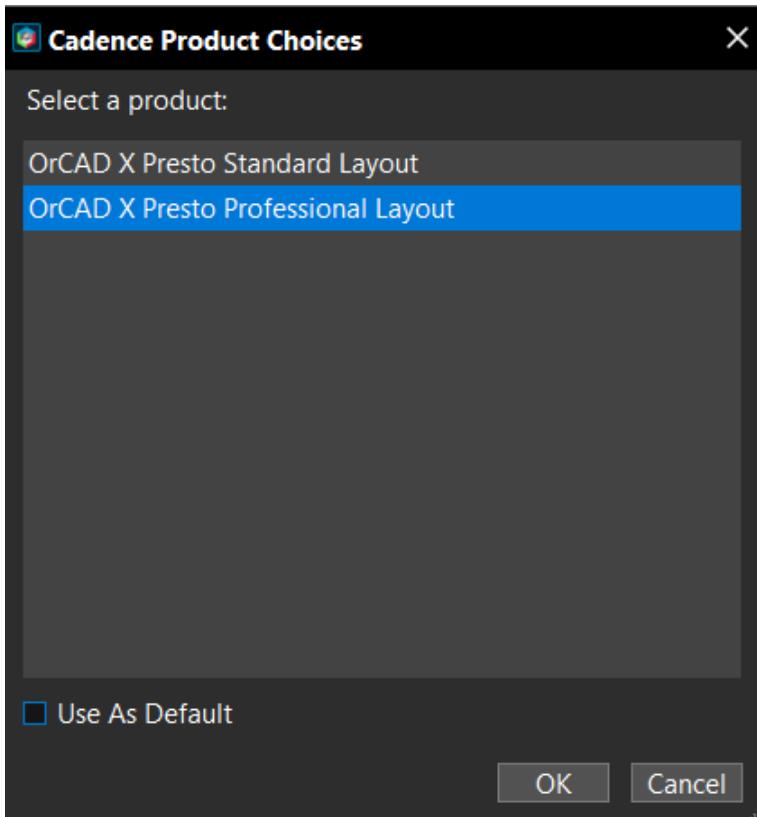
Related Topics

- [Launching LiveDoc](#)
- [LiveDoc User Interface](#)
- [LiveDoc Preferences](#)
- [Using LiveDoc Templates](#)
- [LiveDoc Workflow](#)
- [Adding a Page in LiveDoc](#)
- [Defining Page Title Blocks](#)
- [PCB Views in LiveDoc](#)
- [Placing PCB Views in LiveDoc](#)
- [Viewing and Sharing PCB Views in LiveDoc](#)
- [Defining Dimensions in LiveDoc](#)
- [Defining Annotations in LiveDoc](#)
- [Managing LiveDoc](#)

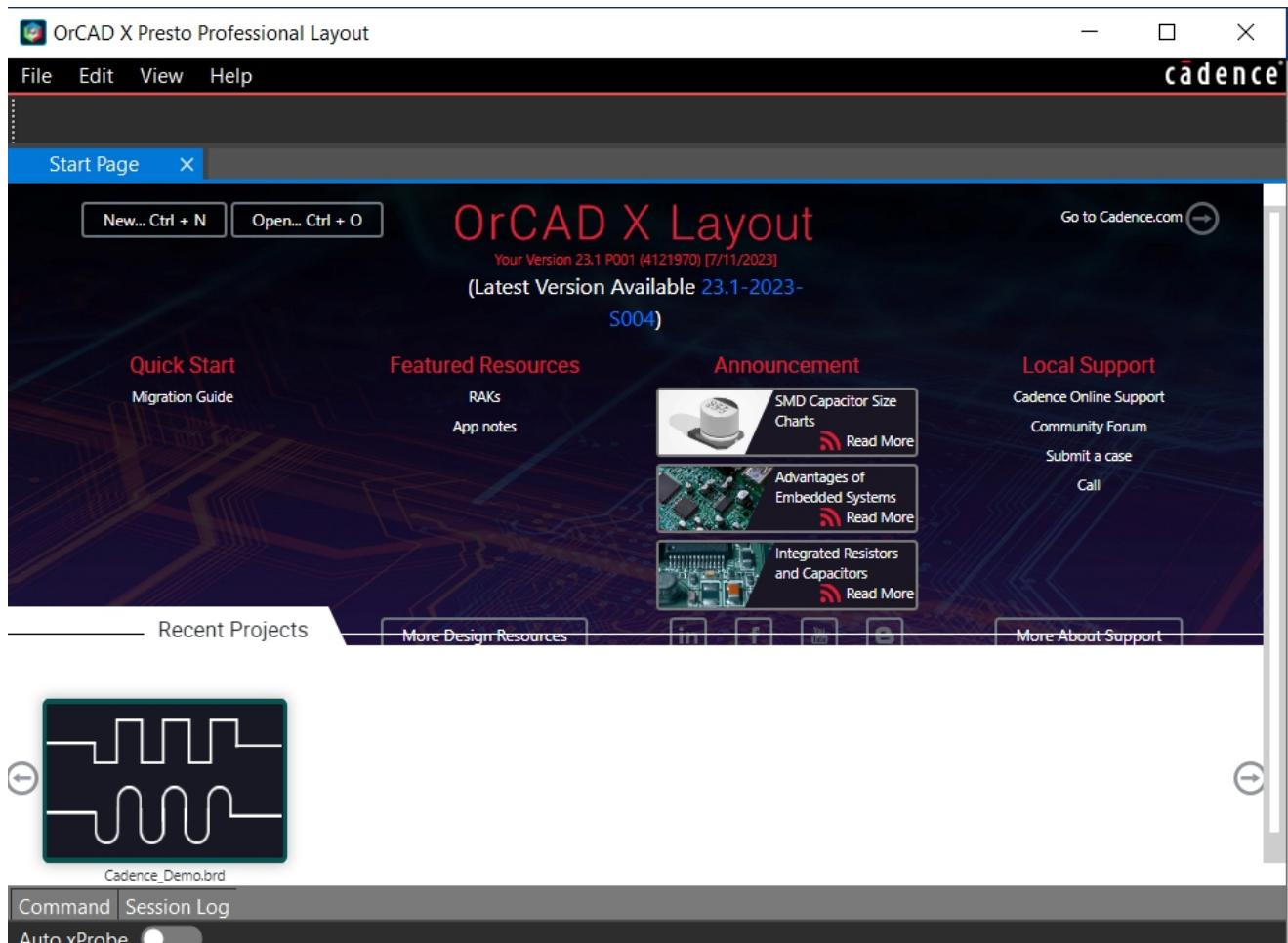
Launching LiveDoc

Start LiveDoc as follows:

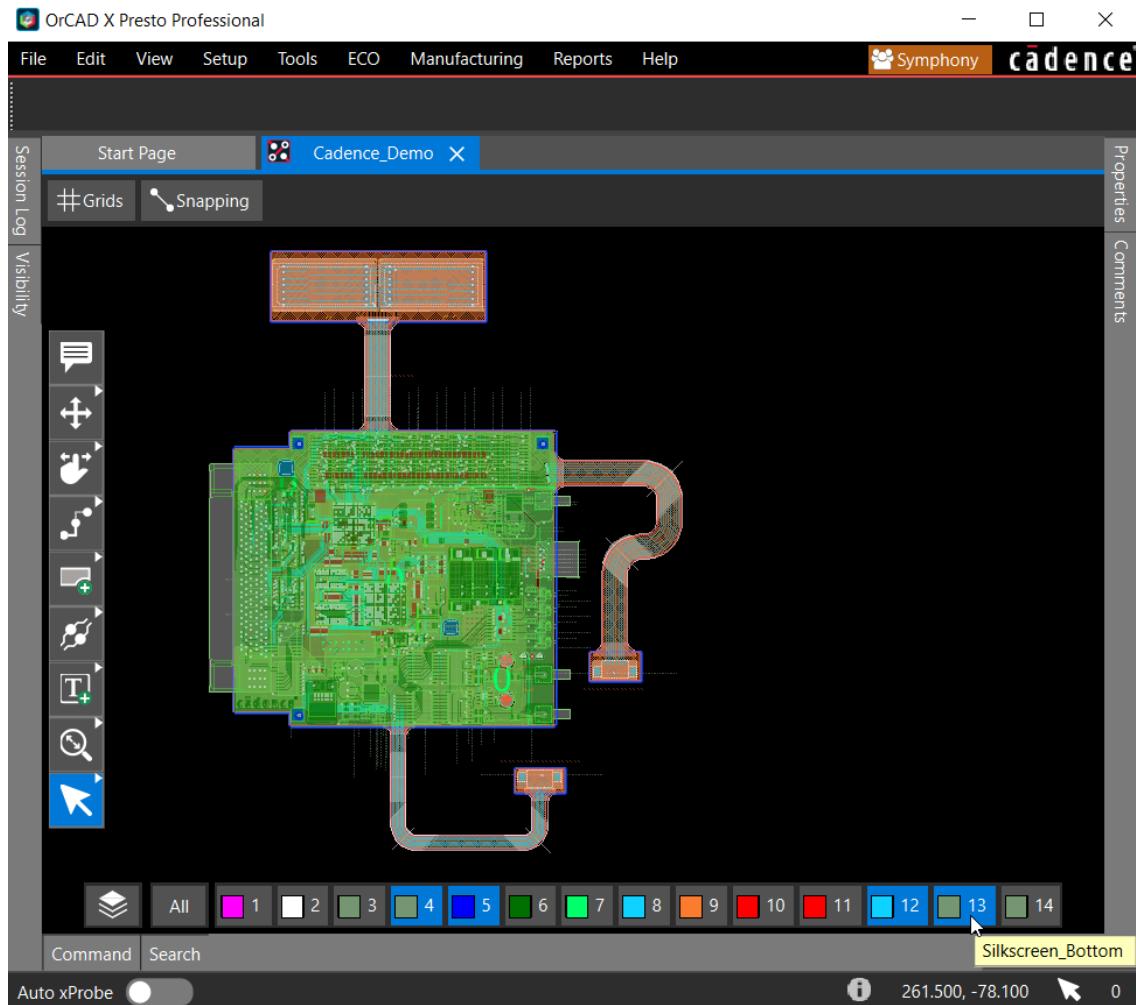
1. Open OrCAD X Presto using one of the following methods:
 - a. Choose *Start – Cadence PCB 2023 – OrCAD X Presto 2023 on Windows*. Alternatively, type `orcadx` & at the Windows command prompt and press `Enter`. The *Cadence Product Choices* dialog box is displayed.



2. Choose a product license and click *OK*.
OrCAD X Layout is launched, and the *Start Page* is displayed.



3. Choose *File – Open* and choose a board design (.brd) file.
The selected board design is loaded in a new tab.

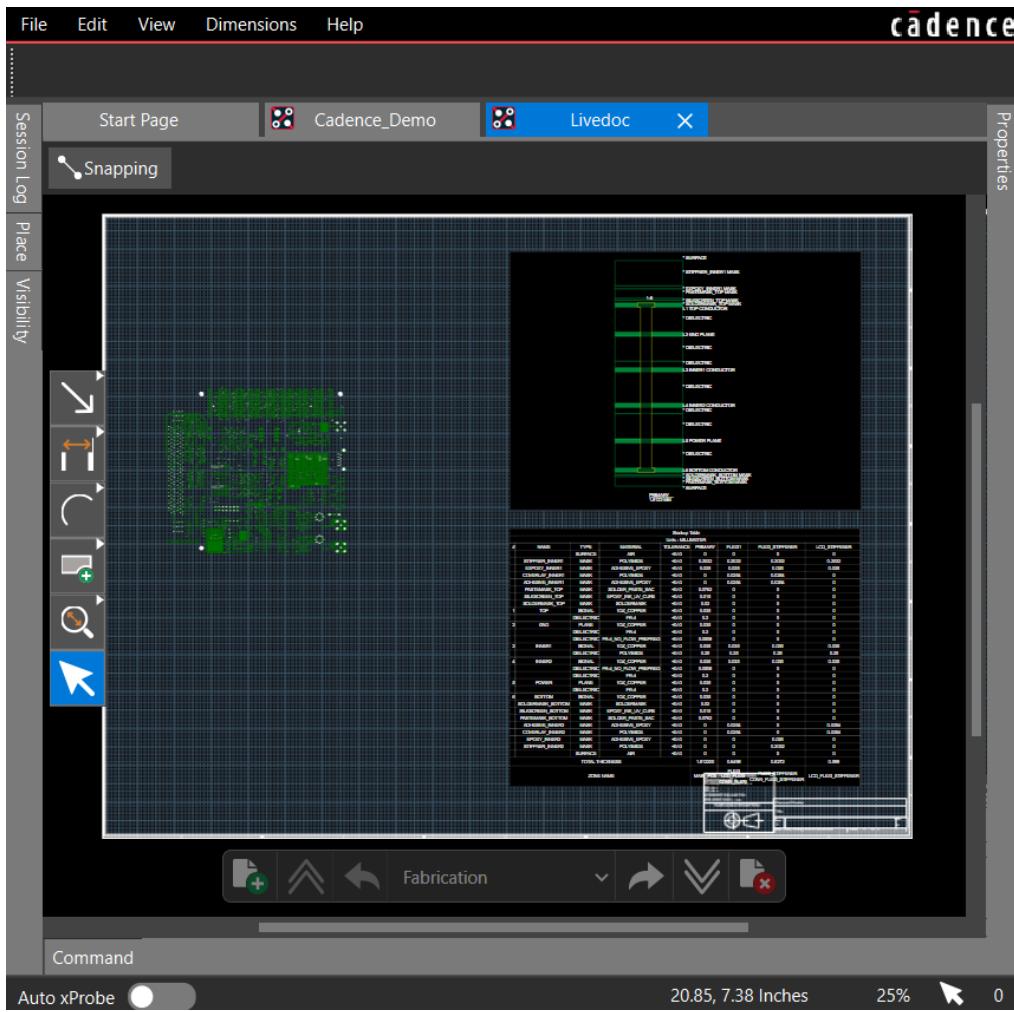


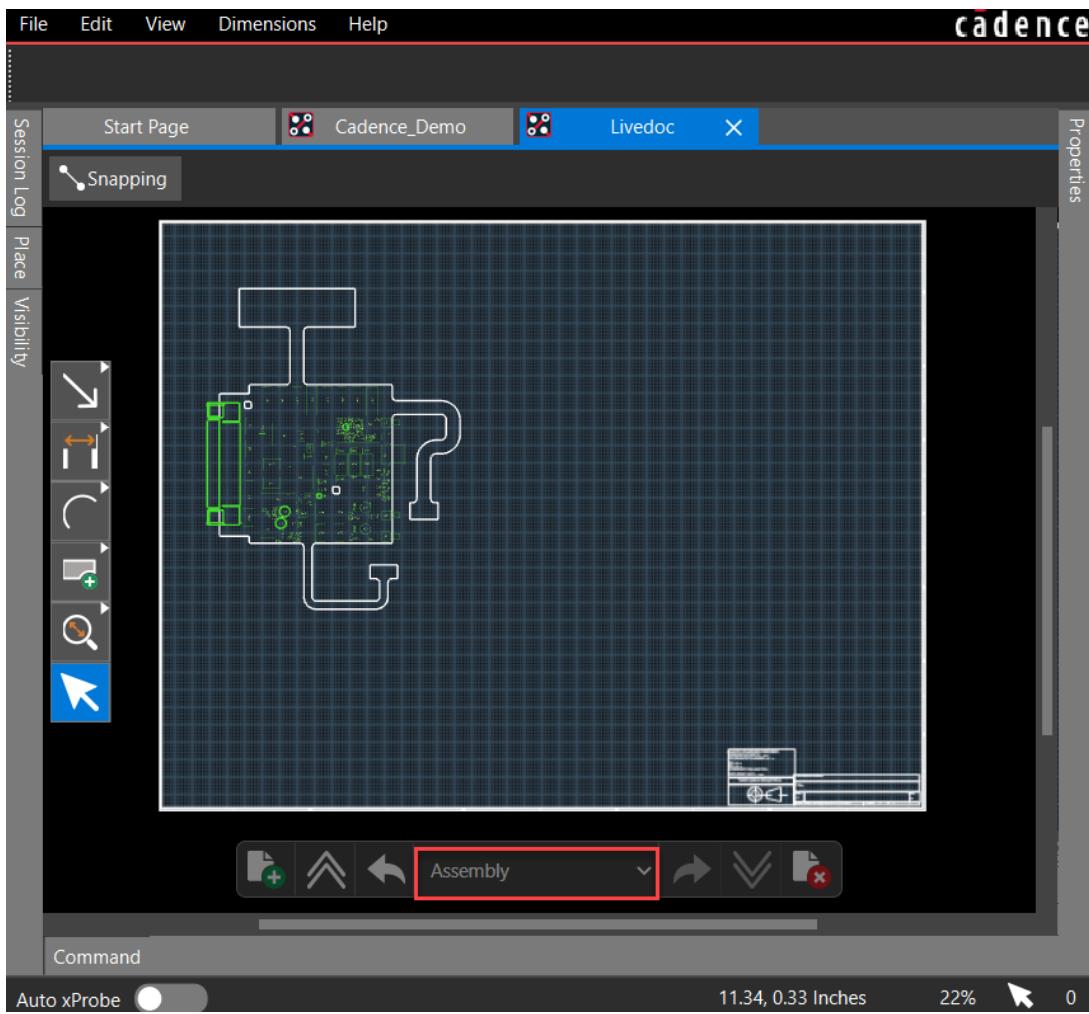
4. Choose *Manufacturing – LiveDoc – Create New From Template*.



A bar shows while LiveDoc is uploading. After the upload, the LiveDoc view of the current board design is opened in a separate tab. Fabrication and assembly pages are loaded automatically for an active design.

OrCAD X Presto User Guide





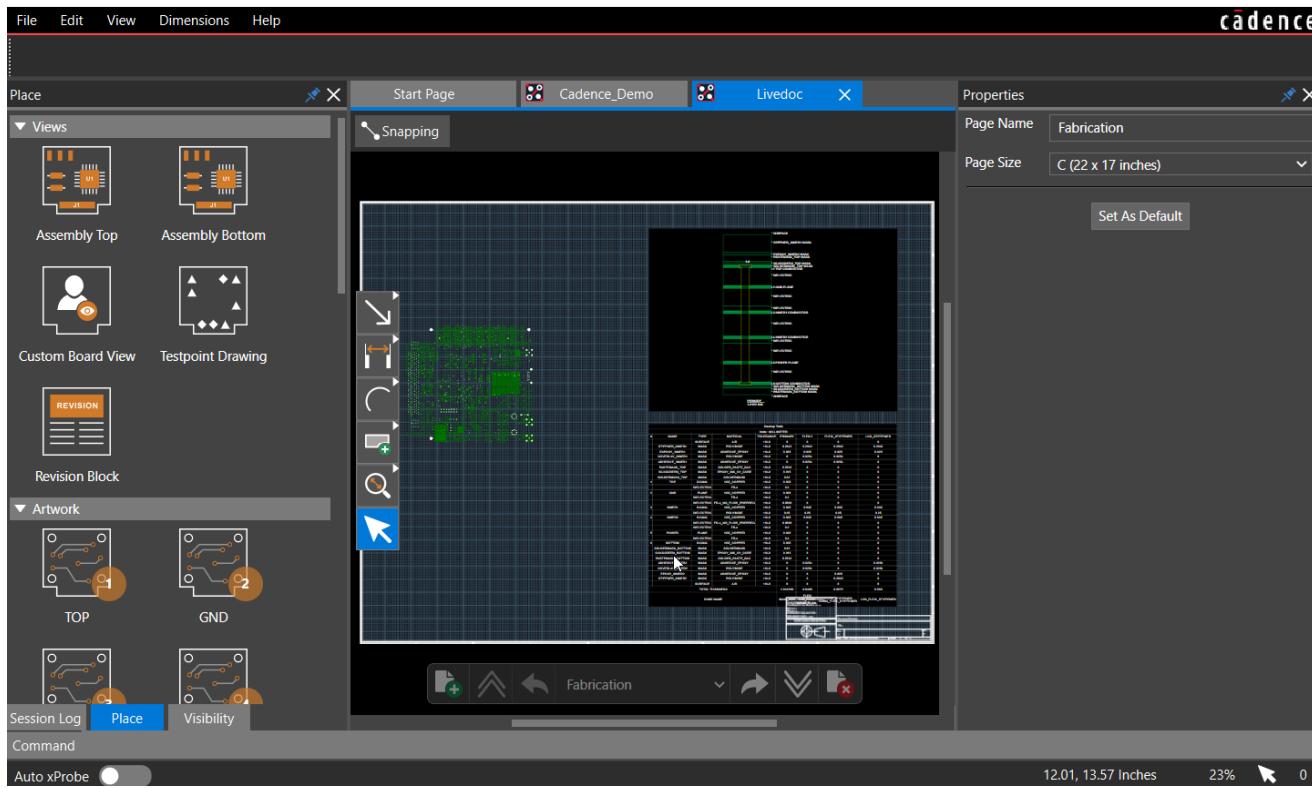
When you close the LiveDoc tab, the LiveDoc data is saved with the design database. It is accessible when you reopen the design.

Related Topics

- [What is LiveDoc?](#)
- [LiveDoc User Interface](#)
- [LiveDoc Preferences](#)
- [Using LiveDoc Templates](#)
- [LiveDoc Workflow](#)
- [Adding a Page in LiveDoc](#)
- [Defining Page Title Blocks](#)
- [PCB Views in LiveDoc](#)
- [Placing PCB Views in LiveDoc](#)
- [Viewing and Sharing PCB Views in LiveDoc](#)
- [Defining Dimensions in LiveDoc](#)
- [Defining Annotations in LiveDoc](#)
- [Managing LiveDoc](#)

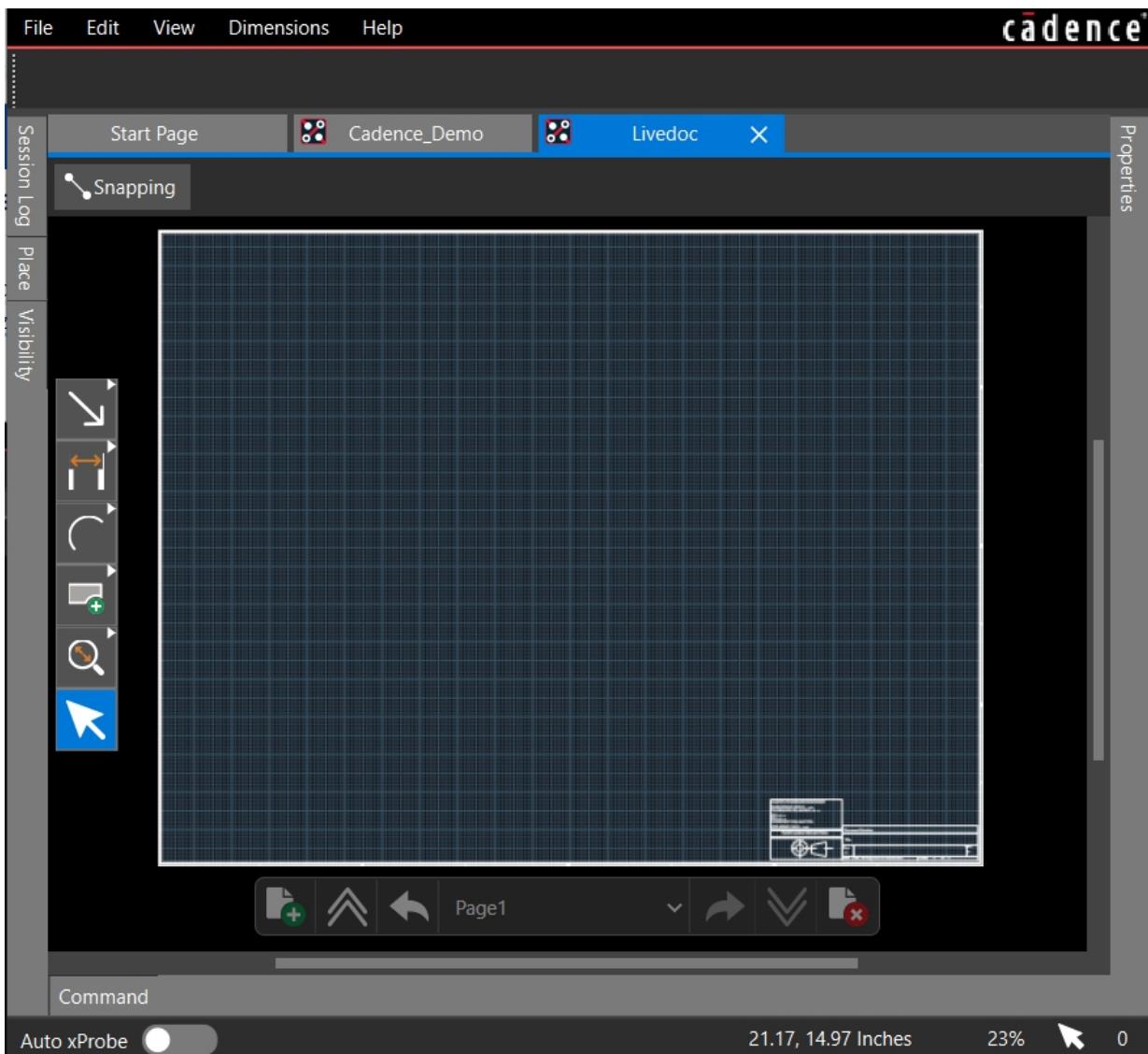
LiveDoc User Interface

The user interface of LiveDoc is similar to the main window of OrCAD X Presto and displays its panels, menus, and toolbars.



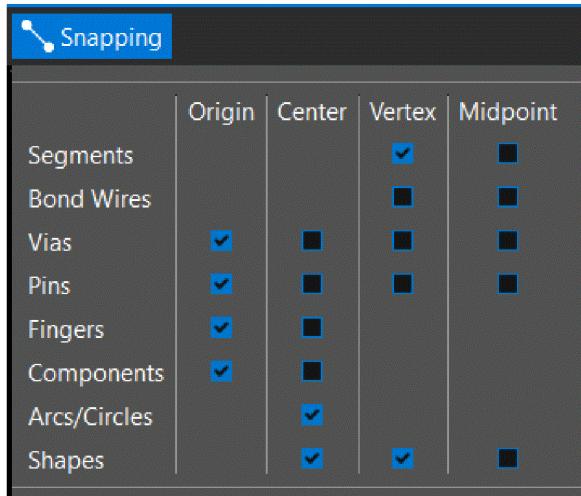
LiveDoc Canvas

The LiveDoc canvas is the center panel of the main window where PCB views are placed. When you open a new page in LiveDoc, a default page border is already placed that can be replaced with a different page border available in the design library. The page border size determines the extent of the LiveDoc canvas. LiveDoc displays only one page at a time.



You can pan a Livedoc canvas using a mouse or arrow keys on the keyboard. Press and hold the middle mouse button and slide the mouse to the left, right, up, and down. You can also use the arrow keys on a keyboard to pan. The panning stops when you move the mouse outside of the Livedoc canvas.

Snapping controls are also accessible from the LiveDoc canvas.



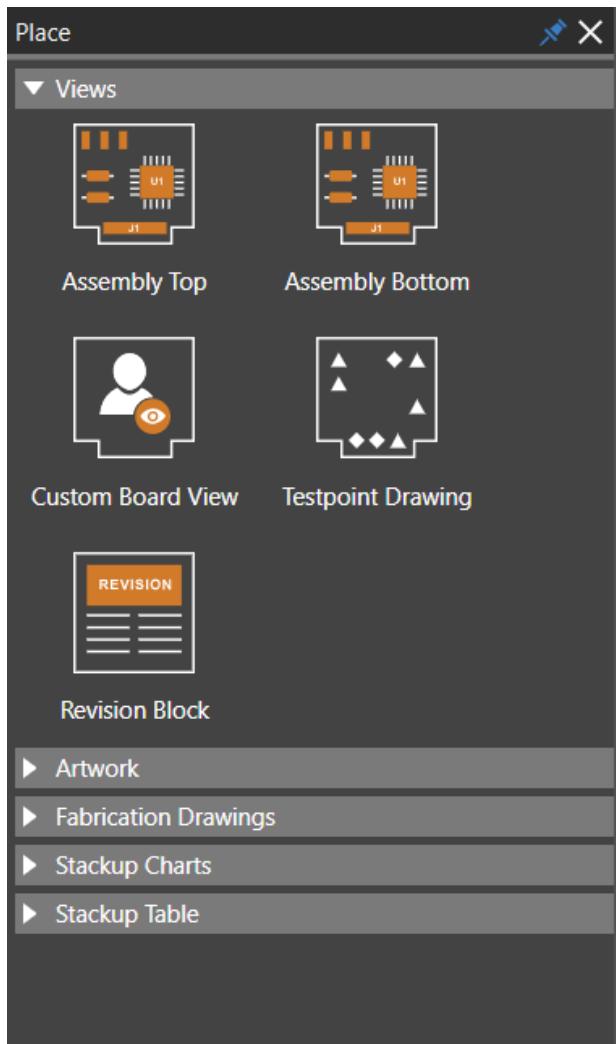
The screenshot shows the 'Snapping' panel in the LiveDoc interface. It is a grid-based configuration table with rows for different PCB elements and columns for snapping options. The columns are labeled 'Origin', 'Center', 'Vertex', and 'Midpoint'. Checkmarks indicate which snapping type is enabled for each element.

	Origin	Center	Vertex	Midpoint
Segments			<input checked="" type="checkbox"/>	<input type="checkbox"/>
Bond Wires			<input type="checkbox"/>	<input checked="" type="checkbox"/>
Vias	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
Pins	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
Fingers	<input checked="" type="checkbox"/>	<input type="checkbox"/>		
Components	<input checked="" type="checkbox"/>	<input type="checkbox"/>		
Arcs/Circles		<input checked="" type="checkbox"/>	<input type="checkbox"/>	
Shapes	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>

 Snapping controls are applied only when adding or modifying dimensions to LiveDoc.

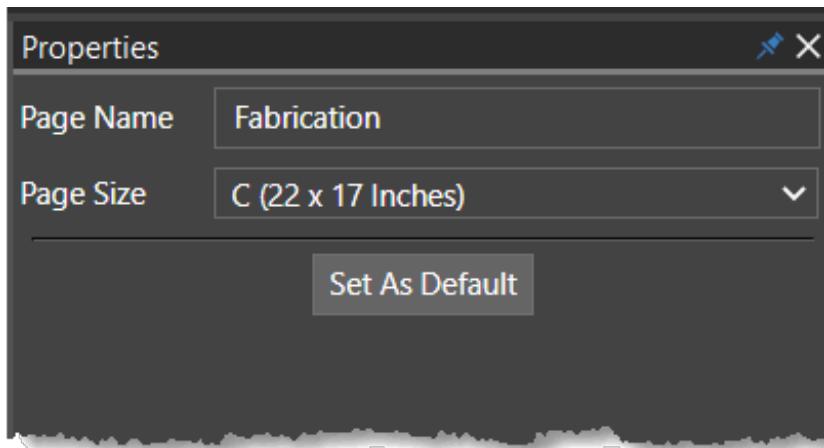
Panels

The *Place* panel contains PCB views for artwork, fabrication drawings, stack-up charts, and tables required for fabrication and assembly drawings. It displays the most common drawings and manufacturing outputs to create production documents. You can choose one of these PCB views and place it anywhere in the LiveDoc canvas.



For example, placing an *Artwork* view populates the view based on the artwork generated from the board design.

The *Properties* panel displays the properties of the selected object. In this panel, you can also edit information about the object that you have selected. When nothing is selected, the page border settings are displayed.



You can use the *Set As Default* option to save the current settings as default. Enabling this option eliminates the need to modify the properties of the same object when added to the LiveDoc canvas. When set as default, the modified property settings are saved with the LiveDoc data and available across sessions for the same design.

- ⓘ To reuse the property settings in other designs, create a new LiveDoc template using the *File – Save As Template* command. Use this template to create LiveDoc for new designs.

The *Properties* panel displays the properties of an object or PCB view selected in the LiveDoc canvas as follows:

Properties

Assembly Drawing

Title

Title

Location

Position Y 100

Size H 1106

Scale

Scale 1x 2x 5x 10x

1

Properties

View	Top	Bottom		
Rotation	0°	90°	180°	270°

Enable Pins and Vias in Visibility Pane to work with holes.

Display Holes

All Holes
 Plated
 Non Plated
 Mechanical
 Component
 Via
 Minimum Diameter Only

Minimum Diameter Millimeters

Use Ctrl+Double click left mouse button to manually customize PCB View area using zoom and pan. This feature is not available when dimensions are annotated on the PCB view.

Set As Default

Properties

Artwork Drawing

Title

Title

Location

Position Y 99

Size H 1106

Scale

Scale 1x 2x 5x 10x

1

Properties

View	Top	Bottom		
Rotation	0°	90°	180°	270°
Undefined Line Width	0.1524	Millimeters		

Use Ctrl+Double click left mouse button to manually customize PCB View area using zoom and pan. This feature is not available when dimensions are annotated on the PCB view.

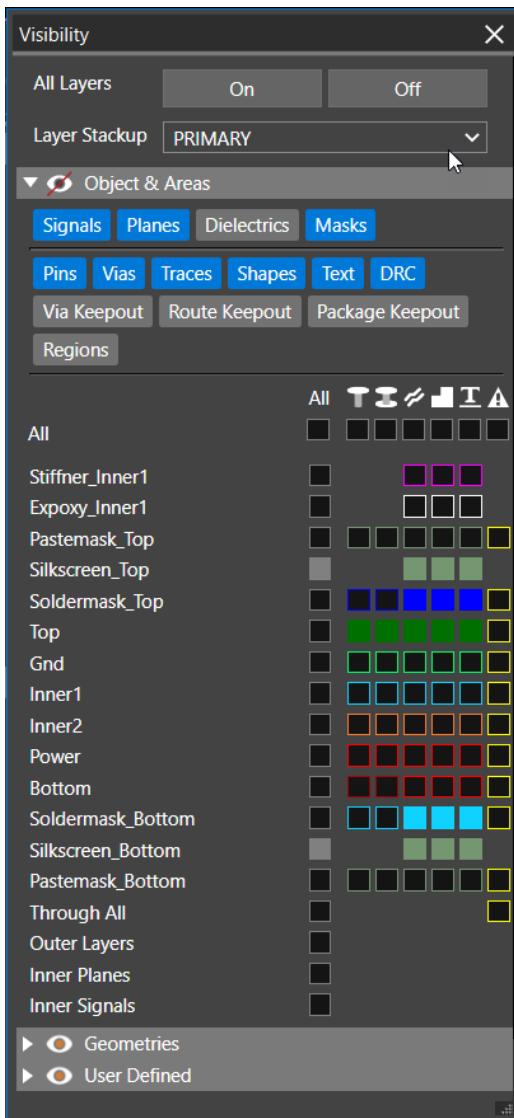
Set As Default

The *Visibility* panel controls the visibility of objects inside a selected PCB view. Selecting a different PCB view changes the settings in the *Visibility* panel.

September 2023

150

Product Version 23.1



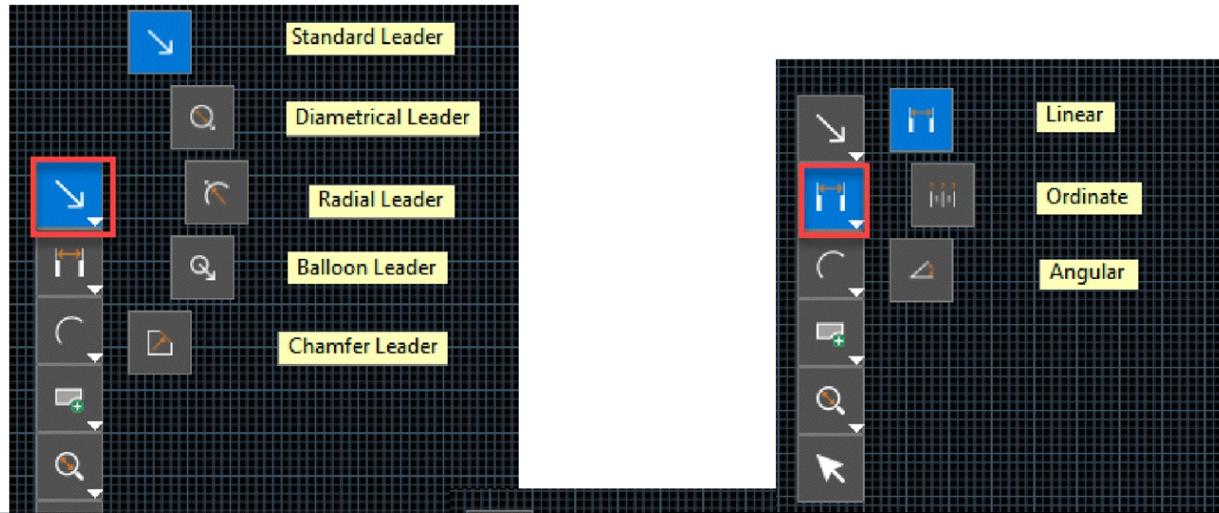
Toolbars

There are two floating toolbars in the main window:

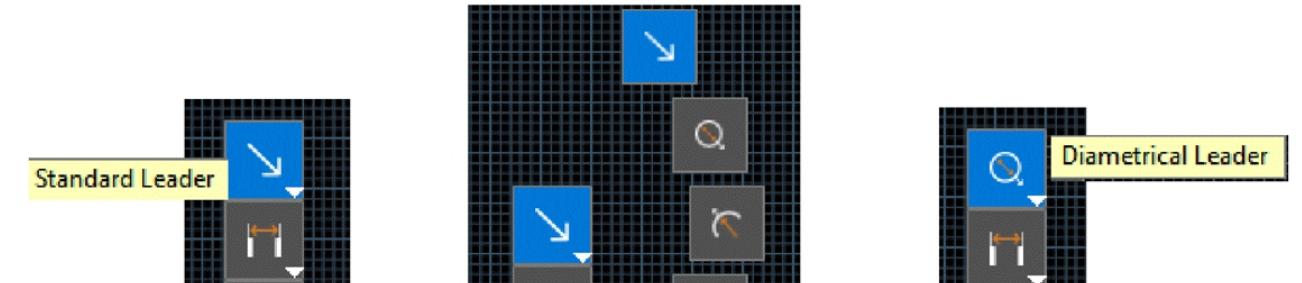
- Command Toolbar
- Page Toolbar

Command Toolbar

The command toolbar is vertically aligned and provides controls to add PCB views, dimensions, annotations, control zoom, and selection of objects.

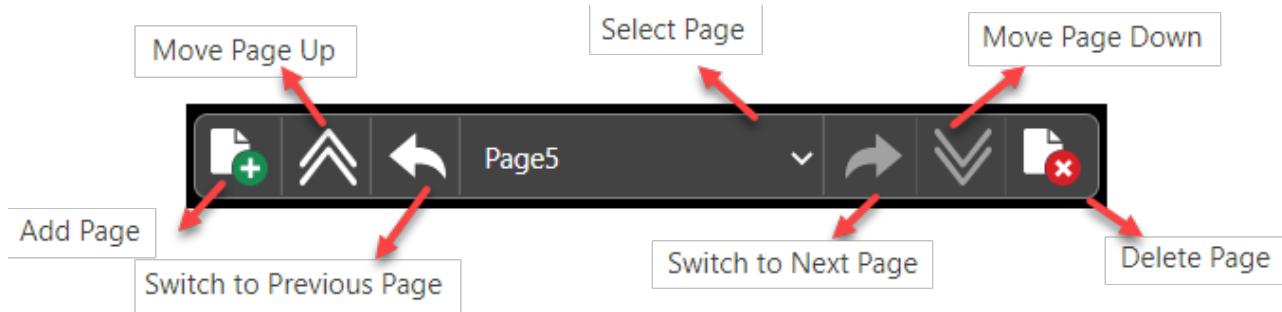


By default, the *Select* icon is enabled in the command toolbar. Remaining icons represent a set of command actions denoted by a small inverted triangle. You can right-click any of the icons to display its associated command icons. When you select an icon, it becomes the default icon in the toolbar.

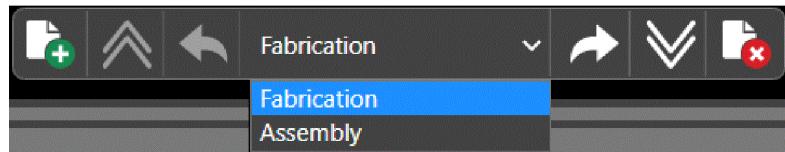


Page Toolbar

LiveDoc supports multiple pages in a template that can be used to create various types of PCB documents. At the bottom of the design canvas, the page toolbar provides controls to include and exclude pages and navigation controls.



By default, *Fabrication* and *Assembly* pages are available in the template.



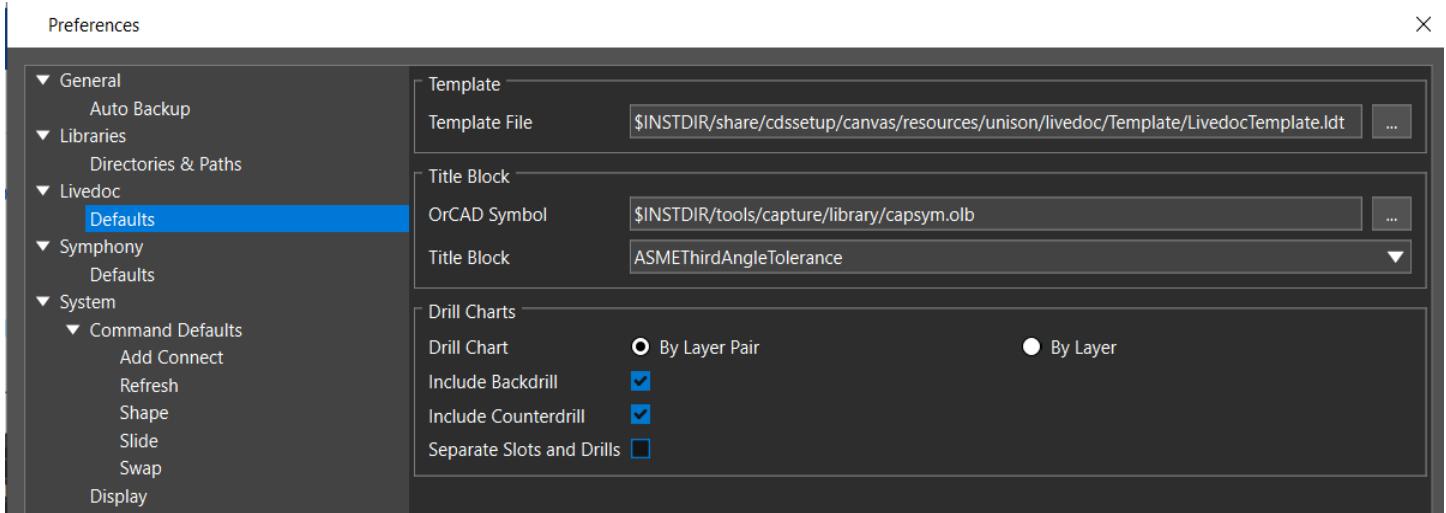
Page settings, such as name and size, can be modified in the *Properties* panel at any time during the documentation process.

Related Topics

- [What is LiveDoc?](#)
- [Launching LiveDoc](#)
- [LiveDoc Preferences](#)
- [Using LiveDoc Templates](#)
- [LiveDoc Workflow](#)
- [Adding a Page in LiveDoc](#)
- [Defining Page Title Blocks](#)
- [PCB Views in LiveDoc](#)
- [Placing PCB Views in LiveDoc](#)
- [Viewing and Sharing PCB Views in LiveDoc](#)
- [Defining Dimensions in LiveDoc](#)
- [Defining Annotations in LiveDoc](#)
- [Managing LiveDoc](#)

LiveDoc Preferences

The LiveDoc application works with a predefined set of preferences specified in the *Preferences* dialog box.



In the *Preferences* dialog box, you can modify the following:

- **Template File:** Specifies the path to the default template file (`LivedocTemplate.ldt`) from the following directory location of the install area:
`<install_directory>/share/cdssetup/canvas/resources/unison/livedoc/Template`
You can also specify the path to any custom template file.

i The new template file is used only for creating a new LiveDoc. All the existing board designs containing LiveDoc data remain unaffected.

- **Title Block:** Specifies the path to the OrCAD symbol library (`.olb`) and the name of the title block symbol used with the default page border. It contains various fields that identify the drawing and other details, such as company logo and address, title, drawing number, revision, sheet size, drawn by, checked by, approved by, with dates, sheet number, drawing scale, and so on.
- **Drill Chart:** Controls the options of the drill chart. A drill chart represents drill hole sizes and locations as a graphical symbol. The location of the symbol marks the location of the drill in the PCB, and the symbol maps to a drill chart that defines the hole size, tolerances, and plating status. You can create a drill chart by layer pair or layer, including backdrill, counterdrill, and separate slots and drills in the drill chart.

Related Topics

- [What is LiveDoc?](#)
- [Launching LiveDoc](#)
- [LiveDoc User Interface](#)
- [Using LiveDoc Templates](#)
- [LiveDoc Workflow](#)
- [Adding a Page in LiveDoc](#)
- [Defining Page Title Blocks](#)
- [PCB Views in LiveDoc](#)
- [Placing PCB Views in LiveDoc](#)
- [Viewing and Sharing PCB Views in LiveDoc](#)
- [Defining Dimensions in LiveDoc](#)
- [Defining Annotations in LiveDoc](#)
- [Managing LiveDoc](#)

Using LiveDoc Templates

A LiveDoc template is a collection of PCB views of the board, tables, and charts placed across one or more pages for a design. A template includes page properties and custom PCB views used to create fabrication and assembly drawing from any board design. For example, if your company creates a three-page documentation of fabrication, assembly drawings, and manufacturing information, you can save this LiveDoc as a template. Any new design using this template automatically generates the same set of PCB documents. The only additional task is to add design-specific data, such as dimensions and notes.

Creating Custom Templates

You can create custom templates by modifying the default template file `LivedocTemplate.ldt` available in the install directory. To create a custom template, do the following:

1. Open LiveDoc using the default template.
2. Add PCB views, charts, and tables to the LiveDoc.
3. Select individual PCB views and change their properties in the *Properties* panel.
4. Save the modified template as a new template using *File – Save As Template*.

 By specifying the template file path in the *Preferences* dialog box, the custom template can be used for new board designs.

Related Topics

- [What is LiveDoc?](#)
- [Launching LiveDoc](#)
- [LiveDoc User Interface](#)
- [LiveDoc Preferences](#)
- [LiveDoc Workflow](#)
- [Adding a Page in LiveDoc](#)
- [Defining Page Title Blocks](#)
- [PCB Views in LiveDoc](#)
- [Placing PCB Views in LiveDoc](#)
- [Viewing and Sharing PCB Views in LiveDoc](#)
- [Defining Dimensions in LiveDoc](#)
- [Defining Annotations in LiveDoc](#)
- [Managing LiveDoc](#)

LiveDoc Workflow

The following sequence of tasks conveys the standard flow of tasks performed in LiveDoc to create PCB documents when the design is ready for production:

- Create LiveDoc using the default template
- Add pages to LiveDoc
- Place default PCB views to LiveDoc
- Create Custom PCB Views by modifying the default PCB view:
 - Visibility of layers
 - Properties of objects
- Supplement data for manufacturing and assembly by adding:
 - Dimensions and leaders
 - Graphical annotations and notes
- Save the customized LiveDoc as a new template
- Export LiveDoc as PDF for review

Related Topics

- [What is LiveDoc?](#)
- [Launching LiveDoc](#)
- [LiveDoc User Interface](#)
- [LiveDoc Preferences](#)
- [Using LiveDoc Templates](#)
- [Adding a Page in LiveDoc](#)
- [Defining Page Title Blocks](#)
- [PCB Views in LiveDoc](#)
- [Placing PCB Views in LiveDoc](#)
- [Viewing and Sharing PCB Views in LiveDoc](#)
- [Defining Dimensions in LiveDoc](#)
- [Defining Annotations in LiveDoc](#)
- [Managing LiveDoc](#)

Adding Page in LiveDoc

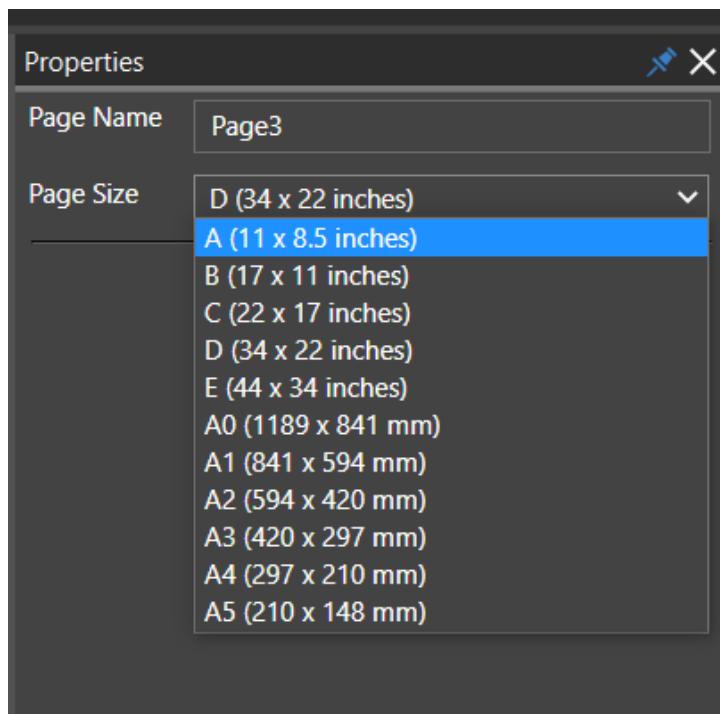
In LiveDoc, you can select different page sizes for creating assembly and fabrication drawings in the *Properties* panel. Do the following to change the default page size:

1. Click the *Add Page* icon in the page toolbar.

The default page is added to the LiveDoc canvas. The *Properties* panel displays its page attributes.

2. Expand the *Page Size* pull-down list in the *Properties* panel.

The list displays default page sizes and their dimensions. The pages from A to E follow American National Standard Institute (ASME) requirements and display size in inches. The pages from A0 to A5 conform to International Organization for Standardization (ISO) and display size in millimeters.



3. Choose another page size.

4. Specify a name for the page.

The selected page is changed in the LiveDoc canvas. All the measurements and the coordinates also start showing in the native unit of the page.

5. Optionally, click *Set as Default*.

The selected page is used when adding new pages to the LiveDoc.

Related Topics

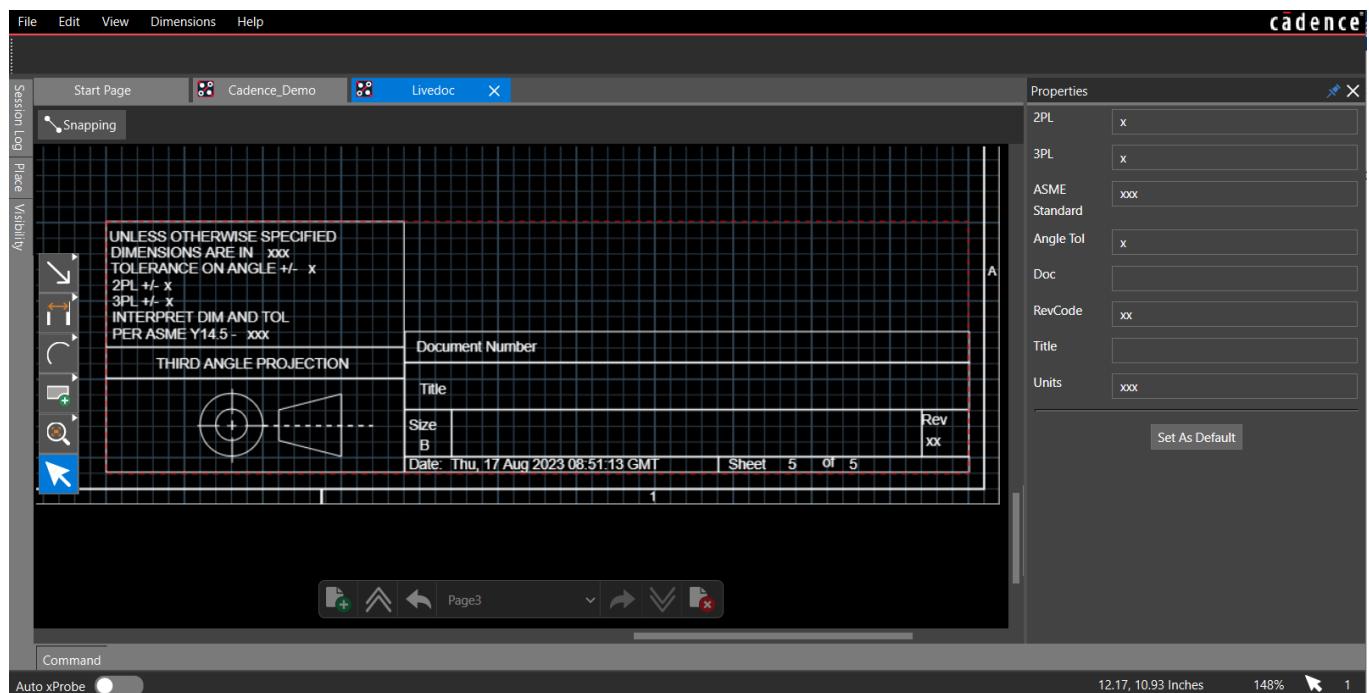
- [What is LiveDoc?](#)
- [Launching LiveDoc](#)
- [LiveDoc User Interface](#)
- [LiveDoc Preferences](#)
- [Adding a Page in LiveDoc](#)
- [LiveDoc Workflow](#)
- [Defining Page Title Blocks](#)
- [PCB Views in LiveDoc](#)
- [Placing PCB Views in LiveDoc](#)
- [Viewing and Sharing PCB Views in LiveDoc](#)
- [Defining Dimensions in LiveDoc](#)
- [Defining Annotations in LiveDoc](#)
- [Managing LiveDoc](#)

Defining Page Title Block

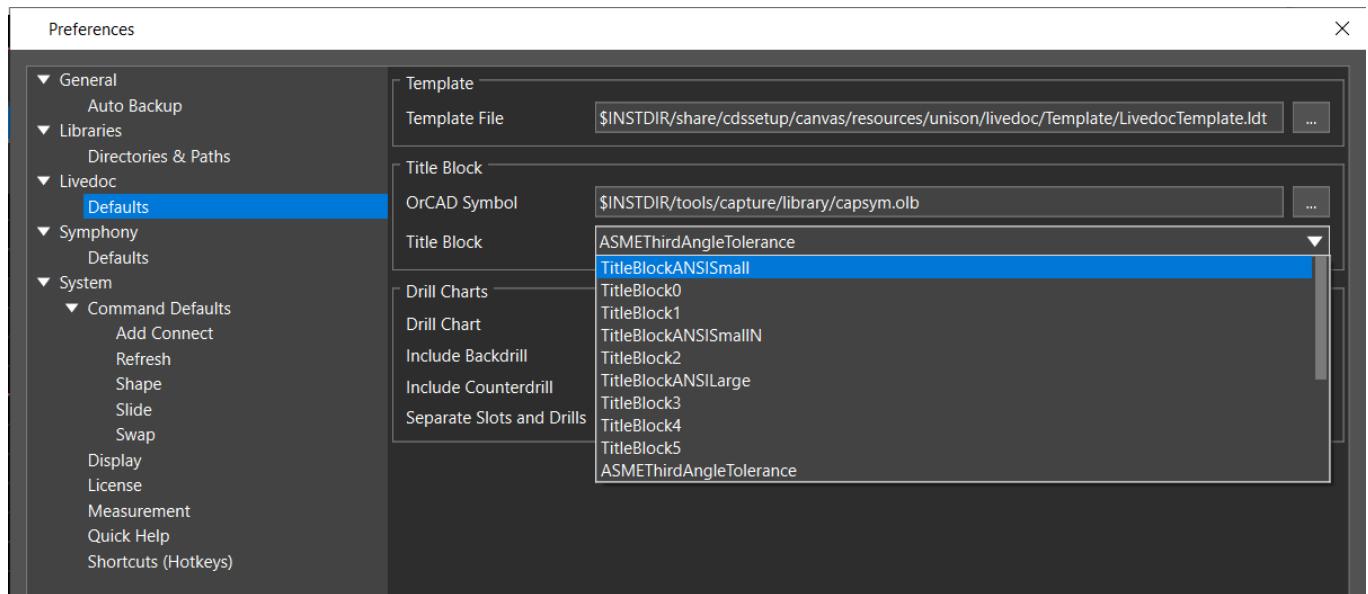
A title block at the bottom right corner of a page indicates drawing details such as the title, document number, revision, and date when the drawing was created. To specify the title block properties, do the following:

1. Open any page.
2. Zoom into the bottom-right corner of the page and select the title block.

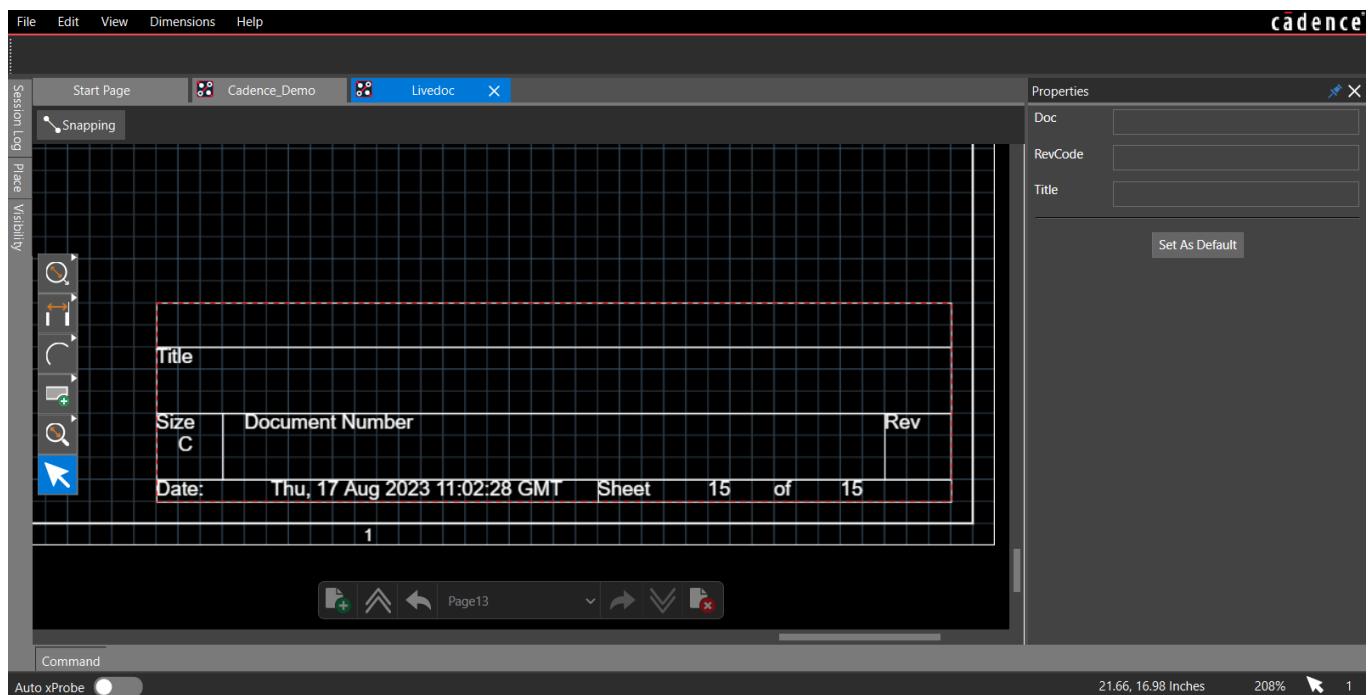
The title block specified in the *Preferences* dialog box is used. The page size, creation date and time, and page number are automatically populated in the title block. and the *Properties* panel displays its properties.



3. Enter the title, document number, and revision in the *Properties* panel.
The values are added to the title block.
4. Alternatively, click *Save a Default* to use the same title block attributes for new pages.
5. Open the *Preferences* dialog box and choose a different title block from the list.



You see a different set of attributes in the *Properties* panel depending on which title block you select.



The selected title block is used only for new pages added to the LiveDoc.

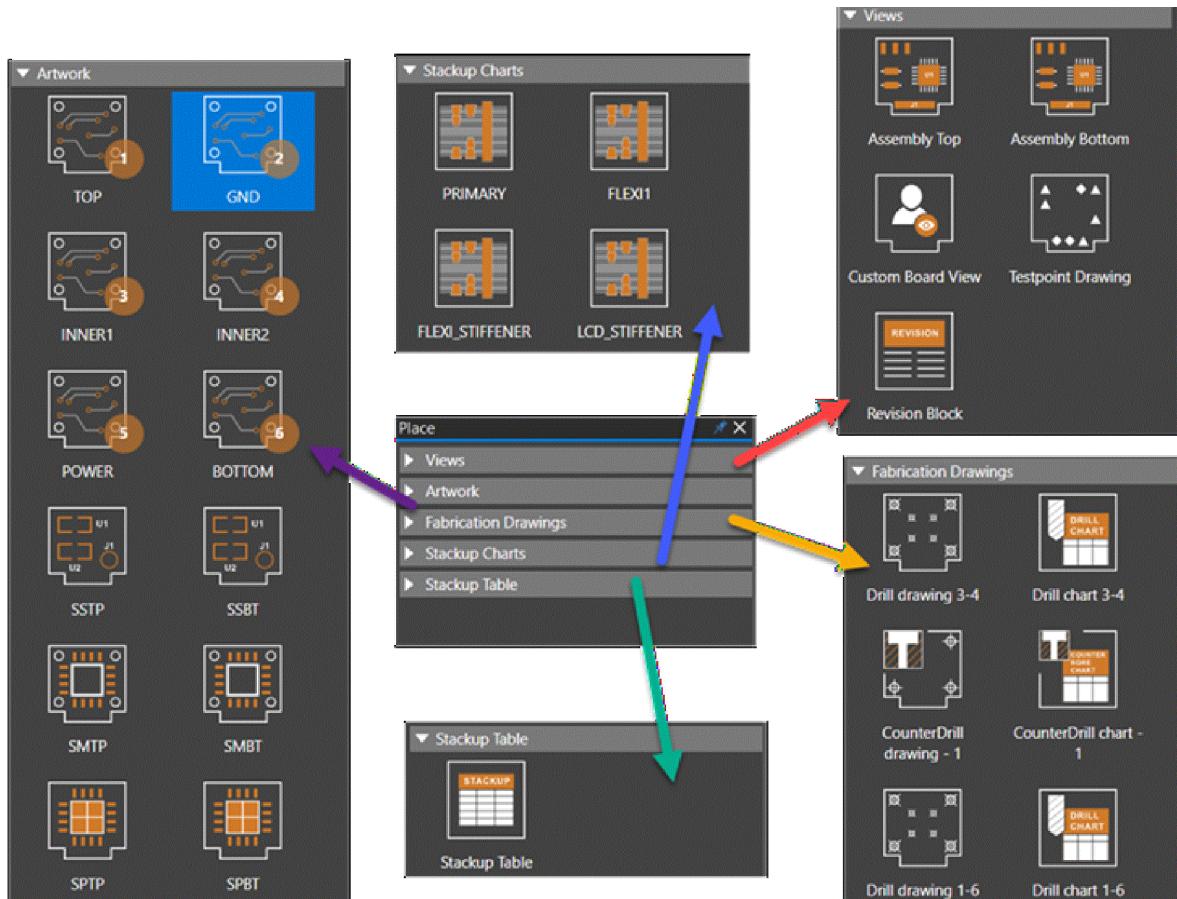
Related Topics

- [What is LiveDoc?](#)
- [Launching LiveDoc](#)
- [LiveDoc User Interface](#)
- [LiveDoc Preferences](#)
- [Defining Page Title Blocks](#)
- [LiveDoc Workflow](#)
- [Adding a Page in LiveDoc](#)
- [PCB Views in LiveDoc](#)
- [Placing PCB Views in LiveDoc](#)
- [Viewing and Sharing PCB Views in LiveDoc](#)
- [Defining Dimensions in LiveDoc](#)
- [Defining Annotations in LiveDoc](#)
- [Managing LiveDoc](#)

PCB Views in LiveDoc

PCB views are different drawing types that include dimensional drawings, assembly drawings and instructions, Gerber files, fabrication documents, and so on. It takes a significant amount of time for complex PCB designs to create these documents to build, view, and archive the final product. Using LiveDoc, you can quickly create the PCB drawings that drive PCB fabrication and assembly. LiveDoc offers a default set of PCB drawings or PCB views that can be placed to document your PCB design. These predefined PCB views are auto-generated from the board design and can be placed and modified as required. You can create and place unlimited PCB views of the design. Each PCB view may have its own display settings and can be formatted independently of the board design.

Predefined PCB Views in LiveDoc



Related Topics

- [What is LiveDoc?](#)
- [Launching LiveDoc](#)
- [LiveDoc User Interface](#)
- [LiveDoc Preferences](#)
- [Using LiveDoc Templates](#)
- [LiveDoc Workflow](#)
- [Adding a Page in LiveDoc](#)
- [Defining Page Title Blocks](#)
- [Placing PCB Views in LiveDoc](#)
- [Viewing and Sharing PCB Views in LiveDoc](#)
- [Defining Dimensions in LiveDoc](#)
- [Defining Annotations in LiveDoc](#)
- [Managing LiveDoc](#)

Placing PCB Views in LiveDoc

You can select PCB views in the *Place* panel and place them in the LiveDoc canvas. Each PCB view can be scaled and formatted independently of the other. You can place as many PCB views as you want on each page. However, placing PCB views outside the extent of LiveDoc canvas is not supported.

- Extend the page border size or increase the view scaling in the Properties panel to fit more PCB views.

You can place specific PCB views of the board outline, assembly top/bottom views of the PCB, drill patterns and charts, stack-up tables, and so on. The default PCB views contain basic information about the design, such as placements of components and layer display. To further minimize rework, any customization for a PCB view is saved with the board design database and available when you reopen LiveDoc.

Related Topics

- [What is LiveDoc?](#)
- [Launching LiveDoc](#)
- [LiveDoc User Interface](#)
- [LiveDoc Preferences](#)
- [Placing PCB Views in LiveDoc](#)
- [LiveDoc Workflow](#)
- [Adding a Page in LiveDoc](#)
- [Defining Page Title Blocks](#)
- [PCB Views in LiveDoc](#)
- [Viewing and Sharing PCB Views in LiveDoc](#)
- [Defining Dimensions in LiveDoc](#)
- [Defining Annotations in LiveDoc](#)
- [Managing LiveDoc](#)

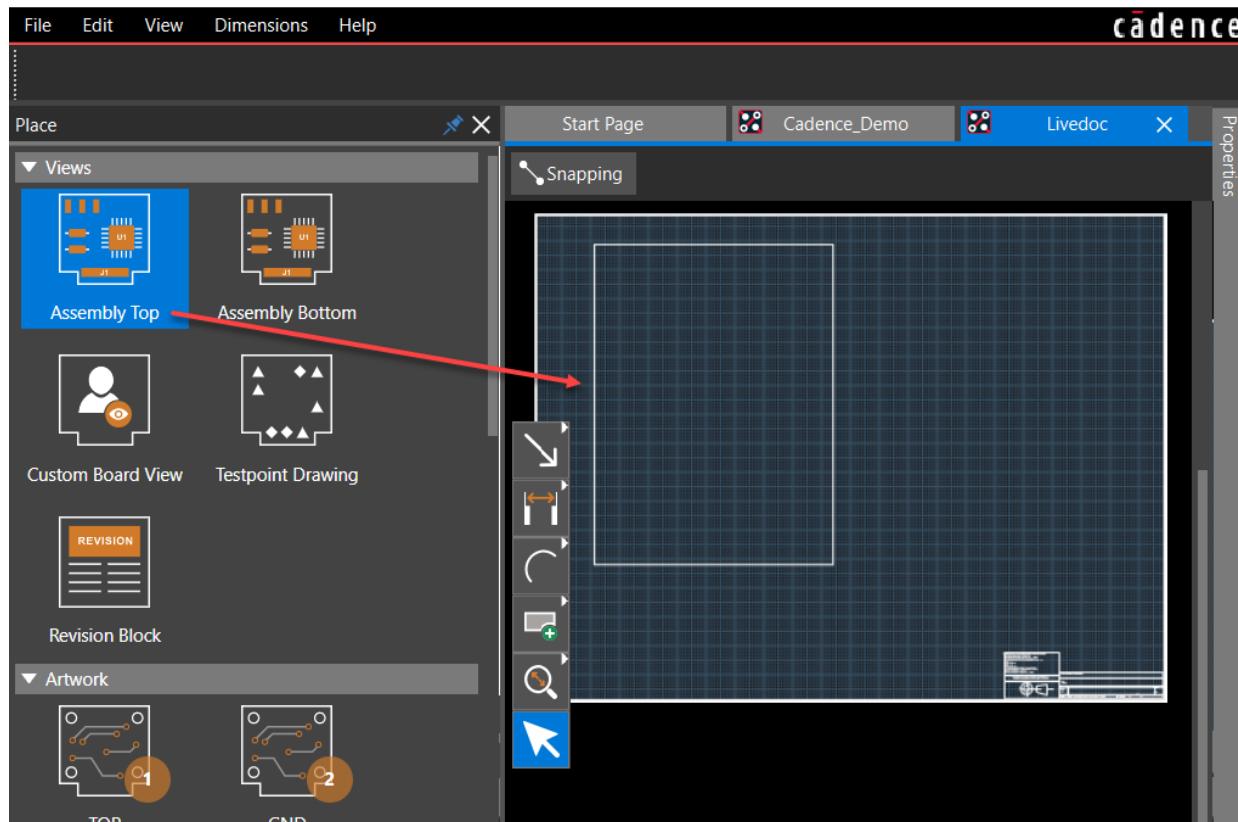
- Placing a PCB View in LiveDoc Canvas
- Placing Stackup Charts and Tables in LiveDoc Canvas
- Placing Drill Charts in LiveDoc Canvas

Placing a PCB View in LiveDoc Canvas

To place a PCB view, do the following:

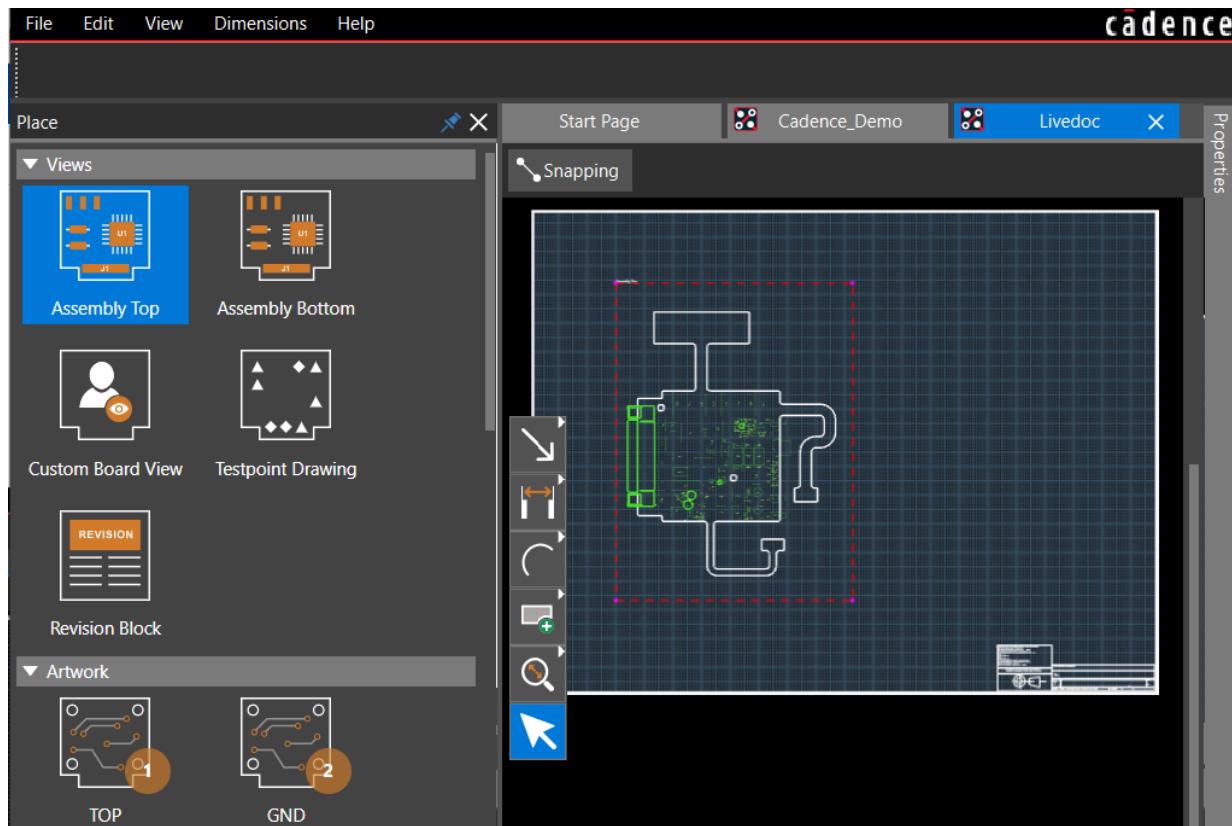
1. Select the icon for a PCB view in the *Place* panel.

The selected PCB view is highlighted in blue, and a white rectangle attached to the mouse cursor specifies the PCB view area on the canvas.



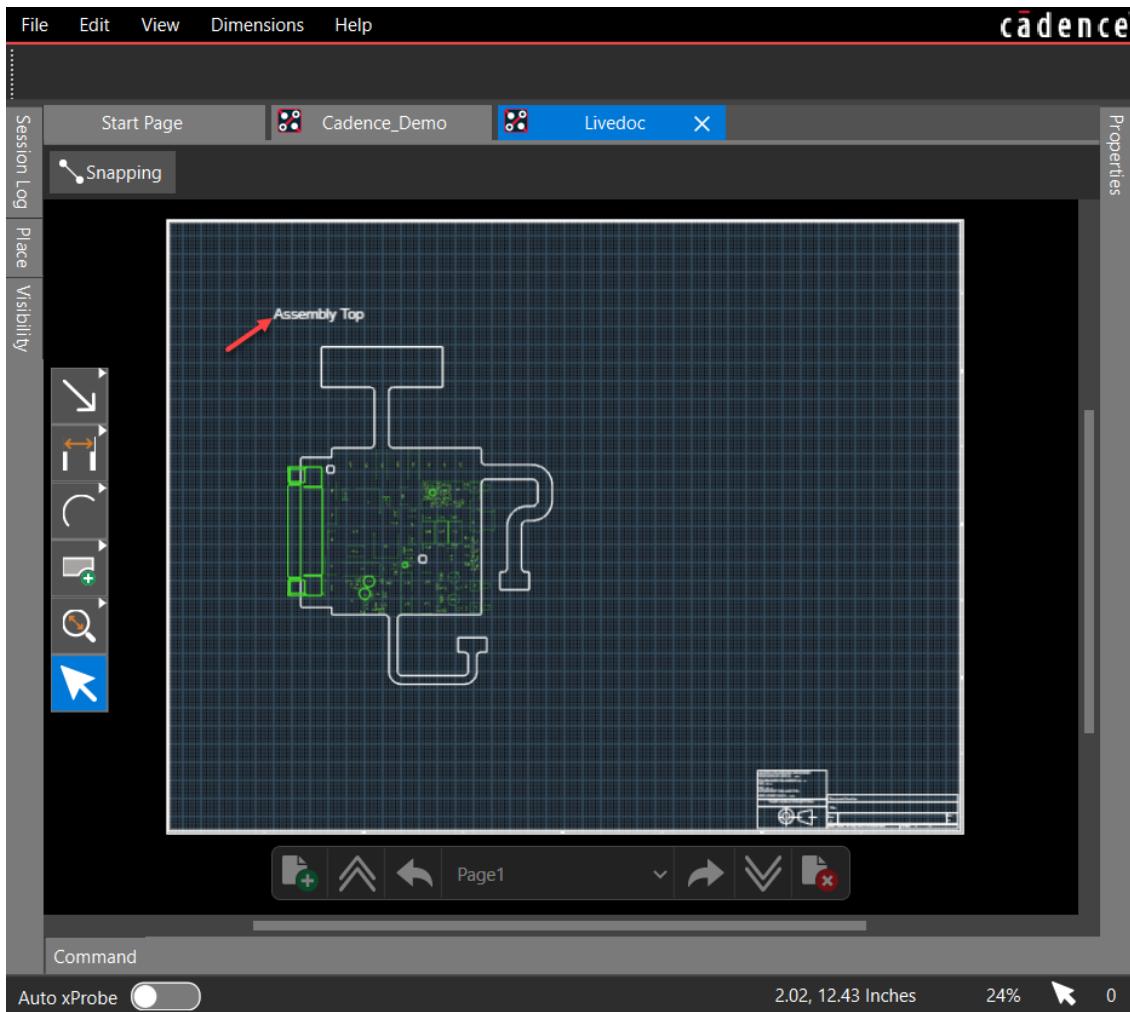
2. Drag and drop the PCB view in the LiveDoc canvas.

The selected view is placed on the LiveDoc canvas inside a red rectangular area.



3. Click to confirm the placement.

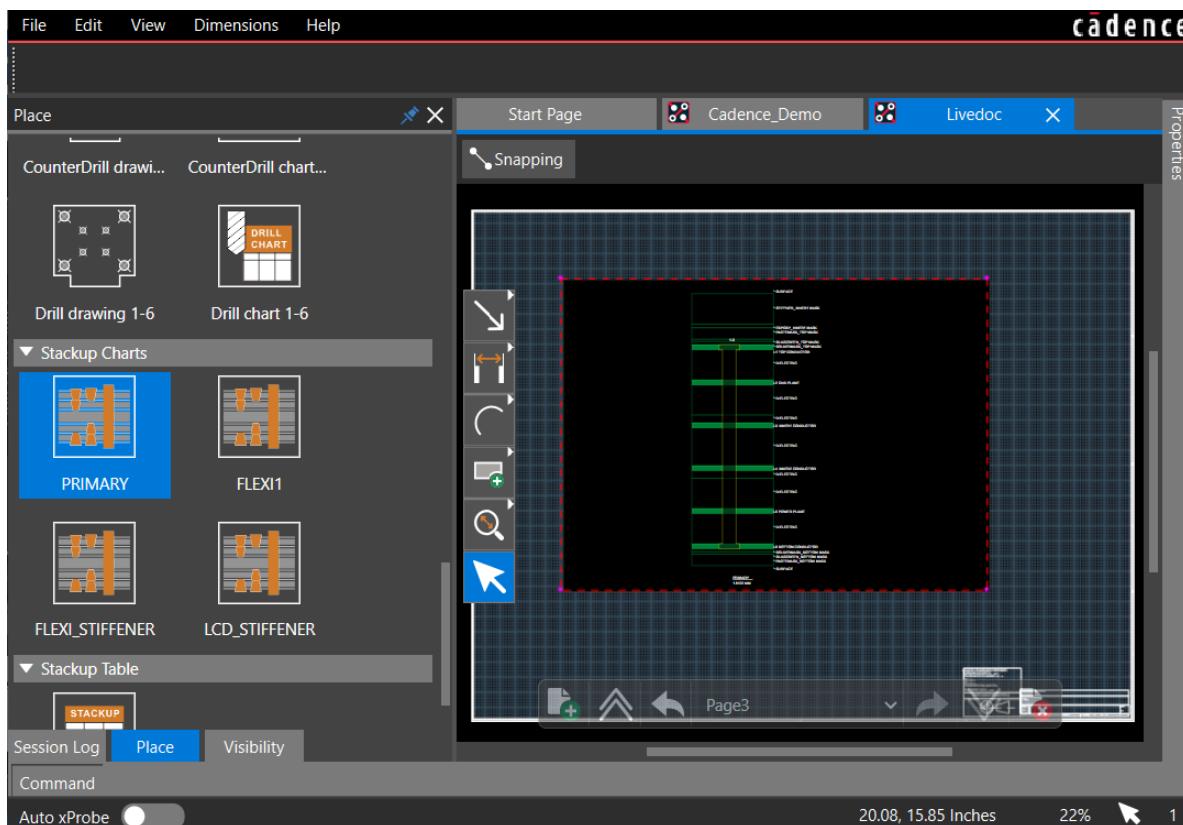
The selected PCB view is uploaded to the LiveDoc canvas, and the *Properties* panel displays the basic information. The name of the PCB view is displayed at the top-left corner of the PCB view area, as shown in the following image:

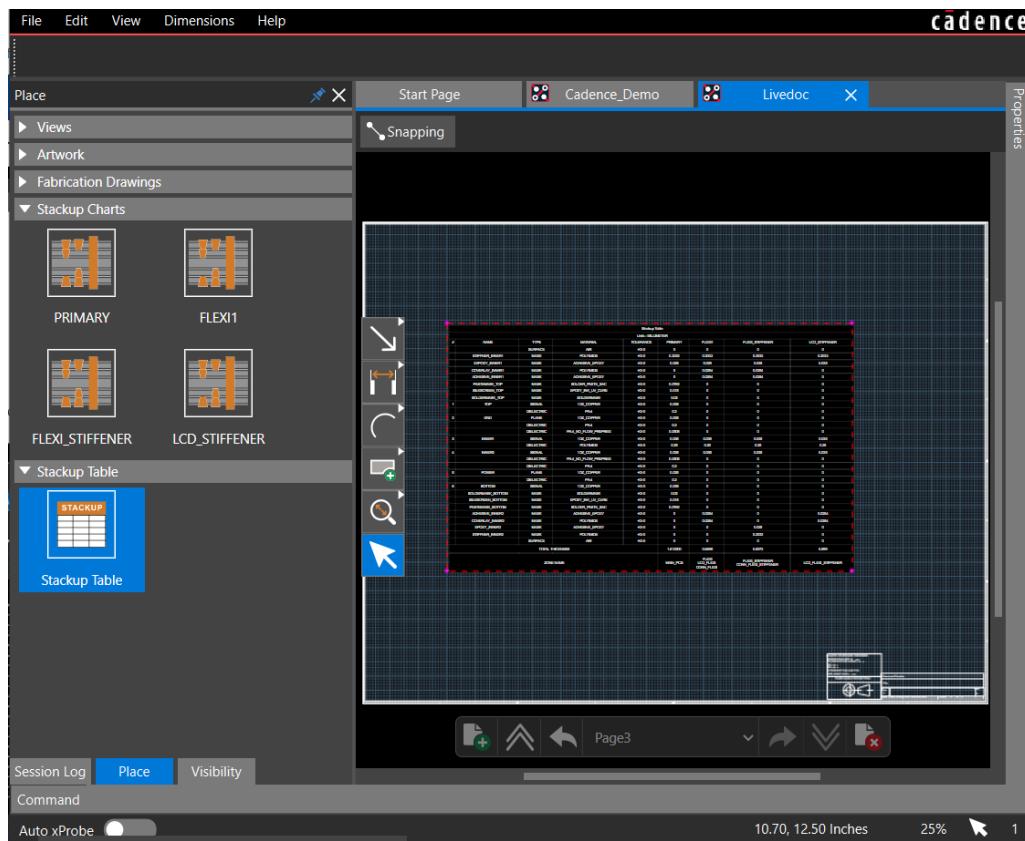


Placing Stackup Charts and Tables in LiveDoc Canvas

To place stackup charts or tables in the LiveDoc canvas, follow these steps:

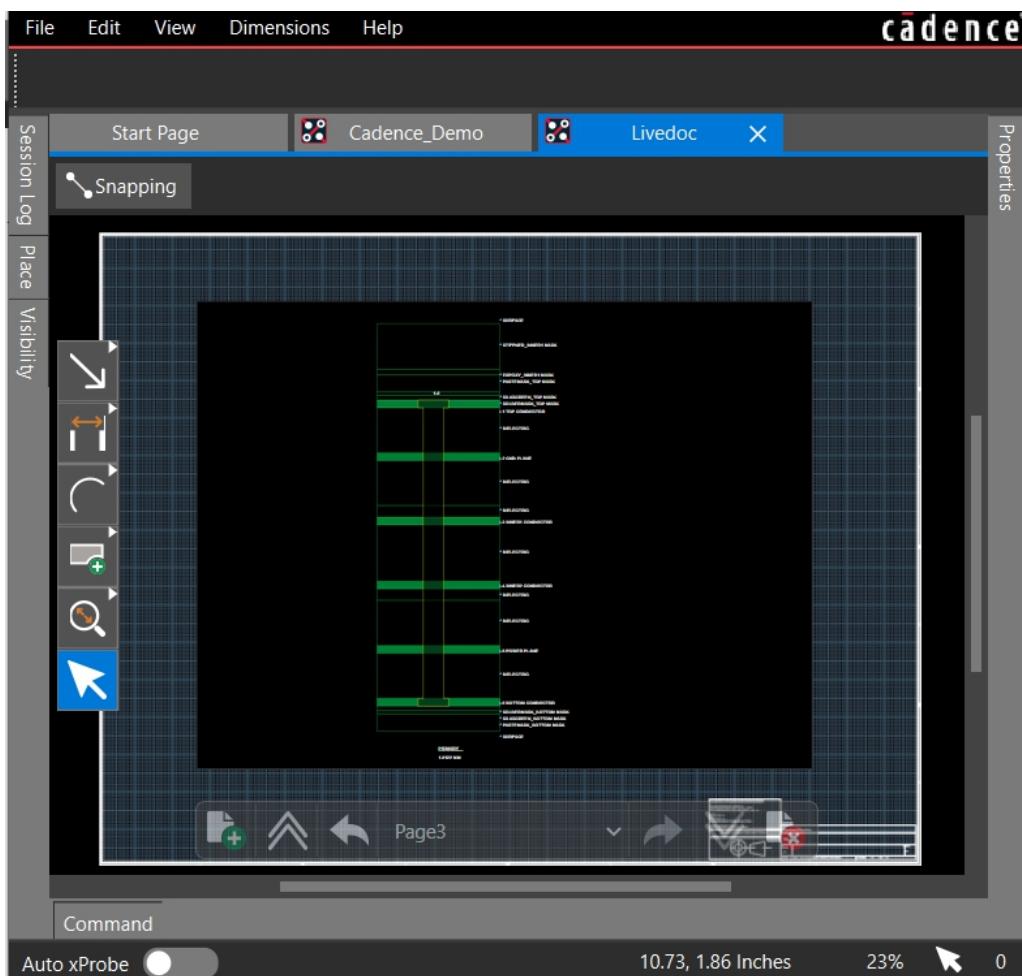
1. Click to select a stackup chart or table in the *Place* panel.
The selected stackup chart or table is highlighted in blue.
2. In the LiveDoc canvas, draw a rectangle to place the selected stackup chart or table.
The selected stackup chart or table is uploaded to the LiveDoc canvas inside a dashed-red border defining the size of the rectangle drawn.

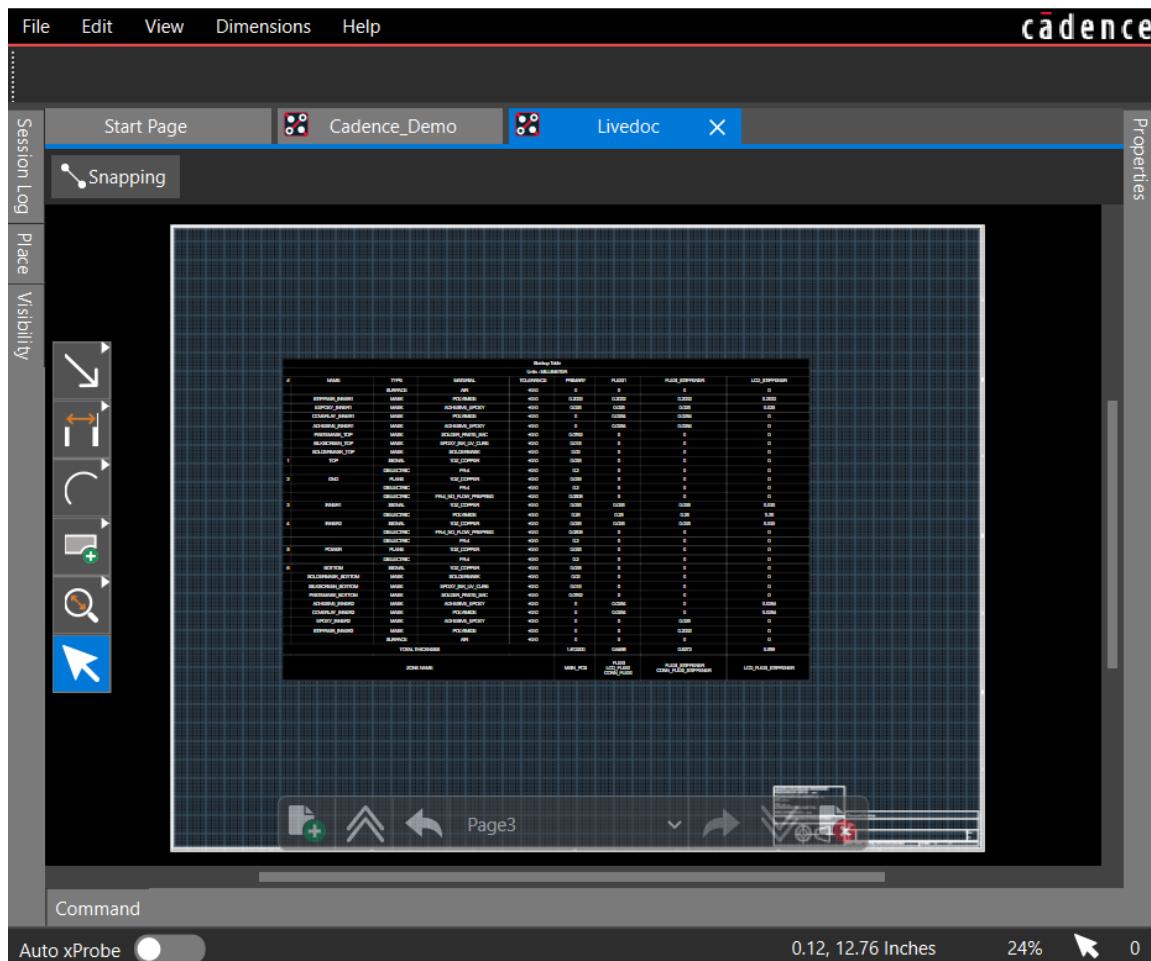




3. Click on the design canvas to complete the placement.

The selected stackup chart or table is uploaded to the LiveDoc canvas inside a red border defining the size of the rectangle drawn.



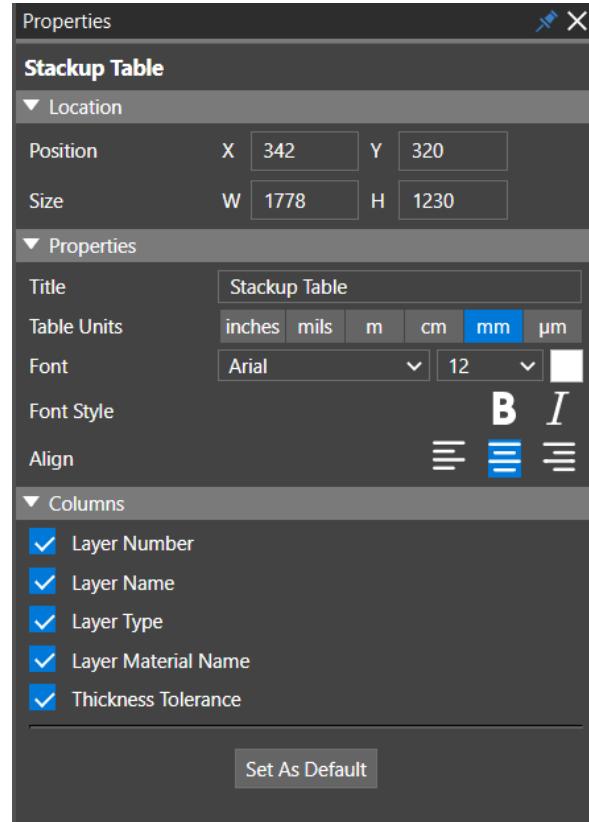
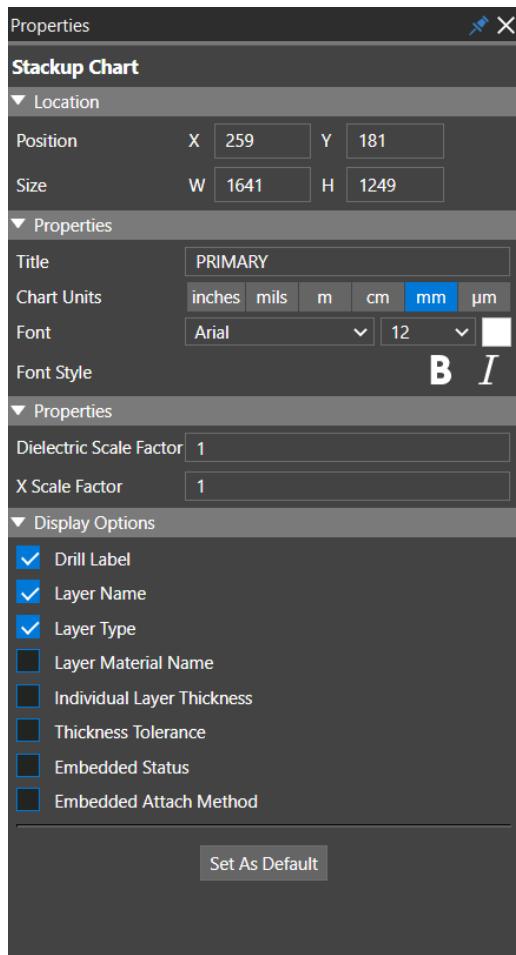


4. Select the chart or table in the LiveDoc canvas.

The *Properties* panel displays the location, text attributes, and display options for the placed chart or table.

5. Modify the chart or table properties in the *Properties* panel.

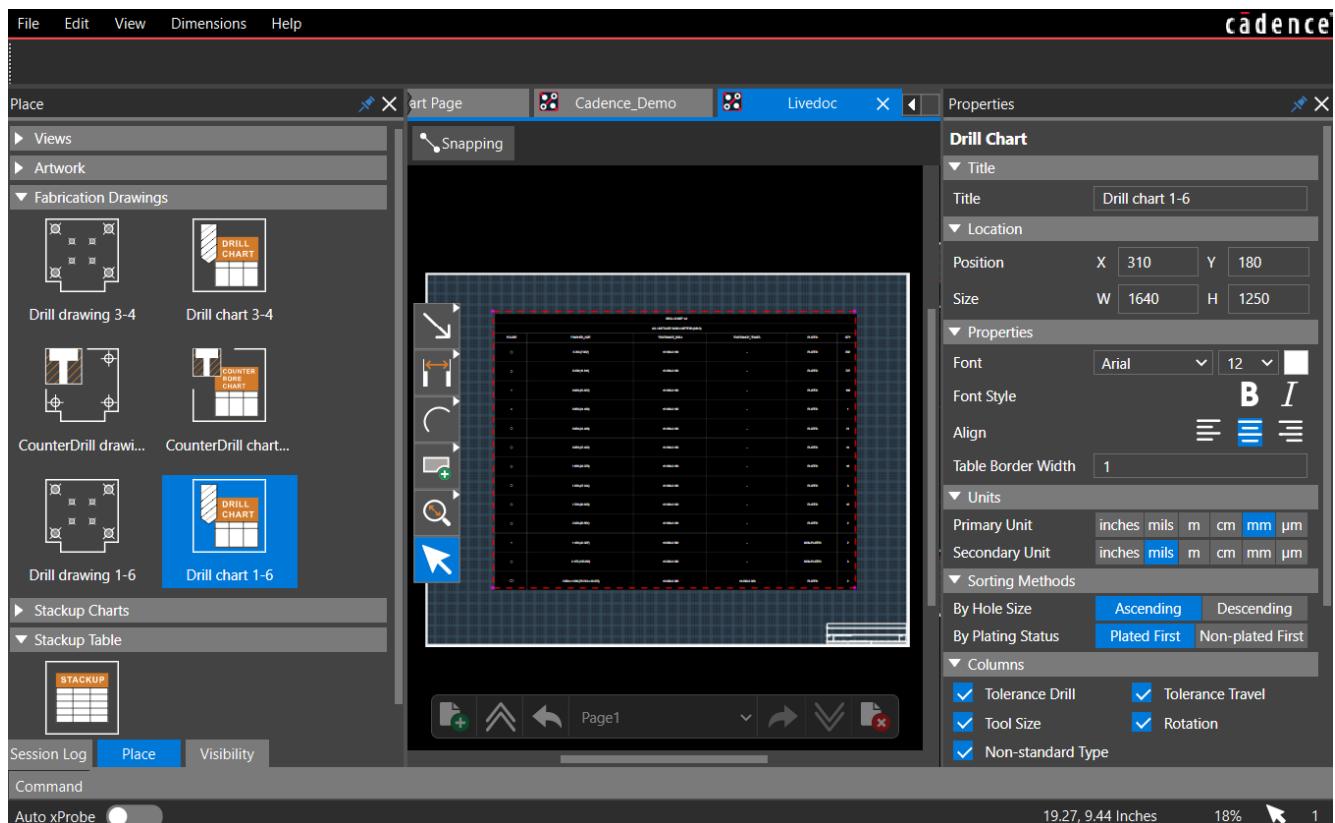
Unselecting checkboxes under the *Display Options* or *Columns* section turns off the text associated with these settings in the chart or table.



Placing Drill Charts in LiveDoc Canvas

To place drill charts in the LiveDoc canvas, follow these steps:

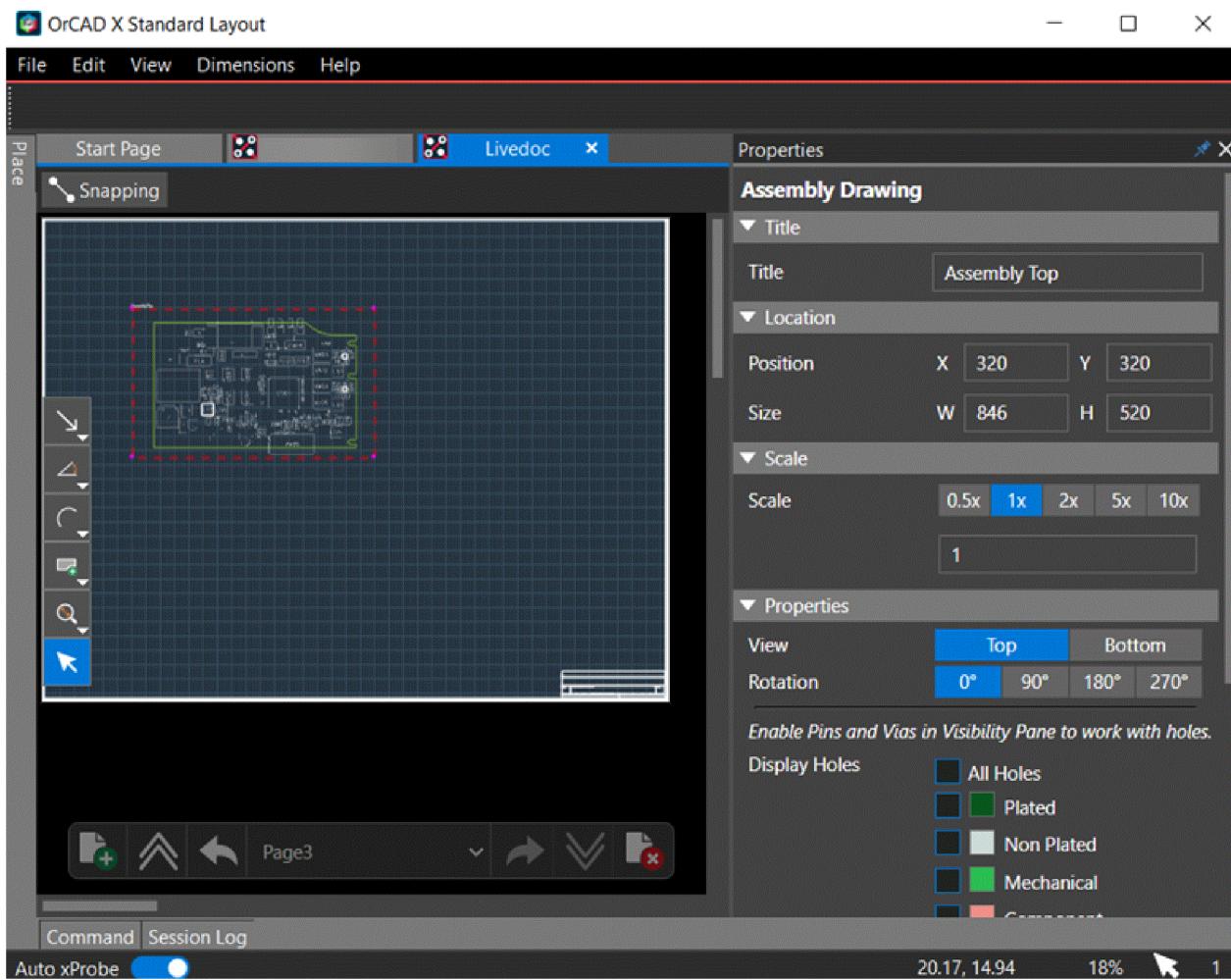
1. Click to select a drill chart from the *Fabrication Drawings* section in the *Place* panel.
The selected drill chart highlights in blue.
2. Draw a rectangle to place the selected drill chart in the LiveDoc canvas.
The selected drill chart is uploaded to the LiveDoc canvas inside a dashed-red border defining the size of the rectangle drawn.
3. Click on the design canvas to complete the placement.
The selected drill chart is uploaded to the LiveDoc canvas inside a red border defining the size of the rectangle drawn. The *Properties* panel displays the location, units, scaling methods, show or hide columns, and display options of the drill chart.



4. Modify the drill chart properties in the *Properties* panel.
Unselecting checkboxes under the *Display Options* or *Columns* section turns off the text associated with these settings in the drill chart.

Viewing and Sharing PCB Views in LiveDoc

When you select a PCB view in the LiveDoc canvas by clicking anywhere inside the PCB view area, a red dashed-border rectangle highlights its boundary, and the *Properties* panel displays its properties. You can now control the display of objects and layers inside the PCB view.



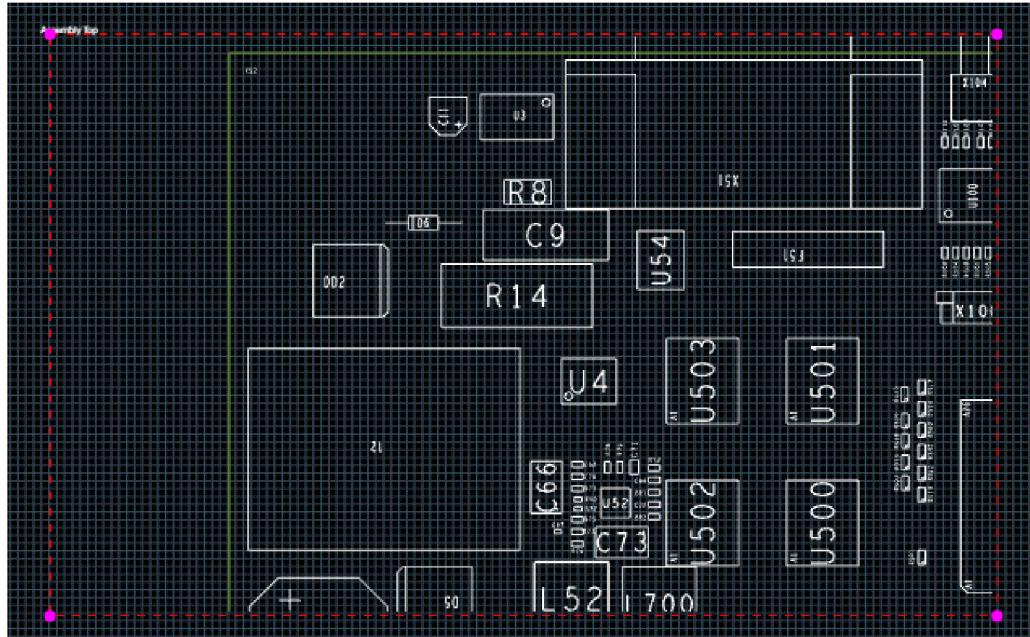
Changing Display of PCB Views

You can resize a PCB view and also zoom in and out or pan to a specific location inside the view.

To zoom and pan inside a PCB view, follow these steps:

1. Hold the `Ctrl` key and double-click anywhere inside the PCB view area.
The cursor changes to a hand pointer.

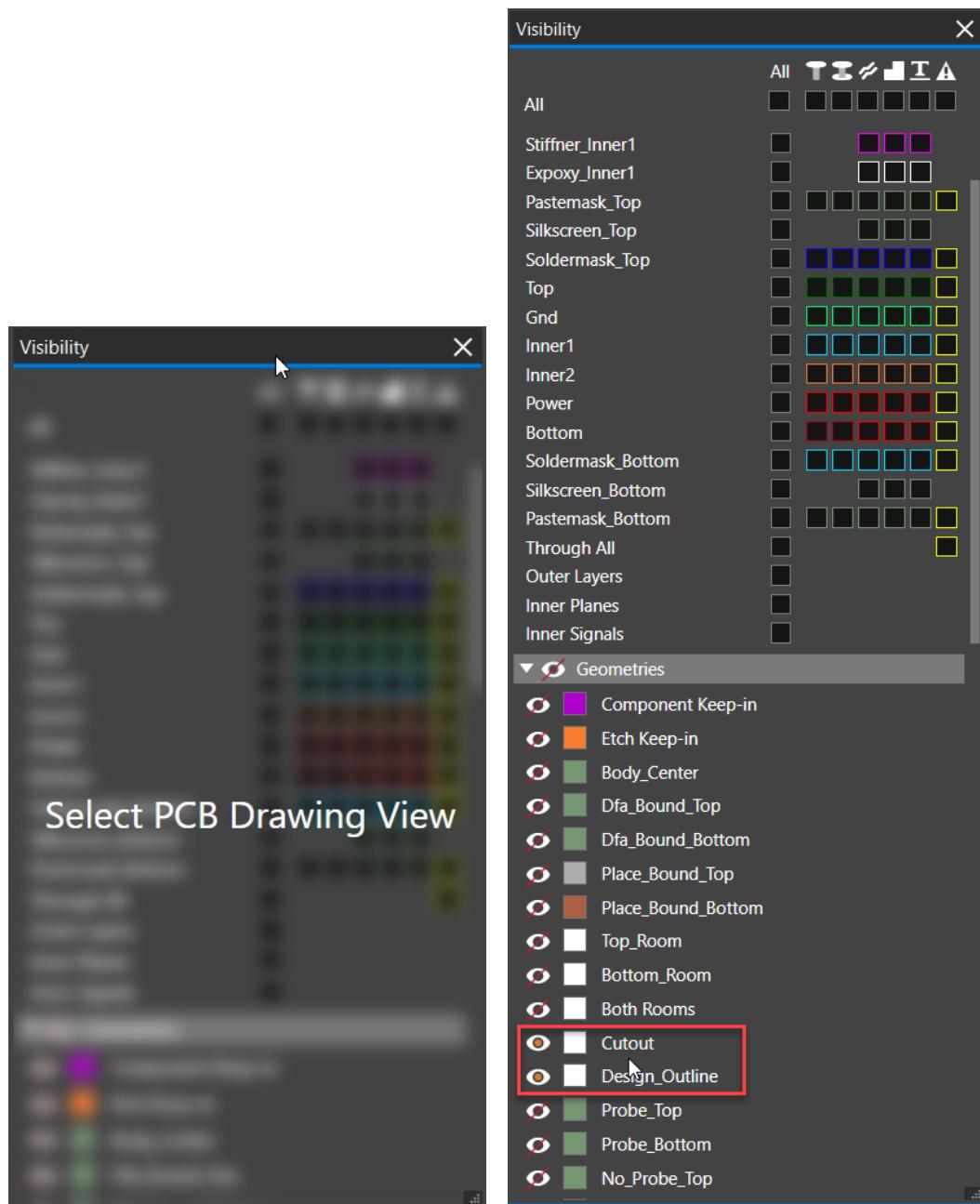
2. Use the left mouse button to pan the PCB view in the specified direction.
3. Use mouse wheel in the forward and backward directions to enlarge or reduce the view.



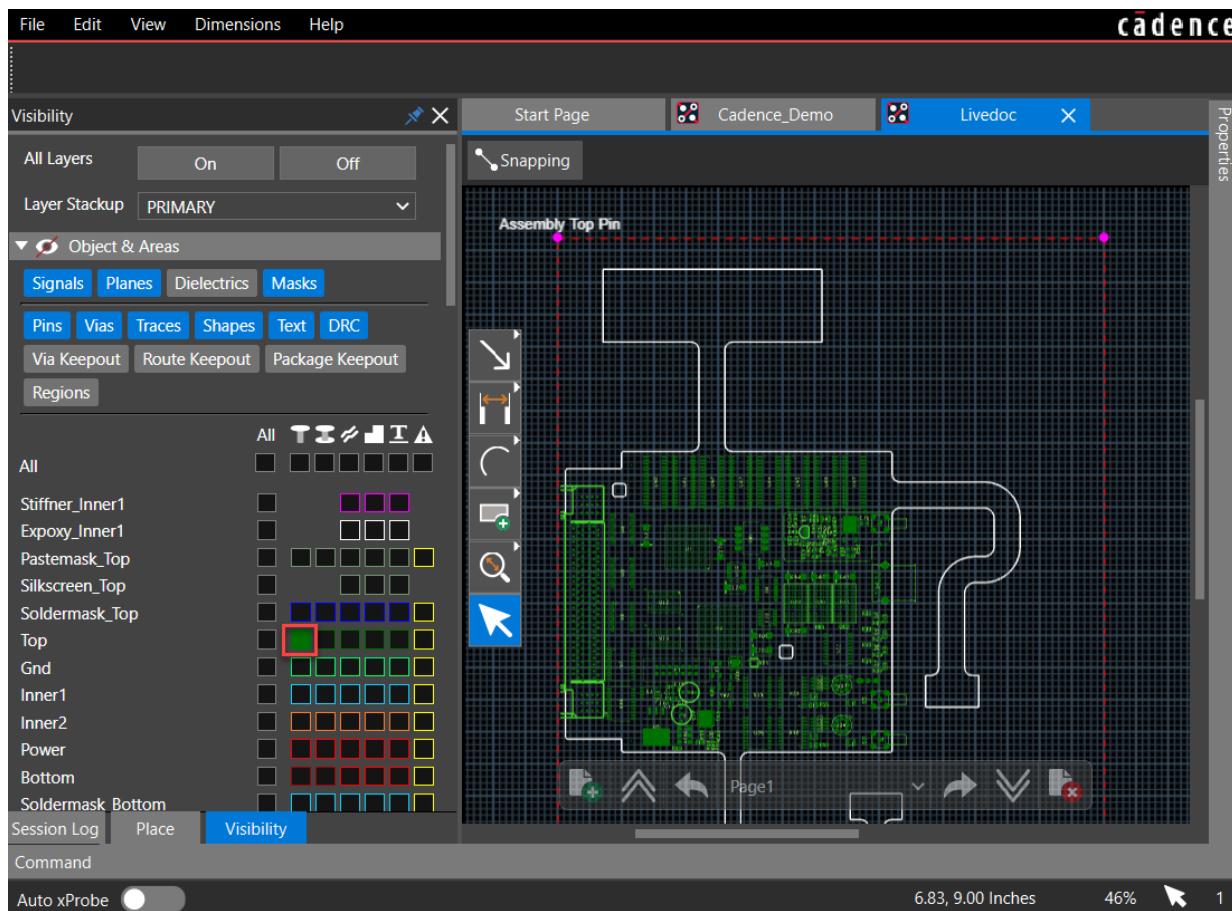
Customizing PCB Views

Any PCB view of the design can be customized by modifying the visibility of the layers in that view. To modify the visibility of layers in a view, follow these steps:

1. Open the *Visibility* panel after placing a PCB view in the LiveDoc canvas. When nothing is selected, the panel is inactive.
2. To change the visibility of any layer, select the PCB view from the LiveDoc canvas and dock the panel to the LiveDoc window. The visibility of all the layers is set to off. Only the valid subclasses based on the selected PCB view are enabled in the *Visibility* panel under the *Geometries* and *User Defined* sections of *Visibility* panel. For example, the following images illustrate the states of the *Visibility* panel of the *Assembly Top* view before and after it is selected in the LiveDoc canvas.



3. In the *Visibility* panel, enable objects on any layer and modify the visibility of the relevant subclasses.
For example, the following image illustrates a custom PCB view *Assembly Top Pin* created from the *Assembly Top* view with pins enabled for the *TOP* layer.



Exporting LiveDoc Data to PDF

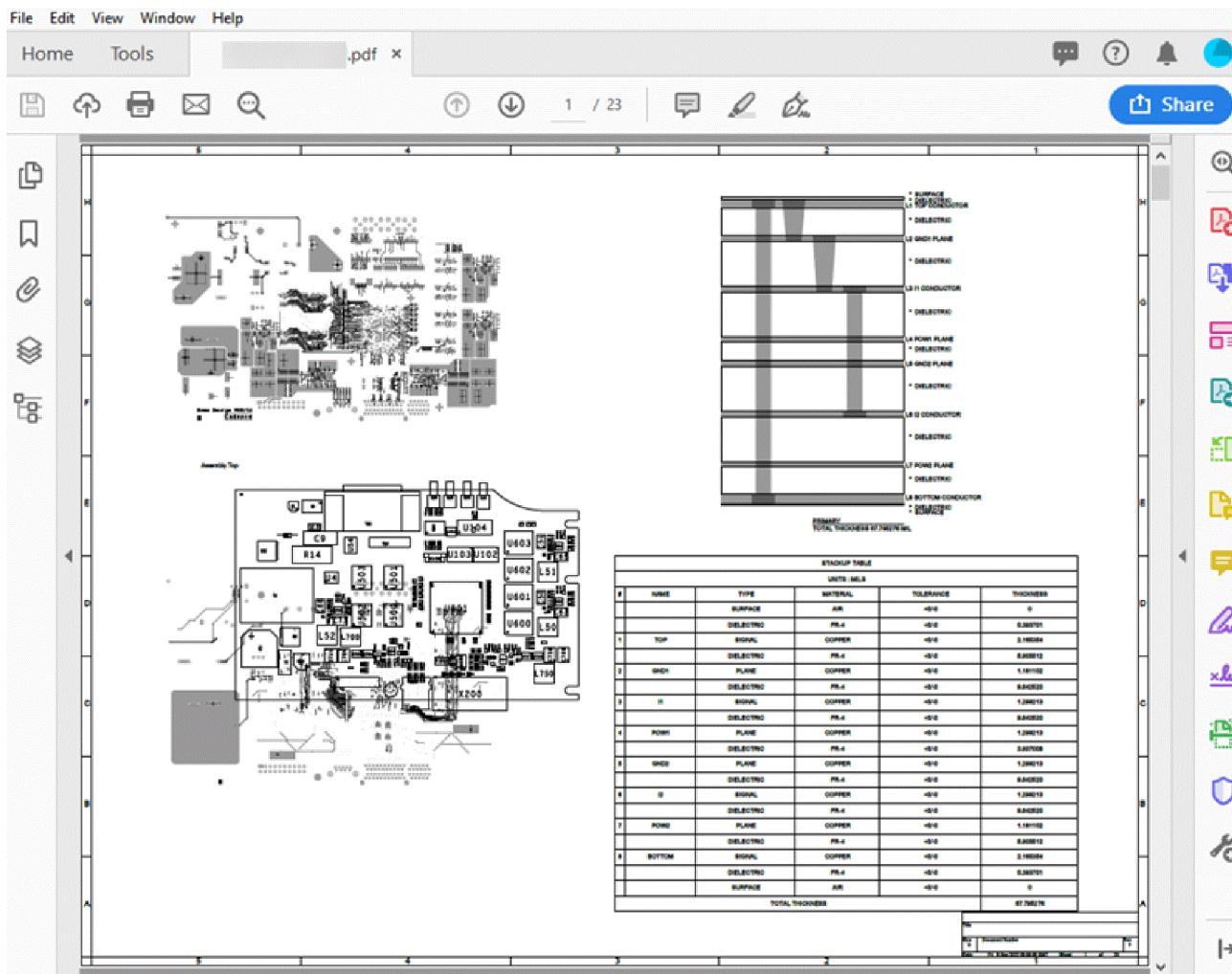
After placing all the default and custom PCB views, you can export Livedoc to a smart PDF format.

- Choose *File – Export PDF* to generate PDF output.

This multi-page single PDF file can be opened in a web browser and shared for review and analysis with manufacturing and assembly teams.

OrCAD X Presto User Guide

Documenting PCB with LiveDoc in OrCAD X Presto--Viewing and Sharing PCB Views in LiveDoc

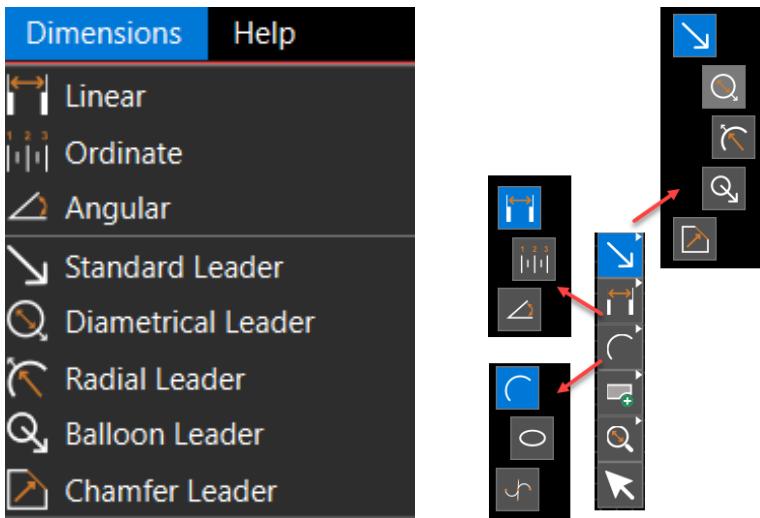


Related Topics

- [What is LiveDoc?](#)
- [Launching LiveDoc](#)
- [LiveDoc User Interface](#)
- [LiveDoc Preferences](#)
- [Viewing and Sharing PCB Views in LiveDoc](#)
- [LiveDoc Workflow](#)
- [Adding a Page in LiveDoc](#)
- [Defining Page Title Blocks](#)
- [PCB Views in LiveDoc](#)
- [Placing PCB Views in LiveDoc](#)
- [Defining Dimensions in LiveDoc](#)
- [Defining Annotations in LiveDoc](#)
- [Managing LiveDoc](#)

Defining Dimensions in LiveDoc

You can add dimensions and leaders to objects in a PCB view to indicate the lengths, sizes, and angles of the object outline or the distance between the selected objects. Choose a dimension from the Dimensions menu or the command toolbar.



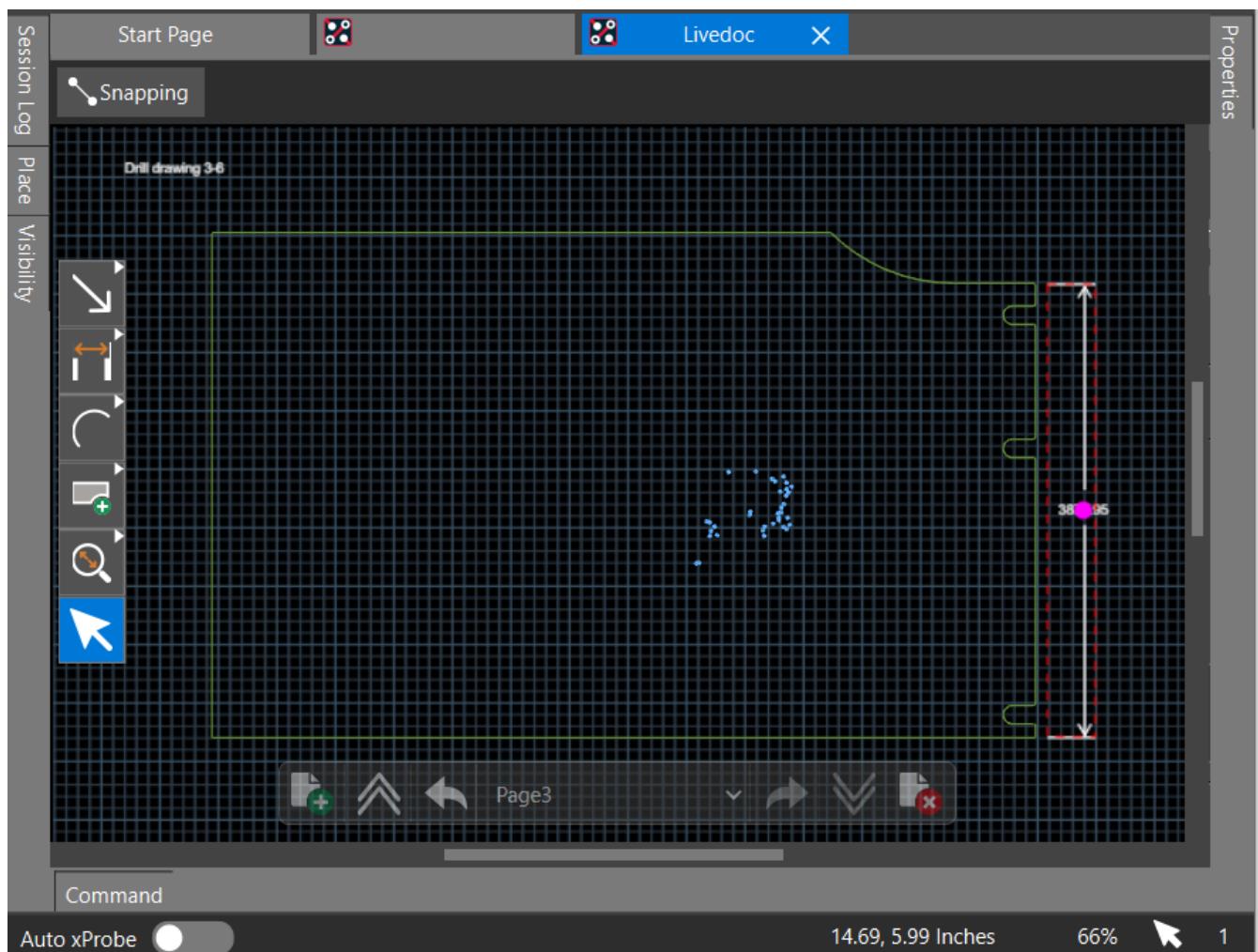
You can add linear, ordinate, and leader dimensions. LiveDoc supports the same dimensioning capabilities as OrCAD X Presto.

Adding Linear Dimensions

You can add linear dimensions to object outlines or between two points. To add linear dimensions, follow these steps:

1. Add a new page in the LiveDoc canvas using the page toolbar.
2. Click to select a PCB view from the *Place* panel and add it to the LiveDoc canvas.
3. To zoom into a location where you want to add dimension, press and hold the `Ctrl` key and move the mouse wheel in the forward direction.
4. Choose *Dimensions – Linear* or select the *Linear dimension* icon  from the command toolbar.
5. Select the first point.
A white dot is displayed as a guideline, and the dimension is attached to the cursor.
6. Select the second point.

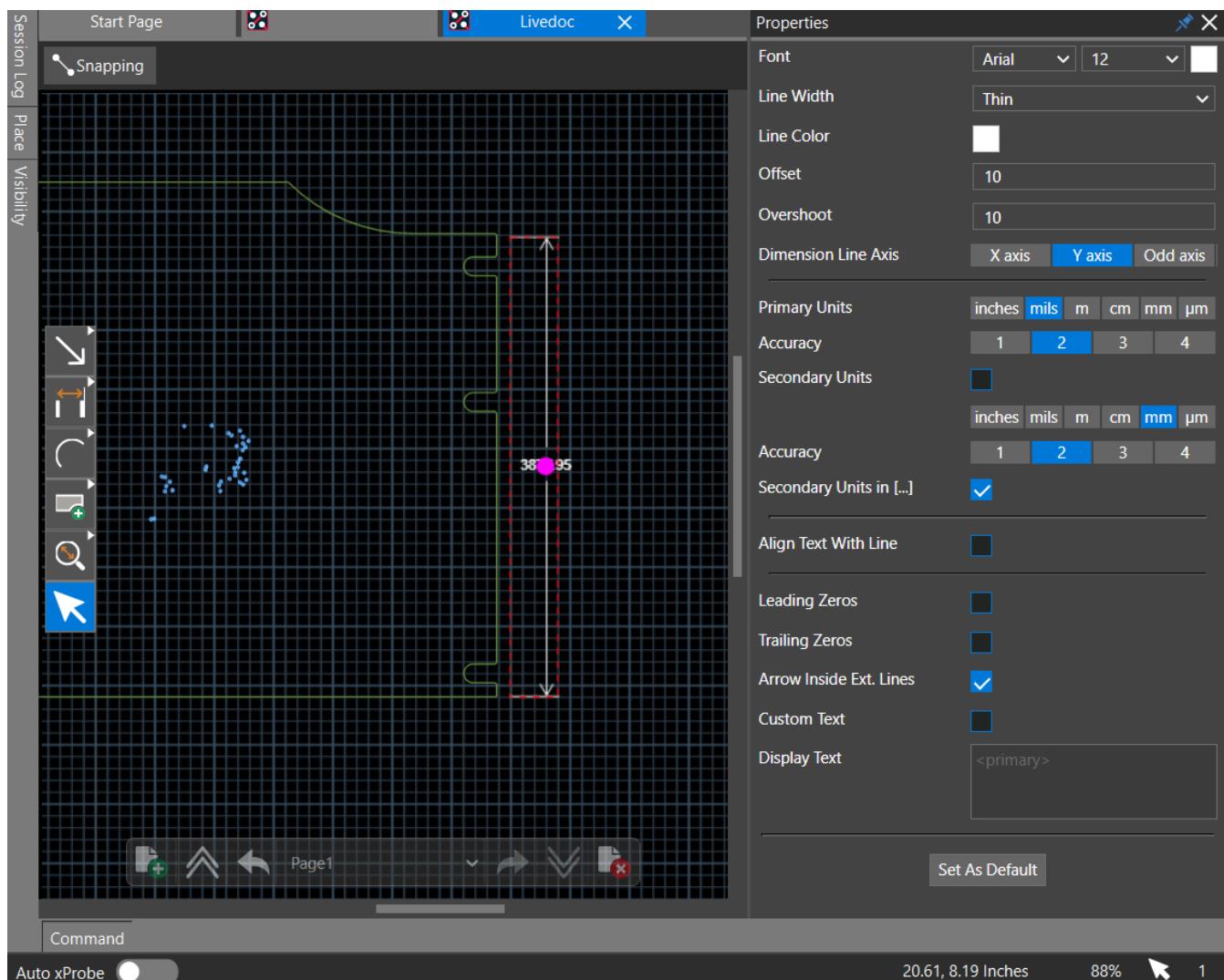
7. Move the cursor to pick a location and click to place the linear dimension.



You can edit dimensioning parameters in the *Properties* panel. To modify a linear dimension, do the following:

1. Click to select a linear dimension in the LiveDoc canvas.

The *Properties* panel displays its properties in blue. A pink dot highlights the dimension value in the canvas.



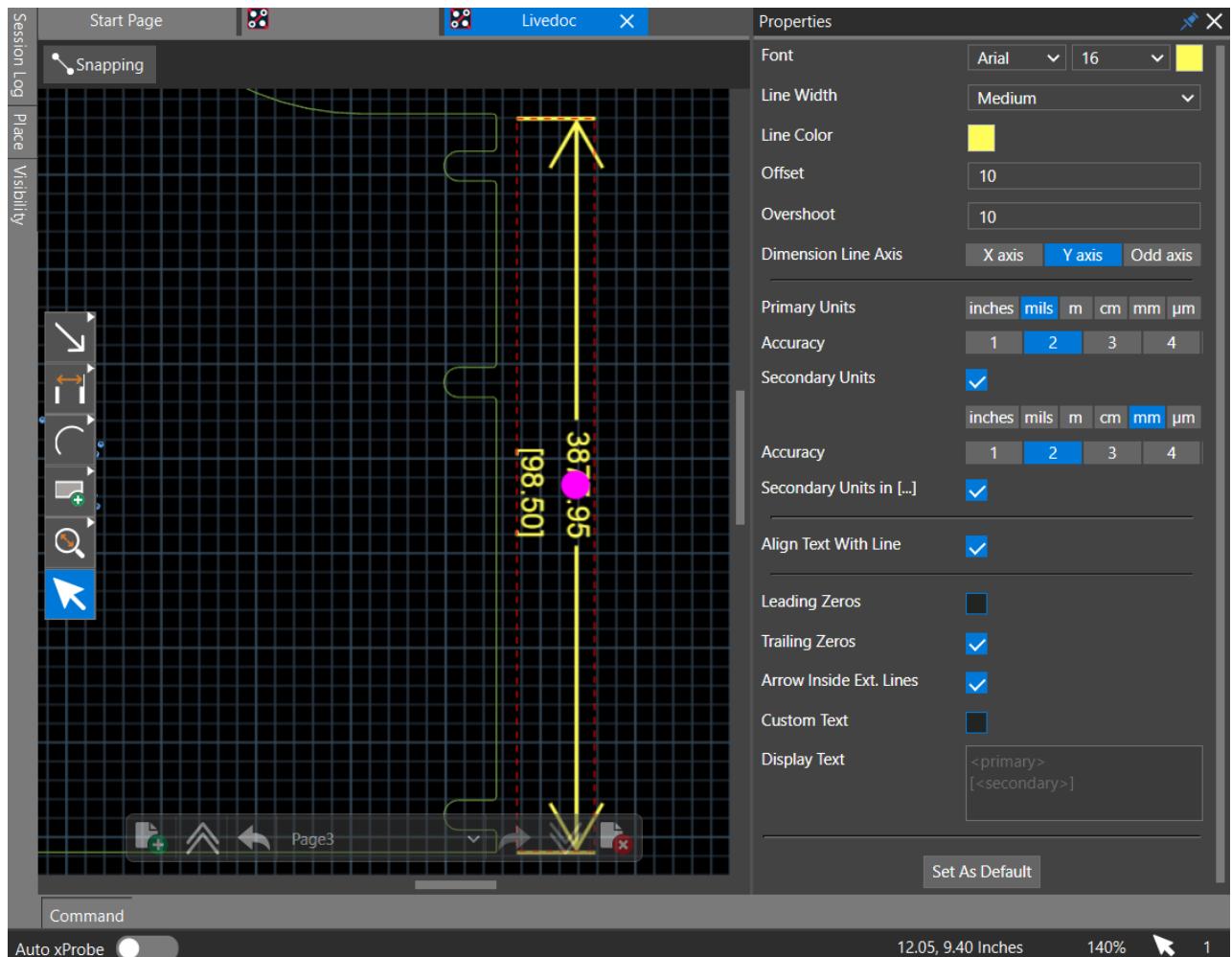
2. In the *Properties* panel, change the following:

- Set the font size to 16 and click the color box for *Font* and choose the yellow color.
The font size of the dimension value increases and is displayed in yellow.
- Change Line Width from *Thin* to *Medium*.
The width of the dimension line increases.
- Click the color box for the *Line Color* and choose yellow from the color palette.
The color of the dimension line changes to yellow.
- Select the *Secondary Units* check box and choose a different unit of measurement to display the dimension value.
The secondary unit starts displaying below the primary units.
- Select the *Secondary Units in the [...]* check box and set the accuracy as needed.

f. Select the *Align Text With Line* checkbox to align the value of the dimension with the line.

g. Select the *Trailing Zeros* checkbox.

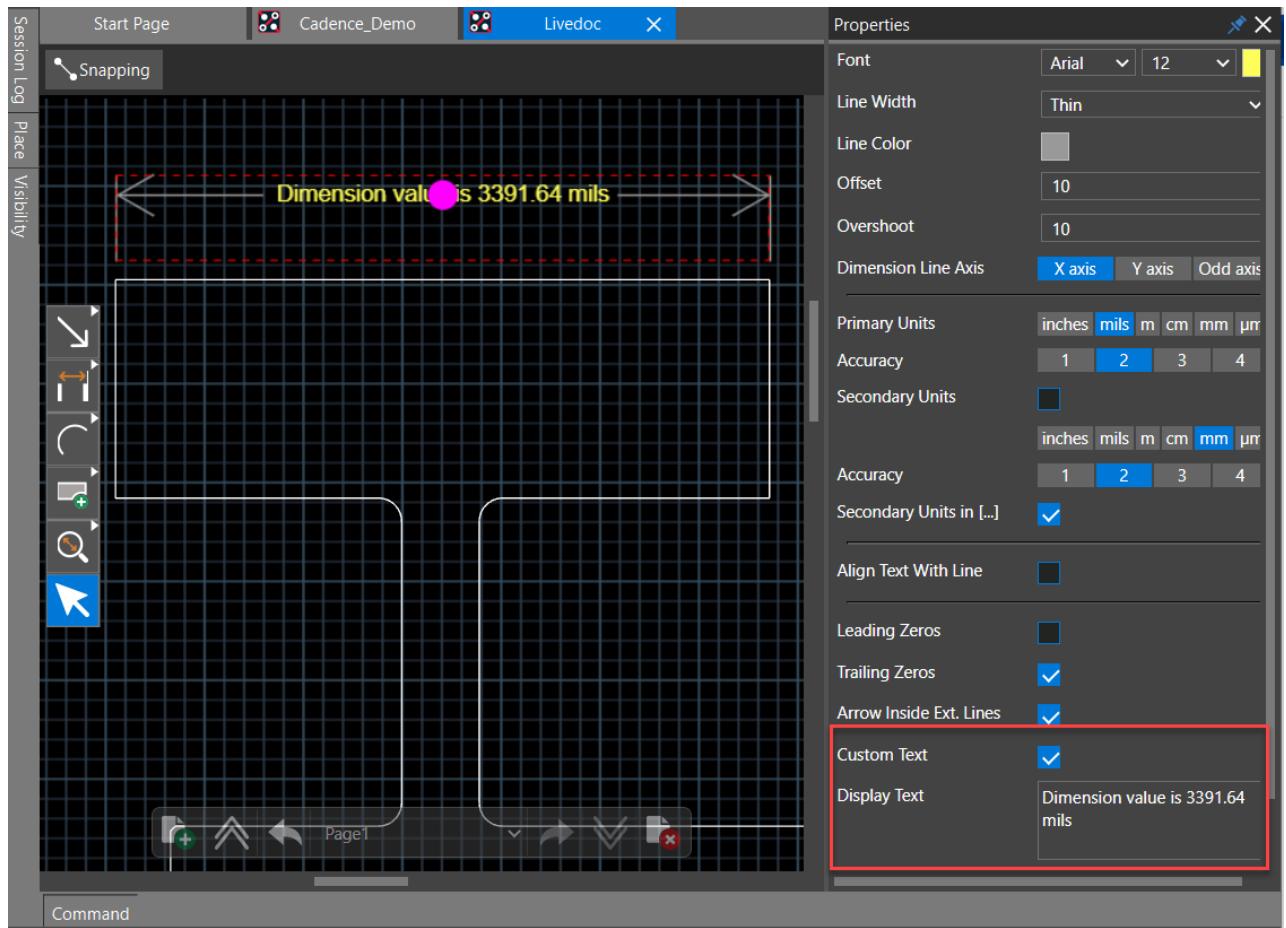
Additional zeros after the decimal are added according to the accuracy set.



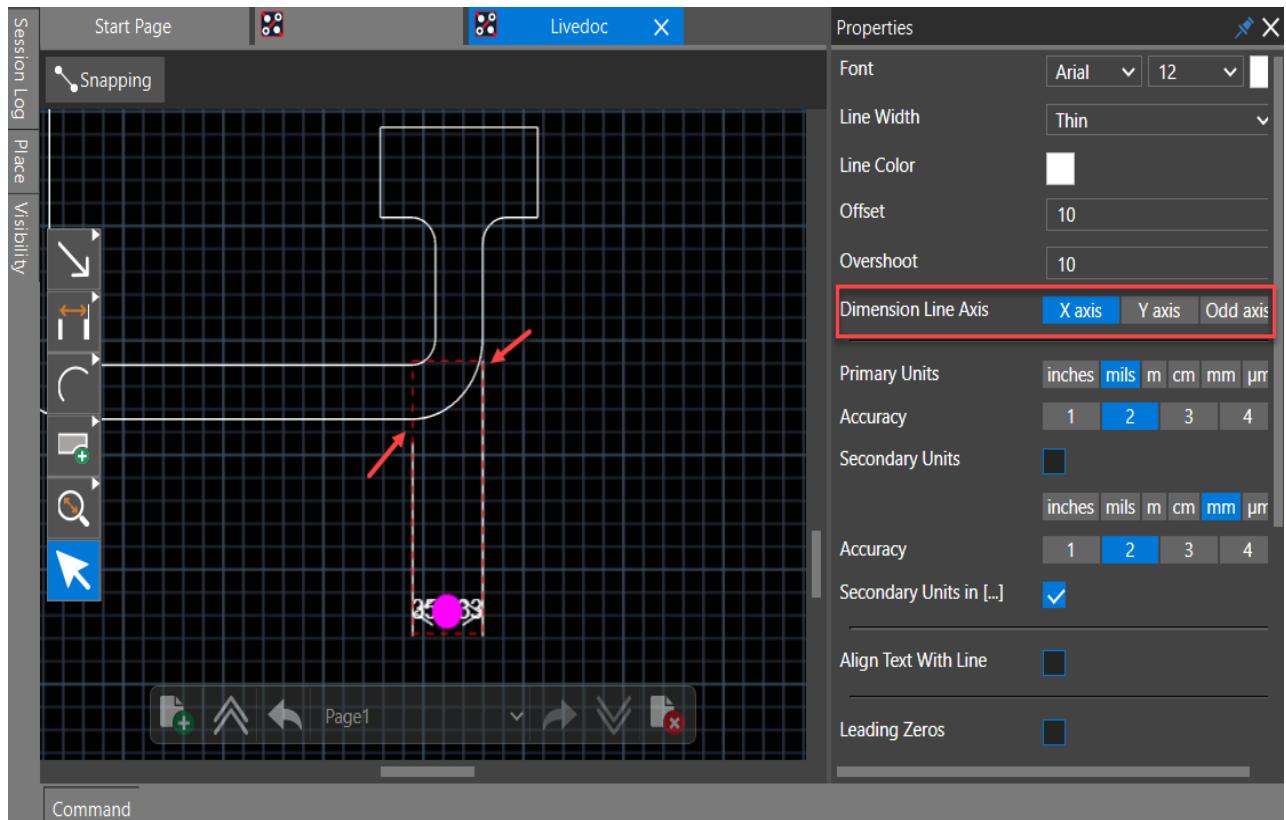
3. Modify dimension text using the following steps:

a. Click the *Custom Text* check box to override the dimension text. You can add prefixes and suffixes to the dimension text. Additional information can also be added in the subsequent lines.

The *Display Text* field becomes active. By default, it displays primary and secondary units.

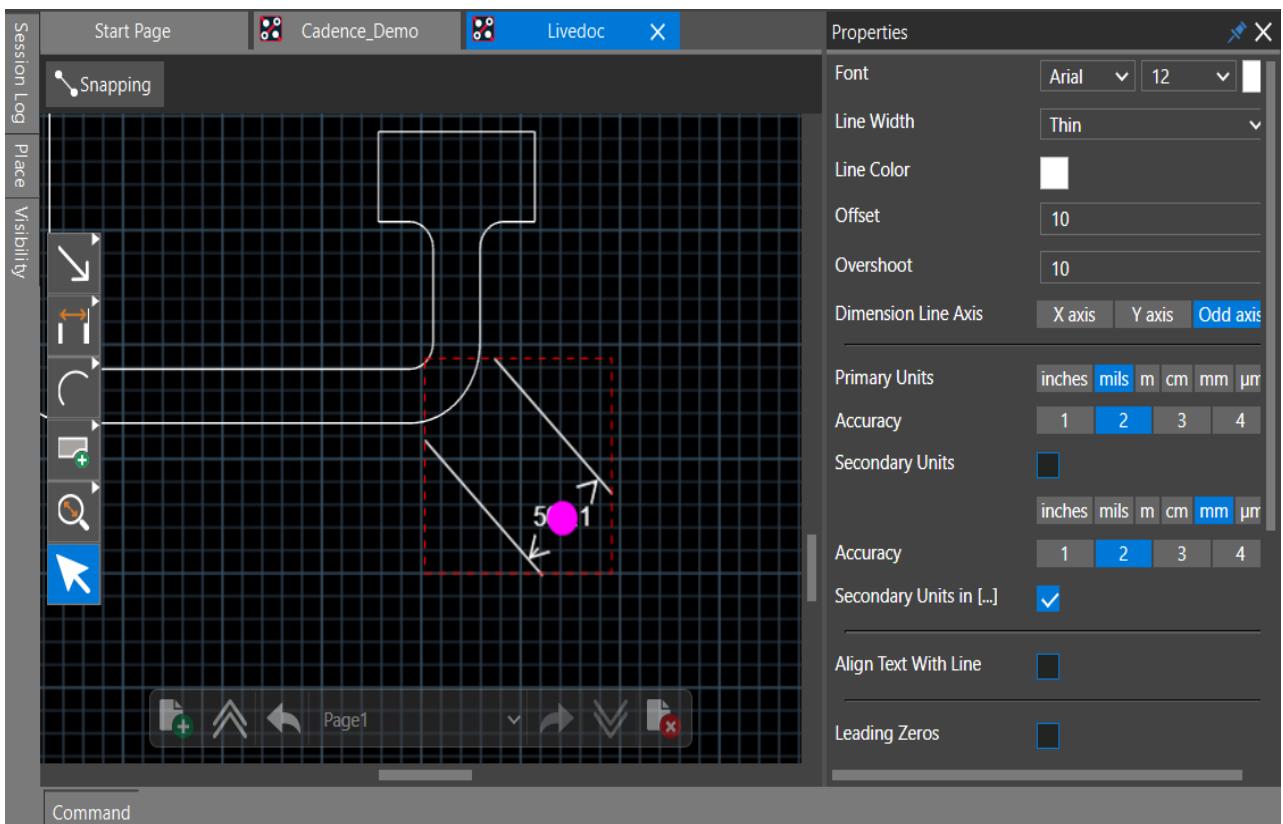


- b. Click *Set as Default* to add new linear dimensions with the same settings.
4. Modify the dimension line axis using the following steps:
 - a. Select two points that are not on the same X-axis coordinates as illustrated in the following image:



- b. Change the *Dimension line axis* to the *Odd Axis*.

LiveDoc calculates and displays the distance between the points along the slanted axis.



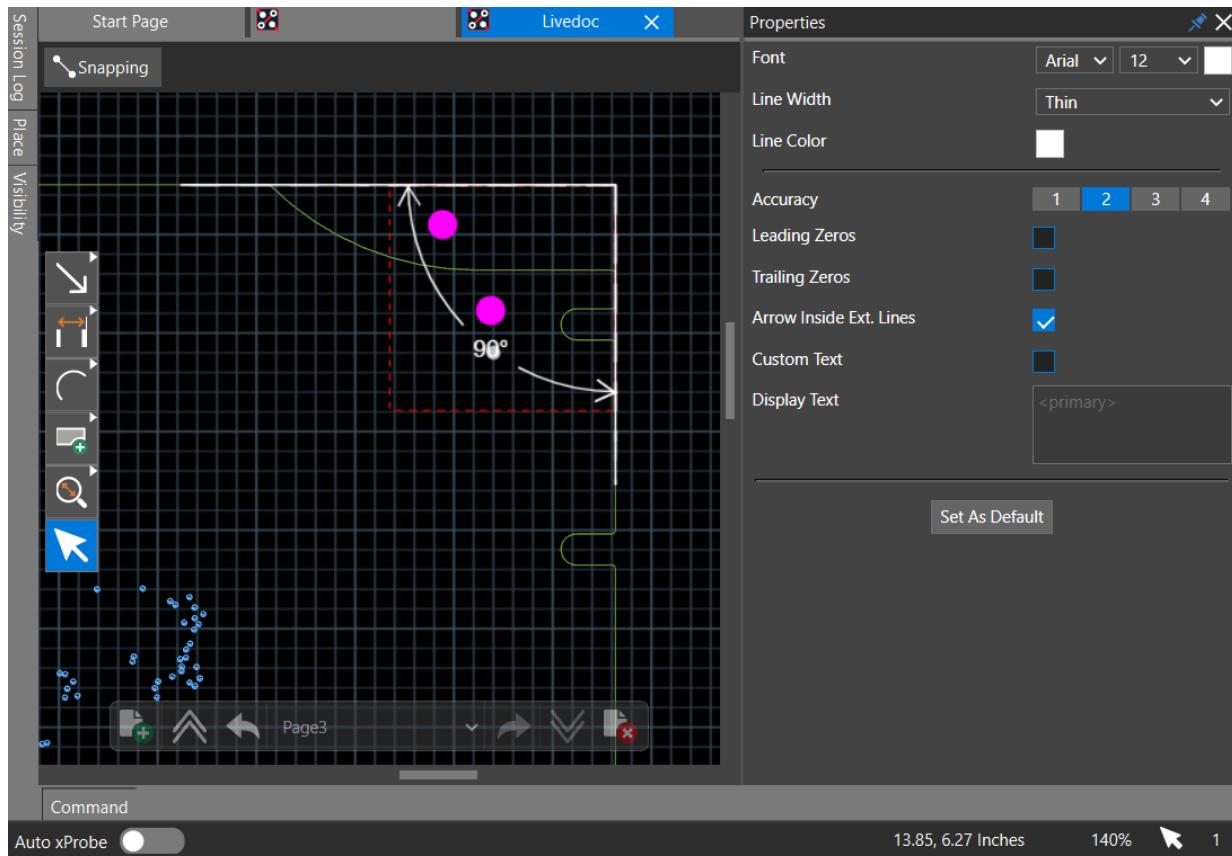
5. To delete a dimension, select it and click the *Delete* button.

Adding Angular Dimensions

Angular dimensioning calculates the angle between two line segments. Angular dimension can be added and modified in the same way as a linear dimension using the following steps:

1. Choose *Dimensions – Angular* or click the  icon from the command toolbar.
2. Click the two line segments and choose a location to place the dimension text.
3. Double-click to select the dimension to modify.

Two pink dots are displayed as guide points for the line and text, respectively.

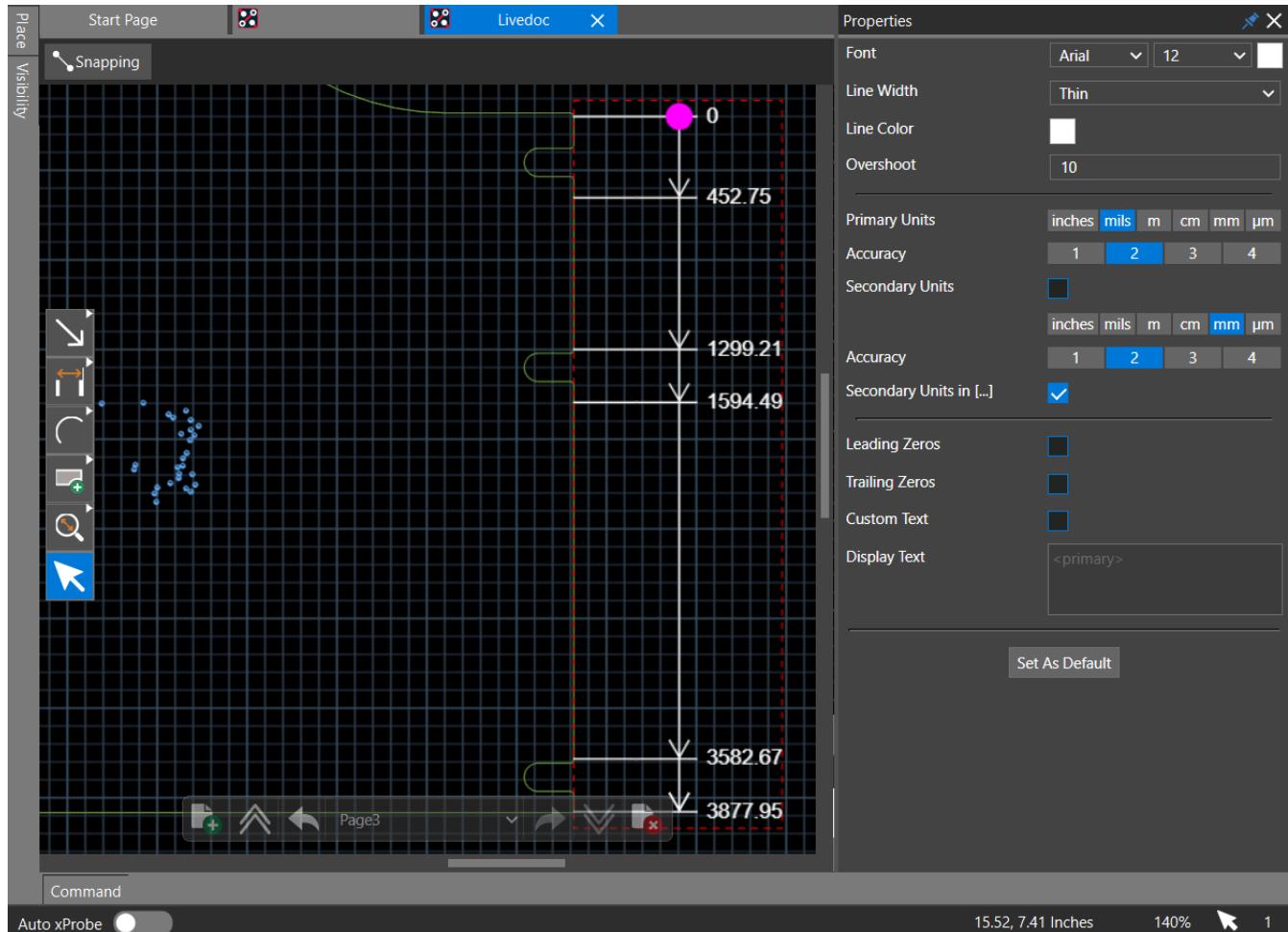


- ✓ Moving the cursor in the inward or outward directions repositions the dimension line. Similarly, moving the cursor along the line repositions the dimension text.

Adding Ordinate Dimensions

The ordinate dimension is a set of multiple linear dimensions that adds dimensions relative to a reference point. The dimensions are placed at the end of a single extension line that can have multiple vertices and is useful to place on the outer edges of a board or object outlines.

1. Choose *Dimensions – Ordinate* or click the  icon from the command toolbar.
2. Select the first dimension point.
A white dot represents the reference point with $x = 0$ and $y = 0$ coordinates.
3. Click to add other dimension points.
4. Double-click to place the dimension.
The other dimensions added are relative to the first point, as illustrated in the following image:



5. To delete the dimension, select it and press the **Delete** button.

Adding Leader-Oriented Dimensions

LiveDoc allows standard leader lines to add a note or dimension to the drawing feature. Leaders are used for adding diametrical and radial dimensions and balloons. Leaders also provide an alternative method for dimensioning chamfers.

LiveDoc provides options to add the following leader types:

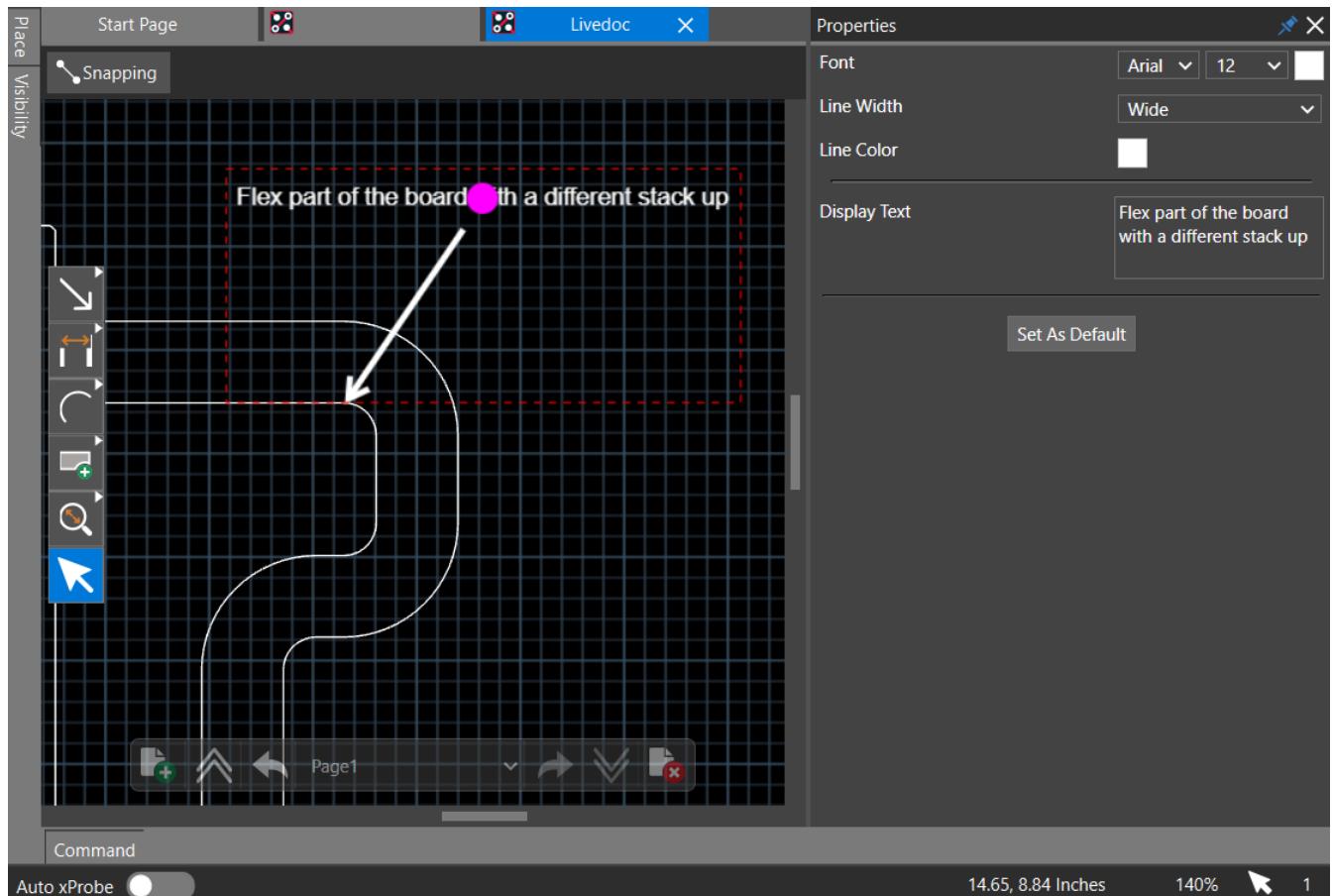
- Standard Leader: Adds standard leader lines with a note.
- Diametrical Leader: Adds dimension to circular objects.
- Radial Leader: Adds dimension to an arc or circle.
- Balloon Leader: Adds a leader line with a balloon at the other end, enclosing an

alphanumeric character string.

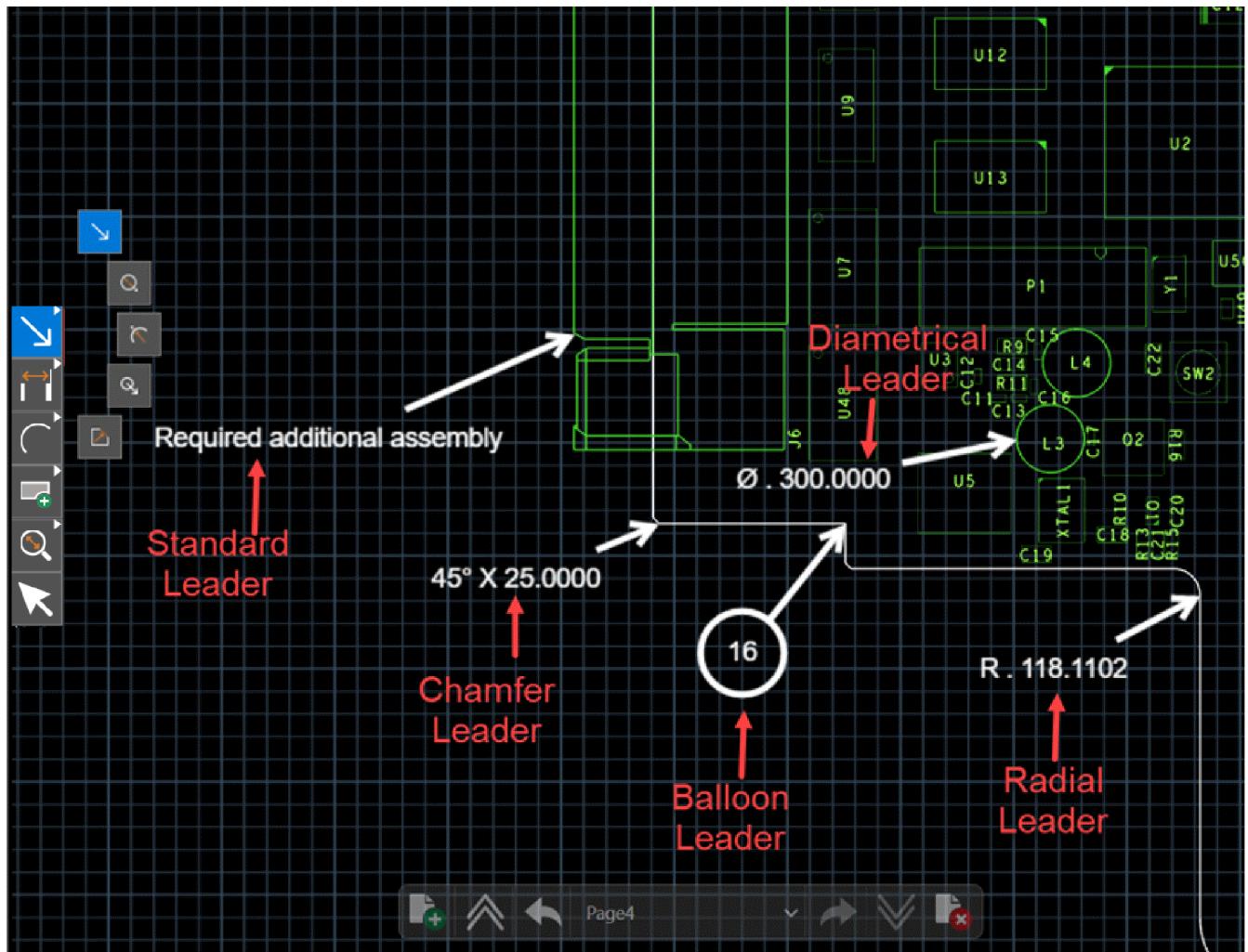
- Chamfer Leader: Adds standard 45-degree chamfer drafting dimensions.

To add leaders in LiveDoc to the objects, do the following:

1. Choose the leader type from the *Dimensions* menu or click a leader icon in the functional toolbar.
2. Position the cursor at a point on an element or in the canvas where you want the leader to point.
3. Click the point on the canvas.
A white dot appears, representing the reference point.
4. Move the cursor and click again to establish the position and length of the leader.
The leader is added to the canvas.
5. Click the *Select* icon  to display leader properties in the *Properties* panel.
6. Modify the leader text in the *Display Text* field in the *Properties* panel.
The *Properties* panel displays the field to modify the default leader text.



7. Select the relevant leader icon from the functional toolbar and repeat steps 2 to 6 to add more leaders.



Related Topics

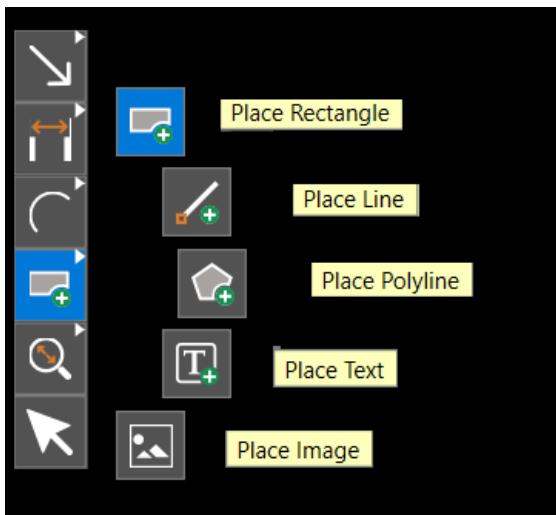
- [What is LiveDoc?](#)
- [Launching LiveDoc](#)
- [LiveDoc User Interface](#)
- [LiveDoc Preferences](#)
- [Defining Dimensions in LiveDoc](#)
- [LiveDoc Workflow](#)
- [Adding a Page in LiveDoc](#)
- [Defining Page Title Blocks](#)
- [PCB Views in LiveDoc](#)
- [Placing PCB Views in LiveDoc](#)
- [Viewing and Sharing PCB Views in LiveDoc](#)
- [Defining Annotations in LiveDoc](#)
- [Managing LiveDoc](#)

Defining Annotations in LiveDoc

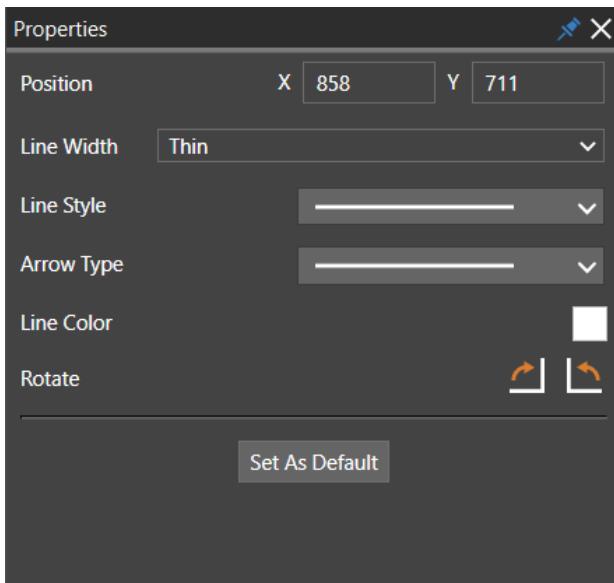
You can point out a specific location of the design, add text, or custom images using graphical tools available in LiveDoc. Various drawing tools are available in LiveDoc to do the following tasks:

- Explicitly draw lines, polylines, and rectangles to create specific assembly instructions or to add inspection notes.
- Specify materials, processes, tolerances, or other design requirements saved as LiveDoc data.

You can place various graphical objects from the functional toolbar, as illustrated in the following image:



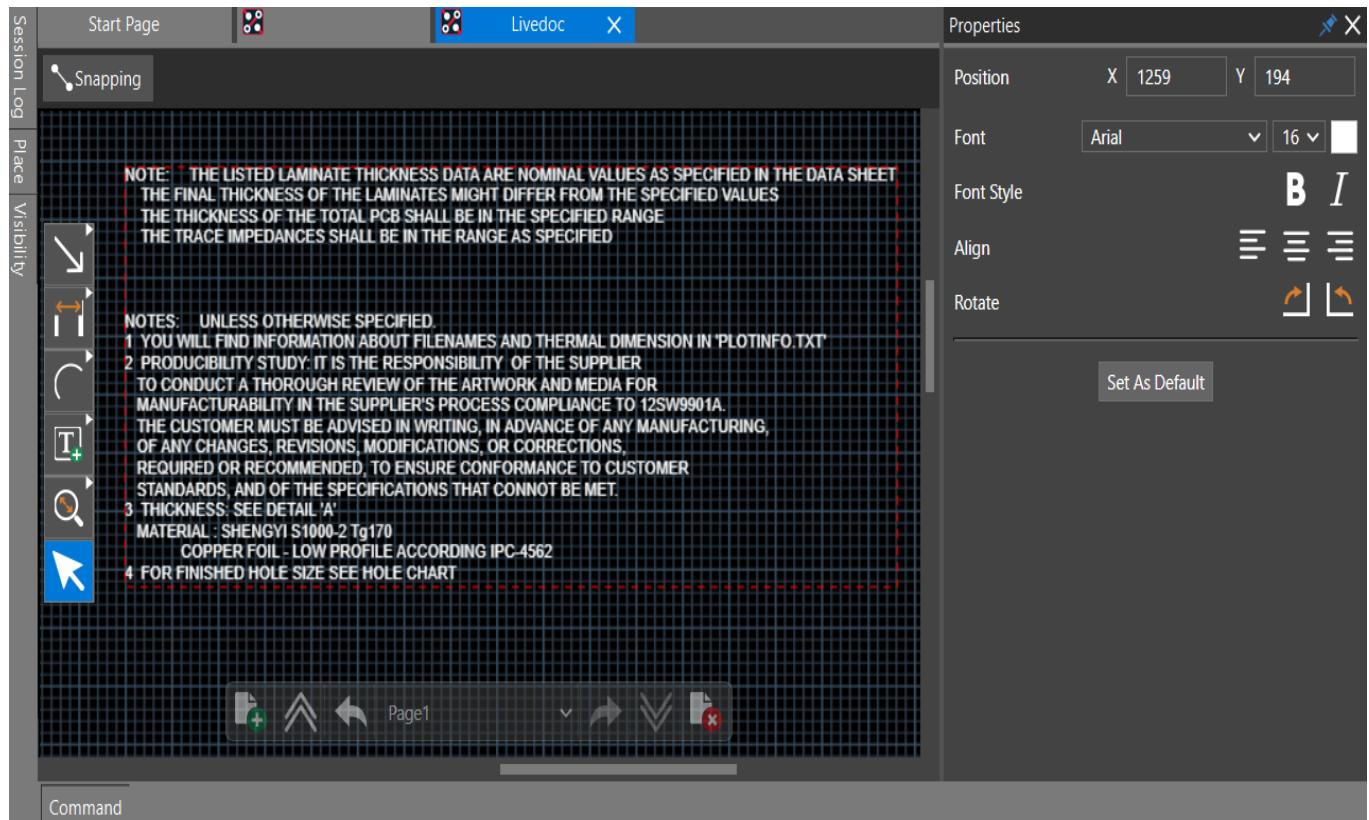
The *Properties* panel displays the attributes of the drawing object when selected in the LiveDoc canvas. You can modify the line width, style, color, and orientation of a rectangle, polyline, or line object.



A text string can be placed directly into the LiveDoc canvas using the *Place Text* icon from the command toolbar. You can add fabrication notes to the manufacturer by using the text option.

To place text in LiveDoc, do the following:

1. Select the *Place Text* icon from the command toolbar.
 2. Click to choose a location in the LiveDoc canvas to place the text.
 3. Type the text string and click anywhere in the LiveDoc canvas.
 4. Select the text using the select icon to modify the text's appearance, such as font, style, alignment, and orientation.
- The attributes of the selected text string are displayed in the *Properties* panel and can be modified as required.



Related Topics

- [What is LiveDoc?](#)
- [Launching LiveDoc](#)
- [LiveDoc User Interface](#)
- [LiveDoc Preferences](#)
- [Defining Annotations in LiveDoc](#)
- [LiveDoc Workflow](#)
- [Adding a Page in LiveDoc](#)
- [Defining Page Title Blocks](#)
- [PCB Views in LiveDoc](#)
- [Placing PCB Views in LiveDoc](#)
- [Viewing and Sharing PCB Views in LiveDoc](#)
- [Defining Dimensions in LiveDoc](#)
- [Managing LiveDoc](#)

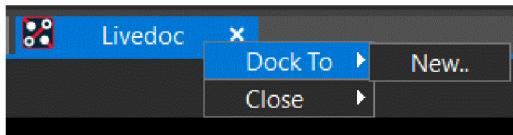
Managing LiveDoc

LiveDoc data is saved with the board design and can be accessed in multiple ways.

Moving LiveDoc Tab to a New Workspace

LiveDoc can work independently of the OrCAD X Presto user interface. The LiveDoc tab can be moved to a new workspace. You can utilize this option to work simultaneously in the OrCAD PCB Editor X and LiveDoc windows in a multi-monitor setup. To switch LiveDoc workspace, do the following:

1. Right-click the *Livedoc* tab and choose *Dock To – New..*.



The new workspace opens in a separate window, and the title bar displays the workspace ID.

2. To move back to the main workspace, choose *LiveDoc – Dock To – MainWorkSpace*.



Closing LiveDoc

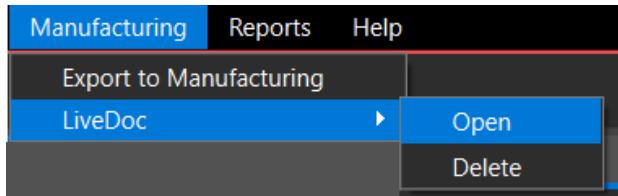
When you close the LiveDoc tab, the LiveDoc data is saved with the database. You can close the LiveDoc tab in one of the following ways:

- Click the cross button on the LiveDoc tab.
- Right-click the LiveDoc tab and choose *Close Tab*.
- Close the board design tab.

Reopening LiveDoc

To reopen an existing LiveDoc view:

- Choose *Manufacturing – LiveDoc – Open*.



Deleting LiveDoc

LiveDoc associated with the board design file (.brd) can be deleted from the active board design.

- Choose *Manufacturing – LiveDoc – Delete*.

Related Topics

- [What is LiveDoc?](#)
- [Launching LiveDoc](#)
- [LiveDoc User Interface](#)
- [LiveDoc Preferences](#)
- [Using LiveDoc Templates](#)
- [LiveDoc Workflow](#)
- [Adding a Page in LiveDoc](#)
- [Defining Page Title Blocks](#)
- [PCB Views in LiveDoc](#)
- [Placing PCB Views in LiveDoc](#)
- [Viewing and Sharing PCB Views in LiveDoc](#)
- [Defining Dimensions in LiveDoc](#)
- [Defining Annotations in LiveDoc](#)

Reviewing and Analyzing Designs in OrCAD X Presto

- Analyzing Design in OrCAD X Presto 3D Environment
- Reviewing Designs Using Comments and Markups in OrCAD X Presto
- Generating Reports in OrCAD X Presto

Analyzing Design in OrCAD X Presto 3D Environment

- Generating 3D Exports in OrCAD X Presto
- Overview of the OrCAD X Presto 3D Environment

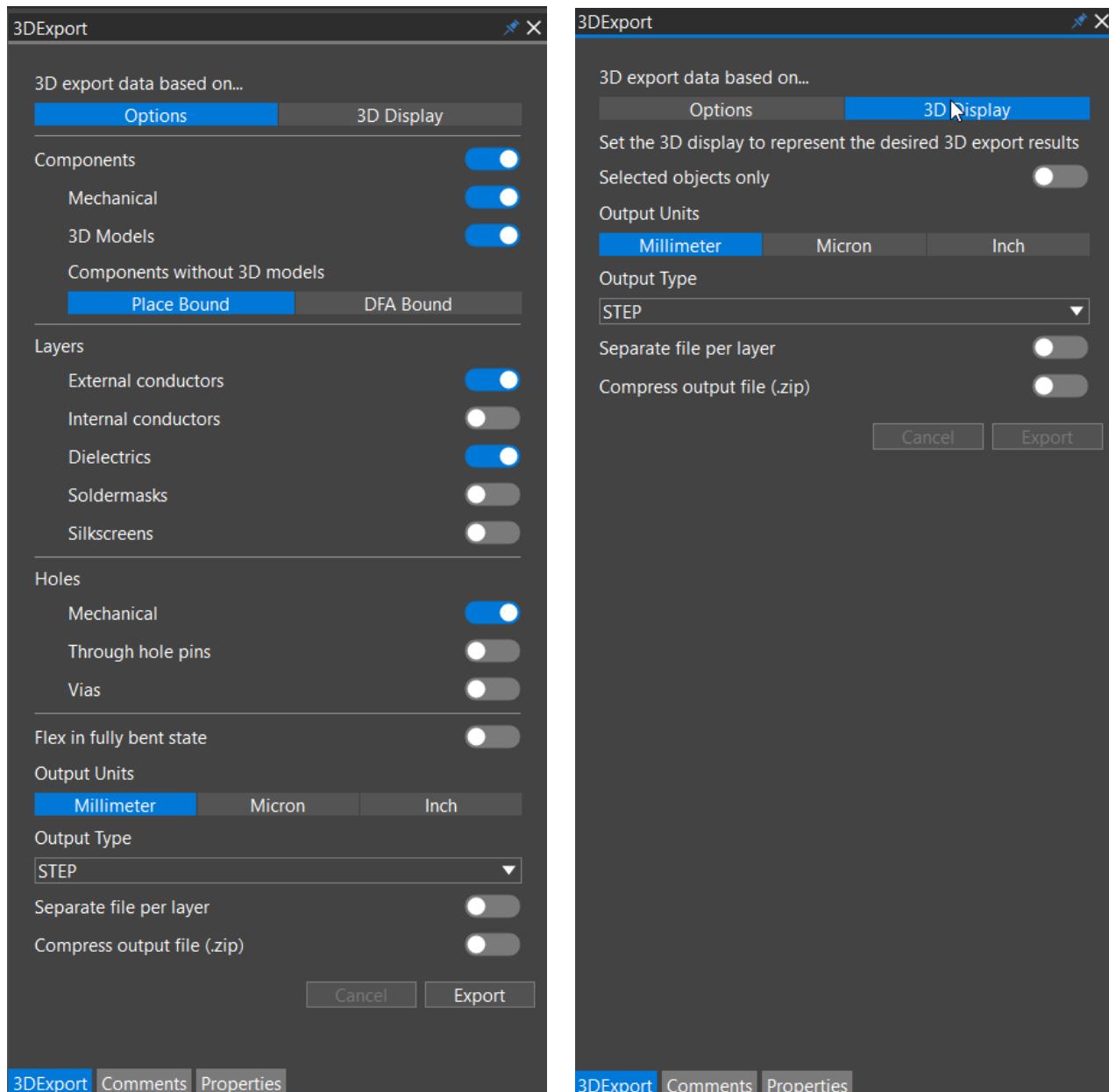
Generating 3D Exports in OrCAD X Presto

For ECAD and MCAD analysis, it is often required to export 3D data of board designs. OrCAD X , Presto, provides option to export 3D design data in multiple output formats. The 3D design data can also be exported for rigid-flex designs when the design is folded or bent. The export option is fully customizable and exports the data depending on the visibility of the straight or foldable state of the design. To generate 3D exports in OrCAD X Presto, do the following:

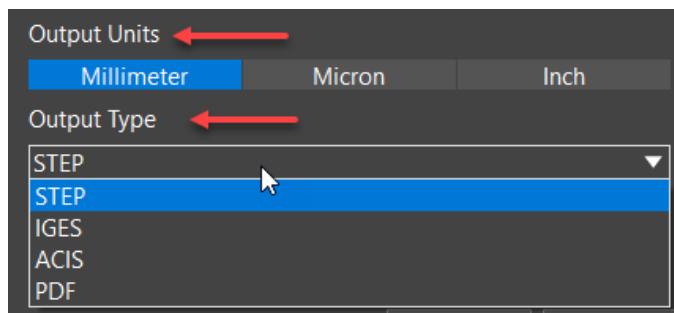
1. Choose *File – 3D Export*.
A docked *3DExport* panel opens in a separate tab at the location of the *Properties* panel.
2. Modify the options for 2D and 3D display in the *Options* and *3D Display* tab. The *3D Display* tab is functional only when the display mode is set to 3D in the *Visibility* panel.

OrCAD X Presto User Guide

Reviewing and Analyzing Designs in OrCAD X Presto--Analyzing Design in OrCAD X Presto 3D Environment



3. Enable *Selected objects only* to output only the objects selected in the design canvas.
4. Select the output units and the output file format.



5. Click *Export*.

A file browser opens.

6. Specify the name and path of the output file and click the *Save* button.

The output file contains the 3D representation of the design based on the visibility settings in the specified directory.

 You cannot modify the display while the 3D export is in process.

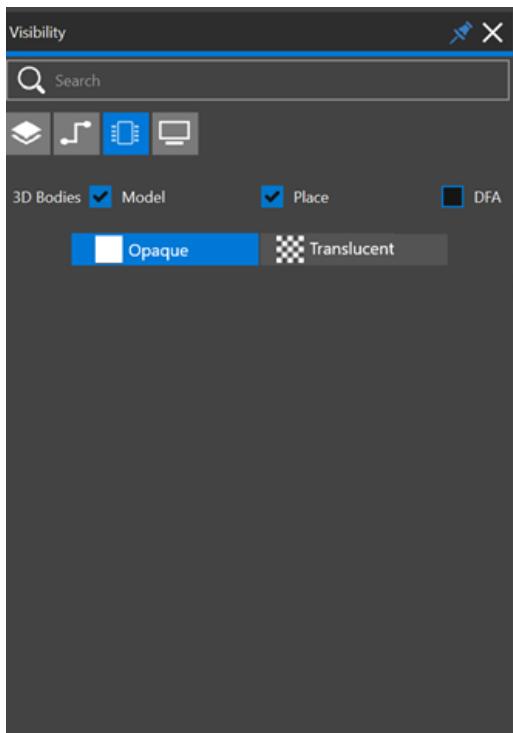
Overview of the OrCAD X Presto 3D Environment

The OrCAD X Presto 3D environment enables you to view and edit footprints, and includes the following 3D capabilities:

Component Shape Visibility

With the *Components* tab of the *Visibility* panel, you can control the visibility state of the Place Bound or DFA Bound extrusions and mapped 3D models, and toggle between the Opaque and Translucent states for enabled representations, as follows:

Visibility Panel Components Tab



Footprint Boundaries

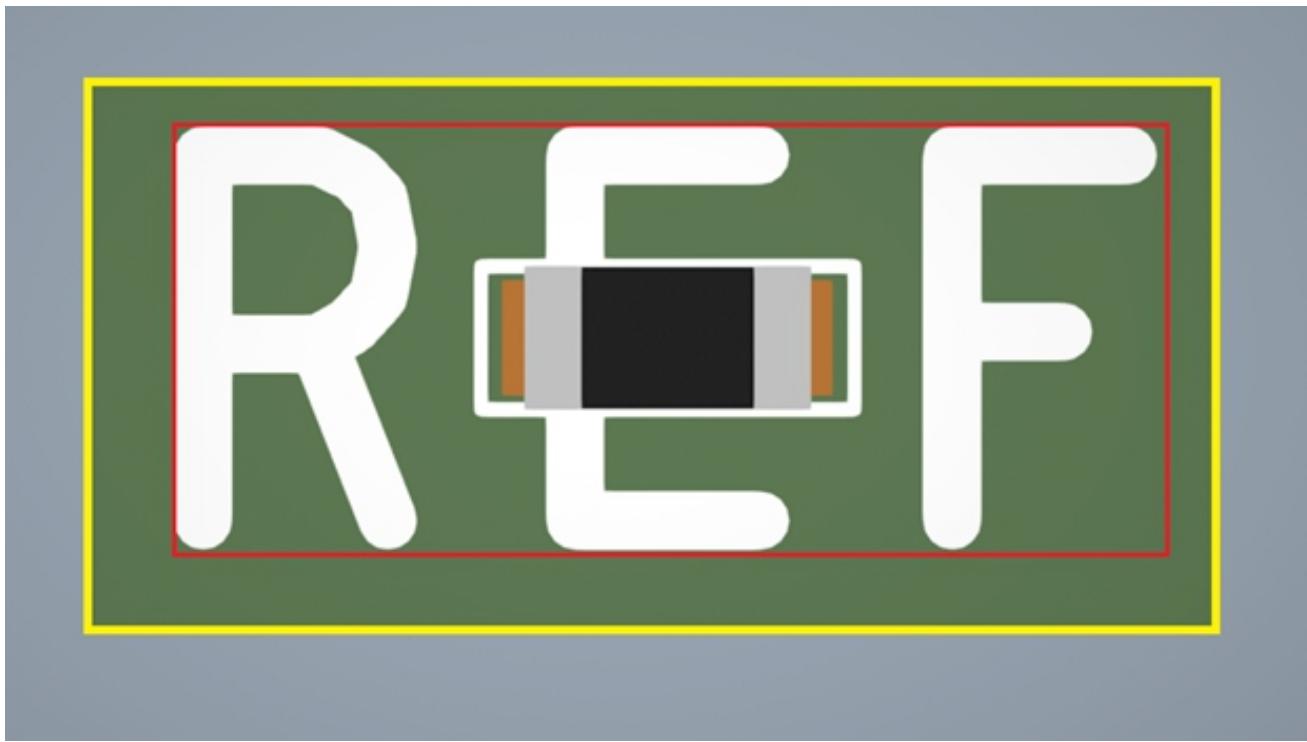
To accurately display the positive material of layers such as solder mask and dielectric, the 3D view requires an overall design boundary. When used for board-level visualization the overall design boundary is taken from the design. However, footprints lack an explicit design boundary.

The overall 2D box extent for the footprint objects, plus an additional apron, will be computed

dynamically and used as the footprint design boundary.

As objects (pins, shapes, and so on) are added, removed, or modified, the boundary is updated to use the *minimum required area* to include all relevant 3D footprint objects. For example, if a silkscreen shape is added outside the current boundary then the boundary will be enlarged just enough to include it.

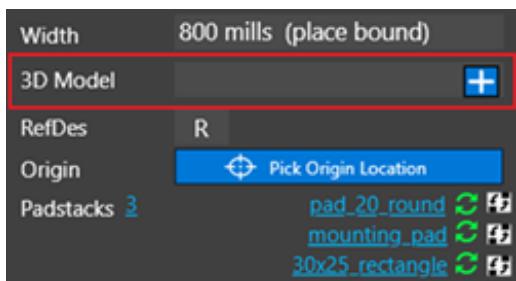
In the following example, the computed 2D extent is outlined in red, while the overall 2D extent (computed + apron) is outlined in yellow.



3D Models

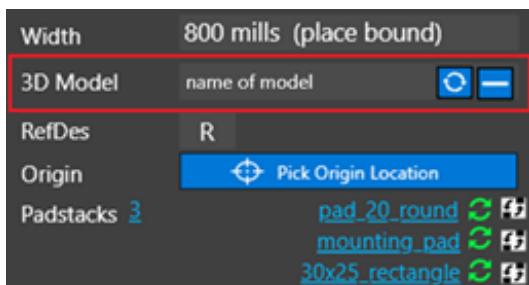
You can control the mapped 3D model for the footprint through the top-level *3D Model* field in the property panel. When there is no existing 3D model mapped to the footprint:

1. The text portion of the “3D Model” field will appear empty and you can use the plus (+) icon button to import a new 3D model for the footprint, as follows:



1. Click the plus icon to open a file dialog and browse to the desired 3D model file.
2. Choose a model file, and it will be imported to the drawing at the drawing origin location, and the 3D view will update to display the model.
 - o Any errors or warnings during this procedure will appear as messages in the session log.

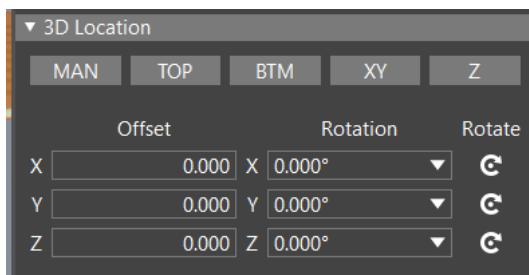
When mapping an existing 3D model to the footprint, the text portion of the *3D Model* field displays the name of the associated model, and you will see two icon buttons, as follows.



1. Use the “refresh” icon to browse to a 3D model file, and import the chosen model while keeping the existing 3D model mapping transform.
2. Use the minus (-) icon to remove the 3D model from the footprint and any mapping transform information for the model.

To edit the 3D model mapping, use the *3D Location* top-level accordion group in the property panel when the footprint has an associated 3D model:

- The mapping transform tools group is visible when 3D is the active view and the footprint has an associated 3D model. Otherwise, you will not see the *3D Location*.



- No active selection is required to perform the mapping operations (for example, alignment, rotation, and so on).
- Other editing operations (text fields, rotate buttons, alignment, and so on) will be similar to those for board-level mechanical model mapping with 3DX.

Reviewing Designs Using Comments and Markups in OrCAD X Presto

Adding comments using markups is a simple design review feature streamlining the PCB design review process across different design teams or domains. The design review solution, integrated within the layout editor, eases the collaboration with assembly and fabrication houses, manages review data efficiently, and keeps it up to date.

This design review functionality works in a single-user mode. You can mark any area in the design canvas using rectangular or arrow markups and add text comments in the *Comments* panel. Each comment is saved with a unique ID. You can add text comments to individual layers with or without markup. A comment may have more than one markup associated with it.

To view all the markups, zoom in to fit the design in the canvas. You can navigate comments in the *Comments* panel or by selecting the markups in the design canvas. The layout editor retains the layer visibility and zoom info when navigating comments or markups.

The markup data is saved with the design database and transferred to other users when the design is shared for task assignments or exchange of information. Using this design review functionality, multiple users can easily collaborate and work on design iterations.

Operations on the design objects are not permitted while working with markups and comments. To work on design objects, click the select icon from the floating toolbar. You can switch between markup and design editing modes by selecting relevant icons.

 The markup functionality is not supported with non-opengl mode, 3D viewing mode, or collaborative design mode (team design and cloud).

- [Adding Comments and Markups in OrCAD X Presto](#)
- [Editing Comments and Markups in OrCAD X Presto](#)
- [Viewing and Navigating Comments and Markups in OrCAD X Presto](#)

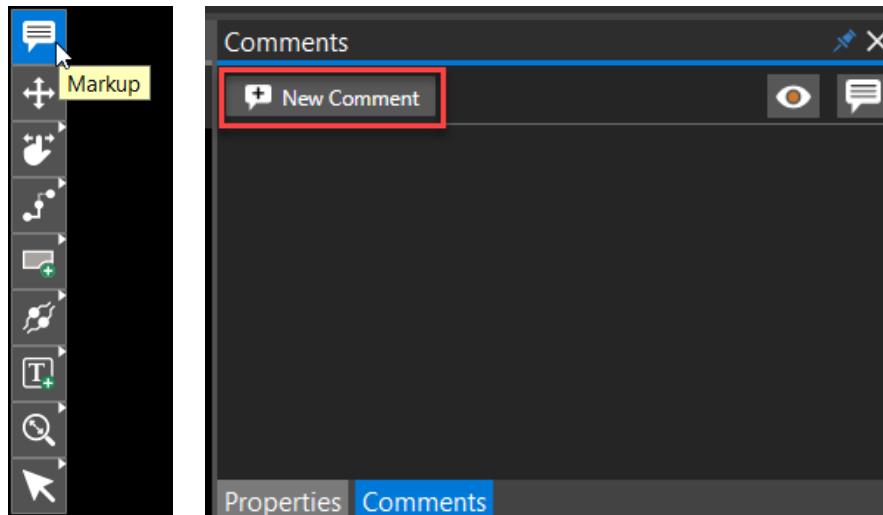
Adding Comments and Markups in OrCAD X Presto

Adding comments when reviewing designs is available only in the single-user mode. Any number of comments can be added to a design. Comments can also be added with or without markups. Furthermore, multiple markups can be associated with a single comment.

To add comments in OrCAD X Presto, do the following:

1. Choose *Tools – Markup* or click the *Markup* icon in the floating toolbar.

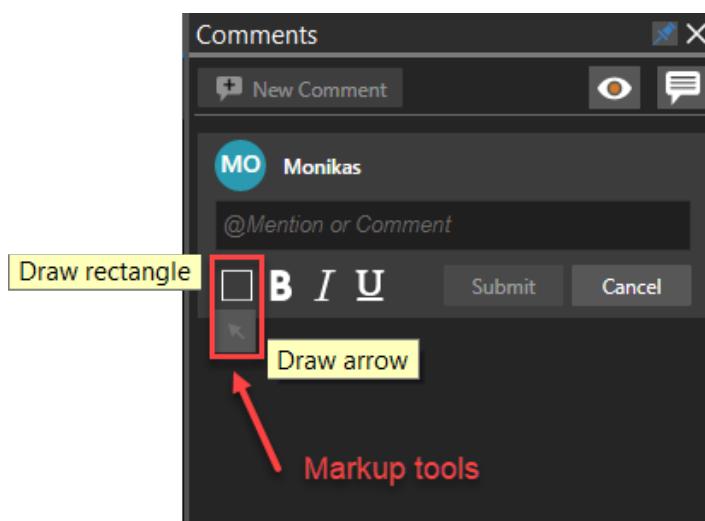
An empty *Comment* panel opens at the right. If the design already has comments, the panel displays pre-populated review comments from previous sessions.



⚠ When the markup command is active, no operations on the design objects can be performed.

2. Click the *New Comment* button.

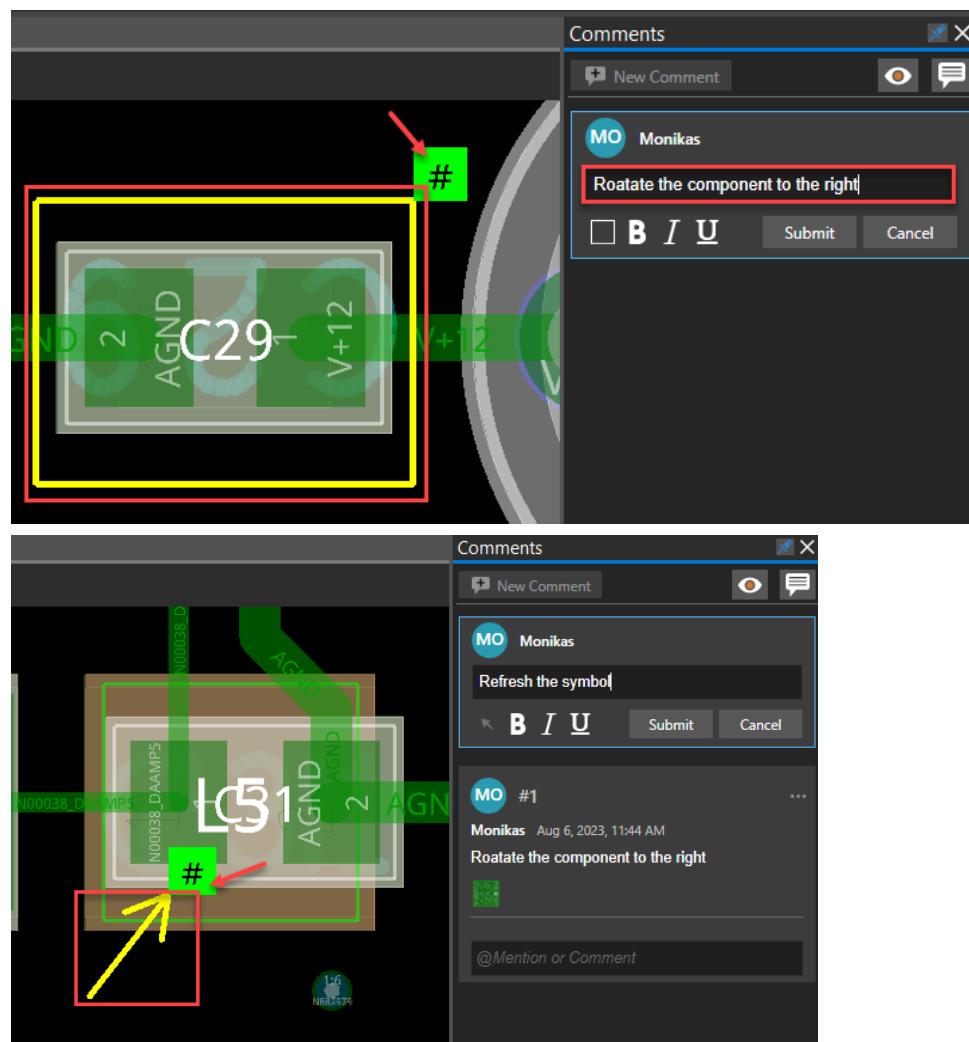
A new comment card opens at the top. Existing comments are pushed below to it.



The comment card displays the user name, user avatar, text field, markup tools, text formatting buttons, and options to save or discard a comment. A comment text can be formatted using the rich text features (bold, italic, underline). A markup is a graphical annotation in the design canvas highlighting the areas where comments are added. The user ID is created from the first two letters of the user name.

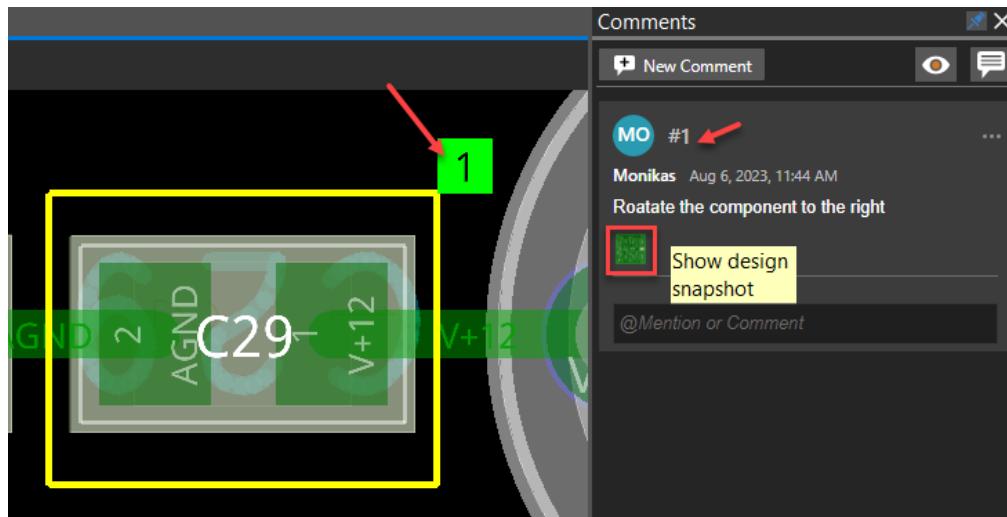
3. Select the markup tool in the comment card. Draw a rectangle to mark a rectangular-shaped area or an arrow to point to an object in the design canvas. Enter the comment in the text field, and press *Submit*.

A markup is added in yellow as a rectangle to the design canvas. The hash character denotes the comment ID during the markup placement.

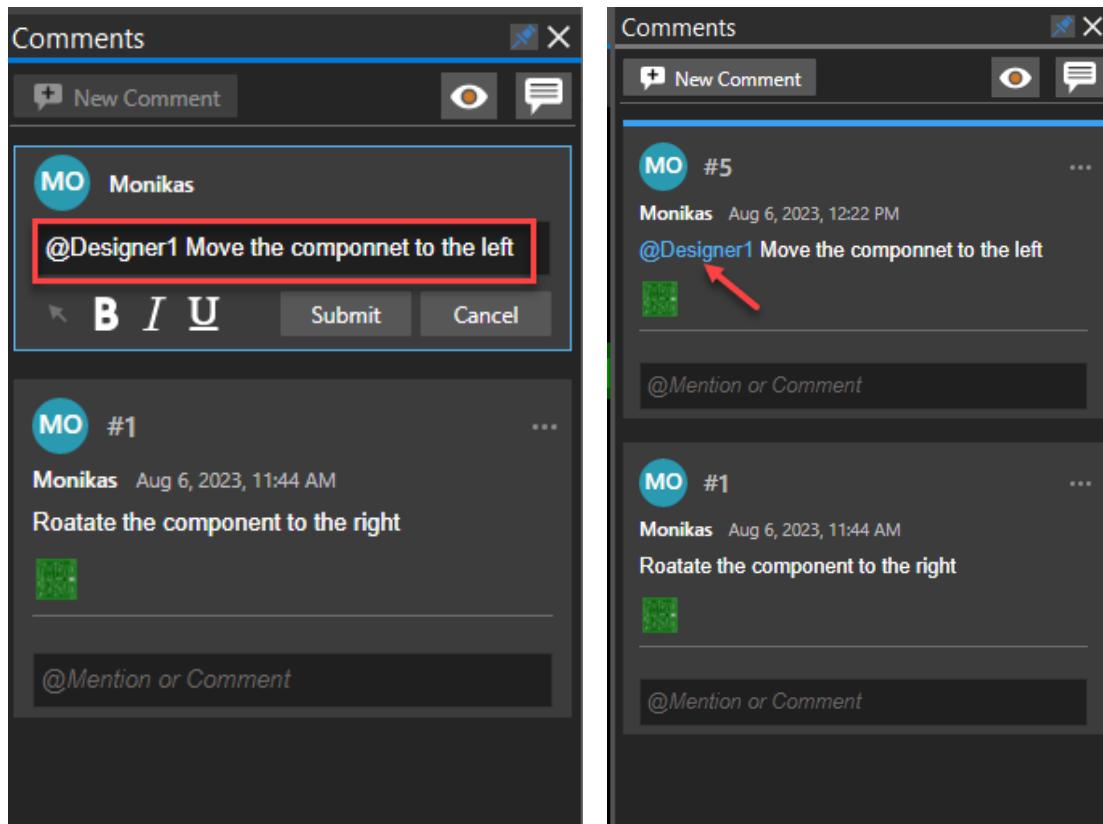


Once the comment is submitted, a unique ID is assigned to it. The markup data is attached to

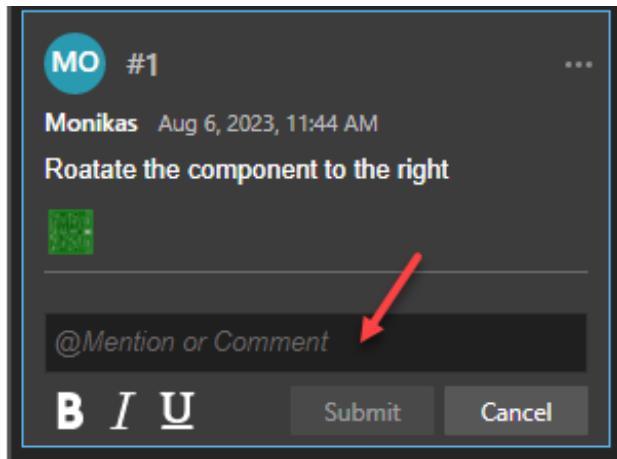
the board design database, enabling other users to view markup data without extra settings when the marked-up design is opened. The markup data includes comment ID, a timestamp that displays the local time zone, and a snapshot of the design (.png) and is displayed in the *Comments* panel. It also includes layer visibility and zoom information of the design when the comment was added.



4. Specify the user ID with a @ prefix to the comment text for assigning tasks to individual users. An assigned label is displayed in blue with that comment. Additionally, a blue bar gets added to the top edge of the comment with a task assigned to it, differentiating it from general comments.



5. Click the text field below a comment to add a reply.

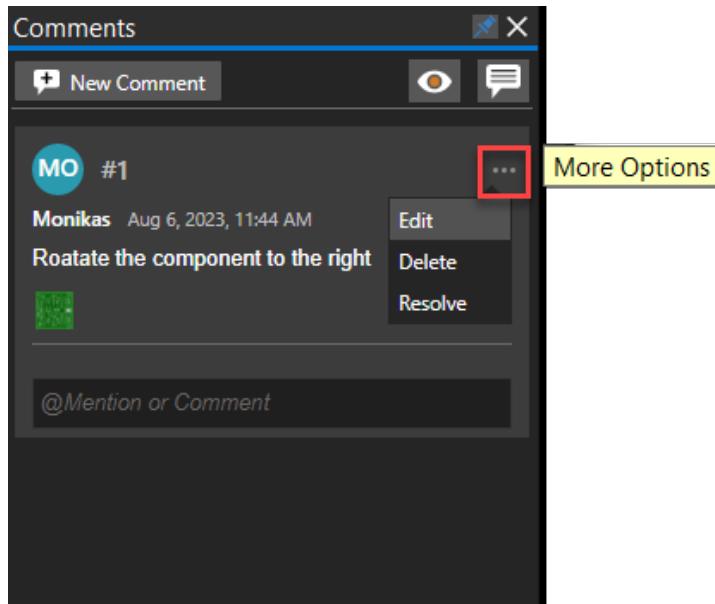


By default, comments are shown in the collapsed state; only the first and the last responses are displayed. Clicking the top-level comment expands the whole thread.

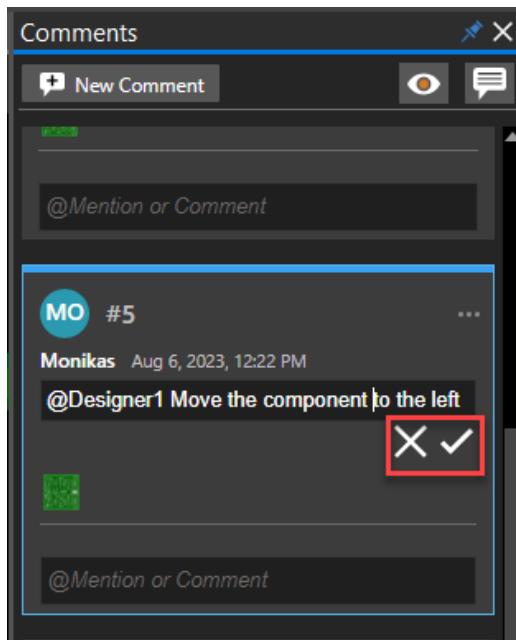
Editing Comments and Markups in OrCAD X Presto

To modify review comments and markups in OrCAD X Presto, do the following:

1. Choose *Tools – Markup* or click the *Markup* icon in the floating toolbar.
2. In the *Comments* panel, select a comment to modify.
3. Click the *More Options* icon to choose options to edit, delete, or resolve a comment.

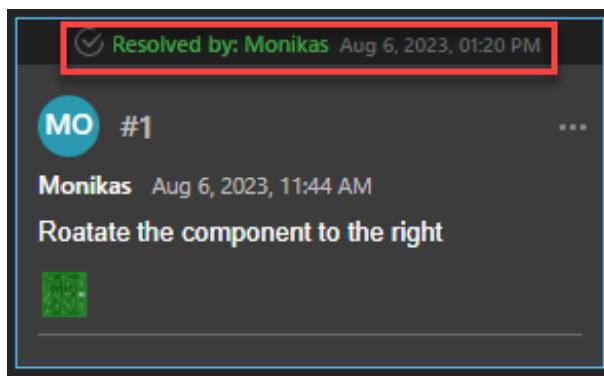


- a. Click *Edit* to modify the comment text. Select the appropriate icons below the text field to save or discard the modified text.



- b. Click *Resolve* to close an open comment. A reply can be added when marking it resolved,

The resolved comment displays the status and the user name who resolved it. A comment, when resolved, removes its markup from the canvas.



- c. Click *More Options* in the resolved comment section and select *Reopen*.

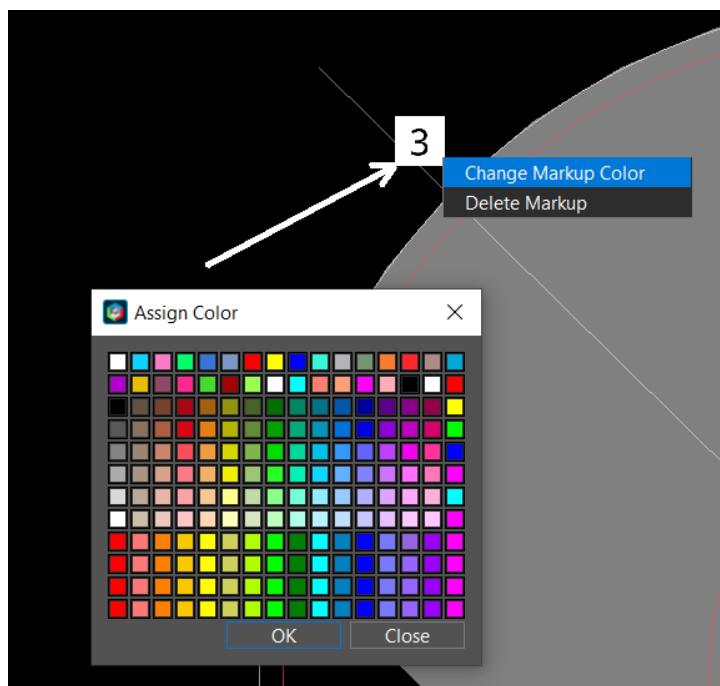
The comment is reopened for further work, and the markup is restored in the design canvas.



- d. Select *Delete* in the *More Options* list of the resolved comment.

The comment is deleted from the markup database and cannot be restored.

4. Select the markup in the design canvas, right-click, and choose *Change Markup Color*. Click a different color in the color palette in the *Assign Color* dialog box.



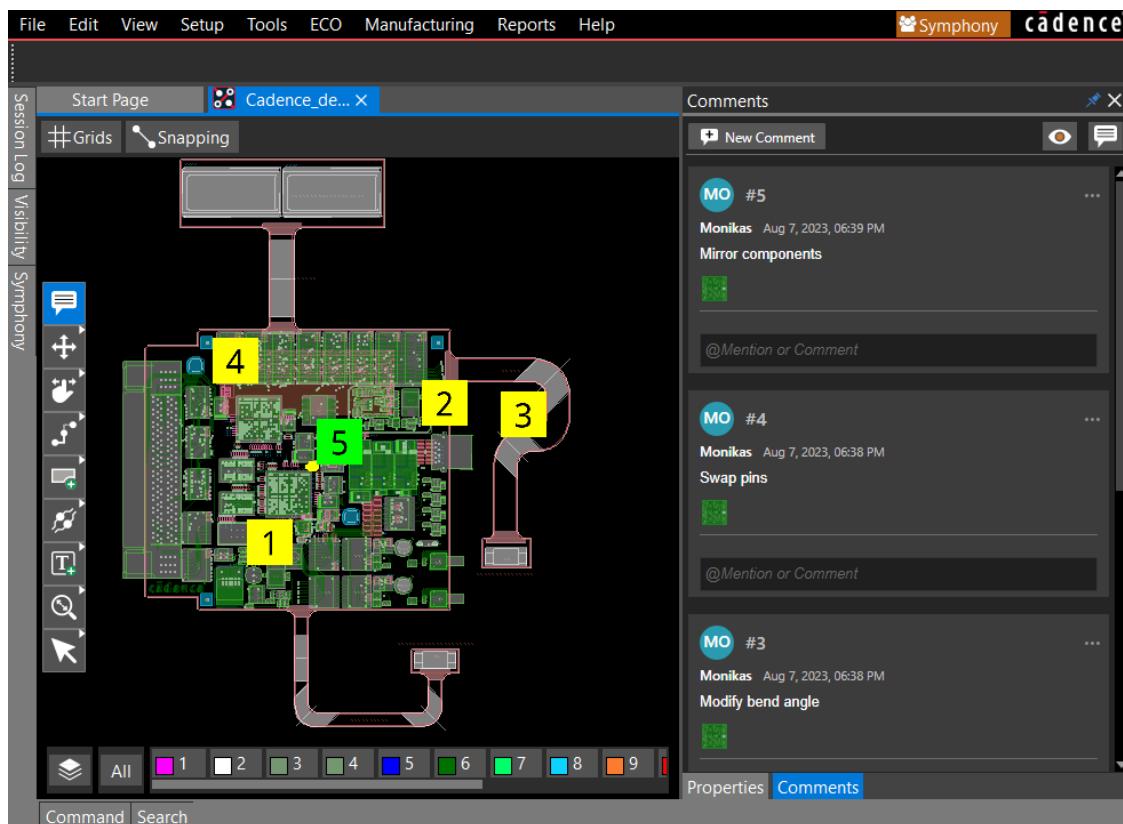
5. Select the markup for the comments that have been resolved. Right-click and choose the *Delete Markup* option.

The markup is deleted from the design canvas. The comments associated with the markup remain in the *Comment* panel until deleted manually.

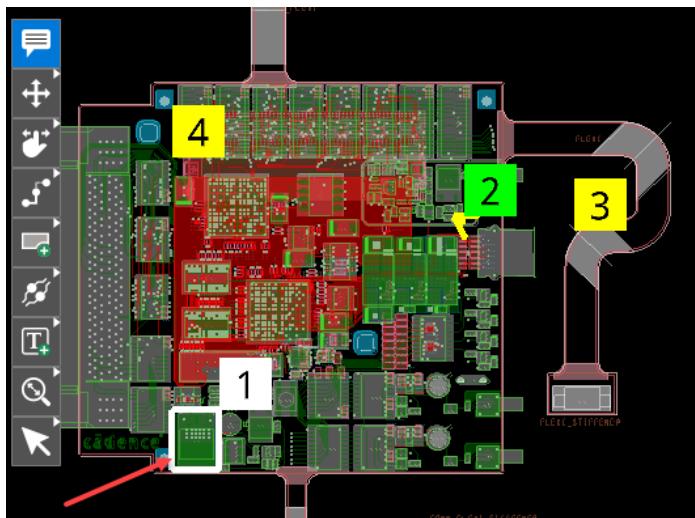
Viewing and Navigating Comments and Markups in OrCAD X Presto

To view and navigate through the comments and markups in OrCAD X Presto, do the following:

1. Open a design with existing comments and activate markups by clicking the *Markup* icon in the floating toolbar or selecting a comment in the *Comments* panel.
2. Choose *View – Zoom – Zoom Fit* or click the *Zoom Fit* icon from the floating toolbar.
Only the markup annotations are displayed in yellow.
3. Click to select a markup annotation in the design canvas.
A markup annotation, when selected, turns green. The comment associated with that markup is also highlighted in the *Comments* panel.



4. Hover the cursor over the markup ID.
The marked-up area gets highlighted.



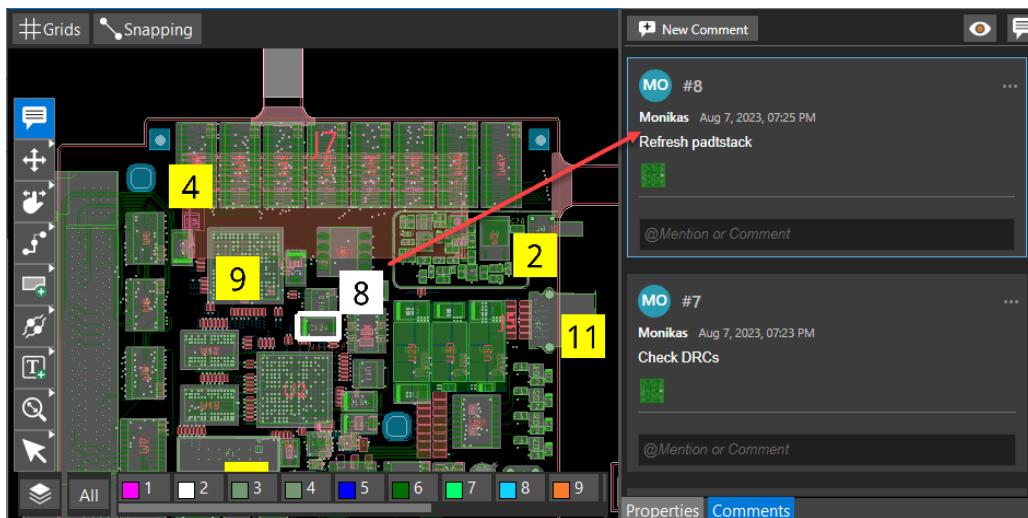
5. Click to select a comment in the *Comments* panel.

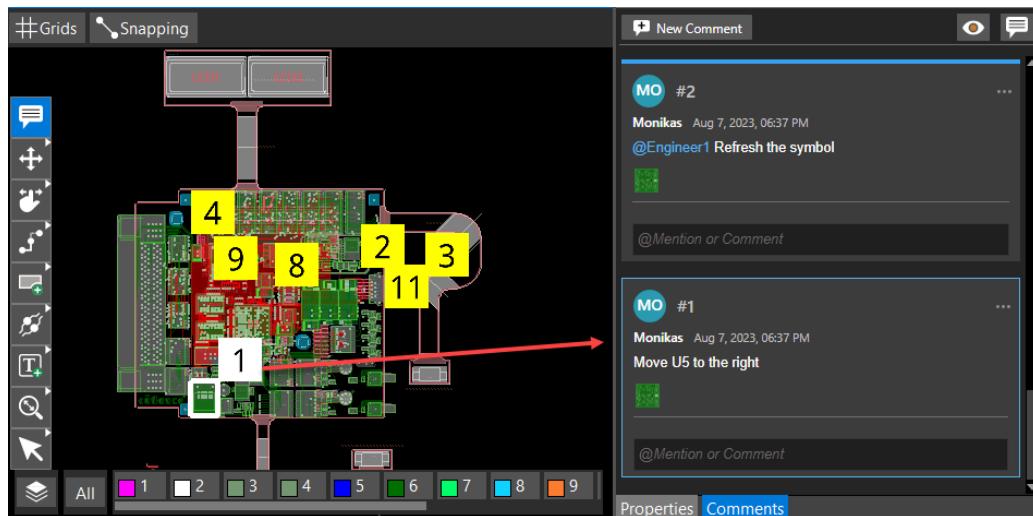
The relevant comment and markup are expanded and highlighted in the *Comments* panel and design canvas. The markup is displayed the same layer visibility and zoom state of the design when the comment was added.



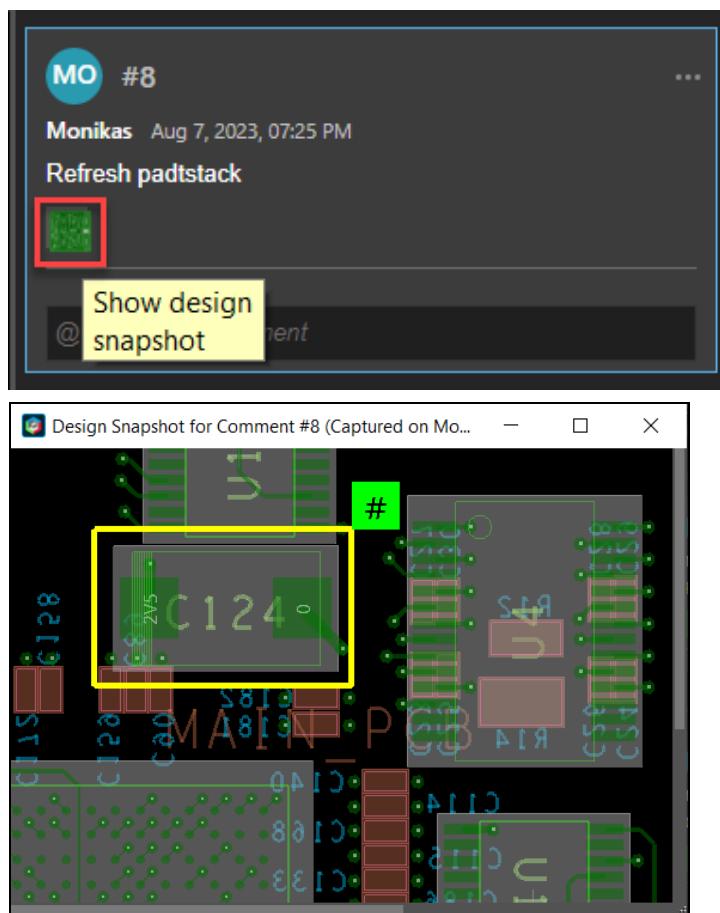


6. Click anywhere in the blank space to deselect the comment.
 7. Keep clicking comments to navigate them in the design. The other way to navigate comments is to select markup annotations in the design canvas.

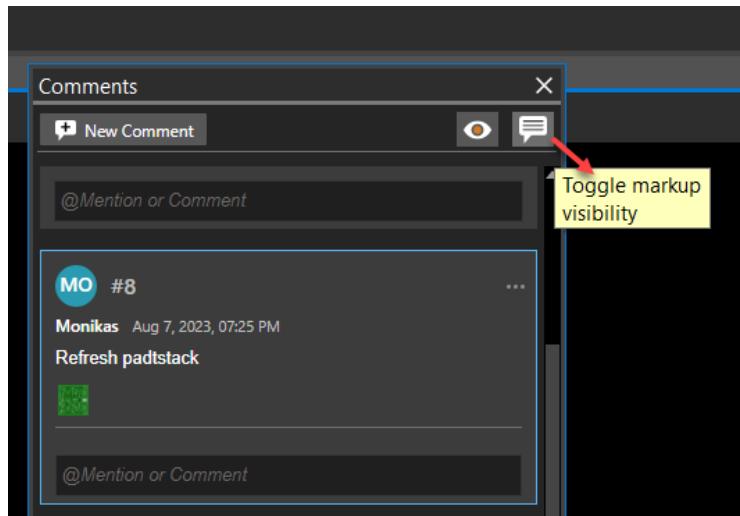




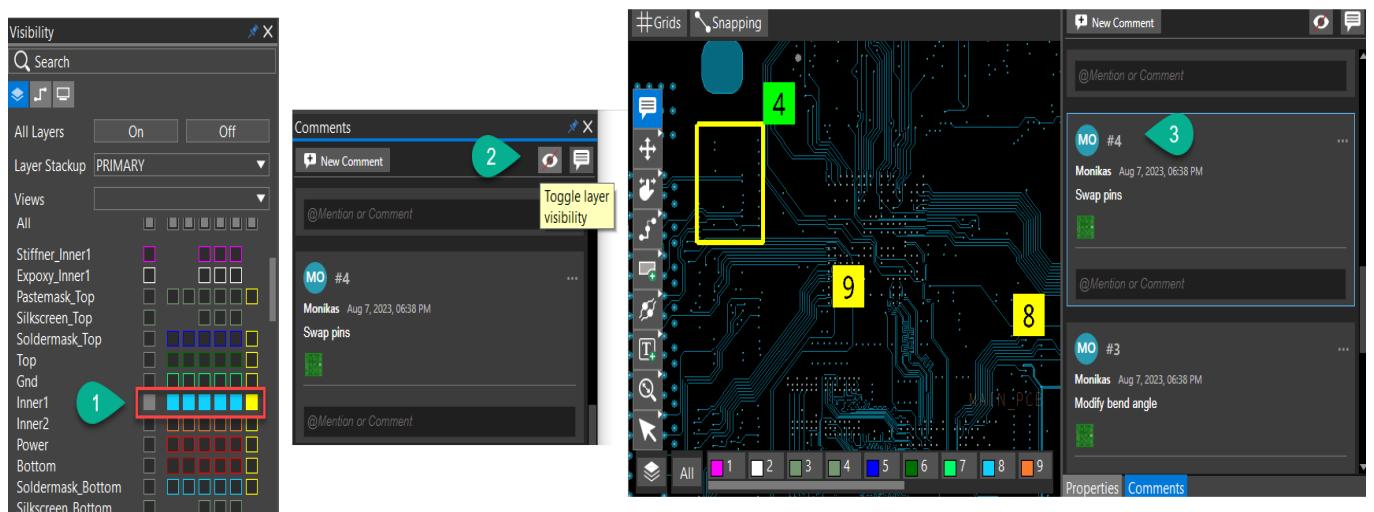
- Click the *Show Design Snapshot* icon in the comment card to view the canvas snapshot in a separate window that displays the state of design with the markup area when the comment was added.



9. Click the *Toggle markup visibility* icon in the *Comments* panel to hide markups in the design canvas. It is useful when making design edits.
All the markups become invisible in the design canvas. The comments remain available in the *Comments* panel.



10. In the *Visibility* panel, turn on the visibility of a specific layer and click the *Toggle layer visibility* icon in the *Comment* panel.
The layer visibility of individual comments is turned off and not used when displaying markups and comments. A comment when selected zoom to the markup location using the layer visibility as specified in the *Visibility* panel.



Similarly, with this option enabled, selecting markup annotations in the design canvas

highlights the relevant comments in the *Comments* panel while retaining the layer visibility.

11. Click zoom icons from the floating toolbar to view the markups in the design canvas.
The markup objects expand or contract with the zoom level. However, the size of the markup annotation remains static and does not change with the zoom changes.

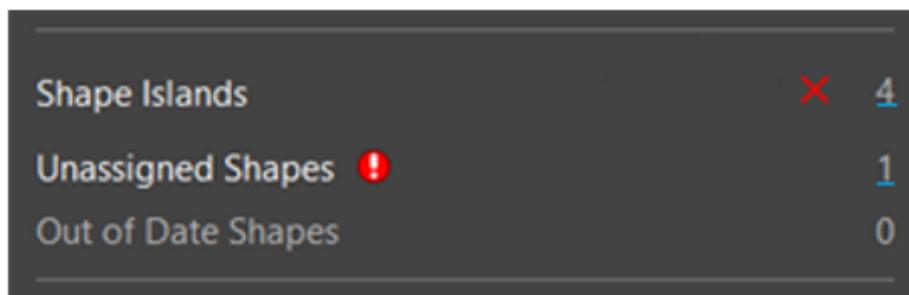
Generating Reports in OrCAD X Presto

OrCAD X Presto has a reporting mechanism that enables you to browse or search through a list of reports and choose specific items on which to focus so you can flag or resolve any issues.

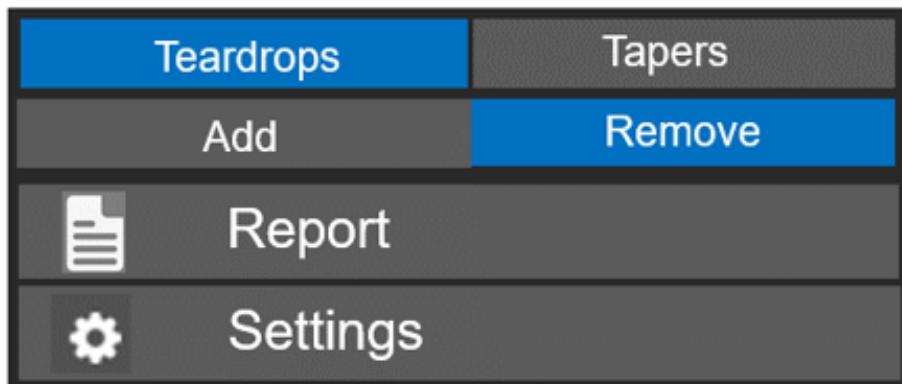
For example, you might want to run reports on shape islands or unassigned shapes; missing teardrops or tapers; and dangling traces, vias and antenna.

You can run and access reports as follows:

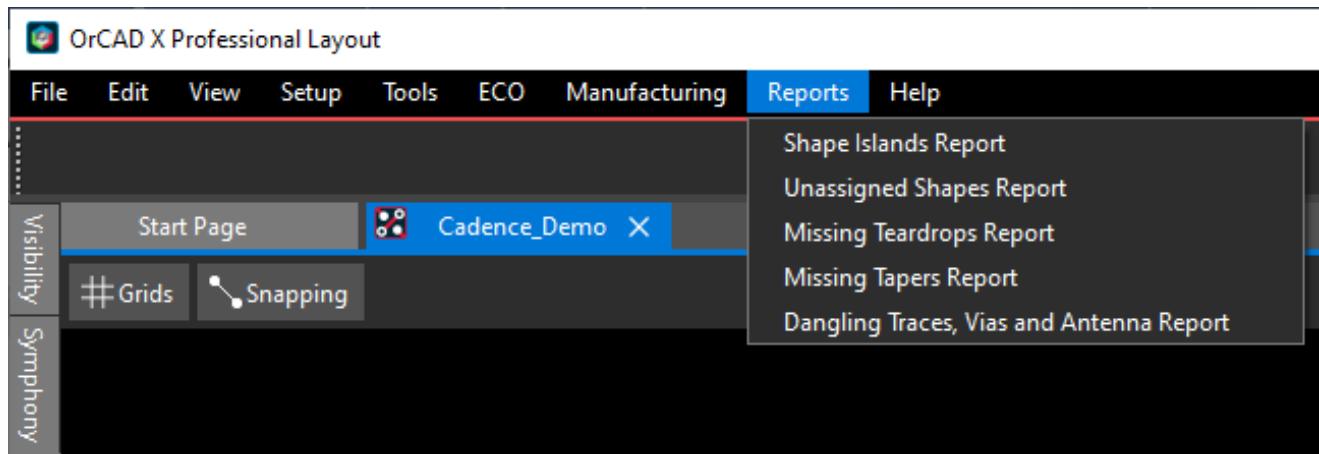
- By clicking on a blue hyperlink from a design-level property panel or object level property (Shape Islands, for example):



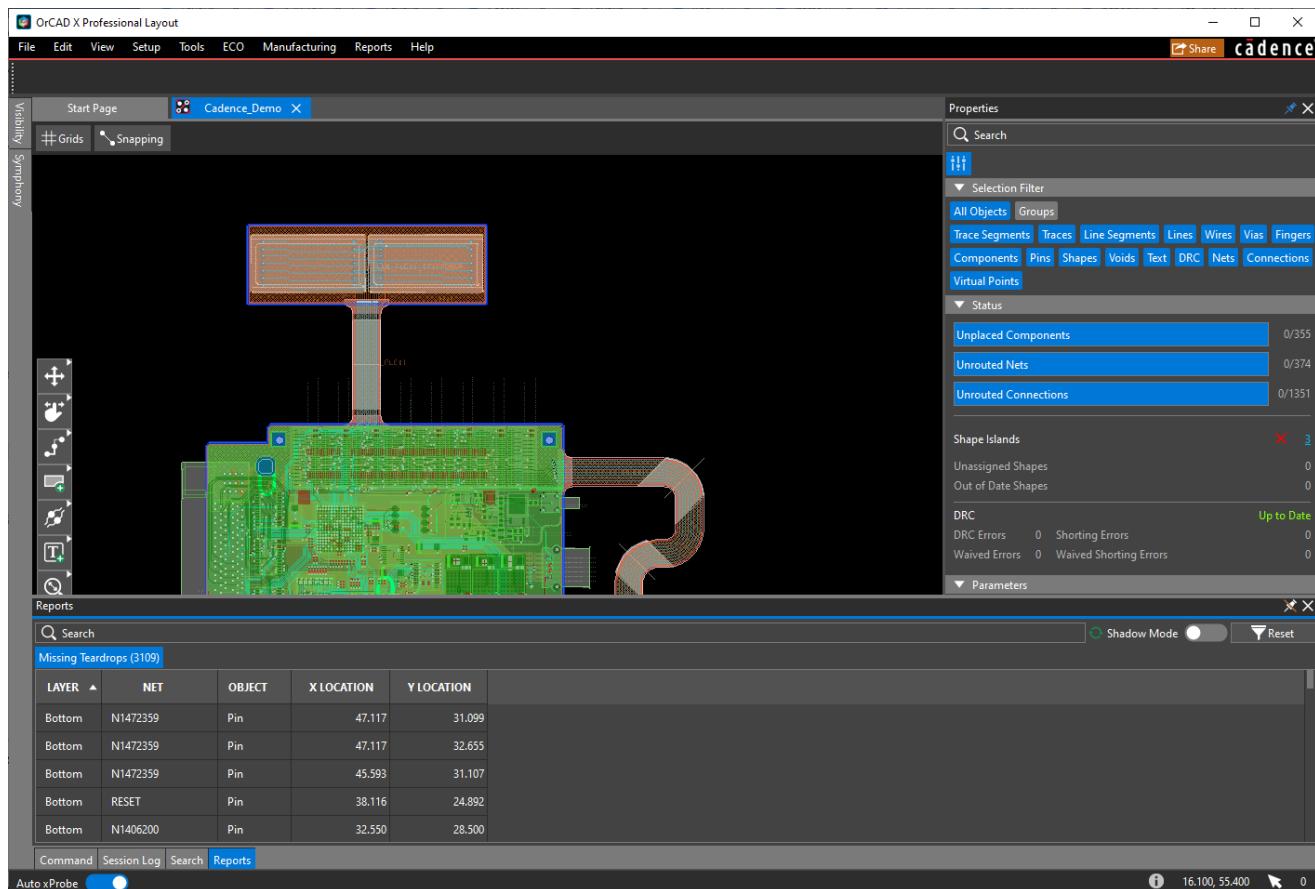
- From an active command (Teardrops, for example):



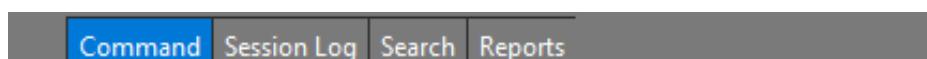
- From the top-level *Reports* menu:



Once you have run a report, it appears in a *Reports* panel, as seen at the bottom of the following Missing Teardrops example:



You can then click through the tabs at the bottom of the *Reports* panel.



To *Search*, for example:

REFDES	PART NUMBER	FOOT PRINT	LAYER	ROTATION	MIRROR	X LOCATION	Y LOCATION	CLASS	TYPE	STATE
C2	CAP_CAPC1608X86N_DISCRETE_47PF_EMA-00000797V22	CAPC1608X86N	Top	90.000°	NO	89.631	15.475	DISCRETE	Electrical Component	Placed
C3	CAP_CAPC2012X71N_DISCRETE_3300PF_EMA-00000756V22	CAPC2012X71N	Bottom	0.000°	YES	66.300	42.000	DISCRETE	Electrical Component	Placed
C4	CAP_CAPC2012X71N_DISCRETE_3300PF_EMA-00000756V22	CAPC2012X71N	Bottom	0.000°	YES	66.300	33.600	DISCRETE	Electrical Component	Placed
C5	CAP_CAPC2012X71N_DISCRETE_3300PF_EMA-00000756V22	CAPC2012X71N	Bottom	0.000°	YES	66.300	39.900	DISCRETE	Electrical Component	Placed
C6	CAP_CAPC2012X71N_DISCRETE_3300PF_EMA-00000756V22	CAPC2012X71N	Bottom	0.000°	YES	66.300	31.500	DISCRETE	Electrical Component	Placed

and find specific data. For example, CADENCE_LOGO:

Match	REFDES	PART NUMBER	FOOT PRINT	LAYER	ROTATION	MIRROR	X LOCATION	Y LOCATION	CLASS	TYPE
100%		CADENCE_LOGO	Top	0.000° NO			-25.527	97.790	Mechanical Component	Mechanical Component

- ✓ You can then either double-click or right-click on any piece of data to, for example, *Zoom*, *Edit*, or *Select on Canvas*.

⚠ Reports generated are a snapshot of the database state at the time the report data is retrieved. This data is static, but you can use the refresh button on the panel or rerun the report for updated data.

Related Topics

- [Shape Islands Report](#)
- [Unassigned Shapes Report](#)
- [Missing Teardrops Report](#)
- [Missing Tapers Report](#)
- [Dangling Traces, Vias and Antenna Report](#)

Dangling Traces, Vias and Antenna Report

The dangling traces, vias and antenna report contains two separate tabs—one for dangling traces,

NET	LAYER	LENGTH	FROM X LOCATION	FROM Y LOCATION	TO X LOCATION	TO Y LOCATION	OK DANGLE
2V5	Bottom	0.520		* 16.800		* 76.000	
CONF_SW3	Inner1	0.005		* 38.455		* 76.950	
Dummy Net	Inner1	0.850		52.000		109.750	
MEMORY.RD7	Top	0.028		* 67.403		* 95.230	
MEMORY.RD6	Top	0.028		* 12.793		* 94.000	
							12.813
							93.980
							No
							No
							No
							No

and one for dangling and antenna vias.

NET	TYPE	PADSTACK	X LOCATION	Y LOCATION	START LAYER	END LAYER	OK DANGLE
0	Antenna	VIA		53.850	106.720	* TOP	BOTTOM
0	Antenna	VIA		32.400	106.700	* TOP	BOTTOM
0	Antenna	VIA		54.739	99.060	TOP	* BOTTOM
0	Antenna	VIA		47.200	99.300	TOP	* BOTTOM
0	Antenna	VIA		40.550	99.700	* TOP	BOTTOM

All columns in the both tabs of the dangling traces, vias and antenna report are read-only and simply informational, except for *OK Dangle*, which enables you to set or reset the `OK_DANGLE` attribute for the related trace or via.

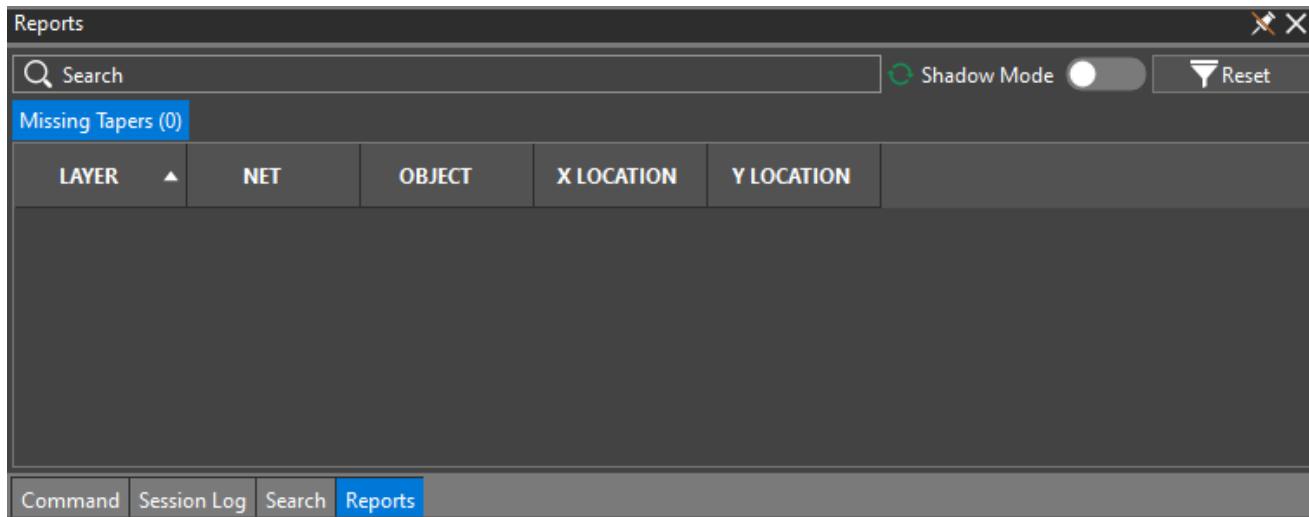
⚠ Note that report results are *not* dynamic, so you must refresh or rerun the report to update results.

Related Topics

- [Generating Reports in OrCAD X Presto](#)
- [Shape Islands Report](#)
- [Unassigned Shapes Report](#)
- [Missing Teardrops Report](#)
- [Missing Tapers Report](#)

Missing Tapers Report

A typical missing tapers report might look as follows:



All fields in the missing tapers report are read-only and simply informational.

Related Topics

- [Generating Reports in OrCAD X Presto](#)
- [Unassigned Shapes Report](#)
- [Missing Teardrops Report](#)
- [Missing Tapers Report](#)
- [Dangling Traces, Vias and Antenna Report](#)

Missing Teardrops Report

A typical missing teardrops report might look as follows:

The screenshot shows the 'Reports' window with the title bar 'Reports'. Below the title bar is a toolbar with a search field containing 'Search', a 'Shadow Mode' toggle switch, and a 'Reset' button. The main area displays a table titled 'Missing Teardrops (3109)'. The table has columns: LAY..., NET, OBJECT, X LOCATION, Y LOCATION, and an empty column on the right. There are five rows of data. The first four rows have 'Bottom' in the LAY... column and 'N1472359' in the NET column. The fifth row has 'Bottom' in the LAY... column and 'RESET' in the NET column. All other columns contain numerical values. At the bottom of the window, there is a navigation bar with tabs: Command, Session Log, Search, and Reports (which is highlighted).

LAY...	NET	OBJECT	X LOCATION	Y LOCATION	
Bottom	N1472359	Pin	47.117	31.099	
Bottom	N1472359	Pin	47.117	32.655	
Bottom	N1472359	Pin	45.593	31.107	
Bottom	RESET	Pin	38.116	24.892	
Bottom	N1406200	Pin	32.550	28.500	

All fields in the missing teardrops report are read-only and simply informational.

Related Topics

- [Generating Reports in OrCAD X Presto](#)
- [Unassigned Shapes Report](#)
- [Missing Teardrops Report](#)
- [Missing Tapers Report](#)
- [Dangling Traces, Vias and Antenna Report](#)

Shape Islands Report

A typical shape islands report might look as follows:

The screenshot shows the OrCAD X Presto software interface with a "Reports" window open. The window title is "Shape Islands (3)". It contains a table with the following data:

LAYER	NET	DELETE	X LOCATION	Y LOCATION
Inner2	0	✗	34.530	162.740
Inner2	0	✗	29.600	162.865
Inner2	0	✗	60.123	99.012

Below the table, there are buttons for "Command", "Session Log", "Search", and "Reports". The "Reports" button is highlighted.

All columns in the shape islands report are read-only and simply informational, except for *Delete*. You can click on the **x** icon in any row of the *Delete* column to disable that row (as the second row, following).

The screenshot shows the same OrCAD X Presto interface with the "Shape Islands (3)" report window. The second row from the previous screenshot has been disabled, indicated by a grayed-out background. The table data is as follows:

LAYER	NET	DELETE	X LOCATION	Y LOCATION
Inner2	0	✗	34.530	162.740
Inner2	0	✗	29.600	162.865
Inner2	0	✗	60.123	99.012

Below the table, there are buttons for "Command", "Session Log", "Search", and "Reports". The "Reports" button is highlighted.

Related Topics

- [Generating Reports in OrCAD X Presto](#)
- [Unassigned Shapes Report](#)
- [Missing Teardrops Report](#)
- [Missing Tapers Report](#)
- [Dangling Traces, Vias and Antenna Report](#)

Unassigned Shapes Report

A typical unassigned shapes report might look as follows:



The *Layer*, *X Location*, and *Y Location* columns in the unassigned shapes report are read-only and simply informational.

The following columns, however, are editable:

- The *Net* column enables you to choose the net you want from the drop-down list.
- The *Delete* column enables you to click on the **x** icon in any row to disable that row and delete the unassigned shape.
- The *Unassigned OK* column enables you to indicate whether it is okay (*Yes*) or not (*No*) to leave the shape net unassigned.

Related Topics

- [Generating Reports in OrCAD X Presto](#)
- [Shape Islands Report](#)
- [Missing Teardrops Report](#)
- [Missing Tapers Report](#)
- [Dangling Traces, Vias and Antenna Report](#)

OrCAD X Presto User Guide

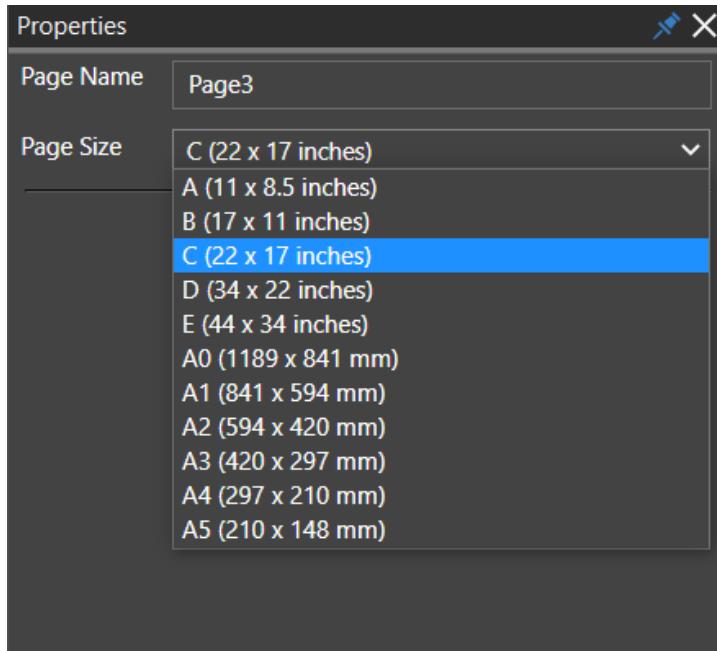
Reviewing and Analyzing Designs in OrCAD X Presto--Generating Reports in OrCAD X Presto

OrCAD X Presto Frequently Asked Questions

Why am I not able to place a PCB view in the LiveDoc canvas?

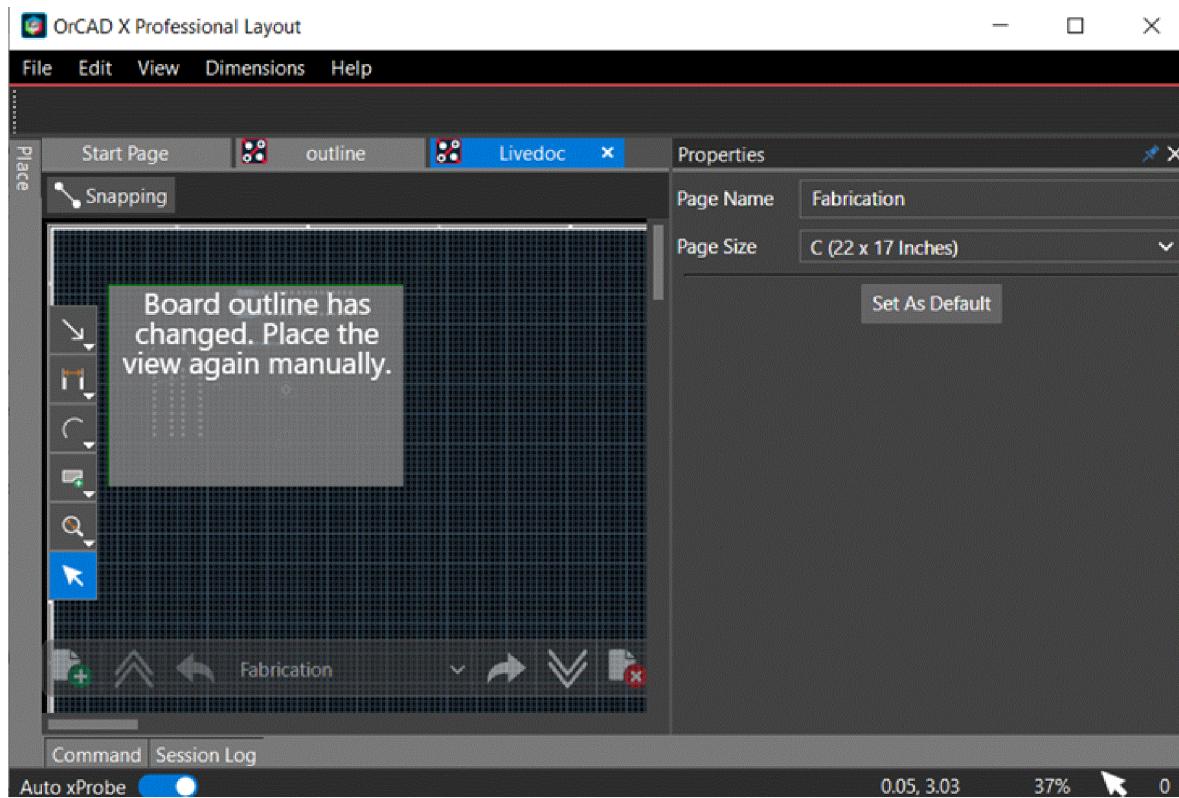
The page border size determines the extent of the LiveDoc canvas. If the view extends beyond the LiveDoc canvas, you might not place a PCB view when changing its scaling or rotating it.

You can either replace the default page size with a page with a larger size or reduce the scaling in the *Properties* panel. You can also set the new page size as the default for creating new pages.



Why do I see incorrect PCB views after editing the board design outline?

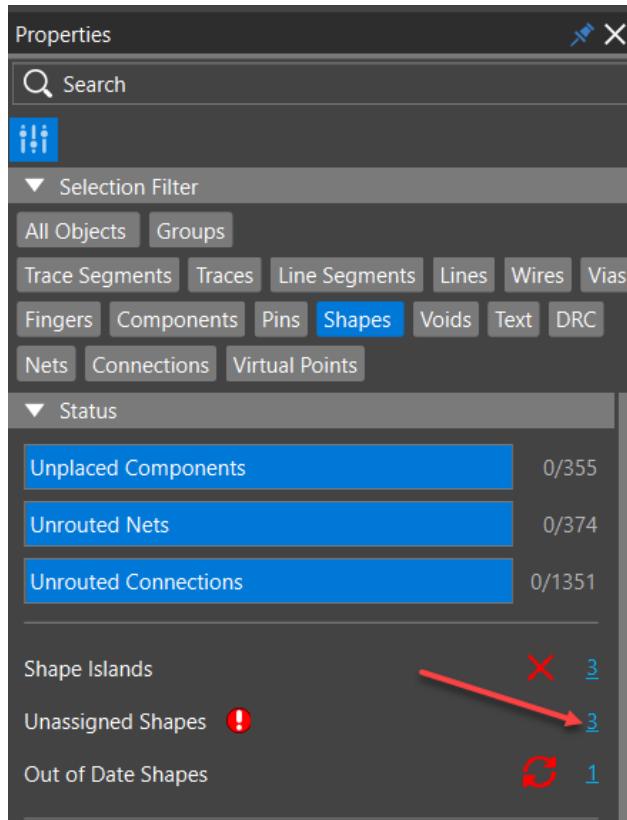
PCB views are placed in the context of the board design outline data. Any changes made to the outline, such as resizing, deleting, or changing the shape, may impact the placement of PCB views in the LiveDoc canvas. A warning is displayed in LiveDoc to indicate the change, and you need to replace the PCB views again to continue working on LiveDoc.



How can I find unassigned shapes?

The *Property* panel displays, as a link, the number of unassigned shapes in a design. To find and fix unassigned shapes, do the following:

1. Open the *Property* panel and ensure no object is selected in the design canvas.
The Status section displays the number of unassigned shapes.



2. Click the numbered link for *Unassigned Shapes* in the *Property* panel.
The *Search* table is populated with the unassigned shapes in the *Reports* panel.

LAYER ▲	NET	DELETE	X LOCATION	Y LOCATION	UNASSIGNED OK	
Top	Dummy Net	✗			No	
Top	Dummy Net	✗	104.200	147.800	No	
Top	Dummy Net	✗	129.350	115.050	No	

3. Double-click an unassigned shape in the table.
The shape is highlighted in the design canvas.

