TEAMCENTER

Xpedition and PADS Integration With Teamcenter

Teamcenter 2412

SIEMENS

Unpublished work. © 2025 Siemens

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

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

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

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

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

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

About Siemens Digital Industries Software

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

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

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

Contents

Integrating Xpedition and PADS with Teamcenter 1-1

Teamcenter EDA architecture 2-1

Integrating Xpedition or PADS or PADS Professional with Teamcenter EDA for design management

Planning your deployment for design management 3-1
Workflow to integrate your ECAD applications with Teamcenter for design management 3-4
Install Teamcenter EDA for design management using Deployment Center 3-5
Installing Teamcenter server for design management through TEM 3-6
- Installing Teamcenter server 3-6
- Install BMIDE to configure derived data for EDA objects 3-8
- Install or update Dispatcher for setting up remote ECAD translation 3-8

Setting up Teamcenter server for design management 3-9
- Overview of setting up and configuring Teamcenter server for design management 3-9
- Populate the Teamcenter database with ECAD components 3-11
- Add EDA relations and EDA objects in Active Workspace Relation Browser 3-12
- Map ECAD attributes with Teamcenter properties 3-14
- Configure derived data for ECAD design objects 3-17
- Configure design variants 3-26
- Manage design files in local storage 3-28
- Apply Teamcenter user's permission on the network shared ECAD design folder 3-29
- Allow ECAD designers to assign a Teamcenter project from the EDA client 3-29
- Set configurations based on the ECAD applications used 3-32
- Configure EDA to access Active Workspace for collaboration and issue management 3-33
- Allow users to view the PCB board requirements for ECAD objects in Active Workspace 3-34
- Customize the acronyms used for ECAD design objects in Teamcenter 3-34
- Configure the EDA client to use the Teamcenter EDA panels instead of Active Workspace 3-35
- Managing and configuring Teamcenter preferences for design management 3-35

Installing the EDA client for design management on the ECAD workstation 3-36
- Install the EDA client for design management 3-36
- Verify the EDA client installation 3-39

Setting up the EDA client for design management 3-39
- Configurations required to set up the EDA client for design management 3-39
- Integrate Teamcenter menus within the ECAD tool 3-40

© 2025 Siemens

Mapping reference designator attributes for your ECAD tools 3-45
Configuring cache and staging directory 3-48
Modify default user properties in the TCEDAClient.properties file 3-58
Set a flatter folder structure for saving ECAD designs 3-59
Modifying the default logging configuration 3-59
Customize the integration by adding additional command calls 3-60
ECAD tool connector configurations and customizations for design management 3-61
Overview of additional customizations required for your ECAD tool 3-61
Connector configurations and customizations for PADS Professional and Xpedition 3-61
Connector configurations and customizations for PADS Layout and PADS Logic 3-72
Setting up ECAD translation 3-74
ECAD translation modes 3-74
Enable PcbToFatf translator 3-75
Supported translator services 3-76
Installing the ECAD viewer 3-77

Integrating EDM with Teamcenter EDA for managing design files

Planning your deployment for EDM integration with Teamcenter 4-1
Workflow to install and setup Teamcenter EDA for EDM integration 4-4
Installing Teamcenter server and the EDA client for EDM integration through Deployment Center 4-4
Installing Teamcenter server for EDM integration through TEM 4-6
Installing the Teamcenter server for EDM integration 4-6
Install BMIDE to configure derived data for EDA objects 4-8
Install or update Dispatcher for setting up remote ECAD translation 4-8
Setting up Teamcenter server for Xpedition EDM integration 4-9
Configurations required to integrate Teamcenter with the Xpedition EDM 4-9
Populate the Teamcenter database with ECAD components 4-10
Add EDA relations and EDA objects in Active Workspace Relation Browser 4-12
Customizing ECAD attribute mappings 4-14
Configure derived data for EDX design files 4-16
Configure design variants 4-26
Allow ECAD designers to assign a Teamcenter project from the EDA client 4-27
Set configurations based on the ECAD applications used 4-31
Allow users to view the PCB board requirements in Active Workspace 4-31
Configure single sign-on for Teamcenter and Xpedition EDM integration 4-32
Set properties on EDM designs during design review 4-32
Customize the acronyms used for ECAD design objects in Teamcenter 4-35
Managing and configuring Teamcenter preferences for design management 4-36
Install the EDA client for EDM integration 4-36
Setting up EDM integration 4-39
Configurations required for setting up EDM integration 4-39
Integrating Xpedition EDM with Teamcenter 4-39
Configure and deploy EDM server with Teamcenter 4-40
EDM queue configuration for Teamcenter EDA integration 4-44
Create an EDM baseline profile to enable automation 4-47

© 2025 Siemens

Set automatic import options 4-54
Export X-Y coordinates and rotation details of the EDX file into Teamcenter 4-55
Retain the EDX files after publishing to Teamcenter 4-60
EDM tool connector configurations for Teamcenter integration 4-60
Overview of connector configurations required for EDM integration 4-60
How variant names are extracted to an EDX package from EDM project 4-60
How design BOM is extracted to EDX package for an EDM project 4-60
Working with intermediate and viewable files 4-60
Working with supported data files 4-61
Current behavior of EDM integration 4-62
Setting up ECAD translation 4-62
ECAD translation modes 4-62
Enable PcbToFatf translator 4-63

Integrating Xpedition or PADS or EDM with Teamcenter EDA for part library management

Planning your deployment for part library management 5-1
Workflow to integrate your ECAD applications with Teamcenter for part library management 5-3
Install Teamcenter EDA for part library management using Deployment Center 5-4
Installing Teamcenter server for part library management through TEM 5-6
Installing the Teamcenter server for part library management 5-6
Install BMIDE for mapping ECAD library objects with Teamcenter objects 5-8
Install or update Dispatcher for translating ACIS parts to NX parts and JT file 5-8
Setting up Teamcenter server for part library management 5-8
Overview of configuring Teamcenter for ECAD part library management 5-8
Create external applications for your ECAD tool in Teamcenter 5-10
Create a Teamcenter part library 5-12
Mapping ECAD part attributes to Teamcenter attributes 5-13
Creating classification mapping for ECAD parts with Teamcenter classes 5-19
How to use a custom part type in the EDA library integration 5-22
Synchronize ECAD library parts with multiple external sites 5-29
Managing and configuring Teamcenter preferences for part library management 5-31
Installing the EDA client for part library management 5-31
Install the EDA client for part library management 5-31
Verify the EDA client installation 5-34
Configure the EDA client for part library management 5-35
Steps to configure the EDA client for part library management 5-35
Create customized gateway configurations for managing ECAD part data 5-36
Create an ECAD library configuration 5-40
Map ECAD library objects and attributes using ODBC 5-42
Set the working library 5-56
Set up EDA part library synchronization with Teamcenter 5-56
Enable acistoprt translator 5-58
Set up automatic synchronization of EDM part library with Teamcenter 5-59
ECAD tool customizations for part library management 5-60

© 2025 Siemens

Overview of ECAD tool customizations required for part library management 5-60
Setting up PADS Library integration 5-60
Setting up Xpedition Library integration 5-61
Setting up EDX Library integration 5-62

Upgrading to a newer version of Teamcenter EDA

Taskflow to upgrade to a newer version of Teamcenter EDA 6-1
Upgrade Teamcenter server 6-1
Configure Teamcenter for design management 6-2
Overview of configurations required for design management 6-2
Modify the style sheet to include EDA derived data relation in Active Workspace 6-2
Add EDA relations and EDA objects in Active Workspace Relation Browser 6-3
Configure Teamcenter for part library management 6-5
Overview of configurations required for part library management 6-5
Synchronize ECAD library parts with multiple external sites 6-5
Steps to perform before you upgrade the EDA client 6-7
Collect Teamcenter server details required during the EDA client upgrade 6-8
Upgrade to a newer version of the EDA client 6-8
Delete the cache and staging directory 6-11

Uninstall the EDA client from the ECAD workstation 7-1

Deploying Teamcenter EDA documentation on your local drive or network 8-1

© 2025 Siemens

1. Integrating Xpedition and PADS with Teamcenter

Teamcenter Electronic Design Automation (EDA) integrates Siemens EDA ECAD applications with Teamcenter so that ECAD designers can move design data into and out of Teamcenter and manage the ECAD part library data in Teamcenter.

Teamcenter supports two types of EDA integrations:

- **ECAD design integration**

  The Teamcenter EDA integration with Xpedition and PADS for design management enables ECAD designers to manage ECAD design data in Teamcenter.

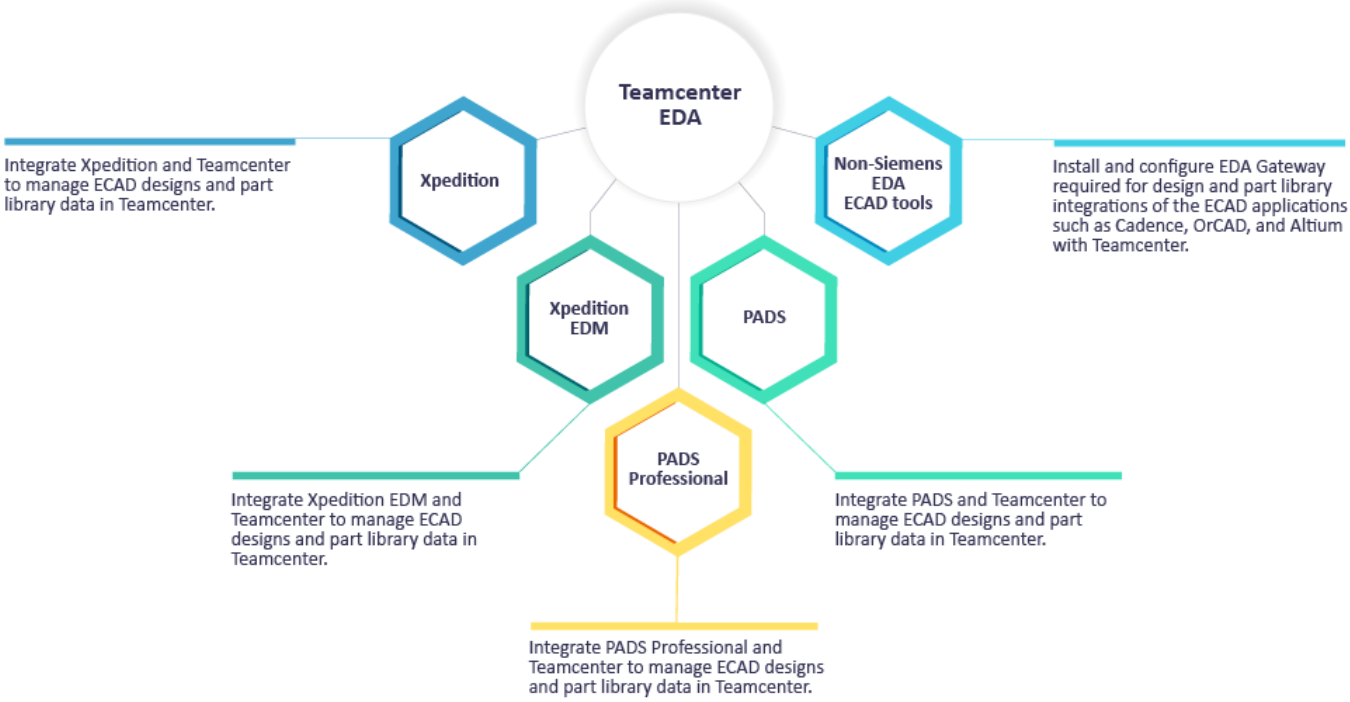
- **ECAD part library integration**

  The Teamcenter EDA integration with Xpedition and PADS for part library management enables ECAD designers and librarians to manage all the parts and part attributes in the ECAD libraries, as well as manage the relationships between these objects. Teamcenter library data can be synchronized with each individual ECAD tool's local library on the workstation so that design team is up to date.

For the ECAD designers and library managers to work with the ECAD design and library parts, you as an administrator must integrate Xpedition and PADS with Teamcenter by:

- Installing and setting up Teamcenter with Xpedition or PADs for design management.
- Installing and setting up Teamcenter with EDM for design management.
- Installing and setting up Teamcenter with Xpedition or PADS or EDM for part library management.

© 2025 Siemens



Where do I go from here?

ECAD Designer	See Managing Xpedition or PADS Designs in Teamcenter for instructions on how to manage ECAD designs in Teamcenter.
Library Administrator	See Managing Xpedition or PADS Part Library in Teamcenter for instructions on how to manage ECAD part library data in Teamcenter.
Administrator	
How do I install and set up design integration for Xpedition and PADS with Teamcenter?	**Planning your deployment for design management** has information about the appropriate components that must be installed on the server (Teamcenter) and client machines (ECAD workstation - Xpedition or PADS). See the **workflow** for guidance on the steps to integrate Xpedition and PADS with Teamcenter for design management.
How do I install and set up design integration for EDM with Teamcenter?	**Planning your deployment for EDM design management** has information about the appropriate components that must be installed on the server (Teamcenter) and client machines (ECAD workstation - Xpedition EDM).

© 2025 Siemens

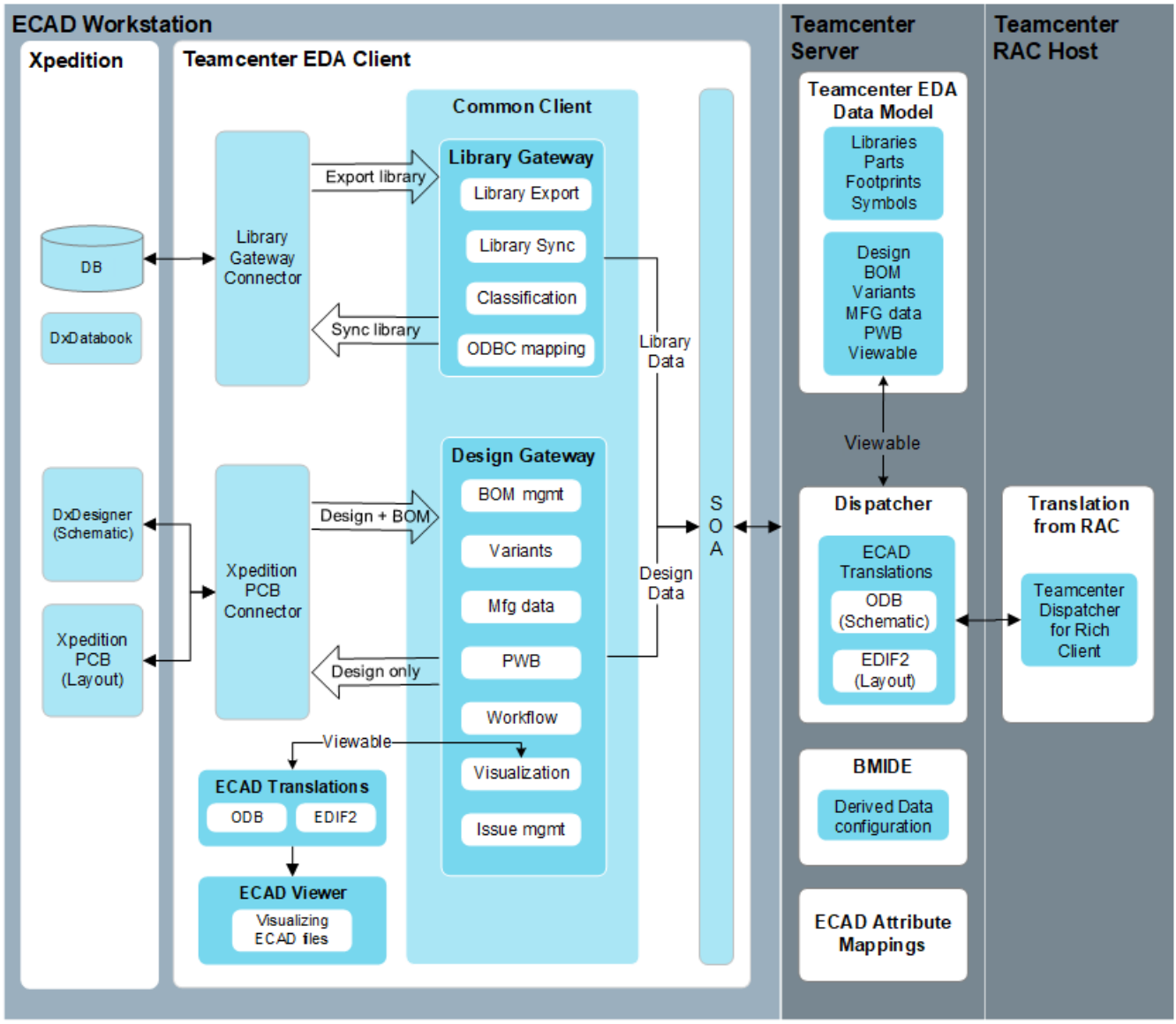
	See the **workflow** for guidance on the steps to integrate Xpedition and PADS with Teamcenter for EDM design management.
How do I install and set up library integration for Xpedition, EDM, and PADS with Teamcenter?	**Planning your deployment for part library management** has information about the appropriate components that must be installed on the server (Teamcenter) and client machines (ECAD workstation - Xpedition or PADS). See the **workflow** for guidance on the steps to integrate Xpedition and PADS with Teamcenter for part library management.
How can I perform additional customizations in the EDA client operations for design and part library integrations?	See Callbacks in Teamcenter EDA and Teamcenter EDA Schema — Reference documentation for information on additional customizations. Additionally, to customize the connector integrations, refer to the Teamcenter EDA Design Connector Integration — Reference, Teamcenter EDA Library Connector Integration — Reference, and Teamcenter Gateway for EDA Integration — Reference.
Where can I get a list of preferences that I need to set for design and part library integrations?	There are several sources from which you can retrieve a list of preferences. You can view the list in Administration data report, Teamcenter rich client, Teamcenter Active Workspace, and raw XML file.
How do I install and configure the EDA Gateway for (non-Siemens EDA) ECAD applications?	See the documentation on Teamcenter EDA Gateway for (Non-Siemens EDA) ECAD Applications.

© 2025 Siemens

© 2025 Siemens

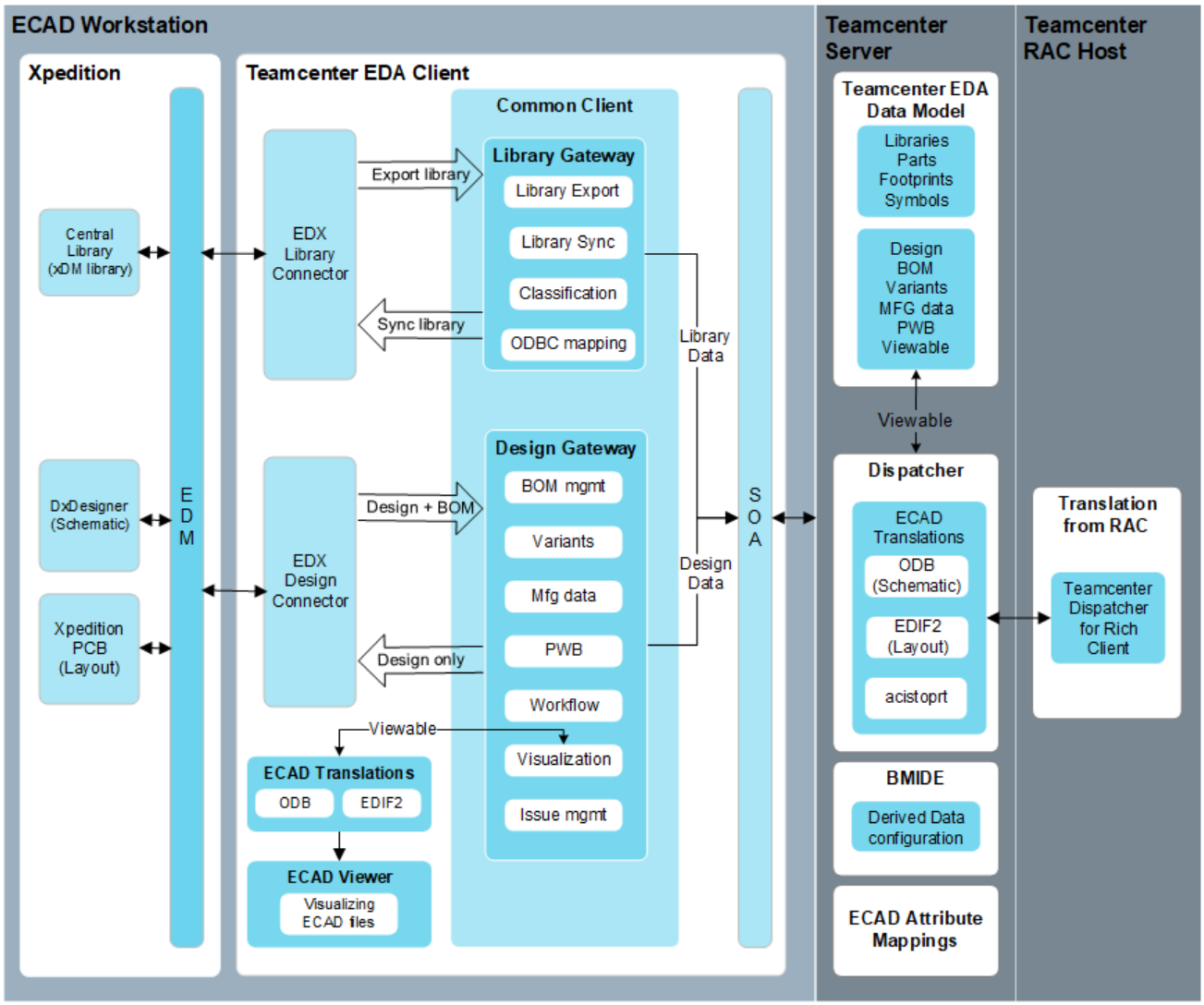
2. Teamcenter EDA architecture

The components of Teamcenter EDA and their interactions for design management and part library management with ECAD tool, for example, Xpedition, are as follows:



The components of Teamcenter EDA and their interactions for design management and part library management with ECAD tool, for example, EDM, are as follows:

© 2025 Siemens



© 2025 Siemens

3. Integrating Xpedition or PADS or PADS Professional with Teamcenter EDA for design management

Planning your deployment for design management

For installing and integrating Teamcenter EDA with Xpedition or PADS or PADS Professional for design management, you must plan your deployment and your environment setup. You can install the Teamcenter EDA server and client components through **Deployment Center** or through **TEM**.

As an administrator, you must install the appropriate components on the server (Teamcenter) and client machines (ECAD workstation) as per the site requirements as mentioned below:

Server components	Requirements
Teamcenter server	Teamcenter server is used to store and track the ECAD designs. For information about the versions of operating systems, third-party software, and Teamcenter software configurations that are certified for the EDA platform, refer to the **Hardware and Software Certifications** knowledge base article on Support Center. If you have an existing Teamcenter environment, you can update it to support the latest EDA version. Ensure that you download the compatible versions from Support Center before starting your deployment.
EDA server support	EDA server support is required to add ECAD data model to the Teamcenter server environment. You must add EDA server support on top of the Teamcenter environment. As the EDA client is supported in a four-tier architecture, the EDA Server Support feature (EDA data model) must also be installed on a four-tier Teamcenter server.
Business Modeler IDE	Business Modeler IDE is required to configure derived data for EDA objects. Before working with Teamcenter EDA objects in Business Modeler IDE, you must install the EDA server support features in Teamcenter Environment Manager (TEM). From the Features panel, choose **Extensions→Mechatronics Process Management→EDA for Business Modeler IDE**. You must also

© 2025 Siemens

Server components	Requirements
	install the EDA Server Support template applicable to your project.
Dispatcher	Dispatcher is required for remote ECAD translations to translate and save the design intermediate file into a viewable dataset in Teamcenter. If Dispatcher is already installed in your existing Teamcenter environment, you can use the same instance. However, you must modify it to include the translators that are specific to ECAD translation.

Client components	Requirements
Teamcenter EDA client	The EDA client moves designs between Xpedition or PADS tools and Teamcenter. The EDA client must be installed on the ECAD workstation using the Teamcenter software kit. Before installing the EDA client, check the **Hardware and Software Certifications** knowledge base article on Support Center to identify which versions of the Xpedition or PADS tools are supported on which versions of Teamcenter EDA. Siemens Digital Industries Software recommends that you first perform the installation on a test system to verify the parameter values required for the installation on the ECAD workstations. **Prerequisites and requirements before installing the EDA client:** • **Supported Operating System:** The EDA client framework is supported only on Windows. • **Supported Java version:** For information about the supported Java versions that are certified for the EDA platform, see the **Hardware and Software Certifications** knowledge base article on Support Center. • **Software requirements specific to the ECAD tools on which the EDA client should be installed:** • If you use PADS and select PADS Logic as the schematic design tool, install PADS Logic or PADS Layout on your client workstations.

© 2025 Siemens

Client components	Requirements
	If you use PADS and select Xpedition Designer as the schematic design tool, install Designer (PADS Flow) on the client workstations. • If you use Xpedition as the ECAD design tool, install Designer and Xpedition PCB on the client workstations. • **Information required for EDA client installation from the Teamcenter installation administrator:** • FMS client cache (FCC) If the FMS client cache (FCC) exists, use the existing one or specify a new one. • The FCC parent information for the EDA client, including the host and port values The minimum information required is the host name and the port value. • The four-tier, middle-tier web application server URI and the connection name for the EDA client • Logon URL and the application ID to enable Teamcenter Security Services • Host name, port value, and license server mode of the Siemens Digital Industries Software common license server Note: Teamcenter software kit contains the 64-bit installer for Windows system. You can select Win32 applications directly from TEM or DC along with the rest of the Win64 applications from within the Teamcenter software kit. • **Other installation considerations** • Only the English versions of ECAD tools are supported. • Item ID, variant name, dataset name, reference designator value, and mapped attribute names and values must contain only ASCII characters. Mapped attributes include

© 2025 Siemens

Client components	Requirements
	CAD attribute mappings and component instance attribute mappings. • The staging directory and all file names must contain only ASCII characters. • PADS Logic does not support variants.
(Optional) ECAD viewer	Install the ECAD viewer as a design-view tool and a markup tool.

Workflow to integrate your ECAD applications with Teamcenter for design management

Scenarios to deploy the Teamcenter EDA server and client components for design management:

Using Deployment Center	*Using TEM*
Install Teamcenter server and the EDA client	Install Teamcenter server, Install BMIDE, (Optional) Install Dispatcher
↓	↓
Set up Teamcenter server for design management	Set up Teamcenter server for design management
↓	↓
Set up the EDA client for design management	Install the EDA client
↓	↓
Perform additional configurations and customizations required for your ECAD applications	Set up the EDA client for design management
↓	↓
Set up ECAD translation	Perform additional configurations and customizations required for your ECAD applications
↓	↓
(Optional) Install the ECAD viewer	Set up ECAD translation

© 2025 Siemens

(Optional) Install the ECAD viewer

Install Teamcenter EDA for design management using Deployment Center

The following procedures assume that you are installing EDA on an existing Teamcenter and that you are familiar with Deployment Center.

Procedure

1. Log on to Deployment Center and select the environment to which you want to add Teamcenter EDA.

2. Go to the **Applications** task. Click **Add or Remove Selected Applications**.

3. In the **Available Applications** panel, use the web browser search to find the Electronic Design Automation (EDA) application. Select the application, and then click **Update Selected Applications**.

   Deployment Center automatically selects any additional dependent applications.

4. In the **Available Applications** panel, under **EDA Client**, select the ECAD tool with which you are integrating Teamcenter EDA for design management and other options as required on the ECAD client machine:

   - (Optional) Select **EDA Translation**.
   - **Mentor PADS**
   - **Mentor Xpedition Designer and Layout**

   By default, all options under **EDA Client** are selected. Remove the tools and applications that are not required for design integration and click **Update Selected Applications**.

   Deployment Center automatically selects any additional dependent applications.

5. Go to the **Components** task.

6. In the **Selected Components** list of the **Components** task, note any remaining components whose configuration status is not **100%**. Select each incomplete component, enter required parameters, and save component settings until all components in the environment show a configuration status of **100%**.

© 2025 Siemens

When all components are fully configured, the **Deploy** task is enabled.

7. Go to the **Deploy** task. Click **Generate Install Scripts** to generate deployment scripts you will use to update affected machines.

   When script generation is complete, note any special instructions in the **Deploy Instructions** panel.

8. Locate deployment scripts, copy each script to its target machine, and then run each script on its target machine.

   For more information about running deployment scripts, see *Run the deployment scripts* in the Deployment Center documentation.

   You can also generate mini-kits for mass client deployment. Running the **generateMiniSoftware** utility extracts only the necessary software from the full software kits and builds a smaller, more portable software kit for the specific client deployment. For more information, see *Generating mini kits for mass client installations* in *Deployment Center — Usage* on Support Center.

   If you face any issues during the deployment, see *Troubleshoot the deployment script* in the Deployment Center documentation.

Installing Teamcenter server for design management through TEM

Installing Teamcenter server

Install Teamcenter

To install Teamcenter EDA, you must first install Teamcenter. To install the Teamcenter corporate server, refer to the Teamcenter Installation on Windows Using TEM help.

If you have an existing Teamcenter environment, you can update it to support the latest EDA version.

Refer to the Install patches on a Teamcenter server topic in the Teamcenter Installation on Windows Using TEM help for instruction on installing Teamcenter patches.

Add EDA support on the Teamcenter server for design management

EDA server support adds the EDA data model to the existing Teamcenter data model.

Before installing the EDA server support, you must install the Teamcenter server.

1. From your Teamcenter environment, start Teamcenter Environment Manager (TEM).

2. Select the **Configuration Manager** option to perform maintenance on an existing installation and click **Next** until the **Select Features** dialog box appears.

© 2025 Siemens

3. In the **Select Features** dialog box:

   a. Select the following options:

      - Choose **Extensions→Mechatronics Process Management** and select **EDA Server Support**.
      - Choose **Extensions→Mechatronics Process Management** and select **EMPS-Foundation**.
      - Choose **Extensions→Mechatronics Process Management** and select **EDA for Business Modeler IDE**.
      - Specify **Active Workspace** server extensions features for EDA. These are available in the **Features** panel in **Teamcenter Environment Manager (TEM),** under **Base Install→Active Workspace→Server Extensions→EDA Server Support for Active Workspace**.
      - Specify **Active Workspace** client extensions features for EDA. These are available in the **Features** panel in **Teamcenter Environment Manager (TEM),** under **Base Install→Active Workspace→Client→Electronic Design Automation for Active Workspace**.

   b. In the **Installation Directory** box, type the location where you want to install Teamcenter (TC_ROOT).

   c. Click **Next**.

4. Enter information as needed in the subsequent panes.

5. In the **Confirm Selections** dialog box, click **Next**.

   The EDA server components are installed.

   The **Add/Remove Components** dialog box confirms if the installation is successful. If the installation is unsuccessful, click **Show Details** and proceed as needed.

6. Obtain the license file that includes EDA licensing and EDA library license (for library support) and install it on the server.

After adding the EDA server support to your Teamcenter environment, you must verify your installation by checking whether the EDA data types are added to Teamcenter.

Verify your installation

Verify the addition of the EDA data types to the server:

1. Run the Teamcenter rich client.

© 2025 Siemens

2. In My Teamcenter, choose **File→New→Item**.

   The **New Item** dialog box is displayed.

3. Verify that the EDA, EDAComp, EDASchem, and EDACCABase item types are added to the list of types you can create.

4. Select one of the EDA item types and create an instance.

5. Verify that the item instance is created.

Install BMIDE to configure derived data for EDA objects

You must install Business Modeler IDE to **configure derived data for EDA objects and to import the sample BMIDE EDA server support template**.

You can install BMIDE either as a stand-alone application or place the BMIDE plug-ins in an existing Eclipse environment.

For instructions on installing BMIDE, refer to the Install the Business Modeler IDE topic or the Add the Business Modeler IDE to an existing Eclipse SDK environment topic in the BMIDE for Data Model Design documentation on Support Center.

Install or update Dispatcher for setting up remote ECAD translation

Teamcenter viewers can only process neutral ECAD formats, and depending on the workflow of your environment, you may need to translate native ECAD PCB and schematic data to this neutral file format.

With ECAD translation, the design intermediate file from the Xpedition or PADS tool is converted into a neutral file format (*XFATF* for PCB designs and *XSCH* for schematic designs). The neutral file is saved as a viewable dataset in Teamcenter.

The following translation modes are supported for ECAD translation:

- Local translation where the viewable file is generated locally.
- Remote translation where the Teamcenter integration initiates a remote translation of the intermediate file into a viewable format.

You require Dispatcher for remote ECAD translations only. You can install Dispatcher as a standalone instance or in an existing Teamcenter environment. If Dispatcher is already installed in your existing Teamcenter environment, you must modify it to include the *PcbToFatF* ECAD translator.

For instructions on installing Dispatcher, refer to the Dispatcher — Deployment and Administration documentation on Support Center. After installing and setting up the dispatcher you must **enable the PcbToFatf translator**.

© 2025 Siemens

Setting up Teamcenter server for design management

Overview of setting up and configuring Teamcenter server for design management

Before you install and enable your design tools and perform client installations, you must perform the following server setup configurations that are required for the integrations to work properly for design management:

Configuration	Why is it required?
Populate the Teamcenter database with ECAD components	Any component that does not already exist in Teamcenter is automatically created when saving the BOM.
Modify the Active Workspace style sheet to include EDA derived data relation	To allow EDA derived datasets to be displayed as item revision attachments in Active Workspace, the **Awp0ItemRevisionSummary** style sheet must be modified.
Add EDA relations and EDA objects in the Active Workspace Relation Browser	To traverse EDA relations and show EDA objects in the **Relations** tab in Active Workspace, EDA information must be included in the Relation Browser configuration.
Map ECAD attributes with Teamcenter properties	Using mapping definition files, the ECAD part attributes can be stored in the Teamcenter database and displayed and modified both in the ECAD design tool and Teamcenter.
Configure derived data for ECAD design objects	Derived data contains information that is derived from an ECAD design and comprises derived items and datasets. Derived items represent parts, subassemblies, and tools. Derived datasets manage data files created by ECAD applications.

Optionally, to change the behavior of the how the Teamcenter and the ECAD tool integration works, you can modify the settings for other configurations as follows. These configurations are available OOTB with set default values:

Configuration	Why is it required?
Configure design variants	To design and develop a product with several *variations*, where each variation has different options or capabilities, you create design variants. Creating design variants avoids the need to create a unique version of the design for each variation. A variant uses the same base design, but the PCB assembly is loaded with the set

© 2025 Siemens

Configuration	Why is it required?
	of components specified by the variation. You can then select a variation when generating the design's manufacturing output (BOM, Pick and Place (P&P), assembly drawings), which will in turn determine how the product is assembled.
Manage design files in local storage	You can configure the default behavior of removing design files in the local storage option after you check in, check out, revise, or perform a save as operation on the design files. The **Remove working files from local storage** check box allows you to remove associated design files after you perform check in, check out, revise, or save as action on the design.
Apply Teamcenter user's permission on the network shared ECAD design folder	During any Teamcenter EDA operation on a design in a network shared folder, the ECAD users accessing the design will have viewing or modifying rights on that shared folder based on the access permissions set for that user in Teamcenter.
Allow ECAD designers to assign a Teamcenter project from the EDA client	While saving a new or an existing EDA design from the EDA client to Teamcenter, the ECAD designers can assign or remove a Teamcenter project for that design. The projects assigned to Printed Circuit Assembly (PCA) objects are propagated to all *ItemRevisions* and *Datasets* associated with the PCA object based on the Teamcenter propagation rule defined for that object. You can also modify the OOTB propagation rules as per your requirements.
Set configurations based on the ECAD applications used	You can configure which preference values should be used for a specific preference based on the connectors configured for your ECAD design tool. This configuration allows all EDA preferences to be overridden by application-specific preferences, if set.
Configure EDA to access Active Workspace for collaboration and issue management	You must enable EDA to access Active Workspace for performing tasks related to the collaboration and issue management functionalities. The tasks can be completed using the **Design Info** and **Inbox** commands from Active Workspace.
Allow users to view the PCB board requirements in Active Workspace	By default, ECAD designers can view the PCB board requirements specifications related to the design revisions in the **Requirements** tab in

© 2025 Siemens

Configuration	Why is it required?
	Active Workspace for the **EDACCABase** and **EDASchem** object types, by default. Users can configure the **Requirements** tab for other item revisions.
Customize the acronyms used for ECAD design objects in Teamcenter	If your ECAD designers use acronyms different from the default acronyms provided, the Teamcenter configurator can override the defaults by creating custom acronyms used for ECAD designs in Teamcenter.
Configure the EDA client to use the Teamcenter EDA panels instead of Active Workspace	By default, the EDA client uses the Active Workspace panels for user actions such as open and save. You can use Teamcenter EDA panels instead, if you prefer to or in case you do not have Active Workspace installed.

Populate the Teamcenter database with ECAD components

Many companies control the ECAD component library items that can be used to create BOMs by checking in the approved components for use to the database. Thereafter, you can create BOMs that use these components.

You can configure how to:

- **Control ECAD component library items for BOM creation.**

  To control component library items for BOM creation, when EDA is first installed, the **EDA_CheckComponentExistence** is set to **1** (true). With this setting, you cannot save design BOMs unless all the necessary components are already in Teamcenter.

- **Populate the database with ECAD components before the designer can save designs and before creating BOMs in Teamcenter using EDA.**

  You can initially populate the database with components by changing the **EDA_CheckComponentExistence** preference to **0** (false). If this preference is set to **0**, any component that does not already exist in Teamcenter is automatically created when saving the BOM.

  > Caution:
  >
  > Do not set this preference to **0** in a production environment.

  1. Set the **EDA_CheckComponentExistence** preference:

     a. Run the rich client on the Teamcenter server and log on as an administrator.

© 2025 Siemens

b. In the My Teamcenter application, choose **Edit→Options**.

c. At the bottom of the **Options** dialog box, click **Filters**.

d. In the **Preferences** dialog box, search for the **EDA_CheckComponentExistence** preference.

e. Click **Edit** and change the value of the preference from **1** (true) to **0** (false).

f. Click **Save** to apply the changes.

2. Locate or create designs that use the approved library components. Use the EDA **Save As** command to save the designs to the Teamcenter server and create BOMs. When the BOMs are created, the components are automatically added to the database.

   For information about using the Save As command, see *Saving designs to Teamcenter* in the *Xpedition and PADS Design Management* documentation.

3. Reset the **EDA_CheckComponentExistence** preference to **1** (true) if you want to restrict users to using only the approved (and checked-in) components. This is the recommended setting in production environments.

   However, if you wish to allow new components to be automatically placed in the database when users create BOMs, leave the **EDA_CheckComponentExistence** preference as is, set to **0**.

   This action bypasses business processes for component evaluation and approval.

Add EDA relations and EDA objects in Active Workspace Relation Browser

To traverse EDA relations and show EDA objects from the **Relations** tab in Active Workspace, EDA information must be included in the Relation Browser configuration. To do this, you must update the **RelationBrowserConf** dataset in the **RB_UIConfigure.xml** file in the Teamcenter server configuration.

To enable the **Relations** tab to show relations between Teamcenter objects and objects in a remote application, do the following:

1. In the Teamcenter rich client, search for the dataset **RelationB***.

2. Right-click the **RelationBrowserConf** dataset and choose **Check-InOut** > **Check-Out**.

3. Right-click **RelationBrowserConf** dataset and choose **Named References**.

4. In the **Named References** dialog box, select **RB_UIConfigure.xml** and click **Download**.

5. Modify the **RelationBrowserConf** dataset in the locally downloaded **RB_UIConfigure.xml** file to add EDA relations and EDA objects as follows.

© 2025 Siemens

```
=================================================================
==
            Filename: RB_UIConfigure.xml

            File used to configure control panel of relation browser.
=================================================================
==

 -->
        <RelationBrowser>
          <OptionPanel>
            <view name="General">
              <ruleName>GenericRule</ruleName>
              <defaultLayout>IncrementalHierarchic</defaultLayout>
              <group name="relations">

                <filter name="EDA" parameterSet="EDARelations"
color="(204,51,0)"/>
              </group>
              <group name="objects">

               <filter name="EDA" parameterSet="EDAObjects"
color="(204,51,0)"/>
               <filter name="Other" parameterSet="WorkspaceObject"
color="(238,236,225)"/>
              </group>
            </view>
            <view name="System">

               <parameterSet name="EDAObjects">
                  <clipsfacts>
                    <fact>EDACCABase</fact>
                    <fact>EDACCABaseRevision</fact>
                    <fact>EDASchem</fact>
                    <fact>EDASchem Revision</fact>
                  </clipsfacts>
               </parameterSet>
                 <parameterSet name="EDARelations">
                   <clipsfacts>
                     <fact>source=GRM,key=EDA,relationType-
=EDAHasDerivedDataset,targetDirection=forward,inputTypes=ItemRevision
</fact>
                     <fact>source=GRM,key=EDA,relationType-
=EDAHasDerivedItem,targetDirection=forward,inputTypes=ItemRevision</
fact>
                     <fact>source=GRM,key=EDA,relationType-
=EDAHasPWB,targetDirection=forward,inputTypes=ItemRevision</fact>
```


© 2025 Siemens

```
                        <fact>source=GRM,key=EDA,relationType-
=EDAHasSchematic,targetDirection=forward,inputTypes=ItemRevision</
fact>
                        <fact>source=GRM,key=EDA,relationType-
=EDAHasSchematic,targetDirection=backward,inputTypes=ItemRevision</
fact>
                        <fact>source=GRM,key=EDA,relationType-
=EDAHasSublayout,targetDirection=forward,inputTypes=Dataset,outputTyp
es=Dataset</fact>
                        <fact>source=GRM,key=EDA,relationType-
=EDAHasVariant,targetDirection=forward,inputTypes=ItemRevision</fact>
                        </clipsfacts>                      </
parameterSet>

            </view>
        </RelationBrowser>
```

6. Delete the original **RB_UIConfigure.xml** file from the **Named References** dialog box and import the new **RB_UIConfigure.xml** file in Teamcenter.

7. Click **Close**.

8. Check in the **RelationBrowserConf** dataset.

Map ECAD attributes with Teamcenter properties

How are ECAD attributes mapped with Teamcenter properties

Using mapping definition files, the ECAD part attributes can be stored in the Teamcenter database and displayed and modified both in the ECAD design tool and Teamcenter.

For the ECAD parts created in Xpedition or PADS, the ECAD design attributes can be mapped to Teamcenter properties while saving the files in Teamcenter. When the designs are saved, the ECAD attributes are added to the corresponding Teamcenter properties. When the designs are opened from the Teamcenter menu in the Xpedition or PADS tool, the properties are mapped back to the corresponding attributes in the ECAD tool.

For the ECAD design tool, choose entries for design, project, and variant attributes such that they apply to the data types that represent the corresponding objects. Mapping definitions for variant objects must be associated with the specific object type that represents the variant BOM.

For details on how to map ECAD attributes, see the Mapping attributes section in the *Teamcenter Integration for NX* documentation.

To understand the correct syntax and symantics for mapping attributes to standard and custom item types, see the Examples of Syntax and Symantics for Mapping Attributes topic in the *Teamcenter Integration for NX* documentation.

© 2025 Siemens

Customize ECAD attribute mapping

You can customize the ECAD attribute mappings in Teamcenter by exporting the attribute mappings to a file, editing the file, and importing the file back into Teamcenter.

1. Open a Teamcenter command prompt.

2. Change to the *bin* directory within TC_ROOT (TC_ROOT is the installed location of Teamcenter).

3. Run the **export_attr_mappings** command to export the mappings to a file, for example:

   export_attr_mappings –file=*test_attribute_mapping* **–u=***Tc-admin-user* **–p=***password*

4. Edit the mappings file to add EDA tool attributes. The format is **attr_from_ECADtool :attr_from_Tc /description="desc"**, where:

   - **attr_from_ECADtool** is the name of the attribute from the ECAD library tool.

   - **attr_from_Tc** is the attribute from Teamcenter.

   - **description** is the information used to describe this attribute.

   For example, adding the following text to the file defines mappings for your Siemens-EDA ECAD tools:

```
{ Dataset type="EDADesMentor"
mgc_type : IM.user_data_1 /description="mgc_type"
design_name : IM.user_data_2 /description="design_name"
}

{ Dataset type="EDAComMentorBS"
GEOM : ItemRevision.ICS(-5488) /description="GEOM"
}

{ Dataset type="EDAComPADS”
Part Number: ItemRevision.ICS(-5485)/description=" Part Number"
Description : ItemRevision.ICS(-5486)/description="Description"
}

{ Dataset type="EDAComExpedition”
Value : ItemRevision.ICS(-6211) /description="Value"}
{Item type=EDAComPart"
Cost: Item.GRM(IMAN_master_form).object_desc /description="Cost"
}

{ Dataset type=" Epl0EDXPackage”
"Tolerance" : ItemRevision.GRM(IMAN_master_form).object_desc /
```


© 2025 Siemens

```
    description="Tolerance"
    }
```

5. Import the edited mapping file back into Teamcenter by running the **import_attr_mappings** command, for example:

   import_attr_mappings –file=*test_attribute_mapping* **–u=***Tc-admin-user* **–p=***password*

The attribute mapping import and export process uses the PLM XML import/export mechanism. For more information, see the Tc XML and PLM XML Configuration for Data Import and Export documentation.

For the attribute mappings to work correctly, you must also perform the **client configurations for mapping RDN attributes** on your EDA client workstation.

Sample ECAD design attribute mapping definitions

The below example defines dataset attribute mappings and variant attribute mappings. In this mapping definition, the `Item` level attribute mapping definition defines the ECAD variant attribute mappings and it is part of the dataset level attribute mapping definition.

ECAD variant mapping for Xpedition tool:

```
{ Dataset type="EDADesExpBrd"
ORGNAME : Item.GRM(IMAN_master_form,Item Master).user_data_1
ORGADDR1 : Item.GRM(IMAN_master_form,Item Master).user_data_2
"DES_NAME" : ItemRevision.object_name
"DES_DESC" : ItemRevision.object_desc
Doc : Item.GRM(IMAN_master_form,Item Master).user_data_3

    { Item type="Item"
        "variant_def_list_tc_description" : ItemRevision.object_desc /
description="Variant Item Revision Description"
    }
}
```

In above example, the attribute mappings are defined with respect to the dataset type **EDADesExpBrd**. Using this dataset attribute mapping functionality, it is possible to map an ECAD design attribute to a property with the Teamcenter object type. In this example, the ECAD attribute **ORGNAME** is mapped to **user_data_1** property of the item master form and **ORGADDR1** is mapped to **user_data_2** property of the item master form in Teamcenter.

Similarly, the ECAD attribute **DES_NAME** is mapped to **object_name** property of **ItemRevision** and **DES_DESC** is mapped to **object_desc** property of **ItemRevision** in Teamcenter.

© 2025 Siemens

In the last line of above example, the attribute mappings are defined for the item type **Item**. However, these mappings are applicable only to the dataset type **EDADesExpBrd**. These mappings are not applicable for other dataset types. To define the attribute mappings between the ECAD variant attributes and the Teamcenter variant object properties, the attribute mappings must be defined on the **Variant Item** type. In above example, the ECAD variant attribute **variant_def_list_tc_description** is mapped to **object_desc** property on **ItemRevision** in Teamcenter.

Configure derived data for ECAD design objects

Understanding how derived data works

Derived data contains information that is derived from an ECAD design and comprises derived items and datasets. Derived items represent parts, subassemblies, and tools. Derived datasets manage data files created by ECAD applications.

You can configure how derived data is created in Teamcenter EDA by using the **EDA Derived Data** folder in Business Modeler IDE, creating an EDA derived data configuration, and configuring how derived data files are named in Teamcenter. For example, a configuration can specify that when a schematic design is saved in Teamcenter EDA, a schematic drawing can be automatically generated from the schematic design and saved along with the schematic item

EDA business objects define the different types of derived data you can generate. To locate EDA business objects, use the **Find** button in the BMIDE view to search for all business objects containing the **EDA** string.

The following item types are children of the EDA business object:

Item Types	Description
EDACCABase	Represents the common electrical CAD (ECAD) design data that is shared between variant circuit card assemblies (CCAs). It is used only for multiple CCA representations.
EDAComp	Represents electrical components contained in the CCA bill of materials (BOM).
EDASchem	Represents the electrical schematic item.

The following relationships are children of the ImanRelation business object:

ImanRelation	Description
EDAHasDerivedDataset	Identifies the associated dataset as a derived dataset.
EDAHasDerivedItem	Identifies the associated item as a derived item.

© 2025 Siemens

How to use the EDA derived data editor to configure derived data in Teamcenter

Use the EDA Derived Data editor in the BMIDE to work with derived data configurations used by the Teamcenter EDA application. Teamcenter EDA integrates Teamcenter with ECAD applications that are used to design electronic components, such as circuit boards.

1. To access the **EDA Derived Data** editor, open the **Extensions→EDA Derived Data** folder, right-click an EDA derived data object, and choose **Open**.

2. To add an item to the EDA derived data object, click the **Add** button next to the **Configure Items** table.

3. To add a dataset to the EDA derived data object, click the **Add** button next to the **Configure Dataset** table.

How to create an EDA derived data configuration

You can configure how derived data is created in Teamcenter by using the **EDA Derived Data** folder in Business Modeler IDE. After creating a derived data configuration, you can specify the name of the configuration in the **EDA_DerivedDataConfigDefault** preference.

1. In BMIDE, open the **Extensions** folder.

2. In the **Extensions** folder, right-click **EDA Derived Data** and choose **New EDA Derived Data**.

   The **New EDA Derived Data** wizard is displayed.

3. In the **EDA Derived Data** dialog box, enter the following information and click **Next**:

Field	Description
Name	Specify a name you want to assign to the new derived data configuration. This is the name used in the **EDA_DerivedDataConfigDefault** preference.
Description	Type a description for the new configuration.

   The **EDA Derived Data Configuration** dialog box in the wizard is displayed.

4. In the **EDA Derived Data Configuration** dialog box, set up how all EDA item and dataset types are to be handled for all contexts.

   a. Click the **Add** button next to the **Configure Items** table.

      The **Add/Edit EDA Derived Item Configuration** dialog box is displayed.

© 2025 Siemens

In the **Add/Edit EDA Derived Item Configuration** dialog box, configure the derived items to be generated. For example, create separate rows for contexts such as schematic, PCB, simulation, and so on, including variations based on the what the parent is, such as **Schematic**, **CCA**, and **CCAVariant**. In this way, you set up how derived data is generated for all combinations of items.

In the **Add/Edit EDA Derived Item Configuration** dialog box, enter the following information and click **Finish**:

Field	Description
Name	Type the name that you want to assign to the derived item configuration. This is the name displayed to the user on the Teamcenter EDA **Derived Item** dialog box during save operations.
Context	**Browse** to select a specific Teamcenter EDA application context from the contexts listed below. When users in Teamcenter EDA save derived data for the following specified data types, derived items are generated according to this configuration. • all • pcb • pcb/simulation • schematic • schematic/pcb • schematic/simulation • simulation For PWB, if you have derived items and derived datasets defined for both schematic and PCB, you must set the preference **EDA_DefinePWBWith** to **schematicANDpcb** so that the PWB and derived data can be exposed and modified in both schematic and PCB tools. By default, the preference value is set to **Schematic**.
Prefix	(Optional) Type a file name string to be attached to the beginning of the parent item ID to distinguish it as being generated by this configuration. The resulting string, including the prefix and postfix, is used in the derived item user interface in Teamcenter

© 2025 Siemens

Field	Description
	EDA as the initial value for the **Derived Item ID** box and **Name** box. This can be overridden by the user.
Postfix	(Optional) Type a file name string to be attached to the end of the item ID to distinguish it as being generated by this configuration. The resulting string, including the prefix and postfix, is used in the derived item user interface in Teamcenter EDA as the initial value for the **Derived Item ID** box and **Name** box. This can be overridden by the user.
EDA Parent	**Browse** to select the derived parent EDA business object to which the derived item is related. (Teamcenter EDA does not support attaching derived items under other derived items). • **CCA** represents a circuit card assembly (CCA). • **CCABase** represents the common design data that is shared between variant circuit card assemblies (CCAs). It is used only for multiple CCA representations. • **CCAVariant** represents the variant design data for a circuit card assembly (CCA). This is the data that is used on top of the CCABase business object. • **PWB** represents a printed wire board (PWB). A PWB is the product of a schematic design and printed circuit board (PCB) layout design and holds all the printed wire board production data created by those designs. • **Schematic** represents the electrical schematic item.
Relation	**Browse** to select the relationship between the derived item and the parent item revision. The **EDAHasDerivedItem** business object and its children are displayed in the selection dialog box.
Item Business Object	**Browse** to select the business object type name for the derived item, for example, EDA.
Add to Bom	Select this option to add the derived item to the bill of materials of the Teamcenter EDA parent. This check box is disabled if the Teamcenter EDA parent is a schematic or a PWB.

The derived item configuration is added to the **Configure Items** table.

© 2025 Siemens

b. Click the **Add** button to the right of the **Configure Dataset** table.

The **Add/Edit EDA Derived Dataset Configuration** dialog box is displayed.

In the **Add/Edit EDA Derived Dataset Configuration** dialog box configure the derived datasets to be generated. For example, create separate rows for contexts such as schematic, PCB, and simulation, including variations based on the what the parent is, such as **Schematic**, **CCA**, and **CCAVariant**. In this way, you set up how derived data is generated for all combinations of datasets.

In the **Add/Edit EDA Derived Dataset Configuration** dialog box, enter the following information:

Field	Description
Name	Type the name that you want to assign to the derived dataset configuration. This is the name displayed to the user on the Teamcenter EDA **Derived Dataset** dialog box during save operations.
Context	**Browse** to select a specific Teamcenter EDA application context from the contexts listed below. When users in Teamcenter EDA save derived data for the following specified data types, derived datasets are generated according to this configuration. • all • pcb • pcb/simulation • schematic • schematic/pcb • schematic/simulation • simulation For PWB, if you have derived items and derived datasets defined for both schematic and PCB, you must set the preference **EDA_DefinePWBWith** to **schematicANDpcb** so that the PWB and derived data can be exposed and modified in both schematic and PCB tools. By default, the preference value is set to **Schematic**.

© 2025 Siemens

Field	Description
Prefix	(Optional) Type a file name string to be attached to the beginning of the parent item ID to distinguish it as being generated by this configuration. The resulting string, including the prefix and postfix, is used in the derived dataset user interface in Teamcenter EDA as the initial value for the **Derived Item ID** box and **Name** box. This can be overridden by the user.
Postfix	(Optional) Type a file name string to be attached to the end of the item ID to distinguish it as being generated by this configuration. The resulting string, including the prefix and postfix, is used in the derived dataset user interface in Teamcenter EDA as the initial value for the **Derived Item ID** box and **Name** box. This can be overridden by the user.
EDA Parent	**Browse** to select the derived parent EDA business object to which the derived dataset is related. In addition to the following item types, the list also includes item configurations you already created. • **CCA** represents a circuit card assembly (CCA). • **CCABase** represents the common design data that is shared between variant circuit card assemblies (CCAs). It is used only for multiple CCA representations. • **CCAVariant** represents the variant design data for a circuit card assembly (CCA). This is the data that is used on top of the CCABase business object. • **PWB** represents a printed wire board (PWB). A PWB is the product of a schematic design and printed circuit board (PCB) layout design and holds all the printed wire board production data created by those designs. • **Schematic** represents the electrical schematic item.
Relation	**Browse** to select the relationship between the derived dataset and the parent item revision. The **EDAHasDerivedDataset** business object and its children are displayed in the selection dialog box.

© 2025 Siemens

Field	Description
Dataset Business Object	**Browse** to select the parent dataset business object type in Teamcenter to represent the derived item, for example, PDF.
Dataset Reference	**Browse** to select the kind of file reference to use for the derived dataset. If the derived data instance comprises more than one file, this field must either be specified as a **ZIPFILE** type or must be specified using a separate derived dataset configuration entry with the same derived data name.
Pathname	Type the path where the derived dataset is to be saved on the user's machine. Path names are evaluated at run time and must be the fully qualified path of the dataset that is to be saved. Path names can be explicitly specified (for example, *D:\EDA\Datasets\readme.txt*) or formed using the variables or file name filters. Derived datasets can contain multiple files. Path names are case sensitive, and the directory delimiters of / or \ are used interchangeably.
Callback Name	Type the EDA callback name to execute. This name is used to identify the configured callback in the EDA configuration file to determine what script to execute. The script is responsible for creating or placing the corresponding derived files to be uploaded as specified by the configured source path name.

The derived item configuration is added to the **Configure Dataset** table.

c. Click **Finish**.

The derived data configuration is added under the **EDA Derived Data** folder

5. To save the changes to the data model, choose **BMIDE→Save Data Model**, or click the **Save Data Model** button on the main toolbar.

6. Deploy your changes to the test server. Choose **BMIDE→Deploy Template** on the menu bar, or select the project and click the **Deploy Template** button on the main toolbar.

7. In the rich client, set the **EDA_DerivedDataConfigDefault** preference to point to the EDA derived data configuration you just created.

8. Choose **Edit→Options**, click the **Search** link at the bottom of the **Options** dialog box, locate the **EDA_DerivedDataConfigDefault** preference, and change its value to the new configuration.

© 2025 Siemens

There may be multiple configurations created in the Business Modeler IDE, but an administrator can point to only one of them through this preference.

9. After deployment, test your new configuration in Teamcenter EDA.

   For example, in your ECAD design tool, choose **Teamcenter→Save Derived Data**. (You can also select the **Generate Derived Data** check box in the **Save As**, **Save**, or **Check In** dialog box.)

   To verify that the derived data is generated, in Teamcenter, expand the item that contains the derived data (for example, a CCA item). The derived dataset entries appear as you expand the tree structure. To see the contents of the derived dataset, right-click the dataset and choose **Named References**. A dialog box appears that shows the files that are contained in the derived dataset.

Configure how derived data files are named in Teamcenter

Update the value of the **EDA_UseDerivedDataSubDirs** preference based on how you want the derived data files to be named in Teamcenter. The values are as follows:

- **True**

  The derived dataset name is the same as the Derived Dataset Configuration Name.

- **False**

  The derived dataset name consists of the parent item ID and an underscore (character) followed by the Derived Dataset Configuration Name.

Using wildcards in the derived data configuration path names

You can specify wildcard characters in the derived data configuration path name as follows:

Wildcard	Definition	Example
Asterisk (*)	Matches any number of characters including none.	**$DESIGN**.doc** Selects all **.doc** files from the child folder under the **$DESIGN** folder. **$DESIGN**PWB*.doc** Selects all **.doc** files with **PWB** in their file names from the child folders under the **$DESIGN** folder. **$DESIGN\PWB**.doc** Selects all **.doc** files that start with **PWB** in their file names from the child folder under the **$DESIGN** folder.

© 2025 Siemens

Wildcard	Definition	Example
Two asterisks (**)	Works like one asterisk (*) but crosses directory boundaries. This syntax is generally used to match complete paths.	**$DESIGN**\MyFile.doc** Selects the **Myfile.doc** file from all child folders under the **$DESIGN** folder.
Question mark (**?**)	Matches exactly one character.	**$DESIGN\Help\Help?.doc** Selects files with one character after their file name like **Help1.doc**, **Help2.doc** and **HelpA1.doc** but not **HelpMe.doc**.
Braces (**{ }**)	Specifies a collection of sub patterns.	**$DESIGN\Help{Me,You}*.pdf** Selects all **.pdf** files from either **HelpMe** or **HelpYou** folder.
Square brackets (**[ ]**)	Conveys a set of single characters or when a hyphen (-) is used a range of characters.	**$DESIGN\Help[123]*.doc** or **$DESIGN\Help[1-3]*.doc** Selects all **.doc** files any one of the folders **Help1**, **Help2**, **Help3**.

If multiple files from different folders match during save of the derived dataset, no files are saved for the derived dataset.

Example:

If your folder structure is like this:

$DESIGN\File1.doc

$DESIGN\Help1\File2.doc

$DESIGN\Help1\File3.pdf

$DESIGN\Help2\File4.doc

$DESIGN\Help2\File5.pdf

$DESIGN\Help2\File6.xls

The wildcards give the following results:

© 2025 Siemens

$DESIGN*.doc** will fail.

$DESIGN.pdf** will fail.

$DESIGN.xls** will succeed.

$DESIGN\Help1*.doc will succeed.

$DESIGN\File4.doc** will succeed.

$DESIGN*\File3.pdf will succeed.

You can also specify that the entire contents of a directory, including all sub-directories and files, are to be saved as a zip file. Specify the **DerivedDatasetConfiguration Dataset Reference** as **ZIPFILE**. Specify the **DerivedDatasetConfiguration Pathname** as a fully qualified directory path ending with the filename filter value ******. Each directory of the path must be an existing directory, no wildcards are allowed. **Derived DatasetConfigurationPathname** variables such as **$DESIGN** are supported.

Example:

$DESIGN/manufacturingData/**

Configure design variants

What are design variants?

To design and develop a product with several *variations* of that product, where each variation has different options or capabilities, you create design variants. Creating design variants avoids the need to create a unique version of the design for each variation. A variant uses the same base design, but the PCB assembly is loaded with the set of components specified by the variation. A variation may then be nominated when generating the design's manufacturing output (such as BOM, Pick and Place (P&P), and assembly drawings), which will in turn determine how the product is assembled.

When you are working with ECAD design files, you can create any number of variations of the same base design, where each component may be configured differently as follows:

- **Fitted**

  This is the default state of a component if the component is fitted. It does not have variations. When you create a new variant, all components default to the state *fitted*.

- **Not Fitted**

  If a component is set to *not fitted*, it still exists on the schematic and is transferred to the PCB, but it is removed from the appropriate output documentation, such as the BOM.

© 2025 Siemens

- **Fitted with Varied Parameters**

  A component can have variations of any of its parameters, as part of the variant definition process. Modifying the value of a parameter is a local variation, only affecting the output documentation. The original schematic and the component whose parameter is being varied are not modified in any way.

- **Alternate Part**

  It is also possible to select an entirely different component as an *alternate part*. Since the alternate part is a different component, only one component is presented on the compiled schematic sheet. The alternate part must also share the same set of pins placed in the same locations as the base part. This is an essential requirement to ensure that the connectivity remains valid when the design is compiled.

An ECAD design is saved as a non-variant design for the first time in Teamcenter. Later, the variants of the ECAD design are added or created in Teamcenter as required. There are two scenarios for how variants are added.

- In the first scenario, you save the ECAD design as a non-variant design in Teamcenter. When new variants of the designs are created in the ECAD tool, you update the non-variant design in Teamcenter with the variants.

- In the second scenario, you save the non-variant ECAD design as a variant design in Teamcenter by creating dummy variants. When the variant designs are created in the ECAD tool, you replace the dummy variants in Teamcenter with the actual variant designs.

  You can **configure how to save a nonvariant ECAD design to Teamcenter as a variant design** using the EDA variant object model.

Configure ECAD design variants in Teamcenter

You can save a nonvariant ECAD design to Teamcenter using the EDA variant object model. To configure the saving of new nonvariant ECAD designs in Teamcenter as variant designs:

1. Update the value of the **EDA_SaveAsForceVariant** preference to **true**. This value specifies that nonvariant ECAD designs not previously saved in Teamcenter are to be saved in Teamcenter, using the EDA variant object model.

2. Update the value of the **EDA_FutureVariantName** preference. The default value is **futureVariant**.

   When you save a nonvariant design to Teamcenter and the value of the **EDA_SaveAsForceVariant** is **true**, the value of the **EDA_FutureVariantName** preference is used as the **Variant Name** in the **PCA Variants** dialog box. The value of the **Variant Name** in the **PCA Variants** dialog box cannot be changed.

Nonvariant ECAD designs already saved to Teamcenter using the EDA nonvariant object model are converted to use the EDA variant object model by the existing nonvariant-to-variant conversion

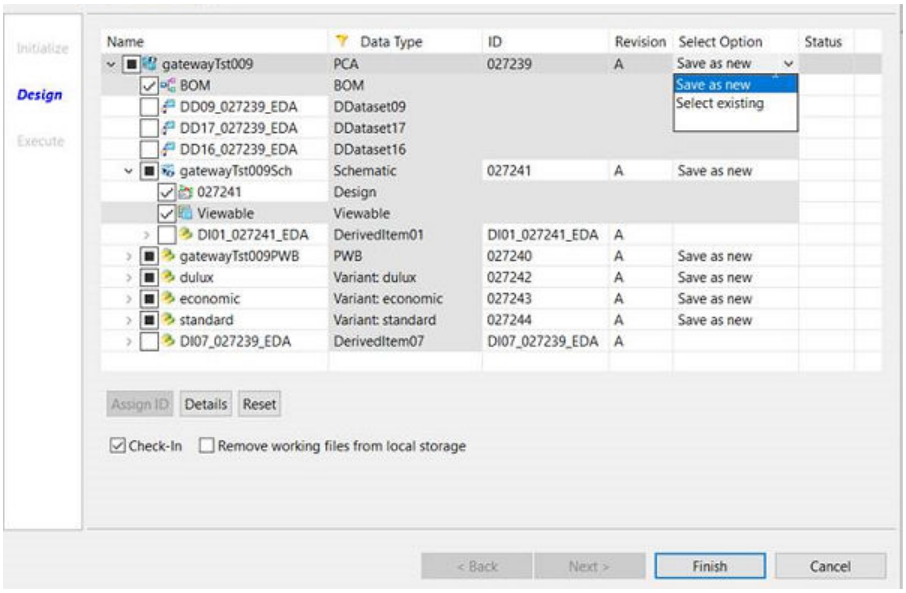
© 2025 Siemens

functionality. This conversion occurs automatically when an ECAD design, previously saved to Teamcenter as a nonvariant design, is saved to Teamcenter after the first variant is added to the ECAD design.

Base BOMs do not support this functionality.

Manage design files in local storage

You can configure the default behavior of removing design files in the local storage option after you check in, check out, revise, or perform a save as operation on the design files. The **Remove working files from local storage** check box allows you to remove associated design files after you check in, check out, revise, or perform a save as operation on the design.



You can configure the default status of this check box by setting the following values in the **EDA_RemoveWorkingFilesOptionDefault** preference:

- **unchecked**: The check box is cleared by default, and the user can select it.
- **checked**: The check box is selected by default, and the user can clear it.
- **Forceunchecked**: The check box is cleared, and the user cannot select it.
- **Forcechecked**: The check box is selected, and the user cannot clear it.

You can set the values of the **EDA_ValidateRemoveWorkingFiles** preference, to validate that the design files are not locked before performing the purge action. When this preference is set to **True**, the design files will be purged only if none of the design files are locked. However, if the preference value is set to **False**, only those design files that are not locked will be purged from the local storage. Locked files will not be purged and the users will be notified about the files not purged.

© 2025 Siemens

Apply Teamcenter user's permission on the network shared ECAD design folder

During any Teamcenter EDA operation on a design in a network shared folder, the ECAD users accessing the design will have viewing or modifying rights on that shared folder based on the access permissions set for that user in Teamcenter.

To apply the Teamcenter user's permissions on the ECAD design folder, the Teamcenter administrator must set the **EDA_EnableFolderPermissions** preference value to **true**.

To specify which Teamcenter user roles must have permission to access the ECAD design folder, the user roles must be set in the **FolderPermissions_ValidRolesForAccess** preference.

Allow ECAD designers to assign a Teamcenter project from the EDA client

While saving a new or an existing EDA design from the EDA client to Teamcenter, the ECAD designers can assign or remove a Teamcenter project for that design.

To make sure that the ECAD designer assigns a Teamcenter project while saving a design, you must set the **EDA_TCProjectRequired** preference to **true**. If this preference is set to **false**, the ECAD designer can save the design without assigning any project. The default value of the preference is **false**. If a default project is assigned to a user in Teamcenter, that project is assigned to the design by default. However, the user can assign additional projects while saving the design.

You must also assign the ECAD users to a project in Teamcenter to allow the users to select that project during the save operation. All projects associated with the ECAD users are displayed for selection to that user during the save operation.

A Teamcenter propagation rule is used for propagating security-related property values, such as project and license assignments, from a source business object to a destination business object. The projects assigned to Printed Circuit Assembly (PCA) objects are propagated to all **ItemRevisions** and **Datasets** associated with the PCA object based on the Teamcenter propagation rule defined for that object. You can also modify the OOTB propagation rules as per your requirements. For more information on how to modify an existing propagation rule, refer to the Configure your business data model in BMIDE on Support Center.

The OOTB propagation rules defined for EDA objects are as follows:

Direction	Source business object	Relation	Destination business object	Prop Group	Operation	Style	Details
Forward	**EDACCABaseRevision**	**IMAN_Rendering**	**Dataset**	Security Group I	All	Merge	Propagation rule between PCA **EDACCABas**

© 2025 Siemens

Direction	Source business object	Relation	Destination business object	Prop Group	Operation	Style	Details
							eRevision and viewable **Dataset** in a variant design.
Forward	**EDACCABaseRevision**	**structure_revisions**	**PSBOMViewRevision**	Security Group I	All	Merge	Propagation rule between PCA **EDACCABaseRevision** and **PSBOMViewRevision** in a variant Design.
Forward	**ItemRevision**	**EDAHasSchematic**	**EDASchemRevision**	Security Group I	All	Merge	Propagation rule between PCA **ItemRevision** and **EDASchemRevision**.
Forward	**EDASchemRevision**	**IMAN_Rendering**	**Dataset**	Security Group I	All	Merge	Propagation rule between **EDASchemRevision** and viewable **Dataset**.
Forward	**ItemRevision**	**EDAHasPWB**	**ItemRevision**	Security Group I	All	Merge	Propagation rule between PCA **ItemRevision** and **Printed Wiring Board**.
Forward	**EDACCABaseRevision**	**EDAHasVariant**	**ItemRevision**	Security Group I	All	Merge	Propagation rule between PCA **EDACCABaseRevision** and variant

© 2025 Siemens

Direction	Source business object	Relation	Destination business object	Prop Group	Operation	Style	Details
							ItemRevision.
Forward	**ItemRevision**	**EDAHasDerivedItem**	**ItemRevision**	Security Group I	All	Merge	Propagation rule between any **ItemRevision** and derived **ItemRevision** in a design.
Forward	**ItemRevision**	**EDAHasDerivedDataset**	**Dataset**	Security Group I	All	Merge	Propagation rule between any **ItemRevision** and derived **Dataset** in a design.

You can also modify the OOTB propagation rules to create custom rules for variant and non-variant designs.

Create a custom propagation rule for variant designs:

Direction	Source business object	Relation	Destination business object	Prop Group	Operation	Style	Details
Forward	**ItemRevision** for the variant item type defined in the **EDA_CCAVariantItemTypeDefault** preference	**structure_revisions**	**PSBOMViewRevision**	Security Group I	All	Merge	Propagation rule between the variant **ItemRevision** and **PSBOMViewRevision**.

© 2025 Siemens

Create a custom propagation rule for non-variant designs:

Direction	Source business object	Relation	Destination business object	Prop Group	Operation	Style	Details
Forward	**ItemRevision** for the PCA item type defined in the **EDA_CCAItemTypeDefault** preference	**IMAN_Rendering**	**Dataset**	Security Group I	All	Merge	Propagation rule between the PCA **ItemRevision** and viewable **Dataset** in a non-variant design.
Forward	**ItemRevision** for the PCA item type defined in the **EDA_CCAItemTypeDefault** preference	**structure_revisions**	**PSBOMViewRevision**	Security Group I	All	Merge	Propagation rule between the variant **ItemRevision** and **PSBOMViewRevision**.

Set configurations based on the ECAD applications used

You can configure which preference values should be used for a specific preference based on the connectors configured for your ECAD design tool. This configuration allows all EDA preferences to be overridden by application-specific preferences, if set. The application specific preferences can be created for all the EDA design preferences by adding a suffix to the OOTB preference name. The suffix is the name of the ECAD tool application as mentioned in the *%TCEDAECAD_ROOT%*****_usrdef.xml** file.

If the preference exists with the suffix of the connector name, this preference is used. Else, the preference without a suffix is used.

Example:

Consider that you are using two different connectors, and you want to set different values for the following preference, depending on which connector is currently being used. You can set different values for the same preference by adding the application name as a suffix to the preference name.

The OOTB preference name is **EDA_CCAVariantItemTypesAllowed**.

© 2025 Siemens

To set the preferences with different values for both the connectors, create two different instances of the preference with the connector names suffixed to the preference.

- Preference to be used if connector 1 is used:

  EDA_CCAVariantItemTypesAllowed_<connectorname_1>

- Preference to be used if connector 2 is used:

  EDA_CCAVariantItemTypesAllowed_<connectorname_2>

Configure EDA to access Active Workspace for collaboration and issue management

You must enable EDA to access Active Workspace for the collaboration and issue management functionalities. The collaboration and issue management tasks can be completed using the **Design Info** and **Inbox** commands from Active Workspace. To enable EDA to access Active Workspace:

- In Active Workspace, click **Create Change** and check if **Issue Report** is part of the available change types.

  If **Issue Report** is not available, perform following:

  - Create a Business Modeler IDE project, and add a condition with the following parameter values:

    - **Name**: **isIssueReportCreatable**
    - **Input parameters**: **Custom**
    - **Signature**: **isIssueReportCreatable (UserSession o)**
    - **Expression**: **Condition::isTrue()**

  - Deploy the Business Modeler IDE template.
  - In the Teamcenter rich client, update the **AWC_DispayableChangeTypes** preference with the value **IssueReportRevision**.

- Update the value of the **ActiveWorkspaceHosting.URL** preference with that of the Active Workspace URL.

  OR

  Create a preference named **ActiveWorkspaceHosting.EDA.URL** and update it with the value of the Active Workspace URL.

© 2025 Siemens

Allow users to view the PCB board requirements for ECAD objects in Active Workspace

ECAD designers can view the PCB board requirements specifications related to the design revisions in the **Requirements** tab in Active Workspace for the **EDACCABase** and **EDASchem** object types, by default.

To allow users to configure **Requirements** tab for other Item Revisions, user can include the following in the respective stylesheet.

```
<page titleKey="tc_xrt_Requirements"
visibleWhen="ActiveWorkspace:SubLocation
 != com.siemens.splm.client.occmgmt:OccurrenceManagementSubLocation">
        <htmlPanel declarativeKey="Eda0RequirementsTraceability" />
</page>
```

Additionally, you can specify the revision rule to configure the requirements BOM structure in the **Requirements** tab by setting the **EDA_AWRequirementBOMRevRule** preference. The default value of the revision rule is set to **Latest Working**.

Customize the acronyms used for ECAD design objects in Teamcenter

If your ECAD designers use acronyms different from the default acronyms provided, the Teamcenter a can override the defaults by creating custom acronyms used for ECAD designs in Teamcenter.

By default, EDA uses the following acronyms to refer to electronic design objects on the user interface:

- **PCA** for printed circuit assembly or **CCA** for circuit card assembly (in earlier versions of EDA).
- **PWB** for printed wire board.

If your ECAD designers use different acronyms, the Teamcenter administrator can override the defaults by creating custom text copies of the **TC_ROOT\lang\textserver***locale***\eda0_text_locale.xml** file. For each *custom-name***_text_locale.xml** file, locate the following lines in the file, and edit the display text as required:

```
<key id="Eda0_PCA">PCA</key>
<key id="Eda0_PCAToolTip">Printed Circuit Assembly</key>
<key id="Eda0_SCH">SCH</key>
<key id="Eda0_SCHToolTip">Schematic</key>
<key id="Eda0_PWB">PWB</key>
<key id="Eda0_PWBToolTip">Printed Wire Board</key>
<key id="Eda0_Design_Metrics_key">Design Metrics</key>
<key id="Eda0_Cad_Baselines_key">CAD Baselines</key>
<key id="Eda0_Edm_Viewer_key">EDM Viewer</key>
```


© 2025 Siemens

> Note:
>
> There is a separate locale file for each supported locale. The example shown is from the **en_US** locale file.

For more information about customizing textserver file text, see the Teamcenter Server Customization help on Support Center.

Configure the EDA client to use the Teamcenter EDA panels instead of Active Workspace

By default, the EDA uses the Active Workspace panels for user actions such as open and save. You can use Teamcenter EDA panels instead, if you prefer to or in case you do not have Active Workspace installed. This can be done by setting the following preferences:

EDA_Use_ActiveWorkspace

This preference specifies whether to display the Active Workspace panels instead of the older client panels in dialog boxes where available. The default value is set to **True**. To switch to Teamcenter EDA client panels, change the preference value to **False**.

The default value is valid only if one of the following preferences is also defined to point to an Active Workspace server.

- **ActiveWorkspaceHosting.URL**
- **ActiveWorkspaceHosting.EDA.URL**

If neither preference is defined, then the **EDA_Use_ActiveWorkspace** preference is ignored, and Active Workspace is not used. If either one of these preferences is defined, **ActiveWorkspaceHosting.URL** takes precedence and the default for value for **EDA_Use_ActiveWorkspace** is **True**.

EDA_Use_ActiveWorkspace_Open

This preference specifies whether to display Active Workspace panel in the **Open** dialog box instead of the Teamcenter EDA client panel. The default value is set to **True**. To switch to Teamcenter EDA client panels, change the preference value to **False**.

The preference **EDA_Use_ActiveWorkspace** must also be set to **True** to use this preference.

Managing and configuring Teamcenter preferences for design management

You can set system-wide preferences on the Teamcenter server that apply to all EDA users. To understand how preferences work and to understand how to access a list of all supported preferences,

© 2025 Siemens

refer to the Managing Preferences help on Support Center. All EDA preferences begin with the prefix **EDA**.

1. Run the rich client on the Teamcenter server and log on as an administrator.
2. In My Teamcenter, choose **Edit→Options**.
3. At the bottom of the **Options** dialog box, click **Index**.
4. In the **Preferences** dialog box, search for the preferences that begin with **EDA_**.
5. Change the preference value and click the **Modify** button to save the new value.

Installing the EDA client for design management on the ECAD workstation

Install the EDA client for design management

Use the **tem.bat** command from the Teamcenter software kit to launch Teamcenter EDA Environment Manager (TEM) to perform installation and maintenance operations of Teamcenter EDA on an ECAD workstation for design integration.

TEM is available with the Teamcenter installation or upgrade images. Contact your Teamcenter installation administrator to obtain the correct product distribution image.

> Caution:
>
> Before installing the EDA client on your ECAD workstation, refer to all the prerequisites and the information required to install the client in the **Planning your deployment** topic.

1. Ensure that the proper version of JRE is installed and the JRE_HOME environment variable (32-bit system) or the JRE64_HOME environment variable (64-bit system) is set.
2. Extract the Teamcenter software distribution image to a temporary location.
3. Specify the path to the Java Runtime Environment (JRE) by setting the **JRE_HOME** environment variable on the ECAD workstation.

   You can also specify the JRE path when you launch TEM from the command line using the **-jre** path argument.
4. Start TEM.

   a. Browse to the root directory of the extracted Teamcenter EDA software distribution image.

© 2025 Siemens

b. Right-click the **tem.bat** program icon and choose **Run as administrator**.

TEM launches and displays the **Installer Language** panel.

c. In the **Installer Language** panel, select a language and click **OK**.

TEM displays the **Welcome to Teamcenter EDA** panel.

Your language selection applies only to the TEM session and not the Teamcenter EDA installation.

5. In the **Welcome to Teamcenter EDA** panel, click **Install**.

6. In the **Configuration** panel, type the configuration ID and the description for the new Teamcenter EDA configuration.

   The configuration ID identifies your Teamcenter configuration when you maintain, upgrade, uninstall, or add features to the configuration. Installation log files are also named based on this ID.

7. Proceed to the **Features** panel and click **Next**.

8. Expand **EDA Client** and select the required options:

   - **ECAD Design Integrations**

     Select the design tool support you require. This integrates the design tool with the EDA client. However, a separate installation of the design tool is required.

     - **Mentor PADS**
     - **Mentor PADS Professional**
     - **Mentor Xpedition Designer and Layout**
     - **Teamcenter Gateway for EDA Design**
     - (Optional) **EDA Translation**

9. In the **Installation Directory** box, browse to a folder where the EDA client will be installed. For example, type **c:\EDA_client** for Windows.

   The client is installed to an **eda** subdirectory, for example, **c:\EDA_client\eda** (Windows). This directory is known as *TCEDAECAD_ROOT*.

10. Click **Next**.

© 2025 Siemens

11. In the **FCC Client Cache (FCC)** panel, select **Use new FCC** and click **Next**.

12. In the **FCC Parents** panel, type the **Host** name of the Teamcenter server in the **FCC Parents** box.

13. In the **4-tier Server Configurations** panel, type the IP address of the Teamcenter server in the **4-tier Servers** box. For example:

    http://*host***:***7001***/tc**

    Replace *host* with the server name.

14. In the **EDA Client** panel:

    a. Change the path if needed in the **Staging Location** box.

    You must enter the full path to the directory where the ECAD designs are saved on the client. This directory is the working directory for EDA.

    If a staging directory already exists, you can continue using the same directory or select another folder as the staging directory.

    > Caution:
    >
    > Do not use spaces in the cache or staging directory path. Some EDA functions do not work if there are spaces in these paths.

    b. Clear the **Will this configuration be shared by multiple users?** check box to create user environment variables after installation. By default, the check box is selected to create system environment variables.

    c. Click **Next**.

15. Review the features selected in **Confirmation** panel and click **Start**.

    The EDA client components are installed.

    The **Install** panel confirms whether the installation is successful. If the installation is successful, click **Close**. If the installation is unsuccessful, click **Show Details** and proceed as needed.

After the installation of EDA client is complete you must verify that the EDA client is installed correctly and **perform the configurations required to set up design management**.

Additionally, you must further **configure and customize your ECAD tool connector** as per your requirement.

© 2025 Siemens

Verify the EDA client installation

Verify the EDA client installation by checking the following:

1. Ensure that Teamcenter client communication system (TCCS) services point to the Teamcenter server with EDA server support.

2. The FMS **fcc.xml** file points to your Teamcenter server.

3. The **FMS_HOME** system environment variable is set and points to the location of the **fcc.xml** file.

4. The **TCEDAECAD_ROOT** user environment variable is set to your EDA directory (for example, **c:\EDA_client\eda** on Windows systems).

   The value of **TCEDAECAD_ROOT** is added to **PATH**.

5. The directory defined by the **TCEDAECAD_ROOT** user variable exists and is populated with files.

6. The relevant ECAD tool subdirectory exists (*<TCEDAECAD_ROOT>\eda*.

   If you are using PADS, verify that the **PADS** subdirectory exists. If you are using Xpedition, verify that the **Mentor_Expedition** subdirectory exists.

7. The **TCEDAClient.properties** file in the *TCEDAECAD_ROOT* directory contains the values you entered using the installation routine, for example:

   ```
   URL=http://hostname:port/tc
   StagingDir=staging-directory-location
   ```

   > Note:
   >
   > When you first run EDA, the staging directories are created from the settings in the **TCEDAClient.properties** file.
   >
   > The paths of the staging directories contain double back slashes (\\) on Windows systems. For example:
   >
   > *C:\\ecad\\EdaStaging*

Setting up the EDA client for design management

Configurations required to set up the EDA client for design management

Before configuring the EDA client for design management, you must first **enable the Teamcenter menu in the ECAD tool** user interface to access the Teamcenter menu within the design tool. This is a one-time process.

© 2025 Siemens

After integrating Teamcenter menus within the ECAD tool user interface, perform the following configurations on the EDA client for design management:

- **Mapping reference designator attributes for your ECAD tools**
- **Configuring cache and staging directory**
- **Modifying default user properties in the TCEDAClient.properties file**
- **Set a flatter folder structure for saving ECAD design**
- **Modifying the default logging configuration**

Integrate Teamcenter menus within the ECAD tool

Enable Teamcenter menu within the ECAD tool

To access the Teamcenter menu within the design tool, you must first enable the Teamcenter menu in the user interface, which is a one-time process.

Before you enable the Teamcenter menus within the ECAD tool, verify that:

- The Teamcenter server is operational.
- The Teamcenter rich client is installed and functional in a four-tier setup.
- The ECAD tool installation is complete.
- The TCEDAECAD_ROOT user environment variable is created by the installer and that it points to the directory where EDA is installed.
- The JAVA_HOME environment variable exists and points to a supported 32-bit or 64-bit JAVA installation.
- The proper Mentor WDIR environment variable is set and that the folder exists.

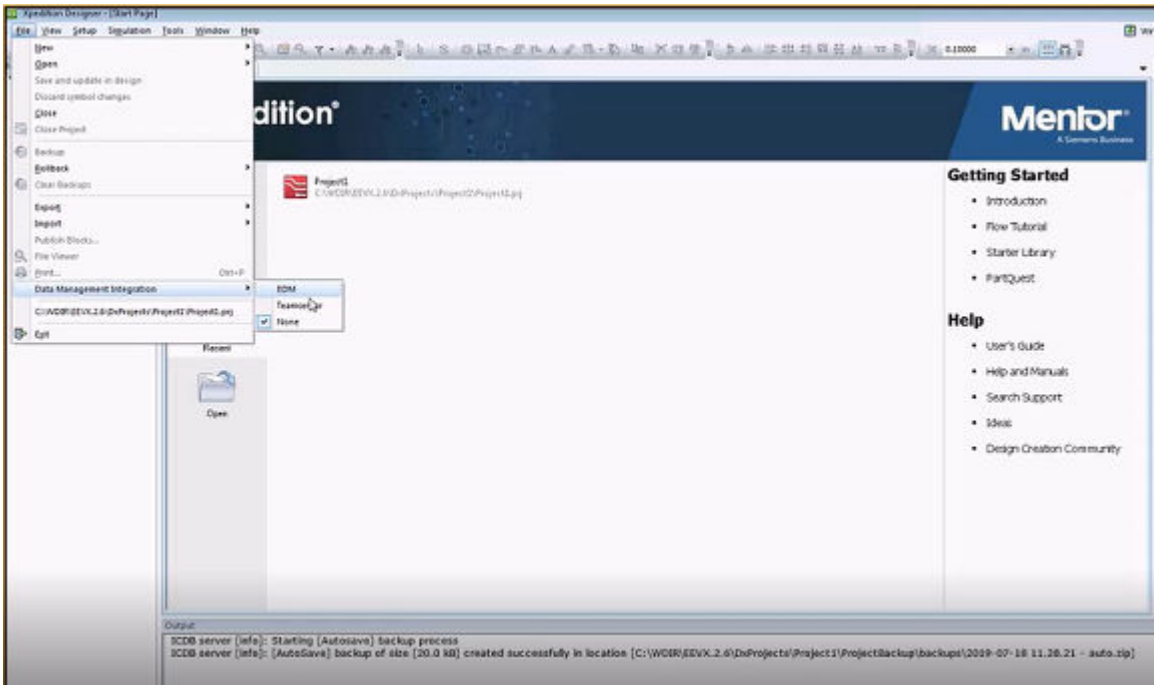
To integrate Teamcenter menu on your design tool perform the following tool specific steps:

To integrate Teamcenter menu within Xpedition

To enable Teamcenter panels within the Designer or the Layout tools:

1. Open the *Xpedition Designer* or *Xpedition Layout*.
2. Click **File→Data Management Integration→Teamcenter**.

© 2025 Siemens



3. In the **Teamcenter EDA Login** window, specify the Teamcenter **User ID** and **Server** details for integration.

   The Teamcenter user must have either the **Author** or the **Occasional User** license to manage ECAD design data in Teamcenter.

4. Click **Login**.

To integrate Teamcenter menu within PADS

> Note:
>
> PADS can be installed to create schematics using either *PADS Logic* or *PADS Designer*. This depends on which design flow is selected during PADS installation. The instructions below are for PADS logic support.
>
> Refer to the instructions explained below for PADS Professional flow integration with Teamcenter to set up *PADS Designer* for PADS.

1. Customize the PADS application by completing these steps. This is a one-time procedure.

   a. Create Windows shortcuts to the launching scripts located in *%TCEDAECAD_ROOT%* **\PADS\wnti\bin**.

© 2025 Siemens

> Tip:
>
> You can save these shortcuts on your desktop for easier access.

b. Launch PADS Logic and choose **Tools→Basic Scripts→Basic Scripts**.

c. Click **Load File** and browse to the **Teamcenter.bas** customization script located in the *%TCEDAECAD_ROOT%***\PADS\wnti\vb** directory.

d. Click **Open** and choose the **In Menu** command.

e. Click **Close**.

f. Launch PADS Layout and customize it in the similar way.

2. From PADS Logic and PADS Layout, choose **Tools→Basic Scripts→Teamcenter** to display the Teamcenter palette menu in PADS.

   The palette menu appears at the top of the PADS editing window. You can select any of the Teamcenter actions by choosing the menu commands.

   You can remove the palette menu by clicking **Close**.

To integrate Teamcenter menu within PADS Professional

To use the setup option, the following steps can be followed to enable the EDA menu items. This procedure must be followed in the cases where English is not both the local language and the displayed language.

Designer with PADS Design Flow setup recommended procedure:

1. Launch PADS Designer tool.

2. Click **Setup→Settings**.

3. Select **Run on Startup**.

4. Enter the full path for the vbs file from *%TCEDAECAD_ROOT%\psw\dxd\pswMenu.vbs*.

   This path must contain only the absolute path to the **vbs** file.

5. Click **OK**.

To bypass the use of the setup option and directly configure the pointers to the EDA menu items, the following steps can be followed:

© 2025 Siemens

PADS Designer setup alternate procedure

1. If WDIR does not contain a **scripts.ini file**, copy the **scripts.ini file** file from *%TCEDAECAD_ROOT%* **\psw\exp\wdir** to WDIR.

2. If WDIR already contains a scripts.ini file, add the following lines to the end of the file (or to the appropriate areas if the **ViewDraw** or **Expedition PCB** sections already exist within the **scripts.ini file** file). The new entries must be arranged in numerical order with no duplicate numbers within a section.

```
[ViewDraw]
Script#0=%TCEDAECAD_ROOT%\psw\dxd\pswMenu.vbs
[Expedition PCB]
Script#0=%TCEDAECAD_ROOT%\psw\exp\pswMenu.vbs
```

PADS Professional Layout tool does not provide a setup menu item to configure the path to the EDA menu items. The installed scripts.ini file must be directly edited. The following steps must be followed.

PADS Professional Layout setup procedure

1. Edit the scripts.ini file located within the installed PADS Professional installation. The file is found at *<PAD Professional installation>\SDD_HOME\standard\automation\startup* location.

2. Add the following lines to the end of the file (or to the appropriate areas if the **ViewDraw** or **Expedition PE** areas already exist). The new entries must be arranged in numerical order with no duplicate numbers within a section.

```
[ViewDraw]
Script#0=%TCEDAECAD_ROOT%\psw\dxd\pswMenu.vbs
[Expedition PE]
Script#2=%TCEDAECAD_ROOT%\psw\exp\pswMenu.vbs
```

> Note:
>
> The second section of the **scripts.ini file** file, the label is **Expedition PE** instead of **Expedition PCB**.
>
> Where both Xpedition and PADS Professional are installed, the **scripts.ini file** file can contain both **Expedition PE** and **Expedition PCB** sections.

Additional connector configurations and customizations for design management

You must configure and customize your ECAD tool connector as per your requirement. Some connectors have unique configuration files named **settings.ini**. All connector specific customizations and configurations as well as the information about using the **settings.ini** files is explained in **ECAD tool customizations for design management** section. The **settings.ini** file allows you to configure

© 2025 Siemens

connector behavior. Some connectors include customization files that need to be copied into folders within the installed ECAD tools. Information about these files is also explained in this section.

Verify the integration of ECAD tool with Teamcenter

You must verify that your ECAD tool is correctly integrated with Teamcenter and the Teamcenter menu is enabled within the ECAD tool.

Designer

1. Launch Designer.
2. Verify that **Teamcenter** button shows up in the menu bar.
3. Open a sample project.
4. Open a sample schematic.
5. Click **Teamcenter→Open**. This will launch the Teamcenter **Open** dialog box.
6. Dismiss the resulting Teamcenter **Open** dialog box.

Xpedition Layout

1. Launch Xpedition Layout.
2. Verify that **Teamcenter** button shows up in the menu bar.
3. Open a sample project.
4. Open a sample layout.
5. Click **Teamcenter→Open**. This will launch the Teamcenter **Open** dialog box.
6. Dismiss the resulting Teamcenter **Open** dialog box.

PADS Layout

1. Launch PADS Layout using the provided launching script.
2. Open the **Teamcenter** menu.
3. Verify that the Teamcenter floating menu appears and contains Teamcenter action buttons.
4. Open a sample layout.

© 2025 Siemens

5. Click **Teamcenter→Open**. This will launch the Teamcenter **Open** dialog box.

6. Dismiss the resulting Teamcenter **Open** dialog box.

PADS Designer

1. Launch PADS Designer.

2. Verify that **Teamcenter** button shows up in the menu bar.

3. Open a sample project.

4. Open a sample schematic.

5. Click **Teamcenter→Open**. This will launch the Teamcenter **Open** dialog box.

6. Dismiss the resulting Teamcenter **Open** dialog box.

PADS Professional Layout

1. Launch PADS Professional Layout.

2. Verify that **Teamcenter** button shows up in the menu bar.

3. Open a sample project.

4. Open a sample layout.

5. Click **Teamcenter→Open**. This will launch the Teamcenter **Open** dialog box.

6. Dismiss the resulting Teamcenter **Open** dialog box.

Mapping reference designator attributes for your ECAD tools

A reference designator identifies a component within a schematic design or on a printed circuit board. The reference designator usually consists of one or two letters followed by a number, for example *R13*, *C1002*. The number is sometimes followed by a letter, indicating that components are grouped or matched with each other, for example, *R17A*, *R17B*. IEEE 315 contains a list of *Class Designation Letters* to use for electrical and electronic assemblies. For example, the letter R is a reference prefix for the resistors of an assembly, C for capacitors, and so on.

In the EDA client definition file (***usrdef.xml**) you can find additional elements that are required for mapping the **ReferenceDesignator** element with the **PSOccurrence** attribute. This process is referred to as reference designator (RDN) mapping. One or more of these files are installed in the *TCEDAECAD_ROOT* folder. Identify the appropriate ***usrdef.xml** files in your installation and modify it to enable RDN attribute mapping for your ECAD authoring tools.

© 2025 Siemens

Tip:

Save a copy of each original file in case you have problems or must revert to an earlier version, without your modifications.

Caution:

Do not modify any other areas of these files.

The **RdnAttrMapDefs** subelement of the following sample ***edadef.xml** file contains additional subelements to enable and configure RDN attribute mapping. You must add this set of elements to your ***usrdef.xml** file in a specific order.

```
<?xml version="1.0" encoding="UTF-8"?>
<EDAClientDef
    application="mentorEDXDes"
    family="mentorEDX"
    className="com.teamcenter.edabase.DefaultEDAFactory"
    bomOptionModel="pcbOnly"
    enableVariants="true"
    supportsBaseBom="true"
    preserveSession="false"
    enableOpenRelatedDatasets="false"
    copyFilesInSaveAsCopy="false"
    allowVariantNameAsItemId="false"
    USASCIIRequired="true"
    enableDoNotKeep="false"
    allowUnsavedVariants="false"
    supportSpacesInStagingDir="true"
    enableEDAUIDs="false"
    forceCheckIn="true">

    <PrimaryDataType name="combined" datasetType="EDADesExpBrd">
        <Include>*.edx</Include>
        <Exclude>odb</Exclude>
        <Exclude>edif</Exclude>
    </PrimaryDataType>

    <IntermediateDataType name="schCad" datasetType="EDAGenSchem"/>
    <IntermediateDataType name="pcbCad" datasetType="EDAGenPCBCAD"/>

    <ViewableDataType name="xsch"  datasetType="SCHFATF"/>
    <ViewableDataType name="xfatf" datasetType="PCBFATF"/>

    <PreferenceDefs/>

    <RdnAttrMapDefs>
```


© 2025 Siemens

```
        <!-- component instance attribute mapping example
        <RdnAttrDesign cadAttrName="Foobar" tcAttrName="<BOMLine
property name or PS
            Occurrence Note property name>" isOccNote="<true or false
depending on the
            specified Tc property type>"/>
        -->
        <!-- component instance constant value mapping example
        <RdnAttrConst  tcAttrName="< BOMLine property name or PS
Occurrence Note
            property name>" isOccNote="<true or false depending on the
specified Tc
            property type>" value="value0"/>
        -->
    </RdnAttrMapDefs>

    <TranslatorDef type="xfatf" translator="odb"/>
    <TranslatorDef type="xsch"  translator="edif200"/>

</EDAClientDef>
```

The suggested order of the first-level subelements of the EDA client definition file is as shown below:

Example:

```
PrimaryDataType
RelatedDataType
IntermediateDataType
ViewableDataType
PartitionDataType
IssueAttachmentDataType
PreferenceDefs
CallbackDefs
RdnAttrMapDefs
TranslatorDef
LibraryDef
EDAGatewayDef
MigrationDef
SaveOptionDefs
OperationData
```

Point to consider while mapping the reference designator elements with PSOccurence attributes:

- The **RdnAttrMapDefs** subelement contains examples that are commented out. Siemens Digital Industries Software suggests that you leave the commented out subelement as examples and then copy them to add mappings. Add one subelement for each mapping, making modifications as needed.

© 2025 Siemens

- The **BOMLine** property name or **PSOccurrenceNote** property name (**tcAttrName**) must be the name of the Teamcenter **BOMLine** property to which the ECAD attribute value is mapped.
- The ECAD attribute name (**cadAttrName**) must be the name of an ECAD attribute that can be extracted from the ECAD component occurrence or RDN.
- The **RdnAttrDesign** subelement of the **RdnAttrMapDefs** element defines a mapping from an ECAD attribute name to a Teamcenter **PSOccurrence** property or **PSOccurrenceNote** property name.
- The Teamcenter property must have write access for all ECAD designers. Otherwise, a write access error is reported during BOM creation or update.
- The Boolean **(isOccNote)** value simply indicates that the Teamcenter property name is a **PSOccurrenceNote** property.
- You must add one **RdnAttrDesign** subelement for each ECAD occurrence attribute that you want to map to a **BOMLine** property.
- The **RdnAttrConst** subelement of the **RdnAttrMapDefs** element defines a Teamcenter **BOMLine** property or **PSOccurrenceNote** property that is set to the string specified by the value attribute. You may not need to define any mappings of this type.

Once you modify the ***usrdef.xml** file, and before you install it, drag it on to your browser. This verifies that you have not introduced any XML syntax errors into the file. If there are any errors, your browser displays an error message. Do not install the modified file until the browser verifies that it is clean, with no errors or warnings.

Configuring cache and staging directory

Understanding how cache and staging directory works

The cache and staging directories temporarily store data as the ECAD designs move between the design tool and Teamcenter. Occasionally, designers may need to refresh a design to synchronize the local copy of the design with the latest version in Teamcenter. Alternatively, they may need to clear the cache to remove all local copies of the designs that they are not working with. EDA uses a staging directory for the local storage of ECAD designs and the cache directory for storing the temporary data.

The EDA cache directory may be created under the staging directory, or it may be created under the user's home directory, depending on the configuration.

The EDA staging directory holds designs for checkin and checkout with Teamcenter. The first time you choose a command on the **Teamcenter** menu, the staging directory is created automatically. The staging directory path is based on the staging directory setting in the *TCEDAECAD_ROOT***TCEDAClient.properties** file. The current OS username may be added to that value as a subfolder and that path is written to a new user-specific **TCEDAClient.properties** file in the **USER_HOME** directory.

© 2025 Siemens

In the **TCEDAClient.properties** file, the environment variable **EDA_StagingDir** takes precedence over the **StagingDir** property. The **EDA_StagingDir** environment variable must be defined as a full path and not a relative path to an existing folder that the user has access to or to a folder that the user can create.

How to configure the checkout of files in EDA

When the designer opens an ECAD design object for checkout, the EDA file services tools must know both the source and the destination for the action. For a checkout, the source is where the data is stored within the Teamcenter file system and is part of the metadata maintained by Teamcenter. The destination is where the designer wants to store the date and therefore varies.

At a minimum, each workstation has a unique file system. For the EDA file system tools, the designer must establish a fixed location known to EDA for these files. This fixed location is defined at the time of EDA installation. The location is stored within a settings file that all EDA tools can access. During each installation, you must specify where the staging directory is defined for that user and host.

Once this is done, the resulting action for a checkout from EDA is that the data source is a Teamcenter server, and the data destination is specified by the user's staging directory setting in an EDA settings file. A different user can also access the same data items. In such a case, the destination is the accessing user's specified staging directory.

How to configure the saving of files in EDA

When the designer saves an ECAD object, the source can be any location in the user's file system, but the destination is always an EDA-provided location within the Teamcenter server. In practice, however, different scenarios are possible. In the case of one ECAD product, a design may be stored within a single file. In another ECAD product, the design data may be stored in multiple files. Many designers may wish to include supplemental data included with the design files. For example, the designer may want to add a *readme* file or add a *specification* document.

To allow designers to save additional files with the design data, EDA stores such files in a separate container. Any files or file objects that are in this container are captured when the ECAD tool executes an EDA **Save**, **Save As** or **Revise** action.

Capturing supplemental or hidden ECAD files

Some ECAD products use hidden or invisible files to hold properties or metadata for use by specific ECAD products. EDA assumes that these files are hidden in the directory that contains the design data. EDA uses this directory as a reference point for all its activities and for capturing supplemental or hidden ECAD files. To avoid the possibility of duplicate names (same names for different objects) if multiple designs are stored within containers in a common staging directory, each directory containing an ECAD design must be given a unique name. The name of this directory must match (be the same as that of) the design's item ID, also known as the part number.

© 2025 Siemens

Create cache and staging directories

The first time you select a command on the Teamcenter menu and log on, the cache and staging directories are created automatically. The staging directory path is based on the staging directory setting in the *TC_ROOT***eda****TCEDAClient.properties** file. The current operating system user name may be added to that value as a subfolder and that path is written to a new user-specific **TCEDAClient.properties** file in the *USER_HOME* directory. The cache directory is also created under the staging directory.

1. Open the design tool.

2. Log on to Teamcenter:

   a. Choose **Teamcenter→Purge Working Files**.

      Teamcenter EDA Login dialog box is displayed.

   b. Type your Teamcenter user name, password, and group, if applicable. A group is required only if you want to log on to a group that is not your default group.

   c. Click **Login**.

   d. In the **Purge Working Files** dialog box, click **Cancel**.

      When you do this for the first time, the cache and staging directories are created automatically, displaying the folders, including the latest ones.

3. Choose **Teamcenter→Logout**.

   You are logged off Teamcenter.

4. Verify that the cache and staging directories are created.

   The directories are created from the settings in the **TCEDAClient.properties** file.

Teamcenter EDA staging directory conventions

Teamcenter EDA requires that you create and work on designs in the staging directory. If you have already created designs in another directory, copy them to the staging directory and then work on them exclusively from the staging directory. All design datasets including schematic, PCB, and simulation data files must be saved in a single design root folder that is associated to a PCA item.

In the following examples, replace the first instance of *username* in the path with the operating system user name, replace the second instance of *username* in the path with the Teamcenter user ID, and *siteid* with the site ID of Teamcenter.

© 2025 Siemens

Follow these conventions:

- **PADS**
  - Save physical designs under a **pads** staging subdirectory. For example, save the latest version as

    `c:\ecad\EDAStaging\`*`username`*`\`*`username_siteid`*`\pads\latest\`*`itemId`*`\brd`

    and open a specified version as

    `c:\ecad\EDAStaging\`*`username`*`\`*`username_siteid`*`\pads\nonLatest\`*`itemId`*`.pcb.`*`rev.ver`*`\brd`

  - Save logical designs under a **pads** staging subdirectory. For example, save the latest version as

    `c:\ecad\EDAStaging\`*`username`*`\`*`username_siteid`*`\pads\latest\`*`itemId`*`\sch`

    and save a specified version as

    `c:\ecad\EDAStaging\`*`username`*`\`*`username_siteid`*`\pads\nonLatest\`*`itemId`*`.sch.`*`rev.ver`*`\sch`

- **Xpedition**

  Save designs under an **expedition** staging subdirectory. For example, save the latest version as

  `c:\ecad\EDAStaging\`*`username`*`\`*`username_siteid`*`\expedition\latest\`*`itemId`*

  and open a specified version as

  `c:\ecad\EDAStaging\`*`username`*`\`*`username_siteid`*`\expedition\nonLatest\`
  `itemId``.pcb.`*`rev.ver`*

Allow concurrent editing and saving of designs from staging folder into Teamcenter

ECAD designers can transfer check-out on an item in Teamcenter to a different user. This will allow the other user to continue to work on the designs checked out by previous user from the same location and then check in the designs as well. To allow this, you must set the **StagingDir** in **TcEDAClient.properties** file to a shared folder on a network drive and set new properties that defines what type of cache to use under which scenario.

Set the StagingDir in the TcEDAClient.properties file:

You can map a drive to a network folder and then you can set it to use that mapped drive as the staging directory.

1. If required, you can map a network drive, for example, *S* drive to a shared folder on network, for example, *\\<machine_name>\Shared*.

© 2025 Siemens

2. Set the staging directory **StagingDir** in the **TCEDAClient.properties** file to the path of the mapped drive.

> Example:
>
> **StagingDir**=*S\:\\StagingDir*
>
> **StagingDir**=*\\\\<machine_name>\\Shared\\StagingDir*

Set a new properties in the TcEDAClient.properties file

> Note:
>
> The old **CacheDir** property is now obsolete and it will not be used even if included in the **TCEDAClient.properties** file.

- **CacheType**

  The new property **CacheType** must be set in the **TCEDAClient.properties** file. The valid values are **Local**, **Shared**, and **ClientServer**. The default value is **Local**.

  - **Local**: This is the type of cache used in previous versions of EDA. The physical cache will be at *%USERPROFILE%***/Teamcenter/EDA**/*<TC_SERVER_SITE_ID>*.

  - **Shared**: This type of cache can be shared by more than one user. The connection to the cache will be closed after each access so that the other users are not blocked. However, this can affect performance on a network file cache. The physical cache will be under the staging folder in a hidden folder name **tcedacache.**

  - **ClientServer**: This cache can be accessed through a client-server environment. This cache can be shared and it also provides better performance than the **Shared** option. However, the server must be **ON**, before it can be accessed. The physical cache will be under the staging folder in a hidden folder name **tcedacache.**

- **CacheHost**

  This is a new property, but is only used if **CacheType**=**ClientServer**. It defines the host port for the Derby server.

- **CachePort**

  This is a new property, but is only used if **CacheType**=**ClientServer**. It defines the host IP address for the Derby server

- **Set Derby Client/Server**

© 2025 Siemens

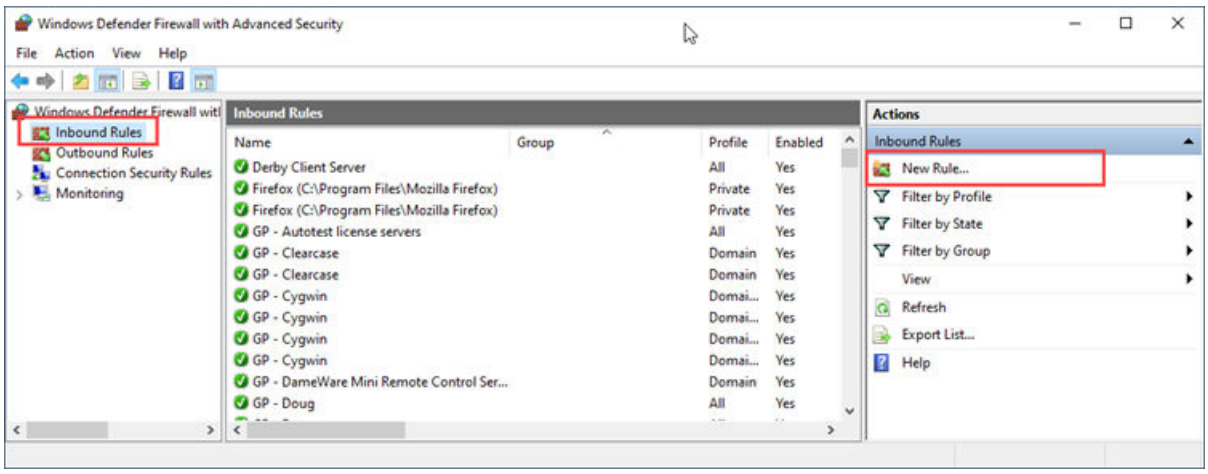
When **CacheType** is set to **ClientServer**, user must start a Derby server on some machine for all the clients to use. The clients will point to that server using the **CacheHost** and **CachePort** properties.

The server machine must be setup to allow inbound and outbound traffic on that port.
To allow port input/output

1. Go to **Settings** > **Windows Defender Firewall.**

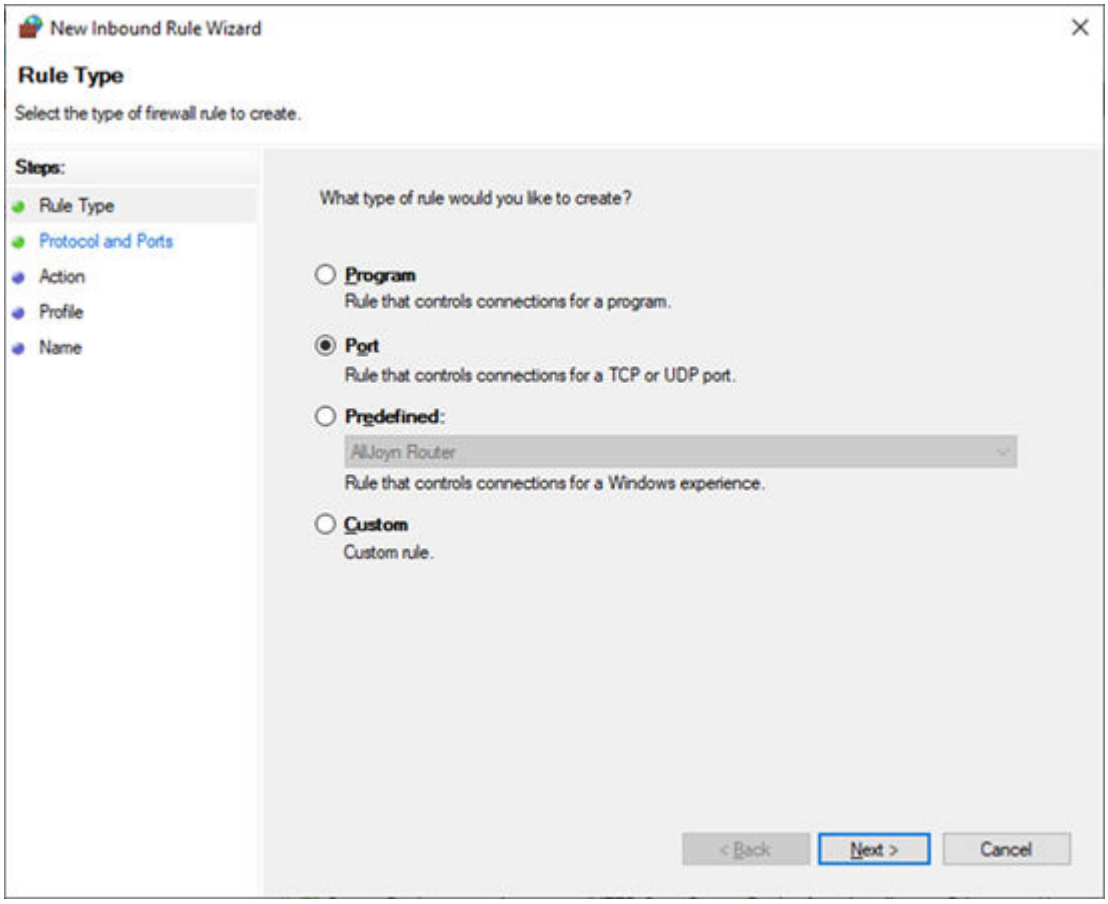
2. Click **Advanced settings**.

3. Select **Inbound Rules**, then click **New Rule...**.



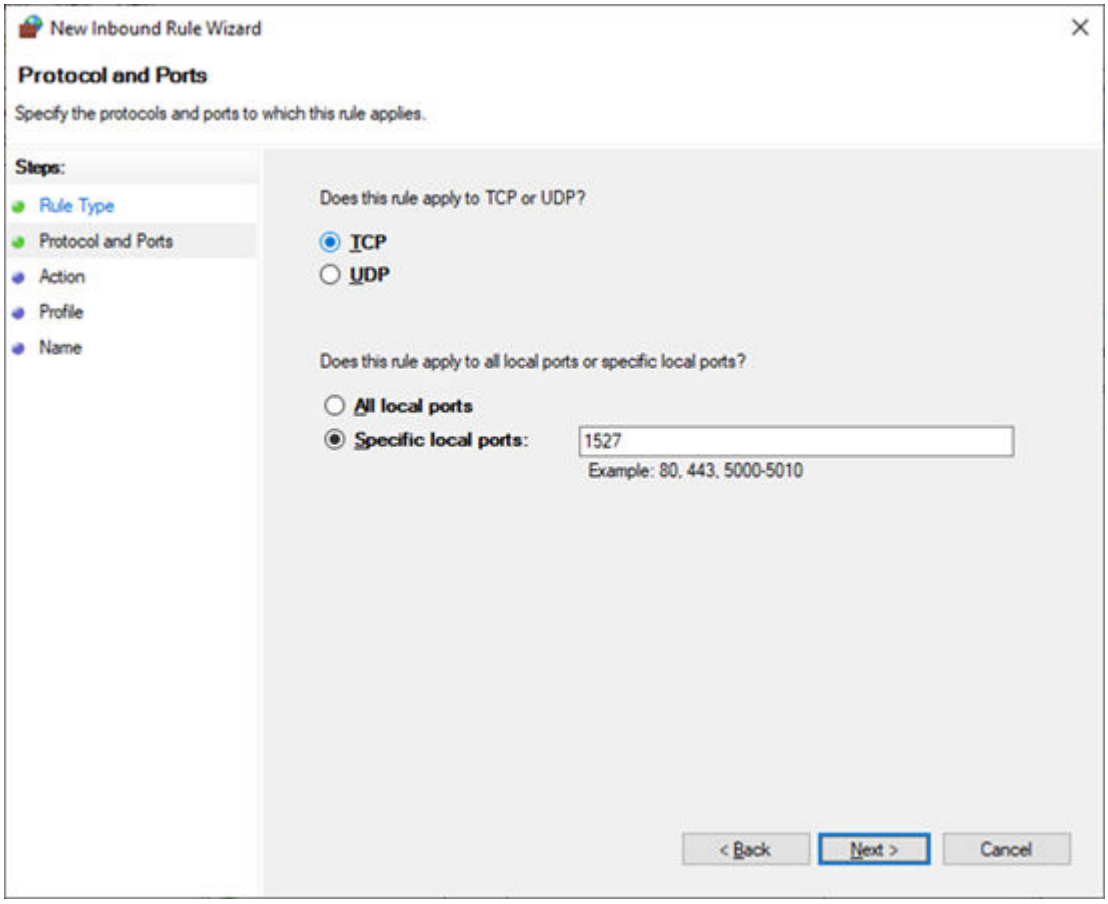
4. Select **Port** then click **Next**.

© 2025 Siemens



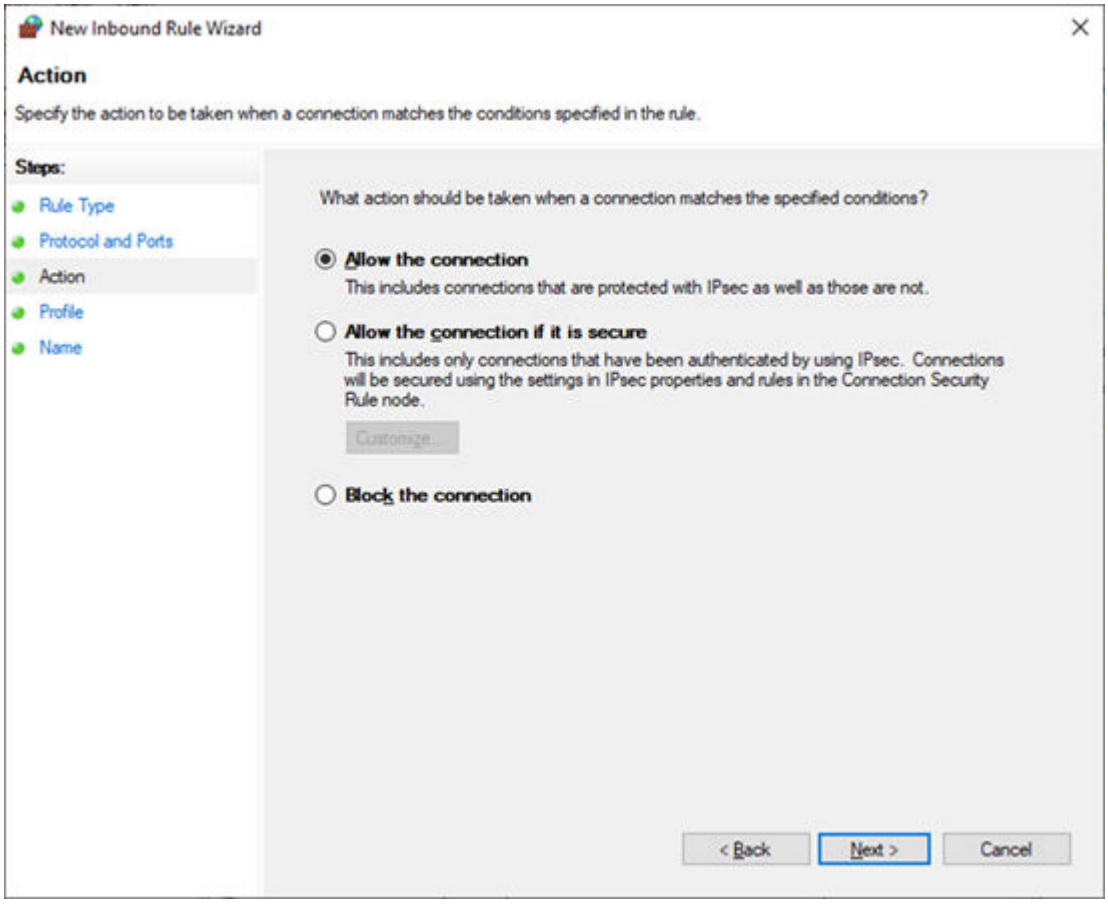
5. Keep **TCP** selected, enter **Specific local ports**, then click **Next**.

© 2025 Siemens



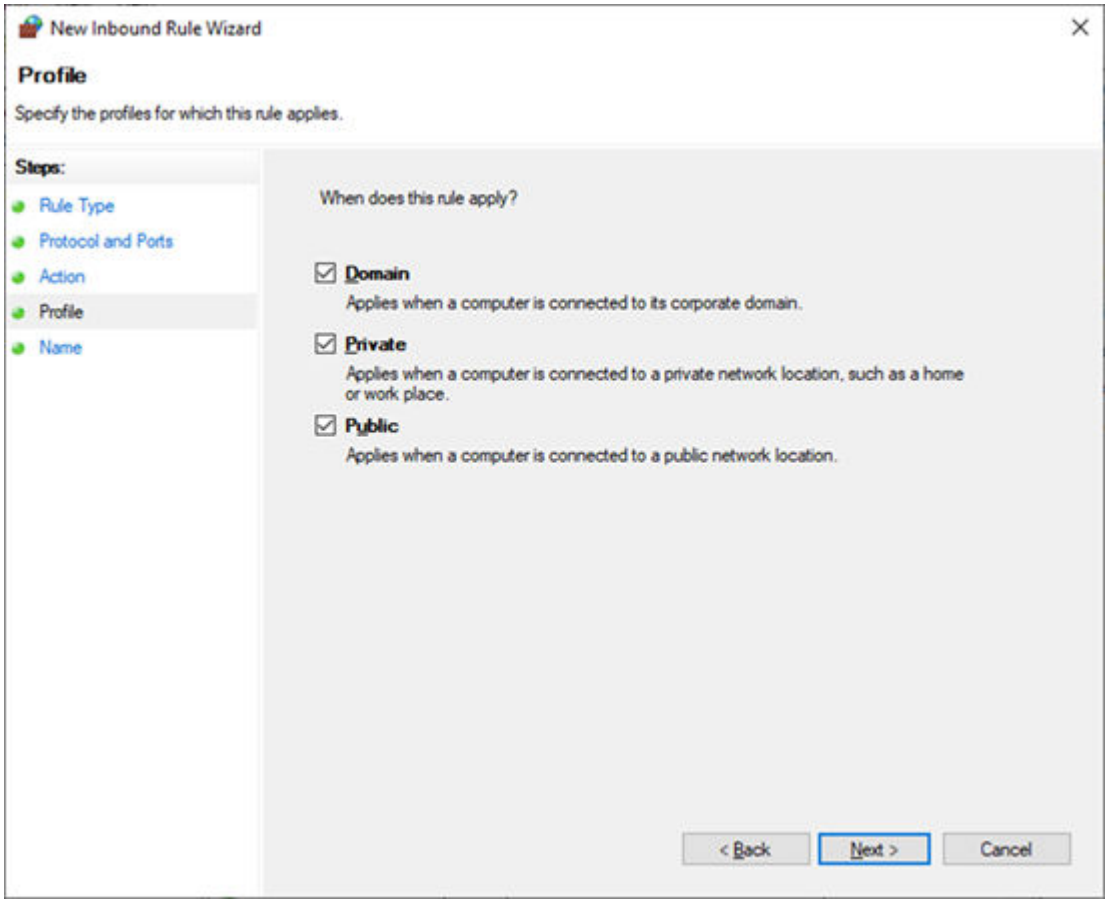
6. Keep **Allow the connection** selected, then click **Next**.

© 2025 Siemens



7. Leave all options about when the rule will apply checked, or set as required, then click **Next**.

© 2025 Siemens



8. Specify a **Name** for the Cache Server, then click **Finish**.

   Repeat the steps to create a similar outbound rule.

- **Start Server**

  To start the cache server, install EDA client on the machine you want to run the server on. Open a console and run the following commands:

  set CLASSPATH=%TCEDAECAD_ROOT%\derbyshared.jar;%TCEDAECAD_ROOT%\derby.jar

  java -jar %TCEDAECAD_ROOT%\derbyrun.jar server start -h <host> -p <port> -noSecurityManager

  The host (-h ipaddr) option value must be replaced with the server machine IP address.

  The port (-p port) option is optional, if not specified, port 1527 is used.

© 2025 Siemens

Example:

java -jar %TCEDAECAD_ROOT%\derbyrun.jar server start -h 10.104.127.167 -p 1529 -noSecurityManager

Modify default user properties in the TCEDAClient.properties file

Caution:

Follow this process only if you must change the properties for the user, the cache directory, or staging directory, this process can be followed. Siemens Digital Industries Software does not recommend this method to switch back and forth between settings but only as a one-time change that should be undertaken only if necessary.

1. Use the EDA client to check in all currently checked-out designs.

2. Use the EDA client to purge the cache.

3. Manually delete the cache and staging directories.

4. Edit the **URL** line in the *TCEDAECAD_ROOT***TCEDAClient.properties** file to point to the new server:

   `URL=http://`*`hostname`*`:`*`port`*`/tc`
   `StagingDir=`*`staging-directory-location`*

   In the example, replace *hostname* with the name of the new server running in the thin client environment, and replace *port* with the port number of that server.

   Note:

   The path of the staging directory must contain double backslashes on Windows systems, for example:

   `C:\\ecad\\EdaStaging\\`

5. Edit the *%USERPROFILE%***Teamcenter\EDA\TCEDAClient.properties** files to match the preceding settings, as necessary.

   Caution:

   Any definitions in this file override the file in *TCEDAECAD_ROOT* even if you edit the properties there.

© 2025 Siemens

Set a flatter folder structure for saving ECAD designs

In scenarios where two or more designers work on a single ECAD design within a shared network while saving the designs, multiple levels of folders are created based on the operating system user name, Teamcenter user name-site ID, project ID, and so on. This causes the file to be nested deep within the structure. This hierarchical folder structure is defined in the *TCEDAECAD_ROOT***TCEDAClient.properties** file.

To make this structure flatter and remove the multiple levels of folders, enable the **EDAStagingDirLevels** property defined in this file. The new structure saved in the staging directory excludes the intermediate folders based on the user's profile.

To enable this property, remove the # sign in the left most line where the property is defined.

> Example:
>
> By default, when a designer is working on an ECAD design, the files are saved in the staging directory format:
>
> **C:\Staging\user\siteid\app\latest\ItemID**
>
> Remove all the multiple folders created while saving ECAD designs by enabling the `EDAStagingDirLevels` property and the files will be saved in the staging directory format:
>
> **C:\Staging\view_<RevID>_PrjName** for read-only and previous revision files.
>
> **C:\Staging\PrjName** for WIP project.

Modifying the default logging configuration

You can examine EDA log files to troubleshoot problems. You can update the **log4j.properties** file that is located in the *TCEDAECAD_ROOT* directory to change the default logging configuration.

In the **log4j.properties** file, you can update the log name, change log values, change the location of log storage, and so on.

After making changes to the **log4j.properties** file, restart the EDA client.

The log files are stored in the location you specified in the **log4j.properties** file and are named as *$user_name*_**TcEDA.log**.

The historical log files are named as *$user_name*_**TcEDA.log_yyyy-mm-dd.log**.

> Note:
>
> Some connectors also produce log files that cover details of their operation. Refer to the **settings.ini** files within the installed connector for additional information.

© 2025 Siemens

Customize the integration by adding additional command calls

EDA integrations uses the following commands while performing integration operations between Teamcenter and ECAD tools. You can use these commands to customize the integration.

Use the **edacli.bat** file to access these commands. The **edacli.bat** file initiates a Java process that passes the required arguments to the EDA integration. The **edacli.bat** file is located in the *TCEDAECAD_ROOT* directory.

Syntax

edacli -*command* **-application=***application_name* **-status=** *file_name*

Arguments

-*command*

Specifies the command to execute. The following commands are available:

Command	Description
checkPartWorkFlow Status	Displays the **Workflow Status** dialog box that shows the status of your workflow requests.
getMyWorklist	Displays the **My Worklist** dialog box that shows your workflow tasks.
logout	Logs you off Teamcenter after displaying the **Logout** dialog box. Note: The **logout** command only releases the Teamcenter license. To release ECAD tool license, use the ECAD tools.
logoutNoConfirm	Logs you off Teamcenter without the **Logout** dialog box. Note: The **logout** command only releases the Teamcenter license. To release ECAD tool license, use the ECAD tools.
purgeCache	Displays the **Purge Working Files** dialog box. The **Purge Working Files** command allows you to clear the cache of the designs that are not checked out.

© 2025 Siemens

-application

Specifies the application name representing the ECAD tool. This value specifies which configuration file to use.

The following applications are available:

Command	Description
expeditionPcb	Xpedition and Designer
padsPcb	PADS Layout
padsSchematic	PADS Logic

-status

Specifies the full path for the EDA status file that the common client creates.

The common client writes status information to this file. The status is either **success** or **failure**. If the status is **failure**, the common client displays a failure message.

ECAD tool connector configurations and customizations for design management

Overview of additional customizations required for your ECAD tool

After the configurations required to set up design management on the EDA client are complete you must further configure and customize your ECAD tool connector as per your requirement. Refer to:

- **Connector configurations and customizations for PADS Professional and Xpedition**
- **Connector configurations and customizations for PADS Layout and PADS Logic**

Connector configurations and customizations for PADS Professional and Xpedition

Customizing ECAD tool menus for Active Workspace

Teamcenter Active Workspace panel is supported from within the EDA client panels. If this is configured, the design workflow and workflow status menu items can be instead accessed from the Teamcenter design info menu item. In such a case, these two menu items can be removed from the user interface.

To remove the Design Workflow and Workflow Status menus from PADS Professional Flow and PADS Designer with PADS Design Flow:

© 2025 Siemens

1. Revise your *%TCEDAECAD_ROOT%***\psw\dxd\pswMenu.vbs** file for PADS Designer.

2. Replace the following lines:

```
addButton4 menu, "&Workflow",_
                     "&Design Workflow...",
"pswDesignWorkflow.vbs",_
                     "&Part Workflow...",
"pswInitiateWorkflow.vbs",_
                     "&Workflow Status...",   "pswWorkflowStatus.vbs"
```

with the following lines:

```
addButton menu,  "Part &Workflow...",     "pswInitiateWorkflow.vbs"
```

To remove the Design Workflow and Workflow Status menus from PADS Professional Flow:

1. Revise your *%TCEDAECAD_ROOT%***\psw\exp\pswMenu.vbs** file for Xpedition PCB.

2. Replace the lines:

```
addButton4 menu, "&Workflow",_
                     "&Design Workflow...",
"pswDesignWorkflow.vbs",
                      "design_wf_16.bmp",_
                     "&Part Workflow...",
"pswInitiateWorkflow.vbs",
                      "library_wf_16.bmp",_
                     "&Workflow Status...",
"pswWorkflowStatus.vbs",
                     "processstate_16.bmp"
```

with the following lines:

```
addButton menu,  "Part &Workflow...",
"pswInitiateWorkflow.vbs",
                      "library_wf_16.bmp"
```

To enable the Design Info, Workflow, and Inbox menus where the Active Workspace panels are not activated:

1. Revise your *%TCEDAECAD_ROOT%***\psw\dxd\pswMenu.vbs** file for PADS Designer.

© 2025 Siemens

OR

Revise your *%TCEDAECAD_ROOT%***\psw\exp\pswMenu.vbs** file for Xpedition PCB.

2. Retain the following lines:

```
addButton4 menu, "&Workflow",_
                      "&Design Workflow...",
"pswDesignWorkflow.vbs",
                      "design_wf_16.bmp",_
                      "&Part Workflow...",
"pswInitiateWorkflow.vbs",
                      "library_wf_16.bmp",_
                      "&Workflow Status...",
"pswWorkflowStatus.vbs",
                      "processstate_16.bmp"
```

Replace other Teamcenter menu options with the following lines:

```
    addButton menu, "Inbox...",
"pswWorkflowWorklist.vbs",
                        "inbox_16.bmp"

    addButton menu, "Design Info...",
"pswShowDesignInfo.vbs",
                        "information_16.bmp"

    addSeparator menu

    addButton4 menu, "&Workflow",_
                      "&Design Workflow...",
"pswDesignWorkflow.vbs",
                         "design_wf_16.bmp",_
                      "&Part Workflow...",
"pswInitiateWorkflow.vbs",
                         "library_wf_16.bmp",_
                      "&Workflow Status...",
"pswWorkflowStatus.vbs",
                         "processstate_16.bmp"

    addSeparator menu
```


© 2025 Siemens

Disable availability of Variant Manager to perform Save, SaveAs, or Checkout actions on design files

When a save, save as, or checkout action is attempted on design files, the EDA connector cannot identify initially if the designs are created with or without variants. To determine if variants are defined, the connector starts Variant Manager and searches for them. However, if you do not have a Variant Manager license, this process fails.

To work around the issue of a missing license or working with a license with limited capabilities, the connector must be customized to ignore the check for variants. This customization must be done in the*%TCEDAECAD_ROOT%***\expeditionPcb_usrdef.xml** configuration file.

Specifically, modify the **enableVariants** property to **False** in the configuration file to ignore the search for design variants. This file is for both boards and schematics.

> Note:
>
> The **padsDxSchematic_usrdef.xml** file is located at the same location as the other configuration files and must not be modified.

Configure connector behavior for Designer, Xpedition PCB, and PADS Professional

The **setting.ini** file allows you to configure connector behavior for Designer, Xpedition PCB, and PADS Professional. PADS layout does not include this feature. This file contains values that are accessed by identifiers, also referred to as keys.

The **setting.ini** file can be found at the following locations:

- For Designer: *%TCEDAECAD_ROOT%***\psw\dxd\settings.ini**
- For Xpedition PCB: *%TCEDAECAD_ROOT%***\psw\exp\settings.ini**

You can perform the following configurations or customizations using the **setting.ini** file:

- **Control component extraction in Partlister**
- **Control BOM values**
- **Review configuration file limitations**
- **Design Reference Designator Name**
- **Partlister delimiter**
- **Enabling logging**

© 2025 Siemens

Control component extraction in Partlister

The Designer tools use a component extraction process that uses the **Partlister** tool. This tool is configured to only extract specified attributes of components and map the ECAD tool internal attribute names with the names as specified in the **setting.ini** file.

The configuration file can be found at the following locations, with the file name depending on tool version:

- For Designer with Xpedition Design Flow: *%TCEDAECAD_ROOT%***\psw\dxd\partlister.v2.xml**
- For PADS Designer with PADS Design Flow: *%TCEDAECAD_ROOT%***\psw\dxd\partlister.pads.xml**
- For PADS Designer with PADS Professional Flow: *%TCEDAECAD_ROOT%***\psw\dxd\partlister.v2.xml**

The only entries that should be changed are those related to the names of the extracted attributes. In these files, look for the lines for column specification and having a type of **PROPERTY**.

They are similar to the following snippet:

```
<Column type="PROPERTY" property="Ref Designator" label="REFDES"
    width="15" key="PRIMARY" sort="ASCENDING" display="Full"/>
<Column type="PROPERTY" property="Part Number" label="PART_NUMBER"
    width="13" display="Full"/>
<Column type="PROPERTY" property="Package" label="PKG_TYPE"
    width="15" display="Full"/>
```

The **tag** property defines the internal ECAD tool attribute name and the tag **label** defines the external attribute name. The **label** value must match the definition within the **settings.ini** file.

Control BOM values

The **settings.ini** file contains several keys prefixed with the word **part**. These attributes control which values are written from Designer into the **edabom.xml** file and sent to Teamcenter. The following table describes which key affects which attributes. The values you specify must match the values generated by the Partlister xml files. These values are only for the Designer connector. Xpedition PCB cannot be configured using this method.

.ini file key	Edabom element	Edabom attribute
part.name	Component	Name
part.itemid	Component	itemId
part.revision	Component	itemRev
part.qty	Component	Qty
part.refdes	Rdn	Name

© 2025 Siemens

Review configuration file limitations

Each user or user company may use different names to define properties that are to be passed to Teamcenter. It is important that the configuration files described here are reviewed and corrected if necessary prior to putting Teamcenter into production use.

Design Reference Designator Name

Within the Xpedition and PADS tools, each part instance is assigned a unique externally visible identifier. This is called a *Reference Designator*. Some tools refer to this identifier as *REFDES* while other tools refer this identifier as *Ref Designator*. To support the use of either of these names, an entry in the **settings.ini** file allows the user to specify the proper name to use within their environment. This setting can be found in the following line:

```
part.refdes=REFDES
```

The schematic and board tools can define independent names for this identifier. The names entered in the **settings.ini** file must match the names entered in the Partlister configuration files found at the location *%TCEDAECAD_ROOT%***\psw\dxd**.

Partlister delimiter

The **input.delimiter** key specifies the delimiter used in the output file produced by partlister. This value must match the value specified in the **Partslister** configuration xml files described above.

Enabling logging

The **log.file** key controls connector logging. If **log.file** is set to a non-blank value, the connector will write logging information to the specified path and file. This information can help diagnosing connector issues.

Configure viewable file generation

The PCB portion of the Teamcenter EDA Connector for Xpedition and PADS Professional is configured to work with ODB++ files. The schematic viewable generator is preset to work with **edif200** files. Your ECAD tools may require additional licenses to generate any viewable file types.

The processing of ODB++ files can be performed either in automatic or manual mode. The ECAD tools require an **xeautoextended** license to enable the EDA connector to automatically generate the ODB++ files. These files can also be manually created using just the base licenses. Because of this need for an additional license, the connector has been developed to work with either automatic ODB++ file generation or to work with ODB+ files manually created before saving the designs to Teamcenter.

© 2025 Siemens

Generating ODB++ files manually

1. The ODB++ files can be manually generated from within the layout tool by selecting **Output→ODB++**.

2. In the **ODB++ creation** dialog box, specify the path for the output files as *.\Output\ODBpp*. This indicates the ODB output folder within the design's PCB folder.

3. Specify the job name for the ODB++ files within the connector software as **designodb**.

4. Select the **Compress Output** option.

5. The generated viewable data must be contained within a single file for the Teamcenter translation tool. Therefore, the ODB++ data (multiple files) must be compressed into a single TGZ file.

The path to the resulting file is *<design>\PCB\Output\ODBpp\designodb.tgz*.

Generating ODB++ files automatically

1. The ODB++ generation engine requires a configuration file that specifies all the characteristics of the content of the generated files.

2. To allow the ECAD tools to indicate the information that it requires to properly generate the ODB++ files, the EDA connector assumes that the user has executed the ODB++ engine at least once and saved the resulting specification files.

3. For this to happen, the user must run the **Output→ODB++** menu item from within either layout tool and set all the required configuration items.

4. The job name for the ODB++ files is hard-coded within the connector software as **designodb**.

5. Select the **Compress Output** option.

After performing the ODB++ file generation manually once, the user can then switch to automatic generation.

> Note:
>
> The configuration files are stored within the PCB design data, so this manual setup must be performed once for each design.

Defining the TcEdaUID attribute in Designer

EDA includes a new attribute, TcEdaUID, on BOM components. This attribute must be defined in Designer, and passed between Designer and Xpedition and/or PADS.

© 2025 Siemens

Define TcEdaUID attribute in Designer property definition list

Launching the property definition editor can be done in one of two ways. Launch either **xDM Library Tools** or **Library Manager**, as appropriate:

1. **From within Designer:**

   a. Launch Designer.

   b. Open the target active design.

   c. In the menu bar, select **Tools→Library Tools**.

   d. Go to step 4.

2. **From Windows Start Menu:**

   a. Select the proper Xpedition version.

   b. Select **Data Management**.

   c. Select **Library Tools**.

   d. Open target **Central Library**.

   e. Go to step 4.

3. **In the xDM Library Tools (or Library Manager) menu bar:**

   Select **Tools→Property Definition Editor**.

4. **In the Property Definition Editor window:**

   a. Click the **Add a property** button.

   b. Select:

      - The first column checkbox
      - Type **User**
      - Name **TcEdaUID**

© 2025 Siemens

- Format **Character String**

c. Click **OK**.

d. Save and close all active windows.

After the change to the Property Definition Editor, this change must be propagated to existing designs that are or will be managed by Teamcenter. To get the added property into an existing design, the user can either select **Tools→Update Libraries** and it reloads the library PRP file, or if Designer is closed and reopened, the library PRP file is reloaded.

If a new design is created, this added property will be imported from the central library.

> Note:
>
> The added property should be added to each central library that may be used in design work.

Enable passing of TcEdaUID attribute to PADS Layout - To use with Designer - PADS Flow only

To enable passing of the TcEdaUID attribute from PADS Designer to PADS Layout, you must modify the PADS configuration file used by the Partlister.

1. Edit the file *%SDD_HOME%***\standard\pads94.cfg** or *%SDD_HOME%***\standard\pads95.cfg**.

2. Add the following line at the end of the **General** section:

```
PKG TcEdaUID
```

An alternate configuration method is to create a unique configuration file for each user.

1. Make a copy of the file *%SDD_HOME%***\standard\pads94.cfg** or *%SDD_HOME%* **\standard\pads95.cfg**.

2. Edit the copy as indicated above.

3. For each user, go to **Designer→Setup→Settings→Project→PCB Configuration**.

4. Edit the config file path to point to the copied file.

Occurrence Attribute Mapping

Depending on what occurrence attribute mapping is required, edit the *%TCEDAECAD_ROOT%* **\eda\padsDxSchematic_usrdef.xml** file.

© 2025 Siemens

Considerations when working with multiple ECAD tools

Designer and Xpedition work on a shared database model for storing EDA information. This allows users to work on a single design from multiple tools simultaneously. However, Designer and Xpedition work differently for saving and opening ECAD files. This can create issues when the designer works on the same design files using the different ECAD tools.

Saving designs using different ECAD tools

Designer automatically saves designs into the database as soon as any changes are made. With Xpedition, designers must periodically save their changes.

Consider a scenario where the designer is working in Xpedition and does not save the design file. The designer then opens the same design in Designer. As soon as the changes are made in Designer, the files are automatically saved to Teamcenter. However, these design files will not have all the latest changes made in Xpedition. Hence, while working on the designs using different ECAD tools, users must make sure that the designs are saved every time any changes are made.

Opening designs using different ECAD tools

The shared database is locked when any tool is active. This can be an issue when the designer wants to check out a dataset from Teamcenter into a directory where multiple tools are operating. In cases when only one ECAD tool is used, the connector downloads a zip file into the design directory, closes the current design, unzips the file, and then reopens the design.

In scenarios where multiple ECAD tools are in use, the shared database is in a locked state, and the files will not unzip. When the connector attempts to unzip a design, it warns the user that the shared database will be overwritten.

Saving BOMS

During a save operation, such as *save*, *save as*, or *check in* from the ECAD tool, the designer is prompted to save a bill of materials. Regardless of the tool being used, user can choose either a *PCB BOM*, or a *schematic BOM*, or no BOM option.

Xpedition can produce both a layout BOM and a schematic BOM. It produces the schematic BOM through the ECAD tool, Partlister.

Although the layout BOM option is prompted when you save the designs from the schematic tool, Designer is only capable of producing a schematic BOM, which is done through the ECAD tool, Partlister. If a user chooses to save a PCB BOM from Designer, a message is displayed that Designer cannot generate the PCB BOM. The options displayed for the users would be to generate either a schematic BOM or no BOM.

It is assumed that the schematic BOM gives the most accurate view of the materials associated with the design.

© 2025 Siemens

The following table summarizes which BOMs can be generated.

ECAD tool	Schematic BOM	Layout BOM	
Xpedition PCB	Yes	Yes	Only in Xpedition Flow or PADS Professional Flow or PADS VX Flow
Designer	Yes	No	
PADS Logic	Yes	No	Only in PADS Flow More information about how these files are generated in the Xpedition and PADS Design Management help.
PADS Layout	No	Yes	

Considerations for creating packaging

The generation of a valid BOM is dependent on component identification being valid. For the scenarios where the user has added or removed components, the user must ensure that the design has been packaged again. If the designs are not packaged after the components are added or removed, the component reference designators may not be correct and the generated BOM will not reflect the latest changes.

Considerations for saving design viewables in Teamcenter

The following table summarizes which viewables can be generated with which tools:

	Schematic viewable	Layout or PCB viewable	
Xpedition PCB	Yes	Yes	Only in Xpedition Flow or PADS Professional Flow or PADS VX Flow
Designer	Yes	No	
PADS Logic	Yes	No	Only in PADS Flow More information about how these files are generated in the Xpedition and PADS Design Management help.
PADS Layout	No	Yes	

Considerations for Xpedition VX integration with the EDA client

If EDA client version 2.4 or a later version is installed on your ECAD workstation and the system environment variable *%TCEDAECAD_ROOT%* is set, the user need not add the EDA menu items manually. Designer and Xpedition have a built-in menu for EDA operations.

To enable the EDA menu, open **Designer**, select **File→Data Management Integration→Teamcenter**.

Follow the same steps for Xpedition to enable EDA menus. Do not configure EDA menu items again, otherwise multiple EDA menus will be configured.

© 2025 Siemens

> Note:
>
> These steps are applicable for enabling EDA menus as the Active Workspace panels in the ECAD tool. For the rich client panels, VB scripts are used to generate the menus.

Troubleshooting ECAD tool integration with Teamcenter

These are a few troubleshooting checks you can perform if the ECAD tool integration with Teamcenter does not work properly:

Issues	Action
Teamcenter menu does not show up in the menu bar.	Verify that the **scripts.ini** file either does not exist in WDIR folder or it does not contain the required lines for the integration to work. The environment variable WDIR was previously shared between all Siemens EDA products. It is now changed to allow each tool version to have a unique WDIR location. The Teamcenter EDA connectors do not need to implement any changes to support the WDIR update. However, each different WDIR locations must contain a copy of the ECAD tool **scripts.ini** file with Teamcenter extensions.
You get an error when a menu item is selected from the Teamcenter menu.	This means that the VBScript scripts copied into WDIR do not contain the correct path to EDA.
No BOM is generated during a save operation when **With BOM** is selected	Make sure that the Partlister file is configured to output a delimited file with a part number attribute as one of the columns. The delimiter in the file and the names of the various columns must match up to those specified in **settings.ini**.
Either Designer, Xpedition, PADS or their related Teamcenter connectors do not launch or report that they have license issues.	Make sure that the proper ECAD tool and Teamcenter licenses are installed and activated.

Connector configurations and customizations for PADS Layout and PADS Logic

Customizing PADS Layout or PADS Logic menus for Active Workspace

Teamcenter Active Workspace is supported within the EDA client panels. If Active Workspace is configured, the **Design workflow** and **Workflow status** menu items can be instead accessed from the

© 2025 Siemens

Teamcenter **Design info** menu item. In that case, these two menu items can be removed from the user interface.

To remove the Design workflow and Workflow status menus from PADS Layout or PADS Logic:

1. Revise your *%TCEDAECAD_ROOT%***\PADS\wnti\vb\Teamcenter.bas** file for PADS Layout or PADS logic.

2. Search for comments with text **Active Workspace** and look for more instructions within the comments to remove the menus.

Defining the TcEDAUID attribute in the PADS Layout attribute dictionary

EDA includes a new attribute, named **TcEDAUID**, on BOM components. This attribute must be defined in the PADS Layout attribute dictionary, so that it will be included in ECO reports for backward annotation. This must be done with an empty design, or one without a TcEDAUID attribute defined.

1. Click **Edit→Attribute Dictionary**.

2. Click **New**.

3. Enter attribute name as **TcEdaUID**.

4. On the **Types** tab, select **Free Text**.

5. On the **Objects** tab:

   - Select **Part**.

   - Select **ECO Registered**.

6. Click **OK**.

7. Click **Close**.

You must now save this updated attribute dictionary to a default startup file, so that it will be used in all new designs that uses the same startup file.

1. Click the **File** menu and select **Save as Start-up File**.

2. Change the **File** name if required.

3. To save the updated attribute dictionary:

   - Select **Attributes** section.

© 2025 Siemens

- Enter a **Startup File Description** if required, for example, EDA.
- Click **OK**.

Back annotating from PADS Layout

PADS Layout to PADS Logic

Refer the *PADS Layout Users Guide and Reference Manual*, chapter *Working with PADS Logic*, section **Backward Annotating from PADS Layout to PADS Logic**. You must use the first method, *Automated Backward Annotation Process* or the third method, **Creating the ECO File in PADS Logic**.

> Note:
>
> The second method, that is, the creating the ECO File in PADS Layout method, will not work because the TcEdaUID properties are updated programmatically, and will not be updated in the ECO file created by Layout.

PADS Layout to PADS Designer

Refer the *PADS Layout Users Guide and Reference Manual*, chapter *Working with Designer*, section **Backward Annotating from PADS Layout to PADS Designer**.

You can use any method, however, do not choose to use an existing ECO file. You must use a method that creates a new ECO file using Netlist comparison.

Forward annotating from PADS Logic

Follow these steps to forward annotate from PADS Logic to PADS Layout:

1. Click the **Tools** menu and select **PADS Layout**.
2. Open the **Design** tab.
3. Select **ECO To PCB**.
4. Click **Disconnect**.

Setting up ECAD translation

ECAD translation modes

The following translation modes are supported for ECAD translation:

- **Local translation**

© 2025 Siemens

The viewable file is generated locally. This option is available only on Windows clients.

- ECAD translation happens by default and is installed out-of-the-box and deployed locally on the EDA client machines with the EDA Standalone Client TEM feature.

- You must edit the **TCEDAClient.properties** file located in the *TCEADAECAD_ROOT* directory to update the path to the translation directory (*Teamcenter EDA application root\emps\translation*).

- **Remote translation**

  The EDA integration initiates a remote translation of the intermediate file into a viewable format.

  ECAD translation is embedded into the Teamcenter Dispatcher framework. Make sure that the **Dispatcher server and required translator are installed**. After installing and setting up the dispatcher you must **enable the PcbToFatf translator**.

  If you select the **Viewable** check box while saving your EDA design, the following automatic translation process takes place:

  - The design intermediate file that can be parsed by the ECAD translator is extracted.

  - The intermediate file is converted into a neutral file format: **XFATF** for PCB designs and **XSCH** for schematic designs.

  - The neutral file is saved as a viewable dataset in Teamcenter.

You can enable either the local translation mode or the remote translation mode. You may also enable both these modes and control which mode to use by editing the **EDA_PreferredLocalTS** preference.

Enable PcbToFatf translator

After you install and configure Dispatcher as per the deployment strategy, you must enable the **PcbToFatf** translator required for the remote translation of the intermediate file into a viewable format.

You can use TEM to install the translator or manually edit the **translator.xml** file to enable the translator. For instructions on how to enable the **PcbToFatf** translator using TEM, see the Dispatcher — Deployment and Administration help on Support Center.

The following is an example of how to edit the **translator.xml** file to enable the translator.

Prerequisites: Verify that the **TransExecutable dir** is correctly specified for this translator.

1. Open the **translator.xml** file from the *Dispatcher_Root***Module\conf** directory.

2. Set the **isactive** attribute to **true** to activate this translator.

© 2025 Siemens

> Note:
>
> By default, this attribute is set to **false**.

3. Open the *translator***.bat** (Windows) from the *Dispatcher_Root***\Module\Translators***translator_name* directory and edit the **CHANGE_ME** tags.

   For more information about **CHANGE_ME** tags, see the **.bat** file for the translator.

4. Run the translator in standalone mode from the command prompt to verify whether it is working properly.

   For more information about running the translator in standalone mode, from a command prompt, change to the **Dispatcher\Translators***translator_name* directory and run the appropriate command for your platform:

   > Note:
   >
   > For more information, see the **Readme** file in the **Translators***translator_name* directory.
   >
   > Windows: *translator_name***.bat -help**

5. (Optional) To set up *Dispatcher-client-proxy-user* as the owner of the translated files, make the following change to the *translator_name***.bat** (Windows) file.

   NxTransDirect translator example:

   ```
   "%UGII_BASE_DIR%\ugmanager\run_nxtrans.bat" -u=Dispatcher-client-proxy-user
   -p=password
   -changeOwnerToCad=false "-pim=yes" %*
   ```

   If you set up *Dispatcher-client-proxy-user* as the owner of the translated files, you must create a dispatcher client access rule. For more information on how to create a dispatcher client access rule, see the Dispatcher — Deployment and Administration help.

   After installation, you must add a rule to the access rule tree permitting the translation service proxy user to update attributes of a **DispatcherRequest** object. If this access rule is not created correctly, the dispatcher client reports errors.

Supported translator services

Following translator services are supported in Teamcenter EDA:

- mentor
- pads

© 2025 Siemens

- xpedition

Installing the ECAD viewer

ECAD Viewer displays separate ECAD views that help you navigate, search, and work with PCB designs, schematic representations, and design for assembly and test output. You can use the ECAD viewer to view the schematic and layout designs. The first steps to consider before you work with the ECAD Viewer is to make sure your native ECAD files are translated to the XFATF or XSCH neutral file format. You can also translate EDIF 2.0 and 3.0 schematic file formats to the neutral file format.

Installing the ECAD viewer is optional; if you do not want to use the ECAD viewer for design view and markups, you can still use the EDA client to check in and check out designs between your ECAD design tool and Teamcenter.

For more information about installing the ECAD viewer, see the Teamcenter Installation on Windows Using TEM help.

For more information about using the ECAD viewer, see ECAD Viewer documentation on Support Center.

© 2025 Siemens

© 2025 Siemens

4. Integrating EDM with Teamcenter EDA for managing design files

Planning your deployment for EDM integration with Teamcenter

For installing and integrating Teamcenter with EDM for EDX design management, you must plan your deployment and your environment setup. You can install the Teamcenter EDA server and client components through **Deployment Center** or through **TEM**.

As an administrator, you must install the appropriate components on the server (Teamcenter) and client machines (ECAD workstation - Xpedition or PADS) as per the site requirements as mentioned below:

Server components	Requirements
Teamcenter server	Teamcenter server is used to store and track the ECAD designs. For information about the versions of operating systems, third-party software, and Teamcenter software configurations that are certified for the EDA platform, refer to the **Hardware and Software Certifications** knowledge base article on Support Center. If you have an existing Teamcenter environment, you can update it to support the latest EDA version. Ensure that you download the compatible versions from Support Center before starting your deployment.
EDA server support	Teamcenter server is used to store and track the ECAD designs. EDA server support is required to add ECAD data model to the Teamcenter server environment. You must add EDA server support on top of the Teamcenter environment. As the EDA client is supported in a four-tier architecture, the EDA Server Support feature (EDA data model) must also be installed on a four-tier Teamcenter server.
Business Modeler IDE	Business Modeler IDE is required to configure derived data for EDA objects. Before working with Teamcenter EDA objects in Business Modeler IDE, you must install the EDA server support features in Teamcenter Environment Manager (TEM). From the Features panel, choose **Extensions→Mechatronics Process**

© 2025 Siemens

Server components	Requirements
	Management→EDA for Business Modeler IDE. You must also install the EDA Server Support template to your project.
Dispatcher	Dispatcher is required for remote ECAD translations to translate and save the design intermediate file into a viewable dataset in Teamcenter. If Dispatcher is already installed in your existing Teamcenter environment, you can use the same instance. However, you must modify it to include the translators that are specific to ECAD translation.

Client components	Requirements
EDA client	EDA client moves designs between Xpedition and Teamcenter. The EDA client must be installed on the ECAD workstation using the Teamcenter software kit. Before installing the EDA client, see the **Hardware and Software Certifications** knowledge base article on Support Center to identify which versions of the Xpedition are supported on which versions of Teamcenter EDA. Siemens Digital Industries Software recommends that you first perform the installation on a test system to verify the parameter values required for the installation on the ECAD workstations. **Prerequisites and requirements before installing the EDA client:** • **Supported Operating System:** The EDA client framework is supported only on the Windows operating system. • **Supported Java version:** For information about the supported Java versions that are certified for the EDA platform, see the **Hardware and Software Certifications** knowledge base article on Support Center. • **Software requirements specific to the ECAD tools on which the EDA client should be installed:** • If you use Xpedition as the ECAD design tool, install Xpedition Designer or Layout on the client workstations.

© 2025 Siemens

Client components	Requirements
	• **Information required for EDA client installation from Teamcenter installation administrator:** • FMS client cache (FCC) If the FMS client cache (FCC) exists, use the existing one or specify a new one. • The FCC parent information for the EDA client, including the host and port value The minimum information required is the host name and the port. • The four-tier, middle-tier Web application server URL, and the connection name for the EDA client • Logon URL and the application ID To enable Teamcenter Security Services • Host name, port, and license server mode of the Siemens Digital Industries Software common license server Note: Teamcenter software kit contains the 64-bit installer for Windows system. You can select Win32 applications directly from TEM or DC along with the rest of the Win64 applications from within the Teamcenter software kit. • **Other installation considerations** • Only the English versions of ECAD tools are supported. • Item ID, variant name, dataset name, reference designator value, and mapped attribute names and values must contain only ASCII characters. • The staging directory and all file names must contain only ASCII characters.
EDM Server	The EDM server administrator must perform configurations to integrate Xpedition EDM with Teamcenter.

© 2025 Siemens

Workflow to install and setup Teamcenter EDA for EDM integration

Scenarios to deploy the Teamcenter EDA server and client components:

Using Deployment Center	*Using TEM*
Install Teamcenter server and the EDA client	Install Teamcenter server, Install BMIDE, Install Dispatcher
↓	↓
Set up Teamcenter server for design management	Set up Teamcenter server for design management
↓	↓
Set up EDM integration	Install the EDA client
↓	↓
Perform additional configurations and customizations required for EDM integrations	Set up EDM integration
↓	↓
Set up ECAD translation	Perform additional configurations and customizations required for EDM integrations
	↓
	Set up ECAD translation

Installing Teamcenter server and the EDA client for EDM integration through Deployment Center

The following procedures assume that you are installing EDA on an existing Teamcenter and that you are familiar with Deployment Center.

Procedure

1. Log on to Deployment Center and select the environment to which you want to add Teamcenter EDA .

2. Go to the **Applications** task. Click **Add or Remove Selected Applications**.

© 2025 Siemens

3. In the **Available Applications** panel, use the web browser search to find the Electronic Design Automation (EDA) application. Select the application, and then click **Update Selected Applications**.

   Deployment Center automatically selects any additional dependent applications.

4. In the **Available Applications** panel, under **EDA Client**, select the required applications for integrating Teamcenter EDA and Xpedition EDM for design management on the ECAD client machine:

   - (Optional) Select **EDA Translation**.
   - **Mentor Xpedition EDM Design**
   - **Teamcenter Gateway for EDA**

   By default, all options under **EDA Client** are selected. Remove the tools and applications that are not required for design integration and click **Update Selected Applications**.

   Deployment Center automatically selects any additional dependent applications.

5. Go to the **Components** task.

6. In the **Selected Components** list of the **Components** task, note any remaining components whose configuration status is not **100%**. Select each incomplete component, enter required parameters, and save component settings until all components in the environment show a configuration status of **100%**.

   When all components are fully configured, the **Deploy** task is enabled.

7. Go to the **Deploy** task. Click **Generate Install Scripts** to generate deployment scripts you will use to update affected machines.

   When script generation is complete, note any special instructions in the **Deploy Instructions** panel.

8. Locate deployment scripts, copy each script to its target machine, and then run each script on its target machine.

   For more information about running deployment scripts, see *Run the deployment scripts* in the Deployment Center documentation.

   You can also generate mini-kits for mass client deployment. Running the **generateMiniSoftware** utility extracts only the necessary software from the full software kits and builds a smaller, more portable software kit for the specific client deployment. For more information, see *Generating mini kits for mass client installations* in *Deployment Center — Usage* on Support Center.

© 2025 Siemens

If you face any issues during the deployment, see *Troubleshoot the deployment script* in the Deployment Center documentation.

Postrequisites

After the successful installation of the server and client components, you must configure the Teamcenter server as well as perform the EDA client configurations for setting up EDM integration.

- **To integrate EDM with Teamcenter EDA for design management:**
  - **Set up Teamcenter server for Xpedition EDM integration**
  - **Set up EDM integration**
  - **Perform EDM tool connector configurations for Teamcenter integration**

Installing Teamcenter server for EDM integration through TEM

Installing the Teamcenter server for EDM integration

Install Teamcenter

To install Teamcenter EDA, you must first install Teamcenter. To install the Teamcenter corporate server, refer to the Teamcenter Installation on Windows Using TEM help.

If you have an existing Teamcenter environment, you can update it to support the latest EDA version.

Refer to the Install patches on a Teamcenter server topic in the Teamcenter Installation on Windows Using TEM help for instruction on installing Teamcenter patches.

Add EDA support on the Teamcenter server for design management

EDA server support adds the EDA data model to the existing Teamcenter data model.

Before installing the EDA server support, you must install the Teamcenter server.

1. From your Teamcenter environment, start Teamcenter Environment Manager (TEM).
2. Select the **Configuration Manager** option to perform maintenance on an existing installation and click **Next** until the **Select Features** dialog box appears.
3. In the **Select Features** dialog box:
   a. Select the following options:

© 2025 Siemens

- Choose **Extensions→Mechatronics Process Management** and select **EDA Server Support**.
- Choose **Extensions→Mechatronics Process Management** and select **EMPS-Foundation**.
- Choose **Extensions→Mechatronics Process Management** and select **EDA for Business Modeler IDE**.
- Specify **Active Workspace** server extensions features for EDA. These are available in the **Features** panel in **Teamcenter Environment Manager (TEM),** under **Base Install→Active Workspace→Server Extensions→EDA Server Support for Active Workspace**.
- Specify **Active Workspace** client extensions features for EDA. These are available in the **Features** panel in **Teamcenter Environment Manager (TEM),** under **Base Install→Active Workspace→Client→Electronic Design Automation for Active Workspace**.

b. In the **Installation Directory** box, type the location where you want to install Teamcenter (TC_ROOT).

c. Click **Next**.

4. Enter information as needed in the subsequent panes.

5. In the **Confirm Selections** dialog box, click **Next**.

   The EDA server components are installed.

   The **Add/Remove Components** dialog box confirms if the installation is successful. If the installation is unsuccessful, click **Show Details** and proceed as needed.

6. Obtain the license file that includes EDA licensing and EDA library license (for library support) and install it on the server.

After adding the EDA server support to your Teamcenter environment, you must verify your installation by checking whether the EDA data types are added to Teamcenter.

Verify your installation

Verify the addition of the EDA data types to the server:

1. Run the Teamcenter rich client.

2. In My Teamcenter, choose **File→New→Item**.

   The **New Item** dialog box is displayed.

© 2025 Siemens

3. Verify that the EDA, EDAComp, EDASchem, and EDACCABase item types are added to the list of types you can create.

4. Select one of the EDA item types and create an instance.

5. Verify that the item instance is created.

Install BMIDE to configure derived data for EDA objects

You must install Business Modeler IDE to **configure derived data for EDA objects and to import the sample BMIDE EDA server support template**.

You can install BMIDE either as a stand-alone application or place the BMIDE plug-ins in an existing Eclipse environment.

For instructions on installing BMIDE, refer to the Install the Business Modeler IDE topic or the Add the Business Modeler IDE to an existing Eclipse SDK environment topic in the BMIDE for Data Model Design documentation on Support Center.

Install or update Dispatcher for setting up remote ECAD translation

Teamcenter viewers can only process neutral ECAD formats, and depending on the workflow of your environment, you may need to translate native ECAD PCB and schematic data to this neutral file format.

With ECAD translation, the design intermediate file from the Xpedition or PADS tool is converted into a neutral file format (*XFATF* for PCB designs and *XSCH* for schematic designs). The neutral file is saved as a viewable dataset in Teamcenter.

The following translation modes are supported for ECAD translation:

- Local translation where the viewable file is generated locally.

- Remote translation where the Teamcenter integration initiates a remote translation of the intermediate file into a viewable format.

You require Dispatcher for remote ECAD translations only. You can install Dispatcher as a standalone instance or in an existing Teamcenter environment. If Dispatcher is already installed in your existing Teamcenter environment, you must modify it to include the *PcbToFatF* ECAD translator.

For instructions on installing Dispatcher, refer to the Dispatcher — Deployment and Administration documentation on Support Center. After installing and setting up the dispatcher you must **enable the PcbToFatf translator**.

© 2025 Siemens

Setting up Teamcenter server for Xpedition EDM integration

Configurations required to integrate Teamcenter with the Xpedition EDM

To integrate Teamcenter with the Xpedition EDM, you must configure the following on the Teamcenter server.

Configuration	Why is it required?
Populate the Teamcenter database with ECAD components	Any component that does not already exist in Teamcenter is automatically created when saving the BOM.
Add EDA relations and EDA objects in Active Workspace Relation Browser	To traverse EDA relations and show EDA objects from the Active Workspace Relation Browser, EDA information must be included in the Active Workspace Relation Browser configuration.
Customizing ECAD attribute mappings	Using mapping definition files, the ECAD part attributes can be stored in the Teamcenter database and displayed and modified both in the ECAD design tool and Teamcenter.
Configure derived data for ECAD design objects	Derived data contains information that is derived from an ECAD design and comprises derived items and datasets. Derived items represent parts, subassemblies, and tools. Derived datasets manage data files created by ECAD applications.
Configure design variants	To design and develop a product with several *variations*, where each variation has different options or capabilities, you create design variants. Creating design variants avoids the need to create a unique version of the design for each variation. A variant uses the same base design, but the PCB assembly is loaded with the set of components specified by the variation. You can then select a variation when generating the design's manufacturing output (BOM, Pick and Place (P&P), assembly drawings), which will in turn determine how the product is assembled.
Allow ECAD designers to assign a Teamcenter project from the EDA client	While saving a new or an existing EDA design from the EDA client to Teamcenter, the ECAD designers can assign or remove a Teamcenter project for that design. The projects assigned to Printed Circuit Assembly (PCA) objects are propagated to all **ItemRevisions** and **Datasets** associated with the PCA object based on the Teamcenter propagation rule defined for that object. You can

© 2025 Siemens

Configuration	Why is it required?
	also modify the OOTB propagation rules as per your requirements.
Set configurations based on the ECAD applications used	You can configure which preference values should be used for a specific preference based on the connectors configured for your ECAD design tool. This configuration allows all EDA preferences to be overridden by application-specific preferences, if set.
Allow users to view the PCB board requirements in Active Workspace	By default, ECAD designers can view the PCB board requirements specifications related to the design revisions in the **Requirements** tab in Active Workspace for the **EDACCABase** and **EDASchem** object types, by default. Users can configure the **Requirements** tab for other item revisions.
(Optionally) Enable single sign-on for Teamcenter and Xpedition EDM server design integration	Teamcenter Security Services eliminates the need for multiple authentication challenges as users move from one Teamcenter application to another. It also provides a common framework to integrate with a site's *single sign-on* solution. With single sign-on configuration extended to the EDM server, users can log on to Teamcenter when they are logged on to the EDM client. Alternatively, they can log on to the EDM client automatically when the user is logged on to Teamcenter.
Set properties on EDM designs during design review	You can configure Teamcenter to initiate a workflow to allow ECAD designers to set properties on the EDM designs during the review process.
Customize the acronyms used for ECAD design objects in Teamcenter	If your ECAD designers use acronyms different from the default acronyms provided, the Teamcenter configurator can override the defaults by creating custom acronyms used for ECAD designs in Teamcenter.
Managing and configuring Teamcenter preferences for EDX design management	You can set system-wide preferences on the Teamcenter server that apply to all EDA users.

Populate the Teamcenter database with ECAD components

Many companies control the ECAD component library items that can be used to create BOMs by checking in the approved components for use to the database. Thereafter, you can create BOMs that use these components.

© 2025 Siemens

You can configure how to:

- **Control ECAD component library items for BOM creation.**

  To control component library items for BOM creation, when EDA is first installed, the **EDA_CheckComponentExistence** is set to **1** (true). With this setting, you cannot save design BOMs unless all the necessary components are already in Teamcenter.

- **Populate the database with ECAD components before the designer can save designs and before creating BOMs in Teamcenter using EDA.**

  You can initially populate the database with components by changing the **EDA_CheckComponentExistence** preference to **0** (false). If this preference is set to **0**, any component that does not already exist in Teamcenter is automatically created when saving the BOM.

  > Caution:
  >
  > Do not set this preference to **0** in a production environment.

  1. Set the **EDA_CheckComponentExistence** preference:

     a. Run the rich client on the Teamcenter server and log on as an administrator.

     b. In the My Teamcenter application, choose **Edit→Options**.

     c. At the bottom of the **Options** dialog box, click **Filters**.

     d. In the **Preferences** dialog box, search for the **EDA_CheckComponentExistence** preference.

     e. Click **Edit** and change the value of the preference from **1** (true) to **0** (false).

     f. Click **Save** to apply the changes.

  2. Locate or create designs that use the approved library components. Use the EDA **Save As** command to save the designs to the Teamcenter server and create BOMs. When the BOMs are created, the components are automatically added to the database.

     For information about using the Save As command, see *Saving designs to Teamcenter* in the *Xpedition and PADS Design Management* documentation.

  3. Reset the **EDA_CheckComponentExistence** preference to **1** (true) if you want to restrict users to using only the approved (and checked-in) components. This is the recommended setting in production environments.

     However, if you wish to allow new components to be automatically placed in the database when users create BOMs, leave the **EDA_CheckComponentExistence** preference as is, set to **0**.

© 2025 Siemens

This action bypasses business processes for component evaluation and approval.

Add EDA relations and EDA objects in Active Workspace Relation Browser

To traverse EDA relations and show EDA objects from the **Relations** tab in Active Workspace, EDA information must be included in the Relation Browser configuration. To do this, you must update the **RelationBrowserConf** dataset in the **RB_UIConfigure.xml** file in the Teamcenter server configuration.

To enable the **Relations** tab to show relations between Teamcenter objects and objects in a remote application, do the following:

1. In the Teamcenter rich client, search for the dataset **RelationB***.

2. Right-click the **RelationBrowserConf** dataset and choose **Check-InOut** > **Check-Out**.

3. Right-click **RelationBrowserConf** dataset and choose **Named References**.

4. In the **Named References** dialog box, select **RB_UIConfigure.xml** and click **Download**.

5. Modify the **RelationBrowserConf** dataset in the locally downloaded **RB_UIConfigure.xml** file to add EDA relations and EDA objects as follows.

```
=====================================================================
==
            Filename: RB_UIConfigure.xml

            File used to configure control panel of relation browser.
=====================================================================
==

 -->
        <RelationBrowser>
          <OptionPanel>
            <view name="General">
              <ruleName>GenericRule</ruleName>
              <defaultLayout>IncrementalHierarchic</defaultLayout>
              <group name="relations">

                <filter name="EDA" parameterSet="EDARelations"
color="(204,51,0)"/>
              </group>
              <group name="objects">

               <filter name="EDA" parameterSet="EDAObjects"
color="(204,51,0)"/>
```


© 2025 Siemens

```
                <filter name="Other" parameterSet="WorkspaceObject"
color="(238,236,225)"/>
               </group>
             </view>
             <view name="System">

                <parameterSet name="EDAObjects">
                   <clipsfacts>
                     <fact>EDACCABase</fact>
                     <fact>EDACCABaseRevision</fact>
                     <fact>EDASchem</fact>
                     <fact>EDASchem Revision</fact>
                   </clipsfacts>
                </parameterSet>
                  <parameterSet name="EDARelations">
                    <clipsfacts>
                      <fact>source=GRM,key=EDA,relationType-
=EDAHasDerivedDataset,targetDirection=forward,inputTypes=ItemRevision
</fact>
                      <fact>source=GRM,key=EDA,relationType-
=EDAHasDerivedItem,targetDirection=forward,inputTypes=ItemRevision</
fact>
                      <fact>source=GRM,key=EDA,relationType-
=EDAHasPWB,targetDirection=forward,inputTypes=ItemRevision</fact>
                      <fact>source=GRM,key=EDA,relationType-
=EDAHasSchematic,targetDirection=forward,inputTypes=ItemRevision</
fact>
                      <fact>source=GRM,key=EDA,relationType-
=EDAHasSchematic,targetDirection=backward,inputTypes=ItemRevision</
fact>
                      <fact>source=GRM,key=EDA,relationType-
=EDAHasSublayout,targetDirection=forward,inputTypes=Dataset,outputTyp
es=Dataset</fact>
                      <fact>source=GRM,key=EDA,relationType-
=EDAHasVariant,targetDirection=forward,inputTypes=ItemRevision</fact>
                      </clipsfacts>                     </
parameterSet>

             </view>
         </RelationBrowser>
```

6. Delete the original **RB_UIConfigure.xml** file from the **Named References** dialog box and import the new **RB_UIConfigure.xml** file in Teamcenter.

7. Click **Close**.

8. Check in the **RelationBrowserConf** dataset.

© 2025 Siemens

Customizing ECAD attribute mappings

1. Open a Teamcenter command prompt.

2. Change to the *bin* directory within TC_ROOT. (TC_ROOT is the installed location of Teamcenter.)

3. Run the **export_attr_mappings** command to output the mappings to a file, for example:

   export_attr_mappings –file=*test_attribute_mapping* **–u=***Tc-admin-user* **–p=***password*

4. Edit the mappings file to add EDA tool attributes.

   Below are the OOTB attribute mappings for EDM design integration:

```
{ Dataset type="EDADesExpBrd"
    "tc_PCA_ID" : Item.item_id /master=iman /description="CCA Item
ID"
    "tc_PCA_revision" : ItemRevision.item_revision_id /master=iman /
description="CCA Item Revision ID"
    "tc_PCA_description" : ItemRevision.object_desc /
description="CCA Item Revision Description"
    "tc_PWB_ID" : ItemRevision.GRM(EDAHasPWB).item_id /master=iman /
description="PWB Item ID"

"tc_PWB_revision" : ItemRevision.GRM(EDAHasPWB).item_revision_id /
master=iman /description="PWB Item Revision ID"
    "tc_PWB_description" : ItemRevision.GRM(EDAHasPWB).object_desc /
description="PWB Item Revision Description"
    "tc_exported_date" : last_mod_date /master=iman /
description="Last modified date"

}
```

   To define any additional mappings, add the mappings under this dataset type.

5. Import the edited mapping file back into Teamcenter by running the **import_attr_mappings** command, for example:

   import_attr_mappings –file=*test_attribute_mapping* **–u=***Tc-admin-user* **–p=***password*

© 2025 Siemens

Example:

The below example defines dataset attribute mappings and variant attribute mappings. In this mapping definition, the `Item` level attribute mapping definition defines the ECAD variant attribute mappings and it is part of the dataset level attribute mapping definition.

ECAD variant mapping for your ECAD tool:

```
{ Dataset type="EDADesExpBrd"
ORGNAME : Item.GRM(IMAN_master_form,Item Master).user_data_1
ORGADDR1 : Item.GRM(IMAN_master_form,Item Master).user_data_2
"DES_NAME" : ItemRevision.object_name
"DES_DESC" : ItemRevision.object_desc
Doc : Item.GRM(IMAN_master_form,Item Master).user_data_3

    { Item type="Item"
        "variant_def_list_tc_description" :
ItemRevision.object_desc /description="Variant Item Revision
Description"
    }
}
```

In above example, the attribute mappings are defined with respect to the dataset type **EDADesExpBrd**. Using this dataset attribute mapping functionality, it is possible to map an ECAD design attribute to a property with the Teamcenter object type. In this example, the ECAD attribute **ORGNAME** is mapped to **user_data_1** property of the item master form and **ORGADDR1** is mapped to **user_data_2** property of the item master form in Teamcenter.

In above example, the attribute mappings are defined with respect to the dataset type **EDADesAltiumBrd**. Using this dataset attribute mapping functionality, it is possible to map an ECAD design attribute to a property with the Teamcenter object type. In this example, the ECAD attribute **ORGNAME** is mapped to **user_data_1** property of the item master form and **ORGADDR1** is mapped to **user_data_2** property of the item master form in Teamcenter.

Similarly, the ECAD attribute **DES_NAME** is mapped to **object_name** property of **ItemRevision** and **DES_DESC** is mapped to **object_desc** property of **ItemRevision** in Teamcenter.

In the last line of above example, the attribute mappings are defined for the item type **Item**. However, these mappings are applicable only to the dataset type **EDADesExpBrd**. These mappings are not applicable for other dataset types. To define the attribute mappings between the ECAD variant attributes and the Teamcenter variant object properties, the attribute mappings must be defined on the **Variant Item** type. In above example, the ECAD variant attribute **variant_def_list_tc_description** is mapped to **object_desc** property on **ItemRevision** in Teamcenter.

The attribute mapping import and export process uses the PLM XML import/export mechanism. For more information, see the Tc XML and PLM XML Configuration for Data Import and Export help on Support Center.

© 2025 Siemens

For details on how to map ECAD attributes, see the Mapping attributes section in the *Teamcenter Integration for NX* documentation.

To understand the correct syntax and symantics for mapping attributes to standard and custom item types, see the Examples of Syntax and Symantics for Mapping Attributes topic in the *Teamcenter Integration for NX* documentation.

Configure derived data for EDX design files

Understanding how derived data works

Derived data contains information that is derived from an ECAD design and comprises derived items and datasets. Derived items represent parts, subassemblies, and tools. Derived datasets manage data files created by ECAD applications.

You can configure how derived data is created in Teamcenter EDA by using the **EDA Derived Data** folder in Business Modeler IDE, creating an EDA derived data configuration, and configuring how derived data files are named in Teamcenter. For example, a configuration can specify that when a schematic design is saved in Teamcenter EDA, a schematic drawing can be automatically generated from the schematic design and saved along with the schematic item

EDA business objects define the different types of derived data you can generate. To locate EDA business objects, use the **Find** button in the BMIDE view to search for all business objects containing the **EDA** string.

The following item types are children of the EDA business object:

Item Types	Description
EDACCABase	Represents the common electrical CAD (ECAD) design data that is shared between variant circuit card assemblies (CCAs). It is used only for multiple CCA representations.
EDAComp	Represents electrical components contained in the CCA bill of materials (BOM).
EDASchem	Represents the electrical schematic item.

The following relationships are children of the ImanRelation business object:

ImanRelation	Description
EDAHasDerivedDataset	Identifies the associated dataset as a derived dataset.
EDAHasDerivedItem	Identifies the associated item as a derived item.

© 2025 Siemens

Configure how derived data files are named in Teamcenter

Update the value of the **EDA_UseDerivedDataSubDirs** preference based on how you want the derived data files to be named in Teamcenter. The values are as follows:

- **True**

  The derived dataset name is the same as the Derived Dataset Configuration Name.

- **False**

  The derived dataset name consists of the parent item ID and an underscore (character) followed by the Derived Dataset Configuration Name.

How to create an EDA derived data configuration

You can configure how derived data is created in Teamcenter by using the **EDA Derived Data** folder in Business Modeler IDE. After creating a derived data configuration, you can specify the name of the configuration in the **EDA_DerivedDataConfigDefault** preference.

1. In BMIDE, open the **Extensions** folder.

2. In the **Extensions** folder, right-click **EDA Derived Data** and choose **New EDA Derived Data**.

   The **New EDA Derived Data** wizard is displayed.

3. In the **EDA Derived Data** dialog box, enter the following information and click **Next**:

Field	Description
Name	Specify a name you want to assign to the new derived data configuration. This is the name used in the **EDA_DerivedDataConfigDefault** preference.
Description	Type a description for the new configuration.

   The **EDA Derived Data Configuration** dialog box in the wizard is displayed.

4. In the **EDA Derived Data Configuration** dialog box, set up how all EDA item and dataset types are to be handled for all contexts.

   a. Click the **Add** button next to the **Configure Items** table.

      The **Add/Edit EDA Derived Item Configuration** dialog box is displayed.

© 2025 Siemens

In the **Add/Edit EDA Derived Item Configuration** dialog box, configure the derived items to be generated. For example, create separate rows for contexts such as schematic, PCB, simulation, and so on, including variations based on the what the parent is, such as **Schematic**, **CCA**, and **CCAVariant**. In this way, you set up how derived data is generated for all combinations of items.

In the **Add/Edit EDA Derived Item Configuration** dialog box, enter the following information and click **Finish**:

Field	Description
Name	Type the name that you want to assign to the derived item configuration. This is the name displayed to the user on the Teamcenter EDA **Derived Item** dialog box during save operations.
Context	**Browse** to select a specific Teamcenter EDA application context from the contexts listed below. When users in Teamcenter EDA save derived data for the following specified data types, derived items are generated according to this configuration. • all • pcb • pcb/simulation • schematic • schematic/pcb • schematic/simulation • simulation For PWB, if you have derived items and derived datasets defined for both schematic and PCB, you must set the preference **EDA_DefinePWBWith** to **schematicANDpcb** so that the PWB and derived data can be exposed and modified in both schematic and PCB tools. By default, the preference value is set to **Schematic**.
Prefix	(Optional) Type a file name string to be attached to the beginning of the parent item ID to distinguish it as being generated by this configuration. The resulting string, including the prefix and postfix, is used in the derived item user interface in Teamcenter

© 2025 Siemens

Field	Description
	EDA as the initial value for the **Derived Item ID** box and **Name** box. This can be overridden by the user.
Postfix	(Optional) Type a file name string to be attached to the end of the item ID to distinguish it as being generated by this configuration. The resulting string, including the prefix and postfix, is used in the derived item user interface in Teamcenter EDA as the initial value for the **Derived Item ID** box and **Name** box. This can be overridden by the user.
EDA Parent	**Browse** to select the derived parent EDA business object to which the derived item is related. (Teamcenter EDA does not support attaching derived items under other derived items). • **CCA** represents a circuit card assembly (CCA). • **CCABase** represents the common design data that is shared between variant circuit card assemblies (CCAs). It is used only for multiple CCA representations. • **CCAVariant** represents the variant design data for a circuit card assembly (CCA). This is the data that is used on top of the CCABase business object. • **PWB** represents a printed wire board (PWB). A PWB is the product of a schematic design and printed circuit board (PCB) layout design and holds all the printed wire board production data created by those designs. • **Schematic** represents the electrical schematic item.
Relation	**Browse** to select the relationship between the derived item and the parent item revision. The **EDAHasDerivedItem** business object and its children are displayed in the selection dialog box.
Item Business Object	**Browse** to select the business object type name for the derived item, for example, EDA.
Add to Bom	Select this option to add the derived item to the bill of materials of the Teamcenter EDA parent. This check box is disabled if the Teamcenter EDA parent is a schematic or a PWB.

The derived item configuration is added to the **Configure Items** table.

© 2025 Siemens

b. Click the **Add** button to the right of the **Configure Dataset** table.

The **Add/Edit EDA Derived Dataset Configuration** dialog box is displayed.

In the **Add/Edit EDA Derived Dataset Configuration** dialog box configure the derived datasets to be generated. For example, create separate rows for contexts such as schematic, PCB, and simulation, including variations based on the what the parent is, such as **Schematic**, **CCA**, and **CCAVariant**. In this way, you set up how derived data is generated for all combinations of datasets.

In the **Add/Edit EDA Derived Dataset Configuration** dialog box, enter the following information:

Field	Description
Name	Type the name that you want to assign to the derived dataset configuration. This is the name displayed to the user on the Teamcenter EDA **Derived Dataset** dialog box during save operations.
Context	**Browse** to select a specific Teamcenter EDA application context from the contexts listed below. When users in Teamcenter EDA save derived data for the following specified data types, derived datasets are generated according to this configuration. • all • pcb • pcb/simulation • schematic • schematic/pcb • schematic/simulation • simulation For PWB, if you have derived items and derived datasets defined for both schematic and PCB, you must set the preference **EDA_DefinePWBWith** to **schematicANDpcb** so that the PWB and derived data can be exposed and modified in both schematic and PCB tools. By default, the preference value is set to **Schematic**.

© 2025 Siemens

Field	Description
Prefix	(Optional) Type a file name string to be attached to the beginning of the parent item ID to distinguish it as being generated by this configuration. The resulting string, including the prefix and postfix, is used in the derived dataset user interface in Teamcenter EDA as the initial value for the **Derived Item ID** box and **Name** box. This can be overridden by the user.
Postfix	(Optional) Type a file name string to be attached to the end of the item ID to distinguish it as being generated by this configuration. The resulting string, including the prefix and postfix, is used in the derived dataset user interface in Teamcenter EDA as the initial value for the **Derived Item ID** box and **Name** box. This can be overridden by the user.
EDA Parent	**Browse** to select the derived parent EDA business object to which the derived dataset is related. In addition to the following item types, the list also includes item configurations you already created. • **CCA** represents a circuit card assembly (CCA). • **CCABase** represents the common design data that is shared between variant circuit card assemblies (CCAs). It is used only for multiple CCA representations. • **CCAVariant** represents the variant design data for a circuit card assembly (CCA). This is the data that is used on top of the CCABase business object. • **PWB** represents a printed wire board (PWB). A PWB is the product of a schematic design and printed circuit board (PCB) layout design and holds all the printed wire board production data created by those designs. • **Schematic** represents the electrical schematic item.
Relation	**Browse** to select the relationship between the derived dataset and the parent item revision. The **EDAHasDerivedDataset** business object and its children are displayed in the selection dialog box.

© 2025 Siemens

Field	Description
Dataset Business Object	**Browse** to select the parent dataset business object type in Teamcenter to represent the derived item, for example, PDF.
Dataset Reference	**Browse** to select the kind of file reference to use for the derived dataset. If the derived data instance comprises more than one file, this field must either be specified as a **ZIPFILE** type or must be specified using a separate derived dataset configuration entry with the same derived data name.
Pathname	Type the path where the derived dataset is to be saved on the user's machine. Path names are evaluated at run time and must be the fully qualified path of the dataset that is to be saved. Path names can be explicitly specified (for example, *D:\EDA\Datasets\readme.txt*) or formed using the variables or file name filters. Derived datasets can contain multiple files. Path names are case sensitive, and the directory delimiters of / or \ are used interchangeably.
Callback Name	Type the EDA callback name to execute. This name is used to identify the configured callback in the EDA configuration file to determine what script to execute. The script is responsible for creating or placing the corresponding derived files to be uploaded as specified by the configured source path name.

The derived item configuration is added to the **Configure Dataset** table.

c. Click **Finish**.

The derived data configuration is added under the **EDA Derived Data** folder

5. To save the changes to the data model, choose **BMIDE→Save Data Model**, or click the **Save Data Model** button on the main toolbar.

6. Deploy your changes to the test server. Choose **BMIDE→Deploy Template** on the menu bar, or select the project and click the **Deploy Template** button on the main toolbar.

7. In the rich client, set the **EDA_DerivedDataConfigDefault** preference to point to the EDA derived data configuration you just created.

8. Choose **Edit→Options**, click the **Search** link at the bottom of the **Options** dialog box, locate the **EDA_DerivedDataConfigDefault** preference, and change its value to the new configuration.

© 2025 Siemens

There may be multiple configurations created in the Business Modeler IDE, but an administrator can point to only one of them through this preference.

9. After deployment, test your new configuration in Teamcenter EDA.

   For example, in your ECAD design tool, choose **Teamcenter→Save Derived Data**. (You can also select the **Generate Derived Data** check box in the **Save As**, **Save**, or **Check In** dialog box.)

   To verify that the derived data is generated, in Teamcenter, expand the item that contains the derived data (for example, a CCA item). The derived dataset entries appear as you expand the tree structure. To see the contents of the derived dataset, right-click the dataset and choose **Named References**. A dialog box appears that shows the files that are contained in the derived dataset.

How to use the EDA derived data editor to configure derived data in Teamcenter

Use the EDA Derived Data editor in the BMIDE to work with derived data configurations used by the Teamcenter EDA application. Teamcenter EDA integrates Teamcenter with ECAD applications that are used to design electronic components, such as circuit boards.

1. To access the **EDA Derived Data** editor, open the **Extensions→EDA Derived Data** folder, right-click an EDA derived data object, and choose **Open**.

2. To add an item to the EDA derived data object, click the **Add** button next to the **Configure Items** table.

3. To add a dataset to the EDA derived data object, click the **Add** button next to the **Configure Dataset** table.

Configure EDA derived data for EDX design files

All the design files contained in an EDX design package can be classified as derived data files with the exception of BOM files, EDIF files, and ODB++ files. Derived data files may be placed anywhere within the package. Files that match your *EDA derived data* configuration are extracted from the EDX package to a temporary folder with the same structure as that in the package. For this, you must prefix your derived data path names with *$DESIGN*.

Example:

To save a file from your package at the **\Design01\Board\DWG\PDF*.pdf** path,

- Edit or create a new EDA derived data configuration using Teamcenter Business Modeler IDE. See the BMIDE help for more details.

- Add or edit a configure dataset entry.

© 2025 Siemens

- In the **Pathname** field, enter **$DESIGN\DWG\PDF*.pdf**

The **$DESIGN** keyword replaces the first two levels of the structure. Make sure you have the Teamcenter preference **EDA_DerivedDataConfigDefault** set to the name of the EDA derived data containing this dataset.

When an EDX file is published to Teamcenter, the files from the EDX package are saved in Teamcenter as derived files based on the derived data configuration.

Using wildcards in the derived data configuration path names

You can specify wildcard characters in the derived data configuration path name as follows:

Wildcard	Definition	Example
Asterisk (*)	Matches any number of characters including none.	**$DESIGN**.doc** Selects all **.doc** files from the child folder under the **$DESIGN** folder. **$DESIGN**PWB*.doc** Selects all **.doc** files with **PWB** in their file names from the child folders under the **$DESIGN** folder. **$DESIGN\PWB**.doc** Selects all **.doc** files that start with **PWB** in their file names from the child folder under the **$DESIGN** folder.
Two asterisks (* *)	Works like one asterisk (*) but crosses directory boundaries. This syntax is generally used to match complete paths.	**$DESIGN**\MyFile.doc** Selects the **Myfile.doc** file from all child folders under the **$DESIGN** folder.
Question mark (**?**)	Matches exactly one character.	**$DESIGN\Help\Help?.doc** Selects files with one character after their file name like **Help1.doc**, **Help2.doc** and **HelpA1.doc** but not **HelpMe.doc**.
Braces (**{ }**)	Specifies a collection of sub patterns.	**$DESIGN\Help{Me,You}*.pdf** Selects all **.pdf** files from either **HelpMe** or **HelpYou** folder.
Square brackets (**[ ]**)	Conveys a set of single characters or when a	**$DESIGN\Help[123]*.doc** or **$DESIGN\Help[1-3]*.doc**

© 2025 Siemens

Wildcard	Definition	Example
	hyphen (-) is used a range of characters.	Selects all **.doc** files any one of the folders **Help1**, **Help2**, **Help3**.

If multiple files from different folders match during save of the derived dataset, no files are saved for the derived dataset.

Example:

If your folder structure is like this:

$DESIGN\File1.doc

$DESIGN\Help1\File2.doc

$DESIGN\Help1\File3.pdf

$DESIGN\Help2\File4.doc

$DESIGN\Help2\File5.pdf

$DESIGN\Help2\File6.xls

The wildcards give the following results:

$DESIGN*.doc** will fail.

$DESIGN.pdf** will fail.

$DESIGN.xls** will succeed.

$DESIGN\Help1*.doc will succeed.

$DESIGN\File4.doc** will succeed.

$DESIGN*\File3.pdf will succeed.

You can also specify that the entire contents of a directory, including all sub-directories and files, are to be saved as a zip file. Specify the **DerivedDatasetConfiguration Dataset Reference** as **ZIPFILE**. Specify the **DerivedDatasetConfiguration Pathname** as a fully qualified directory path ending with the filename filter value **. Each directory of the path must be an existing directory, no wildcards are allowed. **Derived DatasetConfigurationPathname** variables such as **$DESIGN** are supported.

© 2025 Siemens

Example:

$DESIGN/manufacturingData/**

Configure design variants

What are design variants?

To design and develop a product with several *variations* of that product, where each variation has different options or capabilities, you create design variants. Creating design variants avoids the need to create a unique version of the design for each variation. A variant uses the same base design, but the PCB assembly is loaded with the set of components specified by the variation. A variation may then be nominated when generating the design's manufacturing output (such as BOM, Pick and Place (P&P), and assembly drawings), which will in turn determine how the product is assembled.

When you are working with ECAD design files, you can create any number of variations of the same base design, where each component may be configured differently as follows:

- **Fitted**

  This is the default state of a component if the component is fitted. It does not have variations. When you create a new variant, all components default to the state *fitted*.

- **Not Fitted**

  If a component is set to *not fitted*, it still exists on the schematic and is transferred to the PCB, but it is removed from the appropriate output documentation, such as the BOM.

- **Fitted with Varied Parameters**

  A component can have variations of any of its parameters, as part of the variant definition process. Modifying the value of a parameter is a local variation, only affecting the output documentation. The original schematic and the component whose parameter is being varied are not modified in any way.

- **Alternate Part**

  It is also possible to select an entirely different component as an *alternate part*. Since the alternate part is a different component, only one component is presented on the compiled schematic sheet. The alternate part must also share the same set of pins placed in the same locations as the base part. This is an essential requirement to ensure that the connectivity remains valid when the design is compiled.

An ECAD design is saved as a non-variant design for the first time in Teamcenter. Later, the variants of the ECAD design are added or created in Teamcenter as required. There are two scenarios for how variants are added.

© 2025 Siemens

- In the first scenario, you save the ECAD design as a non-variant design in Teamcenter. When new variants of the designs are created in the ECAD tool, you update the non-variant design in Teamcenter with the variants.

- In the second scenario, you save the non-variant ECAD design as a variant design in Teamcenter by creating dummy variants. When the variant designs are created in the ECAD tool, you replace the dummy variants in Teamcenter with the actual variant designs.

  You can **configure how to save a nonvariant ECAD design to Teamcenter as a variant design** using the EDA variant object model.

Configure ECAD design variants in Teamcenter

You can save a nonvariant ECAD design to Teamcenter using the EDA variant object model. To configure the saving of new nonvariant ECAD designs in Teamcenter as variant designs:

1. Update the value of the **EDA_SaveAsForceVariant** preference to **true**. This value specifies that nonvariant ECAD designs not previously saved in Teamcenter are to be saved in Teamcenter, using the EDA variant object model.

2. Update the value of the **EDA_FutureVariantName** preference. The default value is **futureVariant**.

   When you save a nonvariant design to Teamcenter and the value of the **EDA_SaveAsForceVariant** is **true**, the value of the **EDA_FutureVariantName** preference is used as the **Variant Name** in the **PCA Variants** dialog box. The value of the **Variant Name** in the **PCA Variants** dialog box cannot be changed.

Nonvariant ECAD designs already saved to Teamcenter using the EDA nonvariant object model are converted to use the EDA variant object model by the existing nonvariant-to-variant conversion functionality. This conversion occurs automatically when an ECAD design, previously saved to Teamcenter as a nonvariant design, is saved to Teamcenter after the first variant is added to the ECAD design.

Base BOMs do not support this functionality.

Allow ECAD designers to assign a Teamcenter project from the EDA client

While saving a new or an existing EDA design from the EDA client to Teamcenter, the ECAD designers can assign or remove a Teamcenter project for that design.

To make sure that the ECAD designer assigns a Teamcenter project while saving a design, you must set the **EDA_TCProjectRequired** preference to **true**. If this preference is set to **false**, the ECAD designer can save the design without assigning any project. The default value of the preference is **false**. If a default project is assigned to a user in Teamcenter, that project is assigned to the design by default. However, the user can assign additional projects while saving the design.

© 2025 Siemens

You must also assign the ECAD users to a project in Teamcenter to allow the users to select that project during the save operation. All projects associated with the ECAD users are displayed for selection to that user during the save operation.

A Teamcenter propagation rule is used for propagating security-related property values, such as project and license assignments, from a source business object to a destination business object. The projects assigned to Printed Circuit Assembly (PCA) objects are propagated to all **ItemRevisions** and **Datasets** associated with the PCA object based on the Teamcenter propagation rule defined for that object. You can also modify the OOTB propagation rules as per your requirements. For more information on how to modify an existing propagation rule, refer to the Configure your business data model in BMIDE on Support Center.

The OOTB propagation rules defined for EDA objects are as follows:

Direction	Source business object	Relation	Destination business object	Prop Group	Operation	Style	Details
Forward	**EDACCABaseRevision**	**IMAN_Rendering**	**Dataset**	Security Group I	All	Merge	Propagation rule between PCA **EDACCABaseRevision** and viewable **Dataset** in a variant design.
Forward	**EDACCABaseRevision**	**structure_revisions**	**PSBOMViewRevision**	Security Group I	All	Merge	Propagation rule between PCA **EDACCABaseRevision** and **PSBOMViewRevision** in a variant Design.
Forward	**ItemRevision**	**EDAHasSchematic**	**EDASchemRevision**	Security Group I	All	Merge	Propagation rule between PCA **ItemRevision** and **EDASchemRevision**.
Forward	**EDASchemRevision**	**IMAN_Rendering**	**Dataset**	Security Group I	All	Merge	Propagation rule between **EDASchemRevision** and

© 2025 Siemens

Direction	Source business object	Relation	Destination business object	Prop Group	Operation	Style	Details
							viewable **Dataset**.
Forward	**ItemRevision**	**EDAHasPWB**	**ItemRevision**	Security Group I	All	Merge	Propagation rule between PCA **ItemRevision** and **Printed Wiring Board**.
Forward	**EDACCABaseRevision**	**EDAHasVariant**	**ItemRevision**	Security Group I	All	Merge	Propagation rule between PCA **EDACCABaseRevision** and variant **ItemRevision**.
Forward	**ItemRevision**	**EDAHasDerivedItem**	**ItemRevision**	Security Group I	All	Merge	Propagation rule between any **ItemRevision** and derived **ItemRevision** in a design.
Forward	**ItemRevision**	**EDAHasDerivedDataset**	**Dataset**	Security Group I	All	Merge	Propagation rule between any **ItemRevision** and derived **Dataset** in a design.

You can also modify the OOTB propagation rules to create custom rules for variant and non-variant designs.

© 2025 Siemens

Create a custom propagation rule for variant designs:

Direction	Source business object	Relation	Destination business object	Prop Group	Operation	Style	Details
Forward	**ItemRevision** for the variant item type defined in the **EDA_CCAVariantItemTypeDefault** preference	**structure_revisions**	**PSBOMViewRevision**	Security Group I	All	Merge	Propagation rule between the variant **ItemRevision** and **PSBOMViewRevision**.

Create a custom propagation rule for non-variant designs:

Direction	Source business object	Relation	Destination business object	Prop Group	Operation	Style	Details
Forward	**ItemRevision** for the PCA item type defined in the **EDA_CCAItemTypeDefault** preference	**IMAN_Rendering**	**Dataset**	Security Group I	All	Merge	Propagation rule between the PCA **ItemRevision** and viewable **Dataset** in a non-variant design.
Forward	**ItemRevision** for the PCA item type defined in the **EDA_CCAItemTypeDefault** preference	**structure_revisions**	**PSBOMViewRevision**	Security Group I	All	Merge	Propagation rule between the variant **ItemRevision** and **PSBOMViewRevision**.

© 2025 Siemens

Set configurations based on the ECAD applications used

You can configure which preference values should be used for a specific preference based on the connectors configured for y ECAD design tool. This configuration allows all EDA preferences to be overridden by application-specific preferences, if set. The application specific preferences can be created for all the EDA design preferences by adding a suffix to the OOTB preference name. The suffix is the name of the ECAD tool application name exactly as mentioned in the *%TCEDAECAD_ROOT%* ***_usrdef.xml** file.

If the preference exists with the suffix of the connector name, then this preference is used. Else, the preference without a suffix is used.

Example:

Consider that you are using two different connectors, and want to set different values for the following preference, depending on which connector is currently being used. You can set different values for the same preference by adding the application name as a suffix to the preference name. The OOTB preference name is **EDA_CCAVariantItemTypesAllowed**.

To set the preferences with different values for both the connectors, create two different preference instances with the connector names suffixed to the preference.

- Preference to be used if the connector 1 is used:
  EDA_CCAVariantItemTypesAllowed_<connectorname_1>
- Preference to be used if the connector 2 is used:
  EDA_CCAVariantItemTypesAllowed_<connectorname_2>

Allow users to view the PCB board requirements in Active Workspace

ECAD designers can view the PCB board requirements specifications related to the design revisions in the **Requirements** tab in Active Workspace for the **EDACCABase** and **EDASchem** object types, by default.

To allow users to configure **Requirements** tab for other Item Revisions, user can include the following in the respective stylesheet.

```
<page titleKey="tc_xrt_Requirements"
visibleWhen="ActiveWorkspace:SubLocation !=
com.siemens.splm.client.occmgmt:OccurrenceManagementSubLocation">
        <htmlPanel declarativeKey="Eda0RequirementsTraceability" />
</page>
```

Additionally, you can specify the revision rule to configure the requirements BOM structure in the **Requirements** tab by setting the **EDA_AWRequirementBOMRevRule** preference. The default value of the revision rule is set to **Latest Working**.

© 2025 Siemens

Configure single sign-on for Teamcenter and Xpedition EDM integration

Teamcenter Security Services eliminates the need for multiple authentication challenges as users move from one Teamcenter application to another. It also provides a common framework to integrate with a site's *single sign-on* solution. With single sign-on configuration extended to the EDM server, users can log on to Teamcenter when they are logged on to the EDM client. Alternatively, they can log on to the EDM client automatically when the user is logged on to Teamcenter.

Single sign-on is supported for Xpedition EDM and Teamcenter. To enable SSO for the integration, you must:

- **Set up the SSO server**

  To set up the SSO server, refer to *Security Services Installation/Customization* in the *Teamcenter* documentation.

- **Set up the EDM server for SSO support**

  To set up the EDM server for SSO, refer to the *Teamcenter Integration* section in the *Xpedition® EDM Server and Utilities User's Guide* help on Support Center.

- **Set up the integration for SSO support**

  When installing the Teamcenter EDA client, on the EDA Client wizard, select the **Enable SSO** check box and navigate to the **Security Services** wizard.

  Here, specify the Teamcenter login service in the **Login URL** field and the Teamcenter SSO application ID in the **Application ID** field.

Set properties on EDM designs during design review

You can configure Teamcenter to initiate a workflow to allow ECAD designers to set properties on the EDM designs during the review process.

To initiate this workflow during the review of EDM designs, you must:

1. Set the **EDA_Workflows** preference value to initiate a workflow during the review process.

2. Configure the **EDM-set-properties** workflow handler to apply the property-value pair on the EDM designs.

© 2025 Siemens

1. Set the EDA_Workflows preference value to initiate the workflow during the review process

You must specify a Teamcenter workflow that must be initiated during the EDM design review process in the **EDA_Workflows** preference. If this preference value is not set, the workflow is not initiated during the EDM review process.

The **EDA_Workflows** preference specifies the Teamcenter workflow template to be initiated on the CCA **ItemRevision** based on the ECAD design status and the purpose of the review on that design. Specify the value of the preference in the following format:

Design_Status;Design_Intent;TC_Workflow_Template where,

- **Design_Status** is the release status applied to the ECAD design.
- **Design_Intent** is the purpose of review for the ECAD design.
- **TC_Workflow_Template** is the name of the Teamcenter workflow template.

Example:

The value of the **EDA_Workflows** preference can be set as **Released;Release;EDM Set Properties** where

- **Released** is the release status applied to the EDM design.
- **Release** is the design intent.
- **EDM Set Properties** is the Teamcenter workflow template.

2. Configure the EDM-set-properties workflow handler

Handler name: **EDM-set-properties**

Argument	Value
-EDMInboxQueue	This is the Inbox Queue Path location configured in the EDM Server
-CONTAINER:PROPERTY	**VALUE**

- **CONTAINER**

  This is the EDM container type. Its valid values are **DC**, **LDC**, **PDC**, and **PLC**.

Valid values	Examples	Value considerations
DC	Board Container	Any valid EDM property can be updated.

© 2025 Siemens

Valid values	Examples	Value considerations
		Example: **-DC:tc_PCA_ID**
LDC	Logical Design Container	Any valid EDM property can be updated. Example: **-LDC:tc_PWB_revision**
PDC	Physical Design Container	Any valid EDM property can be updated.
PLC	Part List Container	Any valid EDM property can be updated.

- **PROPERTY**

  This is the EDM property name, for example, **tc_PCA_ID**, **tc_PCA_revision**, or **tc_PWB_ID, tc_UID**.

  You can update multiple containers and properties at a time. However, you cannot update containers not present in the original EDX file attached to the design dataset.

- **VALUE**

  This is the value to be set on the EDM container property. This value can be of two types:

  - **Direct value**

    The actual value is passed to the handler for the container property.

  - **Indirect value**

    The valid format is **TCOBJECT:TCPROPERTY**

Format	Description	Valid values
TCOBJECT	It is the Teamcenter object (item) from which a property value should be read.	**PCAITEM, PCAITEMREV, PWBITEM, PWBITEMREV**
TCPROPERTY	It is the property to be read on the given **TCOBJECT**.	object_name

- **PLACEMENT**

  Requires no specific placement. Proper placement depends on the desired behavior of the workflow process and may require coordination with the placement of other handlers, especially in cases where other handlers depend on the results of **EDM-set-property**. Typical placement might be on the **Start** action or **Complete** action.

Additionally, the **EDXClient** utility is also supported for the workflow handler to make sure that the EDX file is copied to the inbox queue path.

© 2025 Siemens

Note:

The **edxclient.exe** is installed on the Teamcenter server by installing the **Xpedition Data Management** > **Xpedition EDM Engineer** client product. Refer to the Xpedition help on Support Center for more details.

The configname property is defined using the **EDM Batch Admin** utility to create a secure login method to EDM, which is used during the handler execution. Because the login configuration is stored in the *HOME* directory of the user that executes the **EDM Batch Admin** utility, the utility should be executed using the same credentials that are used for the Teamcenter Server Manager as this service executes the workflow handler.

To use this utility, the workflow handler must include these additional parameters:

Parameter	Description
-EDXClientPath	This is the absolute path of **edxclient.exe** Example: *%SDD_HOME%***\common\win64\bin\edxclient.exe**
-EDMConfigName	This is the EDM configuration name for **edxclient.exe**. This configuration is created in the EDM batch login.
-EDXQueueName	This is the EDM queue name configured in the EDM server.

If you want to use the **EDM-set-properties** handler in your workflow and, if an argument is not passed or if the value of an argument is empty, then the handler will pick the value set in the preferences available for each of these parameters. You can set the values of the parameters in the preferences listed below:

Parameter	Preference
-EDXClientPath	**EDA_EdxClientPath**
-EDMConfigName	**EDA_EdmConfigName**
-EDXQueueName	**EDA_EdxQueueName**

Customize the acronyms used for ECAD design objects in Teamcenter

If your ECAD designers use acronyms different from the default acronyms provided, the Teamcenter configurator can override the defaults by creating custom acronyms used for ECAD designs in Teamcenter.

By default, EDA uses the following acronyms to refer to electronic design objects on the user interface:

- **PCA** for printed circuit assembly or **CCA** for circuit card assembly (in earlier versions of EDA).
- **PWB** for printed wire board.

© 2025 Siemens

If your ECAD designers use different acronyms, the Teamcenter administrator can override the defaults by creating custom text copies of the **TC_ROOT\lang\textserver***locale***\eda0_text_locale.xml** file. For each *custom-name***_text_locale.xml** file, locate the following lines in the file and edit the display text as required:

```
<key id="Eda0_PCA">PCA</key>
<key id="Eda0_PCAToolTip">Printed Circuit Assembly</key>
<key id="Eda0_PWB">PWB</key>
<key id="Eda0_PWBToolTip">Printed Wire Board</key>
```

> Note:
>
> There is a separate locale file for each supported locale. The example shown is from the **en_US** locale file.

For more information about customizing textserver file text, see the *Server Customization* help.

Managing and configuring Teamcenter preferences for design management

You can set system-wide preferences on the Teamcenter server that apply to all EDA users. To understand how preferences work and to understand how to access a list of all supported preferences, refer to the Managing Preferences help on Support Center. All EDA preferences begin with the prefix **EDA**.

1. Run the rich client on the Teamcenter server and log on as an administrator.
2. In My Teamcenter, choose **Edit→Options**.
3. At the bottom of the **Options** dialog box, click **Index**.
4. In the **Preferences** dialog box, search for the preferences that begin with **EDA_**.
5. Change the preference value and click the **Modify** button to save the new value.

Install the EDA client for EDM integration

Use the **tem.bat** command from the Teamcenter software kit to launch Teamcenter EDA Environment Manager (TEM) to perform installation and maintenance operations of Teamcenter EDA on an ECAD workstation for EDM design integration.

TEM is available with the Teamcenter installation or upgrade images. Contact your Teamcenter installation administrator to obtain the correct product distribution image.

© 2025 Siemens

> Caution:
>
> Before installing the EDA client on your ECAD workstation, refer to all the prerequisites and the information required to install the client in the **Planning your deployment** topic.

1. Ensure that the proper version of JRE is installed and the JRE_HOME environment variable (32-bit system) or the JRE64_HOME environment variable (64-bit system) is set.

2. Extract the Teamcenter software distribution image to a temporary location.

3. Specify the path to the Java Runtime Environment (JRE) by setting the **JRE_HOME** environment variable on the ECAD workstation.

   You can also specify the JRE path when you launch TEM from the command line using the **-jre** path argument.

4. Start TEM.

   a. Browse to the root directory of the extracted Teamcenter EDA software distribution image.

   b. Right-click the **tem.bat** program icon and choose **Run as administrator**.

      TEM launches and displays the **Installer Language** panel.

   c. In the **Installer Language** panel, select a language and click **OK**.

      TEM displays the **Welcome to Teamcenter EDA** panel.

      Your language selection applies only to the TEM session and not the Teamcenter EDA installation.

5. In the **Welcome to Teamcenter EDA** panel, click **Install**.

6. In the **Configuration** panel, type the configuration ID and the description for the new Teamcenter EDA configuration.

   The configuration ID identifies your Teamcenter configuration when you maintain, upgrade, uninstall, or add features to the configuration. Installation log files are also named based on this ID.

7. Proceed to the **Features** panel and click **Next**.

8. Expand **EDA Client** and select the required options:

   - **ECAD Design Integrations**

© 2025 Siemens

Select the design tool support you require. This integrates the design tool with the EDA client. However, a separate installation of the design tool is required.

- **Mentor Xpedition EDM Design**
- **Teamcenter Gateway for EDA Design**
- (Optional) **EDA Translation**

9. In the **Installation Directory** box, browse to a folder where the EDA client will be installed. For example, type **c:\EDA_client** for Windows.

   The client is installed to an **eda** subdirectory, for example, **c:\EDAC_client\eda** (Windows). This directory is known as *TCEDAECAD_ROOT*.

10. Click **Next**.

11. In the **FCC Client Cache (FCC)** panel, select **Use new FCC** and click **Next**.

12. In the **FCC Parents** panel, type the **Host** name of the Teamcenter server in the **FCC Parents** box.

13. In the **4-tier Server Configurations** panel, type the IP address of the Teamcenter server in the **4-tier Servers** box. For example:

    http://*host***:***7001***/tc**

    Replace *host* with the server name.

14. In the **EDA Client** panel:

    a. Change the path if needed in the **Staging Location** box.

       You must enter the full path to the directory where the ECAD designs are saved on the client. This directory is the working directory for EDA.

       If a staging directory already exists, you can continue using the same directory or select another folder as the staging directory.

       > Caution:
       >
       > Do not use spaces in the cache or staging directory path. Some EDA functions do not work if there are spaces in these paths.

    b. Clear the **Will this configuration be shared by multiple users?** check box to create user environment variables after installation. By default, the check box is selected to create system environment variables.

© 2025 Siemens

c. Click **Next**.

15. Review the features selected in **Confirmation** panel and click **Start**.

    The EDA client components are installed.

    The **Install** panel confirms whether the installation is successful. If the installation is successful, click **Close**. If the installation is unsuccessful, click **Show Details** and proceed as needed.

After installing the EDA client, you must **integrate Xpedition EDM with Teamcenter**, **configure the EDA client** and **configure the connectors** for the Teamcenter integration to work correctly.

Setting up EDM integration

Configurations required for setting up EDM integration

After installing the EDA client and integrating Xpedition EDM with Teamcenter, you must:

- **Integrate Xpedition EDM with Teamcenter** .
- **Configure and deploy the EDM server with Teamcenter**.
- **Perform EDM queue configuration**.
- **Create an EDM baseline profile to enable automation**.
- **Set automatic import options**.
- **Export X-Y coordinates and rotation details of the EDX file into Teamcenter**
- **Retain EDX files after publishing to Teamcenter**.
- **Configure the connector for EDM integration**.

Integrating Xpedition EDM with Teamcenter

To integrate Xpedition EDM with Teamcenter, consider the following information and requirements after installing the EDA client:

- You must have *write* permissions to create a file in the *%TCEDAECAD_ROOT%* folder.
- The **API.properties** file is created after installing the EDA client. This file contains the properties required to use the **Mentor EDX SDK** API.

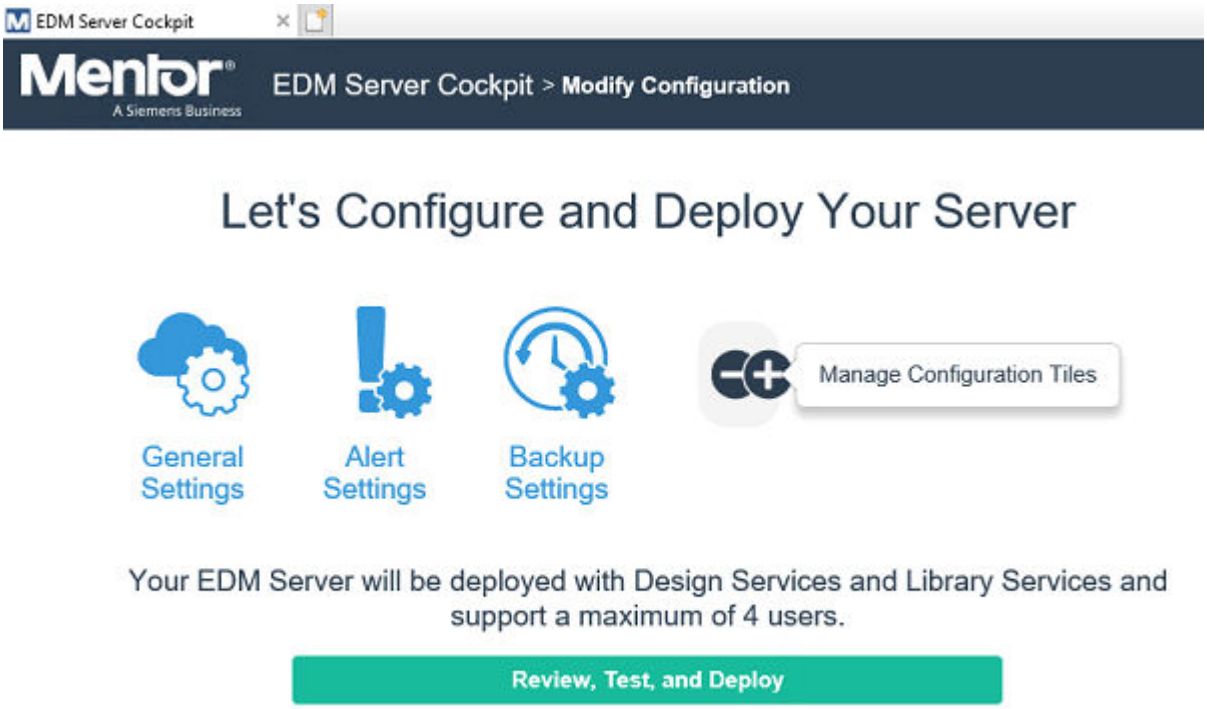
© 2025 Siemens

- The **SDD_HOME** environment variable should be set to **Xpedition** installation location in the following files:
  - *%TCEDAECAD_ROOT%***\MentorEDXDesCon\eda_edx_connector.bat**
  - *%TCEDAECAD_ROOT%***\MentorEDXDesCon\eda_edx_connector_singlejvm.bat**

Configure and deploy EDM server with Teamcenter

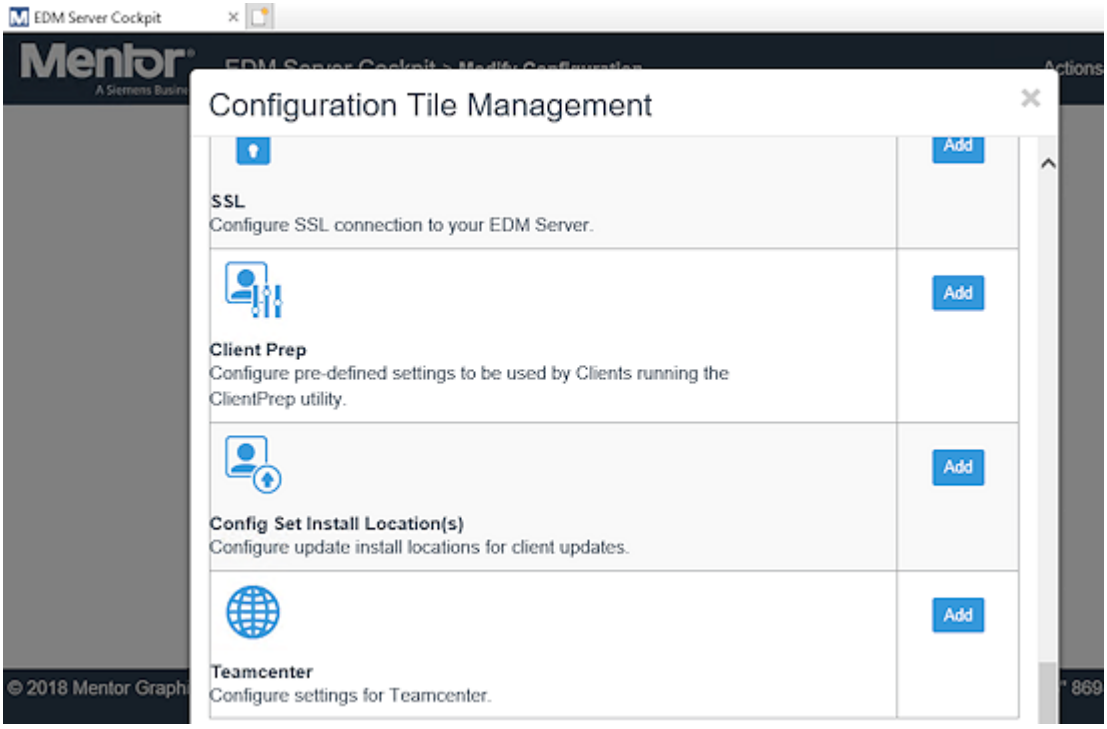
The EDM server administrator can review all current configurations, modify and add to the existing configurations, run pretests, and then deploy the EDM server with Teamcenter configuration.

1. Open the **EDM Server Cockpit** tool.
2. **If deploying EDM server for the first time:**
   - If the EDM server is not previously deployed, you can **Configure and Deploy** it by specifying the Teamcenter configurations.
     a. Click **Manage Configuration Tiles** to add the **Teamcenter** application.



© 2025 Siemens

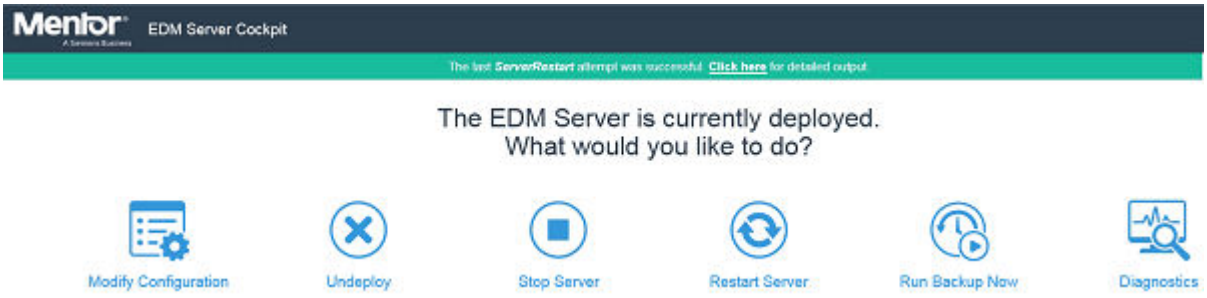
b. In the **Configuration Tile Management** dialog box, click **Add** against the **Teamcenter** icon to add and configure the Teamcenter application.



c. Click the **Teamcenter** tile.

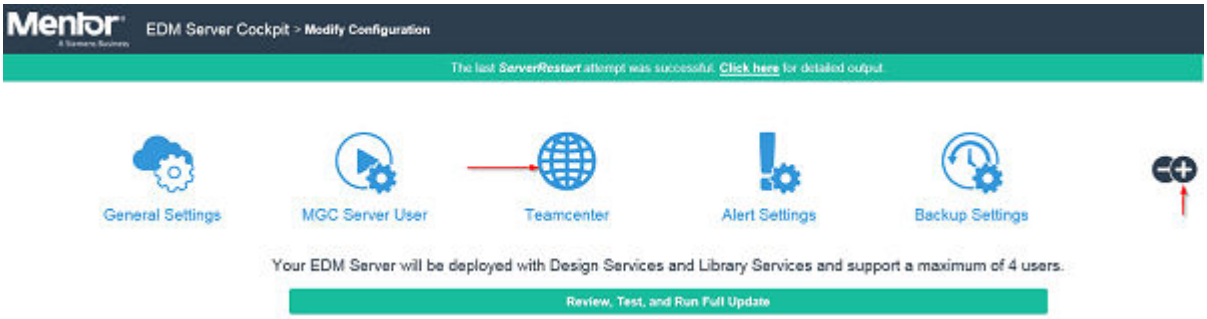
3. **If EDM server is previously deployed:**

a. If the EDM server is already deployed, you can modify it to add Teamcenter configurations and deploy it once again.



b. Click **Modify Configuration**.

© 2025 Siemens



c. Click the **Teamcenter** tile if it is already added. If it is not added, in the **Configuration Tile Management** dialog box, click **Add** against the **Teamcenter** icon to add and configure the Teamcenter application.

4. On the **Teamcenter** application page, click **Yes** to **Enable Teamcenter Integration**.

5. In the **Teamcenter Configuration Zip Path** field, enter the location of **EDXConfigFiles.zip** file.

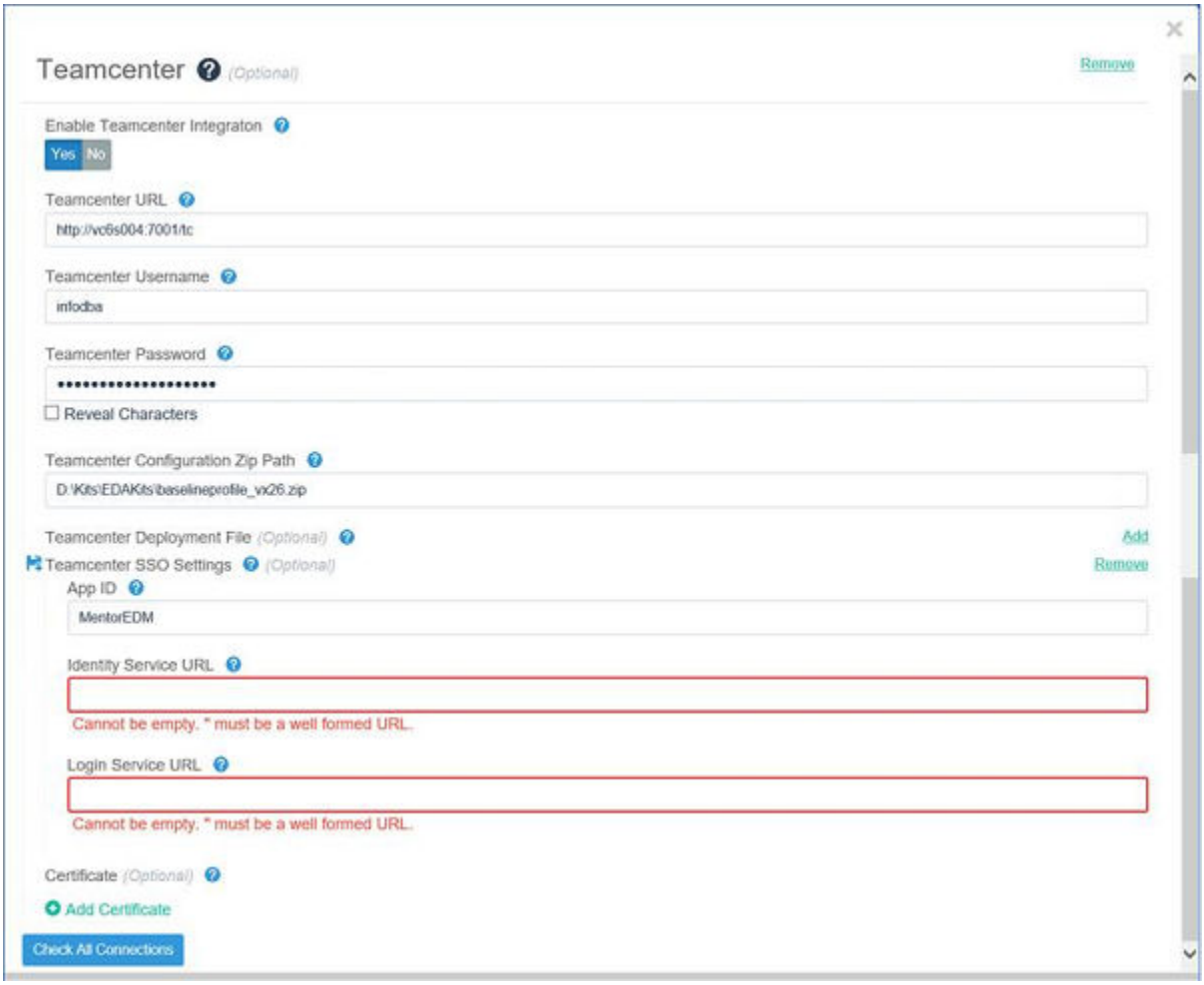
   This file is located at *%TCEDAECAD_ROOT%***\MentorEDXDesCon\server_updates** location.

   The zip file contains the following xml files:

   - **MetaData.xml**: It contains the new metadata created on EDX containers for EDX integration.
   - **SystemConfigurations.xml**: It contains the new callback definitions created for saving EDX to Teamcenter.

6. If the single sign on feature is enabled on the Teamcenter server, to use single sign-on for Teamcenter EDA integration, provide the following information in the **Teamcenter SSO Settings** fields:

© 2025 Siemens



a. **Teamcenter URL**

Teamcenter EDM integration uses the 4-tier URL configured on the EDM server to connect to Teamcenter. This value overrides the URL defined in the *%TCEDAECAD_ROOT%* **\TCEDAClient.properties** file on the client machine.

> Note:
>
> This change is applicable only when Teamcenter is configured in the non-SSO mode on the EDM server. The parent FSC address configured in the *%FMS_HOME%***\fcc.xml** file on the client machine will be in sync with the 4-tier URL configured on the EDM server.

b. **Identity Service URL**

c. **Login Service URL**

© 2025 Siemens

> Note:
>
> The URL information can be obtained from the Teamcenter administrator.

7. Click **Save** to save the configuration.

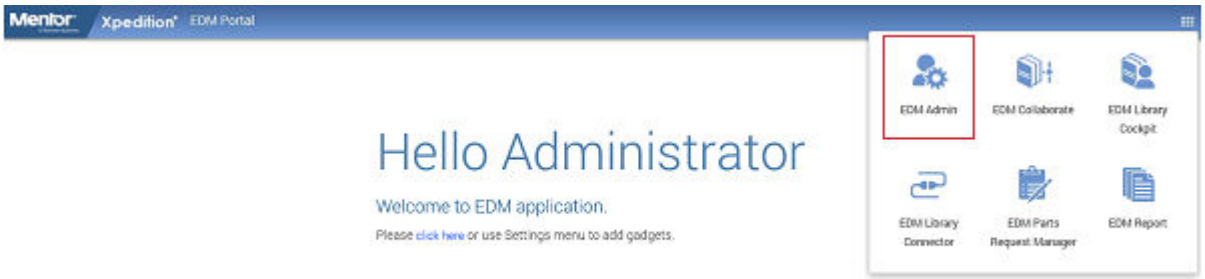
   The Teamcenter configuration tile will be added on the Modify Configurations page.

8. Click **Review, Test, and Deploy** on the main page to update the configuration and deploy the EDM server.

EDM queue configuration for Teamcenter EDA integration

1. Click the **http://<EDM_SERVER_HOST>:<PORT>/xdm** URL to open the **EDM portal** web application, where:

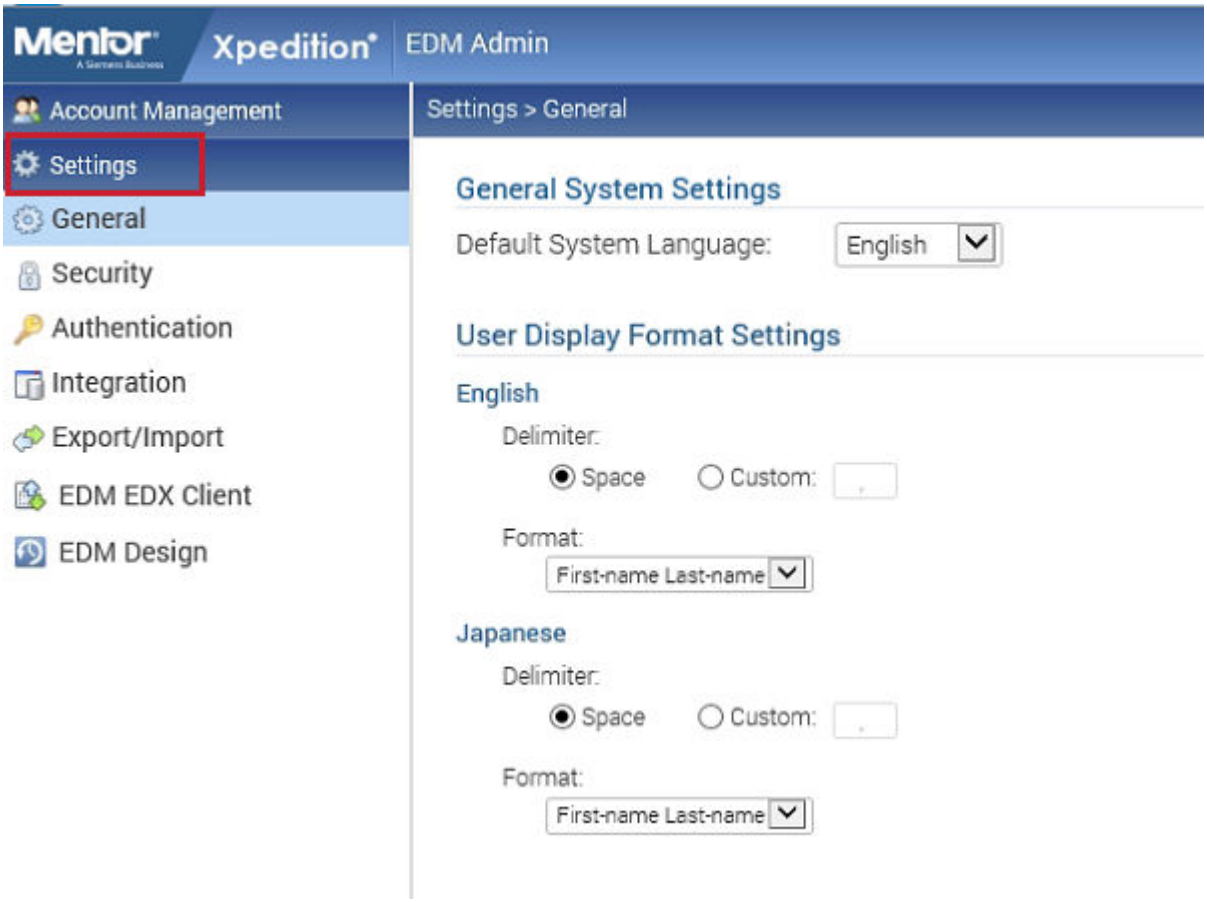
   - **EDM_SERVER_HOST** is the host name of the EDM server where it is installed.
   - **PORT** is the port number on which the EDM server is running, for example, **http://** *edm_server_machine_name***.net.plm.eds.com:31000/xdm**.



2. On the **Xpedition EDM Portal** page, click **EDM Admin**.

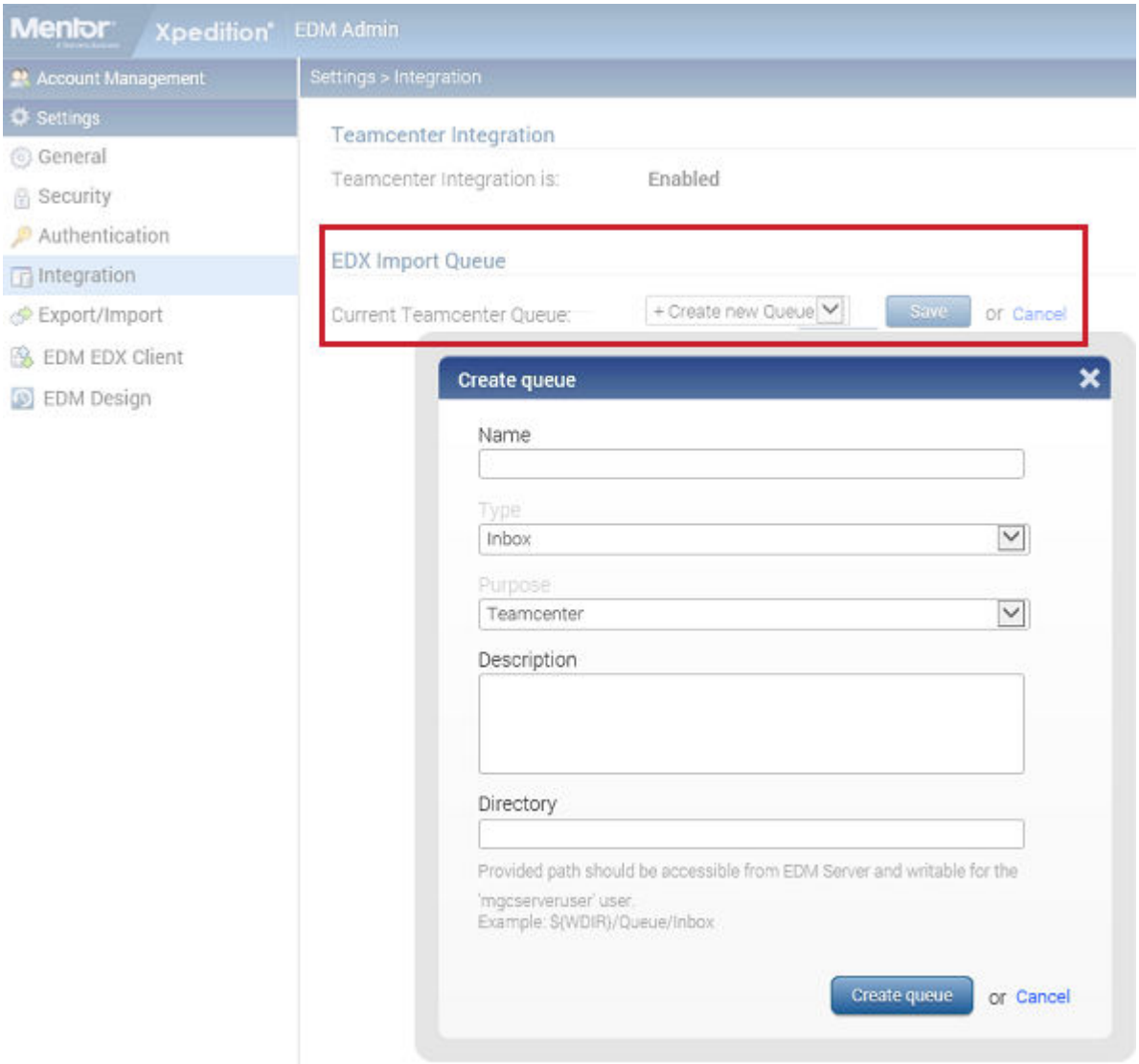
3. Click **Settings** on the navigation panel.

© 2025 Siemens



4. Click **Integration** under **EDX Import Queue**, and select **+ Create new Queue** from the **Current Teamcenter Queue** list.

© 2025 Siemens



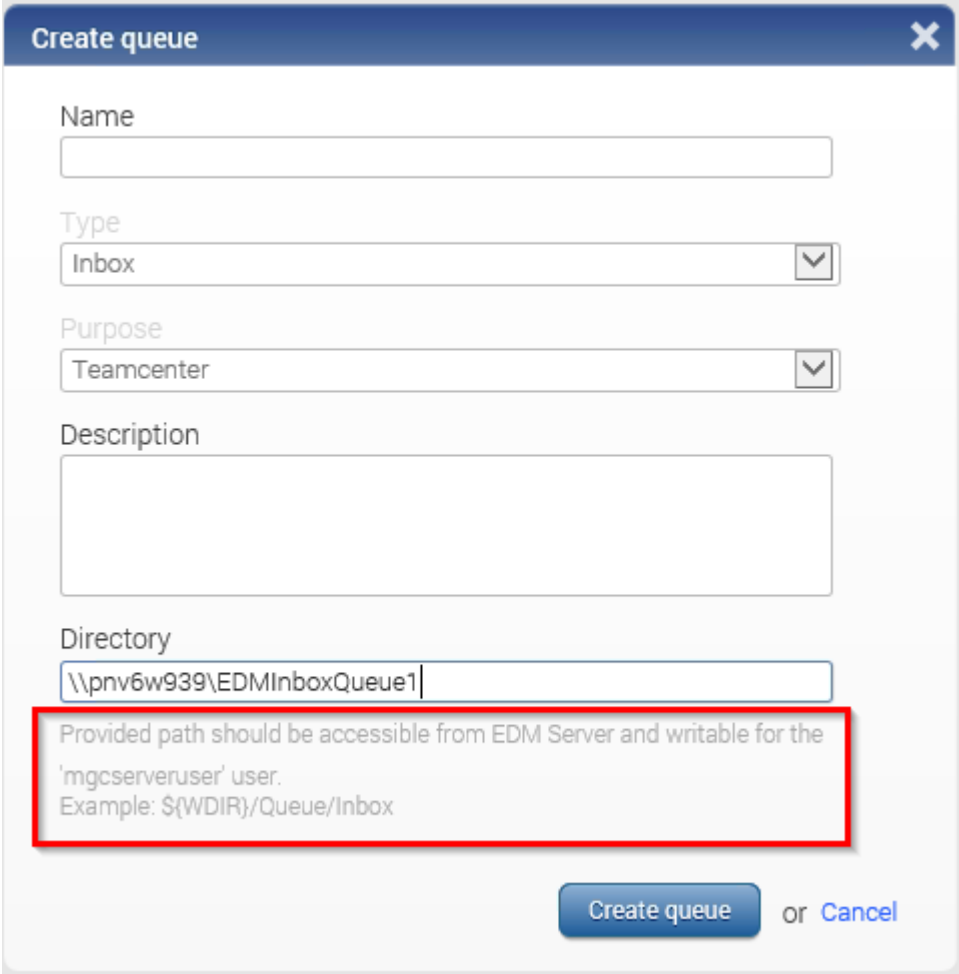
5. In the **Create queue** dialog box, enter a name for the queue in the **Name** field, select **Inbox** from the list for **Type**, and for **Purpose** select **Teamcenter**.

6. In the **Directory** field, provide the directory path for the queue.

   The directory path that you specify here must be accessible from the EDM *Server*. Additionally, the *mgcserveruser* user must have *write* permissions to the inbox queue folder.

   As you are not required to expose the Inbox Queue to the client machine, it is not required that the Inbox Queue is a shared folder. The EDX files are imported to Inbox Queue in a secure manner using the EDX client utility.

© 2025 Siemens



7. Click **Create queue**.

Create an EDM baseline profile to enable automation

Import the baseline profile configuration

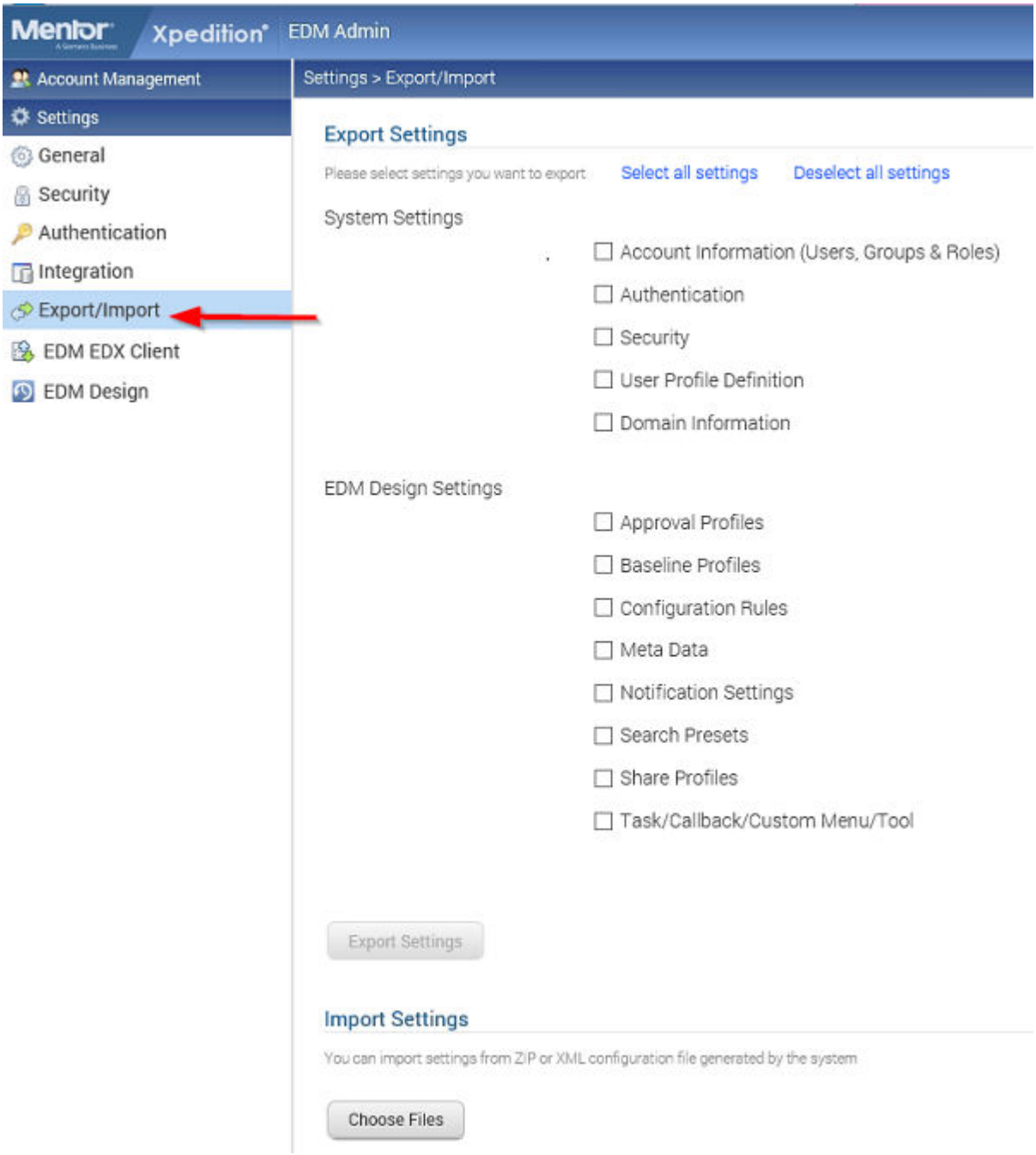
The baseline profile configuration file for publishing EDX file to Teamcenter is available in the **baselineprofile.zip** file located at *%TCEDAECAD_ROOT%***\MentorEDXDesCon\server_updates** location.

This ZIP file contains the **LatchProfiles.xml** file. This file contains the *Publish to Teamcenter* baseline profile to define the part list options for generating design and variant BOM. This is also required to automate the process of creating the baseline and the EDX file. The ZIP file also contains a supporting file that describes the source option for part list generation.

To import the baseline profile:

1. Open the **EDM Admin** web tool.

© 2025 Siemens

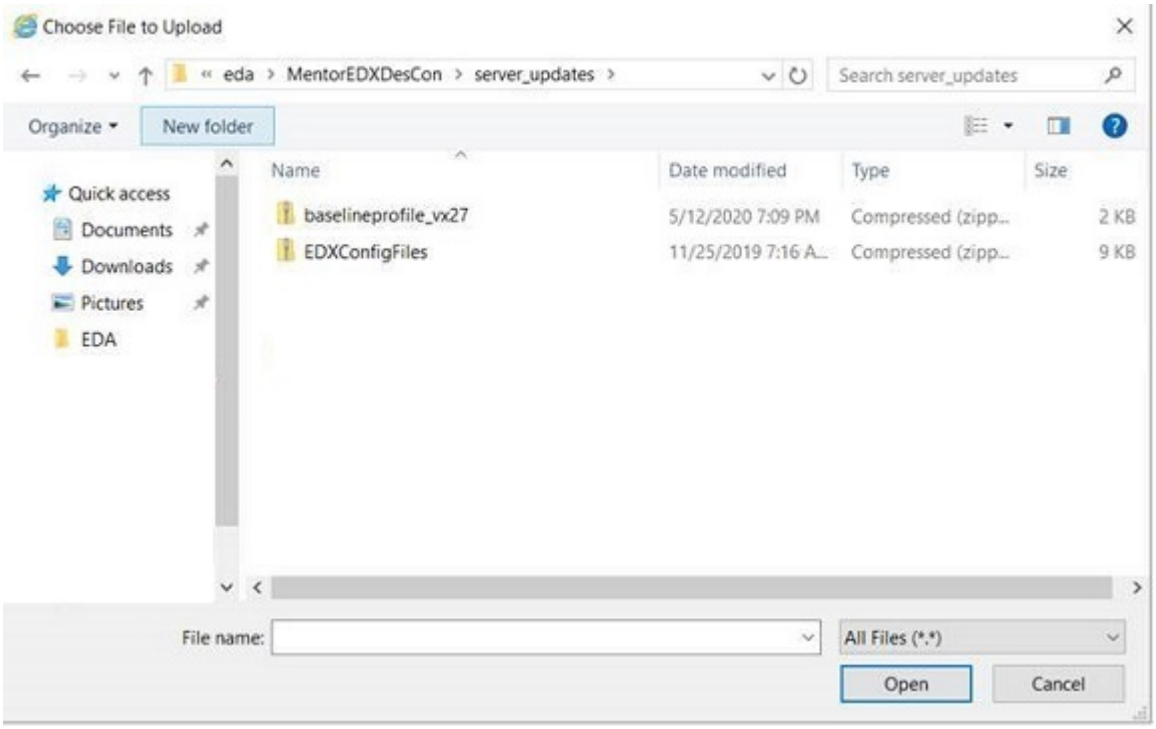


2. Click **Settings→Export/Import**.

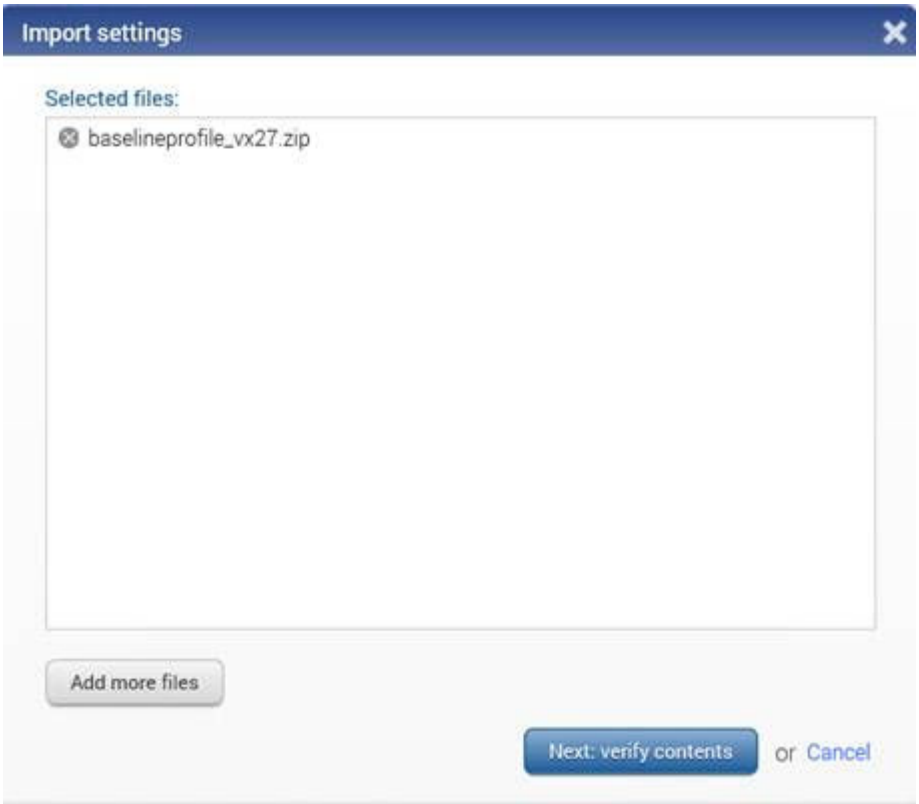
3. Click **Choose Files** under **Import Settings**.

4. Select the **baselineprofile_*.zip** file from the *%TCEDAECAD_ROOT%* **\MentorEDXDesCon\server_updates** location.

© 2025 Siemens

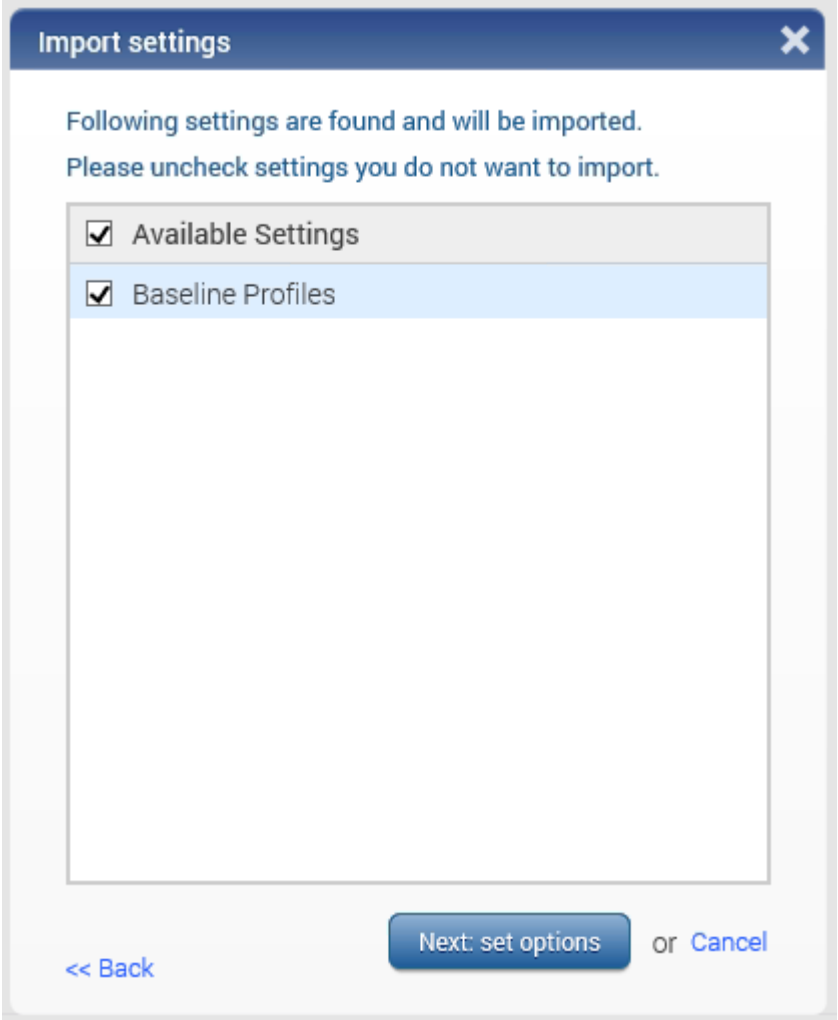


5. Click **Next: Verify Contents** to verify the selected files.



6. Click **Next: set options**.

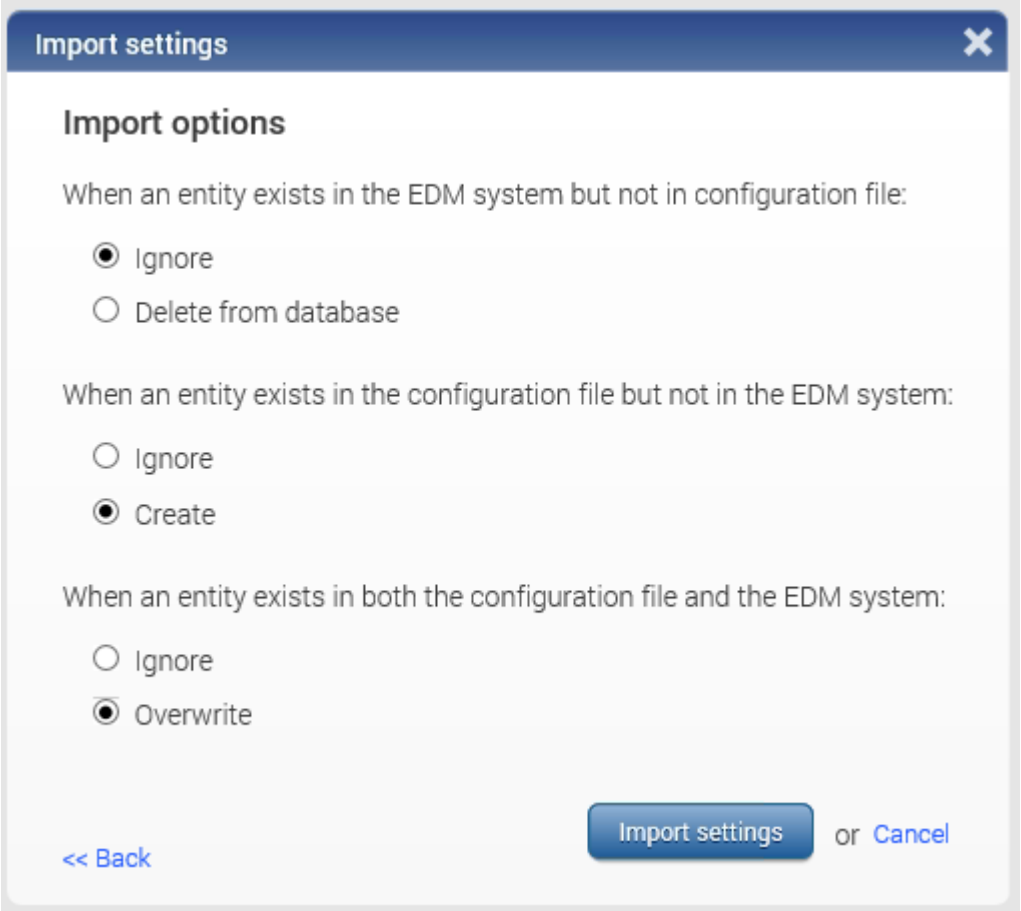
© 2025 Siemens



7. In the **Import settings** dialog box, select the following option for each scenario:

- **Ignore**, when an entity exists in the EDM system but not in the configuration file.
- **Create**, when an entity exists in the configuration file but not in the EDM system.
- **Overwrite**, when an entity exists in both the configuration file and the EDM system.

© 2025 Siemens



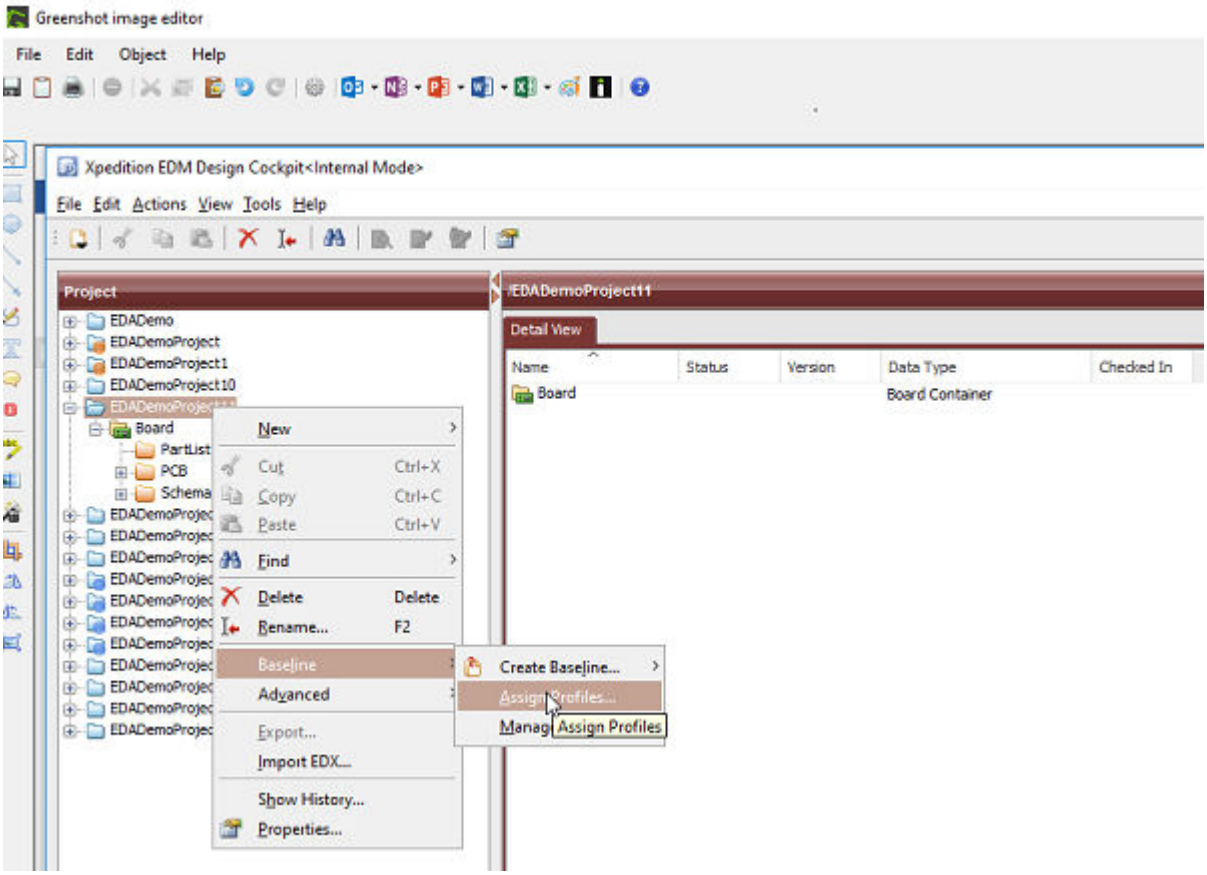
8. Click **Import settings** to import the baseline profile. This creates a baseline profile with the name **Publish to Teamcenter** in EDM.

How the baseline profile works

The baseline profile helps in reducing the number of manual steps such as creating a baseline and the manual inputs required in the **Create EDX** dialog box. It also allows you to configure which files should be included in the EDX file and to include the part list information for saving BOM information in Teamcenter.

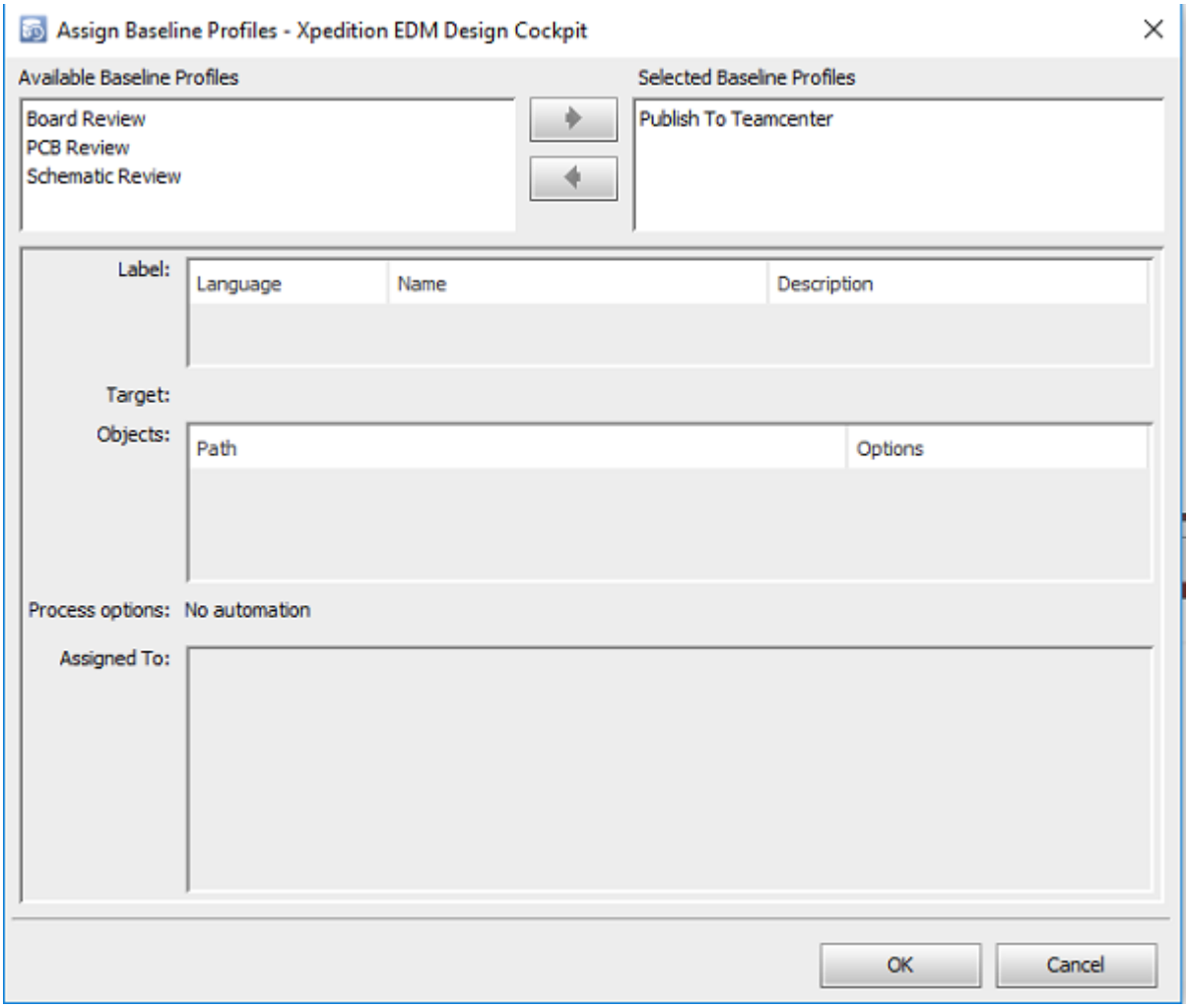
1. In the **EDM Design Cockpit** tool, select a project and assign the baseline profile.

© 2025 Siemens



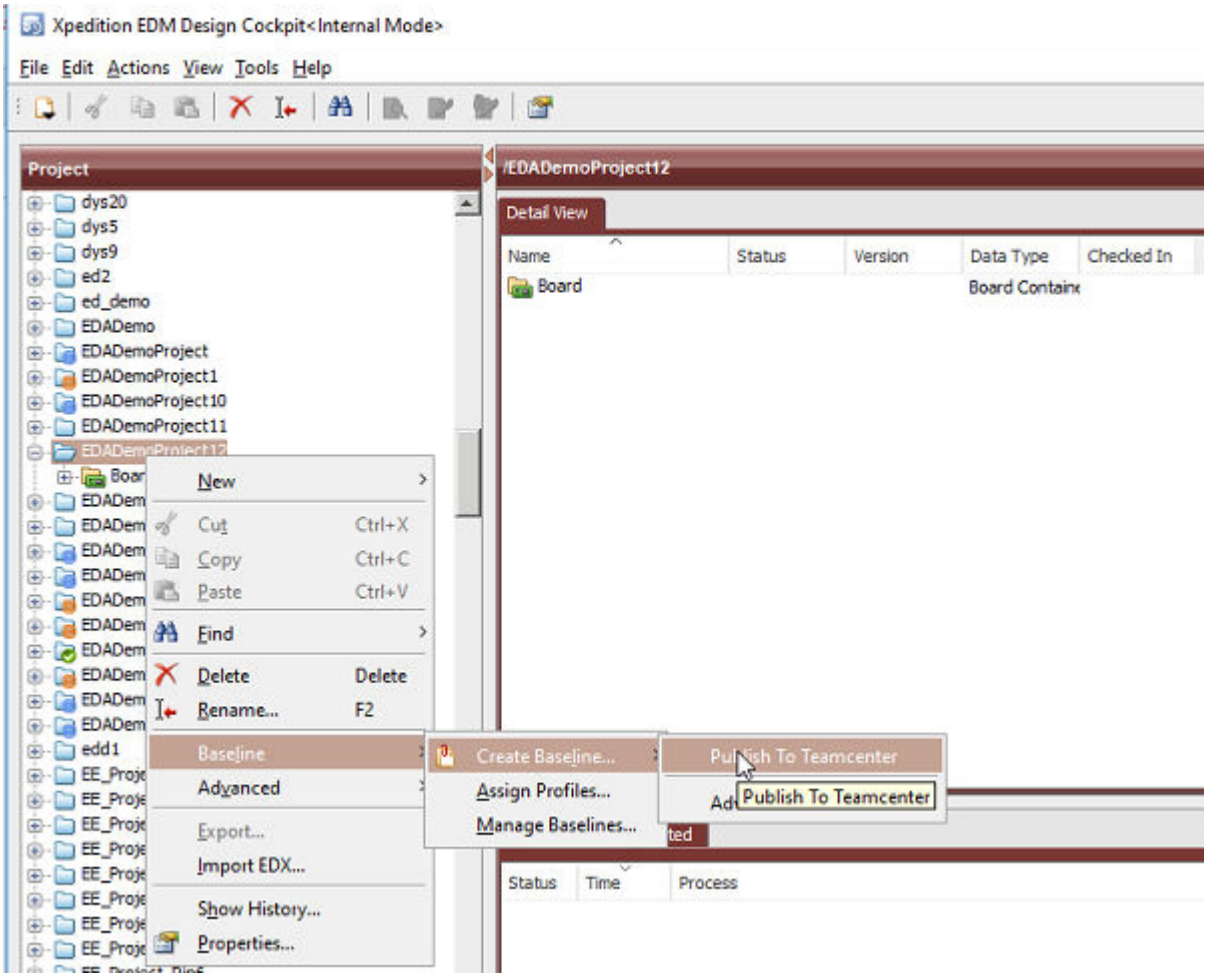
2. Assign **Publish to Teamcenter** profile and click **OK**.

© 2025 Siemens



3. Select the **Project** and create the baseline by selection the **Publish to Teamcenter** profile to automatically create the baseline and EDX files.

© 2025 Siemens

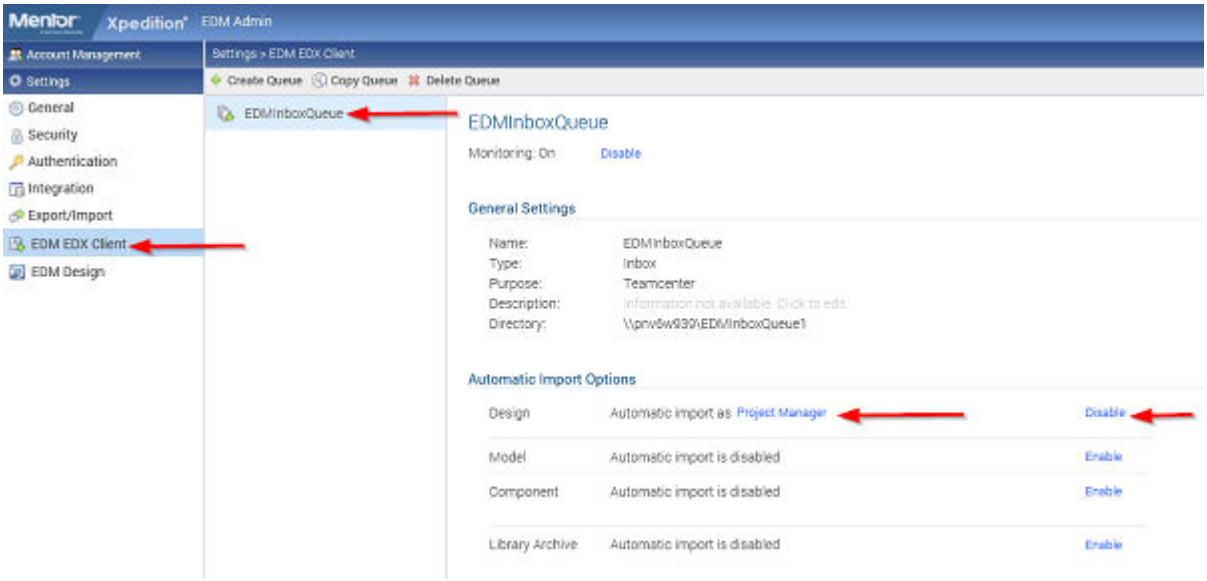


Set automatic import options

1. Open the **EDM Admin** web tool.

2. Go to **Settings** and click **EDM EDX Client** from the navigation panel.

© 2025 Siemens



3. In the inbox queue panel you previously created, first enable the **Automatic Import Options** link and in the **Design** field, from the list of users, select the **Project Manager** user next to **Automatic import as**.

Export X-Y coordinates and rotation details of the EDX file into Teamcenter

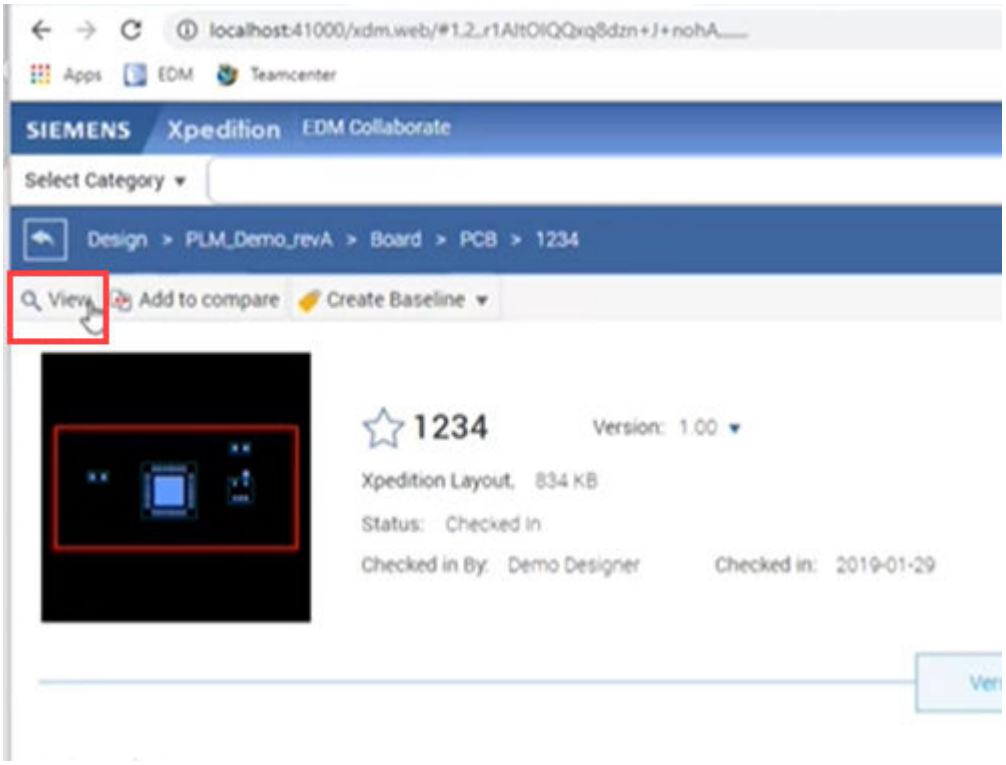
In EDM integration, when you create a design BOM in Teamcenter, you can set the relative 3D position in the BOM line transformation matrix for each component instance. However, for the relative 3D position value to be set in the BOM lines, the X and Y coordinates, surface, and rotation information of the design components must be available in the design EDX files.

Apart from these four attributes, you can also save other reference designator attributes while saving designs from EDM to Teamcenter. To export these additional reference designator attributes from EDM to Teamcenter, you must **map the occurrence attributes** such as reference designator and board side (up or bottom) in the ***usrdef.xml** file.

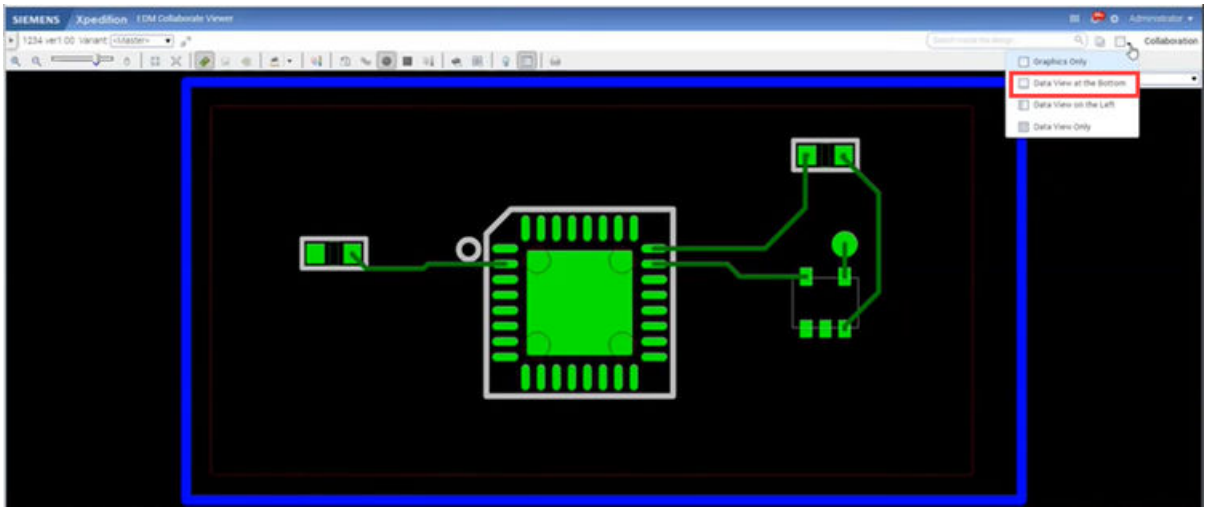
The X and Y coordinates, surface, and rotation information are not available in the EDX files OOTB. For this information to be exported in Teamcenter through the EDX files, you must perform the following configuration in Xpedition EDM.

1. Open a design in Xpedition EDM Collaborate tool. Click **View** to open the design in the viewer.

© 2025 Siemens



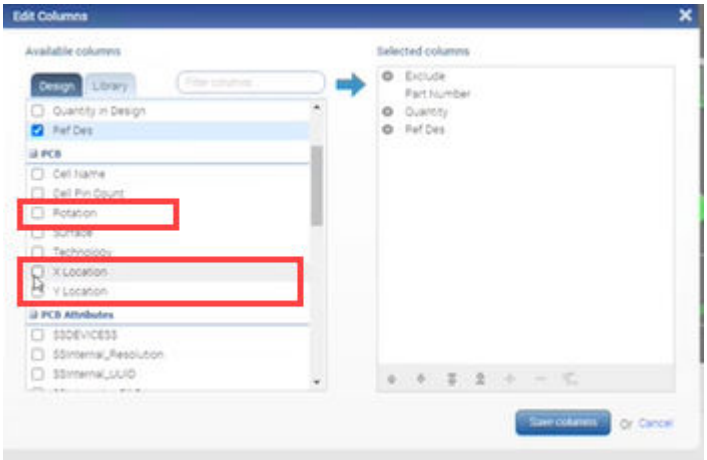
2. At the top-right of the Xpedition EDM Collaborate Viewer tool, click , then choose the **Data View at the Bottom** option from the list.



3. On the toolbar, click .

4. In the **Edit Columns** dialog box, on the **Design** tab, select the **X Location**, **Y Location**, **Surface**, and **Rotation** options that are listed under the **PCB** column.

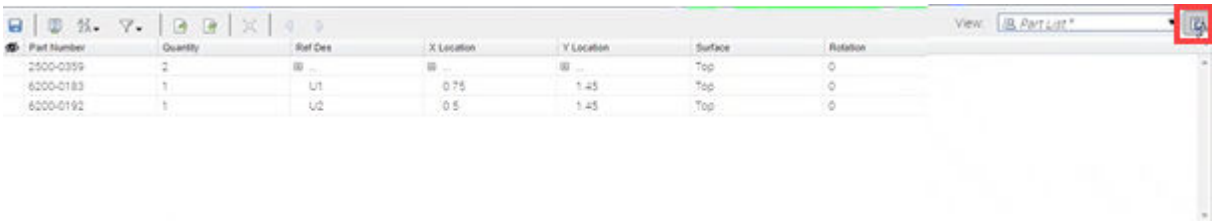
© 2025 Siemens



Apart from these four attributes, you can also save other reference designator attributes while saving designs from EDM to Teamcenter. To export these additional reference designator attributes from EDM to Teamcenter, you must **map the occurrence attributes** such as reference designator and board side (up or bottom) in the ***usrdef.xml** file.

5. Click **Save columns**.

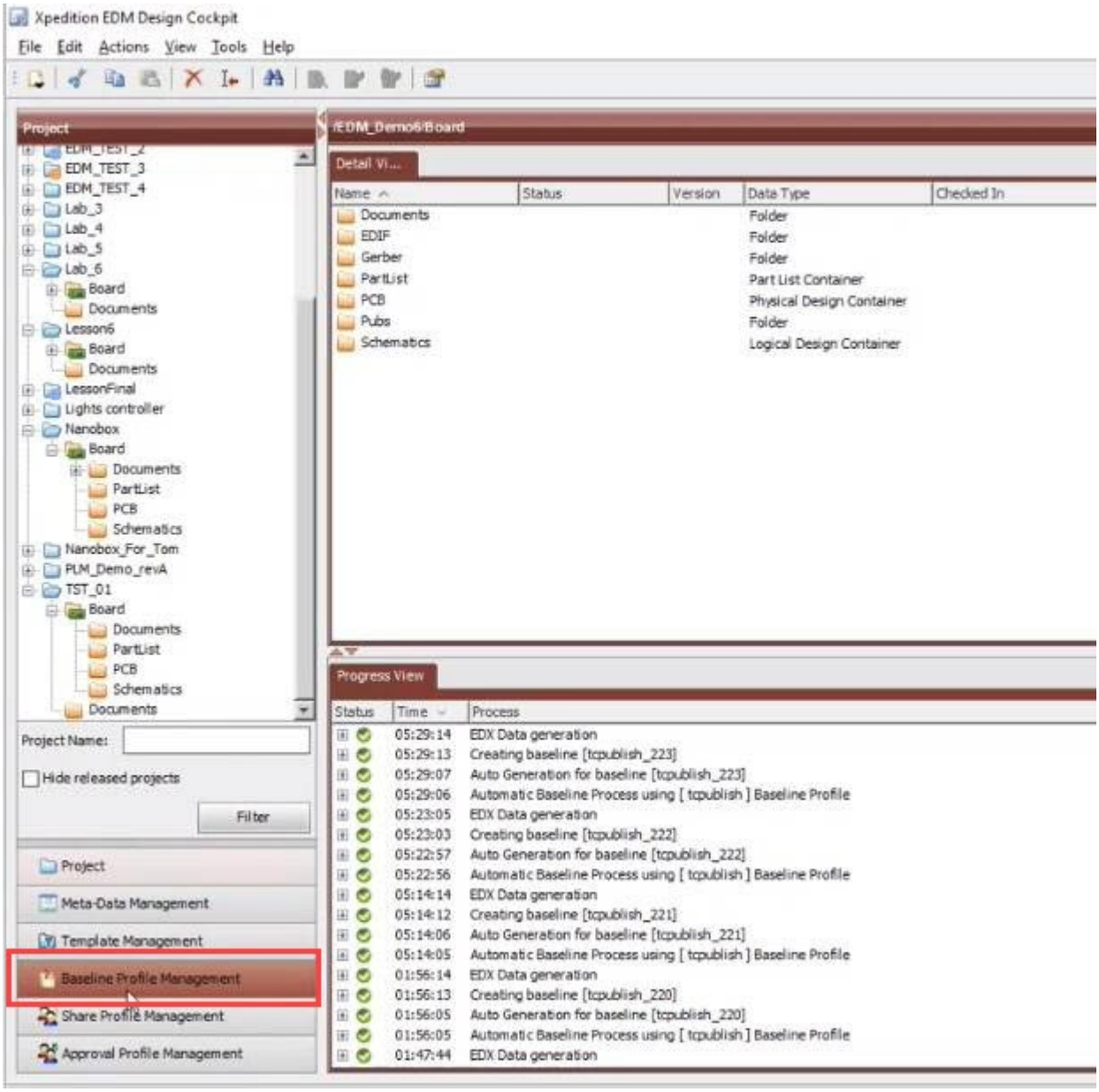
6. Click to save the part list view scheme.



7. In the **Save Scheme** dialog box, click **Save**.

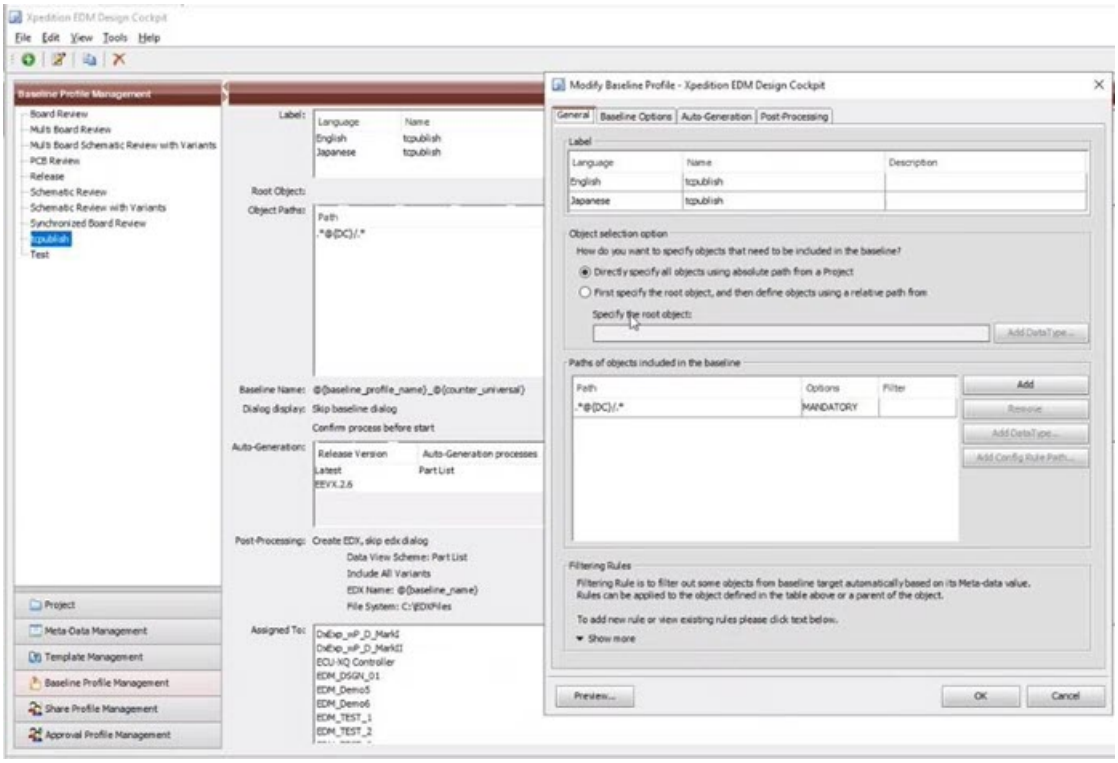
8. To auto generate the baseline information, in the **EDM Design Cockpit** tool, select the **Data View Scheme** as the Part List in the **Baseline Profile Management** tree.

© 2025 Siemens

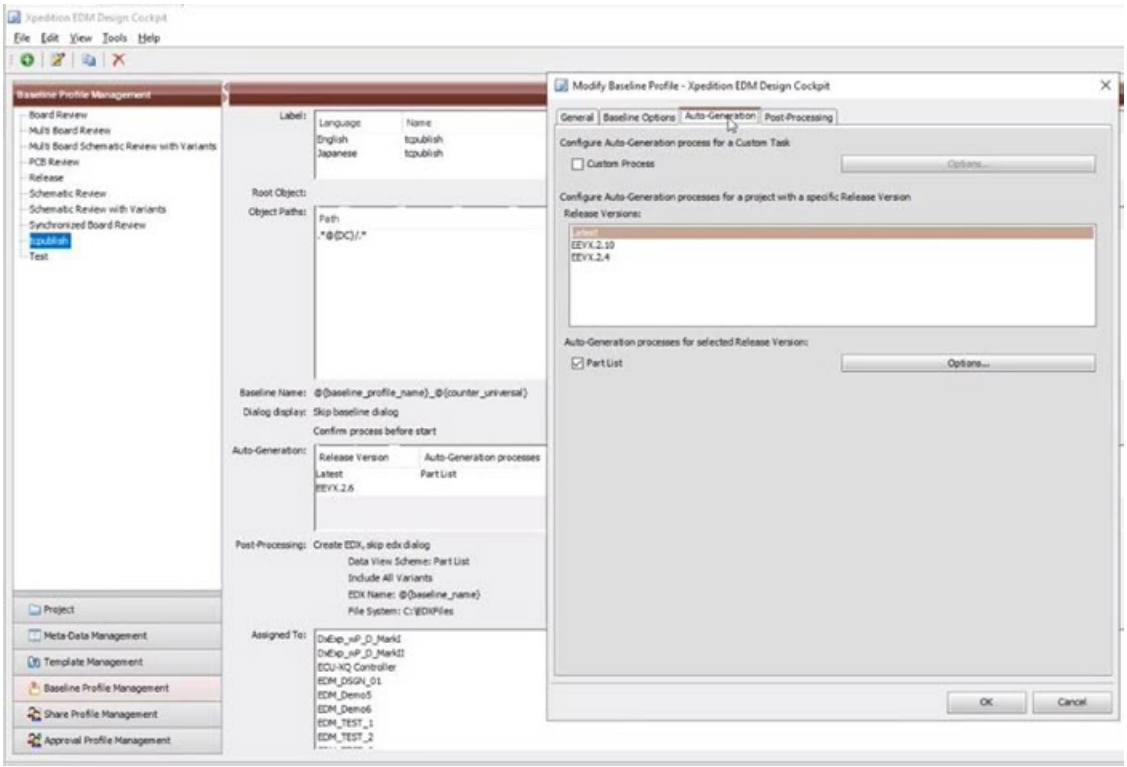


9. Double click **Publish to Teamcenter** option.

© 2025 Siemens



10. In the **Modify Baseline Profile** dialog box, select the **Auto Generation** tab.



11. Click **Options** to open the **Part List Source** dialog box..

© 2025 Siemens

12. Select **Layout** from the list and click **OK**.

13. Save the **Project**.

14. On **Publish Baseline** operation, the EDX file will be generated with the X location, Y location, Surface, and Rotation information and open the Teamcenter Gateway for EDA user interface.

Retain the EDX files after publishing to Teamcenter

To retain the EDX files after they are published to Teamcenter, set the Teamcenter EDA log level to debug mode in the *%TCEDAECAD_ROOT%***\log4j.properties** file.

If the log level is no set to debug mode then after getting published to Teamcenter, the EDX files will be deleted from the client machine.

EDM tool connector configurations for Teamcenter integration

Overview of connector configurations required for EDM integration

After the configurations required to integrate EDM with Teamcenter are complete you must further configure and customize the ECAD tool connector as per your requirement.

How variant names are extracted to an EDX package from EDM project

Variant names are extracted from the variant table data in the **DesignContainer.xml** file created in EDX. The variant BOM is derived from **PartListContainer.xml** file created in EDX for each variant in the design.

How design BOM is extracted to EDX package for an EDM project

The EDX connector extracts component instance information for a design from the **PartListContainer.xml** file present in the EDX package.

The baseline profile **Publish to Teamcenter** is used to include the **PartListContainer.xml** files in the EDX file generated. In the case of variant designs, a **PartListContainer.xml** file is generated for the base BOM and a separate **PartListContainer.xml** file is created for each variant.

These **PartListContainer.xml** files are used in the maintenance of Teamcenter BOMs by using the EDX connector.

Working with intermediate and viewable files

During the save action, *EDIF design files* contained in the EDX design package are extracted from the EDX package to the **edif** subdirectory in the design directory as an input for translation to a schematic *Formatted ASCII Text File (FATF)* file. The input files are referred to as schematic intermediate files. The resulting EDIF translator generated schematic FATF files generated by the EDIF translator and

© 2025 Siemens

the schematic intermediate files, if any, are uploaded and attached to **SCHFATF** and **EDAGenSchem** datasets, respectively.

In addition, during the save action, *ODB++ design files* contained in the EDX design package are extracted to the **odb** subdirectory in the design directory as an input for translation to a **PCBFATF** file. The input ODB++ files are referred to as PCB intermediate files. The resulting ODB++ translator-generated PCB FATF files, and optionally the PCB intermediate files, are uploaded and attached to **PCBFATF** and **EDAGenPCBCAD** datasets, respectively.

To create the intermediate file required for the save operation from Xpedition Layout:

1. Select the **Output→ODB++** menu option.

2. In the **ODB++ Output** dialog box, set the output path to as **.\Output\ODBpp**, and set the file name as **designodb**.

3. Select the **Compress output** option to create the **designodb.tgz** file.

The user created EDIF files can be placed at any location in the **<design>** folder inside the EDX design package. Also, the location of the **<board.eds>** file can be anywhere inside the EDX design package. The user created ODB++ files can be placed at any location in the **<design>** folder. Also, the location of the **designodb.tgz** file can be anywhere inside the EDX design package.

Working with supported data files

Within the xDM Design client, supporting or derived data files are included with an Xpedition design. These files consist of information related to manufacturing, assembly, fabrication, or BOM data.

These attached data files may be of data types unrecognized by the user's operating system (typically, Windows). If these files are not recognized by the system, by default these files cannot be opened or viewed from within the xDM Design client. This is because xDM Design client passes most actions on standard data types to the operating system for execution. The operating system must have default tools defined for the standard data types used within the EDM data.

To add support for these unrecognized file types, the user must configure the Windows operating system to recognize these file types.

1. From a Windows system, open a file system browser.

2. Navigate to and locate the target file.

3. Select the target file, right click on the file and select **Open With**.

   If the tool to be used with the target data type appears in the dialog box, the setup is complete.

© 2025 Siemens

4. If the tool to be used with the target data type does not appear in the dialog box, select the target file, right click on the file and select **Select a program from a list of installed programs** and click **OK**.

5. Select the required tool and click **OK**.

Current behavior of EDM integration

When the EDX baseline is released, the EDX project and the *BoardContainer* are also released in EDM. If the **EDA_Workflows** preference is configured to release the CCA item revision based on the EDX baseline status, the CCA item revision is released each time the EDX files are saved to Teamcenter. So, the subsequent save of EDX files to Teamcenter always result in revise operation. If a new baseline is created for the released EDX project then the baseline is in the **Valid** state but the EDX project is still in the **Release** state.

The EDX API retrieves the status of the EDX files from the EDX project and not from baseline. Due to this reason, when the EDX file is saved using this unreleased baseline, the CCA item revision is released in Teamcenter. The subsequent save action of EDX files to Teamcenter results in a revise action.

Setting up ECAD translation

ECAD translation modes

The following translation modes are supported for ECAD translation:

- **Local translation**

  The viewable file is generated locally. This option is available only on Windows clients.

  - ECAD translation happens by default and is installed out-of-the-box and deployed locally on the EDA client machines with the EDA Standalone Client TEM feature.

  - You must edit the **TCEDAClient.properties** file located in the *TCEADAECAD_ROOT* directory to update the path to the translation directory (*Teamcenter EDA application root\emps\translation*).

- **Remote translation**

  The EDA integration initiates a remote translation of the intermediate file into a viewable format.

  ECAD translation is embedded into the Teamcenter Dispatcher framework. Make sure that the **Dispatcher server and required translator are installed**. After installing and setting up the dispatcher you must **enable the PcbToFatf translator**.

  If you select the **Viewable** check box while saving your EDA design, the following automatic translation process takes place:

© 2025 Siemens

- The design intermediate file that can be parsed by the ECAD translator is extracted.
- The intermediate file is converted into a neutral file format: **XFATF** for PCB designs and **XSCH** for schematic designs.
- The neutral file is saved as a viewable dataset in Teamcenter.

You can enable either the local translation mode or the remote translation mode. You may also enable both these modes and control which mode to use by editing the **EDA_PreferredLocalTS** preference.

Enable PcbToFatf translator

After you install and configure Dispatcher as per the deployment strategy, you must enable the **PcbToFatf** translator required for the remote translation of the intermediate file into a viewable format.

You can use TEM to install the translator or manually edit the **translator.xml** file to enable the translator. For instructions on how to enable the **PcbToFatf** translator using TEM, see the Dispatcher — Deployment and Administration help on Support Center.

The following is an example of how to edit the **translator.xml** file to enable the translator.

Prerequisites: Verify that the **TransExecutable dir** is correctly specified for this translator.

1. Open the **translator.xml** file from the *Dispatcher_Root***Module\conf** directory.
2. Set the **isactive** attribute to **true** to activate this translator.

   > Note:
   >
   > By default, this attribute is set to **false**.

3. Open the *translator***.bat** (Windows) from the *Dispatcher_Root***\Module\Translators***translator_name* directory and edit the **CHANGE_ME** tags.

   For more information about **CHANGE_ME** tags, see the **.bat** file for the translator.

4. Run the translator in standalone mode from the command prompt to verify whether it is working properly.

   For more information about running the translator in standalone mode, from a command prompt, change to the **Dispatcher\Translators***translator_name* directory and run the appropriate command for your platform:

© 2025 Siemens

> Note:
>
> For more information, see the **Readme** file in the **Translators***translator_name* directory.
>
> Windows: *translator_name***.bat -help**

5. (Optional) To set up *Dispatcher-client-proxy-user* as the owner of the translated files, make the following change to the *translator_name***.bat** (Windows) file.

   NxTransDirect translator example:

   ```
   "%UGII_BASE_DIR%\ugmanager\run_nxtrans.bat" -u=Dispatcher-client-proxy-user
   -p=password
   -changeOwnerToCad=false "-pim=yes" %*
   ```

   If you set up *Dispatcher-client-proxy-user* as the owner of the translated files, you must create a dispatcher client access rule. For more information on how to create a dispatcher client access rule, see the Dispatcher — Deployment and Administration help.

   After installation, you must add a rule to the access rule tree permitting the translation service proxy user to update attributes of a **DispatcherRequest** object. If this access rule is not created correctly, the dispatcher client reports errors.

© 2025 Siemens

5. Integrating Xpedition or PADS or EDM with Teamcenter EDA for part library management

Planning your deployment for part library management

For installing and integrating Teamcenter EDA with Xpedition, PADS, PADS Professional, or EDM for part library management, you must plan your deployment and your environment setup. You can install the Teamcenter EDA server and client components through **Deployment Center** or through **TEM**.

As an administrator, you must install the appropriate components on the server (Teamcenter) and client machines (ECAD workstation) as per the site requirements as mentioned below:

Server components	Requirements
Teamcenter server	Teamcenter server is used to store and track the ECAD designs. For information about the versions of operating systems, third-party software, and Teamcenter software configurations that are certified for the EDA platform, refer to the **Hardware and Software Certifications** knowledge base article on Support Center. If you have an existing Teamcenter environment, you can update it to support the latest EDA version. Ensure that you download the compatible versions from Support Center before starting your deployment.
EDA server support and library support	Teamcenter server is used to store and track the ECAD designs. EDA server support is required to add ECAD data model to the Teamcenter server environment. You must add EDA server support on top of the Teamcenter environment. As the EDA client is supported in a four-tier architecture, the EDA Server Support feature (EDA data model) must also be installed on a four-tier Teamcenter server.
BMIDE	BMIDE is required to map ECAD objects with Teamcenter objects.
Dispatcher	Dispatcher is required for remote ECAD translations to translate and save the design intermediate file into a viewable dataset in Teamcenter. For library integrations, the **acistoprt** translator will support the translation of the ACIS part to NX parts. If Dispatcher is already installed in your existing Teamcenter environment, you can use the same instance. However, you

© 2025 Siemens

Server components	Requirements
	must modify it to include the translator **acistoprt**. In addition, you also require NX to be installed on the same machine where the Dispatcher is installed.
Teamcenter Classification Admin	Teamcenter Classification Admin is required to create custom classification schema and set up classification mappings for part library management.

Client components	Requirements
EDA client	EDA client moves designs and library parts between ECAD design tools and Teamcenter. The EDA client must be installed on the ECAD workstation using the Teamcenter software kit. Before installing the EDA client, see the **Hardware and Software Certifications** knowledge base article on Support Center to identify which versions of the Xpedition or PADS or PADS Professional are supported on which versions of Teamcenter EDA. Siemens Digital Industries Software recommends that you first perform the installation on a test system to verify the parameter values required for the installation on the ECAD workstations. **Prerequisites and requirements before installing the EDA client:** • **Supported Operating System:** The EDA client framework is supported only on the Windows operating system. Teamcenter software kit contains the 64-bit installer for Windows system. You can select Win32 applications directly from TEM or DC along with the rest of the Win64 applications from within the Teamcenter software kit. • **Supported Java version:** For information about the supported Java versions that are certified for the EDA platform, see the **Hardware and Software Certifications** knowledge base article on Support Center. • **Information required for EDA client installation from Teamcenter installation administrator:** • FMS client cache (FCC)

© 2025 Siemens

Client components	Requirements
	If the FMS client cache (FCC) exists, use the existing one or specify a new one. • The FCC parent information for the EDA client, including the host and port value The minimum information required is the host name and the port. • The four-tier, middle-tier Web application server URI, and the connection name for the EDA client • Logon URL and the application ID To enable Teamcenter Security Services • Host name, port, and license server mode of the Siemens Digital Industries Software common license server • **Other installation considerations** • Only the English versions of ECAD tools are supported. • Item ID, variant name, dataset name, reference designator value, and mapped attribute names and values must contain only ASCII characters. Mapped attributes include CAD attribute mappings and component instance attribute mappings. • The staging directory and all file names must contain only ASCII characters. • PADS Logic does not support variants.
Teamcenter EDA gateway	Install Teamcenter EDA gateway for part library management. Teamcenter EDA Gateway allows the ECAD tool users to save, check in, or check out native design files, access approved components, populate electrical component BOMs, share fabrication and assembly data, and create and manage other types of derived ECAD data files.

Workflow to integrate your ECAD applications with Teamcenter for part library management

Scenarios to deploy the Teamcenter EDA server and client components:

© 2025 Siemens

Using Deployment Center	*Using TEM*
Install Teamcenter server and the EDA client	Install Teamcenter server and Install BMIDE
↓	↓
Set up Teamcenter server for part library management	Set up Teamcenter server for part library management
↓	↓
Configure the EDA client for part library management	Install the EDA client
↓	↓
Perform additional configurations and customizations required for your ECAD applications	Configure the EDA client for part library management
	↓
	Perform additional configurations and customizations required for your ECAD applications

Install Teamcenter EDA for part library management using Deployment Center

The following procedures assume that you are installing EDA on an existing Teamcenter and that you are familiar with Deployment Center.

Procedure

1. Log on to Deployment Center and select the environment to which you want to add Teamcenter EDA .

2. Go to the **Applications** task. Click **Add or Remove Selected Applications**.

3. In the **Available Applications** panel, use the web browser search to find the Electronic Design Automation (EDA) application. Select the application, and then click **Update Selected Applications**.

   Deployment Center automatically selects any additional dependent applications.

© 2025 Siemens

4. In the **Available Applications** panel, under **EDA Client**, select the ECAD tool with which you are integrating Teamcenter EDA for part library management and other options as required on the ECAD client machine:

   - **Mentor PADS Library**
   - **Mentor PADS Library Win32**
   - **Mentor Xpedition EDM Library**
   - **Mentor Xpedition Library**
   - **Teamcenter Gateway for EDA Library**
   - **Teamcenter Gateway for EDA Library Win32**

   By default, all options under **EDA Client** are selected. Remove the tools and applications that are not required for part library integration and click **Update Selected Applications**.

   Deployment Center automatically selects any additional dependent applications.

5. Go to the **Components** task.

6. In the **Selected Components** list of the **Components** task, note any remaining components whose configuration status is not **100%**. Select each incomplete component, enter required parameters, and save component settings until all components in the environment show a configuration status of **100%**.

   When all components are fully configured, the **Deploy** task is enabled.

7. Go to the **Deploy** task. Click **Generate Install Scripts** to generate deployment scripts you will use to update affected machines.

   When script generation is complete, note any special instructions in the **Deploy Instructions** panel.

8. Locate deployment scripts, copy each script to its target machine, and then run each script on its target machine.

   For more information about running deployment scripts, see *Run the deployment scripts* in the Deployment Center documentation.

   You can also generate mini-kits for mass client deployment. Running the **generateMiniSoftware** utility extracts only the necessary software from the full software kits and builds a smaller, more portable software kit for the specific client deployment. For more information, see *Generating mini kits for mass client installations* in *Deployment Center — Usage* on Support Center.

© 2025 Siemens

If you face any issues during the deployment, see *Troubleshoot the deployment script* in the Deployment Center documentation.

Installing Teamcenter server for part library management through TEM

Installing the Teamcenter server for part library management

Install Teamcenter

To install Teamcenter EDA, you must first install Teamcenter. To install the Teamcenter corporate server, refer to the Teamcenter Installation on Windows Using TEM help.

If you have an existing Teamcenter environment, you can update it to support the latest EDA version.

Refer to the Install patches on a Teamcenter server topic in the Teamcenter Installation on Windows Using TEM help for instruction on installing Teamcenter patches.

Add EDA server support and library support on Teamcenter

EDA Server Support adds the EDA data model to the Teamcenter data model. **ECAD Library Management** adds the EDA part library data model to the existing Teamcenter data model. As the EDA client is supported in a four-tier architecture, the **EDA Server Support** feature (data model) must also be installed on this Teamcenter server.

Because you are adding EDA server support and library support to an already existing Teamcenter server, launch Teamcenter Environment Manager from your Teamcenter environment.

1. Start Teamcenter Environment Manager (TEM).

2. Select the **Configuration Manager** option to perform maintenance on an existing installation and click **Next** until the **Select Features** dialog box appears.

3. In the **Select Features** dialog box:

   a. Choose **Extensions→Supplier Relationship Management** and select **Vendor Management**.

   b. From **Extensions→Mechatronics Process Management**, select the following options:

      - **EDA Server Support**
      - **EMPS-Foundation**
      - **ECAD Library Management**.

© 2025 Siemens

c. Choose **Extensions→Mechatronics Process Management** and select **EDA for Business Modeler IDE**.

d. Specify **Active Workspace** server extensions features for EDA. These are available in the **Features** panel in **Teamcenter Environment Manager (TEM),** under **Base Install→Active Workspace→Server Extensions→EDA Server Support for Active Workspace**.

e. Specify **Active Workspace** client extensions features for EDA. These are available in the **Features** panel in **Teamcenter Environment Manager (TEM),** under **Base Install→Active Workspace→Client→Electronic Design Automation for Active Workspace**.

4. In the **Installation Directory** box, type the location where you want to install Teamcenter (TC_ROOT).

5. Click **Next**.

6. Enter information as needed in the subsequent panes.

7. In the **Confirm Selections** dialog box, click **Next**.

   The EDA server components are installed.

   The **Add/Remove Components** dialog box confirms if the installation is successful. If the installation is unsuccessful, click **Show Details** and proceed as needed.

8. Obtain the license file that includes EDA licensing and EDA library license (for library support) and install it on the server.

After adding the EDA server support and library support to your Teamcenter environment, you must verify your installation by checking whether the EDA data types are added to Teamcenter.

Verify your installation

Verify the addition of the EDA data types to the server:

1. Run the Teamcenter rich client.

2. In My Teamcenter, choose **File→New→Item**.

   The **New Item** dialog box is displayed.

3. Verify that the EDA, EDAComp, EDASchem, and EDACCABase item types are added to the list of types you can create.

4. Select one of the EDA item types and create an instance.

© 2025 Siemens

5. Verify that the item instance is created.

Install BMIDE for mapping ECAD library objects with Teamcenter objects

You must install Business Modeler IDE to **configure mapping of ECAD library objects with Teamcenter objects**.

You can install the BMIDE either as a stand-alone application, or place the BMIDE plug-ins into an existing Eclipse environment.

For instructions on installing BMIDE, refer to the Install the Business Modeler IDE topic or the Add the Business Modeler IDE to an existing Eclipse SDK environment topic in the BMIDE for Data Model Design documentation on Support Center.

Install or update Dispatcher for translating ACIS parts to NX parts and JT file

If Dispatcher is already installed in your existing Teamcenter environment, you can use the same instance. However, you must modify it to include an additional translator to translate the ACIS part files to NX parts.

For library integrations, the **acistoprt** translator will support the translation of the ACIS part to NX part. Further more, to convert the NX part to JT file to visualize the part in Active Workspace, you must enable the **NxToPvDirect** translator.

> Note:
>
> Ensure that the **TC_DATA** and **TC_ROOT** point to correct location in the **Dispatcher\Translators\\nxtopvdirect\nxtopvdirect.bat**.

For instructions on installing Dispatcher, refer to the Dispatcher — Deployment and Administration documentation on Support Center. After installing and setting up the dispatcher you must **enable the *acistoprt* translator**.

Setting up Teamcenter server for part library management

Overview of configuring Teamcenter for ECAD part library management

Before you install and enable your design tools, you must configure your server for part library management.

To enable the part library feature, you can create a custom classification schema, set up the mappings, map the ECAD part attributes and component instance attributes, and configure the import or export process. Additionally, you can change the system preferences as per your requirements.

You must configure the following for part library integration on your Teamcenter server:

© 2025 Siemens

- **Create external applications for your ECAD tool in Teamcenter**.

  External applications are part libraries that are based on site objects. To create an EDA library gateway application configuration file on the EDA client, you must first setup external applications on your Teamcenter server. You can create external applications for setting up your ECAD library tool by using the OOTB application types or by creating a custom application type. You can then create the EDA library gateway application configuration file using the application types available on the Teamcenter server.

- **Create a Teamcenter part library**

  The Teamcenter part library is the enterprise-wide master ECAD library. Users can either export the whole ECAD library into Teamcenter or select a subset of it and then upload it to Teamcenter. You can identify the parts that do not exist in Teamcenter and save them to Teamcenter. After you have enabled and configured the part library to function with Teamcenter, users can start working with the part library management feature.

- **Map ECAD part attributes to Teamcenter properties** and **Map ECAD library objects to Teamcenter objects**.

  ECAD part attributes can be mapped to Teamcenter properties. When parts are imported into Teamcenter, ECAD attributes are placed into the corresponding Teamcenter Classification object. The ECAD attributes can also be mapped to the properties on the **Item** or **ItemRevision** objects. When parts are exported from Teamcenter, the properties are mapped back to the corresponding part attributes in the ECAD tool. All ECAD attributes are passed to Teamcenter during import. However, only the attributes that are mapped to Teamcenter properties are imported.

You can also update or modify the following OOTB configurations or use custom configurations for part library integration on your Teamcenter server:

- **Create a custom classification schema** and **set up classification mapping** for part library management. The classification mapping converts the part category in the ECAD tool library to the class ID defined in Teamcenter Classification and vice versa.

- **Use custom objects or custom relations** by modifying default values in the EDA library preferences.

  Teamcenter EDA provides a basic set of objects and relations. The relations between the object and the item and item revision determines the rules enforced. By default, Teamcenter objects and relations are created during the EDA library integration, which can be configured using the Teamcenter EDA library preferences.

  The administrator can also use custom objects and relations by modifying the default values of the preferences that are used to create the default Teamcenter objects and relations. Before using custom part types, custom objects, and relations, you can refer to the EDA library preferences that are used to create the **OOTB objects** and the **OOTB relations**.

- **Synchronize ECAD library parts with multiple external sites**

© 2025 Siemens

ECAD users should be able to synchronize and reuse the part data across multiple external sites. You can create a query in the Query Builder application to support synchronization of ECAD library parts with multiple external sites.

- **Managing and configuring system preferences for part library management**.

Create external applications for your ECAD tool in Teamcenter

External applications are part libraries that are based on site objects. In the Organization application, you can create, modify, and delete external applications. To create EDA library gateway application configuration file on the EDA client, you must first set up external applications on your server. You can create external applications for setting up your ECAD library tool by:

- **Using the OOTB application types**

  Based on the application types currently supported on the Teamcenter server, the default application names, family names, and primary dataset types for the supported application types is given in the table below for setting up ECAD library integration.

  > Note:
  >
  > The application name in the ***.edadef** file, ***.edadef** file name, and in the Teamcenter application ID must be same.

Library Integration	Application Name	Application Type (External Applications)	Family Name	Primary Dataset type
Mentor Xpedition	mentorExpLib	Mentor Expedition	expedition	EDAComExpedition
Mentor PADS	mentorPadsLib	Mentor Pads	pads	EDAComPADS
Mentor EDM	mentorEDXLib	Mentor EDX	mentorEDX	Epl0EDXPackage

OR

- **Creating a custom application type**

  Instead of using the OOTB application types, you can define a new application type in the **EDALIB_External_Applications** preference. Refer to the preference description for the details.

  After the application type is defined in the **EDALIB_External_Applications** preference, it is displayed in the **Application Types** list along with the OOTB values. You can then create external applications using this custom value. For example, you can create a custom application type named *TcEDA*.

1. Select the top-level **External Applications** node from the **Organization List** tree.

   The **External Applications** pane appears.

© 2025 Siemens

2. Complete the following information:

   a. Type a unique application name in the **Application Name** box using any combination of alphanumeric characters. You can use the hyphen (-) and underscore (_) special characters.

   b. Type a unique application ID in the **Application ID** box. The application ID must be any valid positive integer.

      To automatically generate a unique application ID, click the **Assign** button.

   c. Select the application type from the **Application Type** list. You can use either the OOTB application type or a custom application type to create an external application.

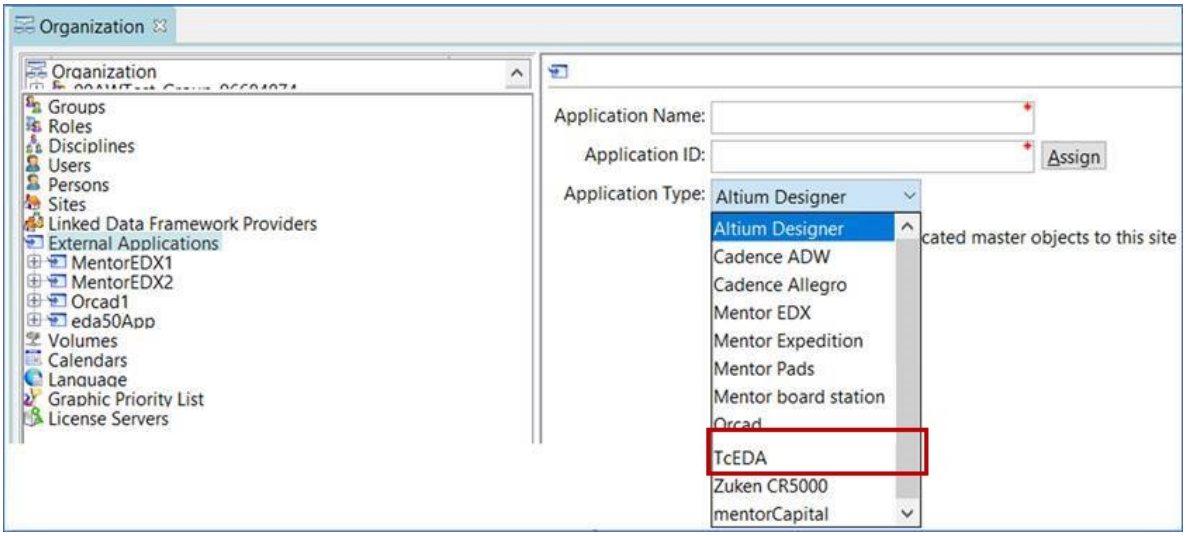
      - **Use the OOTB application type from the available options:**

        - Mentor Expedition
        - Mentor Pads
        - Mentor EDX
        - Mentor board station

        OR

      - **Use a custom application type**

        After the application type is defined in the **EDALIB_External_Applications** preference, it is displayed in the **Application Types** list along with the OOTB values, for example *TcEDA*. Select the custom application type from the list.



© 2025 Siemens

d. Select **Allow deletion of replicated master items to this site** to allow the deletion of replicated master items.

> Note:
>
> The **Uses TCXML Payload** check box only for information purposes and cannot be modified.

3. Click **Create**.

Create a Teamcenter part library

Because the Organization application in Teamcenter is integrated with the ECAD part library, you can create the Teamcenter part library in Organization using the **External Applications** node. This node holds the part libraries created.

1. Open Organization.

2. From the **Organization List** tree, select the **External Applications** node.

   The **External Applications** pane appears.

3. Type a name In the **Application Name** box. This can be any combination of alphanumeric characters.

4. Type an ID in the **Application ID** box.

   This can be any positive integer. If you want the system to generate an ID, click **Assign**.

5. From the application type list, select a type based on your ECAD tool. All available ECAD options are listed here.

> Note:
>
> Select the **PADS** application type only for the **PADS standard** product. For **PADS Professional** product, you must select **Xpedition** application type.

6. Check **Allow deletion of replicated master objects to this site**.

7. Click **Create**.

   The new part is created and listed under the **External Applications** node in the **Organization List** tree.

© 2025 Siemens

Mapping ECAD part attributes to Teamcenter attributes

Overview of mapping ECAD part attributes with Teamcenter properties

ECAD part attributes and objects must be mapped to Teamcenter properties. When the ECAD parts are imported into Teamcenter, ECAD attributes are placed into the corresponding Teamcenter properties. When parts are exported from Teamcenter, the properties are mapped back to the corresponding part attributes in the ECAD tool. All ECAD attributes are passed to Teamcenter during import. However, only the attributes that are mapped to Teamcenter properties are imported.

During the EDA library integration, OOTB mappings between the ECAD tool and Teamcenter are provided by default in the Integration Definition File. You can customize the mappings in Teamcenter by making changes in the Integration Definition File. Refer to the **best practices for mapping ECAD library objects to Teamcenter objects** topic before creating custom mappings.

Best practices for mapping ECAD library objects to Teamcenter objects

- **Use the available EDA type objects in Teamcenter**

  Instead of using the generic item types to represent EDA objects in Teamcenter, use the available EDA types or subtypes. This limits the number of objects for the synchronizer to process and improves performance.

  Teamcenter provides the following EDA objects:

  - **Epl0EDALibPart**
  - **EDAComPart**
  - **EDASymbol**
  - **EDAFootprint**

- **Map different EDA objects in separate types to avoid duplicate item IDs**

- **Use the default type mappings**

  Even though you can give any name for EDA objects in the **EDAlib.xml** file, and then map the object names to Teamcenter business object types, Siemens Digital Industries Software recommends that you use the following types to represent your ECAD objects in the **EDAlib.xml** file:

© 2025 Siemens

ECAD object	Description	Value in the EDAlib.xml file
Library part	Represents a library part	**LibraryPart**
Component/Part	Represents a component or part	**Component**
Symbol/ Schematic Model	Represents a symbol	**Symbol**
Footprint	Represents a footprint	**Footprint**

- The **name** attribute is mandatory and some ECAD applications do not have a corresponding attribute. In this case, the value of the **name** attribute can be **itemID**.

```
<property name="itemId" required="true">
</property> <property name="name" required="true"> </property>
```

- Usually, attributes in the ECAD library tools are in the string format. However, the corresponding Teamcenter properties may be in a different format to leverage the Teamcenter Classification search capabilities. Ensure that the unit in the ECAD attribute matches the Teamcenter attribute unit. The value can then be converted to the Teamcenter attribute format. Otherwise, the import fails. To check the Teamcenter attribute format, refer to Basic Classification — Deployment and Administration help on Support Center.

  Additionally, the Classification application uses the concept of *unit bundles*. For example, distance is measured using units in a bundle, such as millimeters, centimeters, meters, and inches. Among the units in a bundle, one of them is known as the base unit, and its attribute value is saved into the database. However, the Classification import-export layer does not provide APIs to access subunits other than the base unit. This leads to ECAD library integrations failing during an import when the value is not in the base unit. To avoid this problem, do not use unit bundles for ECAD attributes in ECAD tools and Classification.

Map ECAD part attributes to Teamcenter attributes

If you use a custom part or dataset type in your ECAD tool, you must create the corresponding item and dataset in Teamcenter using Business Modeler IDE before mapping the custom objects. Refer to the **Best practices for mapping ECAD library objects to Teamcenter objects** topic before mapping ECAD part attributes to Teamcenter attributes to avoid ECAD library integrations failing during an import.

You can customize the mappings in Teamcenter by downloading the Integration Definition File from the dataset for respective integration, editing the file, and importing the file back into the dataset in Teamcenter.

- Search the dataset in Teamcenter for the respective library integration. Names of the dataset to search for a specific library integration is given in the table below. For instructions on how to search datasets

© 2025 Siemens

in Teamcenter, refer to Searching in Teamcenter topic in the Teamcenter Basics help on Support Center.

Integration	Dataset name	Integration Definition File in the dataset
Mentor EDX Library	**MENTOREDXLIB_BHM_INT_DEF_FILE**	**MENTOREDXLIB_BHMIntegrationDefinition.xml**
Mentor Expedition Library (ODBC)	**MENTOREXPLIB_BHM_INT_DEF_FILE**	**MENTOREXPLIB_BHMIntegrationDefinition.xml**
Mentor PADS Library	**MENTORPADSLIB_BHM_INT_DEF_FILE**	**MENTORPADSLIB_BHMIntegrationDefinition.xml**

- Perform a check-out operation on the selected dataset. For instructions on how to checkout objects in Teamcenter, refer to Check out an Object topic in the Teamcenter Basics help on Support Center.
- Download the Integration Definition File within the dataset.
- Make changes in the Integration Definition File to add the ECAD tool attributes. For instructions on how to modify configurations related to attribute mappings in the Integration Definition File, refer to the Integration Definition File topic in the MBSE Integration Gateway — Deployment help on Support Center.
- Replace the Integration Definition File with the updated one in the dataset.
- Perform the check-in operation on the dataset.

Sample ECAD part attribute mapping definitions

The below example defines sample ECAD part attribute mappings for Mentor PADS library integration within the Integration Definition File.

```
<?xml version="1.0" encoding="utf-8"?>
<!--
@<COPYRIGHT>@
==========================================================================
=======
Copyright 2023.
Siemens Product Lifecycle Management Software Inc.
All Rights Reserved.
==========================================================================
=======
@<COPYRIGHT>@
-->
```


© 2025 Siemens

```
<!--

    This file contains the object mapping definition for all the
    EDA objects created in Teamcenter using Tc EDA integration.

-->
<BHMIntegration xmlns="http://www.plmxml.org/Schemas/bhm"
xmlns:xsi="http://www.w3.org/2001/XMLSchema-instance"
xsi:schemaLocation="http://www.plmxml.org/Schemas/bhm"
applicationName="MENTORPADSLIB">
    <ConnectorClass fullName="" jarFilePath="" />
    <SupportedVersions>
        <Version id="2018.1" />
    </SupportedVersions>

    <ObjectMapping type="Component" behaviorType="MIXED"
tctype="EDAComPart">
        <AttributeMappings>
            <AttributeMapping name="name" tcattr="object_name"
includeinduplicatecheck="false"/>
            <AttributeMapping name="id" tcattr="item_id"
includeinduplicatecheck="false"/>
            <AttributeMapping name="description" tcattr="object_desc"
includeinduplicatecheck="false"/>
            <AttributeMapping name="object_application"
tcattr="object_application" includeinduplicatecheck="false"/>
            <RevisionAttributeMapping>
                <AttributeMapping name="object_application"
tcattr="object_application" includeinduplicatecheck="false"/>
            </RevisionAttributeMapping>
        </AttributeMappings>

        <BHMElement type="mentorPadsLib" tctype="EDAComPADS"
behaviorType="GRM" reltype="IMAN_reference" reftype="mentorPadsLib"></
BHMElement>
        <BHMElement type="Footprint" tctype="EDAFootprint"
behaviorType="BVR" reltype="" reftype=""></BHMElement>
        <BHMElement type="Symbol" tctype="EDASymbol" behaviorType="BVR"
reltype="" reftype=""></BHMElement>
    </ObjectMapping>

    <ObjectMapping type="Footprint" behaviorType="MIXED"
tctype="EDAFootprint">
        <AttributeMappings>
            <AttributeMapping name="name" tcattr="object_name"
includeinduplicatecheck="false"/>
            <AttributeMapping name="id" tcattr="item_id"
includeinduplicatecheck="false"/>
```


© 2025 Siemens

```
            <AttributeMapping name="description" tcattr="object_desc"
includeinduplicatecheck="false"/>
            <AttributeMapping name="object_application"
tcattr="object_application" includeinduplicatecheck="false"/>
            <RevisionAttributeMapping>
                <AttributeMapping name="object_application"
tcattr="object_application" includeinduplicatecheck="false"/>
            </RevisionAttributeMapping>
        </AttributeMappings>
        <BHMElement type="mentorPadsLib" tctype="EDAComPADS"
behaviorType="GRM" reltype="IMAN_reference" reftype="mentorPadsLib"></
BHMElement>
    </ObjectMapping>

    <ObjectMapping type="Symbol" behaviorType="MIXED" tctype="EDASymbol">
        <AttributeMappings>
            <AttributeMapping name="name" tcattr="object_name"
includeinduplicatecheck="false"/>
            <AttributeMapping name="id" tcattr="item_id"
includeinduplicatecheck="false"/>
            <AttributeMapping name="description" tcattr="object_desc"
includeinduplicatecheck="false"/>
            <AttributeMapping name="object_application"
tcattr="object_application" includeinduplicatecheck="false"/>
            <RevisionAttributeMapping>
                <AttributeMapping name="object_application"
tcattr="object_application" includeinduplicatecheck="false"/>
            </RevisionAttributeMapping>
        </AttributeMappings>
        <BHMElement type="mentorPadsLib" tctype="EDAComPADS"
behaviorType="GRM" reltype="IMAN_reference" reftype="mentorPadsLib"></
BHMElement>
    </ObjectMapping>

    <ObjectMapping type="mentorPadsLib" behaviorType="GRM"
tctype="EDAComPADS">
        <AttributeMappings>
            <AttributeMapping name="name" tcattr="object_name"
includeinduplicatecheck="false"/>
        </AttributeMappings>
        <FileMapping>
            <FileMap fileExt="*" tcNameReferencedType="*"  />
        </FileMapping>
    </ObjectMapping>

    <FileMapping>
        <FileMap fileExt="txt" tcDatasetType="Text"
tcNameReferencedType="Text" />
        <FileMap fileExt="png" tcDatasetType="Image"
```


© 2025 Siemens

```
tcNameReferencedType="Image" />
        <FileMap fileExt="html" tcDatasetType="HTML"
tcNameReferencedType="HTML" />
        <FileMap fileExt="jpg" tcDatasetType="JPEG"
tcNameReferencedType="JPEG_Reference" />
        <FileMap fileExt="zip" tcDatasetType="Zip"
tcNameReferencedType="ZIPFILE" />
        <FileMap fileExt="tif" tcDatasetType="TIF"
tcNameReferencedType="TIF_Reference" />
        <FileMap fileExt="gif" tcDatasetType="GIF"
tcNameReferencedType="GIF_Reference" />
        <FileMap fileExt="*" tcDatasetType="MISC"
tcNameReferencedType="MISC_TEXT" />
    </FileMapping>

    <!-- Sample classification mappings -->
    <!-- Do not edit toolClassifybyAttribute value -->
    <!--
    <ClassificationMapping>
        <AtributeCollection  toolClassId="Component">
            <ClassAttribute toolClassAttrId="Description"
tcClassAttrId="-4275" direction="TcToTool"/>
        </AtributeCollection>

        <Class toolClassId="Component" tcClassId="RNC363"
classifyAnchor="false">
        </Class>

        <Class toolClassId="Component"
toolClassifybyAttribute="classificationCategory"
toolClassifybyAttributeValue="TOP" tcClassId="RNC364"
classifyAnchor="false">
            <ClassAttribute toolClassAttrId="Status"
tcClassAttrId="-5489" direction="ToolToTc"/>
            <ClassAttribute toolClassAttrId="Currency"
tcClassAttrId="-4253" direction="ToolToTc"/>
        </Class>
    </ClassificationMapping>
    -->

</BHMIntegration>
```

Map ECAD library objects to Teamcenter objects

You can map ECAD objects with Teamcenter objects. For example, if you want to add the **Block** model from an ECAD tool to Teamcenter, perform the following steps:

1. **Create the object in Business Modeler IDE.**

© 2025 Siemens

If there is no Teamcenter object that maps to the ECAD object, create a new Teamcenter object using Business Modeler IDE.

For example, create an object called **ReusableBlock** as a subtype of **Item** in the Business Modeler IDE.

For more information about creating objects, see the BMIDE for Data Model Design.

2. **Configure the new object for classification.**

   Update the **ICS_classifiable_types** preference to enable classification of the Teamcenter object, for example:

   ```
   ICS_classifiable_types=ReusableBlock, ReusableBlock Revision
   ```

   For more information on classification, refer to the Basic Classification — Deployment and Administration help on Support Center.

3. **Modify the Integration Definition File file to map the library object to Teamcenter object.**

   You can **customize the mappings in Teamcenter** by downloading the Integration Definition File from the dataset for respective integration, editing the file, and importing the file back into the dataset in Teamcenter.

   You can create classification mapping for ECAD parts with Teamcenter classes in the Integration Definition File.

Creating classification mapping for ECAD parts with Teamcenter classes

Overview of classifying ECAD library data

The Classification application utilizes a classification hierarchy to categorize your company's product data. As the system administrator, you use Classification Admin to define the groups, classes, and views that form the classification hierarchy. You also define and format the attributes that, when associated with a class, determine the type of information that is stored.

You can import and export classification hierarchy data using either the import and export features in the Classification Admin application or using command line utilities. Classification Admin supports both legacy XML files, as well as the industry standard PLM XML.

As a library administrator, you can **create a custom classification schema** using the Teamcenter Classification Admin application.

After you import or create the classification schema, **you must set up Classification mapping**. This is a one-time process and the classification mapping must be updated only if the classification schema is changed. Classification mapping converts the part category in the ECAD tool library to the class ID

© 2025 Siemens

defined in Teamcenter Classification and the class ID in Teamcenter Classification to the part category in ECAD tool library.

Because the classification schema may contain a long list of classes, the **edalib_generate_classification_csv_maps** utility is provided to pre-populate the Classification mapping files.

Before you begin

As a library administrator, you must create a classification schema using the Teamcenter Classification Admin application. You can also create a custom classification schema if you are using custom business objects.

Before you start working with the Teamcenter Classification Admin application:

Prerequisites	Your classification administrator must have already created a classification hierarchy in the Classification Admin application before you can use Classification. For details about how to create a schema, see the *Basic Classification—Deployment and Administration* help. If you are using customized business objects (for example, customized item types), your administrator must add the names of these to the values of the ICS_classifiable_types preference.
Enable Classification	Classification does not need to be enabled before you use it. If you have trouble accessing Classification, see your system administrator; it may be a licensing issue. Note: You can log on to Teamcenter only once. If you try to log on to more than one workstation at a time, you see an error message.
Configure Classification	Before you begin using Classification, your Classification administrator must create the classification hierarchy.
Start Classification	Click **Classification** in the navigation pane. If you are not licensed to run the application, you see an error message and you are not able to perform any application functions.

Set up Classification mapping

After you import or create the classification schema, you must set up Classification mapping in the Integration Definition File. This is a one-time process and the classification mapping must be updated only if the classification schema is changed. Using the Integration Definition File, you can define classification of objects and attributes in your modeling tool to objects and attributes in Teamcenter.

© 2025 Siemens

For detailed instructions on how to classify objects and attributes between your ECAD tool and Teamcenter using the Integration Definition File, refer MBSE Integration Gateway for Integrating Modeling Tools help on Support Center.

An example of the classification mapping in the Integration Definition File is as follows:

```
<ClassificationMapping>
    <AtributeCollection  toolClassId="EDABOMComp">
        <ClassAttribute toolClassAttrId="PartNumber"
tcClassAttrId="-5485" direction="TcToTool"/>
        <ClassAttribute toolClassAttrId="Mass" tcClassAttrId="-5491"
direction="ToolToTc"/>
        <ClassAttribute toolClassAttrId="OperatingTemperature"
tcClassAttrId="-5517" direction="ToolToTc"/>
    </AtributeCollection>
    <Class toolClassId="EDABOMComp" tcClassId="RNC363"
classifyAnchor="false">
        <ClassAttribute toolClassAttrId="NumberOfFans"
tcClassAttrId="-5160"/>
        <ClassAttribute toolClassAttrId="FanDiameter"
tcClassAttrId="-5159"/>
        <ClassAttribute toolClassAttrId="UnitCapacity"
tcClassAttrId="-5158"/>
        <ClassAttribute toolClassAttrId="NumbMaximumSupplyPowererOfFans"
tcClassAttrId="-5157"/>
    </Class>
    <Class toolClassId="EDABOMComp"
toolClassifybyAttribute="classificationCategory"
toolClassifybyAttributeValue="top/HEATSINK" tcClassId="RNC364"
classifyAnchor="false">
        <ClassAttribute toolClassAttrId="ThermalResistance"
tcClassAttrId="-6700" direction="TcToTool"/>
        <ClassAttribute toolClassAttrId="Height" tcClassAttrId="-6643"
direction="ToolToTc"/>
        <ClassAttribute toolClassAttrId="Length" tcClassAttrId="-6641"
direction="ToolToTc"/>
        <ClassAttribute toolClassAttrId="FinThickness"
tcClassAttrId="-5182" direction="ToolToTc"/>
    </Class>
</ClassificationMapping>
```

Note:

The mapping files must contain the `classId` value of components mapped to the EDA class only. If any component is mapped to any class other than the EDA classes, for example, mechanical class, such components should not be mentioned in the in the Integration Definition File.

© 2025 Siemens

The value for the *toolClassifybyAttribute* should always be fixed as *classificationCategory*.

How to use a custom part type in the EDA library integration

Overview of custom part types, objects, and relations

Organizations produce several documents that describe, are used by, or in some way relate to an item. For the data to be understood, it has to be associated in a meaningful way to the item or item revision. Teamcenter associates data to items and item revisions using relations.

Teamcenter EDA provides a basic set of part types, objects, and relations. The relations between the object and the item and item revision determines the rules enforced. By default, Teamcenter objects and relations are created during the EDA library integration. These OOTB objects and relations can be configured using the Teamcenter EDA library preferences.

When a user exports, saves, or synchronizes the ECAD library part from the ECAD tool to Teamcenter, the ECAD part is created in Teamcenter and all objects and relationships defined in Teamcenter will be associated with the new library part. You can **use custom part types** by modifying default preference values and modifying the default query details in Teamcenter.

The administrator can also use custom objects and relations by modifying the default values of the preferences that are used to create the default Teamcenter objects and relations. You can refer to the EDA library preferences that are used to create the **OOTB objects** and the **OOTB relations** in Teamcenter.

Use a custom part type for EDA library integration

When an ECAD designer exports, saves, or synchronizes the ECAD library part from the ECAD tool to Teamcenter, the ECAD part is created in Teamcenter and all objects and relationships defined in Teamcenter will be associated with the new library part. Teamcenter EDA provides a basic set of part types, objects, and relations. You can also use custom part types by modifying the default preference values and modifying the default query details in Teamcenter.

To use a custom part type instead of the default part type for the EDA library integration:

1. Specify your custom part type in the respective Object Mapping section in the Integration Definition File.

2. Check the default value defined for the **EDALIB_Sync_FindNewParts** preference. This default value represents the query saved in Teamcenter to find new parts for EDA library synchronization. The default value is **__EDALIB_find_newparts**.

3. In the Query Builder application, search for the **__EDALIB_find_newparts** query and change the default search criteria value defined in this query for the **object_type** attribute with the custom part type defined in the step 1.

© 2025 Siemens

For more information on how to modify and create a query, refer to the Query Builder documentation on Support Center.

The administrator can also use custom objects and relations by modifying the default values of the preferences that are used to create the default Teamcenter objects and relations. You can refer to the EDA library preferences that are used to create the **OOTB objects** and the **OOTB relations** in Teamcenter.

OOTB Teamcenter objects created during the EDA library integration

Teamcenter objects are created during the EDA Library integration with OOTB configurations using the EDA library preferences in Teamcenter.

The following EDA library preferences are used for creating the OOTB Teamcenter objects:

Preference name	Description	Default values
EDAPadstackRelation_relation_primary	This preference is used to define the primary relation objects for the EDAPadstack relation for RAC.	**EDAFootprint Revision** **EDAPackage Revision**
ICS_classifiable_types	This preference is used to identify whether a particular type can be classified or not. It is recommended that the default value should be set as Item/ItemRevision	**EDAComPart** **EDAComPart Revision** **Epl0EDALibPart** **Epl0EDALibPartRevision** **EDASymbol** **EDASymbol Revision** **EDAFootprint** **EDAFootprint Revision** **Vendor** **VendorRevision**
ICS_attribute_mapping_-unit_support	This preference is used to identify whether a particular type of object can be supported for attribute mapping not.	**EDAComMentorBS** **EDAComPADS** **EDAComExpedition** **EDAComCR5000** **EDAComCADStar** **EDAComPCAD**

© 2025 Siemens

Preference name	Description	Default values
		Epl0EDXPackage **EDAComPart** **Epl0EDALibPart** **EDASymbol** **EDAFootprint**

Note:

The **EDA_ComponentItemTypes**, **EDALIB_PartTypeDefault**, **EDALIB_Classification_Object_Type**, **EDALIB_SymbolTypeDefault**, **EDALIB_FootprintTypeDefault**, **EDALIB_PadstackTypeDefault**, **EDALIB_GDETypesMapping**, **EDALIB_DatasetTypesMapping, EDALIB_ManagedObjects**, **EDALIB_AttributesForms_mentorEDX,** and **EDALIB_ItemTypesMapping** preferences are now replaced by mappings in the Integration Definition File for respective library integration.

Based on your connector configurations, specific object types will be used. A list of all supported Teamcenter object types is given below for reference only:

EDAComPart	**EDAComPADS**
EDAComPartMaster	**EDAComPCAD**
EDAComPartVerMaster	**Epl0EDXPackage**
EDAComPart_0_Revision_alt	**EDAComPart Master**
EDAFootprint	**EDAComPart Revision Master**
EDAFootprintMaster	**EDAFootprint Master**
EDAFootprintVerMaster	**EDAFootprint Revision Master**
EDAFootprint_0_Revision_alt	**EDAPackage Master**
EDAPackage	**EDAPackage Revision Master**
EDAPackageMaster	**EDAShape Master**
EDAPackageVerMaster	**EDAShape Revision Master**
EDAPackage_0_Revision_alt	**EDASymbol Master**
EDAPadstackStorage	**EDASymbol Revision Master**
EDAShape	**Epl0AttrForm**

© 2025 Siemens

EDAShapeMaster	**Epl0EDALibPartMaster**
EDAShapeVerMaster	**Epl0EDALibPartRevisionMaster**
EDAShape_0_Revision_alt	**EDAPadstack**
EDASymbol	**EDAComPart Revision**
EDASymbolMaster	**EDAFootprint Revision**
EDASymbolVerMaster	**EDAPackage Revision**
EDASymbol_0_Revision_alt	**EDAPadstackRelation**
Epl0AttrFormStorage	**EDAShape Revision**
Epl0EDALibPart	**EDASymbol Revision**
Epl0EDALibPartMasterS	**Vendor**
Epl0EDALibPartRevMasterS	**VendorRevision**
Epl0EDALibPartRevision	**ManufacturerPart**
EDAComExpedition	**EDAComMentorBS**

OOTB Teamcenter relations configured during the EDA library integration

Teamcenter relations are created by default during the EDA Library integration with OOTB configurations using the EDA library preferences in Teamcenter.

The following EDA library preferences are used for creating the OOTB Teamcenter relations:

Preference name	Description	Default values
EDAPadstack_shown_-relations	This preference is used to specify the relations to display when expanding a node for an object of type **EDAPadstack**.	**IMAN_Rendering** **IMAN_reference**
EDAComPart Revision_-DefaultChildProperties	Lists the properties that can be used as children for the type **EDAComPart Revision**.	**IMAN_master_form_rev** **bom_view_tags** **IMAN_requirement** **IMAN_manifestation** **IMAN_reference** **IMAN_MEView**

© 2025 Siemens

Preference name	Description	Default values
		IMAN_vi_sos **IMAN_external_object_link** **IMAN_specification** **TC_WorkContext_Relation** **TC_AuditLog** **TC_Is_Represented_By** **release_status_list** **vendors** **vendorparts** **structure_revisions** **view** **Epl0HasSimulation**
EDAComPart Revision_PseudoFolder	This preference is used to list the properties that can be displayed as a pseudo-folder under a node for objects of type **EDAComPart Revision** object in the Rich client.	**CAEAnalysis** **MEProcess** **MESetup** **view** **Epl0HasSimulation**
EDAFootprint Revision_-DefaultChildProperties	This preference is used to list the properties that can be used as children for the type **EDAFootprint Revision**.	**IMAN_master_form_rev** **IMAN_reference** **EDAPadstackRelation**
EDAPackage Revision_-DefaultChildProperties	This preference is used to list the properties that can be used as children for the type **EDAPackage Revision**.	**IMAN_master_form_rev** **IMAN_reference** **EDAPadstackRelation**
EDAPadstack_-DefaultChildProperties	This preference is used to list the properties that can be used as children for the type **EDAPadstack** object in the rich client.	**IMAN_Rendering** **IMAN_reference**
Epl0EDALibPartRevision_-DefaultChildProperties	This preference is used to list the properties that can be used as	**IMAN_master_form_rev** **bom_view_tags** **IMAN_requirement**

© 2025 Siemens

Preference name	Description	Default values
	children for the type **Epl0EDALibPartRevision** object in the rich client.	**IMAN_manifestation** **IMAN_reference** **IMAN_MEView** **IMAN_vi_sos** **IMAN_external_object_link** **IMAN_specification** **TC_WorkContext_Relation** **TC_AuditLog** **TC_Is_Represented_By** **release_status_list** **structure_revisions** **view** **Epl0HasSimulation** **Epl0HasComponent**
Epl0EDALibPartRevision_-PseudoFolder	This preference is used to list the properties that can be displayed as a pseudo-folder under a node for objects of type **Epl0EDALibPartRevision** object in the Rich client.	**Epl0HasSimulation** **Epl0HasComponent**
IMAN_reference_relation_-primary	This preference is used to indicate the list of valid primary object types for the relation **IMAN_reference**.	**EDAPadstack**

Note:

The **EDAComPart Revision_<DatasetType>_default_relation**, **EDAFootprint Revision_<DatasetType>_default_relation** , **EDAPackage Revision_<DatasetType>_default_relation** , **EDAPadstack_<DatasetType>_default_relation**, and **EDASymbol Revision_<DatasetType>_default_relation** preferences are now replaced by mappings in the Integration Definition File for respective library integration.

Based on your connector configurations, specific Teamcenter relations will be used. A list of all supported Teamcenter relations is given below for reference only:

© 2025 Siemens

EDAPadstackRelation
Epl0HasAttrForm
Epl0HasComponent
Epl0HasEDXPackage
Epl0HasSimulation
IMAN_reference
view
VMRepresents
TC_vendor_part_rel
IMAN_Rendering
IMAN_master_form_rev
bom_view_tags
IMAN_requirement
IMAN_manifestation
IMAN_MEView
IMAN_vi_sos
IMAN_external_object_link
IMAN_specification
TC_WorkContext_Relation
TC_AuditLog
TC_Is_Represented_By
release_status_list
vendors
vendorparts
structure_revisions

© 2025 Siemens

Transfer mode used for EDA library integration

EDA library integration uses the following Transfer Mode information while creating export records. For more information about the transfer modes and how they work, refer the Tc XML and PLM XML Configuration for Data Import and Export help in the Teamcenter documentation.

Transfer Option Set	Transfer Mode	Closure Rule (Transfer Rule)	Property Set
TIEOptionSetEdaLib	**TIEEdaLibExport**	**TIEEDALibExport**	**TIEExportDefaultPS**

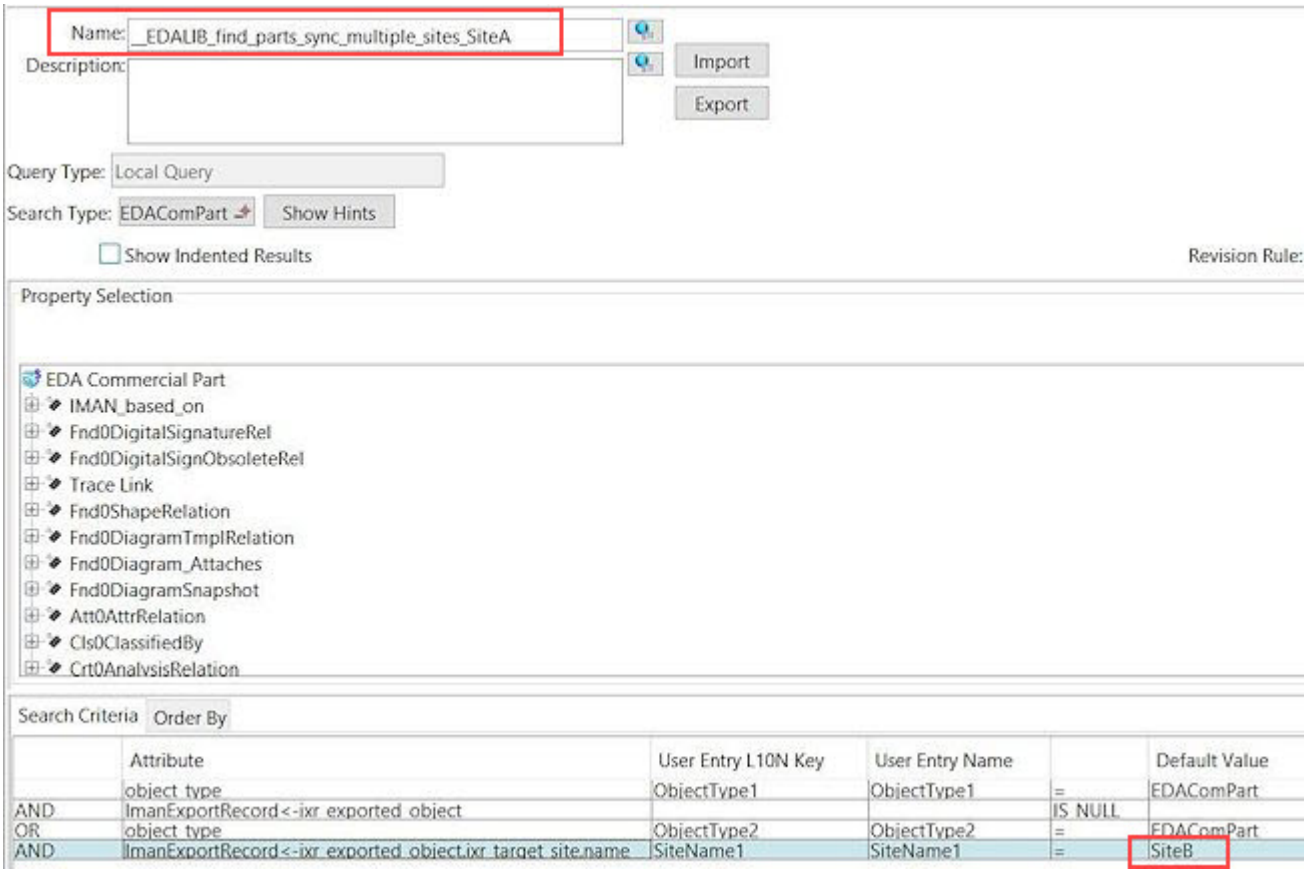
Synchronize ECAD library parts with multiple external sites

To enable ECAD users synchronize and reuse their part data across multiple external sites, you create a query in the Query Builder application to support this synchronization. Consider an example where you want to synchronize a part that is already synchronized with *SiteB* to a different site, *SiteA*.

1. Create a new query **__EDALIB_find_parts_sync_multiple_sites_<site name>** in the Query Builder application using an existing query **__EDALIB_find_parts_sync_multiple_sites** where *<site name>* is the name of the site to which you want to synchronize the parts.

   In our example, this is **__EDALIB_find_parts_sync_multiple_sites_SiteA**, where *SiteA* is the site to which you want to synchronize the parts.

© 2025 Siemens



2. Specify the site name (*SiteB*) in the query to which the part is already synchronized.

   Criteria for creating the query:

   - The query must search for parts that are not yet synchronized to any site (new parts) **AND** the parts that are synchronized to a site (*SiteB*).
   - The above condition to search new parts and existing parts for a site (*SiteB*) is separated by the **OR** function.
   - In each condition, the object type **EDAComPart** is specified with a different **User Entry L10NKey** (*ObjectType1*) and **User Entry Name** (*ObjectType2*) name.

3. Save the query. By default, the type of the query is **Local Query**.

4. To change the above query to a **User Query**, from the Teamcenter command prompt, run the following command:

   %TC_BIN% **\tc_set_query_where_run –u=***Tc-admin-user* **–p=***password* **–g=***group* **-query=***__EDALIB_find_parts_sync_multiple_sites_SiteA*" **–run=***query*

© 2025 Siemens

5. Set the value of the preference **EDALIB_Sync_FindNewParts** with the new query name (**__EDALIB_find_parts_sync_multiple_sites_<site name>**).

   You can also specify this query name in the integration-specific preference. For example, in the Xpedition EDX integration, the value can be set to the preference **EDALIB_Sync_FindNewParts_mentorEDX**.

To modify a query in the Query Builder application, you must first change the query type to **Local Query**. To do this, from the Teamcenter command prompt, run the following command:

%TC_BIN% **\tc_set_query_where_run –u=***Tc-admin-user* **–p=***password* **–g=***group* **-query=***__EDALIB_find_parts_sync_multiple_sites_SiteA*" **–run=***iman*

After making the required changes to the query, run the command specified in the Step 4 to change it to **User Query** again.

Managing and configuring Teamcenter preferences for part library management

You can set system-wide preferences on the Teamcenter server that apply to all EDA users. To understand how preferences work and to understand how to access a list of all supported preferences, refer to the Managing Preferences help on Support Center. All EDA preferences begin with the prefix **EDALIB**.

1. Run the rich client on the Teamcenter server and log on as an administrator.

2. In My Teamcenter, choose **Edit→Options**.

3. At the bottom of the **Options** dialog box, click **Index**.

4. In the **Preferences** dialog box, search for the preferences that begin with **EDALIB_**.

5. Change the preference value and click the **Modify** button to save the new value.

Installing the EDA client for part library management

Install the EDA client for part library management

Use the **tem.bat** command from the Teamcenter software kit to launch Teamcenter EDA Environment Manager (TEM) to perform installation and maintenance operations of Teamcenter EDA on an ECAD workstation for part library integration.

TEM is available with the Teamcenter installation or upgrade images. Contact your Teamcenter installation administrator to obtain the correct product distribution image.

© 2025 Siemens

> Caution:
>
> Before installing the EDA client on your ECAD workstation, refer to all the prerequisites and the information required to install the client in the **Planning your deployment** topic.

1. Ensure that the proper version of JRE is installed and the JRE_HOME environment variable (32-bit system) or the JRE64_HOME environment variable (64-bit system) is set.

2. Extract the Teamcenter software distribution image to a temporary location.

3. Specify the path to the Java Runtime Environment (JRE) by setting the **JRE_HOME** environment variable on the ECAD workstation.

   You can also specify the JRE path when you launch TEM from the command line using the **-jre** path argument.

4. Start TEM.

   a. Browse to the root directory of the extracted Teamcenter EDA software distribution image.

   b. Right-click the **tem.bat** program icon and choose **Run as administrator**.

      TEM launches and displays the **Installer Language** panel.

   c. In the **Installer Language** panel, select a language and click **OK**.

      TEM displays the **Welcome to Teamcenter EDA** panel.

      Your language selection applies only to the TEM session and not the Teamcenter EDA installation.

5. In the **Welcome to Teamcenter EDA** panel, click **Install**.

6. In the **Configuration** panel, type the configuration ID and the description for the new Teamcenter EDA configuration.

   The configuration ID identifies your Teamcenter configuration when you maintain, upgrade, uninstall, or add features to the configuration. Installation log files are also named based on this ID.

7. Proceed to the **Features** panel and click **Next**.

8. Expand **EDA Client** and select the required options:

   - **ECAD Library Integrations**

© 2025 Siemens

Select the library tool support you require. This integrates the ECAD library tool with the EDA client. However, a separate installation of the ECAD tool is required.

- **Mentor PADS Library**
- **Mentor PADS Library Win32**
- **Mentor Xpedition EDM Library**
- **Mentor Xpedition Library**
- **Teamcenter Gateway for EDA Library**
- **Teamcenter Gateway for EDA Library Win32**

9. In the **Installation Directory** box, browse to a folder where the EDA client will be installed. For example, type **c:\EDA_client** for Windows.

   The client is installed to an **eda** subdirectory, for example, **c:\ECAD_client\eda** (Windows). This directory is known as *TCEDAECAD_ROOT*.

10. Click **Next**.

11. In the **FCC Client Cache (FCC)** panel, select **Use new FCC** and click **Next**.

12. In the **FCC Parents** panel, type the **Host** name of the Teamcenter server in the **FCC Parents** box.

13. In the **4-tier Server Configurations** panel, type the IP address of the Teamcenter server in the **4-tier Servers** box. For example:

    http://*host***:***7001***/tc**

    Replace *host* with the server name.

14. In the **EDA Client** panel:

    a. Change the path if needed in the **Staging Location** box.

       You must enter the full path to the directory where the ECAD designs are saved on the client. This directory is the working directory for EDA.

       If a staging directory already exists, you can continue using the same directory or select another folder as the staging directory.

© 2025 Siemens

> Caution:
>
> Do not use spaces in the cache or staging directory path. Some EDA functions do not work if there are spaces in these paths.

b. Clear the **Will this configuration be shared by multiple users?** check box to create user environment variables after installation. By default, the check box is selected to create system environment variables.

c. Click **Next**.

15. Review the features selected in **Confirmation** panel and click **Start**.

    The EDA client components are installed.

    The **Install** panel confirms whether the installation is successful. If the installation is successful, click **Close**. If the installation is unsuccessful, click **Show Details** and proceed as needed.

After the installation of EDA client is complete you must verify that the EDA client is installed correctly and **perform the configurations for setting up part library management**.

You can also perform additional **ECAD library tool customizations** for setting up library management integration.

Verify the EDA client installation

Verify the EDA client installation by checking the following:

1. Ensure that Teamcenter client communication system (TCCS) services point to the Teamcenter server with EDA server support.

2. The FMS **fcc.xml** file points to your Teamcenter server.

3. The **FMS_HOME** system environment variable is set and points to the location of the **fcc.xml** file.

4. The **TCEDAECAD_ROOT** user environment variable is set to your EDA directory (for example, **c:\EDA_client\eda** on Windows systems).

   The value of **TCEDAECAD_ROOT** is added to **PATH**.

5. The directory defined by the **TCEDAECAD_ROOT** user variable exists and is populated with files.

6. The relevant ECAD tool subdirectory exists (*<TCEDAECAD_ROOT>\eda*.

© 2025 Siemens

If you are using PADS, verify that the **PADS** subdirectory exists. If you are using Xpedition, verify that the **Mentor_Expedition** subdirectory exists.

7. The **TCEDAClient.properties** file in the *TCEDAECAD_ROOT* directory contains the values you entered using the installation routine, for example:

```
URL=http://hostname:port/tc
StagingDir=staging-directory-location
```

> Note:
>
> When you first run EDA, the staging directories are created from the settings in the **TCEDAClient.properties** file.
>
> The paths of the staging directories contain double back slashes (\\) on Windows systems. For example:
>
> *C:\\ecad\\EdaStaging*

Configure the EDA client for part library management

Steps to configure the EDA client for part library management

You can perform the following configurations and customizations to set up the EDA client for part library management.

- **Create customized gateway configurations for managing ECAD part data**

  The Teamcenter Gateway for part library management enables one or more ECAD part library applications to connect with the Teamcenter server to manage ECAD part data, customize configuration files, and also manage the relation between the library objects. Teamcenter provides some OOTB configuration files and templates that you can use to configure Teamcenter Gateway for EDA for part library management.

- **Create an ECAD library configuration**

- **Map ECAD library objects to Teamcenter objects using ODBC mapping**

  After mapping ECAD library objects to Teamcenter objects, you can map the properties of ECAD objects to the Teamcenter objects using the *Open Database Connectivity database applications (ODBC)* mapping. The ODBC mapping feature is only available if your ECAD tool supports ODBC databases.

- **Set up EDA part library synchronization with Teamcenter**

  To make specific design teams use only the latest data, ECAD library administrators can establish specific synchronization schedules and processes (or execute the process on demand) between

© 2025 Siemens

Teamcenter and an ECAD tool's local library. The synchronization process identifies any new or updated parts that need to be exported to local libraries. Prior to executing a synchronization process, the library administrator can browse the list to verify which parts should be updated in the local ECAD tool libraries.

Create customized gateway configurations for managing ECAD part data

Overview of configuring Teamcenter Gateway for part library management

The Teamcenter Gateway for part library management enables one or more ECAD part library applications to connect with the Teamcenter server to manage ECAD part data, customize configuration files, and also manage the relation between the library objects. ECAD library administrators can populate libraries, map attributes, modify data, save and check-in check-out ECAD parts, and define access rules from the gateway's user interface.

Before configuring Teamcenter Gateway for part library management, ensure that you install Teamcenter Gateway with library support during the **EDA client installation**.

Teamcenter provides some OOTB **configuration files and templates** that you can use to configure Teamcenter Gateway for EDA. Using these configuration files, ECAD librarians or administrators can:

- Automate the library import and export process.
- Add or update parts.
- Specify scripts in the callback feature to be called for pre- or post- Teamcenter operations.
- Synchronize attribute data with the ECAD tool data.

You can also **create your own configuration file for custom ECAD applications** and **add callbacks**.

You must restart the EDA gateway after performing the configurations.

About configuration files and templates used by Teamcenter Gateway

Configuration files define the various properties required by Teamcenter Gateway for EDA library integration. The configuration file is a readable XML file and can be created and edited using any ASCII text editor. The client configuration must be a valid XML file that is compliant with the supported schema. To help the administrators create their own customized configurations, configuration templates are provided in the EDA installation folder.

Configuration templates are located in the EDA client installation folder and can be used to create new customized gateway configurations. The templates provide defaults and contain place holders for you to configure. You can also edit the out-of-the-box configuration template to create a configuration. The configuration file should be named as *application-name*_**usrdef.xml** and should be located in *<EDA client installation>***/gateway** folder.

© 2025 Siemens

For a basic configuration, the minimum required properties are as follows:

- **Family**

  This is the subdirectory name under the top-level EDA staging directory, for example, **expedition**.

- **Application**

  The initial part of the file name is *application name*_**usrdef.xml**. This name appears in Teamcenter Gateway for EDA when you choose **File→Configuration** to select a configuration.

- **className**

  This refers to the Java class defining the model type for this configuration. These model types should not be modified from the templates.

When you start Teamcenter Gateway for EDA, you can either specify a configuration depending on your requirement or skip specifying the configuration at that point. If you do not specify the configuration to be used, a dialog box to select the configuration is displayed later. You can also switch to an alternate configuration after you start Teamcenter Gateway for EDA by choosing **File→Configuration** and selecting another configuration from the displayed list.

You can add your own elements and attributes as long as the EDA client configuration scheme is followed and the features added can be supported by Teamcenter Gateway for EDA. You can also delete the configurable elements and attributes that are not required.

> Note:
>
> A commented instance of an attribute mapping element example is also shown in the template for reference purposes. However, Siemens Digital Industries Software recommends that you configure this element only if you have a good understanding of Business Modeler IDE and EDA client configuration.

Create a new library gateway configuration file for your ECAD application

You can create the Teamcenter EDA library gateway application configuration file using the **application types available on the Teamcenter server**.

1. Create a library configuration file with the name **<application type>_edadef.xml**. For example, **TcEDA_edadef.xml**.

2. Use this application type as the value for the **application** attribute in *<application type>*_**edadef.xml**.

3. The value for the **family** attribute can be any name related to the ECAD library integration or it can be the default application type value.

© 2025 Siemens

4. Refer to the *%TCEDAECAD_ROOT%\example\docs\SchemaReference.doc* file for details about each XML element and attributes defined in configuration file *<application type>***_edadef.xml** to create the application configuration file.

```
<EDAClientDef
        application="mentorEDXLib"
        className="com.Teamcenter.edabase.lib.DefaultLibraryFactory"
        family="mentorEDX">

    <PrimaryDataType datasetType="Epl0EDXPackage"/>

    <LibraryDef
            SyncLibrary="false"
            SyncDatabase="false"
            SyncFile="true"
            Extension="*.edx"
    />

    <CallbackDefs>
        <callback
                operation="exportLibrary"
                type="java"

command="com.Teamcenter.eda.mentoredx.connector.MentorEDXLibConnector
:exportLibrary"
        />
        <callback
                operation="importLibrary"
                type="java"

command="com.Teamcenter.eda.mentoredx.connector.MentorEDXLibConnector
:importLibrary"
        />
    </CallbackDefs>

    <OperationData operationName="SaveLibraryOperation">
        <Attr name="linkToTCLib" value="true"/>
    </OperationData>

    <EDAGatewayDef/>

</EDAClientDef>
```

5. Copy this configuration file in *%TCEDAECAD_ROOT%***\gateway** folder. When you launch the Teamcenter EDA Gateway, this configuration file is displayed in the list of available configurations.

© 2025 Siemens

Specify scripts in the callback feature for pre- or post- Teamcenter operations

You can add a callback feature to your configuration. The callback configuration feature allows you to specify names of scripts to be called by Teamcenter Gateway for EDA before or after Teamcenter operations such as save or open. The Teamcenter Gateway for EDA integration framework supports a limited number of callback operations specially designed for the EDA integration.

The callback starts a custom script, which may be written to perform ECAD tool-specific tasks appropriate to that process. The name of each callback script and its type are defined in the configuration file. Whenever a callback is available, the gateway client calls it to prepare for pre- or post-Teamcenter operations. Each callback is specified by its operation type, command name, and a list of arguments.

For a detailed explanation of these callbacks, refer to the EDA library integration callback summary in the Callbacks in Teamcenter EDA help on Support Center. This help is also available at *%TCEDAECAD_ROOT%\example\docs\Callbacks.doc* in the EDA install folder.

Callback type	Argument 1	Argument 2	Argument 3	Argument 4
preLoadLibrary	**applicationName**	**edaLibXmlFileName**		
postLoadLibrary	**applicationName**	**edaLibXmlFileName**	**statusFileName**	
preExportLibrary	**applicationName**	**edaLibFileName**		
postExportLibrary	**applicationName**	**edaLibFileName**	**statusFileName**	
preSyncLibrary	**applicationName**	**edaLibFileName**		
postSyncLibrary	**applicationName**	**edaLibFileName**	**statusFileName**	
preSaveLibrary	**applicationName**	**selectedObjectXmlFileName**	**edaLibXmlFileName**	
postSaveLibrary	**applicationName**	**selectedObjectXmlFileName**	**edaLibXmlFileName**	**statusFileName**
preReviseLibrary	**applicationName**	**selectedObjectXmlFileName**		
postReviseLibrary	**applicationName**	**selectedObjectXmlFileName**	**edaLibXmlFileName**	**statusFileName**
preCheckoutLibrary	**applicationName**	**selectedObjectXmlFileName**		
postCheckoutLibrary	**applicationName**	**selectedObjectXmlFileName**	**edaLibXmlFileName**	**statusFileName**
preCheckinLibrary	**applicationName**	**selectedObjectXmlFileName**	**edaLibXmlFileName**	
postCheckinLibrary	**applicationName**	**selectedObjectXmlFileName**	**edaLibXmlFileName**	**statusFileName**

© 2025 Siemens

Callback type	Argument 1	Argument 2	Argument 3	Argument 4
preCancelCheckoutLibrary	**applicationName**	**selectedObjectXmlFileName**		
postCancelCheckoutLibrary	**applicationName**	**selectedObjectXmlFileName**	**edaLibXmlFileName**	**statusFileName**

Create an ECAD library configuration

1. In the EDA Gateway, choose the **Teamcenter→Configure** menu command.

2. In the **Library Integration Configuration** dialog box, click the **Configuration** tab and set the following values:

   - **Library Configuration**

     Specifies the name of the library configuration. To use an existing library configuration, choose a name from the list. Alternatively, you can type a unique name to create a new configuration. You can also click **Delete** to delete an existing configuration.

     > Note:
     >
     > You cannot delete the working library.

   - **Tc Part Library**

     Select the required Teamcenter part library from the list of library names. These part libraries are created in the Organization application in Teamcenter. Only those libraries are listed depending on the ECAD tool being integrated. You cannot enter a new library name.

   - **Configure ECAD Library**

     This option is used to manage the ECAD part library metadata that is stored in Teamcenter. You can use this option for PADS library integration.

     Click **Browse** and ensure that **Library Data Source** points to the correct library path.

     Select the **Set Data Source Folder as Read Only** check box to make the library read-only and protect the data on your workstation after synchronization is complete.

     Specify the following library data source for your ECAD tools:

ECAD tool name	Library data source
PADS	*%SDD_HOME%***Libraries\amd.pt9** *%SDD_HOME%***Libraries\amd.ld9**

© 2025 Siemens

ECAD tool name	Library data source
	*%SDD_HOME%***Libraries\amd.pd9**
	*%SDD_HOME%***Libraries\amd.ln9**

- **Configure Part Metadata Database**

  This option is used to manage the ECAD part library metadata that is stored in the databases that can be accessed using ODBC for the Xpedition, Cadence, Orcad, and Altium library integration.

  a. Select the **Configure Part Metadata Database** check box to configure the part metadata database.

  b. Click **Configure** to configure the data source and a mapping file.

  a. (Optional) In the **Configure ODBC Connector** dialog box, select a data source from the list installed on your workstation and then browse to a mapping file.

     The **Configure ODBC Connector** dialog box maps elements in the EDA part library to the table structure in the part metadata database. In this process, an **EDALibXML** element (component, symbol, or footprint) is mapped to a table, and an EDA property is mapped to a column of the table. Only the modified and new parts are synchronized.

     If the required mapping is not available, click **Create Mapping** to create a new ODBC table mapping, or click **Modify Mapping** to modify the current ODBC table mapping.

  b. (Optional) In the **ODBC Table Mapping** dialog box, the **Teamcenter Library Object** tree shows all available EDA elements. Only components and shapes (that is, footprint, symbol, and padstack) are valid.

     The attributes of the elements are listed in the **edalib.xml** file in the following format:

     ```
     <attr value="68p" itemId="900-2000000-006" name="Value"/>
     ```

     In this example, the displayed attribute name is **Value**.

     A. If the required attribute is not displayed, select the component and click the **Add** button + to add an attribute to the selected EDA element.

        Teamcenter EDA Gateway displays the **Add a property** dialog box, allowing you to define the name of the new attribute.

     B. If the required table is not displayed in the **Table Definition** list, click the **Add** button +.

© 2025 Siemens

Teamcenter EDA Gateway displays the **Add a table** dialog box, allowing you to define the name of the new table.

C. To add properties or columns to the table, select the table and click the **Add** button +.

Teamcenter EDA Gateway displays the **Add a column** dialog box, allowing you to define the name of the new column. The new column is added to the table, but no new column is added to the physical table.

D. Select a property in the **Teamcenter Library Object** tree and a column in the **Table Definition** tree, and then click **Map** button to add the table into the **Teamcenter Library Object** list.

E. When the list is complete, you can click **Save** to save the mapping result in a file in a specified folder.

- **Configure File Mode**

  This option is supported only for EDM library integration.

  In the **Sync Folder** box, type or browse to the location that will create a **.edx** file during the synchronization from Teamcenter. Typically, this location is the EDM inbox queue folder for Xpedition.

3. Click **OK** to close the dialog boxes and complete the configuration process.

Map ECAD library objects and attributes using ODBC

Overview of mapping library parts and Teamcenter object properties using ODBC

ODBC library integration supports the export of the part library data saved in external databases such as Oracle, Microsoft Access, or SQL in the Teamcenter database by using ODBC mapping. As an ECAD library user, you can map the library part attributes with the Teamcenter object properties using ODBC mapping if your ECAD tool supports ODBC databases.

For mappings using the ODBC connector, the primary key must be configured for the database so that the existing database entries are updated instead of inserting a new database entry.

> Note:
>
> ODBC mapping is not supported for PADS and EDM library integrations.

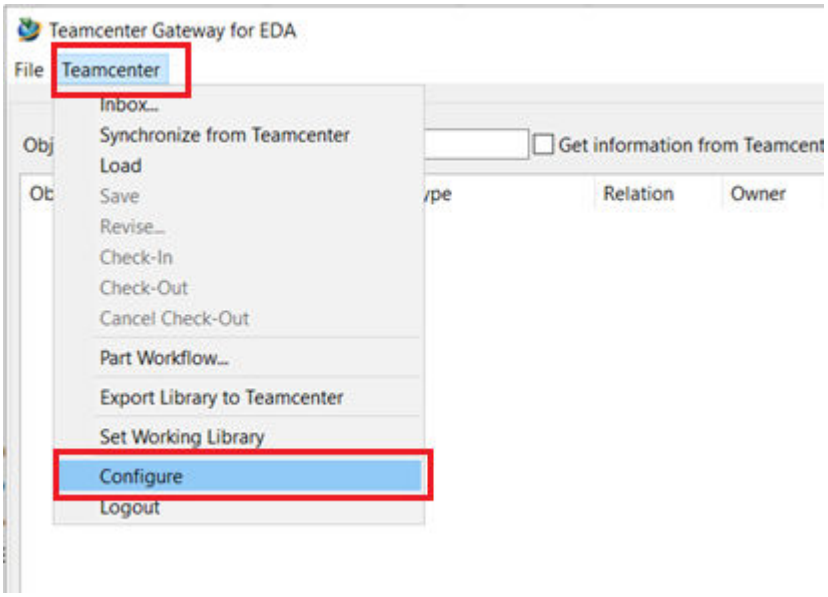
As an ECAD library administrator you can map the ECAD library parts and part attributes stored in a single table or in a multiple table database with the corresponding Teamcenter property.

© 2025 Siemens

Use ODBC to map ECAD part attributes with Teamcenter properties

To map ECAD library part attributes with the Teamcenter object properties using ODBC mapping:

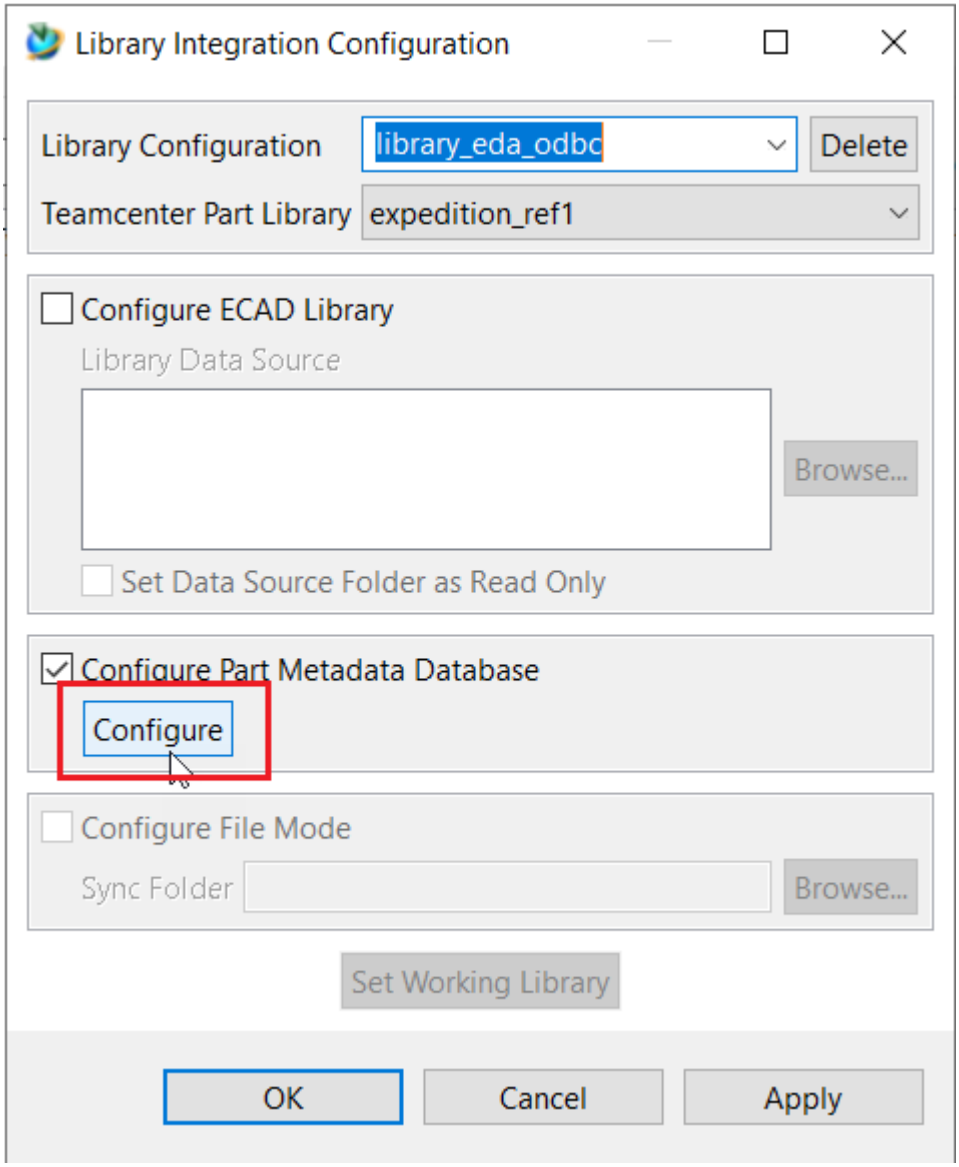
1. In EDA Gateway, choose the **Teamcenter→Configure** menu command.



2. In the **Library Integration Configuration** dialog box, select the **Configure Part Metadata Database** option.

3. Click **Configure** to configure the data source and a mapping file.

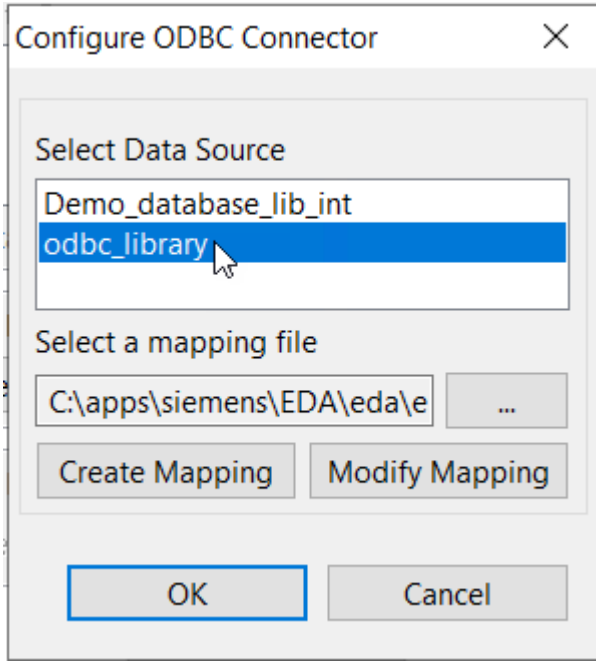
© 2025 Siemens



4. In the **Configure ODBC Connector** dialog box,

   - Select the data source from the list of all the data source names installed on your workstation.
   - Add your custom data sources to this list from Windows by choosing **Start→Control Panel→Administrative Tools→Data Sources**.
   - Add and configure your data source from the **ODBC Data Source Administrator** dialog box.

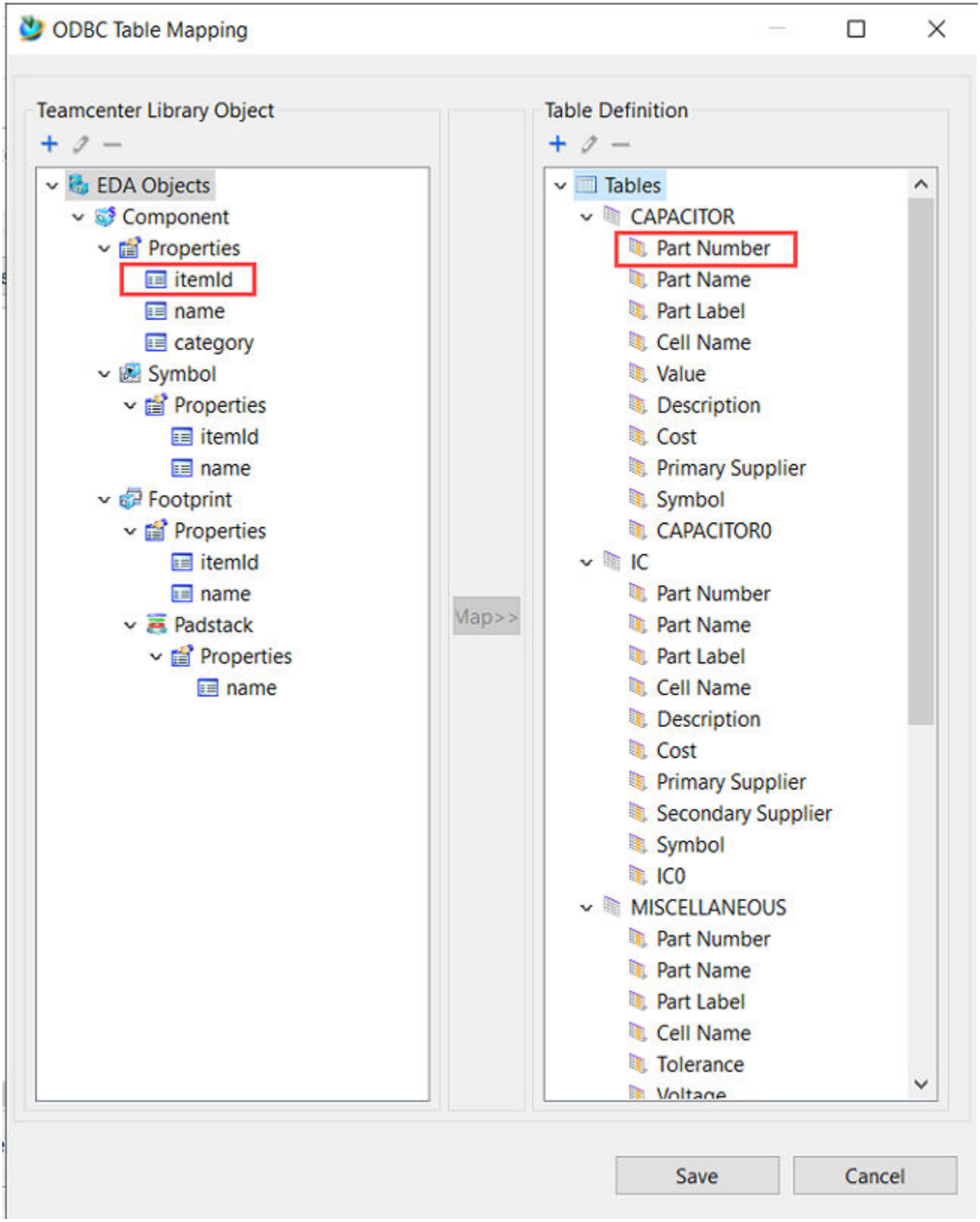
© 2025 Siemens



5. You can map elements in the EDA part library to the table structure in the part metadata database from the **Configure ODBC Connector** dialog box. In the mapping process, an **EDALibXML** element (component, symbol, or footprint) is mapped to a table and an EDA property to a column of the table.

   Browse to an existing mapping file and click **Modify Mapping** to modify the current ODBC table mapping. If the required mapping is not available, click **Create Mapping** to create a new ODBC table mapping.

© 2025 Siemens



6. **Add new EDA objects**

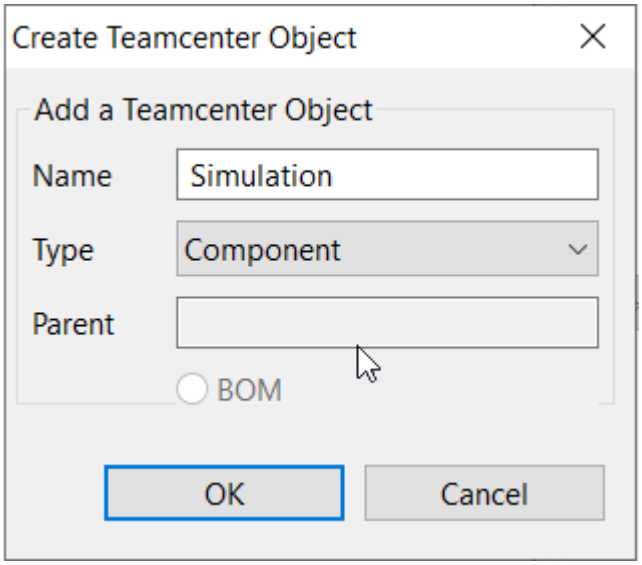
© 2025 Siemens

In the **ODBC Table Mapping** dialog box, the **Teamcenter Library Object** tree shows all the available EDA objects and its properties.

To add new object, for example **Simulation**:

a. Select **EDA objects** and click the **Add** button +.

Teamcenter displays the **Create Teamcenter Object** dialog box.



b. Specify the **Name** and **Type** of the new EDA object.

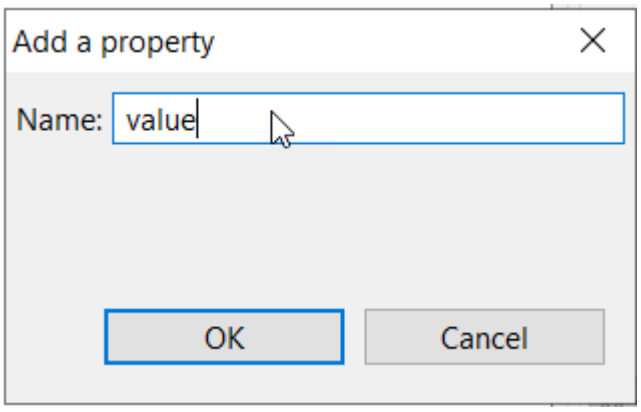
c. Click **OK**.

7. **Add additional properties**

a. Select the **Properties** node and click the **Add** + button.

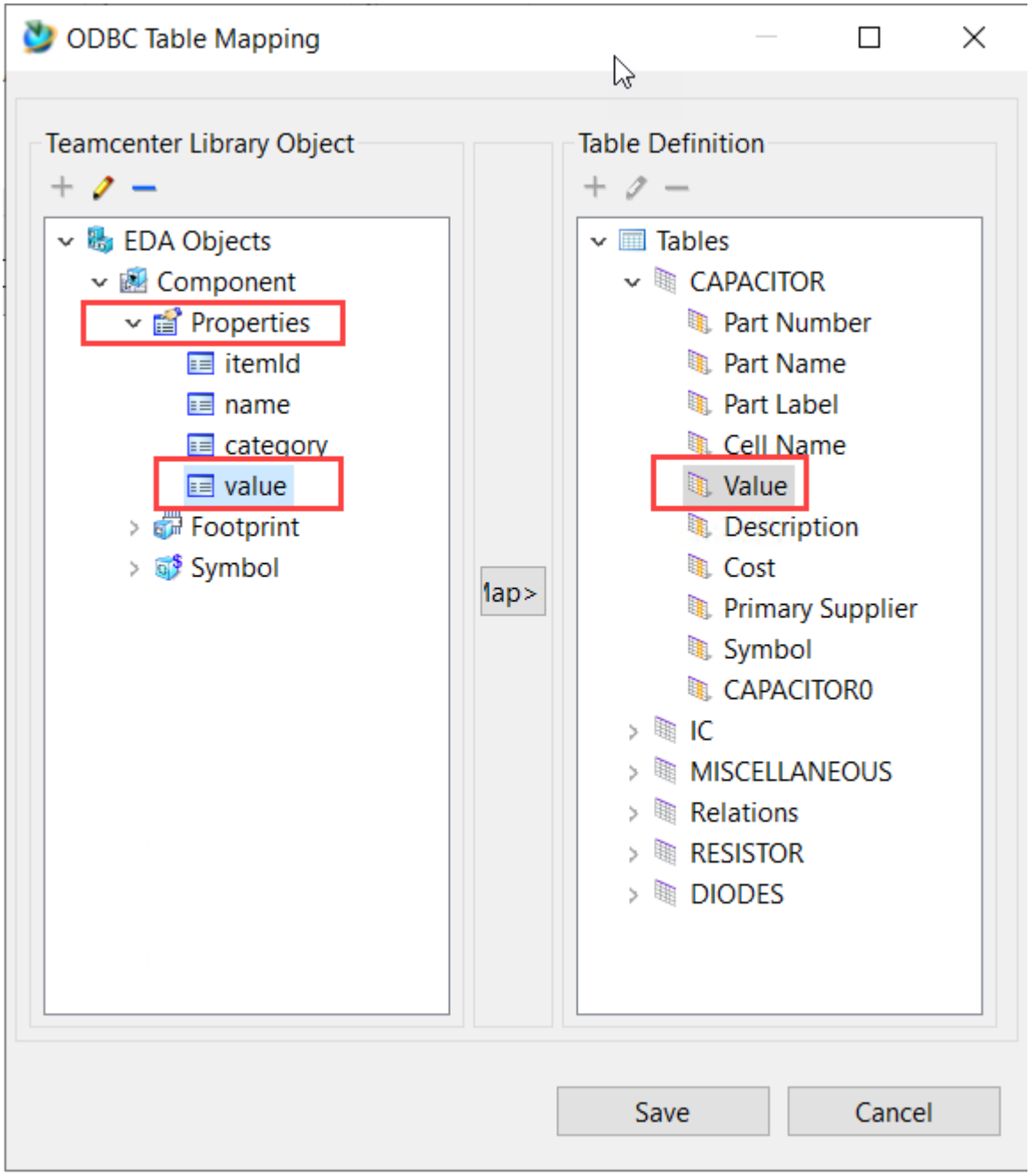
b. Specify a new property in the **Add Property** dialog box.

© 2025 Siemens



A new property *Value* is added under the **Properties** list.

© 2025 Siemens



c. Click **Save**.

> Note:
>
> Apart from the default Teamcenter properties, that is, **itemId** and **name**, if you add any other additional Teamcenter properties to map with the ECAD part attributes in the **ODBC Table Mapping** dialog box, you must **include these new Teamcenter properties in your attribute mapping file**. For example, in the attribute mapping file, we must add the new Teamcenter property, **Value**, for the configuration to work.

© 2025 Siemens

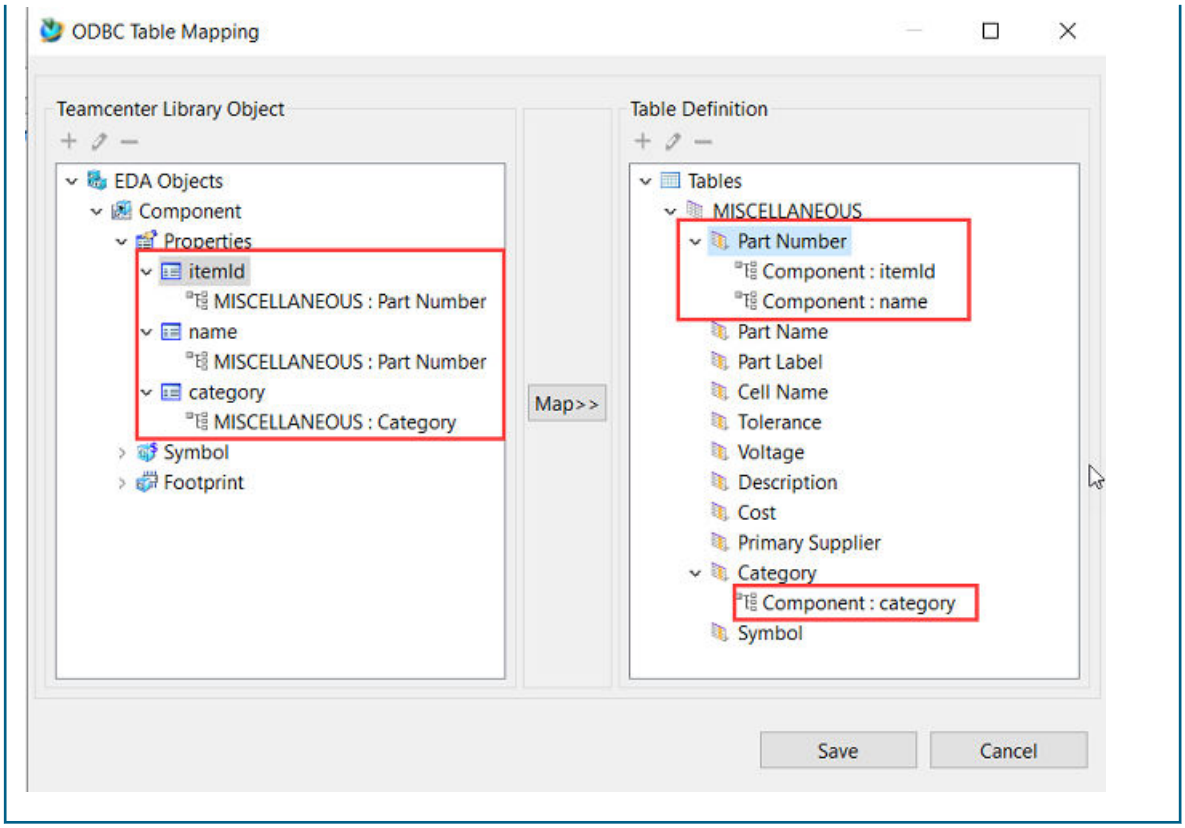
```
{ Item type="EDAComPart"
    MANUFACTURER : ItemRevision.ICS(-5488) /
description="MANUFACTURER"
    DESCRIPTION : Item.GRM(IMAN_master_form).object_desc /
description="DESCRIPTION"
    PACK_TYPE : ItemRevision.ICS(-5487) /description="PACK_TYPE"
    ROHS : ItemRevision.ICS(-4952) /description="ROHS"
    Cost : Item.GRM(IMAN_master_form).object_desc /
description="Cost"
    Value : Item.GRM(IMAN_master_form).object_desc /description="Value"
    ECADVersion :
ItemRevision.GRM(Epl0HasAttrForm,Epl0AttrForm).epl0ecad_version /
description="ECADVersion"
    ECADModified :
ItemRevision.GRM(Epl0HasAttrForm,Epl0AttrForm).epl0ecad_lmd /
description="ECADModified"
}
```

8. **Map ECAD part attributes with Teamcenter properties**

   a. Select a property in the **Teamcenter Library Object** tree and a column in the **Table Definition** tree, and then click **Map >>** to add the table into the **Teamcenter Library Object** list.

   > Note:
   >
   > The **name** attribute is mandatory and some ECAD applications do not have a corresponding attribute. In such a case, you can map the **name** attribute to the same ECAD attribute that you used to map the **item ID**.

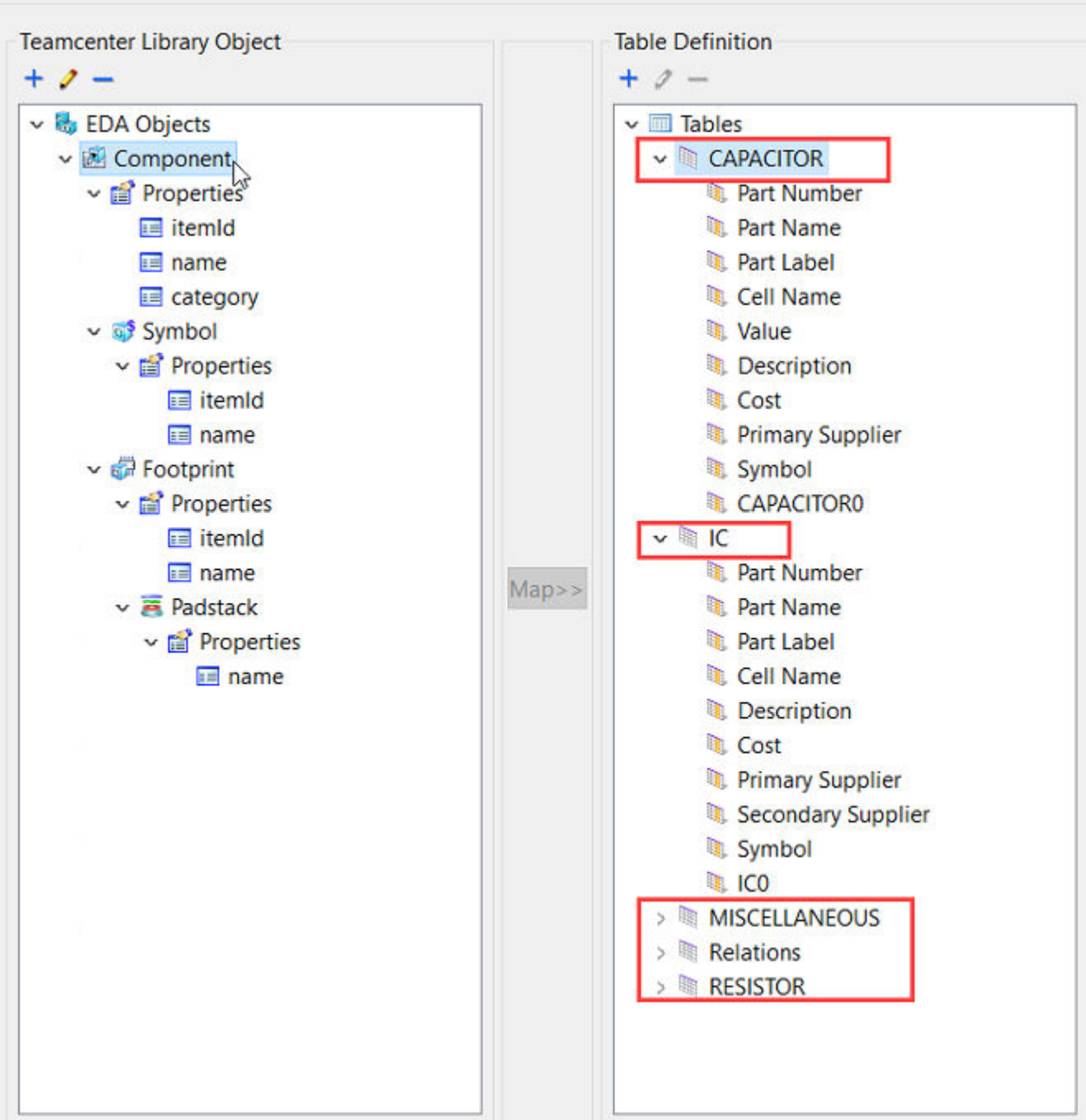
© 2025 Siemens



b. If your database is stored in multiple tables within a database file, you can map a specific column from all tables to the selected property in Teamcenter in a single step.

For example, in a multiple table database, each table represents a component type, for example, CAPCACITOR, IC, and RESITOR. The database can also have a table that contains mixed components types. In such a table, the component type is specified in the column **category**.

© 2025 Siemens

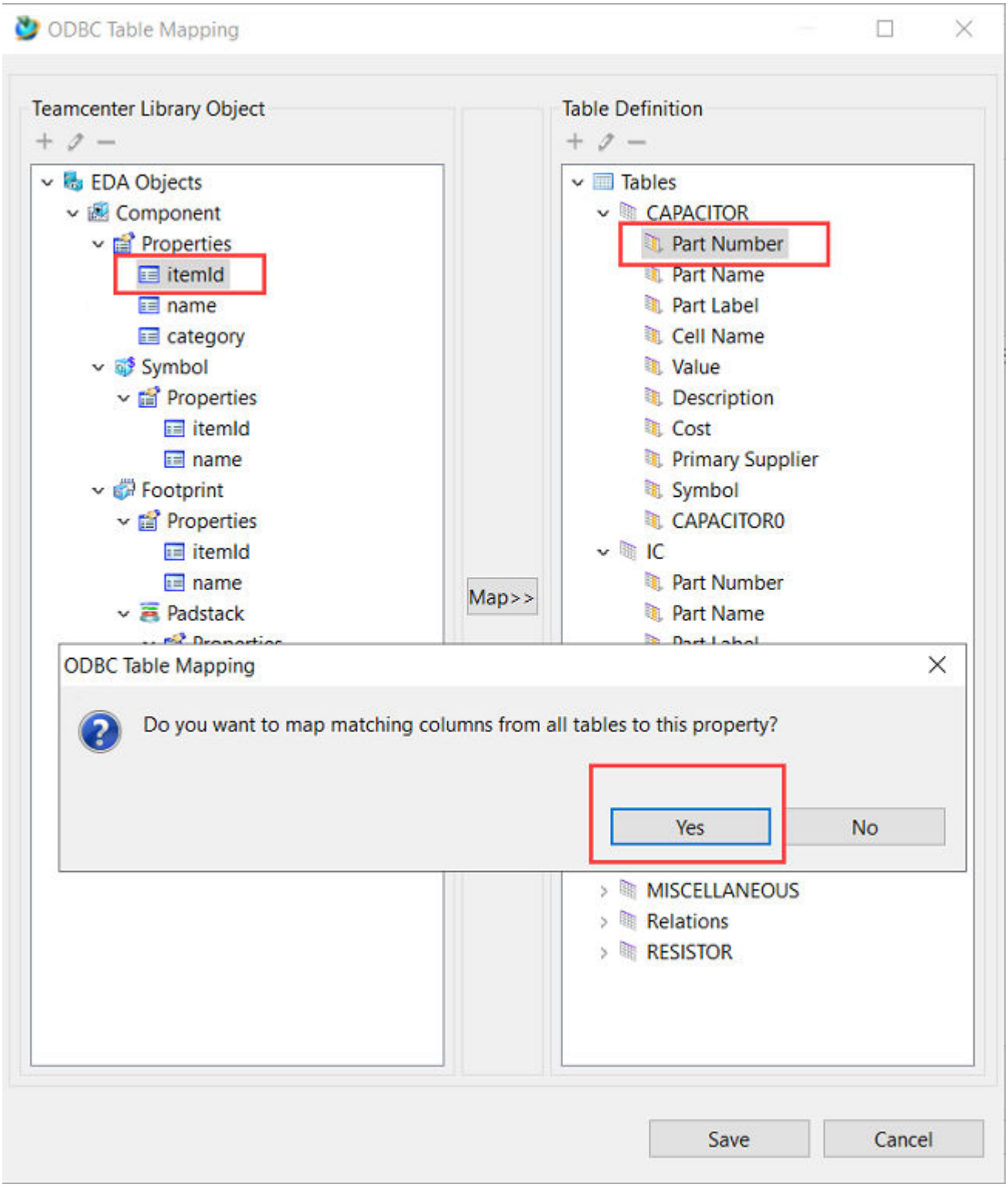


c. In a multiple table database, you can map a property in the **Teamcenter Library Object** tree with all matching columns from all the tables in the **Table Definition** tree in a single step.

   To do this, select a property in the **Teamcenter Library Object** tree and a column in the **Table Definition** tree and click **Map >>**.

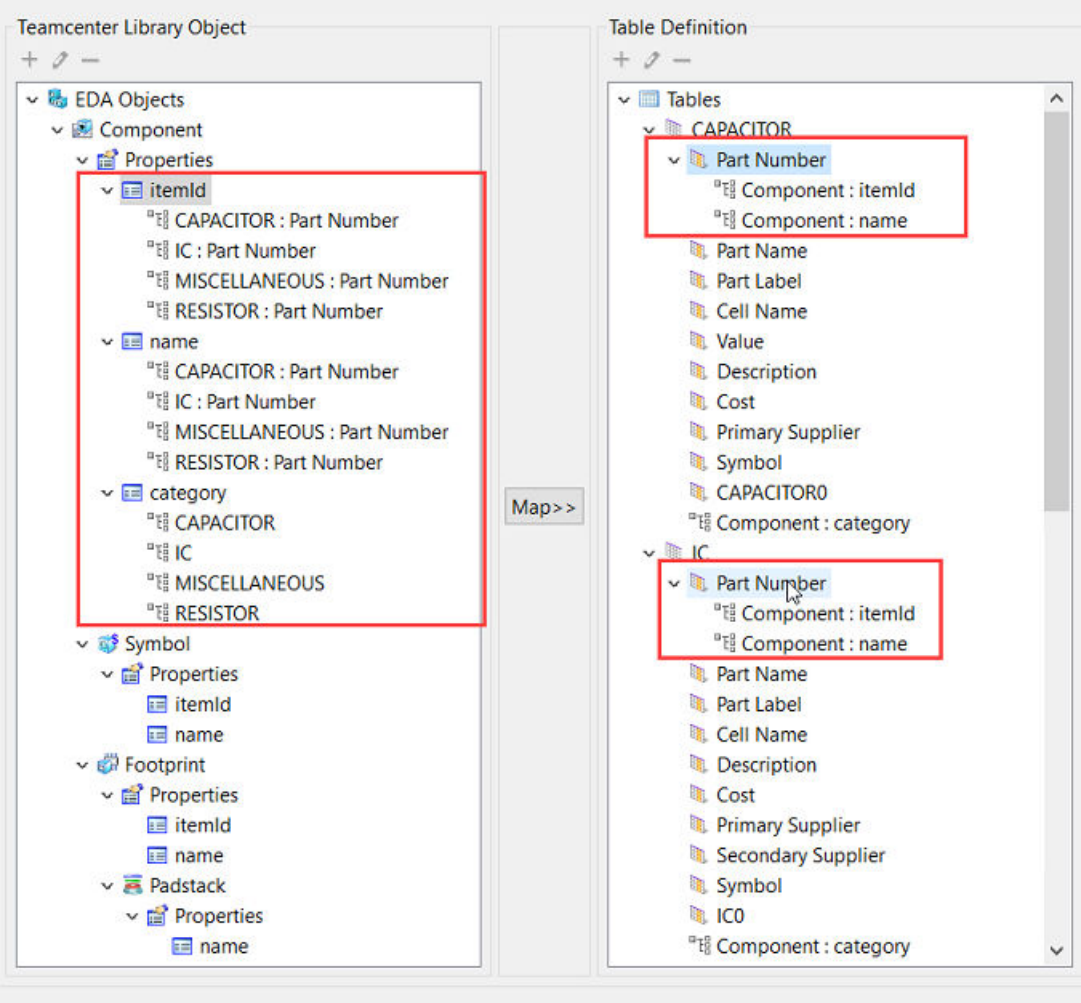
d. Click **Yes** to continue to map the selected property for all matching columns from all the tables in the database. If not, click **No**.

© 2025 Siemens



e. The Teamcenter EDA property is matched with all the matching columns from the multiple component tables.

© 2025 Siemens

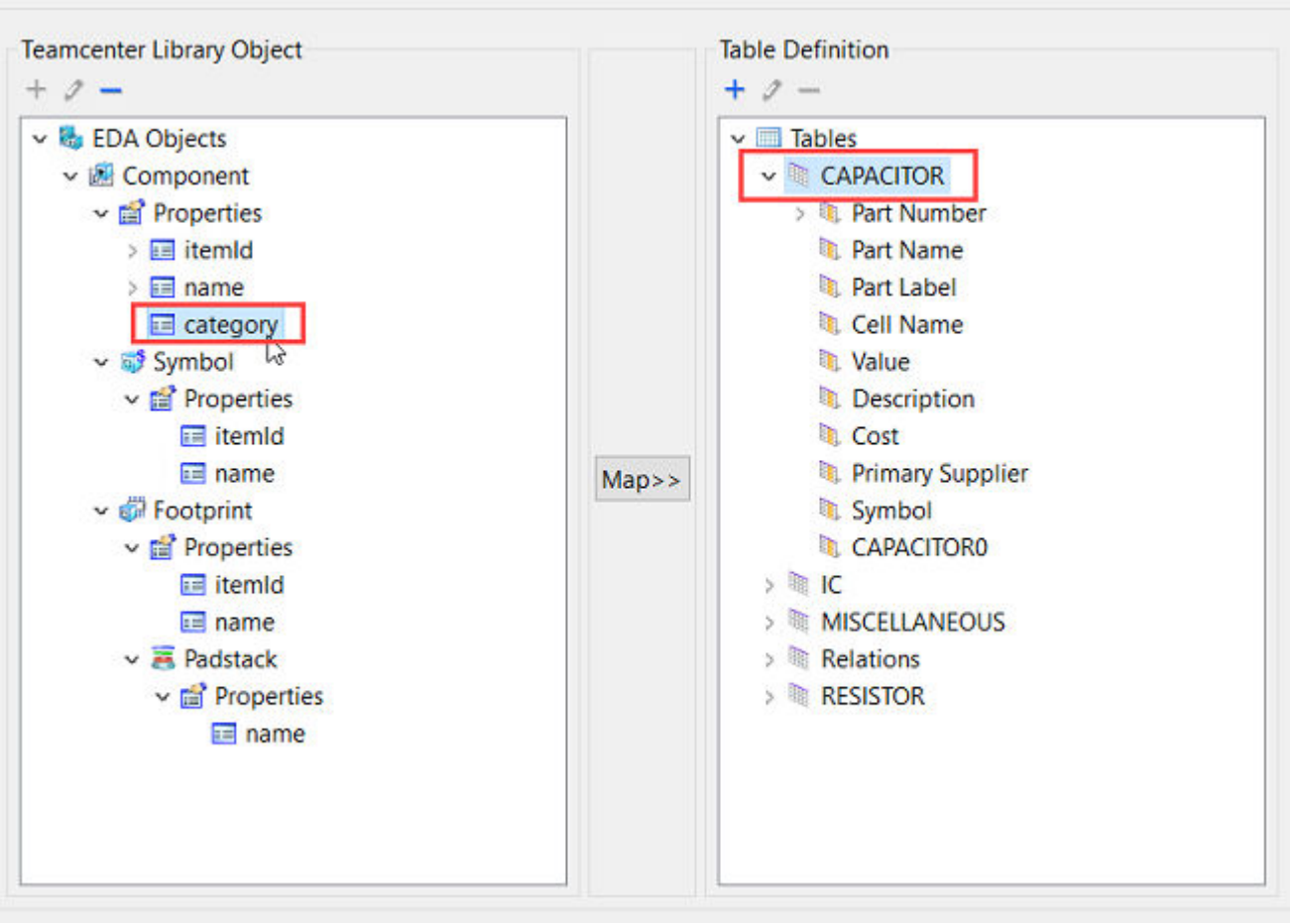


9. **Classify the ECAD component type with the respective Teamcenter EDA object property (category):**

   The table name that represents the ECAD component name, for example, **Capacitor**, can be directly mapped as the Teamcenter EDA **component** property for classifying the ECAD components in Teamcenter. You can map all table names in the **Table Definition** list at a time.

   a. Select the table name (for example, **Capacitor**) in the **Tables Definition** list with the corresponding property **category** in the **Teamcenter Library Object** list.

© 2025 Siemens



b. Click **Map >>** to classify the ECAD component types.

c. For the part data stored in multiple tables and each table name representing a component's category, click **Yes** to map all the table names with the Teamcenter EDA **component** property at a time. Click **No** to continue mapping the selected table only.

d. In scenarios where the database table does not represent a specific component type, but a list of different components whose type is defined in a column of that table, you may want to change the component mapping to a column in that table instead of the table name.

To do this, select the Teamcenter EDA **component** property and select a column of the table in the **Table Definition** list and click **Map >>**.

> Example:
>
> For example, a *Miscellaneous* table may contain different part types, where the part type is defined in the column *category* within that table. You can change the mapping of the **Category** property in the **Teamcenter Library Object** list with the previously

© 2025 Siemens

mapped table name, that is, *Miscellaneous* in the **Tables Definition** list to a column, *category*, within the *Miscellaneous* table.

The category mapping is now applied to the selected column for that table only.

10. When the list is complete, click **Save** to save the mapping result in a file in a specified folder.

11. Click **OK** to close the dialog boxes and complete the configuration process.

Note:

If you reclassify any ECAD part in Teamcenter as another ECAD part type (for example, if a previously classified capacitor part is reclassified as a resistor in Teamcenter), during the synchronization of parts from Teamcenter to the ODBC database, the part is copied to the new table. However, because all columns of both the tables may or may not be the same, the part is also retained in the previous table with **_MOVED_TO_<new table name>** suffix added to the original part name for reference purposes.

For example, a capacitor part *cap_1234* is now reclassified as a resistor part in Teamcenter. During the synchronization process, the part *cap_1234* is added to the *Resistor* table. The same part in the *Capacitor* table is not removed but is renamed as *cap_1234_MOVED_TO_Resistor*.

Set the working library

The ECAD part designer can be configured to maintain one or more ECAD libraries in the same workstation. You can switch between ECAD libraries and the current selection is referred to as the *working library*.

All EDA library operations use the working library. For example, during synchronization EDA will look for modified library objects only in the working library.

1. In Teamcenter Gateway for EDA, choose **Teamcenter→Set Working Library**.

   The **Set Working Library** dialog box appears.

2. Select the working library and click **OK**.

   The EDA library integration traverses the library and collects the object list (parts, symbols, footprints, and padstacks).

Set up EDA part library synchronization with Teamcenter

The EDA part library is synchronized with Teamcenter by running the **eda_sync.bat** command. Perform the following steps to select, edit, and run the script corresponding to your ECAD system.

© 2025 Siemens

1. Create a Teamcenter proxy user account that will be used to perform the library integration sync operations.

2. In the **TcEDAClient.Properties** file, set the **User** value to match the Teamcenter proxy user ID.

3. Generate a file to hold the Teamcenter proxy user password, using the following command:

   Example: `eda_password_file.bat -p <proxy user psw> -pf C:\Siemens\EDASecure\file.txt`

4. Select the script corresponding to your ECAD system from the following list. The selected script is invoked in the task scheduler to synchronize the library periodically.

   Use **start_syncMentorExpLibrary** to synchronize the Xpedition library.

   Use **start_syncMentorEDXLibrary** to synchronize the Xpedition EDM library.

   This script takes an optional argument such as `- syncFolder <folder>`. This argument specifies a folder where the EDX file will be created. This argument overrides the **syncFolder** defined in the library configuration. This command creates an EDX file with the name **eda_sync_yyyyMMdd_HHmmss.edx** where *yyyyMMdd_HHmmss* refers to the current date and time. This file must then be manually imported into the Xpedition EDM library, unless the **syncFolder** has been set up as an EDM inbox queue.

   Use **start_syncMentorPADSLibrary** to synchronize the PADS library.

5. Edit the script that corresponds to your ECAD system as follows:

   Example:

   ```
   eda_sync.bat -syncLibrary -application mentorEDXLib –syncConfig “EDM Library” –pf D:\EDASecure\file.txt
   ```

   Where:

   -pf specifies the file holding encrypted Teamcenter password.

   -syncConfig specifies the synchronization configuration name.

   Example:

   ```
   edacli.bat -syncLibrary -application seexpLib -mode 0 -workingLibraryName X -edaLib X -status statusFile -pf passwordFile
   ```

   Where:

   -pf passwordFile specifies the file holding encrypted Teamcenter password.

© 2025 Siemens

6. In a task scheduler, create an entry that invokes the sync script edited in the previous step at the desired time and frequency.

Enable acistoprt translator

After you install and configure Dispatcher as per the deployment strategy, you must enable the **acistoprt** translator required for translation of 3D models into NX parts.

You can use TEM to install the translator or manually edit the **translator.xml** file to enable the translator. For instructions on how to enable the **acistoprt** translator using TEM, see the Dispatcher — Deployment and Administration help on Support Center.

The following is an example of how to edit the **translator.xml** file to enable the translator.

Prerequisites: Verify that the **TransExecutable dir** is correctly specified for this translator.

1. Open the **translator.xml** file from the *Dispatcher_Root***Module\conf** directory.

2. Set the **isactive** attribute to **true** to activate this translator.

   > Note:
   >
   > By default, this attribute is set to **false**.

3. Open the *translator***.bat** (Windows) from the *Dispatcher_Root***\Module\Translators***translator_name* directory and edit the **CHANGE_ME** tags.

   For more information about **CHANGE_ME** tags, see the **.bat** file for the translator.

4. Run the translator in standalone mode from the command prompt to verify whether it is working properly.

   For more information about running the translator in standalone mode, from a command prompt, change to the **Dispatcher\Translators***translator_name* directory and run the appropriate command for your platform:

   > Note:
   >
   > For more information, see the **Readme** file in the **Translators***translator_name* directory.
   >
   > Windows: *translator_name***.bat -help**

5. (Optional) To set up *Dispatcher-client-proxy-user* as the owner of the translated files, make the following change to the *translator_name***.bat** (Windows) file.

   NxTransDirect translator example:

© 2025 Siemens

```
"%UGII_BASE_DIR%\ugmanager\run_nxtrans.bat" -u=Dispatcher-client-proxy-user
-p=password
-changeOwnerToCad=false "-pim=yes" %*
```

If you set up *Dispatcher-client-proxy-user* as the owner of the translated files, you must create a dispatcher client access rule. For more information on how to create a dispatcher client access rule, see the Dispatcher — Deployment and Administration help.

After installation, you must add a rule to the access rule tree permitting the translation service proxy user to update attributes of a **DispatcherRequest** object. If this access rule is not created correctly, the dispatcher client reports errors.

Set up automatic synchronization of EDM part library with Teamcenter

The EDM part library is synchronized with Teamcenter by running the **EDM_sync.bat** command. Configure the script file corresponding to your EDM system and the script will then run periodically as configured.

For 3D models to be exported from EDM to Teamcenter, EDA gateway must be installed on the EDM server running on windows. If the EDM server is on a Linux machine, the outbox queue must be configured on a Windows machine and the EDA gateway must be installed on the same machine.

1. Edit the following values in the **EDM_sync.bat** script file located at *%TCEDAECAD_ROOT%* **\MentorEDXLibCon\wnti\bin**.

   - **USERNAME**: Specify the username to login to Teamcenter.
   - **PWDFILE**: Specify the path of the file holding encrypted Teamcenter password.
   - **WORKINGLIBNAME**: Specify the working library name from the EDA Client library configuration.

2. **EDM_sync.bat** must be manually invoked once. Later, it will continue to poll the EDM outbox queue folder for any newly generated **.edx** files.

Once the script starts, the *<filename>***.edx** file is generated in the **syncFolder** specified in the **WORKINGLIBNAME** argument in the script file and is automatically added or updated in Teamcenter.

The *<filename>***.RESULT** file is created in the **syncFolder** which provides the status of the sync operation. A *<filename>***.log** file is created that lists the components that fail to get exported to Teamcenter.

During the translation of 3D models from EDM to Teamcenter, the **.sab** file can be removed from the dataset after adding the translated the **.prt** file only if the user performing the translation has write access on the dataset.

© 2025 Siemens

ECAD tool customizations for part library management

Overview of ECAD tool customizations required for part library management

After the configurations required to set up part library management on the EDA client are complete you must further configure and customize your ECAD tool connector as per your requirement.

- **Setting up PADS Library integration**
- **Setting up Xpedition Library integration**
- **Setting up EDX Library integration**

Setting up PADS Library integration

To set up PADS Library integration

1. Open a console window and enter the command **eda_gateway**.
2. On the **Configuration** dialog box, select **mentorPadsLib**.

To modify the default properties

Various properties to control the execution of the library tool may be set in file *%TEMP%* **\eda\mentorPadsLibCon.ini**. This file is created the first time when the integration is run and a configuration is created.

- **LogFile**

  The path and file specification for a log file where the integrations specific logging information is written. This, along with the common-client EDA log file, is required for debugging purposes in case of issues. The default is *%TEMP%***\eda\mentorPadsLibCon.log**.

- **LogLevel**

  The minimum logging level for information to write to the above log file. Options in the decreasing order are:

  - **ERROR**
  - **WARN**
  - **INFO**
  - **DEBUG**

© 2025 Siemens

The default level selected is **INFO**.

- **PadsApplication**

  Defines the PADS application to open for connection. Options are:

  - **PowerLayout**
  - **PowerLogic**

  The default value is **PowerLayout**.

- **PadsVisible**

  Defines the visibility of the PADS application when opened. Options are:

  - **True**
  - **False**

  The default value is **True**.

Setting up Xpedition Library integration

To set up Xpedition Library integration

1. Open a console window and enter the command **eda_gateway**.
2. On the **Configuration** dialog box, select **mentorExpLib**.

To modify the default properties

Various properties to control the execution of the library tool may be set in file *%TEMP%* **\eda\mentorExpLibCon.ini**. This file is created the first time when the integration is run and a configuration is created.

- **LogFile**

  The path and file specification for a log file where the integrations specific logging information is written. This, along with the common-client EDA log file, is required for debugging purposes in case of issues. The default is *%TEMP%***\eda\mentorExpLibCon.log**.

- **LogLevel**

© 2025 Siemens

The minimum logging level for information to write to the above log file. Options in the decreasing order are:

- **ERROR**
- **WARN**
- **INFO**
- **DEBUG**

The default level selected is **INFO**.

- **attrsmap**

  This is used to define the multiple attribute/value format in the **edalib.xml** file while updating the Xpedition part attribute.

  The connector will parse into two pairs, for example, <*attributename*, *attributeValue*> and then use these to update the part information.

  For example, the default is *Value,VALUE;Value2,VALUE2*.

Setting up EDX Library integration

To Install the EDX trusted certificate into the Mentor keystore

The Teamcenter EDA and EDX integration provides a trusted certificate that must be added to the keystore used by your ECAD tool. This is used while synchronizing components back from Teamcenter into the EDX Librarian. The certificate is available in the *%TCEDAECAD_ROOT%\certificate.pem* folder.

To install the certificate into the Mentor keystore, execute the following command from a console:

%KEYTOOL_PATH%\keytool -import -alias TeamcenterEDA20 -file %TCEDAECAD_ROOT%\certificate.pem -keystore %MENTOR_KEYSTORE% -validity 3650

where,

- KEYTOOL_PATH is the location of the keytool program in the ECAD tool installation.

  For example, *C:\MentorGraphics\EEVX.1.2\EEVX.1.2\SDD_HOME\common\win64\jre\default\bin*

- *TCEDAECAD_ROOT* is the eda installation folder. This environment variable is set during installation.

- MENTOR_KEYSTORE is keystore folder used by your ECAD tools.

© 2025 Siemens

For example, *C:\MentorGraphics\EEVX.1.2\EEVX.1.2\SDD_HOME\common\win64\jre\default\lib\sec*

- The **-validity 3650** tells the tool that the key should be valid or 3650 days (10 years). The keytool command will prompt for a password to the Mentor keystore, you must enter it when prompted. You will also be prompted to trust this certificate, answer "y"

To set up EDX Library integration

To use the EDX Library integration,

1. Open a console window and enter the command **eda_gateway**.

2. On the **Configuration** dialog box, select **mentorEDXLib**.

Ensure that you have permission to create a file in the *%TCEDAECAD_ROOT%* folder. A file named **API.properties** must be created there by EDA

© 2025 Siemens

© 2025 Siemens

6. Upgrading to a newer version of Teamcenter EDA

Taskflow to upgrade to a newer version of Teamcenter EDA

You can upgrade the Teamcenter EDA client installations on the ECAD workstations to a new version of the client. For successful upgrade of the EDA client you may be required to perform updates on the Teamcenter server as well. Siemens Digital Industries Software recommends that you first perform the upgrade on a test system to verify the parameter values required for the installation on ECAD workstations.

Taskflow to upgrade to newer version of Teamcenter EDA:

Tasks to be performed on the Teamcenter server	*Tasks to be performed on the EDA client*
Update Teamcenter server to a compatible version	**Perform the pre-upgrade steps**
↓	↓
Configure Teamcenter server for **design management** or **part library management**	**Upgrade the EDA client to a newer version**
	↓
	Delete the cache and staging directories

Upgrade Teamcenter server

Verify whether the current Teamcenter server installation is compatible with the latest version of the EDA client.

If you do not have compatible Teamcenter server version, you must upgrade the Teamcenter server to a version compatible with the new EDA client version. For information on how to perform Teamcenter upgrade, refer to the Teamcenter Upgrade Using TEM documentation.

If you want to install a patch on the existing installation using TEM, refer to the how to install patches on a Teamcenter server topic in Teamcenter Upgrade Using TEM documentation.

If you want to install a patch on the existing installation using Deployment Center, refer to the how to Upgrade or patch your environment topic in Deployment Center documentation.

You must also perform additional configurations on the Teamcenter server for the integration with the EDA client to work properly for **design management** or **part library management**.

© 2025 Siemens

Configure Teamcenter for design management

Overview of configurations required for design management

You must perform the following server side configurations for the integrations to work properly for design management.

Configuration	Why is it required?
Modify the style sheet to include EDA derived data relation in Active Workspace	To allow EDA derived datasets to be displayed as item revision attachments in Active Workspace, the **Awp0ItemRevisionSummary** style sheet must be modified.
Add EDA relations and EDA objects in Active Workspace Relation Browser	For Relation Browser to traverse EDA relations and show EDA objects in Active Workspace, EDA information must be included in the Active Workspace Relation Browser configuration.

Modify the style sheet to include EDA derived data relation in Active Workspace

The *commercial-off-the-shelf (COTS)* value of the EDA preferences used by Active Workspace to display the EDA item revision types is defined in the **Awp0ItemRevSummary** style sheet.

To allow EDA derived datasets to be displayed as ItemRevision attachments in Active Workspace, you must modify the **Awp0ItemRevSummary** style sheet on the Teamcenter server.

1. Log on to Teamcenter and start the rich client.

2. In the navigation pane, click **My Teamcenter**.

   a. Search for the **Awp0ItemRevSummary** dataset.

   b. Double-click the dataset.

      The dataset is automatically checked out and the xml file is launched with the dataset.

3. In the **Awp0ItemRevSummary** style sheet, modify the **XRT_files** page by appending the **EDAHasDerivedDataset** relation to the attachments page objectSet source attribute value as follows. The modified line is marked with a plus (+) sign on the left most column.

```
<page titleKey="attachments">
    <section titleKey="tc_xrt_Files">
         <objectSet
```


© 2025 Siemens

```
source="IMAN_specification.Dataset,IMAN_reference.Dataset,

IMAN_manifestation.Dataset,IMAN_Rendering.Dataset,TC_Attaches.Dataset
,

IMAN_UG_altrep.Dataset,IMAN_UG_scenario.Dataset,IMAN_Simulation.Datas
et,
                    EDAHasDerivedDataset.Dataset"
defaultdisplay="listDisplay"
                    sortby="object_string"
sortdirection="ascending"><tableDisplay>
                          <property name="object_string"/>
                          <property name="object_type"/>
```

Add EDA relations and EDA objects in Active Workspace Relation Browser

To traverse EDA relations and show EDA objects from the **Relations** tab in Active Workspace, EDA information must be included in the Relation Browser configuration. To do this, you must update the **RelationBrowserConf** dataset in the **RB_UIConfigure.xml** file in the Teamcenter server configuration.

To enable the **Relations** tab to show relations between Teamcenter objects and objects in a remote application, do the following:

1. In the Teamcenter rich client, search for the dataset **RelationB***.

2. Right-click the **RelationBrowserConf** dataset and choose **Check-InOut** > **Check-Out**.

3. Right-click **RelationBrowserConf** dataset and choose **Named References**.

4. In the **Named References** dialog box, select **RB_UIConfigure.xml** and click **Download**.

5. Modify the **RelationBrowserConf** dataset in the locally downloaded **RB_UIConfigure.xml** file to add EDA relations and EDA objects as follows.

```
=====================================================================
==
            Filename: RB_UIConfigure.xml

            File used to configure control panel of relation browser.
=====================================================================
==

 -->
       <RelationBrowser>
```


© 2025 Siemens

```
          <OptionPanel>
            <view name="General">
              <ruleName>GenericRule</ruleName>
              <defaultLayout>IncrementalHierarchic</defaultLayout>
              <group name="relations">

                <filter name="EDA" parameterSet="EDARelations"
color="(204,51,0)"/>
              </group>
              <group name="objects">

               <filter name="EDA" parameterSet="EDAObjects"
color="(204,51,0)"/>
               <filter name="Other" parameterSet="WorkspaceObject"
color="(238,236,225)"/>
              </group>
            </view>
            <view name="System">

               <parameterSet name="EDAObjects">
                  <clipsfacts>
                    <fact>EDACCABase</fact>
                    <fact>EDACCABaseRevision</fact>
                    <fact>EDASchem</fact>
                    <fact>EDASchem Revision</fact>
                  </clipsfacts>
               </parameterSet>
                 <parameterSet name="EDARelations">
                   <clipsfacts>
                     <fact>source=GRM,key=EDA,relationType-
=EDAHasDerivedDataset,targetDirection=forward,inputTypes=ItemRevision
</fact>
                     <fact>source=GRM,key=EDA,relationType-
=EDAHasDerivedItem,targetDirection=forward,inputTypes=ItemRevision</
fact>
                     <fact>source=GRM,key=EDA,relationType-
=EDAHasPWB,targetDirection=forward,inputTypes=ItemRevision</fact>
                     <fact>source=GRM,key=EDA,relationType-
=EDAHasSchematic,targetDirection=forward,inputTypes=ItemRevision</
fact>
                     <fact>source=GRM,key=EDA,relationType-
=EDAHasSchematic,targetDirection=backward,inputTypes=ItemRevision</
fact>
                     <fact>source=GRM,key=EDA,relationType-
=EDAHasSublayout,targetDirection=forward,inputTypes=Dataset,outputTyp
es=Dataset</fact>
                     <fact>source=GRM,key=EDA,relationType-
=EDAHasVariant,targetDirection=forward,inputTypes=ItemRevision</fact>
                     </clipsfacts>                      </
```


© 2025 Siemens

```
parameterSet>

            </view>
        </RelationBrowser>
```

6. Delete the original **RB_UIConfigure.xml** file from the **Named References** dialog box and import the new **RB_UIConfigure.xml** file in Teamcenter.

7. Click **Close**.

8. Check in the **RelationBrowserConf** dataset.

Configure Teamcenter for part library management

Overview of configurations required for part library management

ECAD users should be able to **synchronize and reuse ECAD library parts with multiple external sites**. You can create a query in the Query Builder application to support synchronization of ECAD library parts with multiple external sites.

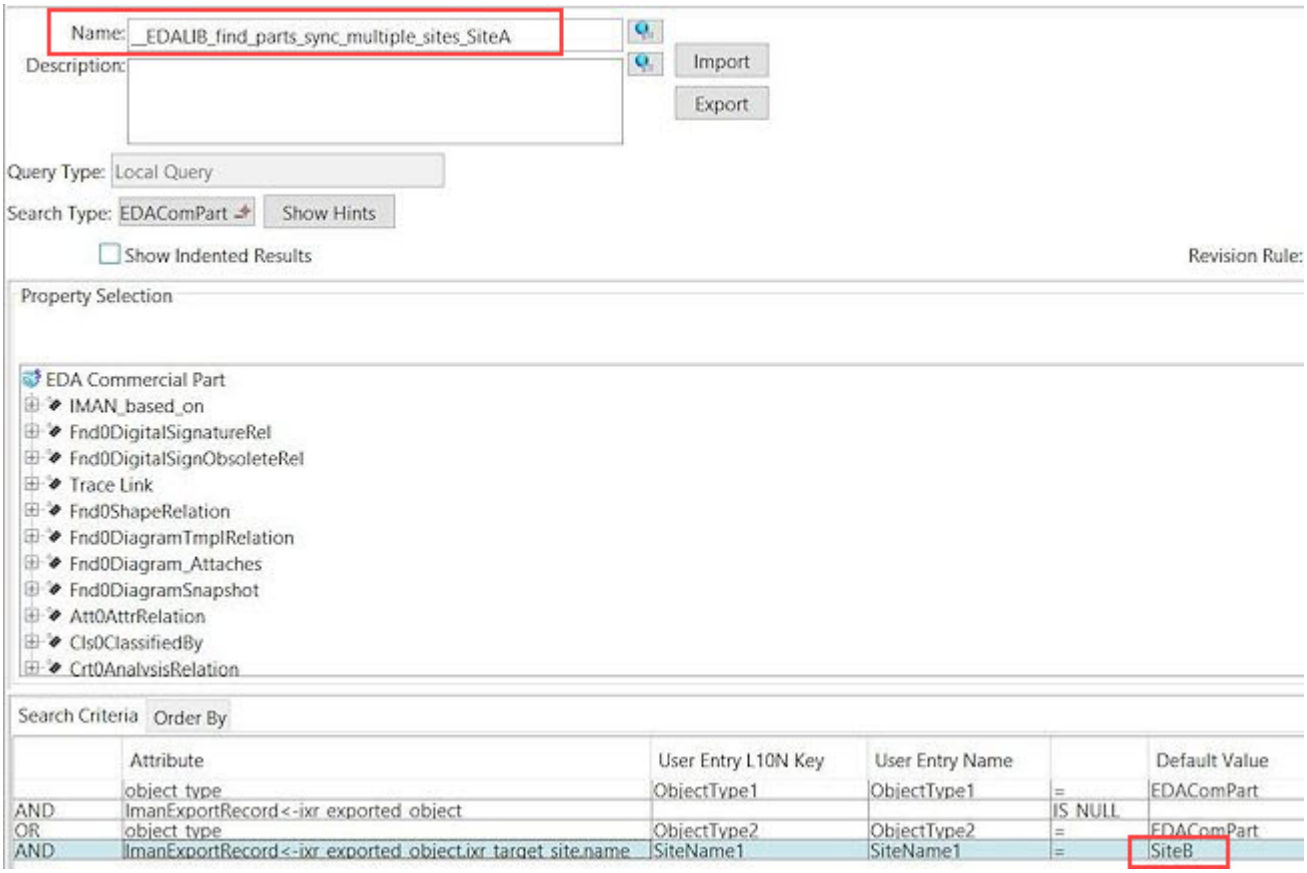
Synchronize ECAD library parts with multiple external sites

To enable ECAD users synchronize and reuse their part data across multiple external sites, you create a query in the Query Builder application to support this synchronization. Consider an example where you want to synchronize a part that is already synchronized with *SiteB* to a different site, *SiteA*.

1. Create a new query **__EDALIB_find_parts_sync_multiple_sites_<site name>** in the Query Builder application using an existing query **__EDALIB_find_parts_sync_multiple_sites** where *<site name>* is the name of the site to which you want to synchronize the parts.

   In our example, this is **__EDALIB_find_parts_sync_multiple_sites_SiteA**, where *SiteA* is the site to which you want to synchronize the parts.

© 2025 Siemens



2. Specify the site name (*SiteB*) in the query to which the part is already synchronized.

   Criteria for creating the query:

   - The query must search for parts that are not yet synchronized to any site (new parts) **AND** the parts that are synchronized to a site (*SiteB*).
   - The above condition to search new parts and existing parts for a site (*SiteB*) is separated by the **OR** function.
   - In each condition, the object type **EDAComPart** is specified with a different **User Entry L10NKey** (*ObjectType1*) and **User Entry Name** (*ObjectType2*) name.

3. Save the query. By default, the type of the query is **Local Query**.

4. To change the above query to a **User Query**, from the Teamcenter command prompt, run the following command:

   %TC_BIN% **\tc_set_query_where_run –u=***Tc-admin-user* **–p=***password* **–g=***group* **-query=***__EDALIB_find_parts_sync_multiple_sites_SiteA*" **–run=***query*

© 2025 Siemens

5. Set the value of the preference **EDALIB_Sync_FindNewParts** with the new query name (**__EDALIB_find_parts_sync_multiple_sites_<site name>**).

   You can also specify this query name in the integration-specific preference. For example, in the Xpedition EDX integration, the value can be set to the preference **EDALIB_Sync_FindNewParts_mentorEDX**.

To modify a query in the Query Builder application, you must first change the query type to **Local Query**. To do this, from the Teamcenter command prompt, run the following command:

%TC_BIN% **\tc_set_query_where_run –u=***Tc-admin-user* **–p=***password* **–g=***group* **-query=***__EDALIB_find_parts_sync_multiple_sites_SiteA*" **–run=***iman*

After making the required changes to the query, run the command specified in the Step 4 to change it to **User Query** again.

Steps to perform before you upgrade the EDA client

Before upgrading the EDA client on your ECAD workstation:

- Download the Teamcenter software kit on your ECAD workstation to perform the upgrade of the EDA client.
- Make note of the current EDA application directory. You may require it after the upgrade.
- During the patch install, the existing contents of the installation are zipped to the *<root>***\install\install\patches\backup\<date>.zip** file. After installation, you must extract the **TCEDAClient.properties** file from this zip folder and copy it back to the *<root>***\eda** to apply the existing configurations after the update.
- Verify that the ECAD designers check in all ECAD designs.

  > Warning:
  >
  > Failure to check in designs may result in loss of data.

- Ensure that the ECAD designers purge working files from their staging locations and back up design files to a location where they can be accessed after the upgrade.
- Delete the cache and staging directories after taking backup.
- Close all the EDA client sessions and confirm that all users are logged off.

Before you upgrade the EDA client, you must collect **Teamcenter server details** required during the upgrade process.

© 2025 Siemens

Collect Teamcenter server details required during the EDA client upgrade

Before you begin the upgrade, you must get the correct product distribution image as well as obtain the following information from your Teamcenter installation administrator:

- The configuration ID.

  The configuration ID identifies your Teamcenter configuration when you maintain, upgrade, uninstall, or add features to the configuration. Installation log files are also named based on this ID.

- The application root directory.

- The FMS client cache (FCC). If one exists, use the existing FCC or use a new one.

- The FCC parent information for the EDA client. The minimum information required is the host name and port.

- The four-tier and middle-tier Web application server URI and connection name for the EDA client.

- The EDA client staging location.

- The logon URL and the application ID in order to enable Teamcenter Security Services.

- The host name, port, and license server mode of the Siemens Digital Industries Software common license server.

Upgrade to a newer version of the EDA client

You can upgrade the EDA client installations on the ECAD workstations to a new version of the client using the Teamcenter software kit. Siemens Digital Industries Software recommends that you first perform the upgrade on a test system to verify the parameter values required for the installation on ECAD workstations.

Once you have completed **the pre-upgrade steps** and have all the **Teamcenter server details** required for the upgrade, perform the following steps to begin the upgrade process:

Teamcenter EDA updates (major or minor releases and patches) are posted for download on Support Center when available:

https://support.sw.siemens.com

For major or minor release upgrade:

1. Extract the Teamcenter software distribution image to a temporary location.

© 2025 Siemens

2. Verify that the **JRE_HOME** environment variable specifies the required Java Runtime Environment (JRE) on the ECAD workstation.

3. Start TEM.

   a. Browse to the root directory of the extracted Teamcenter software distribution image.

   b. Right-click the **tem.bat** program icon and choose **Run as administrator**.

      TEM launches and displays the **Installer Language** panel.

   c. In the **Installer Language** panel, select a language and click **OK**.

      TEM displays the **Welcome to Teamcenter EDA** panel.

      Your language selection applies only to the TEM session and not the Teamcenter EDA installation.

4. In the **Welcome to Teamcenter EDA** panel, click **Upgrade**.

   Click **Upgrade** to upgrade Teamcenter EDA.

5. In the **Old Application Root** panel, verify that you are upgrading the correct Teamcenter EDA installation.

   Click **Next**.

6. In the **Configuration** panel, type the configuration ID and the description for the new Teamcenter EDA configuration.

7. The **New Application Root** panel shows a list of features in your current EDA installation and the status of each feature as follows:

   a. **Upgrade** implies that the feature will be upgraded.

   b. **Deprecated** implies that the feature will not be upgraded.

   In the **New Application Root Directory** box, type the location where you want to upgrade your Teamcenter EDA and click **Next**.

8. In the **File Client Cache (FCC)** panel, select one of the following options:

   a. **Use current FCC**

      Select this option if you want to reuse the existing FCC.

© 2025 Siemens

b. **Use new FCC**

Select this option if you want to use a new FCC.

Click **Next**.

9. In the **FCC Parents** panel, specify the settings for the parent FMS server caches (FSC) that are used by the FMS client cache (FCC) and click **Next**.

   The FSCs are used in the priority that you specify.

10. In the **4-tier Server Configurations** panel, type the address for the SOA service running on the EDA Web server in the **SOA URL** box. For example:

    http://*host***:***port***/tc**

    Replace *host* with the server name and *port* with the server port and click **Next**.

11. In the **EDA Client** panel, verify the staging location and SSO settings.

    The staging location and SSO settings may change during the upgrade. If the staging location changes, you have to perform additional configurations following the upgrade.

    Click **Next**.

12. In the **Flex License Client** panel, specify the license settings.

    Click **Next**.

13. In the **Confirmation** panel, review your selections.

    If you need to make a change, click **Back**.

    When you are ready to upgrade, click **Start**.

    The **Upgrade Features** panel displays the status of the upgrade.

14. If the installation is unsuccessful, review the upgrade log and take corrective action.

    If the installation is successful, and if you have changed the staging location during upgrade, perform the **additional upgrade** steps on each ECAD workstation where the EDA application is upgraded.

© 2025 Siemens

For patch install:

If the patch you downloaded has a corresponding installer patch file (*patch-id*_**install.zip**), download this installer patch and update Teamcenter Environment Manager (TEM) to the latest version using the following steps.

1. Extract the *patch-id*_**install.zip** file to your **install** directory under *TC_ROOT* (the existing EDA installation install folder that contains the **tem.bat** file), to overwrite existing files.

2. Extract the *patch-id*_**win64.zip** file to a temporary folder.

3. Run the **tem.bat** from the existing EDA installation folder using the updated **tem.bat** from step 1.

4. Select **Updates Manager** and click **Next**.

5. Enter the **Update kit location** using the temporary folder location from step 2.

6. Click **Next** and continue through the update process.

Delete the cache and staging directory

Inspect the *USERPROFILE***Teamcenter\EDA** directory of each ECAD user of upgraded ECAD workstation. If this directory exists and contains a **TCEDAClient.properties** file, perform the following actions:

> Note:
>
> If the directory does not exist, or the **TCEDAClient.properties** file does not exist, you can skip the following steps.

- **Delete the cache directory of each ECAD designer on the ECAD workstation.**

  The default location of the cache directory is *%USERPROFILE%***Teamcenter\EDA**<*TC_SERVER_SITE_ID*>.

- **Delete the staging directory of each ECAD designer on the ECAD workstation.**

  To find the location of the staging directory, look for the value of the **StagingDir** property in the *USERPROFILE***Teamcenter\EDA\TCEDAClient.properties** file.

  If the **TCEDAClient.properties** file does not exist, or the property is not specified, or the value of the property is empty, skip this step.

- **Delete the TCEDAClient.properties file of each ECAD designer on the ECAD workstation.**

  The **TCEDAClient.properties** files of the ECAD designers are located in the *USERPROFILE***Teamcenter\EDA** directories.

© 2025 Siemens

© 2025 Siemens

7. Uninstall the EDA client from the ECAD workstation

1. Start TEM.

   You can start TEM in Windows by clicking **Start→Programs→EDA installation name→Environment Manager**.

2. In the **Maintenance** panel, select the **Configuration Manager** option.

   Click **Next**.

3. In the **Configuration Manager** panel, select **Remove Configuration (uninstall)**.

   Click **Next**.

4. In the **Configuration Display** panel, verify the configuration to uninstall.

   Click **Next**.

5. In the **Complete Uninstall** panel, select **Remove all files including user files** option.

   Click **Next**.

6. In the **Warning** panel, select the **I want to permanently remove these items** option.

   Click **OK**.

7. In the **Confirmation** panel, click **Start**.

Teamcenter EDA uninstalls the EDA application.

> Note:
>
> If necessary, manually remove the customization files that were copied into the ECAD tool folders during installation and restore any files that were deleted.

© 2025 Siemens

© 2025 Siemens

8. Deploying Teamcenter EDA documentation on your local drive or network

You can deploy the Teamcenter EDA documentation on your local drive or network. See the **Siemens Help Server documentation** for instructions to deploy the documentation on your local drive or network.

© 2025 Siemens