TEAMCENTER

ECAD Quick Tour

Teamcenter 2412

SIEMENS

Unpublished work. © 2025 Siemens

This Documentation contains trade secrets or otherwise confidential information owned by Siemens Industry Software Inc. or its affiliates (collectively, “Siemens”), or its licensors. Access to and use of this Documentation is strictly limited as set forth in Customer’s applicable agreement(s) with Siemens. This Documentation may not be copied, distributed, or otherwise disclosed by Customer without the express written permission of Siemens, and may not be used in any way not expressly authorized by Siemens.

This Documentation is for information and instruction purposes. Siemens reserves the right to make changes in specifications and other information contained in this Documentation without prior notice, and the reader should, in all cases, consult Siemens to determine whether any changes have been made.

No representation or other affirmation of fact contained in this Documentation shall be deemed to be a warranty or give rise to any liability of Siemens whatsoever.

If you have a signed license agreement with Siemens for the product with which this Documentation will be used, your use of this Documentation is subject to the scope of license and the software protection and security provisions of that agreement. If you do not have such a signed license agreement, your use is subject to the Siemens Universal Customer Agreement, which may be viewed at **https://www.sw.siemens.com/en-US/sw-terms/base/uca/**, as supplemented by the product specific terms which may be viewed at **https://www.sw.siemens.com/en-US/sw-terms/supplements/**.

SIEMENS MAKES NO WARRANTY OF ANY KIND WITH REGARD TO THIS DOCUMENTATION INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF MERCHANTABILITY, FITNESS FOR A PARTICULAR PURPOSE, AND NON-INFRINGEMENT OF INTELLECTUAL PROPERTY. SIEMENS SHALL NOT BE LIABLE FOR ANY DIRECT, INDIRECT, INCIDENTAL, CONSEQUENTIAL OR PUNITIVE DAMAGES, LOST DATA OR PROFITS, EVEN IF SUCH DAMAGES WERE FORESEEABLE, ARISING OUT OF OR RELATED TO THIS DOCUMENTATION OR THE INFORMATION CONTAINED IN IT, EVEN IF SIEMENS HAS BEEN ADVISED OF THE POSSIBILITY OF SUCH DAMAGES.

TRADEMARKS: The trademarks, logos, and service marks (collectively, "Marks") used herein are the property of Siemens or other parties. No one is permitted to use these Marks without the prior written consent of Siemens or the owner of the Marks, as applicable. The use herein of third party Marks is not an attempt to indicate Siemens as a source of a product, but is intended to indicate a product from, or associated with, a particular third party. A list of Siemens’ Marks may be viewed at: **www.plm.automation.siemens.com/global/en/legal/trademarks.html**. The registered trademark Linux® is used pursuant to a sublicense from LMI, the exclusive licensee of Linus Torvalds, owner of the mark on a world-wide basis.

About Siemens Digital Industries Software

Siemens Digital Industries Software is a global leader in the growing field of product lifecycle management (PLM), manufacturing operations management (MOM), and electronic design automation (EDA) software, hardware, and services. Siemens works with more than 100,000 customers, leading the digitalization of their planning and manufacturing processes. At Siemens Digital Industries Software, we blur the boundaries between industry domains by integrating the virtual and physical, hardware and software, design and manufacturing worlds. With the rapid pace of innovation, digitalization is no longer tomorrow’s idea. We take what the future promises tomorrow and make it real for our customers today. Where today meets tomorrow. Our culture encourages creativity, welcomes fresh thinking and focuses on growth, so our people, our business, and our customers can achieve their full potential.

Support Center: **support.sw.siemens.com**

Send Feedback on Documentation: **support.sw.siemens.com/doc_feedback_form**

Contents

© 2025 Siemens

© 2025 Siemens

1. ECAD Quick Tour

Overview of the ECAD Quick Tour

The ECAD Viewer provides you with a rich set of viewing tools, including net and page connector navigation. You can quickly search for various PCB or schematic components. The software includes detailed annotation (markup) tools to help you communicate and collaborate during the PCB design lifecycle, which includes pre-translated text listings. Plus, you can select existing markup text or create your own common messages that can be included as markups during the design process. Cross probing, detailed measurement markups (distance, radial, clearance, Manhattan length and routed length), creating reports, printing, and working with Design for Assembly standards are tools and features that are also included with the ECAD Viewer.

> Note:
>
> The ECAD Viewer is supported by Teamcenter and the standalone application viewer, and its features are identical. This tutorial and its sample data are only intended to be used in the standalone application viewer.

Download and setup demo files

In this activity, you will:

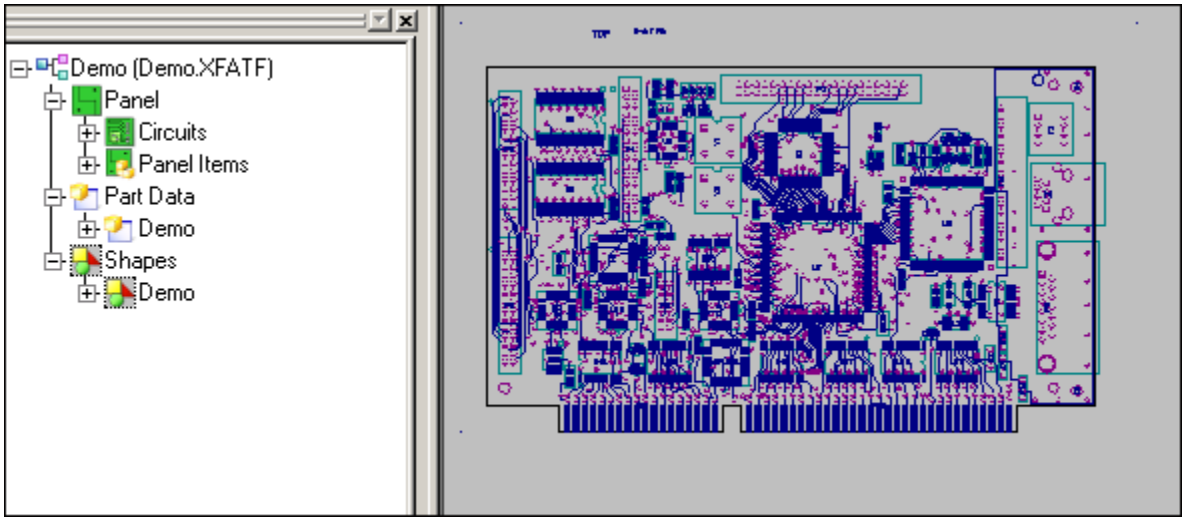
- Download and set up demo files.
- Modify preferences to view changes made to the PCB image.

1. On your desktop or the appropriate network drive, create a folder named **ecad_demo**.
2. Click the **Download** link in the online Help version of this topic to download the demo files.
3. Save *demo.zip* to the **ecad_demo** folder.
4. Extract *demo.zip* in the **ecad_demo** folder so that you have access to the ECAD documents.
5. Start Teamcenter lifecycle visualization and open *demo.xfatf*.
6. Right-click in the Viewing window and choose **Preferences**.
7. On the Viewing page of the **Preferences** dialog box, change the **Highlight Color** to yellow.

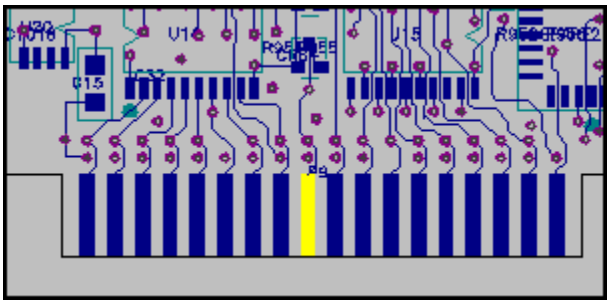
Notice the new feedback color in the second graphic.

PCB demo in the Viewing window

© 2025 Siemens



Highlighted ECAD object using zoom view



Modify the view of PCB designs

In this activity, you will:

- Learn how to use the shortcut keys to change the PCB view to display the top and bottom sides, to pan (move) the design in the Viewing window, to rotate the design, and to zoom in or out of the design.

1. Open *demo.xfatf*.

   The default color scheme is blue for the top side and green for the bottom side.

2. To view only the bottom side, press the **B** key on the keyboard.

   > Note:
   >
   > Notice the PCB objects are mostly green. This is a result of setting the layer color preference. Also, notice the **View Top and Bottom Side** toolbar option is automatically selected.

3. Switch between viewing the top and bottom sides. First, press the **T** key on the keyboard and then press the **B** key.

© 2025 Siemens

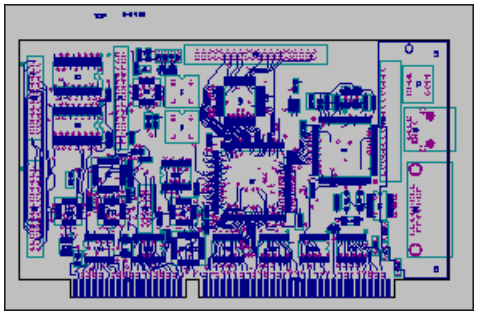
4. Press one of the arrow keys to determine which way the design moves.

5. Rotate the design 90 degrees clockwise by pressing the **R** key on the keyboard.

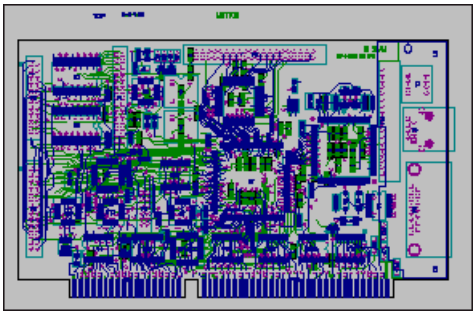
6. Zoom in or out of the design by pressing the **+** or the **-** keys.

Example:

Top side view demo.xfatf



Bottom side view demo.xfatf



Find ECAD objects

In this activity, you will:

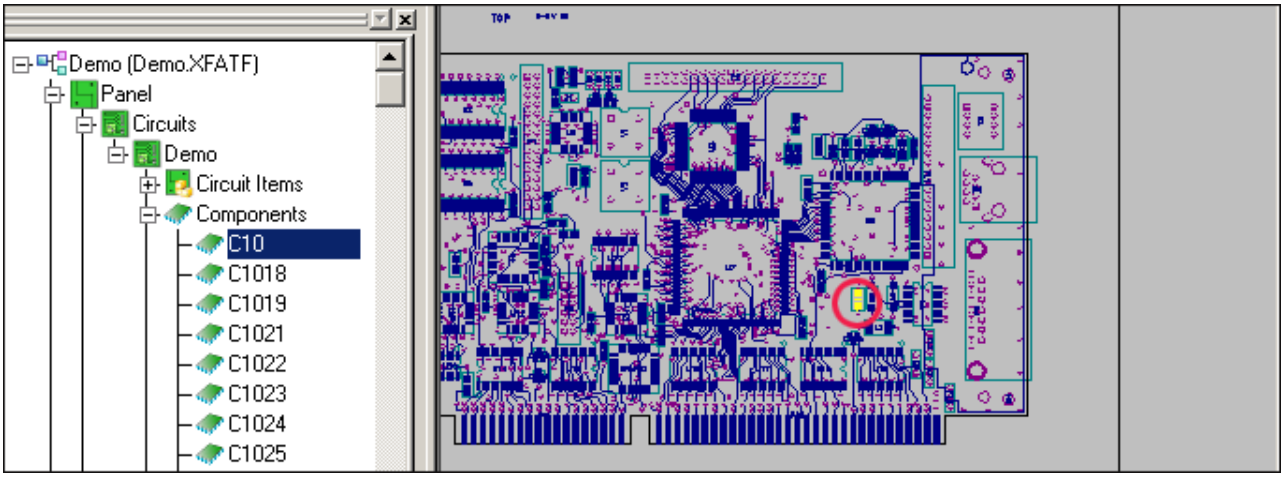
- Learn how to find an ECAD object located in the Viewing window by selecting it in the Assembly view.

1. Set the Viewing window to the Top side and in the Assembly view right-click and select **Find**.

2. In the **Find** dialog box, enter **C10** and then click **Find Next**.

   The component **C10** is highlighted in the **Assembly** view and it is highlighted (yellow) and centered in the Viewing window.

© 2025 Siemens



Try it yourself	
Find *U9*	The component, *U9* is centered in the Viewing window and highlighted here and in the **Assembly** view.
Enter **p** and click **Find next**	Cycle through components that begin with the letter *p* until you are comfortable controlling components in the **Assembly** view and in the Viewing window.

View objects and display additional details about them

In this activity, you will:

- Learn how to use the **Browse** toolbar to display ECAD objects and object **Attributes**.

1. Open *demo.xfatf*.

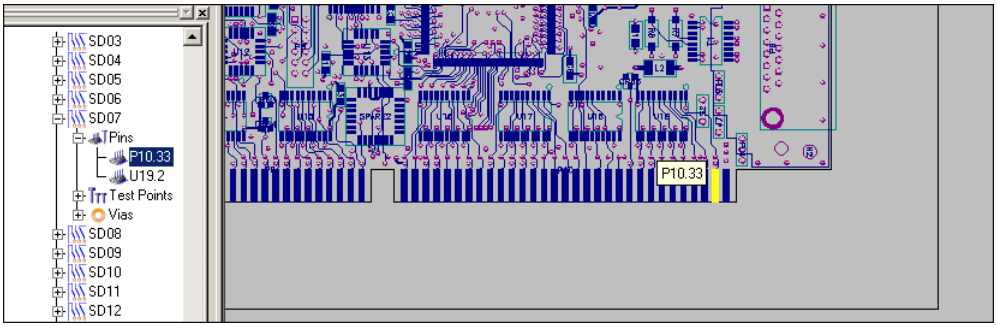
2. On the **ECAD Viewing** toolbar, select **Browse** .

3. Slowly move your cursor around the document in the Viewing window. Notice object names are displayed in a tooltip.

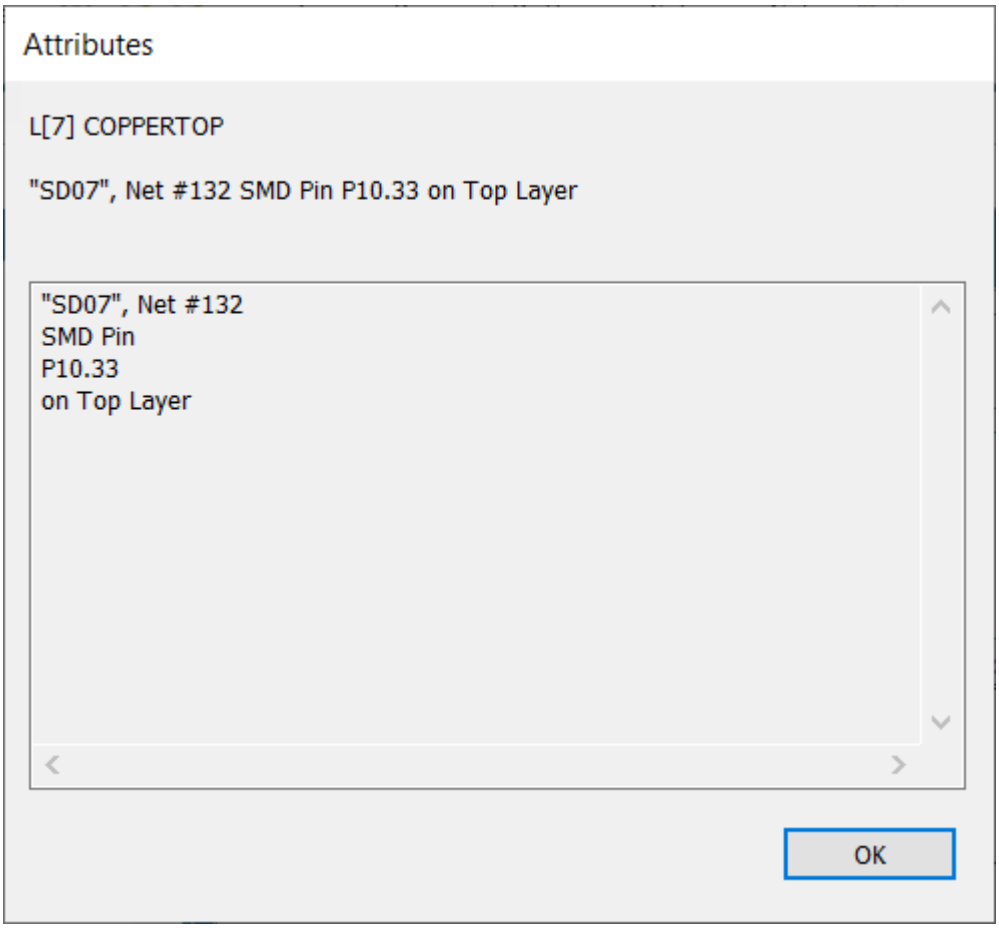
4. In the **Assembly** view, navigate to object **P10.33**.

   Notice the object is highlighted (yellow) in the Viewing window and it is highlighted in the **Assembly** view.

© 2025 Siemens



5. Display the attributes of this object by double-clicking **P10.33**. The **Attributes** dialog box provides you with additional details about the selected object.



Navigate nets and page connectors

In this activity, you will:

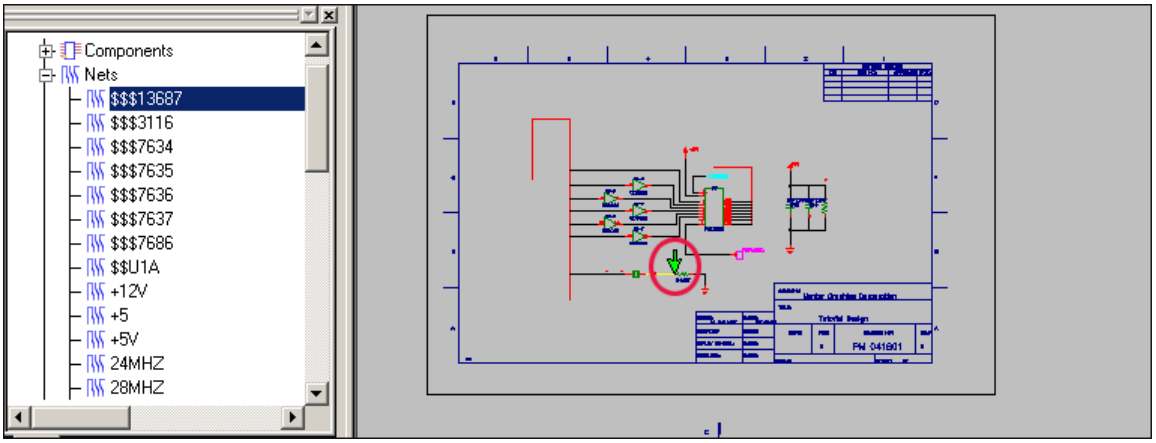
© 2025 Siemens

- Learn how to navigate nets, located on a schematic document, and page connectors used by some schematic documents.

> Note:
>
> You can also navigate nets associated with the PCB document.

1. Open the *demo.xsch* file.

2. On the *demo.xsch* file, navigate to the **Nets** section in the **Assembly** view, and highlight **$$ $13687**.

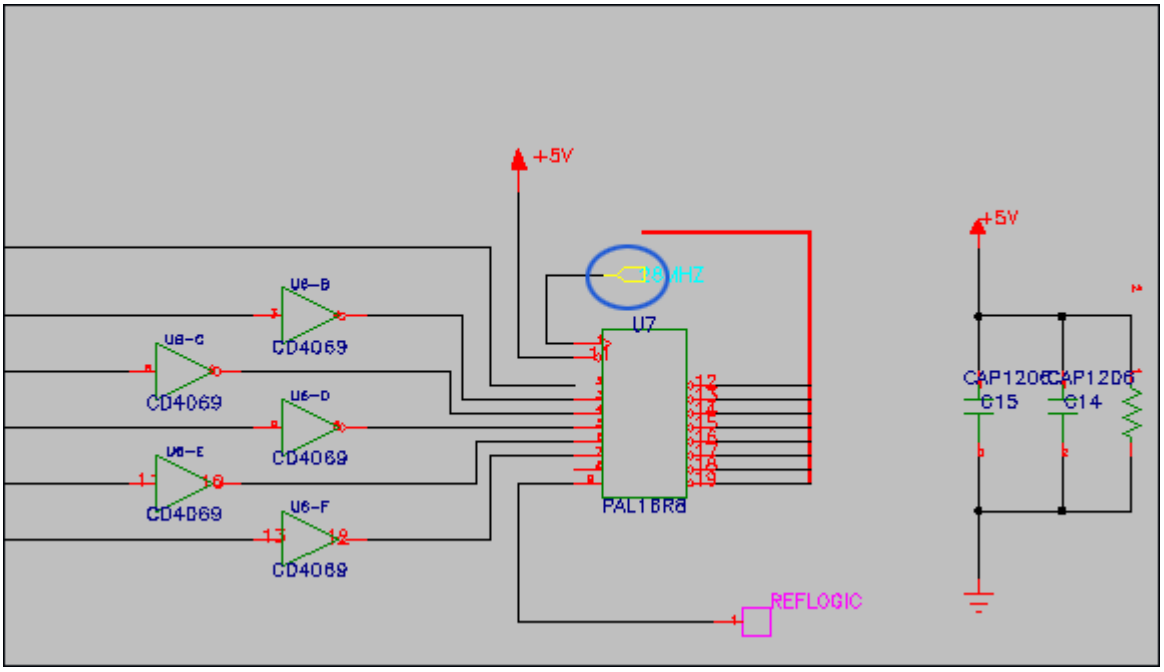


3. On the **ECAD Base** toolbar click either **Next**, or **Previous** to move from net marker to marker.

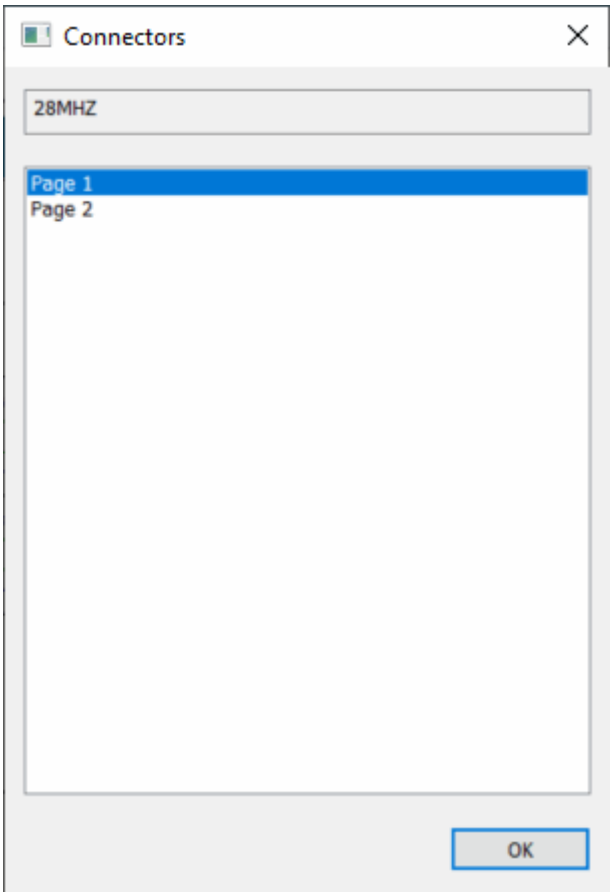
4. (Optional) Use shortcut keys to navigate through the net. Type either **P** or **N** on the keyboard to move from net marker to net marker.

5. Open *demo.xsch* and highlight the page connector on page one.

© 2025 Siemens

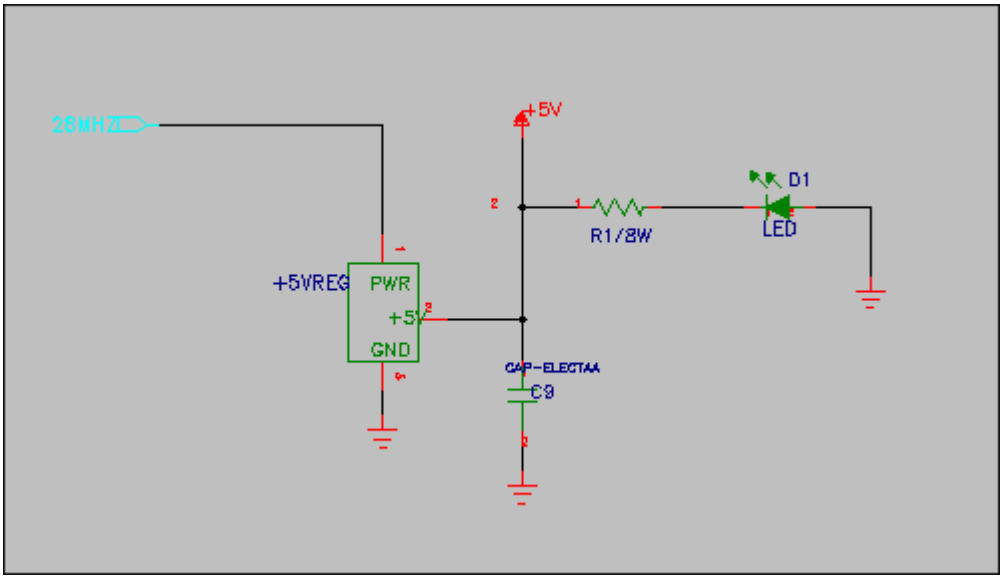


6. Use **Browse** to highlight the object, and right-click to open the **Connectors** dialog box.



© 2025 Siemens

7. (Optional) Use shortcut keys to navigate to the next page. Type either **P** or **N** on the keyboard to move from page to page in the schematic document.



Try it yourself	
Try net navigation using the PCB file. Highlight additional nets on *demo.xfatf*.	In the **Assembly** view highlight several nets and navigate the net markers for each one. Start with net #280 and then open nets of your choice.

Add a text markup

In this activity, you will:

- Learn how to use the **Rubber Stamp** to add predefined text markups to ECAD documents.
- Learn how to add a text markup that is associated with an existing markup.

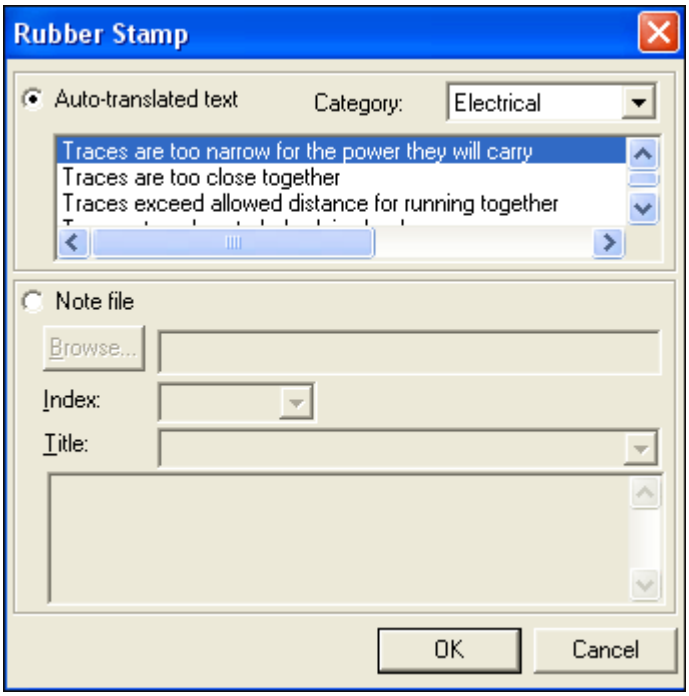
1. Open *demo.xsch*.
2. Highlight **L[1] Component Name** to make this layer the active layer.
3. On the **ECAD Markup** toolbar, select **Enable Markup**, and click **Create New Markup**.
4. Click in the Viewing window and enter a markup name in the **Markup Topic** dialog box, and then click **OK**.

© 2025 Siemens

> Note:
>
> The cursor changes to *No-Symbol* until you select a markup tool.

5. On the **ECAD Markup** toolbar, select **Rubber Stamp**.

   In this example, the **Rubber Stamp** dialog box is set to **Auto-translate text**, and the **Category** is *Electrical*.



6. Highlight the text string **Traces are too close together** and click **OK**.

7. Use the cursor to frame a text box that automatically displays the predefined text string, **Traces are too close together**.

   For this tutorial, frame the text box location as it is in the example.

8. To add a text markup that is associated with an existing markup, do the following:

   a. On the **ECAD Markup** toolbar, choose **Select Markup** and highlight the Rubber Stamp text markup bounding box.

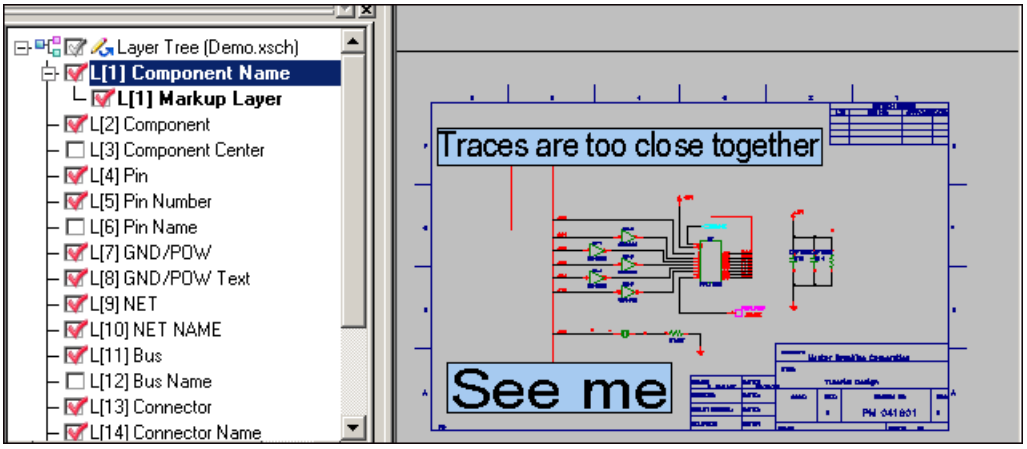
   b. Right-click on the border of the markup and select **Add to This Revision**.

© 2025 Siemens

c. On the **ECAD Markup** toolbar, select **Text**  , and in the **Edit** dialog box enter the text **See me**.

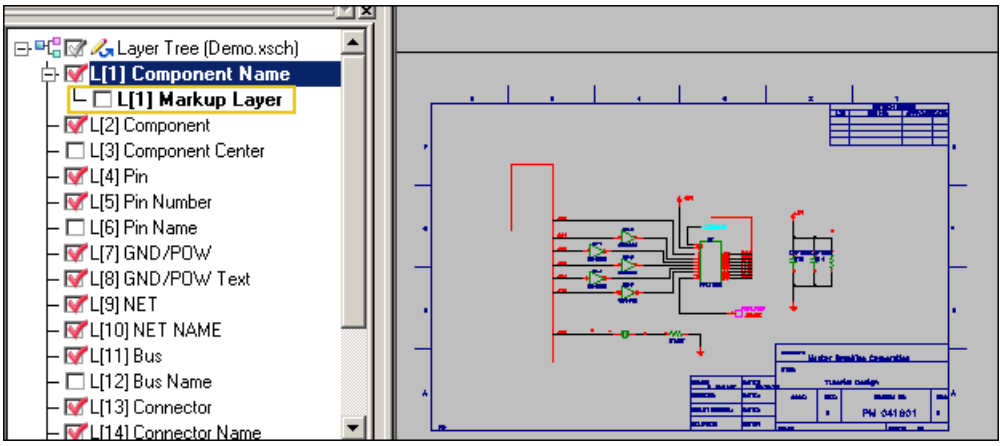
d. Use the cursor to frame a text box that automatically displays the text string, **See me**.

For this tutorial, frame the text box location as it is in the example.



Tip:

One benefit to adding markups on different ECAD document layers is to organize markups based on the contents of the layer. You can also use layers to clear and select markups so they are displayed or hidden in the Viewing window. Navigate to **L[1] Component Name** and clear and then select the **L[1] markup Layer** check box. Notice that you can clear all markups from the view so you can review the entire image without displaying the markups.



Add a markup drawing

In this activity, you will:

© 2025 Siemens

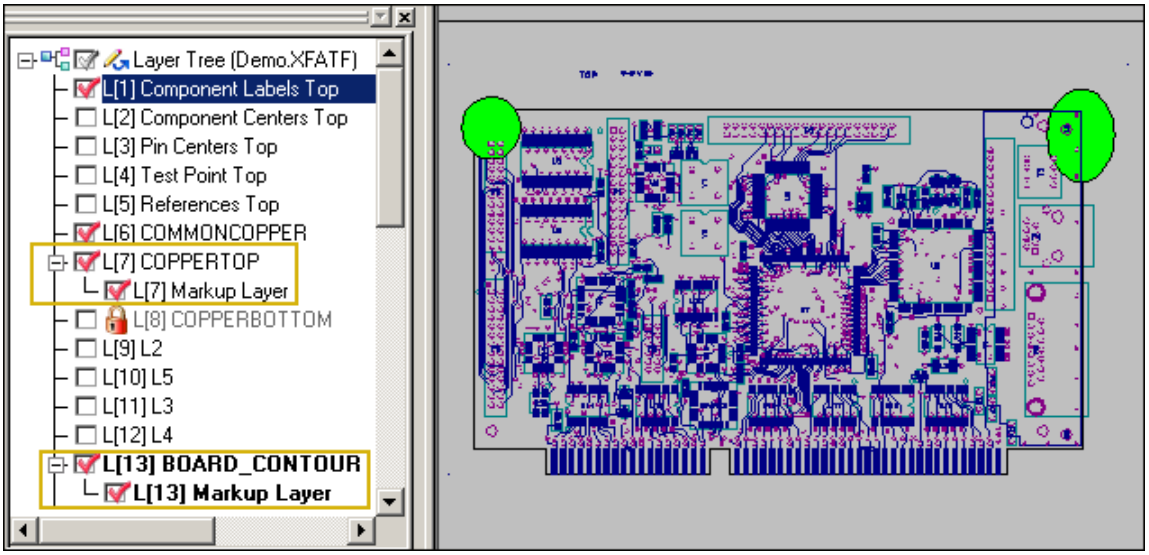
- Learn how to add markup drawings to the PCB image.

1. Open *demo.xfatf*.
2. Highlight **L[7] COPPERTOP** to make this layer the active layer.
3. On the **ECAD Markup** toolbar, select **Enable Markup**, and then click **Create New Markup**.
4. Click in the Viewing window and enter a markup name in the **Markup Topic** dialog box, and then click **OK**.

> Note:
>
> The cursor changes to *No-Symbol* until you select a markup tool.

5. On the **ECAD Markup** toolbar, select **Ellipse** and add an oval markup.
6. Highlight **L[13] BOARD CONTOUR** and repeat the steps to add another oval markup.

   These markups appear as follows:



> Note:
>
> - Notice that you select and add markups to the active layer. The graphic above indicates the active layer is **L[13] BOARD_CONTOUR**, and the other markup is on an inactive layer, **L[7] COPPERTOP**.

© 2025 Siemens

- You can select and clear markup layer check boxes to show and hide markups.
- On the **ECAD Markup** toolbar, the **Markup Log** lists markup details including the layer, topic, author, and date and time of the markup.

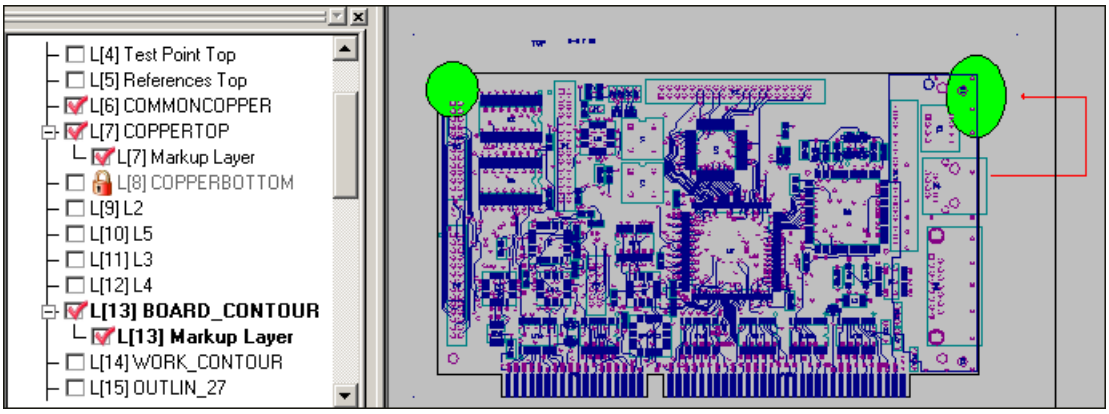
7. Select additional markup options.

   a. On the **ECAD Markup** toolbar, choose **Select Markup** and highlight the right most oval markup.

   b. Right-click on the border of the oval and select **Add to This Revision**.

   c. On the **ECAD Markup** toolbar, select **Leader Line** and try to duplicate adding the markup leader line as shown below.

Add a leader line revision



Note:

The active layer is **L[13] BOARD_CONTOUR**. The new leader line markup is added to this layer.

Try it yourself	
Open the **Markup Log**.	The **Markup Log** displays information about markups added to the document.
Experiment with options displayed on the markup shortcut menu.	• Add another new markup to this revision. • Add a **New Revision**. • Try the **Iconize** and **Expand** options.

© 2025 Siemens

Try it yourself	
	• Use **Properties** to change the line color. • Notice the additional markup details in the **Markup Log** dialog box.

Add distance measurement markups

In this activity, you will:

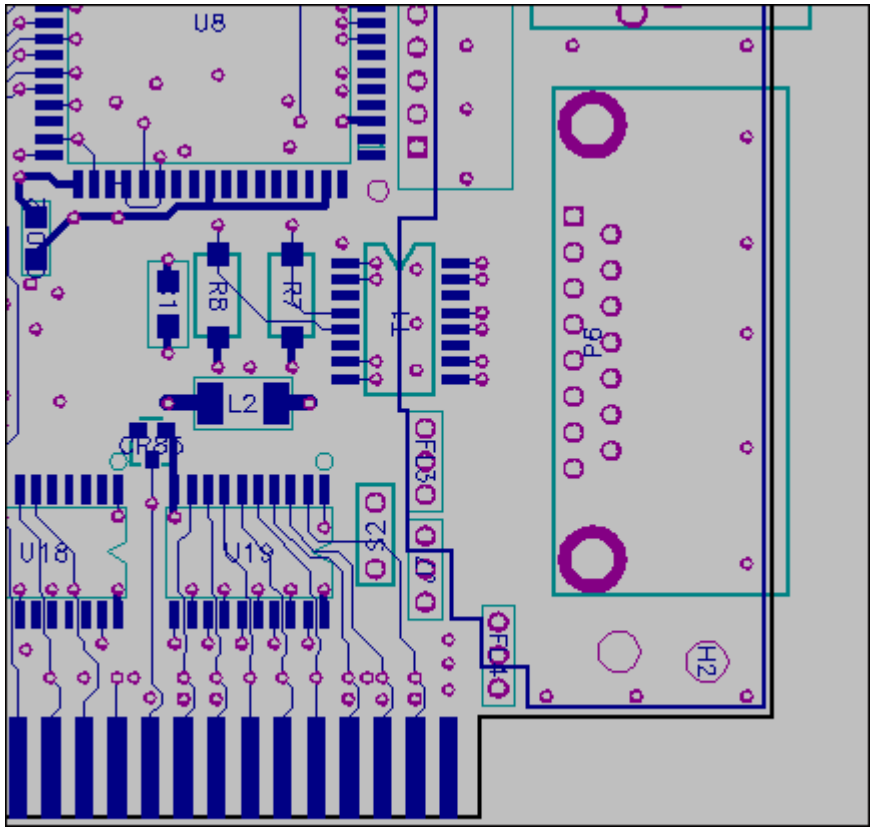
- Learn how to add measurement markups to the ECAD document. You will learn how to:

Distance measurements are taken between two points, between a point and a line, or between two parallel lines. The points and lines can be snap-to portions of a PCB entity. A snap-to element is essentially a measurement point of interest, for instance it might be the outer edge of a trace width or a vertex of a pad edge. You can also specify a point in free space, anywhere on the PCB document.

1. Open *demo.xfatf*.

2. (Optional) For illustration purposes, find the ECAD object **P6** in the Components section, and zoom until you approximate the view shown below.

© 2025 Siemens



3. On the **ECAD Markup** toolbar, select **Enable Markup**, and then click **Create New Markup**.

4. Click in the Viewing window and enter a markup name in the **Markup Topic** dialog box, and then click **OK**.

> Note:
>
> The cursor changes to *No-Symbol* until you select a markup tool.

5. On the **ECAD Markup** toolbar, select **Distance Measurement**.

6. To find the first snap-to point, move the cursor to the end point of the first line and click.

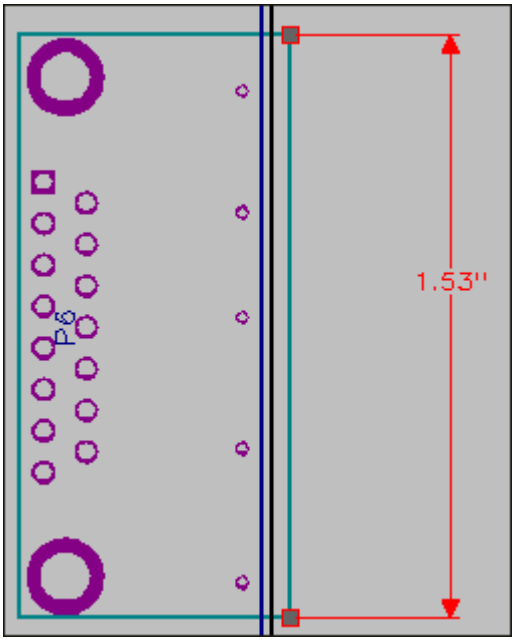
7. Move down the line until the second snap-to point appears, and click.

   You can move the measurement lines and values, as desired.

8. (Optional) To change the measurement properties (font size and line color), on the **ECAD Markup** toolbar, choose **Select Markup**, click the measurement value of the existing measurement markup (1.53”), and then right-click and select **Properties**.

© 2025 Siemens

Change the display properties, as desired.



Try it yourself	
Experiment with Distance Measurement markups.	In addition to point-to-point measurement markups try all of the following: • Vertical distance • Horizontal distance • Point–to–line distance • Line–to–line distance. For details about measurement markups, see Overview of measuring distances in ECAD files in *ECAD Viewer*.

Add radial measurement markups

In this activity, you will:

- Learn how to add radial measurement markups to the ECAD document.

Radial measurements cover the distance associated with ring and circle ECAD objects. Rings can be displayed in different forms, including circular shape with circular hole, circular shape with polygonal hole, polygonal shape with circular hole, and polygonal shape with polygonal hole. Radial

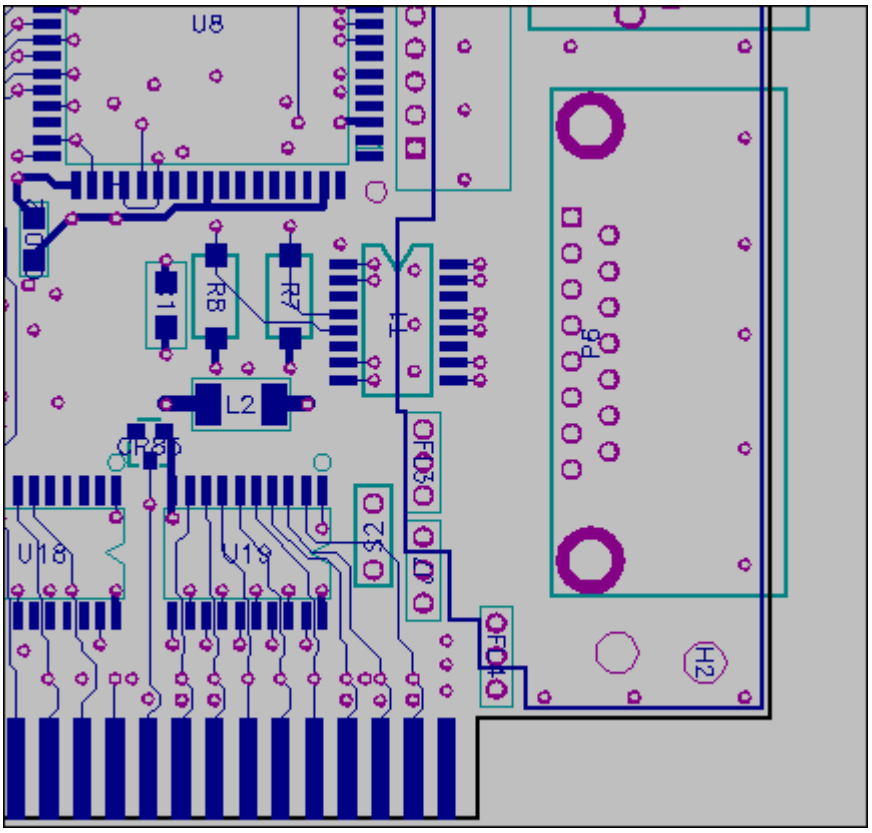
© 2025 Siemens

measurements require just one mouse click on or near a highlighted snap-to point. Three types of radial measurements are supported: the hole radius, the ring radius, and the radial width of the ring (the difference between the outer radius and the hole radius).

This activity demonstrates how to create radial measurements.

1. Open *demo.xfatf*.

2. (Optional) For illustration purposes, find the ECAD object **P6** in the Components section, and zoom until you approximate the view shown below.



3. On the **ECAD Markup** toolbar, select **Enable Markup**, and then click **Create New Markup**.

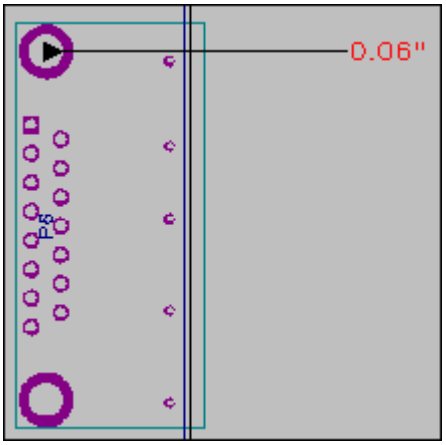
4. Click in the Viewing window and enter a markup name in the **Markup Topic** dialog box, and then click **OK**.

> Note:
>
> The cursor changes to *No-Symbol* until you select a markup tool.

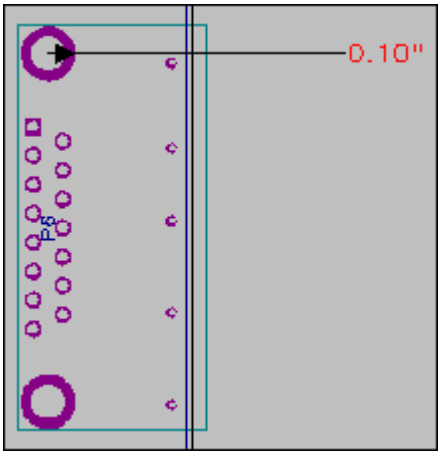
5. On the **ECAD Markup** toolbar, select **Radial Measurement**.

© 2025 Siemens

6. Move the cursor to the circle identified in the sample file. Click the cursor when you see the snap-to marker. This action measures the hole radius.

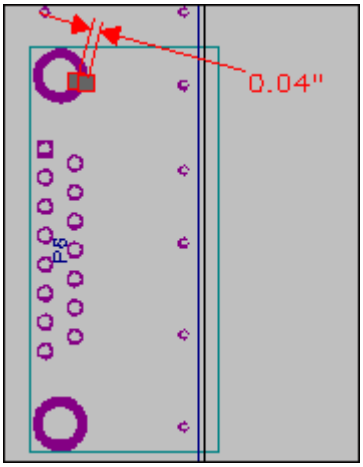


7. Move the cursor so that the marker highlights both the inner and outer rings and click. This action measures the ring radius.



8. Move the cursor so the snap-to marker is visible. Hold the Shift key and click. This action measures the inner and outer ring width.

© 2025 Siemens



Add clearance measurement markups

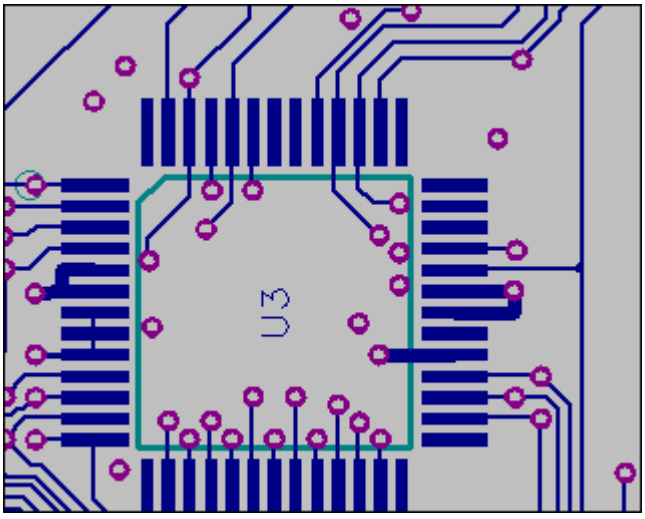
In this activity, you will:

- Learn how to add clearance measurement markups to the ECAD document.

Clearance measurement determines the amount of minimum distance, or clearance, between any two PCB objects. When you move the mouse around the PCB, underlying objects are highlighted when you approach them.

1. Open *demo.xfatf*.

2. (Optional) For illustration purposes, find the ECAD object **U3** in the Components section, and zoom until you approximate the view shown below.



3. On the **ECAD Markup** toolbar, select **Enable Markup**, and then click **Create New Markup**.

© 2025 Siemens

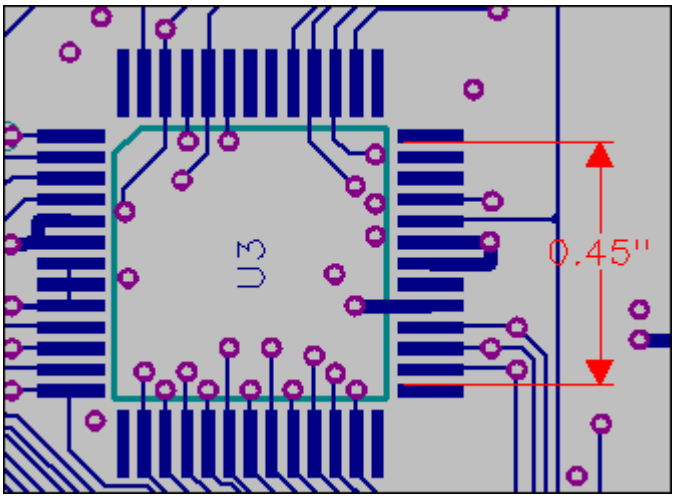
4. Click in the Viewing window and enter a markup name in the **Markup Topic** dialog box, and then click **OK**.

> Note:
>
> The cursor changes to *No-Symbol* until you select a markup tool.

5. On the **ECAD Markup** toolbar, select **Clearance Measurement**.

6. Do the following:

   - Move the cursor to the ECAD object of interest and click.
   - Next move the cursor to the next object of interest and click.



Add Manhattan and Routed measurement markups

In this activity, you will:

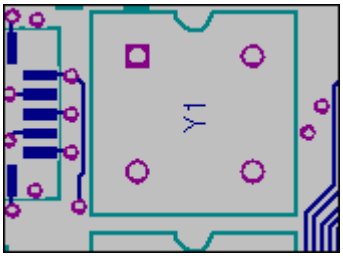
- Learn how to add Manhattan and Routed measurement markups to the ECAD document.

Manhattan or Routed measurements measure the length between various points along traces and vias associated with the same net. Use snap-to pick points to mark the beginning and ending measurement points.

1. Open *demo.xfatf*.

2. (Optional) For illustration purposes, find the ECAD object **Y1** in the Components section, and zoom until you approximate the view shown below.

© 2025 Siemens



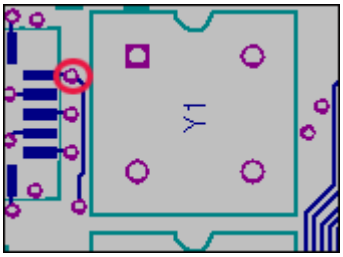
3. On the **ECAD Markup** toolbar, select **Enable Markup**, and then click **Create New Markup**.

4. Click in the Viewing window and enter a markup name in the **Markup Topic** dialog box, and then click **OK**.

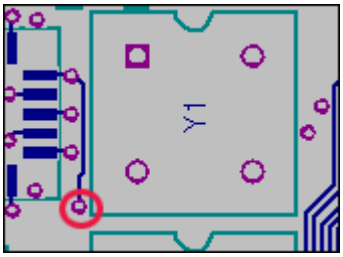
> Note:
>
> The cursor changes to *No-Symbol* until you select a markup tool.

5. On the **ECAD Markup** toolbar, select **Manhattan Measurement**.

6. Move the cursor until you notice the snap-to marker on the top of the trace, then click.

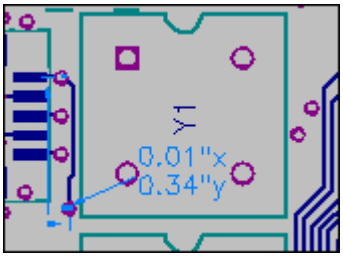


7. Move the cursor until you notice the snap-to marker on the bottom of the trace, then click.



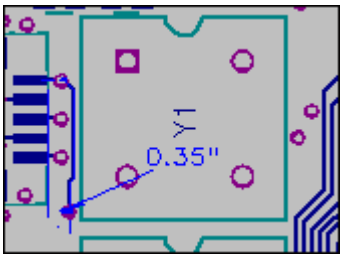
The Manhattan distance between two snap-to markers is the sum of the X and Y measurements. In this example, the Manhattan length is *0.35”*.

© 2025 Siemens



8. To measure the distance between these same two trace points using **Routed Measurement**, follow steps 1 to 6 above. On the **ECAD Markup** toolbar, select **Routed Measurement** .

   The Routed distance between two snap-to markers is the sum of the distance traced along its path. In this example, the distance is *0.35"*.



> Note:
>
> The length measured between the same two snap-to markers may not always be equal.

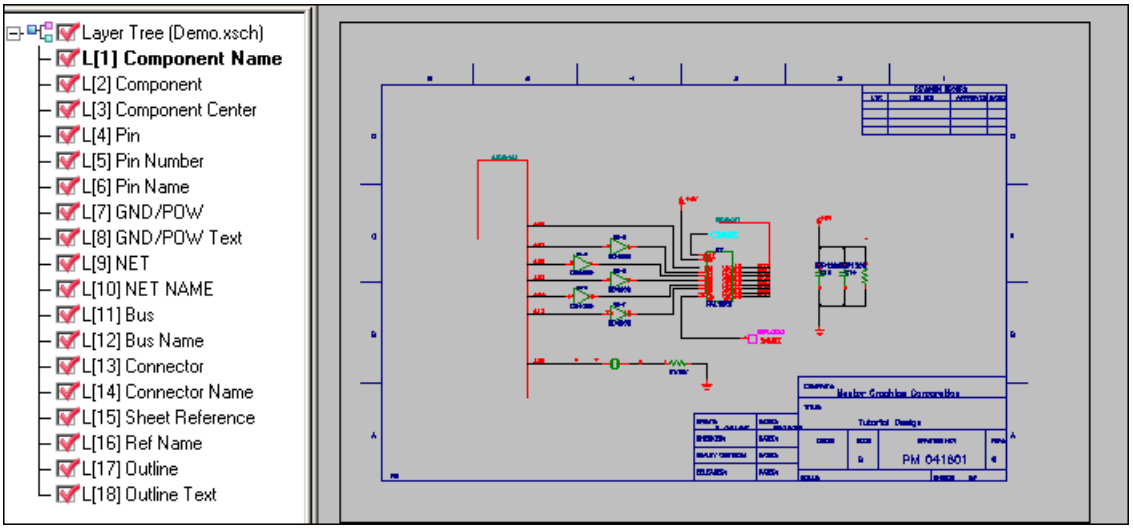
Work with layers

In this activity, you will:

- Learn the important role layers play when you work with ECAD designs. This includes displaying and hiding layers and changing layer colors.

1. Open *demo.xsch*.

© 2025 Siemens

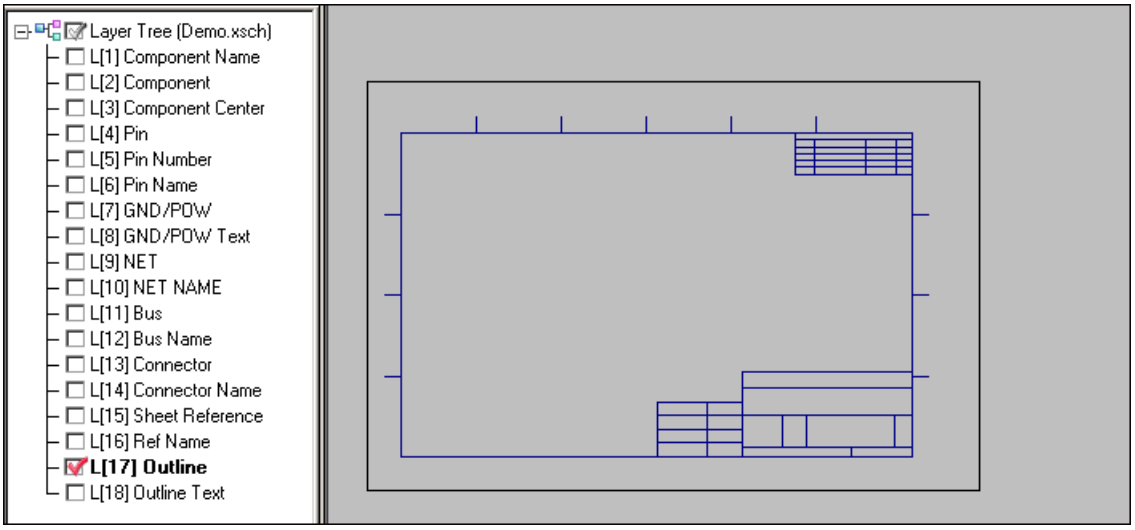


2. In the **ECAD Layers** view, clear the check boxes of all layers.

   This creates an empty ECAD document, which is a good starting point to understanding layers.

3. Select **L[17] Outline**.

© 2025 Siemens

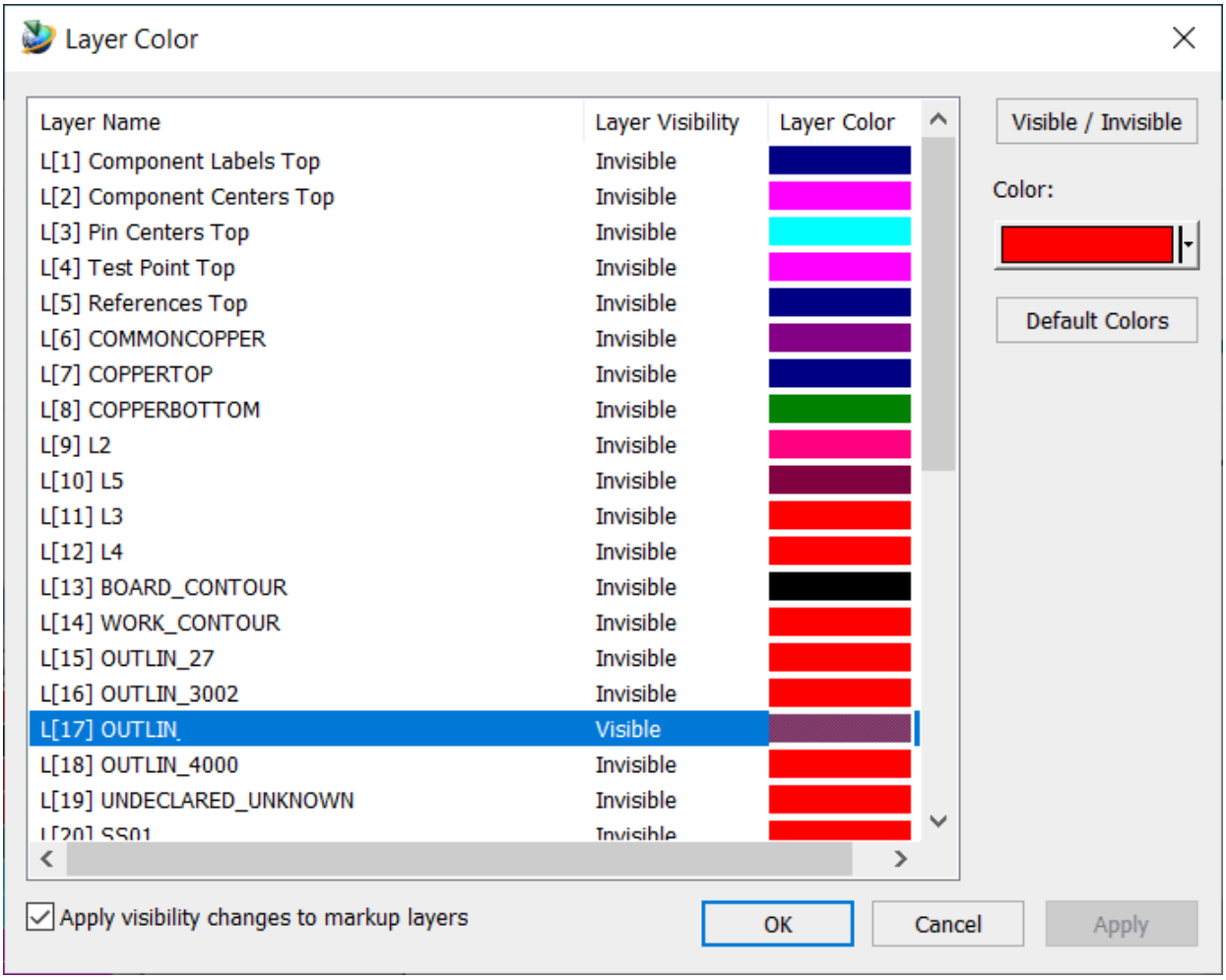


4. Change the **L[17] Outline** layer color from blue to red.

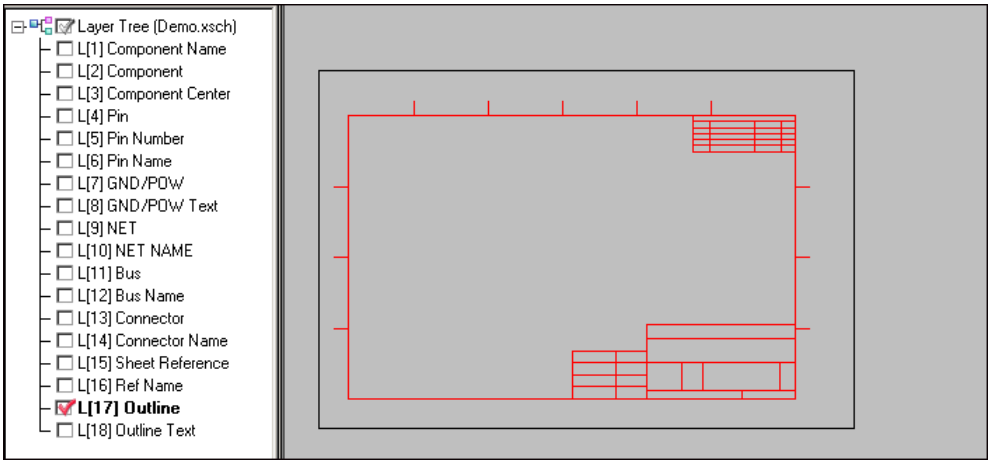
   a. On the **ECAD Base** toolbar, select **Layer Color** .

   b. Highlight **L[17] Outline**.

© 2025 Siemens



c. From the **Color** section, select red.



5. Change the layer color for **L[17] Outline** back to blue.

© 2025 Siemens

6. Continue to experiment by selecting and clearing layers.

> Note:
>
> To reset the visibility state of your ECAD document, right-click in the **ECAD Layers** view, and then select **Reset Visibility State**.

Cross probe a schematic and PCB

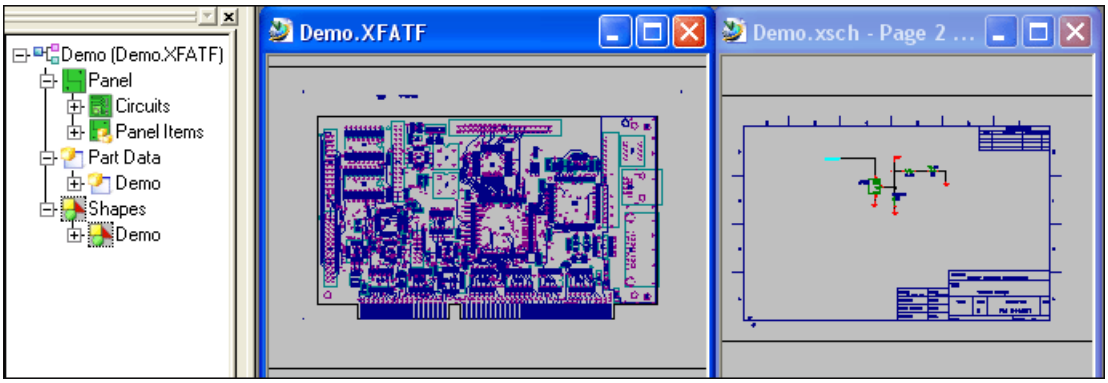
In this activity, you will:

- Learn how to cross probe between a PCB and its associated schematic design.

1. Open both demonstration files – *demo.xsch* and *demo.XFATF*.

2. On the **ECAD Base** toolbar, select **Cross Probe**  .

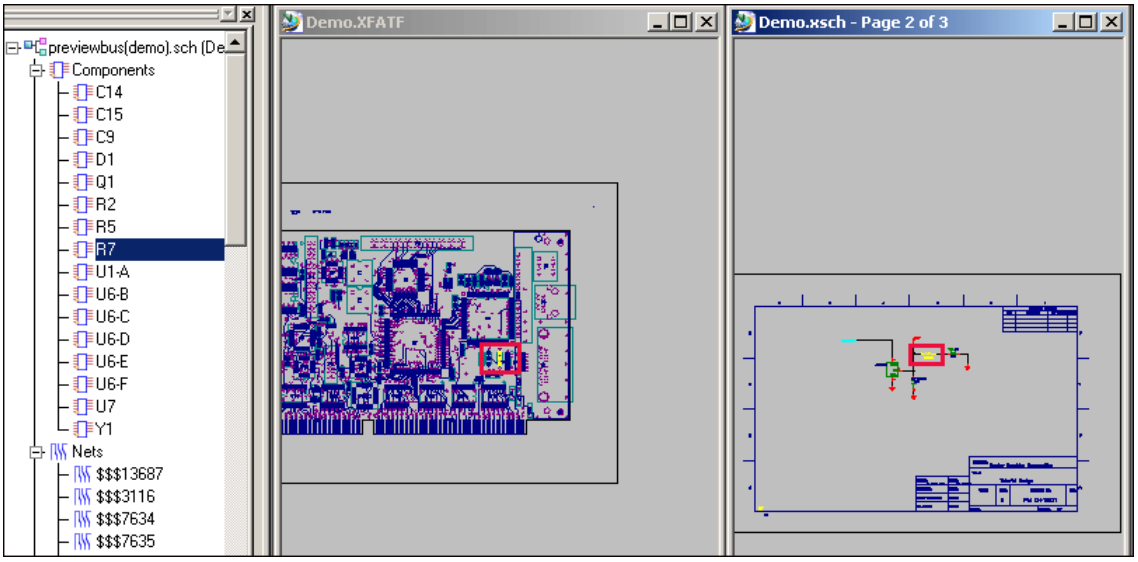
   In this example, the PCB and schematic files are displayed side-by-side so that you can select ECAD components in the **Assembly** view.



> Note:
>
> Notice that *Demo.XFATF* has focus in this example, and the component structure in the **Assembly** view is associated with *Demo.XFATF*. Change focus to *Demo.xsch* and the **Assembly** view changes to this file.

3. Set focus on *Demo.xsch* and highlight **R7** in the Components section in the **Assembly** view.

© 2025 Siemens



Note:

Notice the component **R7** is displayed on both the PCB and schematic design.

Try it yourself	
Find all common components using Cross Probe.	Navigate through each component listed in the **Assembly** view in this example. Hint: There are six common components associated with these two ECAD documents.

Create DFx custom rules

In this activity, you will:

- Learn how to create a custom rule file for Design for Assembly testing. This activity includes working with the DFx rules editor to modify the value of a rule used in a custom test.

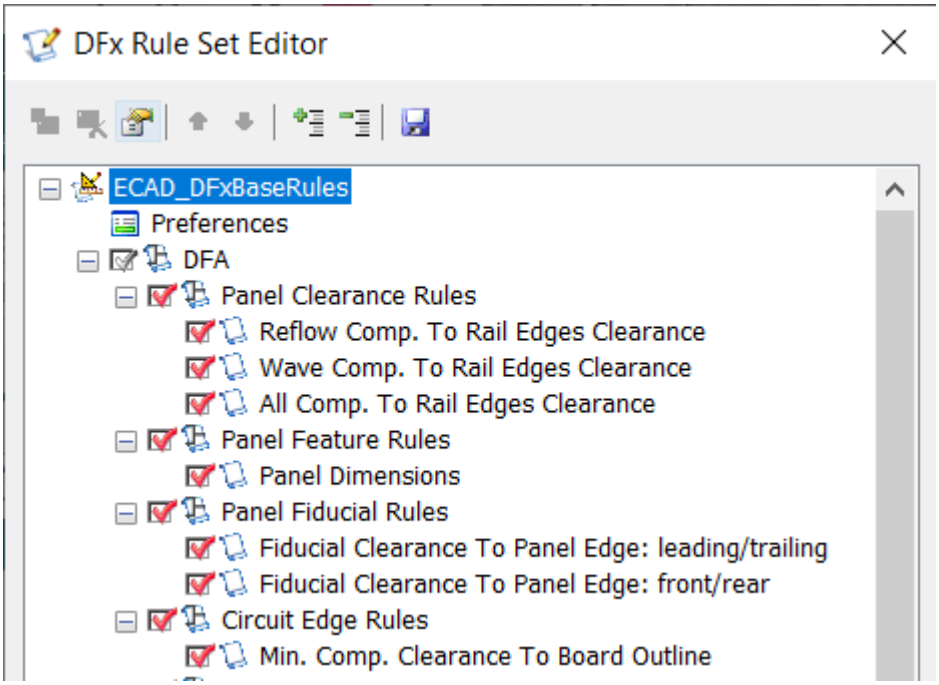
1. Open *demo.xfatf*.

2. On the **ECAD DFx** toolbar, click **Open Rules File** .

3. In the **Open Rules** dialog box, select a rules file and then click **Open**.

4. On the **ECAD DFx** toolbar, click **Edit Rules File** to open the **DFx Rule Set Editor** .

© 2025 Siemens



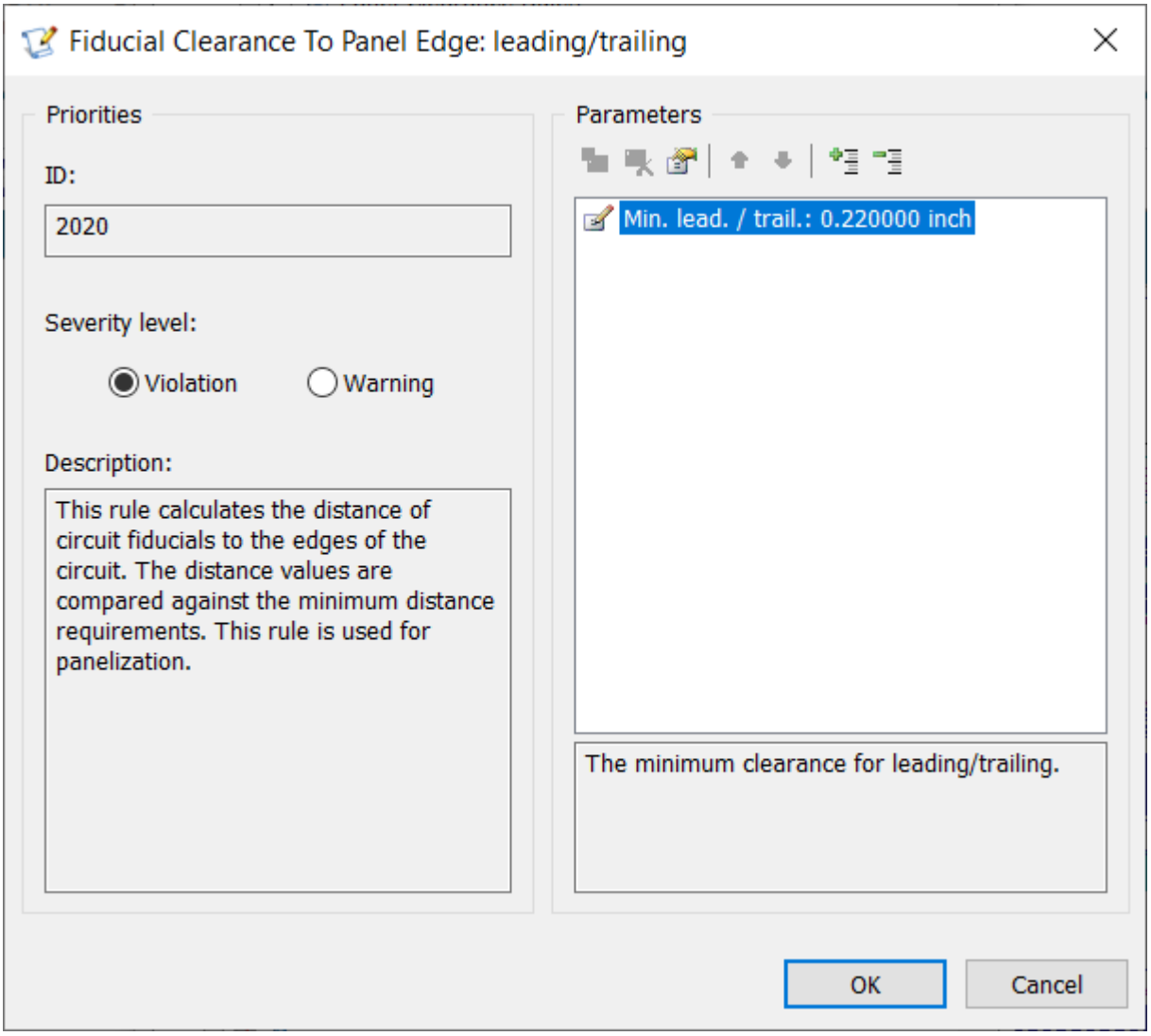
5. Create a custom **demo_test** rule.

   a. Clear **Panel Clearance Rules**, **Panel Feature Rules**, and **Circuit Edge Rules**.

   b. Double-click **Fiducial Clearance To Panel Edge: leading/trailing**.

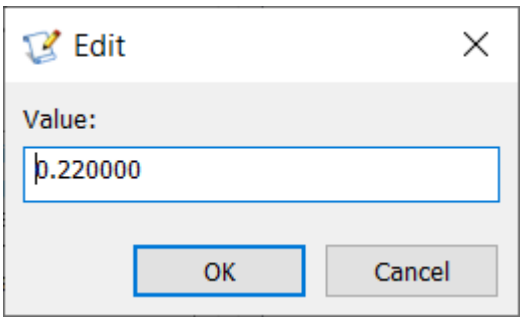
   c. (Optional) In the **Description** section, review the definition of this rule and in the **Parameters** section, review its current value.

© 2025 Siemens



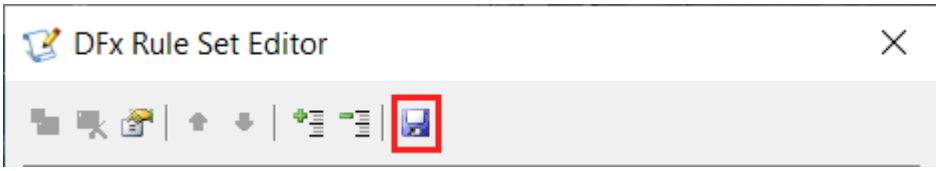
d. Double-click **Min. lead. / Trail.: 0.220000 inch**.

e. Change the value of this rule in the **Edit** dialog box.

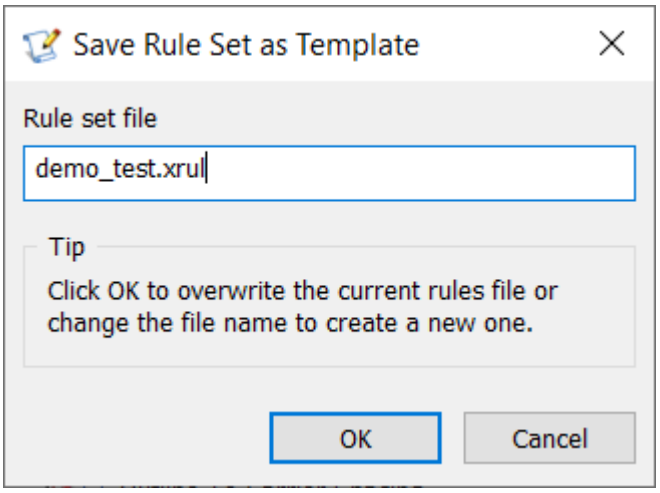


6. In the **DFx Rules Editor** dialog box, click **Save**.

© 2025 Siemens



7. In the **Save Rule Set as Template** dialog box, enter the name of your custom rule file.



8. To review your new rule, do the following:

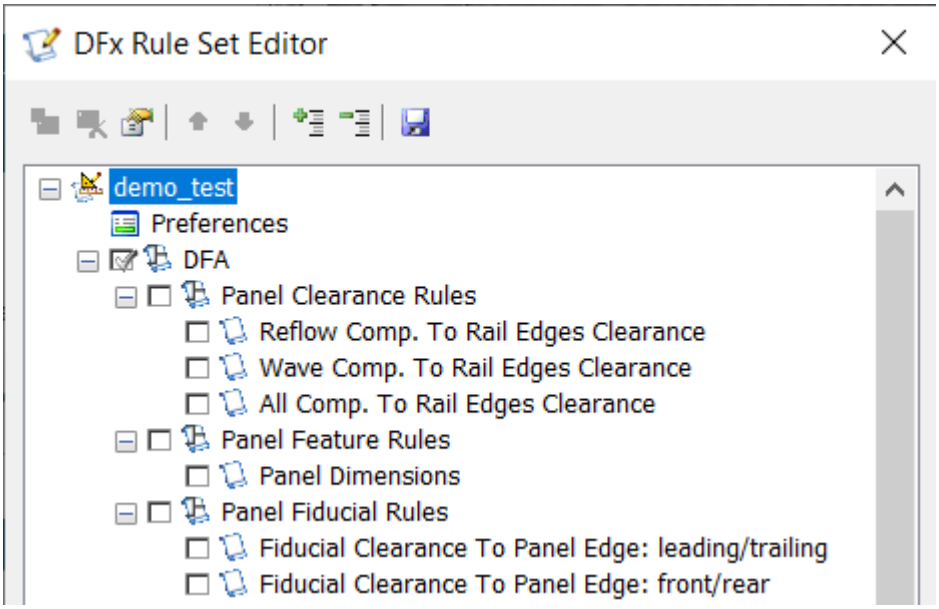
   a. On the **ECAD DFx** toolbar, click **Open Rules File**.

   b. Select **demo_test.xrul** from the list and then click **Open**.

   c. On the **ECAD DFx** toolbar, click **Edit Rules File**.

   d. Use the **DFx Rule Set Editor** dialog box to review the new rules in your custom rules file.

© 2025 Siemens



© 2025 Siemens