TEAMCENTER

ECAD Viewer

Teamcenter 2412

SIEMENS

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

Working with 2D snapshots

Managing ECAD markups

Using cross probing

Comparing ECAD documents

Working with Design for Manufacturing

Creating reports

Printing ECAD documents

System administrator reference for configuring ECAD default preferences

1. Overview of using the ECAD Viewer

Teamcenter lifecycle visualization is the application used by collaborators who do not have access to your corporate database. ECAD documents are imported from and exported to both applications to improve collaboration while maintaining database integrity. To use the ECAD Viewer, you must install Teamcenter lifecycle visualization. Refer to the Add lifecycle visualization to the Teamcenter rich client topic in *Installation* in the Teamcenter help.

The ECAD Viewer (Electrical Computer Aided Design) supports the Printed Circuit Board (PCB) design process by providing PCB and schematic viewing and markup tools and options. The PCB design and manufacturing industry relies on numerous CAD software programs that help designers create printed circuit boards. These native PCB and schematic files must first be translated to a neutral file format before they can be processed and opened in the viewer.

As a result, one of the first steps to consider before you work with the ECAD Viewer is to make sure your native CAD files are translated to the XFATF or XSCH neutral file format. However, it is possible that you can skip this step if your system administrator has already translated your data.

Using Teamcenter engineering process management, your team can translate over 30 PCB file formats, including those supplied by Cadence, Mentor Graphics, and Zuken, to a neutral file format that can be opened and manipulated in the ECAD Viewer. You can also translate EDIF 2.0 and 3.0 schematic file formats to the neutral file format. The software 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 vocabulary 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.

> Tip:
>
> Use the tutorial to get a practical understanding of many basic features and functions included with the ECAD Viewer.

2. Getting started with the ECAD Viewer

ECAD Viewer interface

Overview of the ECAD Viewer interface

The ECAD Viewer user interface includes several viewing frames, toolbars, and shortcut menu options. This section offers a brief introduction to these navigation and command option tools.

The ECAD Viewer interface appears when you open an ECAD document.

To activate ECAD toolbars, right-click the area just above the Viewing window to open a list of toolbars. Select and clear the toolbars as needed. The tasks you perform dictate which user interface frames that are exposed. Each frame contains shortcut menus that provide easy access to options associated with that user interface section. You access shortcut menu options by right-clicking inside a specific viewing section.

Introduction to ECAD toolbars

Exploring ECAD toolbars

Toolbars provide quick access to most ECAD options and features that you use to achieve collaborative design results. Toolbars help you with viewing options, markup tools, net navigation rendering, printing, creating reports, and cross probing reviews between a PCB and its associated schematic.

ECAD Viewing toolbar

Use the **ECAD Viewing** toolbar to quickly manage or change how you view ECAD documents.

	Use this option	To do this
	**Fit All**	Fit the entire design into the Viewing window.
	**Base View**	Size the view to the extents of the PCB dimensions.
	**Browse**	Display the name of ECAD objects as you move the cursor around the Viewing window.
	**Zoom Area**	Zoom to a specific area of the document.

	Use this option	To do this
	Seek	Center a point on the document.
	Pan	Move the document around the Viewing window.
	Zoom	Move closer to or farther from the document.
	Zoom In	Move the document closer to your view.
	Zoom Out	Move the document farther from your view.
	Rotate Counterclockwise	Rotate the document 90 degrees counterclockwise.
	Rotate Clockwise	Rotate the document 90 degrees clockwise.
	Flip Horizontal	Flip the document 180 degrees horizontally.
	Flip Vertical	Flip the document to a vertical position.
	Copy image	Copy the image in the Viewing window to the clipboard.
	Copy region	Copy a section of the image to the clipboard.

Snapshots toolbar

Use the **Snapshots** toolbar to add and manipulate snapshots that are displayed.

	Use this option	To do this
	Add	Add a snapshot of the Viewing window.
	Insert	Insert a snapshot between two existing snapshots.
	Delete	Delete the selected snapshot.
	Replace	Replace the selected snapshot with a snapshot of the Viewing window.

Note:

The **Toggle View** and **Preferences** options are only supported by 3D snapshots.

ECAD Base toolbar

Use the **ECAD Base** toolbar to manage specific ECAD functionality such as specifying side views, changing layer characteristics, navigating nets, creating reports, and setting cross probing for both the PCB and schematic.

	Use this option	To do this
	View Top and Bottom Side	Specify the order in which you view document layers.
	View Opposite Side	View the top or bottom side depending on the side that is currently showing.
	Layer Color	Change layer colors and layer visibility states. Use the **Layer Color** dialog box to manage these options.
	Search	Search for objects by using the options associated with the **Search** dialog box.
	Create Report	Open the **Reports** dialog box to create and run various ECAD reports.
	Cross Probe	Explore both the PCB and schematic in one Viewing window.
	Previous	Move the net marker to the previous point in navigating the net.
	Next	Move the net marker to the next point in navigating the net.
	Visual Compare	Compare the differences between two ECAD documents.
	PCB Properties	Display document property data by using the **Properties** dialog box.

ECAD Markup toolbar

Use the **ECAD Markup** toolbar options to manage how you want to mark up ECAD documents. In most cases, markup support applies to both the PCB and schematic documents.

	Use this option	To do this
	Enable Markup	Activate and deactivate the ECAD markup feature.
	Create New Markup	Open the **Markup Topic** dialog box. After selecting this option, click in the Viewing window and enter a markup topic.
	Select Markup	Select a markup that is displayed in the Viewing window.
	Freehand Line	Draw a freehand line.
	Leader Line	Draw a straight line and optionally add leader ends to the line.
	Ellipse	Draw an ellipse or circle on the ECAD document.
	Polygon	Draw a polygon on the ECAD document.
	Rectangle	Draw a rectangle or a square on the ECAD document.
	Text	Open the **Text Editor** dialog box and add text markups.
	Rubber Stamp	Open the **Rubber Stamp** dialog box. Select either auto-translate selected predefined text or select existing predefined text from a note file.
	Distance Measurement	(PCB only) Measure the distance between one point to another point, between one point to a specific location on a line, or between one line and another line.
	Radial Measurement	(PCB only) Measure the radial distance on any ring or circle.
	Clearance Measurement	(PCB only) Measure the minimum distance between two objects.
	Manhattan Length	(PCB only) Measure the distance between any point along the trace and via contained within the same net. Note: Manhattan length refers to the sum of the X- and Y-distances between the two selected points.

	Use this option	To do this
	Routed Length	(PCB only) Measure the distance between any point along the trace and via contained within the same net. Note: Routed length refers to sum of each segment along the trace between the two selected points.
	Markup Log	Open the **Markup Log** dialog box to display markup activity related to the ECAD document.
	Save Markups	Save the markup in the established dataset.
	Save Markups As	Save the markup in a different dataset.
	Markup Profile Preferences	Display the **Markup Profiles** dialog box which contains markup profile names and markup preferences.

ECAD DFx toolbar

Use the **DFx** toolbar to manage how you want to evaluate and analyze your designs against manufacturing rules.

	Use this option	To do this
	Open Rules File	Open the **Open Rules** dialog box and select a configuration file containing the rules for evaluating the open PCB.
	Edit Rules File	Open the **DFx Rule Set Editor** dialog box to review and modify Design for Assembly and Design for Test rules.
	Open Results File	Open a previously generated results file for the current PCB.
	Save DFx Files	Save the active rules file and results using the current dataset name.
	Save DFx Files As	Save the active rules file and results. You will be prompted for a new dataset name.
	Run DFx Rules	Start the design evaluation on the active PCB.

	Use this option	To do this
	Display DFx Summary Report	Open the most recently saved summary results report for the active PCB.
	Display DFx Detail Report	Open the most recently saved detail results report for the active PCB.

Note:

Most of these options require a DFx authoring license.

ECAD Multipage toolbar

Use the **ECAD Multipage** toolbar to navigate among pages in a multipage ECAD document.

	Use this option	To do this
	First Page	Go to the first page in a multipage file.
	Previous Page	Go to the previous page in a multipage file.
	Go To Page	Go to a specific page in a multipage file.
	Last Page	Go to the last page in a multipage file.
	Next Page	Go to the next page in a multipage file.

Introduction to ECAD shortcut menus

Exploring shortcut menus

Shortcut menus appear when you right-click specific sections of the ECAD interface. Use shortcut menus to quickly access features and functionality to help you work with many aspects affecting ECAD document review and collaboration. Different shortcut menus are displayed depending on what you are doing and where you are in the interface.

Markup shortcut menu

Use the Markup shortcut menu to edit markups, manipulate markup locations, and modify markup properties.

Use this option	To do this
Add to This Revision	Add markup revisions to any markup.
New Revision	Prepare to move a markup by putting its content on the clipboard.
Edit Topic	Copy the contents of the markup to the clipboard.
Iconize	Display expanded and selected markups as icons.
Expand	Display icon markups as fully expanded text.
Delete	Highlight and delete selected markups.
Edit Text	Add a markup to the PCB image.
Select All	Select all markups. This option is unavailable if a markup is already highlighted.
Properties	Display the markup properties (color, fill, font, and so forth) of one or more selected markups. If you authored the markup, you can change the markup properties; otherwise, the properties are view only.

Viewing shortcut menu

Use the Viewing shortcut menu to change how the ECAD document is displayed. For example, use it to change layer colors, viewing preferences, and properties.

Use this option	To do this
Fit All	Fit the entire design into the Viewing window.
Base View	Size the view to the extents of the ECAD dimensions.
Browse	Display the name of ECAD objects as you move the cursor around the Viewing window.
Zoom Area	Zoom to a specific area of the document.
Seek	Center a point of the design.
Pan	Move the board around the Viewing window.
Zoom	Move closer to or farther from the design.
Preferences	Change how designs are viewed and loaded into the Viewing window. When you click this option, you open the **Preferences** dialog box.

Use this option	To do this
Layer Color	Change layer colors. When you click this option, you open the **Layer Color** dialog box.
Properties	Display metadata about this design. When you click this option, you open the **Properties** dialog box.

ECAD Layers shortcut menu

Use the **ECAD Layers** shortcut menu options to easily change how you manage and display markups. The top level of the ECAD Layers page displays the dataset name and revision of the opened file.

Use this option	To do this
Save Markups	Save your markup by overwriting the current markup dataset.
Save Markups As	Save your markup and with a new dataset name. Note: The existing dataset can be overwritten.
Unload Markups	Remove markups from the Viewing window. Unsaved markups will be deleted unless you choose to save them.
Expand	Expand entries in the **ECAD Layers** view.
Collapse	Collapse entries in the **ECAD Layers** view.
All Markups On	Turn on all markups.
All Markups Off	Turn off all markups.
Expand Markups	Expand all markups.
Collapse Markups	Collapse all markups.
Find	Find markups.
Reset Visibility State	Reset layers to the initial view.

Assembly view shortcut menu

Use the **Assembly** view shortcut menu options to easily display additional component details and to find component layer objects.

Use this option	To do this
Attributes	Open the **Attributes** dialog box, which displays additional component details.
Expand	Point to and expand an object that contains sub-objects (noted by a plus sign to the left of the object).
Collapse	Point to and collapse an object that contains sub-objects (noted by a minus sign to the left of the object).
Find	Display the **Find** dialog box and enter objects of interest.
Instances	Display the instance of part data and shapes contained in the ECAD document.

ECAD compare shortcut menu

Use the **Compare** view shortcut menu options to highlight and view entity differences between two ECAD documents.

Use this option	To do this
Expand	Point to and expand an object that contains sub-objects (noted by a plus sign to the left of the object).
Collapse	Point to and collapse an object that contains sub-objects (noted by a minus sign to the left of the object).
Stop Compare	End the compare instance and delete all data in the **Compare** view.
View Differences	List differences between the two ECAD entities and, when selected, display the individual element in the Viewing window. Note: Layers are turned off. To turn layers on, select **Reset Visibility State**.
Find	Display the **Find** dialog box and enter objects of interest.
Reset Visibility State	Return the viewing mode to the state it was in prior to selecting **View Differences**.

DFx shortcut menu

The **DFx** view displays two distinct sections – the **rules** and the results sections. These sections help to manage and view DFx evaluation outcomes. Each section contains a shortcut menu that is slightly different. Use the DFx shortcut options to quickly manage how you work with tools for analyzing PCBs.

Use this option	To do this
Save	Save the current rule and result file. Note: If working with an existing file, the report will be saved to the same file name. If the file is new, the Save As dialog appears.
Save As	Save the current rule and result file to a new dataset (ECADDFX).
Expand	Highlight and expand the node structure.
Collapse	Highlight and collapse the node structure. Display expanded and selected markups as icons.
Turn All Rules On	(If the rules were off) Turn all rules on so they are displayed in the view, along with associated results, if any. Note: • Rules are on when the check box contains a red check mark. You can turn individual rules on or off by clearing the check mark. • Turning rules on or off helps to organize and display the results you want in the view. This action does not modify or customize the test or results.
Turn All Rules Off	Turn off all rules and results.
Find	Find terms (text or numbers) associated with the rules tree data.

Use shortcut keys to navigate documents

You can use your keyboard to quickly modify the view of your ECAD document.

- Do one or more of the following:
  - For **PCB** only:

Use this key	To do this
t	View the top PCB panel.
b	View the bottom PCB panel.

- For **schematic** only:

Use this key	To do this
Page Up	Move to the previous page.
Page Down	Move to the next page.
Home	Move to the first page.
End	Move to the last page.
n	(If on a page connector) Move to the next page connector.
p	(If on a page connector) Move to the previous page connector.

- For both **PCB** and **schematic**:

Use this key	To do this
n	Move to the next net navigation point.
p	Move to the previous net navigation point.
r	Rotate the image 90 degrees clockwise.
R	Rotate the image 90 degrees counterclockwise.
Arrow keys	Move the image incrementally in the direction of the arrow. Note: If a markup is selected, you will move the markup in the direction of the arrows.
c	Center the viewport in the Viewing window.
F	View data which is both inside and outside the bounding box.
f	View data only contained inside the bounding box.

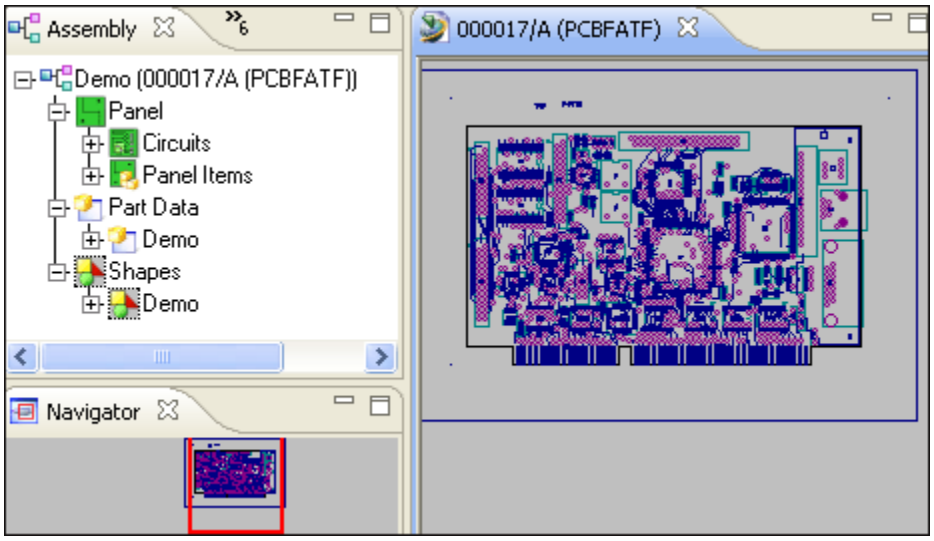
Overview of the ECAD Views

Lifecycle Viewer displays separate ECAD Views that help you navigate, search, and work with PCB designs, schematic representations, and design for assembly and test output.

Assembly view

The **Assembly** view displays a hierarchical structure of PCB circuit items, components, nets, panel items, part data, and shapes. To help visually recognize and identify PCB elements, it may be useful to review the **Assembly PCB icons** that are displayed in the view.

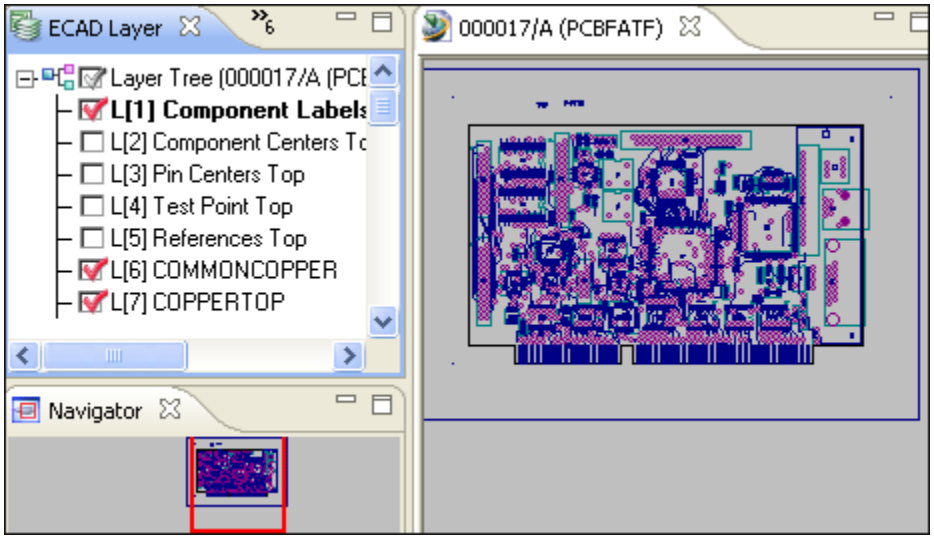
The view for schematic designs includes hierarchical structure for components, nets, shapes, and page views. For information on the icons associated with schematic designs, review the **Assembly schematic icons**.



ECAD Layers view

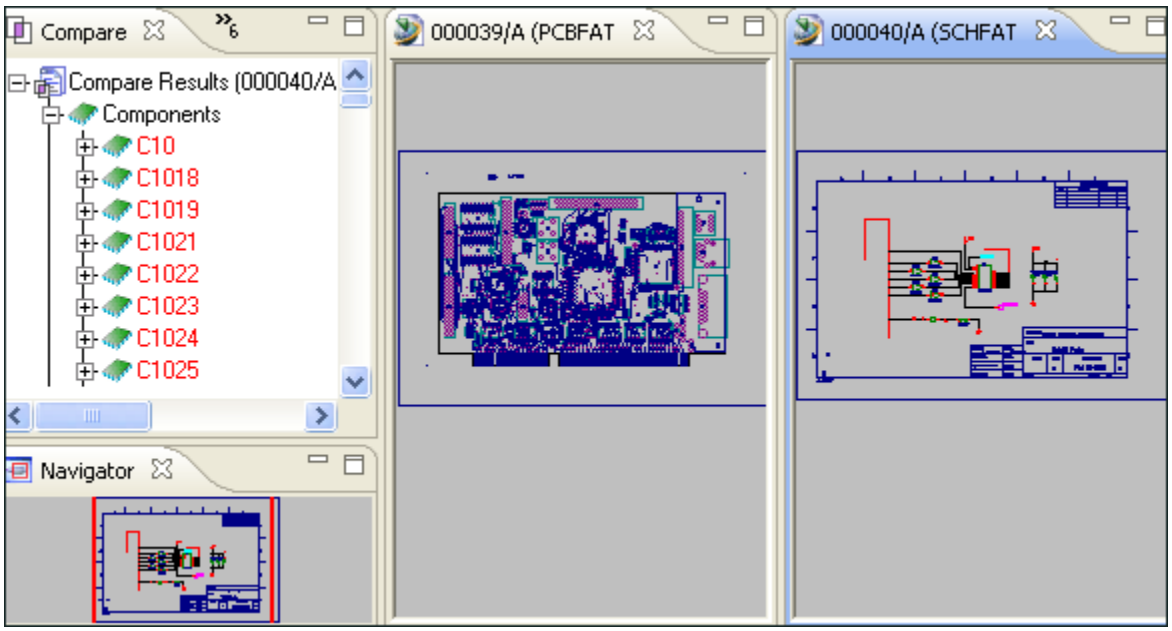
The **ECAD Layers** view displays the various layers associated with the viewable PCB and schematic document. You can modify layer preferences, for example visibility and layer color, and you can add an annotation (markup) on individual layers.

It may be useful to review the meaning of the **ECAD layer icons**.



Compare view

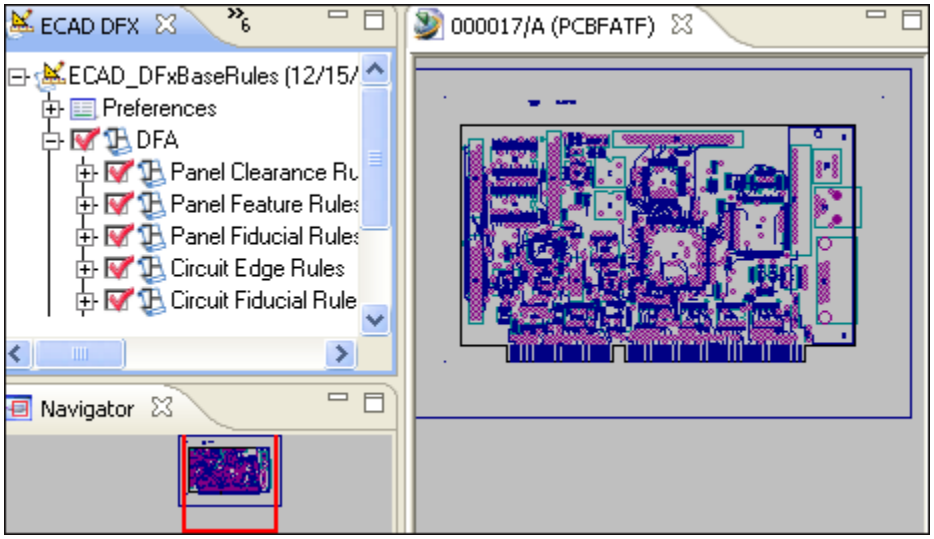
Use the **Compare** view to display ECAD document differences between two opened ECAD documents. You can display differences from one design revision to other revisions, and you can display differences between a PCB and its schematic design. When you highlight differences on the **Compare** view, the ECAD object is centered and highlighted in the Viewing window. You can also display differences by selecting **View Differences** from the compare shortcut menu.



ECAD DFX view

The **ECAD DFX** view provides both customized and default rules or preferences that you set. The rules are used to analyze specific characteristics of the designed PCB. The **ECAD DFX** view displays the rules and the results of the analysis.

Review the **DFx icons** associated with this view.



Object icons

Use the following icons to help visually understand objects that appear in the **Assembly** view, the **ECAD Layer** view, and the **ECAD DFX** view.

PCB Assembly icons

PCB Assembly icons	Icon description
	Root node
	Panel
	Circuit
	Circuit item
	Panel item
	Glue
	Mark
	Hole
	Component
	Nets

PCB Assembly icons	Icon description
	Pin
	Test points
	Vias
	Nail
	Edited nail
	Offrule nail
	Pin SMD
	Pin THT
	Unconnected pin
	Invalid via
	Invalid via with a nail
	Valid via
	Valid via with a nail
	Via with a nail
	Part data
	Part number
	Shapes
	Shape

Schematic Assembly icons

Schematic Assembly icons	Icon description
	Root node
	Circuit
	Circuit item
	Schematic component
	Schematic net
	Schematic bus
	Schematic pin
	Ground – Power
	Power
	Ground
	Part data
	Part number
	Shapes
	Page views
	Page item
	Sheet references
	Sheet
	Connector
	ECAD job – purpose
	Page

Schematic Assembly icons	Icon description
	Pagemark
	Pages

ECAD layer icons

ECAD layer icons	Icon description
	Root node
	A fully loaded and visible layer
	A fully loaded and partially visible layer
	A loaded layer that is not visible
	A locked layer

ECAD DFx icons

ECAD DFx icons	Icon description
	Circuit
	Rules file
	Rule preference
	Unsaved results
	Saved result report or individual result
	Rule group
	Rule
	DFA result
	DFT result

ECAD DFx icons	Icon description
	Number of access per net
	Clearance does not meet minimum
	A selected and loaded rule
	An unloaded but available rule

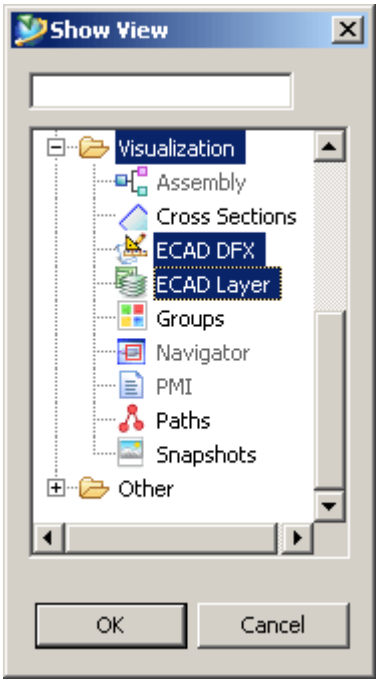
Open ECAD documents

From Teamcenter, you highlight the desired ECAD document to open, and then send it to the viewer.

1. From the specific **Home** folder location, find and highlight the desired ECAD dataset.

2. Right-click and choose **Send to→Lifecycle Viewer**.

3. (If necessary, show the ECAD views) Choose **Window→Show View→Other**. In the **Visualization** section, highlight **Assembly, ECAD DFX, and ECAD Layer**.



4. Activate a desired toolbar by choosing **Toolbars** and select the toolbar of interest.

Note:

You may need to give focus to the Lifecycle Viewer by highlighting it in the view.

Supported PCB and EDIF files

Supported PCB CAD files

Cadence Allegro	Mentor
CADES-G	Mentor Expedition
Cadnetix / Cadnetix EDIF	Valor ODB
Cad-ul Ariadne	Orcad Layout for Windows
Computamation Vutrax	Orcad Layout for DOS PCB386
ComputerVision Autoboard / THEDA	PADS-Perform, PowerPCB, PADS-PCB
DDE ECAD, Superman DDE	P-CAD PDIF
Dedale / DeCad	Prisma
DIF (C-Link)	Protel Protel DOS (2.8)
Docica	Protel 3 for Windows
Encore PCB (Scicards)	Tango
Gencad	Tango Pro
Gencam	Topcad
Harris EDA SCI-Cards CII	Ultiboard
Hewlett-Packard PCDS	VanGuard
IBM CBDS Version 4	Zuken-Redac Visula cadEpert
IBM CBDS Version 5, Unicad	Zuken-Redac CR3000
Integra	Zuken-Redac CR5000

Supported EDIF 2.0 and 3.0 schematic files

Cadence Allegro (capture, HDL)	Zuken CR5000
Mentor Expedition (dx designer)	Mentor Board Station
Cadence ORDAD (capture)	Mentor PADS (dx designer)

Translating CAD files

Overview of translating CAD files

Numerous native CAD applications and associated file formats exist in the industry. Teamcenter viewers can only process neutral CAD formats, and depending on the workflow of your environment, you may need to translate native CAD PCB and schematic data to this neutral file format.

> Note:
>
> We recommend that you contact your system administrator to determine the specific translation process for your company.

Translate CAD files to PCB neutral formats

> Note:
>
> For complete details about translation services, see Dispatcher — Deployment and Administration.

1. From My Teamcenter, select the **Home** folder to create a new subfolder.

2. Choose **File→New→Folder** and in the **New Folder** dialog box, enter a name for the folder, for example *my_pcb_folder*.

   The new folder appears within the **Home** folder structure.

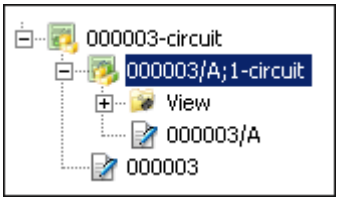
3. Select the new folder to create a new item.

4. Choose **File→New→Item** and in the **New Item** dialog box, select **PCBCircuit**.

5. In the **New Item** dialog box, click **Next**, then click **Assign** to name the new item, for example *circuit*, and then click **Finish**.

6. Click **Close**.

7. Expand the *my_pcb_folder* folder and select the *circuit* item revision.



> Note:
>
> The *my_pcb_folder* and *circuit* item revision are sample names.

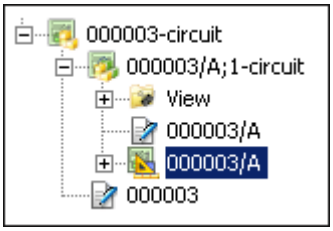
8. Choose **File→New→Dataset** and in the **New Dataset** dialog box, select **EDAGenPCBCAD** as the type for the new dataset.

9. Select **Import** to import your CAD specific PCB file.

10. In the **Import File** dialog box, select the CAD file you want to import into My Teamcenter and then click **Import**.

11. Click **OK** to close the **New Dataset** dialog box and your file is imported as the new dataset type.

12. Select the imported revision and choose **Translation Services→Translate** to translate your CAD file to the PCB neutral file type XFATF. Select the newly created **EDAGenPCBCAD** dataset.



13. In the **Select translator for dataset** dialog box, in the **Provider** section, select **UGS**.

14. In the **Translator** section, select a translator that can translate your native CAD file.

15. Click **OK** to start the translation process.

16. (Optional) Choose **Translation Services→Translation Administrator Console** to see the progress of the various translation stages.

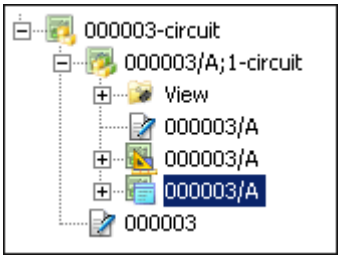
17. (Optional) In the **Translation Request Administration** dialog box, type an asterisk in the **Service** section and then click **Query**.

> Note:
>
> The asterisk is a wildcard symbol.

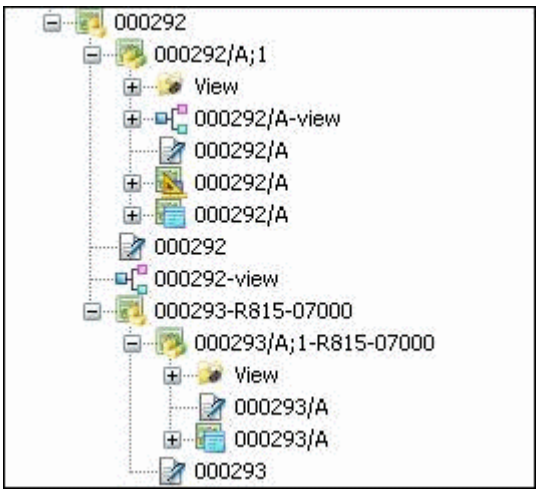
18. Select the desired translation request item and click **Refresh**. Close this dialog box when the translation is finished.

19. To display the new XFATF file, select the **circuit** node, right-click and choose **Refresh**.

The XFATF item revision appears in the **Demo** folder.



20. If the CAD file in a **PCBCircuit** item contains multiple PCB designs (for example, panelized circuits), after translation the child items of the **PCBCircuit** item are created. Each child item contains translated version of an individual PCB design.



Translate CAD files to Schematic neutral formats

> Note:
>
> For complete details about translation services, see Dispatcher — Deployment and Administration.

1. From My Teamcenter, select the **Home** folder to create a new subfolder.

2. Choose **File→New→Folder** and in the **New Folder** dialog box, enter a name for the folder, for example *my_ecad_folder*.

   The new folder appears within the **Home** folder structure.

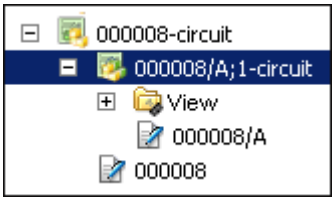
3. Select the new folder to create a new item.

4. Choose **File→New→Item** and in the **New Item** dialog box, select **PCBCircuit**.

5. In the **New Item** dialog box, click **Next**, then click **Assign** to name the new item, for example *circuit*, and then click **Finish**.

6. Click **Close**.

7. Expand the *my_ecad_folder* folder and select the *circuit* item revision.



> Note:
>
> The *my_ecad_folder* folder and *circuit* item revision are sample names.

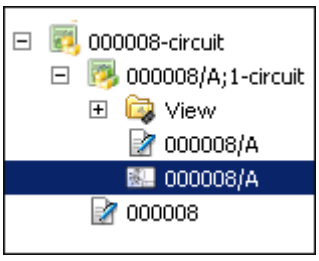
8. Choose **File→New→Dataset** and in the **New Dataset** dialog box, select **EDAGenSchem** as the type for the new dataset.

9. Select **Import** to import your CAD file.

10. In the **Import File** dialog box, select the CAD file you want to import into My Teamcenter and then click **Import**.

11. Click **OK** to close the **New Dataset** dialog box and your file is imported as the new dataset type.

12. Select the imported revision and choose **Translation Services→Translate** to translate your CAD file to the neutral format type **XSCH**. Select the newly created **EDAGenSchem** dataset.



13. In the **Select translator for dataset** dialog box, in the **Provider** section, select **UGS**.

14. In the **Translator** section, select a translator that can translate your native CAD file.

15. Click **OK** to start the translation process.

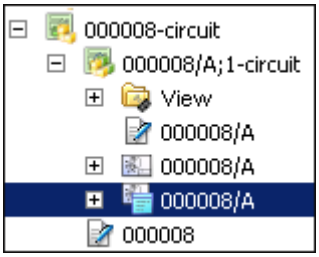
16. (Optional) Choose **Translation Services→Translation Administrator Console** to see the progress of the various translation stages.

17. (Optional) In the **Translation Request Administration** dialog box, type an asterisk in the **Service** section and then click **Query**.

> Note:
>
> The asterisk is a wildcard symbol.

18. Select the desired translation request item and click **Refresh**. Close this dialog box when the translation is finished.

19. To display the new XSCH file, select the **circuit** node, right-click and choose **Refresh**.



3. Viewing ECAD documents

Overview of viewing ECAD documents

Use the following options to help organize how you load and view ECAD documents. Whether you are working with a PCB, a schematic, or with a PCB design analysis, use these options to quickly manipulate the view so you can display the documents and work with them as needed. For example, numerous viewing options exist when you open ECAD documents in the **Assembly** view, in the **ECAD Layer** view, or when you want to view the rules and results of PCB analysis using the **ECAD DFX** view. In addition to immediate, but temporary viewing preferences, you can also configure default preferences that load and set viewing conditions for every document that you open.

Working with standard ECAD viewing options

Display the entire image in the Viewing window

On the **ECAD Viewing** toolbar, click **Fit All**  .

The ECAD document, including data outside the bounding box, is displayed in the Viewing window.

Display the base view

Use the **Base View** option to hide markup images that are embedded outside the bounding box.

- On the **ECAD Viewing** toolbar, click **Base View**  .

Rotate the view

You may find it helpful to rotate the ECAD document in order to get a better view of specific objects.

On the **ECAD Viewing** toolbar, choose one of the following rotation tools:

- **Rotate Counterclockwise**  or **Rotate Clockwise** 

  The view rotates by 90 degrees.

- **Flip Horizontal**  or **Flip Vertical** 

  A mirror image of the board is displayed in the Viewing window.

Move the entire image around the Viewing window

1. On the **ECAD Viewing** toolbar, click **Pan**  .
2. From inside the Viewing window, drag the image to the desired position.

> Tip:
>
> Quickly change to the zoom mode by holding Shift and moving the right mouse button.

Change the center point of the view

You may need to move the center of focus so that you improve the view of various image areas.

1. On the **ECAD Viewing** toolbar, click **Seek**  .
2. Click the desired point on the image.

   The point you select becomes the new center point in the Viewing window.

Zoom in and out of a specific area

When you select a specific area, you can zoom in or out of the area to change the viewing details.

1. On the **ECAD Viewing** toolbar, click **Zoom Area**  .

   Inside the Viewing window, your cursor appears as a magnifying glass.
2. Draw a rectangle around the desired area.

> Tip:
>
> Once the Zoom area is selected, you can zoom in or out of the area and you can pan (move) the image using your mouse. On the **ECAD Viewing** toolbar, click **Zoom**  and move the cursor up and down to change the zoom. To pan the image, press the Shift key and move the cursor.

Zoom by incremental amounts

On the **ECAD Viewing** toolbar, select one of the Zoom tools.

- Click **Zoom In**  one or more times.

The image is increased by incremental amounts.

- Click one or more times on **Zoom Out**  one or more times.

  The image is decreased by incremental amounts.

Working with advanced ECAD viewing options

Working with advanced viewing options

ECAD documents can consist of numerous elements and objects. As a result, tasks such as viewing specific PCB sides, navigating nets, highlighting specific objects, searching for and finding object instances and object names are critical to the PCB design and review process.

View the opposite (top or bottom) side of the PCB

Use **View Opposite Side**  to switch between the top and bottom panels and lock the layers associated with the opposite panel.

On the **ECAD Base** toolbar, do one of the following:

To do this	Use this option
View the top panel. The top panel is displayed and the bottom panel layers are locked.	Turn off **View Opposite Side** 
View the bottom panel. The bottom panel is displayed and the top panel layers are locked.	Turn on **View Opposite Side** 

> Note:
>
> Schematic design does not support this option.

View the top and bottom side of the PCB

Use **View Top and Bottom Side**  to switch between locking and unlocking the opposite panel.

On the **ECAD Base** toolbar, do one of the following:

To do this	Use this option
View both the top and bottom panels. The top panel is forward (closest to you) and the layers are unlocked.	Turn on **View Top and Bottom Side** and turn off **View Opposite Side**
View both the top and bottom panels. The bottom panel is forward (closest to you) and the layers are unlocked.	Turn on both **View Top and Bottom Side** and **View Opposite Side**

Note:

Schematic design does not support this option.

Navigate nets and page connectors

PCB boards and schematic representations can be saturated with ECAD elements. It can be difficult to follow the direction and details of nets on your ECAD document. Using net navigation, when you highlight a net, a marker displays on a navigation point of the net. You can then navigate along the net moving the marker from key point to key point. When navigating nets on the schematic, you can navigate to different pages, depending on the course of the net.

1. In the **Assembly** view, select the desired net.

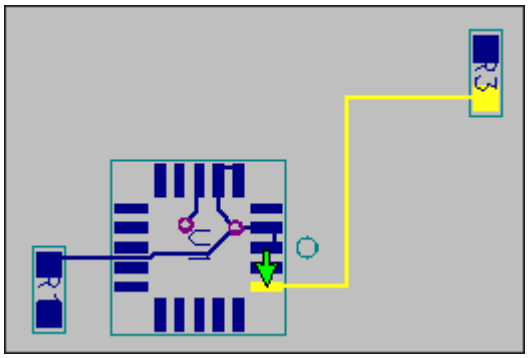
2. On the **PCB Base** toolbar, select either **Next** or **Previous** .

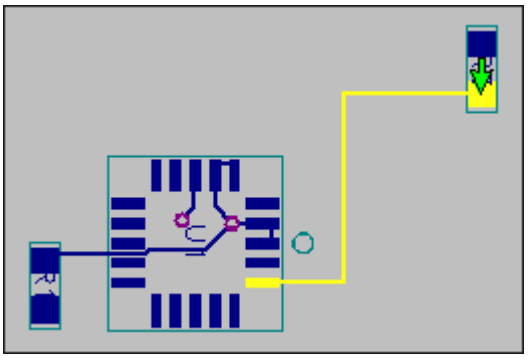
3. (Optional) Use these same navigation steps when the schematic design contains a page connector.

Tip:

Use keyboard shortcuts to move to the next or previous net by using the *N* or *P* keys.

Example:





Note:

When net navigating, you may navigate to an object that is located on a layer that you have configured as invisible. The object displays in the layer color you previously configured for invisible layers.

Highlight and center component objects

In the **Assembly** view, select a PCB object (a component, net, pin, part data, or shape).

The object is centered and highlighted in the Viewing window and highlighted in the **Assembly** view.

Tip:

You can also **browse** over objects to highlight and identify details about them.

Find instances of part data and shapes

At times, one object will represent several instances when you view an ECAD document. For example, both part items and shapes can be repeated in a document as multiple components. You can use the shortcut menu to display all instances of these objects.

1. In the **Assembly** view, select a PCB shape or part data.

2. Right-click and select **Instances**.

3. (Optional) Highlight one or more instances contained in the **Instances** dialog box.

   The object is highlighted and centered in the Viewing window.

4. (Optional) Double-click an instance of interest or click **Attributes** to view the attributes of the object.

> Tip:
>
> You can also **browse** over PCB objects to highlight and identify details about them.

Display file properties

Use **Properties** to display important data and information about the metadata associated with the basic PCB or schematic file.

On the **ECAD Base** toolbar, click **Properties**  .

The **Properties** *<file name>* dialog box appears with metadata about the ECAD document.

> Note:
>
> The **Properties** dialog box displays the following metadata:
>
> - Design of the document
> - The description of the document
> - The creator of the document
> - The date the document was created
> - If the document was modified, the name of the person who modified it
> - The date of the last modification
> - Revision status
> - The dimensions of the document
> - The unit of measurement
> - The read/write status of the document

Display object attributes

You may find it helpful to quickly view details about component attributes associated with the opened ECAD document. For objects that support attributes, you can highlight the object in the **Assembly** view and the object is highlighted and centered in the Viewing window.

Right-click a selected object in the **Assembly** view and select **Attributes**.

The **Attributes** dialog box displays the attributes of the selected object.

> Note:
>
> Not all objects support attributes, for example, header objects such as the introductory nets.

Display object names

1. On the **ECAD Viewing** toolbar, click **Browse**  .

2. Move your cursor over the object and its name is displayed in a text box.

3. (Optional) Click an object in the Viewing window to highlight it on the PCB or schematic as well as in the **Assembly** view.

4. (Optional) Double-click the object in the Viewing window to highlight it and to open the **Attributes** dialog box.

   The **Attributes** dialog box displays additional data and information about the object.

You can also select an object in the **Assembly** view to highlight and center it in the Viewing window.

Searching ECAD documents

Overview of searching ECAD documents

Use search tools to find components, nets, pins, and more. Objects that match your search conditions are displayed in the **Objects found** section of the **Search** dialog box.

Use standard search tools to create the search using saved search filters or using specific object types and then entering search criteria.

Use advance search tools to create searches using filters. You can create, edit, and delete search filters. Use filters to specify greater search details for various objects or for detailed conditions. For PCB design files, for example, you can set the board side for the search. Or, for both PCB and schematics you can set or restrict the rotation orientation of the object.

For both standard and advanced searches, when you highlight objects in the **Objects found** section the objects are also highlighted in the Viewing window. You can also double-click an element in the **Objects found** section to display its attributes in the **Attributes** dialog box. If attributes do not exist for an object, the dialog box is empty.

Standard searches

1. On the **ECAD Base** toolbar, click **Search**  .

2. In the **Search** dialog box, from the **Saved Filters** section, accept the default filter.

   If you have previously created and used advance searches, saved search filters are also available.

3. In the **Simple Search Type** section, select an ECAD object for your search.

4. In the **Search for** section, type additional details for your search.

   Tip:
   - Wildcard symbols are supported.
   - Use the semicolon to add a series of objects to your search. For example, *C100;C105*.
   - To search for all possible conditions, in **Search for**, type an asterisk.

5. Click **Search**.

   Objects that match your search are displayed in the **Objects found** section of the dialog box.

6. (Optional) Click an object in the **Objects found** section.

   The object is centered and highlighted in the Viewing window, and it is highlighted in the **Assembly** view.

7. (Optional) Click **Highlight All** to highlight all objects in the **Objects found** section.

Advanced searches

1. On the **ECAD Base** toolbar, click **Search**  .

2. In the **Search** dialog box, click **Advanced**.

3. (Optional) In the **Saved Filters** section, select the name of a saved search filter.

   The filter values appear in the **Filter** section. You can run the search, create additional filter conditions, or delete filter conditions.

> Tip:
>
> In the **Saved Filters** section, click **Delete** to remove the saved filter data.

4. In the **Edit Filter** section, select an ECAD object from **Type**.

> Note:
>
> Optional filter conditions are associated with the ECAD object selected in **Type**. For example, if you select **Components**, component filter conditions are displayed. If you select **Nets**, net filter conditions are displayed.

5. In the **Filter** section, select **Filter** and then click **Create**.

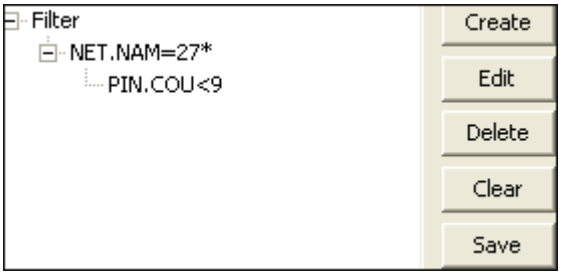
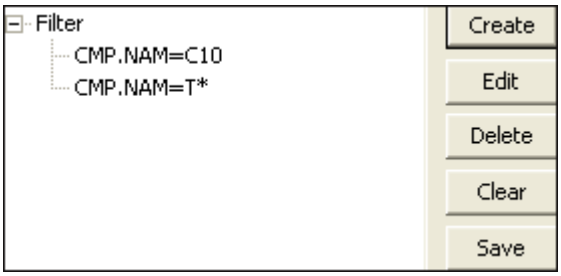
6. In the **Edit Filter** dialog box, select the **Criterion** and search values for the search type.

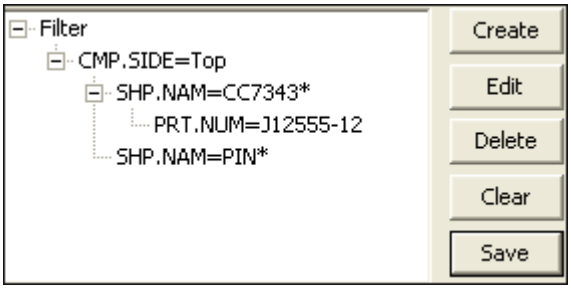
7. (Optional) To add search filters to your search do the following:

   - Highlight a filter and click **Create**.
   - In the **Edit Filter** dialog box, select the **Criterion** and search values for the search type.
   - Repeat these steps for additional search conditions, making sure that you highlight the correct filter for the desired search outcome.

> Example:
>
> For these examples, open the *demo.xfatf* file.

To create this search	Do this
Search for all nets named 27* **AND** those nets with 9 or less pins.	a. Highlight **Filter** and then click **Create**. b. In the **Edit Filter** dialog box, select **Net** and in the value section type **#27***. c. Highlight **NET.NAM=#27***, and click **Create**. d. In the **Criterion** section, select **Number of pins**, and then select the less than symbol. Next, select **9**. e. Click **Search**. The search results appear in the **objects found** section.

To create this search	Do this
	The filter structure should look like the following graphic. 
Search for a component named C10 **OR** all components starting with the letter **T**.	a. Highlight **Filter** and then click **Create**. b. In the **Edit Filter** dialog box, select **Component Name** and then select **C10** from the list. c. Highlight **Filter** again, click **Create**, and then type **T*** in the value section. d. Click **Search**. The search results appear in the **objects found** section. The filter structure should look like the following graphic. 
Search the top side for shape names **CC7343*** **AND** an object with part number **J12555–12**, **OR** a shape named object **PIN***.	a. Highlight **Filter** and then click **Create**. b. In the **Edit Filter** dialog box, in **Criterion**, select **Side** from the list, and then select **Top** from the list. c. Highlight **CMP.Side=Top**, click **Create**, then select **Shape Name** in the **Criterion** section. Scroll to **CC7343**, and add an asterisk to **CC7343**.

To create this search	Do this
	d. Highlight **SHP.NAM=CC7343***, click **Create**, and then select **Part Number** from the **Criterion** section. Scroll to **J12555–12** and click **OK** e. Highlight **CMP.SIDE=Top** again, click **Create** and add **Shape Name** from the **Criterion** list. Then, type **PIN*** in the value section. f. Click **Search**. The search results appear in the **objects found** section. The filter structure should look like the following graphic. 

8. (Optional) To modify an existing filter, highlight the filter and click **Edit**, and make changes in the **Edit Filter** dialog box.

9. (Optional) Click **Delete** to remove a search filter and its child filters.

10. (Optional) Click **Clear** to remove all filters.

11. (Optional) Click **Save** to save the conditions of this search.

    Type the name of the saved search filter. The name appears in the **Saved Filter** section.

12. Click **Search**.

    Objects that are found in the search are displayed in the **objects found** section of the **Search** dialog box.

Note:

- The first object that is found is highlighted in three different locations: in the **objects found** section, in the Viewing window, and in the **Assembly** view.
- Double-click an object in the **objects found** section and the **Attributes** dialog box displays the attributes associated with this entity.

13. (Optional) Click **Highlight All** to display all **objects found** in the Viewing window.

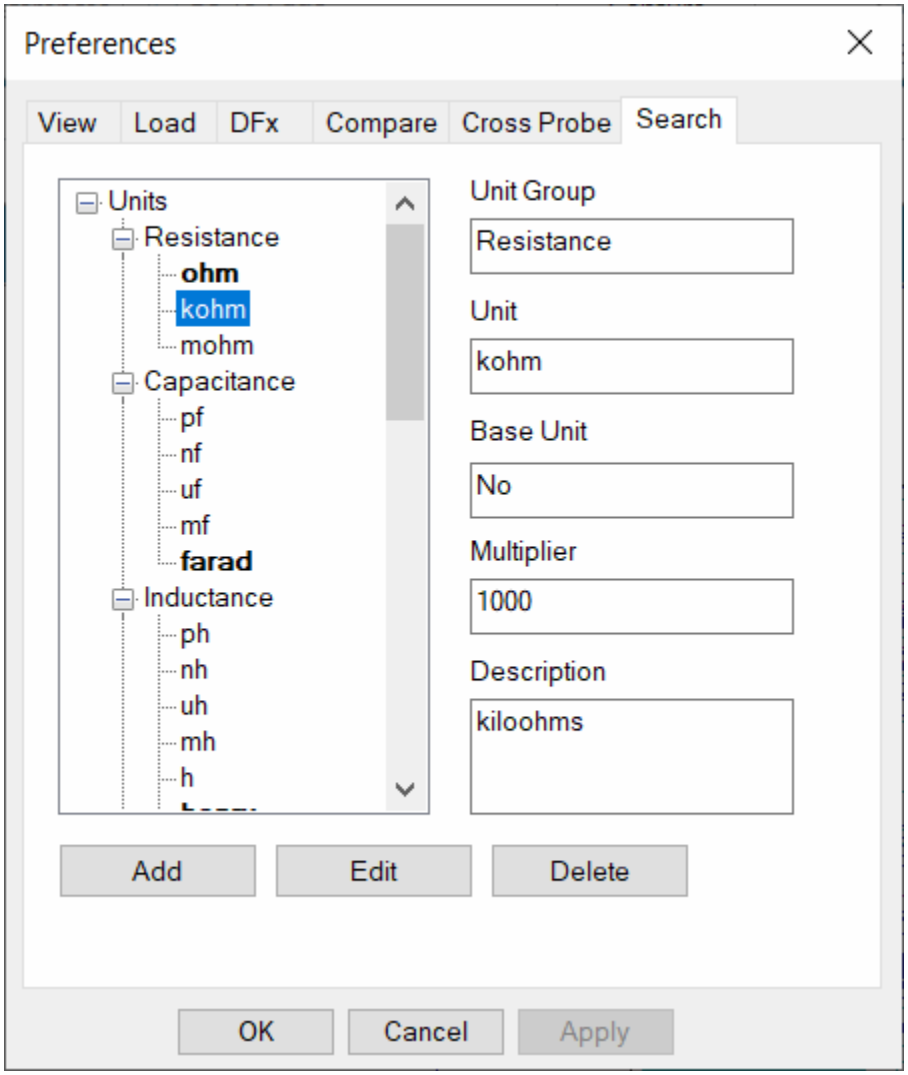
    The objects found are not highlighted in the **Assembly** view.

Use search preferences to change unit values

Search preferences consist of established unit values. Unit measurements include resistance, capacitance, voltage, and current, among other units. Each unit has standard values. For example, for resistance ohm, kohm, and mohm, values are automatically provided. Use search preferences to edit, delete, and add unit attributes and their values.

Caution:

Unit definitions and values are PCB board and schematic dependent. If you want to include unit measurements in your search, you must create unit definitions and values if they are not already included in the preferences.



1. Open an ECAD document.
2. Right-click in the Viewing window and select **Preferences**.
3. Click the **Search** tab.
4. Highlight the unit attribute of interest, and click **Add** and then, in the **Unit Preference** dialog box, define the new attribute and its values.
5. (Optional) Highlight the unit attribute of interest, and click **Edit** and then, in the **Unit Preference** dialog box, make changes to available fields.
6. (Optional) Click **Delete** to remove the attribute.

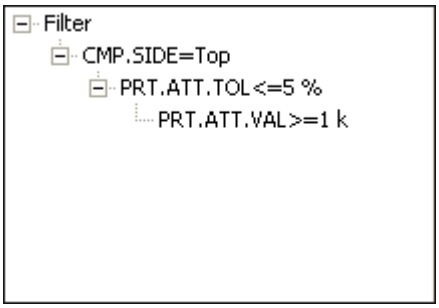
Example:

The example uses the file *demo.xfatf*.

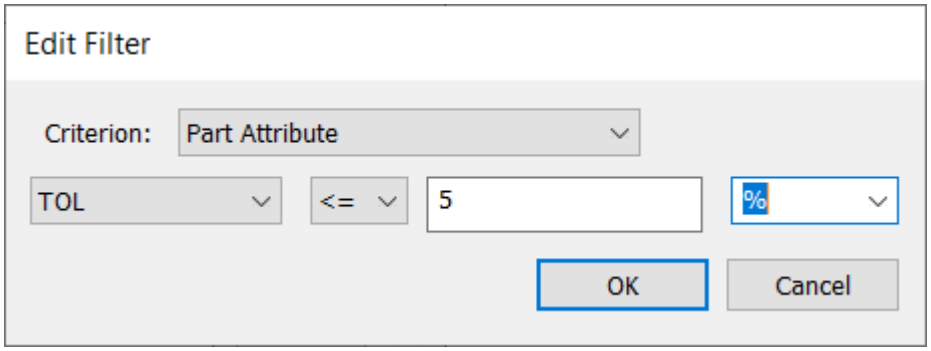
This search looks at the **Top** side of the board for part attributes with a tolerance that is less than or equal to 5%, and a value greater than or equal to 1k (1kilo ohm).

Since many of these search values are not defined in established search preferences, you need to add them.

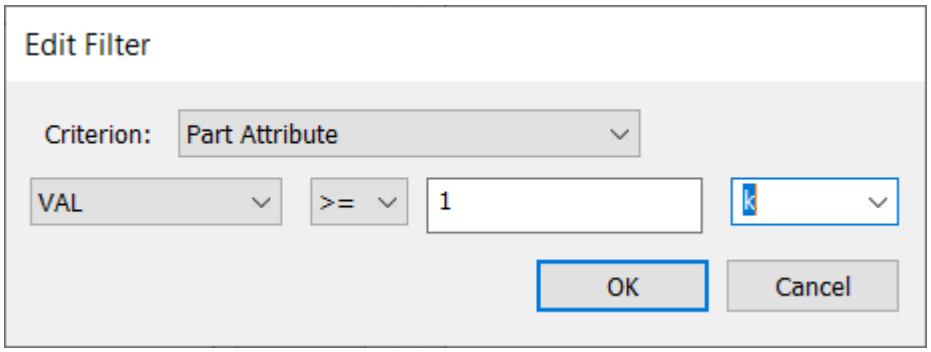
1. Open **Preferences** and click the **Search** tab.
2. Highlight **Resistance** and click **Add**.
3. Type the following values:
   - In **Unit**, type **k**.
   - In **Multiplicator**, type **1000**.
   - In **Description**, type **kiloohms**.
4. On the **ECAD Base** toolbar, click **Search**  .
5. In the **Search** dialog box, click **Advanced**.
6. Process the advance search using the following search structure:



   - For **Part Attribute** values for **TOL**, follow this example:



- For **Part Attribute** values for **VAL**, follow this example:



Change the color of a layer

1. On the **ECAD Base** toolbar, click **Layer Color** .
2. From the **Layer Color** dialog box, highlight the desired layer.
3. From the **Color** list, choose a layer color.
4. Do one of the following:
   - Click **Apply** to make your change and to keep the **Layer Color** dialog box open for more changes.
   - Click **OK** to make change and close the **Layer Color** dialog box.

Note:

Click **Default Colors** to reset layer colors to their initial state.

Change the visibility state of a layer

Modify the visibility state of a layer in one of these ways:

From the	Do this
ECAD Layer view	Select or clear the active layers check box to hide or make visible the desired layer. Note: To hide all layers clear the top node check box in the **ECAD Layer** view.
ECAD Base toolbar	1. Click **Layer Color**. 2. From the **Layer Color** dialog box, highlight the desired layer. 3. Click **Visible/Invisible** to specify your change. 4. Select **Apply visibility changes to markup layers**. 5. Do one of the following: • Click **Apply** to make your change and to keep the **Layer Color** dialog box open for more changes. • Click **OK** to make your change and to close the **Layer Color** dialog box.
ECAD Layer view	Select or clear the active layers check box to hide or make visible the desired layer. Note: To hide all layers clear the top node check box in the **ECAD Layer** view.

Note:

All layers must have a color property even if you set the layer to invisible. You may need to display objects even if they are on invisible layers.

Specify ECAD viewing preferences

Use options in the **Preferences** dialog box to specify default viewing behavior.

1. Right-click anywhere in the Viewing window, and choose **Preferences**.

2. In the **Preferences** dialog box, click the **View** tab and do one or more of the following:

Use this option	To do this
Show border	Display or hide a border surrounding the document.
Flip mouse direction for zoom	Specify how you use the mouse to zoom in the Viewing window. Note: The default behavior for **Zoom** is to zoom out by dragging *up* and zoom in by dragging *down*. You can reverse this behavior by selecting this option. The option also reverses the direction you roll the mouse wheel to zoom in or out. This one preference affects zooming in 2D and 3D Viewing windows.
Background Color	Specify the background color on which the document is displayed.
Feedback Color	Specify the color that appears when you initiate adding a markup or when you highlight an area to zoom. The initial default feedback color is **red**.
Initial View Rotation	Select how you want to rotate images when you open them in the Viewing window. Choose one of the following options: • **No Rotation** • **90 Degrees CW** • **90 Degrees CCW** • **180 Degrees**
Initial View Mode	Specify the default view mode. The initial default view mode is **Pan**.
Units	Specify the default unit of measurement.

3. Click **Apply**.

Configure loading preferences

Use options in the **Preferences** dialog box to specify default loading characteristics. These preferences are used to determine how you want files to be opened.

1. Right-click anywhere in the Viewing window, and click **Preferences**.

2. In the **Preferences** dialog box, click the **Load** tab and do one or more of the following:

Use this option	To do this
Display markups when loaded	Display all embedded markups on all visible layers when you open the file. Note: **Display markups when loaded** is the initial default value. When you clear this setting, markups are loaded but they are not visible.
Show markups as icons	Display markups as icons when you open a file. Note: The default value is to display markups as full text.
Initial Viewport – Base View	Specify the opening view is set to the **Base View**. Note: **Base View** is the default value.
Initial Viewport – Fit All	Specify the opening view is set to **Fit All**.
Component Origin – CAD origin	Display the component marker at the origin of the component.
Component Origin – Component center	Display the component marker at the center of the component.
Font Name	Change the font for the text associated with the ECAD document. Note: The default font is *Hershey*. You can use this feature to change the font to match your locale.

3. Click **Apply**.

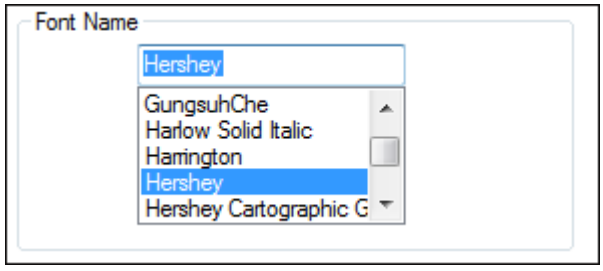
Change the font in ECAD documents

You can change the font in ECAD documents. The default font is *Hershey*. Use **Preferences** to change the text font as needed. You can also change the default font by modifying the *Initialization.xml* file.

1. Right-click anywhere in the Viewing window, and click **Preferences**.

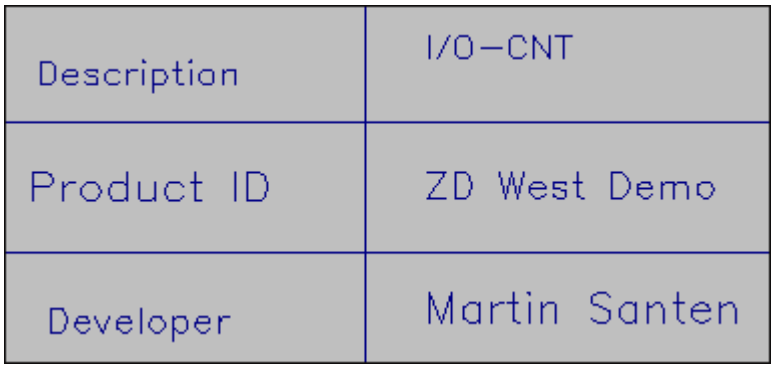
2. In the **Preferences** dialog box, click the **Load** tab.

3. From the **Font Name** list, select a font.



4. To make the font change take place, close and reopen the ECAD document.

   The following snippet uses the *zuken.xsch* file, which is located in the installation **Examples** folder.



Hershey font

Times New Roman font

5. (Optional) To change the default font for ECAD documents, do the following:

a. Navigate to *<installation_folder>\Teamcenter9\Visualization\etc* and open *Initailization.xml*.

b. In the ECAD section, change the font value to your new default font.

```
<Section_ECAD>
  <Version value="3" />
  <TextFont overwrite="no" value="Hershey"/>
  <ViewPreferences>
    <!-- OverWriteViewPreferences: "yes" "no" -->
    <OverWriteViewPreferences state="No" />
```

4. Working with 2D snapshots

Overview of working with 2D snapshots

Snapshots capture the current state of your view. A snapshot saves view magnification, position, orientation, markup layer visibility, and the current page for multiple page images.

Note:

- Snapshots do not save the layer visibility state of images with multiple image layers.
- The shortcut menu option **Save as a Teamcenter Snapshot** is only available in the Lifecycle Visualization viewer if you are working with data from Teamcenter.

Snapshots appear as thumbnail images in the **Snapshots** view.

You can double-click a snapshot to make the current view match the snapshot.

Snapshot commands are in the **Snapshots** toolbar, in the **Snapshots** menu, and in the shortcut menu that you access by right-clicking inside the **Snapshots** view.

Add a 2D snapshot

1. Manipulate the contents of the Viewing window until you create a view you want to save.
2. Do one of the following:
   - In the **Snapshots** view, right-click and choose **Add**.
   - On the **Snapshots** toolbar, click **Add**  .

Insert a 2D snapshot

1. Click the **Snapshots** view to activate Snapshots.
2. In the **Snapshots** view, select a snapshot.
3. Manipulate the image until the Viewing window displays the view of interest.
4. Do one of the following:
   - In the **Snapshots** view, right-click a snapshot and choose **Insert**.

- On the **Snapshots** toolbar, click **Insert**  .

The snapshot is inserted in the **Snapshots** view before the previously selected snapshot.

Replace one 2D snapshot with another

1. Click the **Snapshots** view to activate Snapshots.
2. In the **Snapshots** view, select the snapshot to be replaced.
3. Manipulate the view until the Viewing window displays the view you want to save.
4. Do one of the following:

   - Right-click a snapshot and click **Replace**.
   - On the **Snapshots** toolbar, click **Replace**  .

   The selected snapshot disappears and it is replaced by the new snapshot.

Add a name to the 2D snapshot

1. Click the **Snapshots** view to activate Snapshots.
2. In the **Snapshots** view, right-click a snapshot, and then click **Name**.
3. In the **Snapshot Name** dialog box, in **Name**, type a new name for this snapshot.

   The snapshot name is displayed when you hold the cursor on the snapshot.

Rearrange 2D snapshots

1. In the **Snapshots** view, click a snapshot and drag it to another position.
2. To drag multiple snapshots, hold Ctrl as you select the snapshots, and then drag the group to another position.
3. To drag a series of snapshots, hold Shift as you select the first and last snapshots in the series, and then drag the series to another position.

Delete 2D snapshots

1. Click the **Snapshots** view to activate Snapshots.

2. Do one of the following:

   - In the **Snapshots** view, right-click the snapshot and choose **Delete**.
   - In the **Snapshots** view, select the snapshot you want to delete, and then on the **Snapshots** toolbar, click **Delete**  .

Display an existing 2D snapshot

1. Click the **Snapshots** view to activate Snapshots.
2. Display an existing snapshot in one of these ways:

   - In the **Snapshots** view, double-click the desired snapshot.
   - Right-click the snapshot and choose **Recapture Image**.

   The existing snapshot opens in the Viewing window.

5. Managing ECAD markups

Overview of working with ECAD markups

Markup tools, developed specifically for the PCB design community, help make collaboration and communication a reality. In this collaborative industry, you can use markup tools to minimize design and manufacturing miscalculations by providing comments and suggestions in the review process. Markup features include robust textual markup options, graphical markups such as lines and shapes, and important measurement markups, which include distance, radial, clearance, and length measurements.

ECAD markup profiles

Using markup profiles

Use markup profiles to create a set of common markup preferences. Markup profile preferences include specifying the unique name of the profile and configuring markup appearance preferences for this profile. For example, during a phase in the design project you determine that you want to collaborate with design team members, with management, and with a third party. You can create a markup profile for each group. This helps organize and structure collaboration and communication between these groups for the lifecycle of the project.

Create a new markup profile

1. On the **ECAD Markup** toolbar, click **Enable Markup**  .

2. On the **ECAD Markup** toolbar, click **Markup Preference Profiles**  .

3. In the **Markup Profiles** dialog box, click **Create**.

4. In the **Create Markup Profile** dialog box, do the following:

Use this option	To do this
Profile name	Enter the name of your markup profile.
Initialize from	Select an existing profile from the **Initialize from** list. Note: This action sets initial markup preferences. You can later edit these preferences.

5. Click **OK**.

   The new markup profile is added to **Available Profiles** in the **Markup Profiles** dialog box.

Manage markup profiles

When you manage markup profiles, you can create new markup profiles as well as view, edit, or delete existing profiles. Markup profiles maintain markup appearance preferences for elements such as line, fill, and text color.

1. On the **ECAD Markup** toolbar, click **Enable Markup**  .

2. On the **ECAD Markup** toolbar, click **Markup Preference Profiles**  .

3. In the **Markup Profiles** dialog box, do any of the following:

Use this option	To do this
Active	Select the active profile from **Available Profiles** list.
View	View the appearance preferences of the selected markup profile.
Edit	Edit the appearance preferences of the selected markup profile. Note: You cannot edit the default profile.
Delete	Delete an existing markup profile. Note: You cannot delete either the default profile or the active profile.
Create	Create a new profile name and set the markup appearance preferences for the new profile.

4. Click **OK**.

   The selected markup profile becomes the active markup preferences for future markups.

Changing the appearance of markups

Once you create a new markup profile, you can accept the default appearance options for specific markup tools or you can modify the appearance options for each markup tool. Access the **Preferences** dialog box by clicking **Edit** when you select the profile you want to change.

You can modify the following markup options:

Use this option	To do this
Line	Change line characteristics including style, color, thickness, and leader markers.
Edge	Change how you want the edges to appear by changing the style, size, and thickness of the edge. For example, you may want a red edge around an transparent oval.
Fill	Change the fill style, color, and select or clear transparency.
Font	Change the font name, style, and color.
Predefined text	Change several options for setting predefined text. You can select auto-translation, and the category of predefined text (electrical, manufacturing, and so forth). Plus, you can create and use your predefined text.
Measurement	Change the size of displayed measurement values (ranging from extra small to extra large), the color of the text and notation lines, and precision setting of values to the right of the decimal point (0 to 9).

Text markups for ECAD documents

Using text markups

Text markups provide vital information to other design team members. When you add new text markups or edit existing ones, you are communicating design direction that improves processing PCBs. Using predefined text and auto translated messages also improves how you collaborate with others around the world.

Add a text markup

1. On the **ECAD Markup** toolbar, click **Enable Markup**  .

> Note:
>
> The cursor changes to *NoSymbol* until you select a markup tool.

2. On the **ECAD Markup** toolbar, click **Create New Markup**.

3. Click in the Viewing window and type a markup topic name in the **Markup Topic** dialog box.

4. On the **ECAD Markup** toolbar, click **Text**.

5. Use the cursor to create a bounding box in the Viewing window, and enter your text message in the **Text Editor** dialog box.

6. (Optional) To resize the text bounding box, choose **Select Markup**, point to the desired markup, and move the bounding box handles to the desired position.

> Note:
>
> You can move the markup after it is placed. First, highlight the markup using the **Select Markup** option, and then drag the bounding box to a new location.

Add predefined text markups

You may find that you are typing the same markup text repeatedly or that you communicate standard messages during the project. If so, you can automate this process by selecting markup text from an existing text file.

1. On the **ECAD Markup** toolbar, click the following:

   a. **Enable Markup**

   b. **Create New Markup**

   c. **Rubber Stamp**

2. From the **Rubber Stamp** dialog box, select **Note file**.

   You can **Browse** to an existing predefined note file or you can choose one from the **Index** or **Title** lists.

Example rubber stamp markup file

Automatically translate text messages

Several standard text messages have been predefined and translated in languages supported by the software. When you open a document containing a predefined text message, the text of the message is automatically translated to the default language of the computer that opened the file.

1. On the **ECAD Markup** toolbar, click **Enable Markup**  .

2. On the **ECAD Markup** toolbar, click **Markup Preference Profiles**  .

3. In the **Markup Profiles** dialog box, click **Edit**.

4. In the **Preferences** dialog box, click the **Predefined Text** tab and select **Auto-translate text**.

5. Choose a general topic from the **Category** list, and then select a text message from the list of messages.

For example, automatically translated categories and text messages include the following:

Category	Text message
Electrical	Traces are too narrow for the power they will carry Traces are too close together Traces exceed allowed distance for running together Traces too close to holes/vias/pads High voltage/low voltage isolation rule violations Needs more bypass caps Test point to Test point spacing Test point clearance to Component Clearance to Circuit edge Inaccessible object
Manufacturing	Trace turns too sharp Missing tear-drops Missing thermals Bad or wrong pad stack
Fabrication	Object blocking solder wave Problems with solder mask Markings/silk screen not to standards

Category	Text message
	Split power plane problems
	Questions regarding possible etching undercut on a narrow trace
Mechanical	Holes too close to edge
	Wrong size holes/drills for specified hardware
	Missing features
	Physical part interference
	Panel width and length
	Component Height

Note:

To add this predefined text markup message, add a markup and then on the **ECAD Markup** toolbar, click **Rubber Stamp**  .

Add to an existing markup

In a collaborative environment, you may need to provide updated revisions to an existing text or graphic markup. You can add markup revisions to an existing markup without losing previous markup information.

1. On the **ECAD Markup** toolbar, click **Enable Markup**  .

2. On the **ECAD Markup** toolbar, click **Select Markup**  .

3. In the Viewing window, highlight the target markup.

4. Right–click the markup and select **Add Markup Revision**.

5. Create your markup using the markup tools.

   The target markup and all revisions to it appear as one markup in the **Markup Log**. These markups appear in the Viewing window as separate markup entries, but you can view them together as an icon or as text.

Display predefined text automatically

If you find that you are accessing predefined text options regularly, you can set preferences to always open the **Predefined Text** dialog box.

1. On the **ECAD Markup** toolbar, click **Enable Markup**  .

2. On the **ECAD Markup** toolbar, click **Markup Preference Profiles**  .

3. In the **Markup Profiles** dialog box, select an active profile from **Available Profiles** and click **Edit**.

4. In the **Preference** dialog box, click the **Predefined Text** tab, and select **Display dialog on tool activation**.

Add line or graphic markups

While a text markup is the most common markup, you will find that line and graphic markups can enhance and clarify your design direction.

1. On the **ECAD Markup** toolbar, click **Enable Markup**  .

   > Note:
   >
   > The cursor changes to *NoSymbol*  until you select a markup tool.

2. From the **ECAD Markup** toolbar, click **Create New Markup**  .

3. Click in the Viewing window and enter a markup topic name in the **Markup Topic** dialog box.

4. Select one or more of the following line or graphic markups:

	Use this option	To do this
	**Freehand Line**	Draw a freehand line.
	**Leader Line**	Draw a leader line by doing the following: a. Click in the Viewing window to place one endpoint. b. Click to place each vertex. c. Double-click to place the last endpoint. Tip: To make the line exactly horizontal or vertical, press the Shift key as you drag or click the mouse.

	Use this option	To do this
	**Ellipse**	Draw an ellipse by doing the following: a. Click in the Viewing window to place the starting point of the graphic. b. Drag the cursor to size the graphic as desired. Tip: To draw a circle, hold the Shift key as you drag the cursor.
	**Polygon**	Draw a polygon by doing the following: a. Click in the Viewing window to place one endpoint. b. Click to place each vertex. c. Double-click to place the last endpoint.
	**Rectangle**	Draw an rectangle by doing the following: a. Click in the Viewing window to place one corner. b. Drag the cursor to place the opposite corner. Tip: To draw a square, hold the Shift key as you drag the cursor.

Measurement markups for ECAD documents

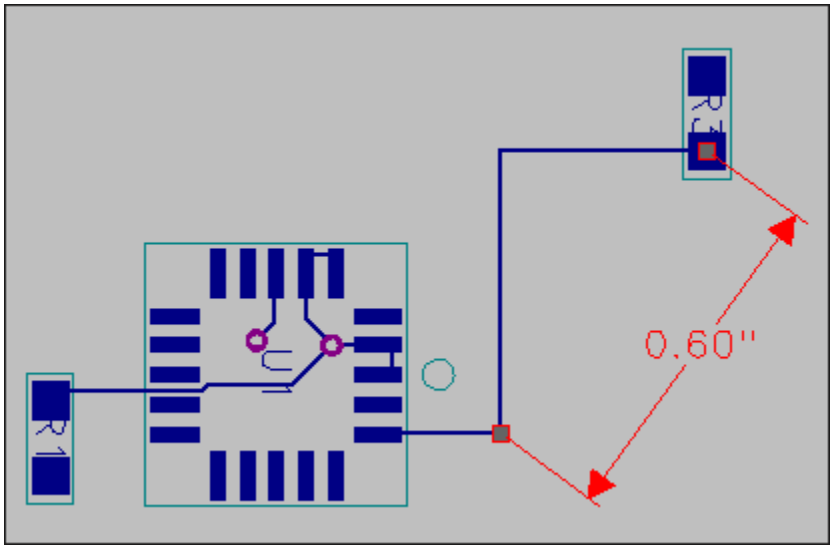
Using measurement markups

There are several instances where you need measurement data from the PCB. Measurement tools, accessed from the **ECAD Markup** toolbar, provide you with options for distance, radial, clearance, and length data. You can save these measurement markups to your database or file system and they become a part of the PCB data session. Measurement tools are even available to you when the PCB has been checked out by someone else; essentially the PCB is in a read-only state. You can take measurements when you are viewing a read-only PCB, but you can not save (persist) them.

Measurements are generated when you set beginning and ending markers or points on the PCB.

Tip:

You can use keyboard keys to quickly modify the markers or points. The Shift key and clicking creates a *vertical* measurement. The Ctrl key and clicking creates a *horizontal* measurement. Use the Backspace key to remove a marker and return to the initial pointer.



Measuring distance in ecad files

Overview of measuring distances in ECAD files

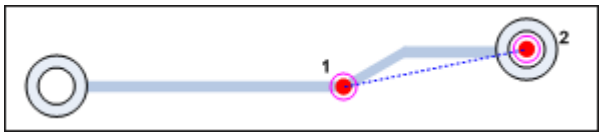
Distance measurements are taken between two points, between a point and a line, or between two parallel lines. The points and lines can be snap-to portions of a PCB entity. A snap-to element is the outer edge of a trace width or a vertex or midpoint of a pad edge. You can also specify a point in free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

When you move your mouse around the PCB, feedback markers are displayed indicating that you are approaching snap-to points or lines.

Point to Point measurements

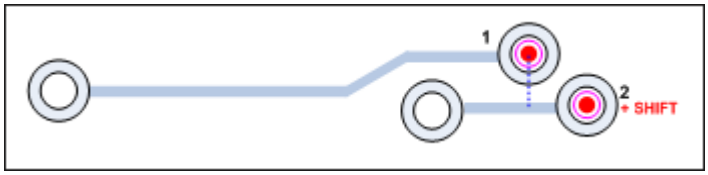
Euclidean Distance

Euclidean Distance is the default, minimum distance between any two points.



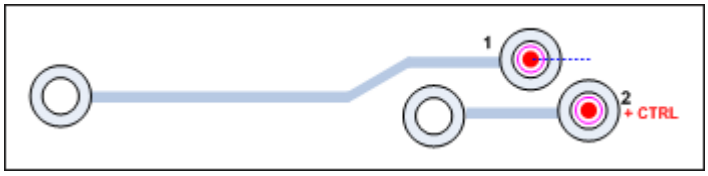
Vertical Distance

Vertical Distance is the between two points when you use the Shift key and click for the second point.



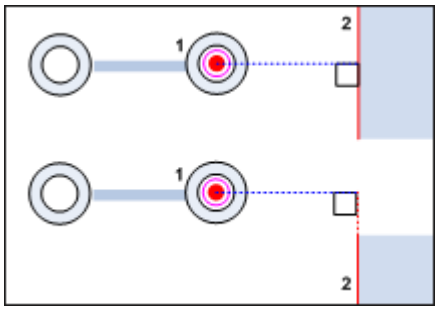
Horizontal Distance

Horizontal Distance is the between two points when you use the Ctrl key and click for the second point.



Point-to-Line

Use Point-to-line measurement to determine the distance between the line and a point formed by a perpendicular line between the point and the line segment. You can extend the point beyond the actual line segment (a point that appears to be in free space).



Line-to-Line

Use line-to-line measurement to determine the distance between two parallel lines.

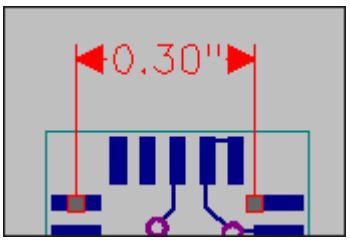
> Note:
>
> For line-to-line measurements, non-parallel lines are unsupported pick points.

Measure distance between points and lines

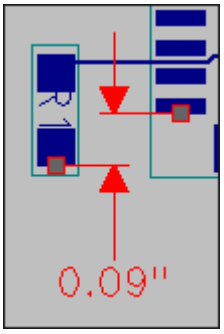
1. On the **ECAD Markup** toolbar, click **Distance Measurement** .

2. Click the beginning measurement point.

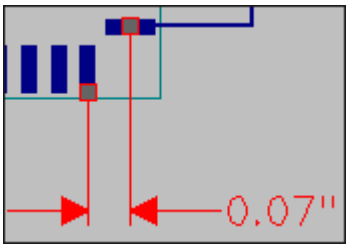
3. Click the ending measurement point.



4. (Optional for vertical measurements) Hold the Shift key and click at the ending measurement point.



5. (Optional for horizontal measurements) Hold the Ctrl key and click at the ending measurement point.



Measuring radials in ECAD files

Overview of measuring radials

Use radial measurement when you want the distance between a ring and circle PCB 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

measurements require just one click on or near a highlighted snap-to point or shape. Three 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).

For circle entities, only one measurement is supported.

Radial measurements

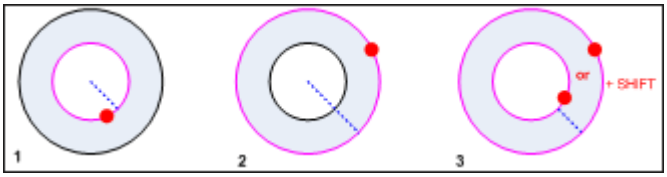
Circle-to-Circle ring measurement	Circle-to-Circle ring measurements occur when your cursor nears the hole edge or ring edge. When you click while either of these edges is highlighted your result is a radial circle-to-circle measurement.
Polygon-to-Circle ring measurement	Polygon-to-Circle ring measurements occur when your cursor nears the hole edge or a snap point on the ring edge. When you click under these instances your result is a radial polygon-to-circle measurement.
Circle-to-Polygon ring measurement	Circle-to-Polygon ring measurements occur when your cursor nears a snap point of an inner shape and its outer edge. When you click under these instances your result is a radial circle-to-polygon measurement.
Polygon-to-Polygon ring measurement	Polygon-Polygon ring measurements occur when your cursor nears a snap point on either the inner or outer shape. When you click under these instances your result is a radial polygon-to-polygon measurement.

Measure circle-to-circle radials

1. On the **ECAD Markup** toolbar, click **Radial Measurement**  .
2. Click the inside edge to measure the radius of the inner circle.
3. Click the outside edge to measure the radius of the entire circle.
4. Click the inside or outside edge while clicking the Shift key to measure the width of the ring.

Example:

Circle-to-Circle



Measure polygon-to-circle radials

1. On the **ECAD Markup** toolbar, click **Radial Measurement** .

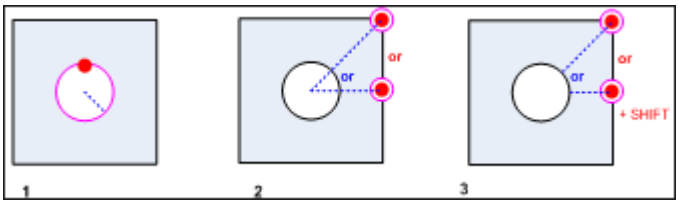
2. Click the hole edge to measure the radius of the hole.

3. Click a snap point on the shape edge (vertex or midpoint) to measure the distance between the center of ring and the selected point on the outer shape.

4. Click a snap point on the shape edge while clicking the Shift key to measure the distance between the outside edge of the ring to the selected snap point on the shape.

Example:

Polygon-to-Circle



Measure circle-to-polygon radials

1. On the **ECAD Markup** toolbar, click **Radial Measurement** .

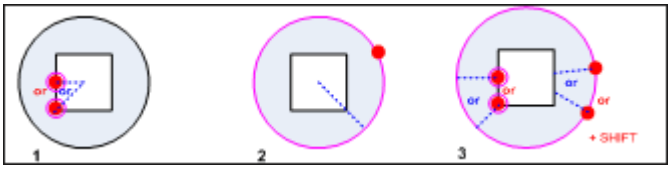
2. Click a snap point on the hole edge to measure the distance between ring center and selected hole point.

3. Click the ring edge to measure the radius of the ring.

4. Click either a snap point on the hole or on the ring edge while clicking the Shift key to measure the radial distance between the selected hole point and ring edge.

Example:

Circle-to-Polygon



Measure polygon-to-polygon radials

1. On the **ECAD Markup** toolbar, click **Radial Measurement** .

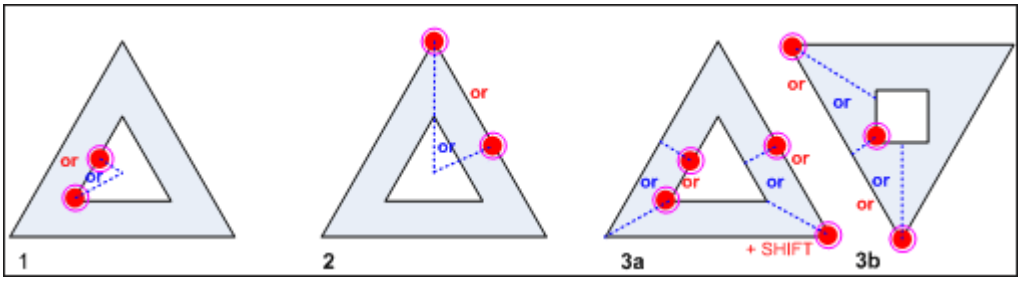
2. Click a snap point on the hole edge to measure the distance between the ring center and the selected hole point.

3. Click a snap point on the shape edge to measure the distance between the ring center and the selected ring edge point.

4. Click a snap point on either the shape edge or the hole edge while clicking the Shift key to measure the radial distance between the selected hole or ring point and the intersecting point.

Example:

Polygon-to-Polygon



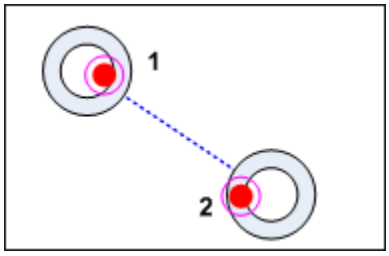
Measuring clearance distances

Overview of measuring clearance distances

Use clearance measurement to determine the amount of minimum distance, or clearance, between any two PCB objects. When you move the cursor around the PCB, underlying objects are highlighted when you approach them.

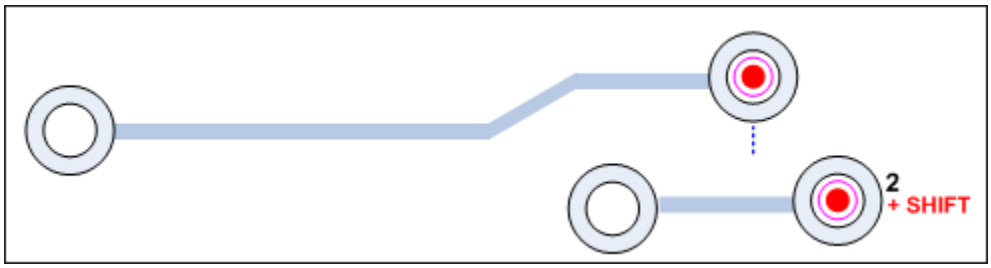
Clearance

Use clearance to measure the minimum Euclidean distance between the two PCB objects.



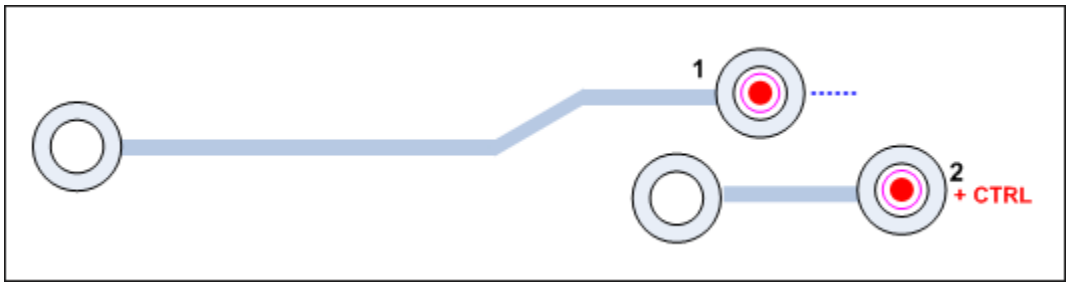
Vertical clearance

Use vertical clearance to measure the distance between two objects by using the Shift key while selecting the second object.



Horizontal clearance

Use horizontal clearance to measure the distance between two objects by using the Ctrl key while selecting the second object.



Measure clearance distance

1. On the **ECAD Markup** toolbar, click **Clearance Measurement** .
2. Click the beginning clearance measurement point.
3. Click the ending clearance measurement point.
4. (Optional for vertical clearance measurements) Hold the Shift key and click at the ending clearance measurement point.

5. (Optional for horizontal clearance measurements) Hold the Ctrl key and click at the ending clearance measurement point.

Measuring traces and vias using Manhattan and Routed measurements

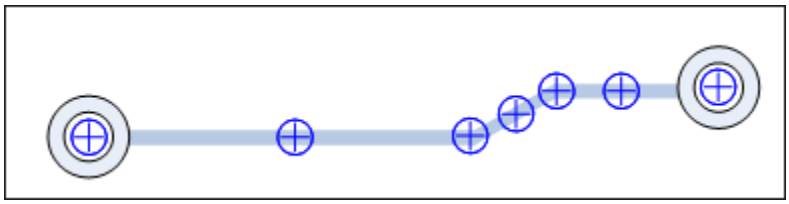
Overview of Manhattan and Routed measurements

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

Snap-to points

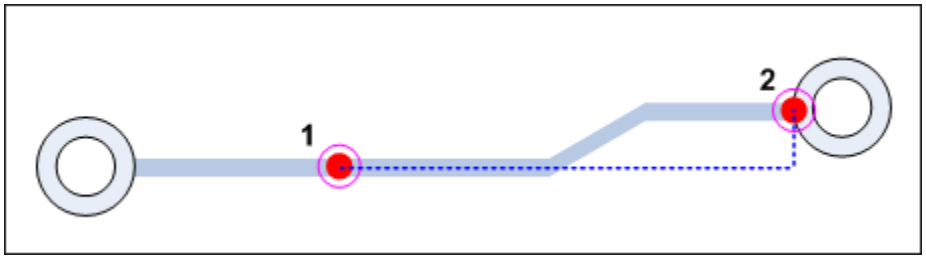
Choose a length measurement (Manhattan or Routed) and then as you move your cursor over a length of trace, feedback indicators highlight snap-to pick points. Snap-to points are points of measurement interest provided by the software. You can measure from one point to the next.

For length measurement, you must click a highlighted snap-to point.



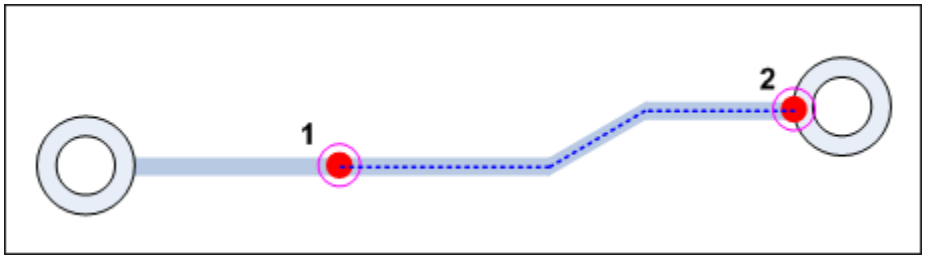
Manhattan length

Use Manhattan length measurement to determine the distance (the sum of the X- and Y-coordinates) between two selected points.



Routed length

Use Routed length measurement to determine the distance (the sum of each segment length along the trace) between two selected points.



Measure traces and vias using Manhattan or Routed measurements

1. On the **ECAD Markup** toolbar, click one of the following:

   - **Manhattan Length**
   - **Routed Length**

2. Move your cursor over the target trace or via and select the beginning snap-to (highlighted) measurement point, and click.

3. Continue navigating the PCB until you find a feedback highlighted marker for your second measurement snap-to point, and click.

Save markups

When you update a markup, the update is saved to the current markup dataset. You can also use **Save As** to save the markup to a different or new dataset.

Do one of the following:

- To save markups, on the **ECAD Markup** toolbar, click **Save**.
- To save markups to a different dataset, on the **ECAD Markup** toolbar, click **Save As**.

> Note:
>
> - **Save** and **Save As** are unavailable if the file is read-only.
> - Saved markups are loaded automatically the next time you open the document, though the markup may not be displayed automatically. Markups are automatically displayed when the file is opened if **Display markups on load** is selected.
> - You can also save markups as a work session using session files. When you open a session file your work is restored exactly as you saved it. You can also save your work session as file archives

used to package your work session using session package. You can then move the zipped work session from your local file session to any location.

> Note:
>
> Saving Design for Manufacturing data as session files or in session packages is not supported.
>
> You can save Design for Manufacturing outside the session file structure and open this saved data.

Display the markup as an icon or text

Some PCB and schematic documents can be visually overwhelming because of the number of objects and details contained on them. When you include markup information, you will add more objects to the view. To help organize these details you can choose to display markups as an icon and then move the icon to a desirable location. You can also represent markups as fully expanded text and graphics.

1. On the **ECAD Markup** toolbar, click **Enable Markup**  .

2. On the **ECAD Markup** toolbar, click **Select Markup**  .

3. In the Viewing window, use **Select Markup** to select the desired markup.

4. Right-click the highlighted markup, and select **Iconize** or **Expand**.

Unload markups

There may be times when you need to unload the currently active (and saved) markup from the viewer. You may also need to unload all unsaved markups from the viewer.

Right-click in the **Assembly** view and select **Unload Markups**.

The currently saved markup is unloaded from the view. If you have unsaved markups in the view, you are warned that these markups will be deleted.

Display markup activity

Use the **Markup Log** to display summary markup information. This information includes the markup topic, the revision, and the author. You can also create and open markup log reports.

1. On the **ECAD Markup** toolbar, click **Enable Markup**  .

2. On the **ECAD Markup** toolbar, click **Markup Log**  .

   The **Markup Log** dialog box appears.

3. (Optional) If **Delete** is active, you can delete markups associated with the markup log topic.

4. (Optional) Create **Markup Log** reports.

Note:

- When you highlight a markup topic, the markups associated with it are highlighted in the Viewing window.
- After you select **Delete**, you are asked to confirm your decision.
- All subordinate revisions must first be deleted before you can delete the first revision.
- Click a column label to sort data in the **Markup Log** dialog box.
- The metadata contained in the markup log is read–only. You cannot write to or edit this information.

Create a markup log report

1. On the **ECAD Markup** toolbar, click **Enable Markup**  .

2. On the **ECAD Markup** toolbar, click **Markup Log**  .

   The **Markup Log** dialog box appears.

3. In the **Markup Log** dialog box, click **Report**.

4. In the **Report Selection** dialog box do one of the following:

   a. Select the report creation method by choosing **Run report wizard** or **Use a previously saved report**, and then click **Next**.

   b. In the **Report Definition Wizard** dialog box, choose or edit the report title and then click **Next**.

   c. From the list, select or clear the data you want in the report and then click **Next**.

   d. (Optional) Select or edit filters that you want applied to the report. Click **Next**.

e. In the **Enter Name** dialog box, enter a name for this report and then click **Next**.

> Note:
>
> You can also click **Skip** is you want to use the default report name.

f. In the **Create Report Output** dialog box, select the report output format (HTML, CSV, TXT, XML), or a custom output, and specify the location for the generated file. Then, click **Finish**.

Modify markup properties

You can change markup properties (such as fill color, edge, and so forth).

1. On the **ECAD Markup** toolbar, click **Enable Markup**  .
2. On the **ECAD Markup** toolbar, click **Select Markup**  .
3. Click the markup you want to modify.

   A bounding box appears around the markup indicating the markup has been selected.
4. Right-click the markup, choose **Properties**, and modify the properties.

Edit markup appearance preferences

Use **Markup Preference Profiles** to modify markup properties such as line color, font type, and fill color. These changes will take effect on future markups.

1. On the **ECAD Markup** toolbar, click **Enable Markup**  .
2. On the **ECAD Markup** toolbar, click **Markup Preference Profiles**  .
3. From the **Markup Profiles** dialog box, click **Edit**.
4. Change the markup appearance using the following options:

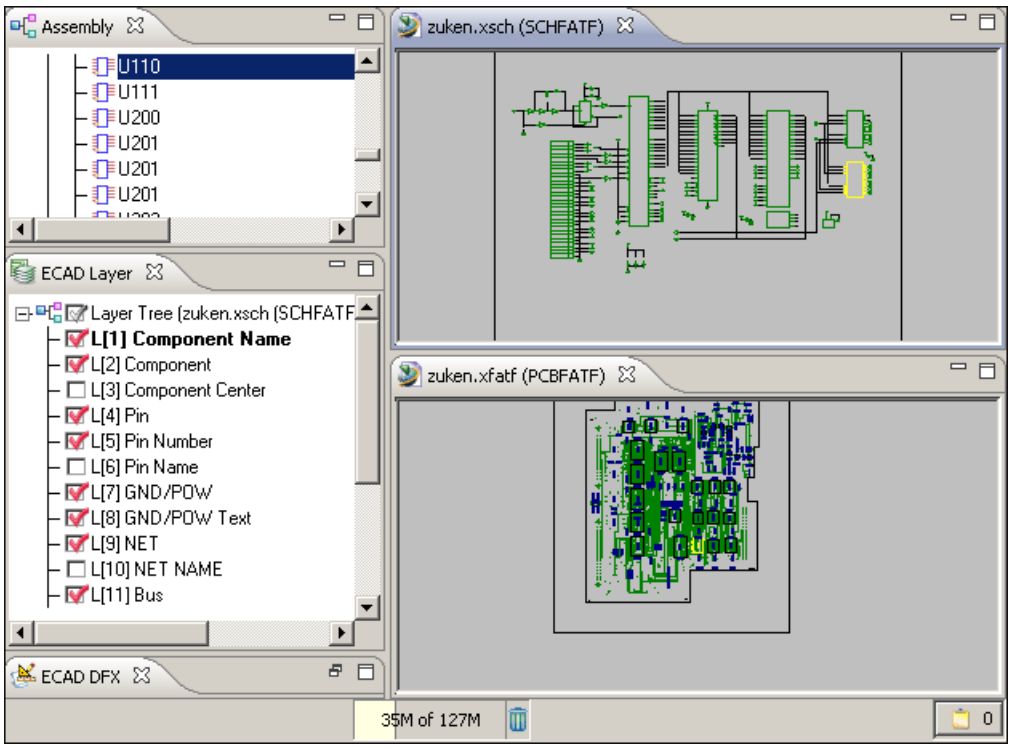
On this page	Use this option	To do this
Line	**Style**	Select a line type from the **Style** list.
	Color	Select a line color from the **Color** list.
	Thickness	Enter a line thickness.

On this page	Use this option	To do this
	Ends	Select start or stop (or both) leader ends. Enter the leader end width and length.
Edge	**Edge style**	Select an edge type from the **Edge style** list.
	Edge color	Select an edge color from the **Edge color** list.
	Thickness	Enter an edge thickness.
Fill	**Fill style**	Select the fill type from **Fill style** list.
	Fill color	Select a fill color from the **Fill color** list.
	(Under) **Transparency**, select **Enable**	Specify that your markup fill color will not block data on the board.
Font	**Font name**	Select a font name from the available font list.
	Font style	Select a font style from the available options.
	Color	Select the text color from the **Color** list.
Predefined Text	**Auto_translatable text message**	Choose from a set of predefined words and phrases that have been translated for numerous locales.
	Note file	Choose a text stamp from preexisting text files. You can **Browse** through the existing notes and then choose a specific note file by selecting the **Index** or **Title** for the note from the list.
	Display dialog on tool activation	Specify the **Predefined Text** dialog box will be displayed when you create a text markup.

6. Using cross probing

Overview of using cross probing

There may be times during the design process where you need to verify similarities of and differences between the schematic and the PCB, between one PCB and another PCB, or between one schematic and another schematic. Also, you may need to determine how the board design was implemented. In these situations, you can use cross probing to find common parts that are included in the objects.



Note:

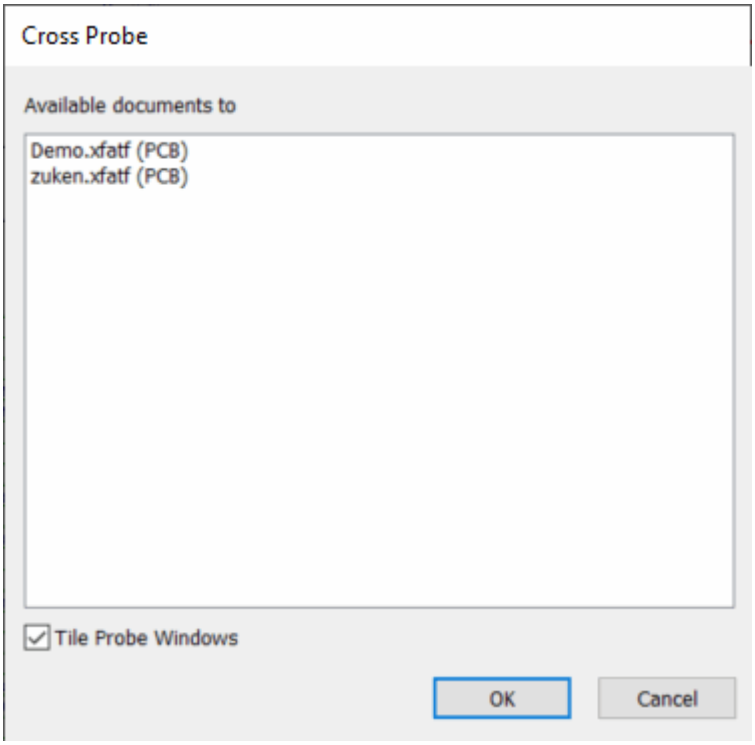
The frames in this graphic have been arranged to display the benefit of cross probing. The frames in your default application may be different. You can drag and drop the views and frames in the user interface as desired.

Cross probe ECAD documents

1. Open the target ECAD documents (PCB and schematic, PCB and PCB, or schematic and schematic) that you want to cross probe.

2. From the **ECAD Base** toolbar, click **Cross Probe** .

3. (Optional – if you have more than two documents open in the Viewing window) In the **Cross Probe** dialog box select the target ECAD documents (PCBs, schematics, or both).



> Note:
>
> The **Assembly** view that is enabled corresponds to the document that is highlighted (either a PCB or a schematic).

4. Select a component from the **Assembly** view, or from the Viewing window.

   Matching components will be highlighted in both ECAD documents.

> Tip:
>
> Most PCB and schematic displays are rich with components. Since it can be difficult to notice all selected objects, you can change the highlight color of the PCB, or the schematic, or both.
>
> 1. Select the PCB to give it focus.
> 2. Choose **View→Preferences**, and in the **View** tab select a color from **Highlight Color**.
> 3. Repeat these steps for other ECAD documents.

7. Comparing ECAD documents

Overview of comparing ECAD documents

You can use this feature to open and examine differences in schematic and PCB documents. You can open any combination of the following ECAD documents:

- Two different PCB revisions.
- Two different schematic revisions.
- A PCB design and its associated schematic.

As examples, you can use compare feature to understand and visually inspect the differences in component strength, placement, or currency between one revision of the schematic design to another. Or, you can use the feature to validate ECAD objects contained in the physical design against objects contained in the accompanying schematic design.

Depending on which ECAD document combination you select, specific preferences are available to help narrow your results. For example, if you are comparing two PCBs, you can choose to compare both sides of the board or you can narrow the results by selecting just one side to compare.

The two documents that you want to compare are visually displayed in the Viewing window. The compare results are also displayed in the **Compare** view.

You can also modify how compare results are displayed in the Viewing window by modifying compare preferences. For example, you can change highlight colors, how the two documents are displayed in the Viewing window, and specifying which ECAD objects and their variables that you want compared.

You can also generate comparison reports, choosing from a variety of output options including HTML and TXT.

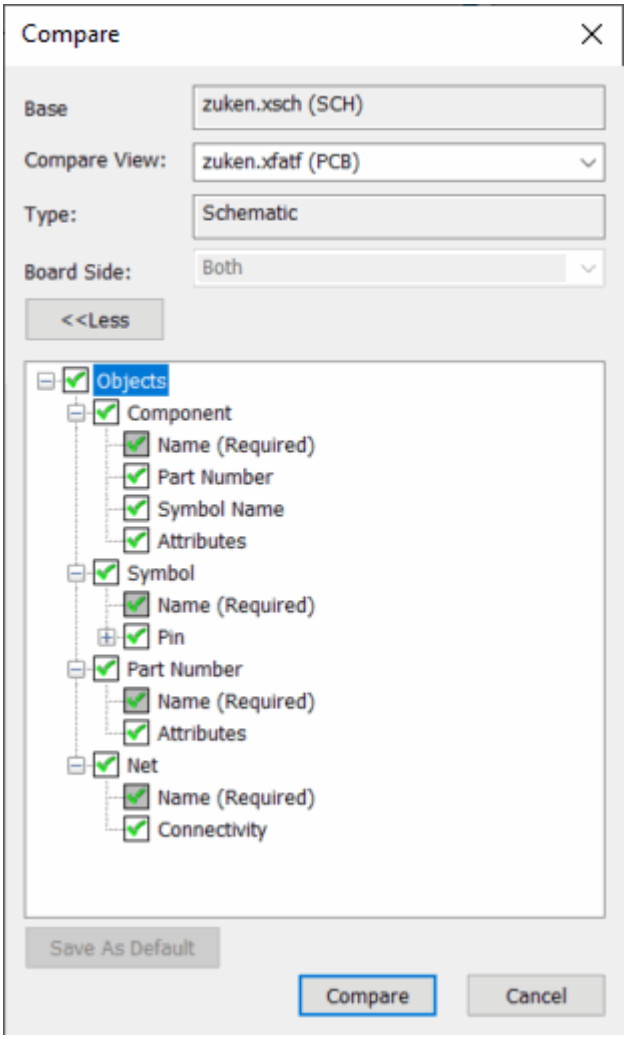
Compare ECAD documents

1. Open two ECAD documents to compare.
2. On the **ECAD Base** toolbar, click **Visual Compare**  .
3. (Optional) If more than two ECAD documents are opened, in **Compare View**, select the two documents that you want to compare.
4. (Optional) If both documents are PCB boards, in **Board Side**, select **Both**, **Top**, or **Bottom** to specify how you want to compare the board.

5. If you are satisfied with the default compare settings, click **Compare**.

   The ECAD documents appear in the Viewing window and the compare results appear in the **Compare** view.

6. (Optional) Click **More** and select options and values for detailed processing, and then click **Compare**.



Note:

- The values and options you select are used for this processing session. If you want to save these options for future processing, click **Save As Default**.
- The ECAD documents are displayed in the Viewing window in accordance with the compare **configuration preferences** you set. By default, the compare results are displayed with red highlight and the documents are tiled horizontally.

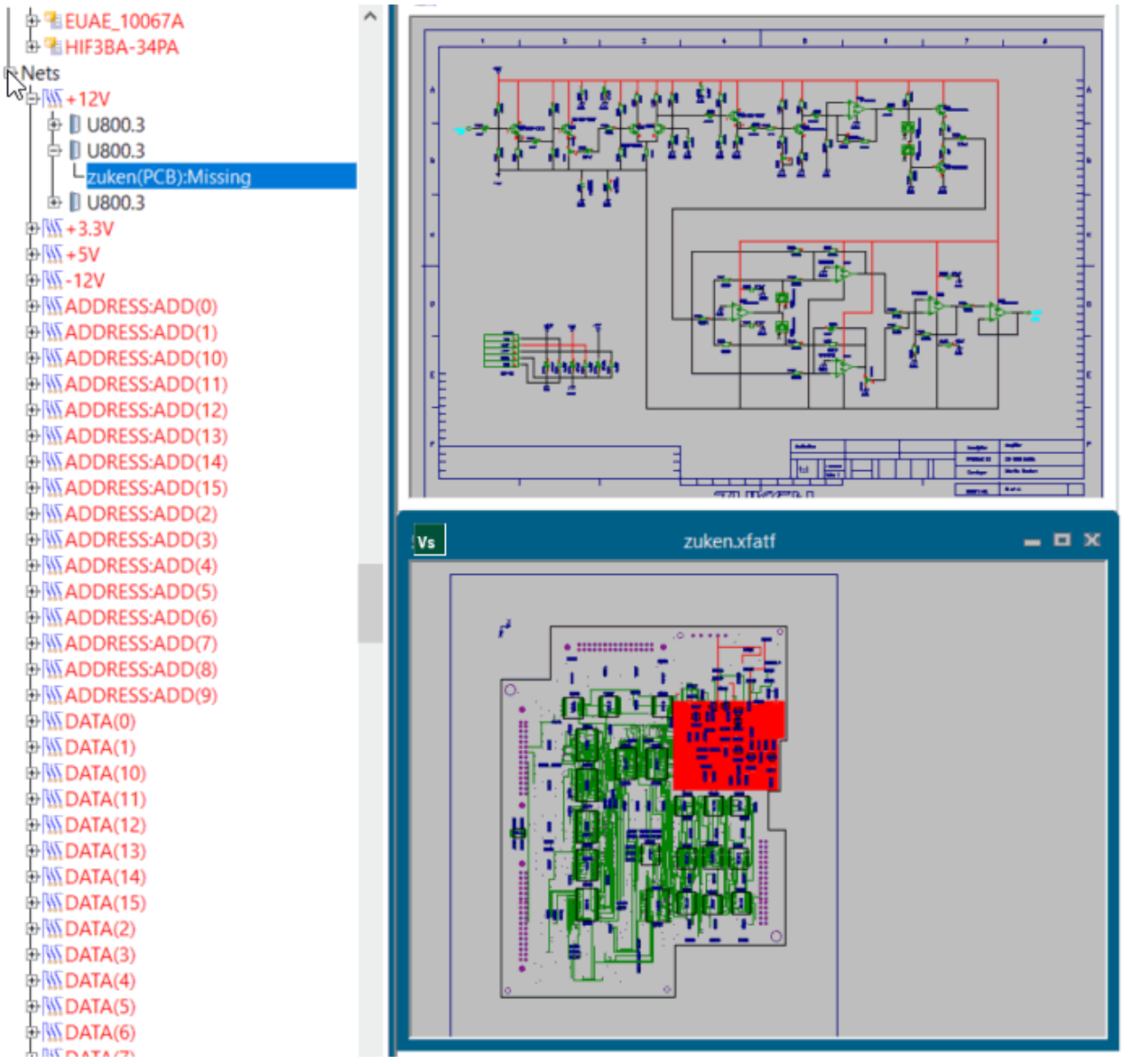
Display compare results

General compare results are displayed on the **Compare** view, and in the Viewing window.

To display only the differences between ECAD documents, In the **Compare** view, select **View Differences** from the shortcut menu.

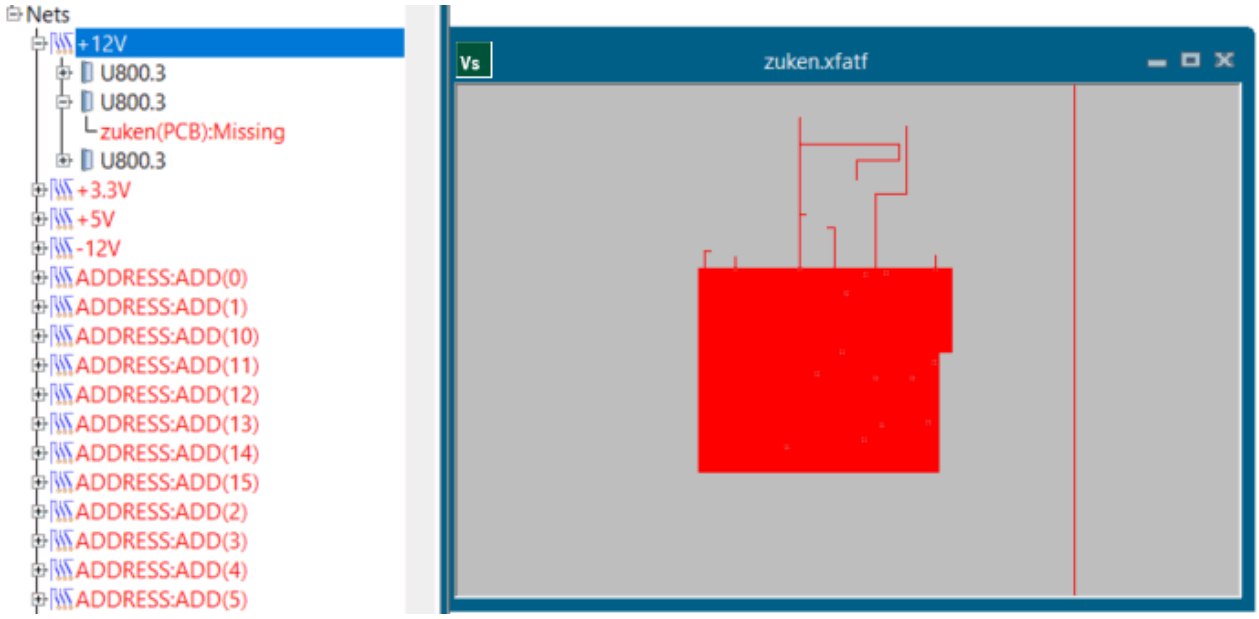
Example:

This example shows general differences.



Example:

This example shows just the differences.



Note:

- Notice that all layers are turned off in this view. To return to the previous viewing mode, select **Reset Visibility State** from the shortcut menu.
- Highlight any one entity in the **Compare** view, and that entity is displayed in the Viewing window.

Change compare preferences

1. Open an ECAD document, and then in the Viewing window, right-click and select **Preferences**.
2. In the **Preferences** dialog box, select the **Compare** tab.
3. From the **Compare Color** list, choose a highlight color.

   Note:

   Red is the default color.

4. From the **Compare Type** list, choose **Schematic** or **PCB**.

The available list of objects and their compare values depend on the type of ECAD document you select.

5. In the **Objects** section, choose the ECAD objects and their attributes that you want to include in the compare process.

6. (Optional) Specify how you want to display your ECAD compare documents by selecting or clearing the **Tile Compare Windows** check box.

   Selecting the check box tiles the documents and clearing it removes tiling.

7. (If you tile documents) Select the **Horizontal** or **Vertical** check boxes.

Generate compare reports

1. On the **ECAD Base** toolbar, click **Create Report**  .

   > Note:
   >
   > You must have previously generated compare analysis for ECAD documents.

2. In the **Report Template** section, select **ECAD Compare Report**.

3. Click **Run** to create the compare report.

4. (Optional) Choose an output format from the **Standard report output format** section.

   You can choose from HTML, CVS, TXT, or XML.

5. (Optional) Select **Custom report output format** and then use **Browse** to navigate to an appropriate stylesheet location.

6. (Optional) In the **Specify a location for the report output file** section, click **Browse** to specify the location to save the output file.

7. (Optional) Select or clear the **Automatically open report output file** check box.

8. Click **Finish**.

9. Optionally, you can also

   - Click **Customize** to open the **Report Definition Wizard** and customize the report template.
   - Click **Preview** to open a preview view of the generated report.

8. Working with Design for Manufacturing

Overview of working with Design for Manufacturing

Design for Manufacturing consists of performing two critical examinations during the PCB design lifecycle: Design for Assembly (DFA) and Design for Test (DFT).

Design for Assembly

Use Design for Assembly to manage an analysis of your PCB designs against a default or customized set of manufacturing rules.

Analyzing results against these rules, DFA highlights problem issues that relate to panel dimensions and clearances, fiducial quantity and clearance, circuit edge clearance, component-to-component clearance and height of placed components.

You can establish custom rules that apply to different manufacturing facilities. Each facility may support a different set of rules and assembly processes. Each team can define the severity of problem issues. For example, the number of fiducials used for a given circuit length may warrant a warning for one facility process, but result in a violation for another. A warning notification typically alerts the PCB designer to less than optimal situations that require communications to the manufacturing engineer prior to release. Violation level notifications usually require extra or manual processing.

Design for Test

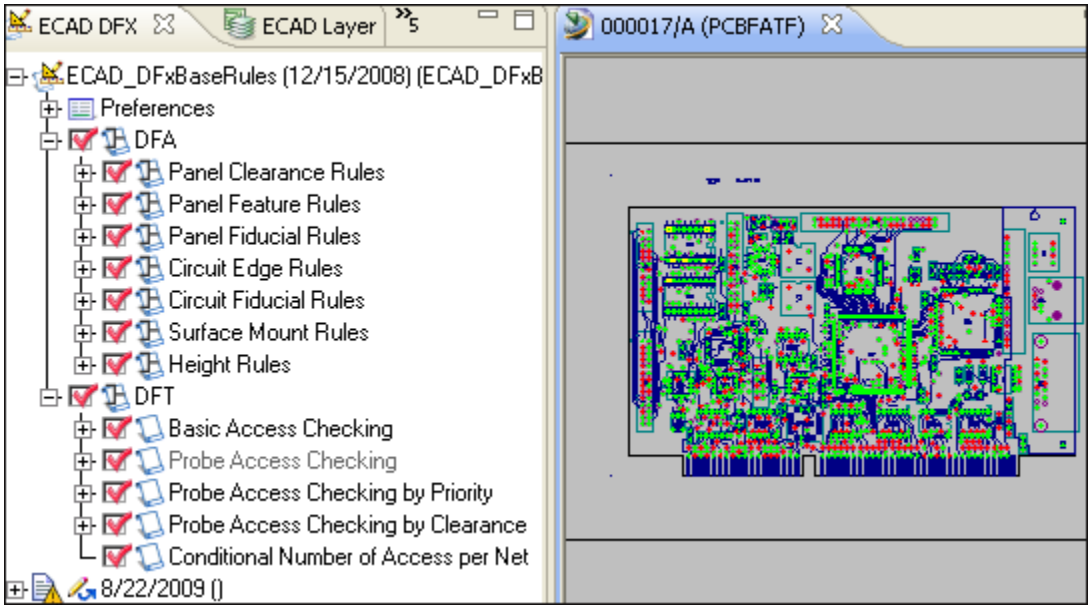
Use Design for Test to examine the integrity of various access points on the PCB. You can select various DFT rules to use in testing, but the master rule, Basic Access Checking, is the minimum rule that can be processed. This rule provides a complete analysis of the PCB to inventory all possible predefined access points. Access points include vias, pins and pads, and test points. DFT testing results provide data about the access clearance between various ECAD objects. During testing of the physical PCB, test pins touch and generate specific electrical current to determine if the elements are designed and functioning properly.

DFT testing provides data to review and repair critical access distance before the PCB goes to manufacturing.

Processing the analysis

Processing either DFA or DFT analysis begins with using options associated with the **DFx** toolbar. When you run DFA and DFT, both their rules and the results are displayed on the **ECAD DFX** view. The results are organized and shown as warnings or as violations. You can view the exceptions in the **Results** section in the **ECAD DFX** view. When you highlight an object on the DFx page, the object is also highlighted and centered in the Viewing window.

Providing you want to maintain your results, make sure to save them before leaving the Design for Manufacturing session. You can save results to a new dataset (ECADDFx) in the database. The **Rule** file is saved to *\Documents and Settings\<User>\Application Data\Teamcenter\DFx*.



Note:

We do not recommend that you save custom rules to the default rules file. Your organization may need to update the default rules file and then push it to all local machines. This process overwrites your custom rules file if you save it as the default rules file.

Review the **DFx graphics** associated with the views to understanding of the purpose of the displayed objects.

Run Design for Manufacturing analysis

1. Open an ECAD document.
2. On the **ECAD DFx** toolbar, click **Open Rules File** .
3. In the **Open Rules** dialog box, select a rules file and click **Open**.
4. On the **ECAD DFx** toolbar, click **Run DFx Rules** .
5. (Optional) View the **results** of the DFx analysis by navigating to the **ECAD DFX** view.
6. (Optional) To open previously saved Design for Manufacturing results, on the **ECAD DFx** toolbar, click **Open Results File** .

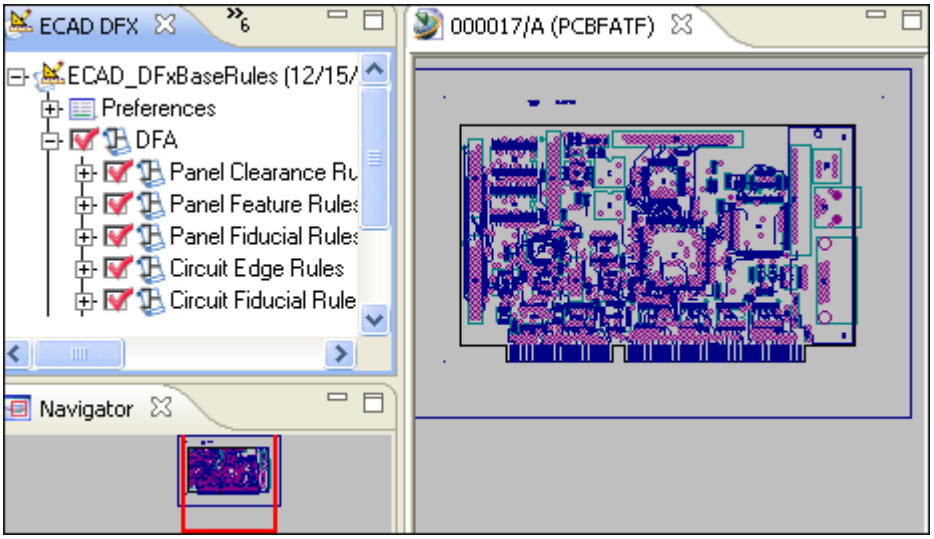
Results are saved to any location on your local file system. Result files include a **.xres** file extension.

Note:

- You can use the default rules to effectively determine the values and thresholds of your PCB. You may also find it useful to create customized rules for design analysis.
- When you change a preference in the rules file, you should rerun the test. For example, if you change the highlight color for violations, the change is automatically displayed in the Viewing window and in the **ECAD DFX** view. Still, rerunning the test is recommended.
- When you change rule files, all test data is automatically removed from the Viewing window and in the **ECAD DFX** view. Rerun the test to display the new visual results.

View Design for Manufacturing test results

1. Navigate to the **ECAD DFX** view.



2. In the **ECAD DFx** view, do any of the following:

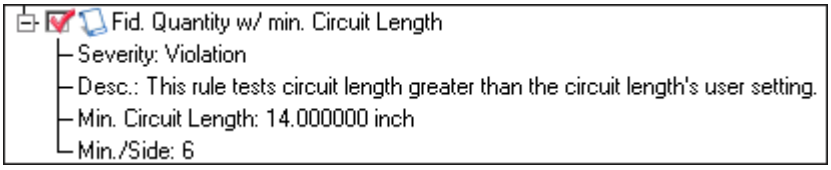
   a. Select or clear specific rules of concern.

   The displayed results showing warnings or violations, if any, are also selected or cleared.

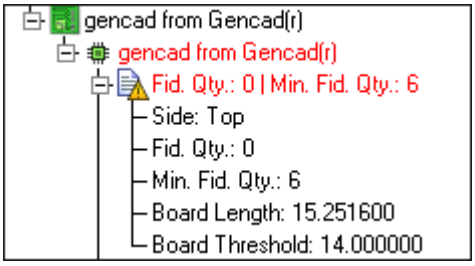
   Tip:

   To help organize the data, you can select or clear rules and collapse or expand the rules and results.

b. To review the severity classification, the description of the test, and the expected default or user determined value, expand the rule by clicking the plus sign to the left of the rule.

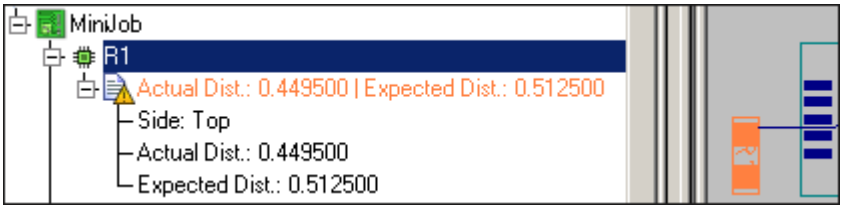


c. To review the test results, including the actual measured value against the predetermined threshold value, expand the result by clicking the plus sign to the left of the result.



> Note:
>
> The severity in this instance is classified as a violation, the PCB element text is red (the violation color of this system), and the measured value is 15.251600 against a threshold value of 14.000000.

d. Click a tested object in the **ECAD DFX** view and the object is highlighted and centered in the Viewing window.



> Note:
>
> The severity in this example is classified as a warning. The text and object colors reflect warning colors for this system, and actual and expected values are also displayed.

> Tip:
>
> For complex PCBs, the ECAD object may not be readily visible in the Viewing window. Since the object is centered in the Viewing window, you can use the cursor to zoom into the design until the object is visible.

You can also view attributes of the object by clicking the **Browse**  . Move the cursor around the Viewing window and ECAD object names are displayed. Click an object and review the attributes in the **Attributes** dialog box.

Save Design for Manufacturing rules and results

1. Run the DFx test.

2. Do one of the following:

   - On the **ECAD DFx** toolbar, click **Save**  .

     For Lifecycle Viewer, if you are working with existing DFx files, the DFx files are saved using the same file name. Otherwise, the **Save As** dialog box appears.

   - On the **ECAD DFx** toolbar, click **Save As**  .

     In the **Item Revision** dialog box, type the name of the dataset.

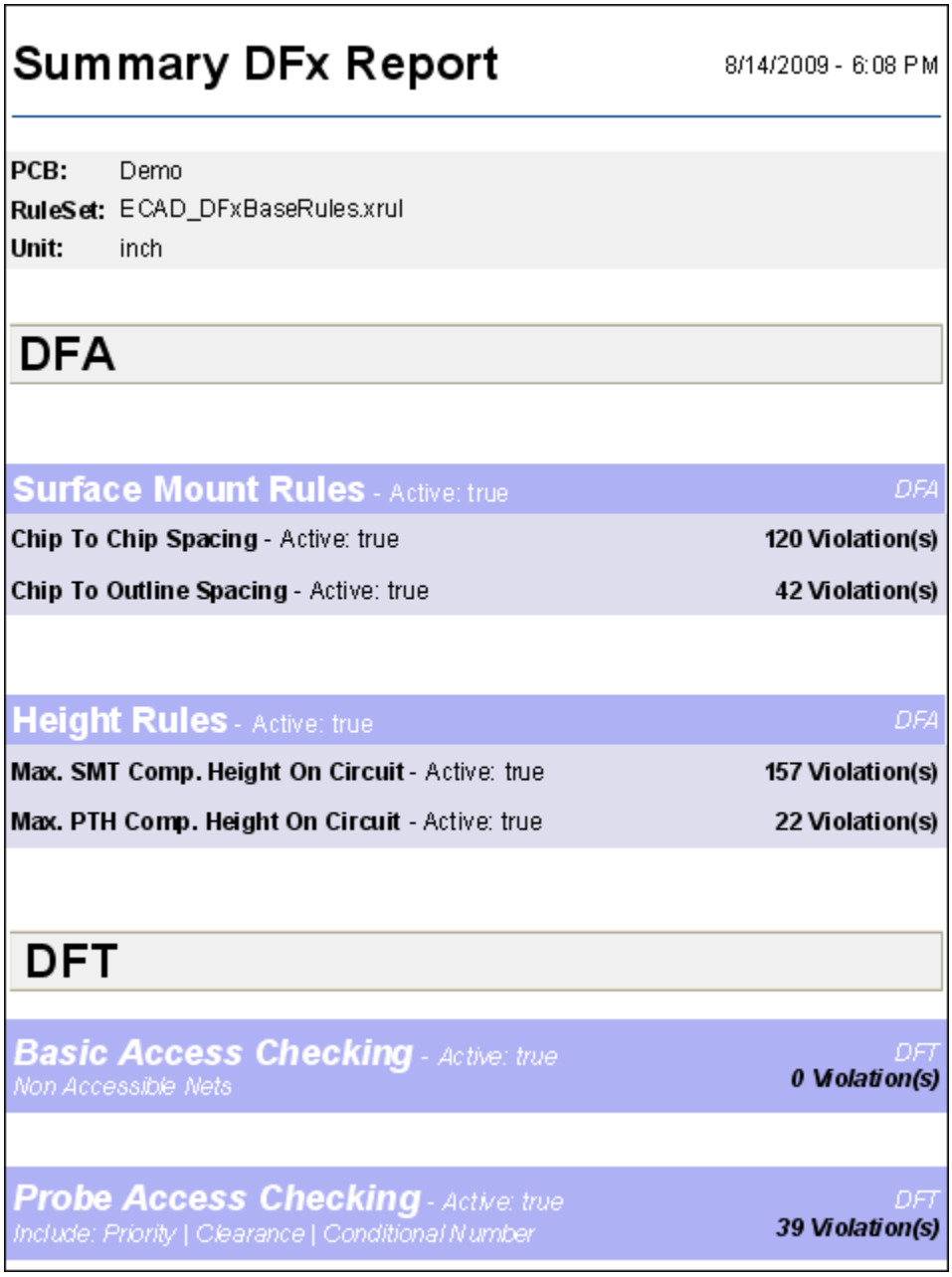
Generate Design for Manufacturing reports

When you launch a new design analysis, both the summary and detailed reports are automatically available. When you choose one of these reports, it opens automatically in your browser. You can also open these reports for tests that you have saved.

Do one of the following:

- For a summary report, on the **ECAD DFx** toolbar, click **Display DFx Summary Report**  .

  The report opens in your browser.



Summary DFx Report

8/14/2009 - 6:08 PM

PCB: Demo
RuleSet: ECAD_DFxBaseRules.xrul
Unit: inch

DFA

Surface Mount Rules - Active: true — DFA
Chip To Chip Spacing - Active: true — 120 Violation(s)
Chip To Outline Spacing - Active: true — 42 Violation(s)

Height Rules - Active: true — DFA
Max. SMT Comp. Height On Circuit - Active: true — 157 Violation(s)
Max. PTH Comp. Height On Circuit - Active: true — 22 Violation(s)

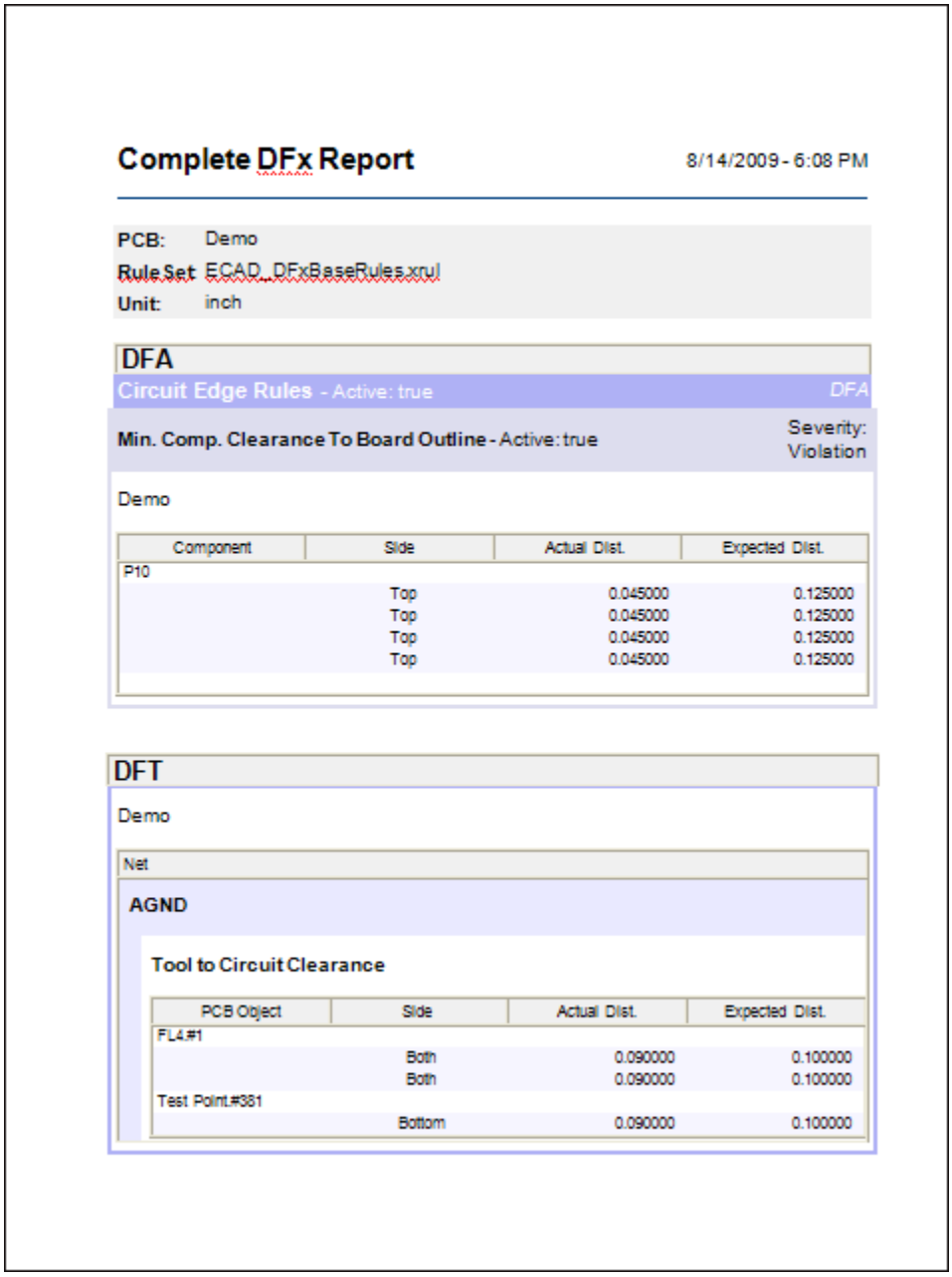
DFT

Basic Access Checking - Active: true — DFT
Non Accessible Nets — 0 Violation(s)

Probe Access Checking - Active: true — DFT
Include: Priority | Clearance | Conditional Number — 39 Violation(s)

- For a detailed report, on the **ECAD DFx** toolbar, click **Display DFx Detail Report** .

  The report opens in your browser.



Complete DFx Report 8/14/2009 - 6:08 PM

PCB: Demo
Rule Set: ECAD_DFxBaseRules.xrul
Unit: inch

DFA

Circuit Edge Rules - Active: true *DFA*

Min. Comp. Clearance To Board Outline - Active: true Severity: Violation

Demo

Component	Side	Actual Dist.	Expected Dist.
P10			
	Top	0.045000	0.125000
	Top	0.045000	0.125000
	Top	0.045000	0.125000
	Top	0.045000	0.125000

DFT

Demo

Net

AGND

Tool to Circuit Clearance

PCB Object	Side	Actual Dist.	Expected Dist.
FL4:#1			
	Both	0.090000	0.100000
	Both	0.090000	0.100000
Test Point:#381			
	Bottom	0.090000	0.100000

Note:

The **Summary DFx Report** provides the following information:

- The PCB name that is being analyzed.
- The file name of the rules file.

- DFA results.
- DFT results.
- The list of processed rules.
- The status for each rule. Rule status is either active or inactive.
- The number of violations or warnings.

The **Complete DFx Report** provides the following information:

- The PCB name that is being analyzed.
- Time and date.
- The file name of the rules file.
- The unit of measurement used in the results.
- DFA results.
- DFT results.
- The list of processed rules.
- For **Basic Access Checking**, **Probe Access Checking**, and **Probe Access Checking by Priority** list the nets that are not accessible.
- For **Probe Access Checking by Clearance**, list of nets that are not accessible and explain for each one the expected minimal clearance and the best possible clearance.
- For **Conditional Number of Access per Net**, list non–accessible nets with the information of maximum number of possible accesses based on clearance or and priority constraints.
- The status for each rule. Rule status is either active or inactive.
- The number of violations or warnings.

Create and save Design for Manufacturing custom rules

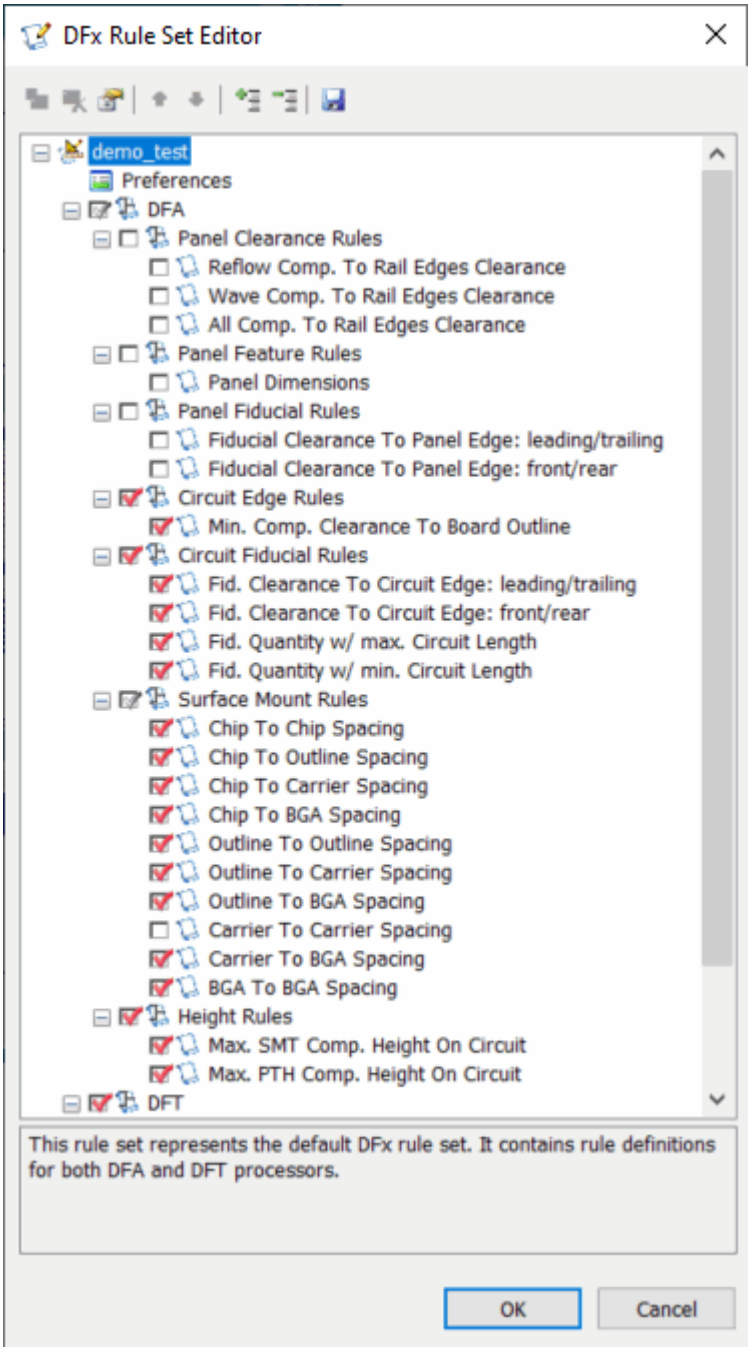
You can customize the attributes of existing DFM rules that are used in an analysis. You customize the attributes of analysis rules to better define manufacturing rules for your facility or environment. For example, you can change the severity level (violation or warning) of the rule, and you can change the

value of the rule. You can also select or clear each rule in the Design for Assembly or Design for Test section of the **DFx Rule Set Editor**.

1. Open an ECAD document.

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

3. To open the **DFx Rule Set Editor,** Click **Edit Rules File** .



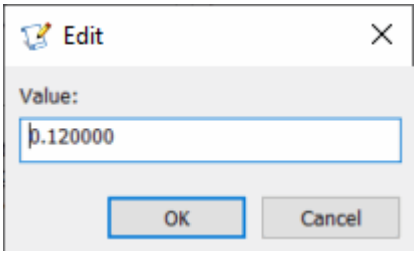
4. Double-click a rule or a sub-rule of interest and its dialog box opens. Do the following:

   a. (Optional) In the **Priorities** section, review the text in the **Description** section to understand what will be tested.

      In the **Parameters** section, you can also review a definition of specific rules.

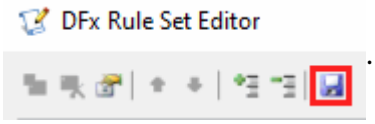
   b. (Optional) In the **Priorities** section, select **Violation** or **Warning**.

   c. (Optional) Double-click the parameter to open the **Edit** dialog box.



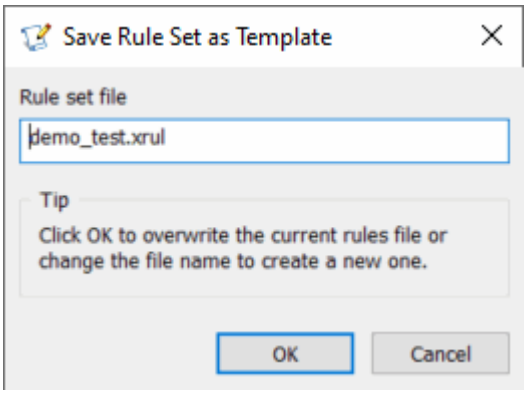
   d. (Optional) Type a new value and click **OK** in the **Edit** dialog box and in the rule or sub-rule dialog box.

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



.

6. In the **Save Rule Set as Template** dialog box, type the rule file name and click **OK**.



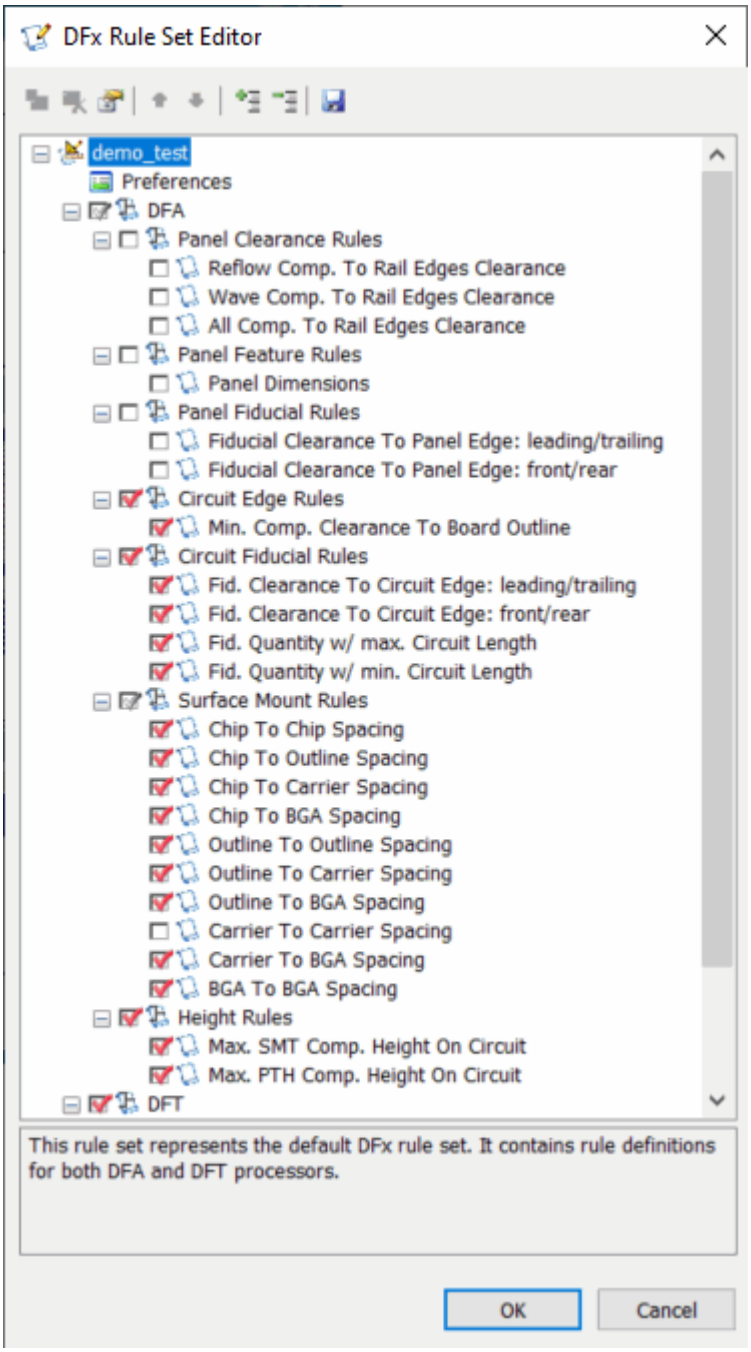
Configure Design for Manufacturing preferences

You can use preference options to specify default Design for Manufacturing preferences. You can set warning and violation highlight colors, and specify Design for Assembly and Design for Test preferences. For example, DFA preferences include specifying leading edges, solder process, and prefix attributes, while DFT preferences include changing the color and symbol of your probing tools.

1. Open an ECAD document.

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

3. To open the **DFx Rule Set Editor,** Click **Edit Rules File** .

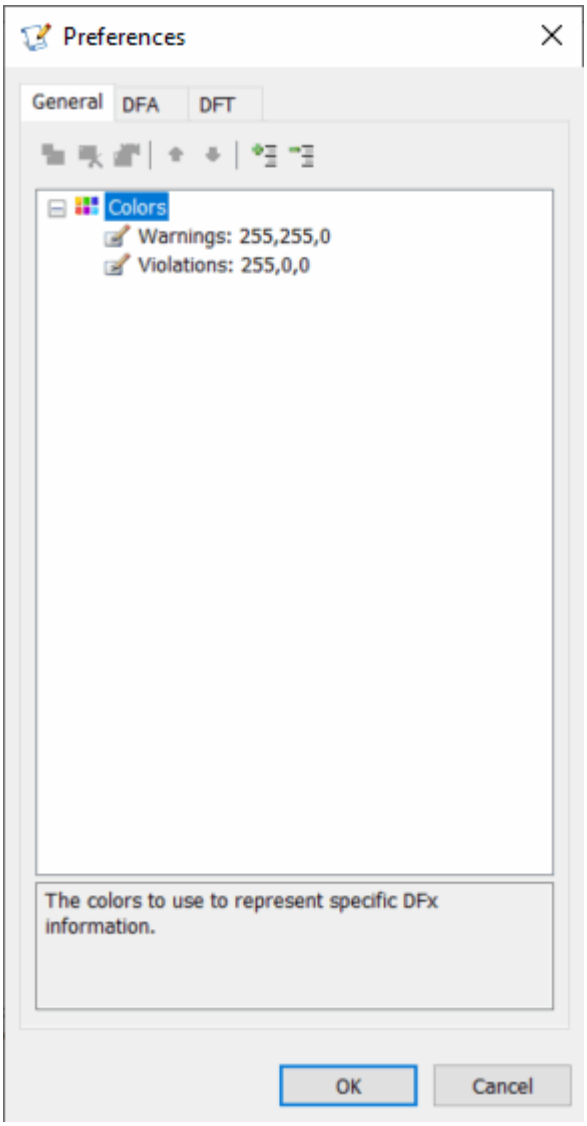


4. Double-click **Preferences**.

5. In the **Preferences** dialog box, navigate to the one of the following tabs:

   - **General**
   - **DFA**
   - **DFT**

   Review the definition of each attribute.

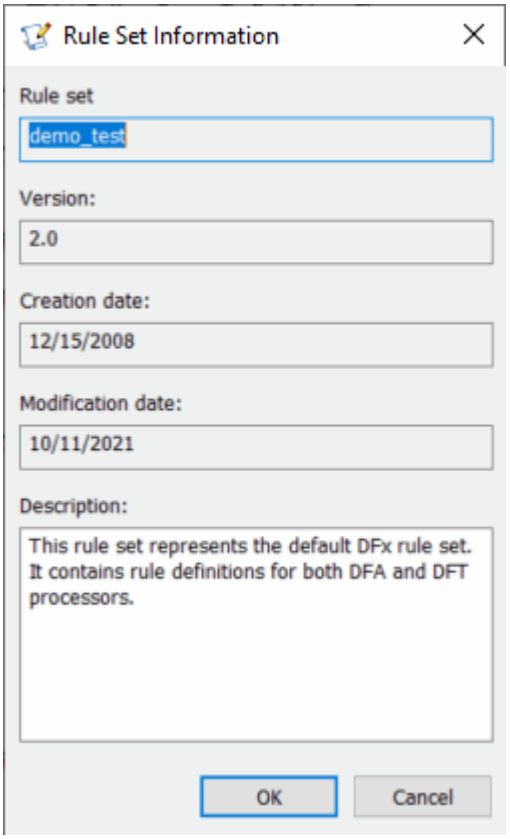


6. (Optional) To change the value of the attribute, double-click the attribute value and make your changes.

7. Click **OK**.

Note:

(Optional) You can modify the description of the rules by double-clicking the rule name. Type your changed text in the **Description** section of the **Rule Set Information** dialog box.



Overview of Design for Test rule definitions and dependencies

DFT rule definitions

Basic Access Checking

This rule performs a complete analysis of the PCB to inventory all possible access points. This is the minimum rule that is executed with DFT. Other rules cannot be processed without running this rule. You cannot change parameters of this rule, such as constraint clearance or conditional number of access per net.

Probe Access Checking

This rule uses access points to check component accessibility. The rule does not require you to define the priority between access points or to define clearance between probing tools. The processing only checks if the net can be accessed through any of pre-defined access types (via, pin/pad and test point) according to their parameters. You must also select **Basic Access Checking**.

Probe Access Checking by Priority

This rule checks that components can be accessed by following a priority order of access. For example, you might define processing to check access first through vias and then check access through pins/pads. You might also exclude accessibility through defined and specific access points. You must also select **Basic Access Checking**.

Probe Access Checking by Clearance

This rule checks component accessibility using access points by applying clearance between probing tools. Consider this rule more like a constraint applied to the **Probe Access Checking** or to **Probe Access Checking by Priority**. This rule cannot be processed alone.

Conditional Number of Access per Net

This rule defines the number of access per net. These set access elements become specific to a particular PCB. By default, the number of access per net is set to 1. Consider this rule more like a constraint applied to the **Probe Access Checking** or to **Probe Access Checking by Priority**. This rule cannot be processed alone.

Dependencies

The **Basic Access Checking** rule is the master rule. You cannot select other rules if you uncheck this rule.

When you select the **Basic Access Checking** rule you can then select both the **Probe Access Checking** and the **Probe Access Checking by Priority** rules. Other rules remain unavailable.

When you select the **Probe Access Checking** rule, all the remaining rules are enabled.

When you select the **Probe Access Checking by Priority** rule, the **Probe Access Checking** becomes automatically unavailable. Other rules remain available.

Working with Design for Test document layers

After DFT processing, you can display two distinct DFT document layers in the **ECAD Layers** view. These DFT layers display access testing points that are dependent on the selected DFT rules.

The first layer, **All Access Points**, represents the complete list of all access points that were found during testing. This layer corresponds to the **Basic Access Checking** rule.

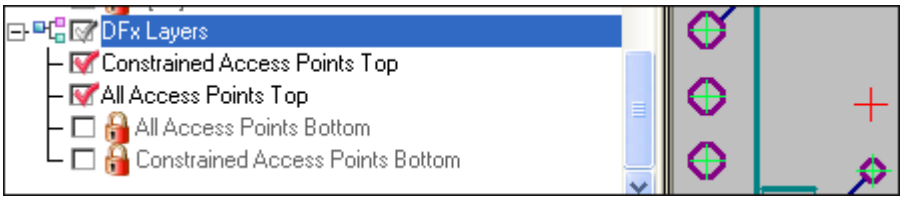
The second layer, **Constrained Access Points**, represents the list of access points once constraints are applied. This layer corresponds to the other DFT rules, excluding the **Basic Access Checking** rule.

DFT layers are displayed in the Viewing window and you can clear and select one or both layers.

How access points are displayed is predefined by the application for some rules and open for changing for others. In the examples that follow, **all** access points are defined as a green cross and **constrained** access points are defined as a red cross.

Example:

Both constrained and all access points are selected.



All access points are selected.

Constrained access points are selected.

When you view different access points, you may find potential access candidates that were not generated from tested DFT constraints. You may be able to change the attribute values of the constraint so that it becomes an available access point for testing.

Change Design for Test document layers

1. Open an ECAD PCB document.
2. Right-click in the Viewing window and select **Preferences**.
3. In the **Preferences** dialog box, click the **DFx** tab.
4. In the **DFT Layer Visibility** section, clear or select one or both of the following:
   - **Turn ON All Access Points**
   - **Turn ON Constrained Access Points**

The next time you open a PCB, your changes are displayed in the **ECAD Layers** view.

For DFT document layers details, see **Working with Design for Test document layers**.

9. Creating reports

Overview of creating reports

As you review and analyze ECAD documents, you may find it useful to generate specific reports for your records or for collaborate with others. You can generate reports for several ECAD objects (bus, nets, parts, and so forth) by selecting the report template that meets your needs. You can then customize this report, and you can preview the report before it is generated. Select the default output formats (HTML, XML, Text, and CSV), or generate a report based on your own output format.

> Note:
>
> You can also generate a markup log report, which may be important and useful when you collaborate with others.

Select a report template

1. On the **ECAD Base** toolbar, click **Create Report**  .

2. In the **Reports** dialog box, from **Report Template** select the ECAD object for the topic of your report.

3. (Optional) From the **Content** section, select **All** to create a report for all selected object types, or choose **Selected** to generate a report for objects that are highlighted in the active ECAD viewing session.

You can then choose to **customize**, **preview**, or **run** the report.

> Note:
>
> Customized report templates are created from an existing report template. You first open an appropriate report template, and then you customize it to suit your needs.

Preview the report

1. On the **ECAD Base** toolbar, click **Create Report**  .

2. In the **Reports** dialog box, click **Preview**.

   The **Report Definition Wizard** dialog box appears, displaying values that will be included in this report.

3. Select **Done** to return to the **Reports** dialog box and select another template, run the report, or end the report session.

4. (Optional) Select **Next**.

5. (Optional) From the **Create Report Output** dialog box that appears, select the output format and the location and file name for the report. Then click **Finish**.

Run the missing net report

The missing net report requires that you have both a PCB (.xfatf) and an associated Schematic (.xsch) file open in the Viewing window.

1. On the **ECAD Base** toolbar, click **Create Report**  .

2. In the **Reports** dialog box, from **Report Template** select **Missing Nets Report**.

3. (Optional) Click **Preview**.

4. (Optional) In the **Available documents to select** dialog box, highlight the ECAD file that appears and then click **OK**.

5. (Optional) In the **Report Definition Wizard** dialog box that appears, review the data in this report.

   You can cancel the report by clicking **Done**, or you can go to the **Create Report Output** dialog box by clicking **Next**.

6. Click **Run**.

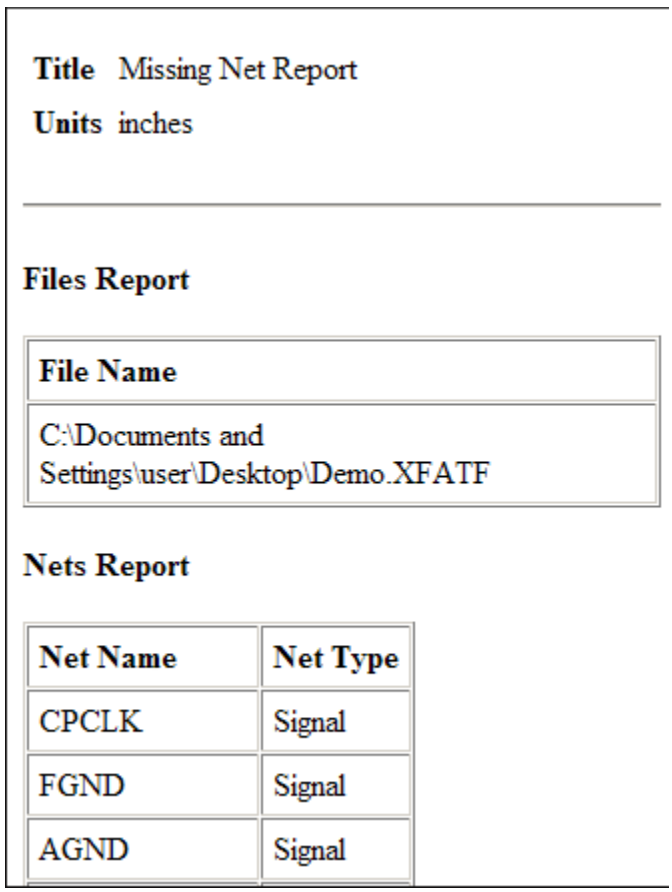
7. In the **Available documents to select** dialog box, highlight the ECAD file that appears and then click **OK**.

8. In the **Create Report Output** dialog box, choose a report output and specify a location for the report file, and then click **Finish**.

   The report appears in your specified location.

Example:

HTML output of the Missing Net Report.



Title Missing Net Report

Units inches

Files Report

File Name
C:\Documents and Settings\user\Desktop\Demo.XFATF

Nets Report

Net Name	Net Type
CPCLK	Signal
FGND	Signal
AGND	Signal

Run reports

1. On the **ECAD Base** toolbar, click Create Report  .

2. In the **Reports** dialog box, click **Run**.

3. In the **Create Report Output** dialog box, choose an output format from the list.

4. (Optional) In the **Create Report Output** dialog box, select an output file format and the location of its style sheet.

5. In the **Create Report Output** dialog box, **Browse** to the location and enter the file name for this report.

6. (Optional) In the **Create Report Output** dialog box, select or clear the **Automatically open report output file** check box.

Customize the report

1. In the **Reports** dialog box, click **Run**.

   The **Report Definition Wizard** dialog box appears.

2. Follow the instructions to create your custom report template.

> Note:
>
> The report options vary for each ECAD object.

10. Printing ECAD documents

Preview documents before printing

1. On the **Printing** toolbar, click **Print Preview**  .

   A preview of what will be printed appears in the **Print Preview** dialog box.

2. (Optional) In the Viewing window, manipulate the PCB or schematic as desired for printing.

   Changes are displayed in the **Print Preview** dialog box.

3. (Optional) To change the printer configuration and options, click **Setup**.

4. (Optional) Click **Print**.

Print ECAD documents

1. (Optional) Preview documents to review how the document will be printed.

2. On the **Printing** toolbar, click **Print**  .

3. In the **Print** dialog box, specify any of the following options:

On this page	Choose this option	Then do this
Print	**Name**	Select a printer to print the documents. Tip: Click **Properties** to view and modify available printer options.
	Default print settings	Select one of the following: • To use the default printer settings, click **Print Defaults**. • To use the printer settings used the last time you printed, click **Last Used.** • To save the current printer settings, click **Save As**, and in the **Save Settings** dialog

On this page	Choose this option	Then do this
		box, type a name for the printer settings, and click **Save**.
	Open Documents	Select the names of open documents that you want to print. Tip: You can also choose **Select None** to clear all selections, or **Select All** to select all documents. If your document is multipaged, you can use the **Page** command button to display the **Page Selection** dialog box. Use the **Print Range** to select the pages you want to print.
Print Layout	**Orientation**	Select one of the following page layouts: • **Portrait** • **Landscape** • **Best Fit** Note: If you choose **Best Fit**, all pages print with the same orientation.
	Print Scale	Select one of the following print scale options: • **Fit to Page** • **1:1 scale** • **Scale** Tip: Use the arrows to increase or decrease the scale percentage, or type in a percentage.
	Quality	Do one of the following:

On this page	Choose this option	Then do this
		• Move the slider to improve the clarity or readability of images previewed or printed. • **Print Image** Moving the slider to the right may improve the print quality of some images. • **Preview Image** Moving the slider to the right may improve the readability and clarity of some images. • Select **Auto Calculate Print Quality** to have the software calculate a solid balance between printing speed and quality for each print job.
	Special Settings	Select from the following: • Select **Print as Monocolor** to print any image (color or grayscale) black and white. • Select **Print Full View** to always print the entire image even if you zoom in or out on the image.
Advanced	**Image Alignment**	Specify where you want an image placed (printed) on the page.
	Margins	Set page margins before printing.
	Treat 2D Vector as Raster	Select or clear letting the software handle how the text is sent to the printer. Note: This setting is useful when printing Vector images that contain font style and font characteristics that are not supported by the local computer. If the Vector image includes a font style that is not supported by the local computer, the software tries to select a close approximation of the font. This may result in text output that is slightly off.

On this page	Choose this option	Then do this
	2D Line Font Relative Spacing	Select this setting to print dashed lines. Note: The patterns of font lines are not a fixed size. They are sized relative to the size of the output.
	MDS File Settings	Select **Page Centric** to set MDS position values to absolute. Example: A starting point for copyright text is set to one inch from the top left corner of the page. The text will always be at this location regardless of the size of the page. Select **Geometry Centric** to scale the location and size of the text with the rest of the image.
Header and Footer	**Font**	To make the Header or Footer section available, select one or both of them. Click **Select Font,** and then choose a font name, style, size, and color.

	Header	Type the header text in the space provided.
		• For additional header options, right-click, then point to **Insert**, and then choose one of following:
		  • **Date**
		  • **Time**
		  • **File**
		  • **Path and File**
		  • **UserID**
		  • **Page**

On this page	Choose this option	Then do this

	Footer	Type the footer text in the space provided.
		• To make the Header or Footer section available, select one or both of them.
		• For additional footer options, right-click, then point to **Insert**, and then choose one of following:
		  • **Date**
		  • **Time**
		  • **File**
		  • **Path and File**
		  • **UserID**
		  • **Page**

Watermark	**Font**	Select **Watermark** to make these options available. Choose **Select Font** to select a font name, style, size, and color. Select **Auto Scale Font Size** to automatically fit the text to the available space.

	Watermark	Type the watermark text in the space provided.
		• For additional watermark options, right-click, then point to **Insert**, and then choose one of following:
		  • **Date**
		  • **Time**
		  • **File**
		  • **Path and File**
		  • **UserID**

On this page	Choose this option	Then do this
		• **Page** Note: If the administrator creates an MDS file, the **Watermark** option is unavailable.
	The button to the left of **Watermark text location**	Choose the direction of the watermark text. Note: Options include text that is displayed diagonally, from lower left to upper right or upper left to lower right, or horizontally in the center of the page.
	Opaque	No further action is required. Note: An opaque watermark can cover text and graphics.
	Transparent	No further action is required. Note: A transparent watermark lightly covers text and graphics. About 50< of the covered data displays on the printed document.
	Background Color	Check **Background Color**, and then select a color. Note: • By default, Background Color is unchecked. • By default, the background color is white.

4. Click **Print**.

11. System administrator reference for configuring ECAD default preferences

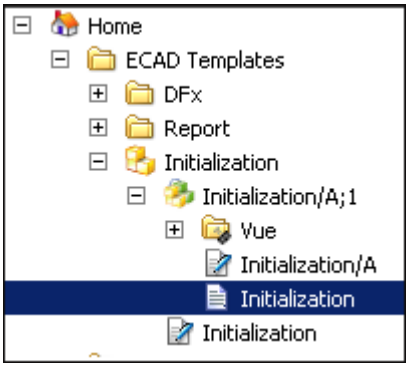
Overview of system administrator configuring ECAD default preferences

As system administrator, you can modify default ECAD preferences such as viewer preferences, compare preferences, or cross probe preferences. Your changes affect the default behavior of working with ECAD files. Default preferences are set values that appear when various viewing dialog boxes are opened.

How to change ECAD default preferences

1. For Teamcenter, your installation created an ECAD Templates in your Home folder. This folder contains ECAD related templates such as Report and DFX templates. The folder also includes the *Initialization.xml* file.

   Example:

2. Use a text editor to open, review, and change the values associated with this XML file.

   Tip:

   For each section that you configure, be sure to configure the `OverWrite` preference to `“Yes”` if the current value is set to `“No”`.

   Here is sample section of an *Initialization.xml* that shows default ECAD options and their values.

```
<Section_ECAD>
    <Version value="3" />
    <TextFont overwrite="no" value="Hershey"/>
    <ViewPreferences>
      <!-- OverWriteViewPreferences: "yes" "no" -->
```

```
      <OverWriteViewPreferences state="No" />
      <!-- State Options: "on" "off" -->
      <DisplayMarkupsOnLoad state="on" />
      <MarkupsAsIcons  state="on" />
      <Border  state="on" />
      <!-- InitialBrowseMode Options:"Browse" "ZoomArea" "Seek" "Pan"
"Zoom" -->
      <InitialBrowseMode  mode="Pan"/>
      <!--  InitialViewPort  Options: BaseView "yes" FitAll "no" -->
      <InitialViewPort  baseView="yes"/>
      <BGColor  red="0xBF" green="0xBF" blue="0xBF" />
      <FBColor  red="0xFF" green="0x00" blue="0x00" />
      <HLColor  red="0xFF" green="0xFF" blue="0x19" />
      <!--Display units  value: "inches" "millimeters" "centimeters" "mils"
-->
      <Units  value="inches" />
      <!--  Component Marker  value: "origin" "center" -->
      <ComponentMarker  value="center" />
    </ViewPreferences>

    <ViewerPreferences>
    </ViewerPreferences>

    <PCBLayerProperties>
      <!-- OverWriteLayerProperties: "yes" "no" -->
 <OverWriteLayerProperties state="Yes" />
 <layer id="ElectricalTop"      on="Yes" red="0x00" green="0x00"
blue="0x84" />
 <layer id="ElectricalBottom"   on="Yes" red="0x00" green="0x82"
blue="0x00" />
 <layer id="ElectricalInner1"   on="No" red="0xFF" green="0x00"
blue="0x80" />
 <layer id="ElectricalInner2"   on="No" red="0x80" green="0x00"
blue="0x40" />
 <layer id="ElectricalInner3"   on="No" red="0xFF" green="0x80"
blue="0x40" />
 <layer id="ElectricalInner4"   on="No" red="0x80" green="0x80"
blue="0xFF" />
 <layer id="ElectricalInner5"   on="No" red="0x00" green="0xFF"
blue="0x40" />
 <layer id="ElectricalInner6"   on="No" red="0xFF" green="0x80"
blue="0xFF" />
 <layer id="ElectricalInner7"   on="No" red="0x80" green="0x80"
blue="0xC0" />
 <layer id="ElectricalInner8"   on="No" red="0x80" green="0x40"
blue="0x00" />
 <layer id="ElectricalInner9"   on="No" red="0x00" green="0x80"
blue="0x80" />
 <layer id="ElectricalInner10"  on="No" red="0x00" green="0x80"
```

```
blue="0xFF" />
 <layer id="ElectricalInner11"  on="No" red="0xFF" green="0x80"
blue="0x80" />
 <layer id="ElectricalInner12"  on="No" red="0xFF" green="0x80"
blue="0x00" />
 <layer id="ElectricalInner13"  on="No" red="0x00" green="0x40"
blue="0x40" />
 <layer id="ElectricalInner14"  on="No" red="0xo0" green="0xFF"
blue="0xFF" />
 <layer id="ElectricalInner15"  on="No" red="0x40" green="0x00"
blue="0x80" />
 <layer id="ElectricalInner16"  on="No" red="0x80" green="0x00"
blue="0x00" />
 <layer id="ElectricalCommon"     on="Yes" red="0x84" green="0x00"
blue="0x84" />
 <layer id="BoardContourCommon" on="Yes" red="0x00" green="0x00"
blue="0x00" />
 <layer id="DocCommon"           on="No" red="0xFF" green="0x00"
blue="0x00" />
 <layer id="MaskTop"              on="No"    red="0xFF" green="0x00"
blue="0x00" />
 <layer id="MaskBottom"         on="No" red="0xFF" green="0x00"
blue="0x00" />
 <layer id="AssemblyTop"        on="Yes" red="0x00" green="0x82"
blue="0x84" />
 <layer id="AssemblyBottom"       on="Yes" red="0x00" green="0x00"
blue="0x00" />
 <layer id="SilkScreenTop"       on="No" red="0xFF" green="0x00"
blue="0x00" />
 <layer id="SilkScreenBottom"     on="No" red="0xFF" green="0x00"
blue="0x00" />
 <layer id="SilkRefTop"           on="No" red="0xFF" green="0x00"
blue="0x00" />
 <layer id="SilkRefBottom"        on="No" red="0xFF" green="0x00"
blue="0x00" />
 <layer id="PasteTop"           on="No" red="0xFF" green="0x00"
blue="0x00" />
 <layer id="PasteBottom"        on="No" red="0xFF" green="0x00"
blue="0x00" />
 <layer id="CompLabelsTop"       on="Yes" red="0x00" green="0x00"
blue="0x84" />
 <layer id="CompLabelsBottom"     on="Yes" red="0x00" green="0x00"
blue="0x84" />
 <layer id="CompCentersTop"       on="No" red="0xFF" green="0x00"
blue="0xFF" />
 <layer id="CompCentersBottom"     on="No" red="0xFF" green="0x00"
blue="0xFF" />
 <layer id="PinCentersTop"       on="No" red="0x00" green="0xFF"
blue="0xFF" />
```

```
 <layer id="PinCentersBottom"     on="No" red="0x00" green="0xFF"
blue="0xFF" />
 <layer id="CADOriginCommon"     on="Yes" red="0x00" green="0x00"
blue="0x84" />
 <layer id="BodyLimitsTop"        on="No" red="0xFF" green="0x00"
blue="0x00" />
 <layer id="BodyLimitsBottom"     on="No" red="0xFF" green="0x00"
blue="0x00" />
 <layer id="ReferencesTop"       on="No" red="0x00" green="0x00"
blue="0x84" />
 <layer id="ReferencesBottom"    on="No" red="0x00" green="0x00"
blue="0x84" />
 <layer id="TestPointsTop"       on="No" red="0xFF" green="0x00"
blue="0xFF" />
 <layer id="TestPointsBottom"    on="No" red="0xFF" green="0x00"
blue="0xFF" />
    </PCBLayerProperties>

    <SCHLayerProperties>
      <!-- OverWriteLayerProperties: "yes" "no" -->
      <OverWriteLayerProperties state="Yes" />
 <layer id="ComponentName"      on="Yes" red="0x00" green="0x00"
blue="0x84" />
 <layer id="SCHPartName"      on="Yes" red="0x00" green="0x00"
blue="0x84" />
 <layer id="Component"         on="Yes" red="0x00" green="0x82"
blue="0x00" />
 <layer id="ComponentCenter"     on="No" red="0xFF" green="0x00"
blue="0x00" />
 <layer id="Pin"              on="Yes" red="0xFF" green="0x00"
blue="0x00" />
 <layer id="PinName"          on="No" red="0xFF" green="0x00" blue="0x00" />
 <layer id="PinNumber"         on="Yes" red="0xFF" green="0x00"
blue="0x00" />
 <layer id="GndPower"         on="Yes" red="0xFF" green="0x00"
blue="0x00" />
 <layer id="GndPowerText"      on="Yes" red="0xFF" green="0x00"
blue="0x00" />
 <layer id="Net"                 on="Yes" red="0x00" green="0x00"
blue="0x00" />
 <layer id="NetName"          on="No" red="0xFF" green="0x00" blue="0x00" />
 <layer id="Bus"              on="Yes" red="0xFF" green="0x00"
blue="0x00" />
 <layer id="BusName"          on="No" red="0x00" green="0x82" blue="0x84" />
 <layer id="Connector"         on="Yes" red="0x00" green="0xFF"
blue="0xFF" />
 <layer id="ConnectorName"        on="Yes" red="0x00" green="0xFF"
blue="0xFF" />
 <layer id="SheetReference"      on="Yes" red="0xFF" green="0x00"
```

```
blue="0xFF" />
 <layer id="RefName"          on="Yes" red="0xFF" green="0x00"
blue="0xFF" />
 <layer id="Outline"          on="Yes" red="0x00" green="0x00"
blue="0x84" />
 <layer id="OutlineText"          on="Yes" red="0x00" green="0x00"
blue="0x84" />
    </SCHLayerProperties>

    <SCHCompareObjects>
       <!-- OverWriteSCHCompareOptions: "yes" "no" -->
       <OverWriteSCHCompareOptions state="no" />
       <!-- CompareOptions: on ="1" off="0" -->
       <Component    Name="1" PartNumber="1" SymbolName="1"
Attributes="1" />
       <Symbol Name="1">
            <Pin Number="1"  Name ="1"/>
       </Symbol>
       <PartNumber Name="1" Attributes="1"/>
       <Net Name="1" Connectivity="1"/>
    </SCHCompareObjects>

    <PCBCompareObjects>
       <!-- OverWritePCBCompareOptions: "yes" "no" -->
       <OverWritePCBCompareOptions state="no" />
       <!-- CompareOptions: on ="1" off="0" -->
       <Contour    BoardContour="1" />
       <Component    Name ="1" Placement="1" PartNumber="1"
FootprintName="1"
                Attributes="1" />
       <Footprint Name="1" Body="1">
            <Pin Placement="1"  Number="1" Name ="1"/>
            <Padstack Electrical="1" Assembly="1" Silkscreen="1"
                Boardcontour="1" DXF="1"
 <Masking="1" Documentation="1"/>
       </Footprint>
       <PartNumber Name="1" Attributes="1"/>
       <Net Name="1" Connectivity="1" Routing="1"/>
       <Via Placement="1" >
           <Padstack Electrical="1" Assembly="1" Silkscreen="1"
            Boardcontour="1" DXF="1" Masking="1" Documentation="1"/>
       </Via>
       <Other>
           <Padstack Electrical="1" Assembly="1" Silkscreen="1"
            Boardcontour="1" DXF="1" Masking="1" Documentation="1"/>
       </Other>
    </PCBCompareObjects>

    <ComparePreferences>
```

```
   <!-- OverWriteComparePreferences: "yes" "no" -->
   <OverWriteComparePreferences state="No" />
   <CompareColor  red="0xFF" green="0x00" blue="0x00" />
   <!--  CompareTiling  tile: Vertical-"vert" Horizontal-"horz" -->
   <CompareTiling  tile="horz"/>
</ComparePreferences>

<CrossProbePreferences>
   <!-- OverWriteCrossProbePreferences: "yes" "no" -->
  <OverWriteCrossProbePreferences state="No" />
  <!--  CrossProbeTiling  tile: Vertical-"vert" Horizontal-"horz" -->
  <CrossProbeTiling  tile="vert"/>
</CrossProbePreferences>

<MarkupProperties>
  <!-- OverWriteMarkupProperties: "yes" "no" -->
  <OverWriteMarkupProperties state="yes" />
  <!-- Line Style Options:
          <0= solid
          <1= dash
          <9= dot
          <10= dashdot
          <11= dashdotdot
      -->
  <!-- Line Start and End Terminator Options:
          <1= no terminator
          <2= filled arrowhead
          <3= filled circle/ellipse
          <4= inverted filled arrowhead
   <5= "X" terminator
      -->
  <!-- Line Start and End Height and Width Notes:
      <Currently the start and end terminators are sized together
        according to the values
        <specified by headWidth and headHeight.
  -->
  <Line style="0" thickness="0.01" startTerm="1" endTerm = "2"
        red="0x00" green="0x00" blue="0x00"
          <headWidth="0.02" headHeight="0.02" tailWidth="0.02"
                tailHeight="0.02" />
  <!-- Edge Style Options:
          <0= solid
          <1= dash
          <9= dot
          <10= dashdot
          <11= dashdotdot
   <99= no edge
   -->
  <Edge style="0" thickness="0" red="0x00" green="0x00" blue="0x00" />
```

```
      <!-- Fill Style Options:
             <1= solid
             <3= hatch
             <4= no fill
      -->
      <!-- Fill Hatch Style Options: (applicable only when above
            Fill Style = hatch)
             <0= no hatch
             <1= horizontal
             <2= vertical
             <3= forward diagonal
             <4= backward diagonal
       <5= crossed vertical & horizontal
             <6= crossed diagonals
      -->

      <!-- Fill Transparency Options:
        <1.0= transparency;
        <0.0= opaque   -->
      <Fill style="1" hatch="0" transparency="0.0" red="0x00"
            green="0xff" blue="0x00" />

    <!-- Font Name Options: "name" field includes all style qualifiers
        of the font appended to the main font name, e.g. "Arial",
        "Arial Bold", or "Arial Bold Italic". If the font specified by the
        Font Name field is not installed on the system, this setting is
ignored.
          -->
      <Font name="Arial" red="0x00" green="0x00" blue="0x00" />

      <!-- FixedTextType Options:
        <0= Use Auto-translate text
        <1= Use Stamp note
      -->
      <!-- AutoTranslateMsg Options: Index of Auto Translate Message to Use
-->
<!-- NotesFile Options: Complete Path and Name to Notes File, i.e.,
    c:\notes.txt -->
<!-- NotesIndex Options: Index of Notes File to Use -->
<!-- DisplayOnActivation Options: "yes" "no"  -->
<!-- NotesFile: for demonstration purpose, a sample notes file is provided
    with each installed product. For example: "C:\Program Files\Siemens\
    Teamcenter[version]\Visualization\Examples\2D\stdnotes.txt"
      -->

      <Text    FixedTextType="0" AutoTranslateMsg="100" NotesFile=""
NotesIndex="1"
DisplayOnActivation="yes" />
```

```
      <!-- Measurement Preferences: -->

      <!-- Size Options: 0=XX-Small  1=XSmall  2=Small 3=Medium 4=Large
            5=X-Large  6=XX-Large -->
      <!-- DisplayPrecision Options: 0-6; number of digits displayed to
            right of decimal -->

      <Measurement    Size="3" red="0x00" green="0x00" blue="0x00"
            DisplayPrecision="2" />
    </MarkupProperties>

    <DFxProperties>
      <!-- OverWriteViewPreferences: "yes" "no" -->
      <!-- Reports: empty strings("") revert to system defaults.  -->
      <OverWriteDFxPreferences state="No" />
      <Reports         DefaultXSL=""  SummaryXSL=""
DefaultReportOutput=""  />
    </DFxProperties>

    <SearchPreferences>
    <!-- OverWriteSearchPreferences "yes" "no" -->
      <OverWriteSearchPreferences state="no" />
      <!-- For each unit group only one unit should be specified as the
            default. -->
      <!-- The defaults multiplier should be 1-->
      <Resistance>
<Unit name="ohm"  default="yes" multiplier="1" description="ohms" />
<Unit name="kohm"  default="no" multiplier="1000" description="kiloohms" />
<Unit name="mohm"  default="no" multiplier="1000000"
description="megaohms" />
      </Resistance>
      <Capacitance>
<Unit name="farad"  default="yes" multiplier="1" description="farad"/>
<Unit name="mF"  default="no" multiplier="0.001" description="millifarad"/>
<Unit name="uF"  default="no" multiplier="0.000001"
description="microfarad"/>
        <Unit name="nF"  default="no" multiplier="0.000000001"
              description="nanofarad"/>
        <Unit name="pF"  default="no" multiplier="0.000000000001"
              description="picofarad"/>
      </Capacitance>
      <Inductance>
<Unit name="henry"  default="yes" multiplier="1" description="henry"/>
<Unit name="H"  default="no" multiplier="1" description="henry"/>
<Unit name="mH" default="no" multiplier="0.001" description="millihenry"/>
<Unit name="uH" default="no" multiplier="0.000001"
description="microhenry"/>
<Unit name="nH" default="no" multiplier="0.000000001"
description="nanohenry"/>
```

```
<Unit name="pH" default="no" multiplier="0.000000000001"
        description="picohenry"/>
      </Inductance>
      <Voltage>
            <Unit name="volt"  default="yes" multiplier="1"
description="volt"/>
            <Unit name="V"  default="no" multiplier="1" description="volt"/>
      </Voltage>
      <Current>
            <Unit name="ampere"  default="no" multiplier="1"
description="amp"/>
            <Unit name="amp"  default="yes" multiplier="1"
description="amp"/>
               <Unit name="A"  default="no" multiplier="1"
description="amp"/>
      </Current>
      <Power>
            <Unit name="watt"  default="yes" multiplier="1"
description="watt"/>
            <Unit name="W"  default="no" multiplier="1" description="watt"/>
      </Power>
      <Energy>
  <Unit name="joule"  default="yes" multiplier="1" description="joule"/>
      </Energy>
      <Frequency>
            <Unit name="hertz"  default="no" multiplier="1"
description="hertz"/>
            <Unit name="Hz"  default="yes" multiplier="1"
description="hertz"/>
      </Frequency>
      <Time>
<Unit name="s"  default="yes" multiplier="1" description="second"/>
<Unit name="ms"  default="no" multiplier="0.001" description="millisecond"/>
<Unit name="us"  default="no" multiplier="0.000001"
description="microsecond"/>
        <Unit name="ns"  default="no" multiplier="0.000000001"
                description="nanosecond"/>
        <Unit name="ps"  default="no" multiplier="0.000000000001"
                description="picosecond"/>
      </Time>
      <Tolerance>
            <Unit name="%"  default="yes" multiplier="1"
description="percent"/>
      </Tolerance>
      <Distance>
            <Unit name="in"  default="no" multiplier="1000"
description="inch"/>
            <Unit name="mils"  default="yes" multiplier="1"
description="mils"/>
```

```
            <Unit name="m"  default="no" multiplier="39370"
description="meter"/>
        <Unit name="cm"  default="no" multiplier="393.70"
                description="centimeter"/>
        <Unit name="mm"  default="no" multiplier="39.370"
                description="millimeter"/>
      </Distance>
    </SearchPreferences>


  </Section_ECAD>
```