

The Siemens logo is displayed in a white rectangular box in the top left corner. The background of the entire page is a low-angle photograph of a modern glass skyscraper reaching towards a blue sky with scattered white clouds.

SIEMENS

Teamcenter EDA 5.1

Installing and Configuring EDA Gateway for (Non- Mentor) ECAD Applications

EDA001 - 5.1

Contents

Overview of setting up EDA Gateway for ECAD applications 1-1

Installing and Configuring EDA Gateway for (Non-Mentor) ECAD Applications for design management

Planning your deployment for design management	2-1
Installing Teamcenter server and the EDA client for design management through Deployment Center	2-3
Prerequisites for installing Teamcenter server and the EDA client through Deployment Center	2-3
Install Teamcenter server and the EDA client through Deployment Center for design management	2-5
Installing Teamcenter server for design management through TEM	2-10
Installing Teamcenter server	2-10
Install BMIDE to configure derived data for EDA objects	2-13
Install or update Dispatcher for setting up remote ECAD translation	2-13
Setting up Teamcenter server for design management	2-14
Overview of setting up and configuring Teamcenter server for design management	2-14
Populate the Teamcenter database with ECAD components	2-17
Import new or updated EDA preferences in Teamcenter to support the latest EDA version	2-18
Import new or updated Active Workspace preferences in Teamcenter to support latest EDA version	2-20
Modify the Active Workspace style sheet to include EDA derived data relation	2-21
Add EDA relations and EDA objects in Active Workspace Relation Browser	2-22
Update the EDA SOA client policy file	2-24
Install xml rendering stylesheet for issue management and collaboration functionality support	2-24
Map ECAD attributes with Teamcenter properties	2-24
Configure derived data for ECAD design objects	2-26
Configure design variants	2-33
Manage design files in local storage	2-34
Apply Teamcenter user's permission on the network shared ECAD design folder	2-35
Allow ECAD designers to assign a Teamcenter project from the EDA client	2-36
Set configurations based on the ECAD applications used	2-39
Configure EDA to access Active Workspace for collaboration and issue management	2-39
Allow users to view the PCB board requirements for ECAD objects in Active Workspace	2-40
Customize the acronyms used for ECAD design objects in Teamcenter	2-41
Configure the EDA client to use the Teamcenter EDA panels instead of Active Workspace	2-41
Managing and configuring Teamcenter preferences for design management	2-42

Installing the EDA client for design management on the ECAD workstation	2-43
Install the EDA client for design management	2-43
Verify the EDA client installation	2-45
Setting up the EDA client for design management	2-46
Configurations required to set up the EDA client for design management	2-46
Configuring Teamcenter Gateway for EDA for design management	2-46
Mapping reference designator attributes for your ECAD tools	2-50
Configuring cache and staging directory	2-52
Modify default user properties in the TCEDAClient.properties file	2-54
Set a flatter folder structure for saving ECAD designs	2-55
Modifying the default logging configuration	2-55
Customize the integration by adding additional command calls	2-56
Setting up ECAD translation	2-57
ECAD translation modes	2-57
Enable PcbToFatf translator	2-58
Installing the ECAD viewer	2-59

Installing and Configuring EDA Gateway for (Non-Mentor) ECAD Applications for part library management

Planning your deployment for part library management	3-1
Installing Teamcenter server and the EDA client for part library management through Deployment Center	3-3
Prerequisites for installing Teamcenter server and the EDA client through Deployment Center	3-3
Install Teamcenter server and the EDA client through Deployment Center for part library management	3-4
Installing Teamcenter server for part library management through TEM	3-10
Installing the Teamcenter server for part library management	3-10
Install BMIDE for mapping ECAD library objects with Teamcenter objects	3-11
Setting up Teamcenter server for part library management	3-12
Overview of configuring Teamcenter for ECAD part library Management	3-12
Import new or modified EDA library preferences in Teamcenter to support latest EDA version	3-13
Create external applications for your ECAD tool in Teamcenter	3-14
Create a Teamcenter part library	3-16
Mapping ECAD part attributes to Teamcenter attributes	3-17
Creating classification mapping for ECAD parts with Teamcenter classes	3-22
How to use a custom part type in the EDA library integration	3-25
Synchronize ECAD library parts with multiple external sites	3-34
Managing and configuring Teamcenter preferences for part library management	3-36
Installing the EDA client for part library management	3-36
Install the EDA client for part library management	3-36
Verify the EDA client installation	3-38
Configure the EDA client for part library management	3-39
Steps to configure the EDA client for part library management	3-39
Create customized gateway configurations for managing ECAD part data	3-40

Create an ECAD library configuration	3-44
Map ECAD library objects to Teamcenter objects using ODBC mapping	3-46
Set the working library	3-52
Set up EDA part library synchronization with Teamcenter	3-52

Upgrading to a newer version of Teamcenter EDA

Taskflow to upgrade to a newer version of Teamcenter EDA	4-1
Upgrade Teamcenter server	4-1
Configure Teamcenter for design management	4-2
Overview of configurations required for design management	4-2
Import new or updated EDA preferences in Teamcenter to support the latest EDA version	4-2
Import new or updated Active Workspace preferences in Teamcenter to support latest EDA version	4-4
Modify the Active Workspace style sheet to include EDA derived data relation	4-5
Add EDA relations and EDA objects in Active Workspace Relation Browser	4-6
Update the EDA SOA client policy file	4-8
Install xml rendering stylesheet for issue management and collaboration functionality support	4-8
Configure Teamcenter for part library management	4-9
Overview of configurations required for part library management	4-9
Import new or modified EDA library preferences in Teamcenter to support latest EDA version	4-9
Steps to perform before you upgrade the EDA client	4-10
Collect Teamcenter server details required during the EDA client upgrade	4-11
Upgrade to a newer version of the EDA client	4-11
Delete the cache and staging directory	4-13

Uninstall the EDA client from the ECAD workstation 5-1

Deploying Teamcenter EDA documentation on your local drive or network 6-1

1. Overview of setting up EDA Gateway for ECAD applications

EDA Gateway is required to integrate your ECAD applications with Teamcenter, to allow designers to move ECAD data into and out of Teamcenter. The integration enables ECAD designers to open and save native design files, access approved parts, generate visualization files, share fabrication and assembly data, create a bill of materials (BOM) containing both mechanical and electrical parts, and collaborate with other domains and suppliers.

Teamcenter allows different types of EDA integrations:

- **Setting up EDA Gateway for design management**
Teamcenter EDA design management allows the ECAD designer to automatically log on to Teamcenter and open, save and request parts, and check in and check out design data. The ECAD tool data is stored as its native design archive. The designer can also store derived data extracted from the ECAD tools, such as fabrication and assembly data, as well as Teamcenter-generated ECAD/MCAD interchange, visualization, and BOM files.
- **Setting up EDA Gateway for part library management**
Teamcenter EDA part library management enables ECAD designers and librarians to capture, track, and manage all the parts, symbols, footprints, padstacks, and attributes in the part libraries, as well as the relationships between these objects. To prevent design teams from using unapproved, obsolete or outdated parts and to ensure that accurate and consistent information is available throughout the organization, Teamcenter library data can be synchronized with each individual ECAD tool's local library. During the synchronization process, Teamcenter automatically identifies any new or updated parts that must be exported.

Note:

This help covers steps required to install and configure EDA Gateway required for design and part library integrations of your ECAD applications with Teamcenter. This deliverable does not include steps to integrate the ECAD applications with Teamcenter. For steps on integrating Teamcenter with your ECAD applications you must refer to the respective connector documentation. If you want to integrate Mentor ECAD tools with Teamcenter EDA, refer to the *Integrating Mentor Xpedition and PADS with Teamcenter* documentation.

2. Installing and Configuring EDA Gateway for (Non-Mentor) ECAD Applications for design management

Planning your deployment for design management

For setting up EDA Gateway 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	<p>Teamcenter server is used to store and track the ECAD designs.</p> <p>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.</p> <p>If you have an existing Teamcenter environment, you can update it to support the latest EDA version.</p> <p>Ensure that you download the compatible versions from Support Center before starting your deployment.</p>
EDA server support	<p>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.</p> <p>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.</p>
Common Integration Services	<p>EDA uses the <i>Common Integration Services (CIS)</i> to manage ECAD applications data in Teamcenter.</p>
Business Modeler IDE	<p>Business Modeler IDE is required to configure derived data for EDA objects.</p> <p>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</p>

Server components	Requirements
	install the EDA Server Support template applicable to your project.
Dispatcher	<p>Dispatcher is required for remote ECAD translations to translate and save the design intermediate file into a viewable dataset in Teamcenter.</p> <p>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.</p>

Client components	Requirements
Teamcenter EDA client	<p>The EDA client moves designs between ECAD tools and Teamcenter. The EDA client must be installed on the ECAD workstation.</p> <p>Before installing the EDA client, check the Hardware and Software Certifications knowledge base article on Support Center to identify which versions of the ECAD tools are supported on which versions of Teamcenter EDA.</p> <p>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.</p> <p>Prerequisites and requirements before installing the EDA client:</p> <ul style="list-style-type: none"> • 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. • Information required for EDA client installation from the Teamcenter installation administrator: <ul style="list-style-type: none"> • FMS client cache (FCC) If the FMS client cache (FCC) exists, use the existing one or specify a new one.

Client components	Requirements
	<ul style="list-style-type: none"> • 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 <div data-bbox="565 730 1317 932" style="border: 1px solid black; padding: 10px; margin: 10px 0;"> <p>Note:</p> <p>Both 32-bit and 64-bit installers are available for Windows systems. The available options vary between the two choices.</p> </div> <ul style="list-style-type: none"> • Other installation considerations <ul style="list-style-type: none"> • 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.
(Optional) ECAD viewer	Install the ECAD viewer as a design-view tool and a markup tool.

Installing Teamcenter server and the EDA client for design management through Deployment Center

Prerequisites for installing Teamcenter server and the EDA client through Deployment Center

You can use Deployment Center to perform a new installation of Teamcenter EDA or upgrade from a previous version. Deployment Center is a centralized web application for deploying software to

Teamcenter environments. It simplifies the process of installing and updating software and automates the deployment process.

Before you begin, ensure that you review the following prerequisites to deploying Teamcenter EDA through Deployment Center:

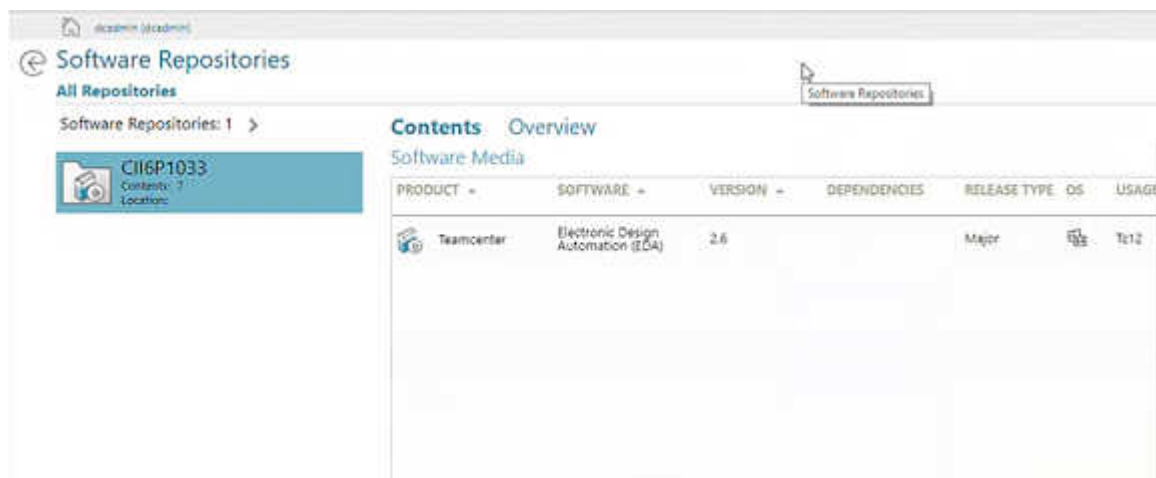
- **Install Deployment Center**

You must install and use the latest available Deployment Center to install Teamcenter EDA. This centralized web application for deploying software to your Teamcenter environments simplifies the process of installing software and automates the deployment.

For instructions on installing and using Deployment Center, see the Deployment Center documentation on Support Center.

- **Put required software kits into the repository**

1. Download the EDA client software kit for the software versions that you want to deploy.
2. Unzip the software kit and copy the unzipped directories to the **software** subdirectory in the repository.
3. Log on to Deployment Center, and click **SOFTWARE REPOSITORIES**.
The **Software Repositories** page opens the **Contents** of the repository and displays the **Software Media** table.




4. Check the list of software to verify that it is correct and complete for your planned deployment. Note whether there are missing dependencies as noted. If so, retrieve the missing software and copy it (unzipped) into the repository and check again.

If you experience a problem in adding software to the Deployment Center repository, you can try to troubleshoot the repository service. See the *Troubleshoot the repository service* topic in the Deployment Center help.

Install Teamcenter server and the EDA client through Deployment Center for design management

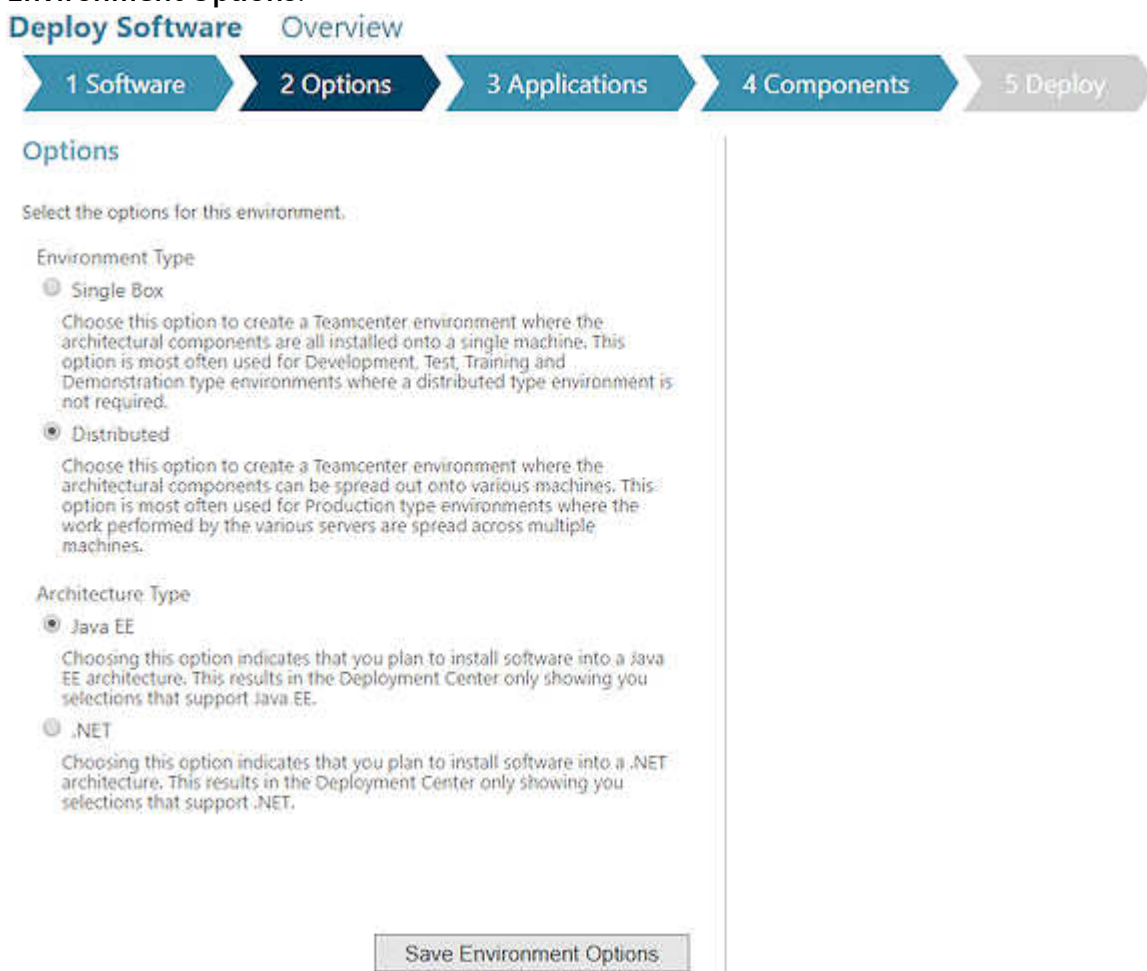
This topic documents how to install Teamcenter EDA using Deployment Center. For detailed information about how to use Deployment Center, see the Deployment Center documentation on Support Center.

Using Deployment Center you can install server and client components in the same distributed environment by providing their machine names. Deployment Center generates separate scripts for each machine.

1. Start the Deployment Center web server.
2. Access the Deployment Center web application from a web browser. The format of the Deployment Center URL is `http://host:serverPort/deploymentcenter`. Here, host is the server where Deployment Center is installed, and serverPort is the port number specified by the **-serverPort** argument in the installation script.
3. Enter the user name and password as specified in the **-user** and **-password** arguments of the Deployment Center installation script.
4. From the Deployment Center home page, open the **Environments** page and choose the environment where you want to install or update software. The **Deploy Software** page provides access to the deployment tasks.
5. In the **Software** task, click  **Edit Selected Software** to add software. The **Available Software** panel displays the **Electronic Design Automation (EDA)** software that you have added to the Deployment Center software repository.
6. Select the following software, and click **Go to Options**:
 - **Teamcenter Electronic Design Automation (EDA)**
 - **Teamcenter Foundation**
 - **Teamcenter MBSE Integration Gateway**



7. In the **Options** panel, select the **Environment Type** and **Architecture Type**, and click **Save Environment Options**.



8. Go to **Applications** panel and click **Edit Selected Applications**.
9. In the **Available Applications** panel, select the following if you are installing Teamcenter EDA for design management:

- **ECAD Design Integrations**


Select the ECAD tool with which you are integrating Teamcenter EDA.

- **Teamcenter Gateway for EDA**

- (Optional) Select **EDA Translation**.

10. Click **Update Selected Applications** to add the applications to the **Selected Applications** list.

11. Click **Go to Components**.

12. In the **Selected Components** panel, click  to add the following components and click **Update Selected Components**. By default, the **Corporate Server**, the **Database Server**, **FSC**, the **Licensing Server**, and the **Teamcenter EDA** options are added to the panel.

- **Server Pool (Java EE)**

- **Teamcenter Web Tier (Java EE)**

For performing derived data configurations, you can add the **Business Modeler IDE 2 tier** or the **Business Modeler IDE 4 tier** component to the **Selected Components** panel.

13. In the **Components** panel, below the **Selected Components** list, click **Start Configuration**. Alternatively, you can click the **Start** link against each component to configure the selected component. Components that are not yet installed display the **Pending Install** status. Components that are installed but require updates to support your selected applications display the **Pending Update** status.

Deploy Software Overview

1 Software 2 Options 3 Applications

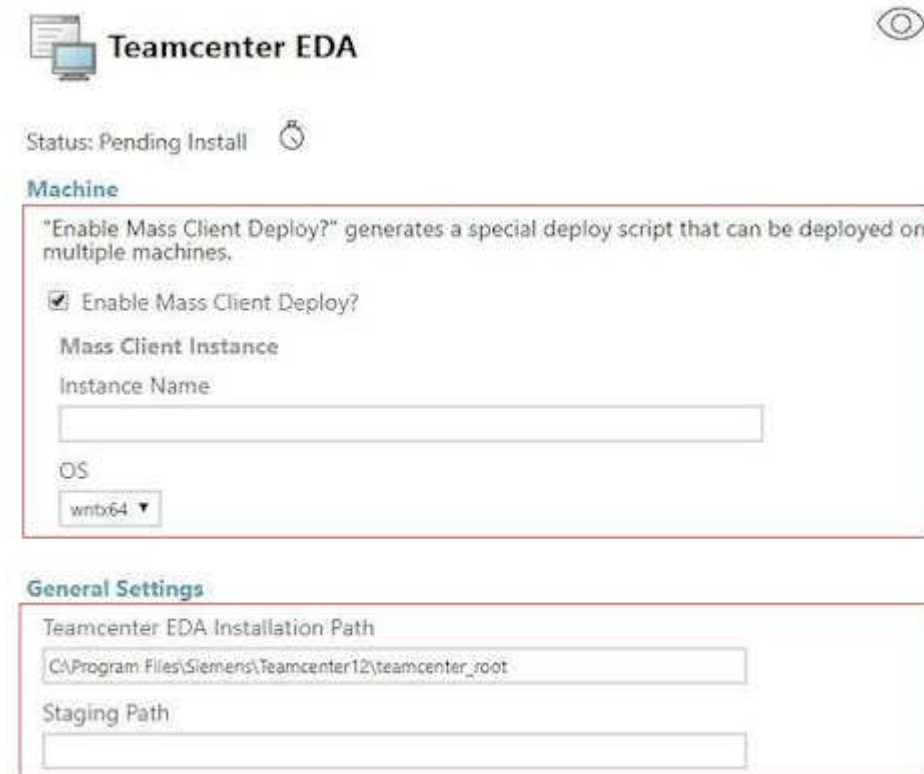
Selected Components 

COMPONENT ▲	MACHINE	OS	COMPLETE	STATUS
 Corporate Server			Start	
 Database Server			Start	
 FSC			Start	
 Licensing Server			Start	
 Server Pool (Java EE)			Start	
 Teamcenter EDA			Start	
 Teamcenter Web Tier (Java EE)			Start	

14. In the right panel, specify the configuration values for each of the selected components. For the **Teamcenter EDA** component, specify the following details:

Parameter	User action
Enable Mass Client Deploy?	<p>Select this option to generate deploy script for installing the Teamcenter EDA client on multiple machines.</p> <p>Leave this option unchecked to generate a deploy script for installation on a single machine.</p>
Instance Name (in case of multiple client installation) OR Machine Name (in case of single client installation)	Specify a name for the zip file created for client installation.
OS	Select the operating system of the Teamcenter EDA client host computer.

Parameter	User action
Teamcenter EDA Installation Path	Specify the path where the EDA client will be installed.
Staging Path	Specify the staging directory location on the client machine.



Teamcenter EDA

Status: Pending Install

Machine

"Enable Mass Client Deploy?" generates a special deploy script that can be deployed on multiple machines.

☒ Enable Mass Client Deploy?

Mass Client Instance

Instance Name

OS

wnb64

General Settings

Teamcenter EDA Installation Path

C:\Program Files\Siemens\Teamcenter12\teamcenter_root

Staging Path

15. Click **Save Component Settings**.
16. After specifying all the required parameters in the **Components** panel, click **Go to Deploy**.
17. In the **Deploy** panel, click **Generate Install Scripts**.
Deployment Center generates installation scripts and submits information about the scripts in the panel on the right.
Deploy Scripts displays the ZIP files that were generated for each server along with the associated component names. Each ZIP file contains the installation scripts for a single server. If all the components were installed on a single machine, there is only one ZIP file. The ZIP file name ends with the target machine name where you run the script.
18. Go to the ZIP file location, copy the Teamcenter EDA deployment script to the host computer where you want to install it, and extract the contents of the file.
19. Open a command prompt window and navigate to the location where you extracted the file contents. Run the **deploy.bat** (Windows) script using the following arguments:

-dcusername	Specify the user name for Deployment Center as defined when installing Deployment Center.
-dcpassword	Specify the password for Deployment Center as defined when installing Deployment Center.
-softwareLocation	Specify the location of the Teamcenter EDA software kit in the -softwareLocation argument, for example: - softwareLocation=D:\deploy_software

When the installation is complete, the command prompt returns the message **Deployment action successfully completed.**

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 Server Installation on Windows 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 Server Installation on Windows 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:
 - Choose **Extensions**→**Mechatronics Process Management** and select **EDA Server Support**.

- Choose **Extensions**→**Mechatronics Process Management** and select **EMPS-Foundation**.
 - 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.
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 and set up Common Integration Services

Teamcenter EDA uses *Common Integration Services (CIS)* to manage ECAD application data in Teamcenter.

The CIS framework uses an *Integration Definition file* to define the default Teamcenter EDA objects that are created during the save-as, save, check-in, and revise operations. These EDA objects are PCA, schematic, PWB, variants, derived items, derived datasets, design dataset, viewable datasets, and BOM components. This file is imported to Teamcenter when CIS services are deployed on the Teamcenter server from the EDA install kit.

The CIS services require an *integration definition file* to be imported to the Teamcenter server. This file is used to define the relation between these EDA objects whether they are related with each other using *Generic Relationship Management (GRM)* relationship or *BOM View Revision BVR* relationship. The relationship behavior is defined OOTB for all the EDA objects with the default Teamcenter object types.

To install and set up common integration services on the Teamcenter server, you must:

1. Update the Teamcenter installer to include the CIS feature.
2. Install Common Integration Services using TEM.

Update the Teamcenter installer to include Common Integration Services

1. Copy the EDA kit to a temporary location.
2. Run Teamcenter Environment Manager (TEM) from the `%TC_ROOT%\install` folder on the Teamcenter server as an administrator.
3. Select **Updates Manager**, click **Next**.
4. For the **Update kit location** field, browse to the `<EDA_KIT>\server_updates\cif\<PLATFORM>` folder. Click **Next**.
5. Proceed through the remaining panels in TEM, entering the required information for the features you selected.
6. When TEM displays the **Confirmation** panel, click **Start** to begin the installation.
7. When the installation is complete, close TEM.

Install the Common Integration Services

1. Run Teamcenter Environment Manager (TEM) from the `%TC_ROOT%\install` folder on the Teamcenter server as an administrator.

2. In the **Maintenance** panel, choose **Configuration Manager**.
3. In the **Configuration Maintenance** panel, choose **Perform maintenance on an existing configuration**.
4. In the **Old Configuration** panel, select the configuration you want to modify.
5. In the **Feature Maintenance** panel, select **Add/Remove Features**.
6. In the **Features** panel, select the following features:
 - **Extensions→Model Management→Server→Common Integration Framework**
7. Proceed through the remaining panels in TEM, entering the required information for the features you select.
8. When TEM displays the **Confirmation** panel, click **Start** to begin the installation.
9. When the installation is complete, close TEM.

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 [Configure your business data model in BMIDE](#) 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 ECAD 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 Installing and Configuring Dispatcher documentation on Support Center. After installing and setting up the dispatcher you must **enable the PcbToFatf translator**.

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.
Import new or updated EDA preferences in Teamcenter to support the latest EDA version	The EDA client requires the latest preferences to be imported in the Teamcenter environment for the client to work correctly.
Import new or updated Active Workspace preferences in Teamcenter to support the latest EDA version	The EDA client requires the latest EDA and Active Workspace preferences to be imported in the Teamcenter environment for the client to work correctly.
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 <i>Awp0ItemRevisionSummary</i> style sheet must be modified.
Add EDA relations and EDA objects in Active Workspace Relation Browser	For Active Workspace Relation Browser to traverse EDA relations and show EDA objects, EDA information must be included in the Active Workspace Relation Browser configuration.
Update the EDA SOA client policy file	You must copy the EDA SOA client policy file from the EDA kit to the Teamcenter server installation folder.

Configuration	Why is it required?
Install xml rendering stylesheet for issue management and collaboration functionality support	For the EDA client Issue Manager and Collaboration functions to work correctly, the <i>IsM1ProblemReportRevisionSummary</i> stylesheet rendering XML file must be installed on the Teamcenter server.
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 <i>variations</i> , 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.
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

Configuration	Why is it required?
	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 for ECAD objects 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 <i>EDACCABase</i> and <i>EDASchem</i> 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 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.

To populate the database with ECAD components:

1. Set the **EDA_CheckComponentExistence** preference:
 - a. Run the rich client on the Teamcenter server and log on as an administrator (for example, **infodba**).
 - 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.

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.

Import new or updated EDA preferences in Teamcenter to support the latest EDA version

The EDA client requires the latest preferences to be imported in the Teamcenter environment for the client to work correctly. Before installing or updating the client, you must import preferences that are specified in an xml file included at the following EDA kit path:

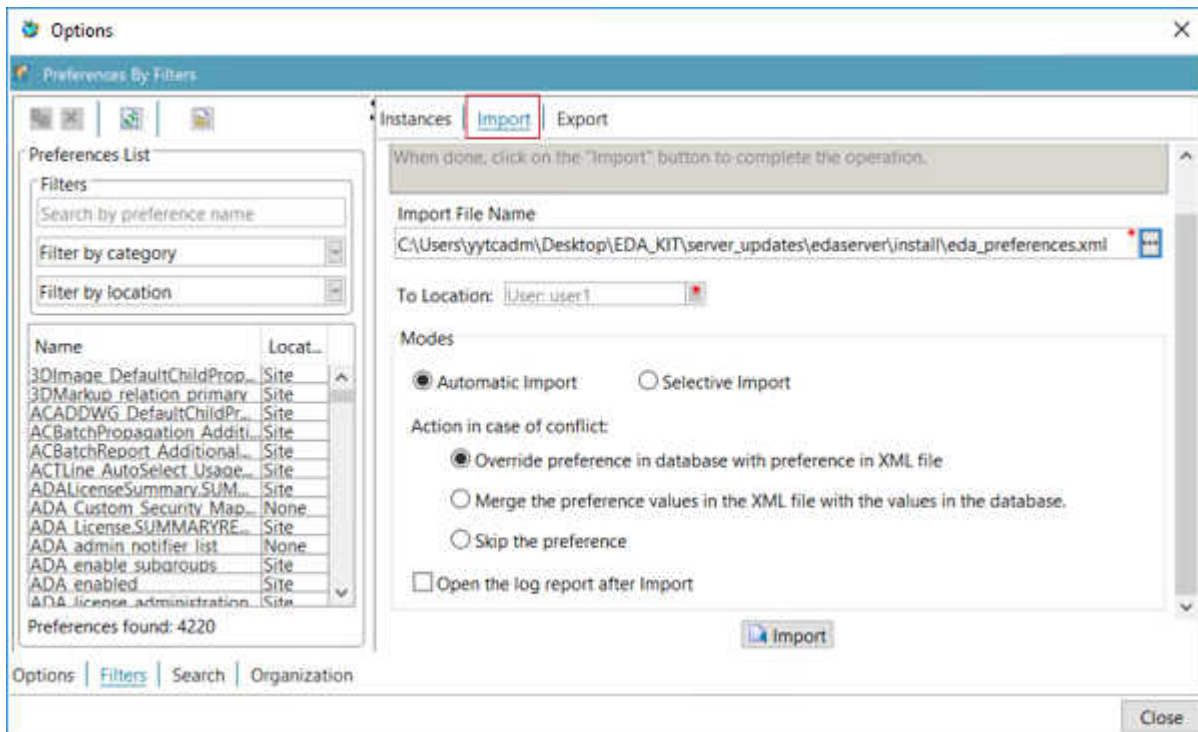
```
<EDA_KIT>\server_updates\edaserver\install\eda_preferences.xml
```

If the preferences are not updated since the Teamcenter base install, this file is not available in the EDA kit.

You can import these preferences in Teamcenter from the rich client or using the Teamcenter command prompt.

Importing preferences in Teamcenter from the rich client:

1. Log on to Teamcenter and start the rich client.
2. Choose **Edit**→**Options** to display the **Options** dialog box.
3. Click **Filters** to open the preferences view.
4. Click **Import**.



5. Browse to the EDA kit location where the preferences file exists and select the *eda_preferences.xml* file for import.
6. Select **Site** from the **To Location** list.
7. Choose **Automatic Import**.
8. If the preference exists in Teamcenter, you can select any one of the **Action in case of conflict** options to override, merge, or skip that preference.
9. Click **Import**.

Importing preferences in Teamcenter from the command prompt:

1. Open a Teamcenter command prompt.
2. Run the following command:
preferences_manager -u=<user id> -p=<password> -g=dba -mode=import -scope=SITE -action=SKIP -file=<EDA_KIT>\server_updates\edaserver\install\eda_preferences.xml

Import new or updated Active Workspace preferences in Teamcenter to support latest EDA version

The EDA client requires the latest EDA and Active Workspace preferences to be imported in the Teamcenter environment for the client to work correctly. Before installing or updating the client, you must import preferences that are specified in an xml file included at the following EDA kit path:

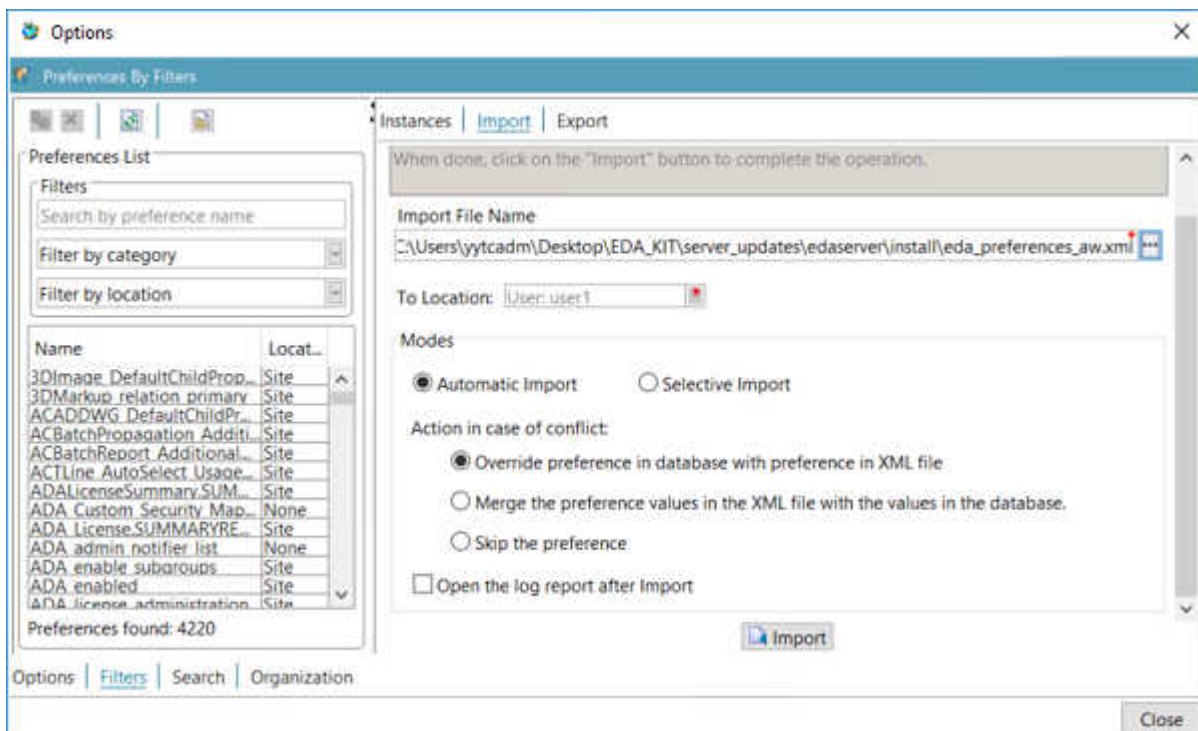
```
<EDA_KIT>\server_updates\edaserver\install\eda_preferences_aw.xml
```

If the Active Workspace preferences are not updated since the Teamcenter base install, this file is not available in the EDA kit.

You can import these preferences in Teamcenter from the rich client or using the Teamcenter command prompt.

Importing preferences in Teamcenter from the rich client:

1. Log on to Teamcenter and start the rich client.
2. Choose **Edit**→**Options** to display the **Options** dialog box.
3. Click **Filters** to open the preferences view.
4. Click **Import**.



5. Browse to the EDA kit location where the preferences file exists and select the *eda_preferences_aw.xml* file for import.
6. Select **Site** from the **To Location** list.
7. Choose **Automatic Import**.
8. If the preference exists in Teamcenter, you can select any one of the **Action in case of conflict** options to override, merge, or skip the preference.
9. Click **Import**.

Importing preferences in Teamcenter from the command prompt:

1. Open a Teamcenter command prompt.
2. Run the following command:
`preferences_manager -u=<user id> -p=<password> -g=dba -mode=import -scope=SITE -action=SKIP -file=<EDA_KIT>\server_updates\edaserver\install\eda_preferences_aw.xml`

Modify the Active Workspace style sheet to include EDA derived data relation

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 *Awp0ItemRevisionSummary* style sheet.

To allow EDA derived datasets to be displayed as ItemRevision attachments in Active Workspace, you must modify the *Awp0ItemRevisionSummary* 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 *Awp0ItemRevisionSummary* dataset.
 - b. Double-click the dataset.
The dataset is automatically checked out and the xml file is launched with the dataset.
3. In the *Awp0ItemRevisionSummary* 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
source="IMAN_specification.Dataset, IMAN_reference.Dataset,
```

```

+.....IMAN_manifestation.Dataset,IMAN_Rendering.Dataset
,TC_Attaches.Dataset,
+.....IMAN_UG_altrep.Dataset,IMAN_UG_scenario.Dataset,I
MAN_Simulation.Dataset,
+.....EDASHasDerivedDataset.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

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

1. Log on to Teamcenter and start the rich client.
2. In the navigation pane, click **My Teamcenter**.
 - a. Search for the *RelationBrowserConf* dataset.
 - b. Double-click the dataset.
The dataset is automatically checked out and the xml file is launched with the dataset.
3. Modify the *RelationBrowserConf* dataset in the *RB_UIConfigure.xml* file to add EDA relations and EDA objects as follows. Modified lines are represented with the plus (+) sign on the extreme left.

```

=====
==
      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>
+.....<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>

```

Update the EDA SOA client policy file

Eclipse provides the framework for interface dialog boxes and provides extension points for communication with the ECAD design tool. All designs are opened and saved in the staging directory whereas the cache holds Teamcenter item data. Communication from client services to Teamcenter is accomplished using SOA services.

You must copy the EDA SOA client policy file from the EDA kit to the Teamcenter server installation folder.

Copy *EDAClientPolicy_1_0.xml* from `<EDA_KIT>\server_updates\tcdata_soa_policies` to `<TC_DATA>\soa\policies`.

This file may not exist in a patch kit if it has not been updated since the base version install.

If a previous version of the EDA SOA client policy file exists, make a backup copy of the file before replacing it with the new file. After copying the new file, you must carry forward all customizations performed in the old policy file into the new file.

Install xml rendering stylesheet for issue management and collaboration functionality support

For the issue management and collaboration functions to work correctly in the EDA client, the xml file rendering the *IsM1ProblemReportRevisionSummary* stylesheet must be installed on the Teamcenter server.

Copy the EDA kit to the Teamcenter server machine and perform the following updates. This file may not exist in a patch kit if it has not been updated since installing the base version.

Open a Teamcenter command prompt and run the following command:

```
install_xml_stylesheet_datasets -u=<user id> -p=<password> -g=dba-input=<EDA_KIT>
\server_updates\edaserver\install\import_eda_stylesheets.txt -filepath=<EDA_KIT>\server_updates
\edaserver\install
```

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 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 *Syntax for mapping attributes with Teamcenter Integration for NX* topic in the Application Administration documentation.

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=infodba —p=infodba
4. Edit the mappings file to add EDA tool attributes.
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=infodba —p=infodba

The attribute mapping import and export process uses the PLM XML import/export mechanism. For more information, see the *PLM XML/TC XML Export Import Administration* documentation.

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 Altium tool:

```
{ Dataset type="EDADesAltiumBrd"
  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"
  }
}
```

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.

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.
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	<p>Type the name that you want to assign to the derived item configuration.</p> <p>This is the name displayed to the user on the Teamcenter EDA Derived Item dialog box during save operations.</p>
Context	<p>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.</p> <ul style="list-style-type: none"> • all • pcb • pcb/simulation • schematic • schematic/pcb • schematic/simulation • simulation
Prefix	<p>(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.</p> <p>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.</p>
Postfix	<p>(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.</p> <p>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.</p>
EDA Parent	<p>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).</p>

Field	Description
	<ul style="list-style-type: none"> • 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	<p>Browse to select the relationship between the derived item and the parent item revision.</p> <p>The EDAHasDerivedItem business object and its children are displayed in the selection dialog box.</p>
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.

- 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.

Field	Description
	This is the name displayed to the user on the Teamcenter EDA Derived Dataset dialog box during save operations.
Context	<p>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.</p> <ul style="list-style-type: none"> • all • pcb • pcb/simulation • schematic • schematic/pcb • schematic/simulation • simulation
Prefix	<p>(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.</p> <p>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.</p>
Postfix	<p>(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.</p> <p>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.</p>
EDA Parent	<p>Browse to select the derived parent EDA business object to which the derived dataset is related.</p> <p>In addition to the following item types, the list also includes item configurations you already created.</p> <p>.</p>



Field	Description
	<ul style="list-style-type: none"> • 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	<p>Browse to select the relationship between the derived dataset and the parent item revision.</p> <p>The EDAHasDerivedDataset business object and its children are displayed in the selection dialog box.</p>
Dataset Business Object	<p>Browse to select the parent dataset business object type in Teamcenter to represent the derived item, for example, PDF.</p>
Dataset Reference	<p>Browse to select the kind of file reference to use for the derived dataset.</p> <p>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.</p>
Pathname	<p>Type the path where the derived dataset is to be saved on the user's machine.</p> <p>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, <i>D:\EDA\Datasets\readme.txt</i>) 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.</p>
Callback Name	<p>Type the EDA callback name to execute.</p> <p>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</p>

Field	Description
	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

- To save the changes to the data model, choose **BMIDE**→**Save Data Model**, or click the **Save Data Model** button  on the main toolbar.
- 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.
- In the rich client, set the **EDA_DerivedDataConfigDefault** preference to point to the EDA derived data configuration you just created.
- 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. There may be multiple configurations created in the Business Modeler IDE, but an administrator can point to only one of them through this preference.
- 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.

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.

- 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:

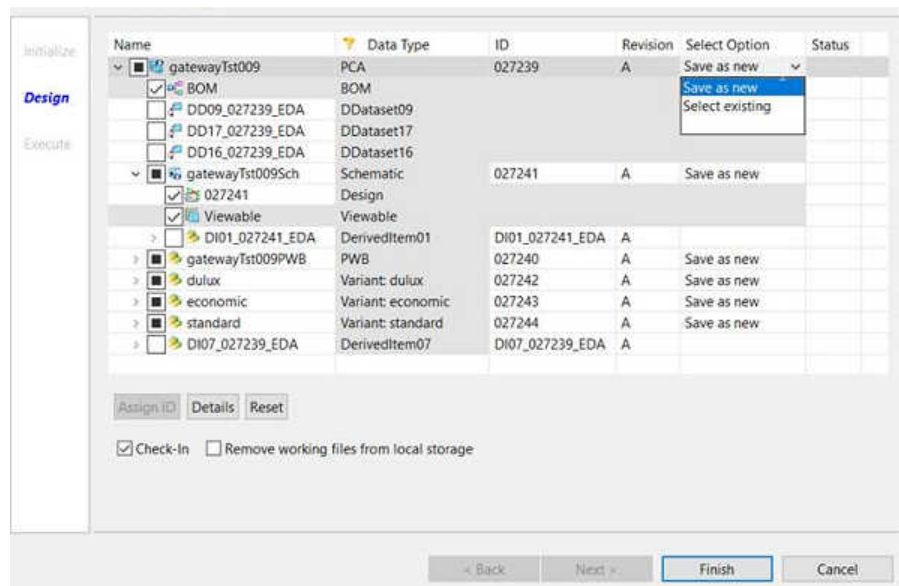
1. Update the value of the **EDA_SaveAsForceVariant** preference to **true**. This value specifies that nonvariant ECAD designs not previously saved in Teamcenter are to be saved in Teamcenter, using the EDA variant object model.
2. Update the value of the **EDA_FutureVariantName** preference. The default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

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:

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

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 EDACCABaseRevision and design Dataset in a variant design.
Forward	EDACCABaseRevision	structure_revisions	PSBOMView Revision	Security Group I	All	Merge	Propagation rule between PCA EDACCABaseRevision and PSBOMView Revision in a variant Design.

Direction	Source business object	Relation	Destination business object	Prop Group	Operation	Style	Details
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 design 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

Direction	Source business object	Relation	Destination business object	Prop Group	Operation	Style	Details
							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 .

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 design Dataset in a non-variant design.
Forward	ItemRevision for the	structure_revisions	PSBOMViewRevision	Security Group I	All	Merge	Propagation rule

Direction	Source business object	Relation	Destination business object	Prop Group	Operation	Style	Details
	PCA item type defined in the EDA_CCAItemTypesDefault preference						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**

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.

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 administrator 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).
- **PCABase** for printed circuit assembly base or **CCABase** for circuit card assembly base (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>
```

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.

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

By default, the EDA client uses 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 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 an 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, 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 to launch Teamcenter EDA Environment Manager (TEM) to perform installation and maintenance operations of Teamcenter EDA on an ECAD workstation for design management.

TEM is available with the Teamcenter EDA 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 EDA 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 **Mechatronics Process Management** and select the required options:
 - (Optional) **EDA Translation**
EDA translation allows converting a PCB or a schematic design into the Teamcenter Visualization ECAD format.
 - **ECAD Design Management**
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.
 - **Teamcenter Gateway for EDA Design**
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**.
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**.

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.
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 cache and staging directories are created from the settings in the **TCEDAClient.properties** file.

The paths of the cache and staging directories contain double back slashes (\\) on Windows systems. For example:

```
C:\\ecad\\EdaCache
```

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

Setting up the EDA client for design management

Configurations required to set up the EDA client for design management

Perform the following configurations on the EDA client for design management:

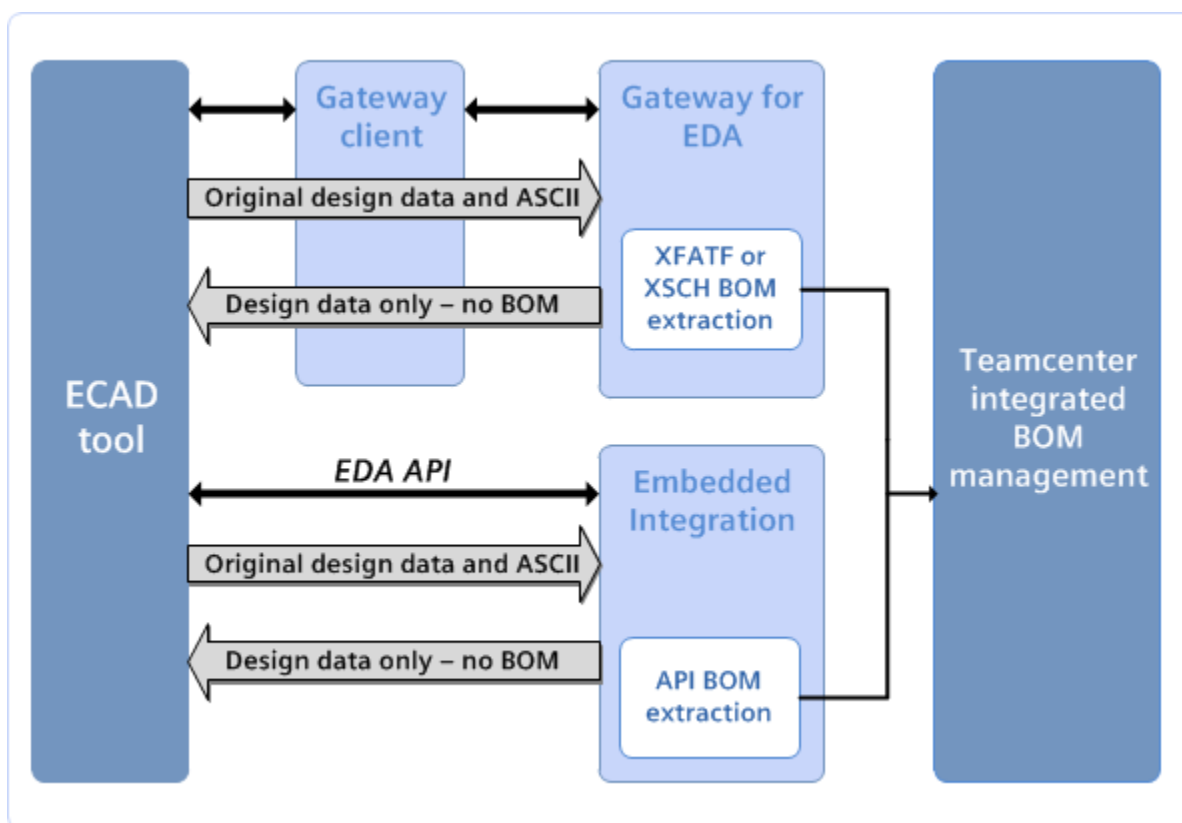
- **Mapping reference designator attributes for your ECAD tools**
- **Configuring cache and staging directory**
- **Modifying default user properties in the TCEDAClient.properties file**
- **Remove multiple levels of folders created while saving ECAD designs**
- **Modifying the default logging configuration**

Configuring Teamcenter Gateway for EDA for design management

About Teamcenter Gateway for EDA

Teamcenter Gateway for EDA is used to integrate ECAD design tools that do not support a customizable user interface. Teamcenter Gateway for EDA provides the same Teamcenter menu as that used in the embedded integrations. You can perform regular Teamcenter functions, such as opening, checking out, and saving designs, including schematic designs with or without variants. The application can extract BOM information from a PCB **xfatf** or Schematic **xsch** file.

The following graphic illustrates Teamcenter Gateway for EDA and the ECAD integration as compared to the embedded menu integration.



About configuration files and templates used by Teamcenter Gateway

Configuration files define the various characteristics required by Teamcenter Gateway for EDA for EDA library integration. These configurations are only included as templates for customization.

When you start Teamcenter Gateway for EDA, you can either specify a configuration depending on your requirement or skip specifying the configuration. If you do not specify the configuration to be used, a dialog box to select the configuration is displayed.

You can switch to an alternate configuration after you start Teamcenter Gateway for EDA by choosing **File→Configuration** and selecting another configuration from the displayed list.

Configuration templates are located in the `TCEDAECAD_ROOT\eda\example\gateway` directory and can be used to create new customized gateway configurations.

The following are the out-of-the-box templates provided:

- **gatewayCombined_edadef.xml**
This configuration allows ECAD designers to save all design data to Teamcenter as one dataset under a CCA item/item revision. The dataset usually contains all related design data such as schematic, simulation, and layout design files, but may contain only one type of the design files.
- **gatewaySchematic_edadef.xml**

gatewayPcb_edadef.xml**gatewaySimulation_edadef.xml**

These configurations are generally used together to allow ECAD designers to save schematic, layout designs, and simulation data in separate datasets under the same CCA item/item revision. The layout and simulation datasets are saved directly under the CCA item/item revision. The schematic design dataset is saved under a schematic item/item revision. The schematic item is a child under the BOM view of the CCA item. All datasets saved under the same CCA item/item revision are called related datasets in a family. These configurations can also be used independently to save different types of design datasets in separate CCA item/item revisions.

The templates provide defaults and contain place holders for you to configure. The following configurable elements and attributes are included in the templates:

- Application name
- Family name for the toolset
- Dataset types
- Translators
- Callbacks

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.

Creating a configuration file for design integrations

The Teamcenter Gateway for EDA configuration file is a readable XML file and can be created and edited using any ASCII text editor. 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 *TCEDAECAD_ROOT/gateway* folder.

For a basic configuration, the minimum required configuration is as follows:

- **Family**
This is the subdirectory name under the top-level EDA staging directory.
- **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.
- **PrimaryDataType** and **datasetType**
PrimaryDataType specifies the design type. Valid types include **schematic**, **pcb**, and **simulation**. **datasetType** represents the Teamcenter dataset type to be created when saving designs using this configuration.
- **RelatedDataType**
The data type that is associated with the primary data type for the concurrent design process.

Adding a callback configuration for Teamcenter Gateway for EDA

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 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.

Callback type	Argument 1	Argument 2	Argument 3
extractCADFromPCB	designFolder	cadFolder	
extractCADFromSCH			
extractBOMFromPCB	designFolder	edaDesignFile	
extractBOMFromSCH			
openDesign	designFolder		
reOpenDesign	designFolder		
setDesignReadOnly	designFolder		
setDesignCheckedOut	designFolder		
closeECAD			
prepareVariantInfo	designFolder		
generateDesignBOMs	designFolder	true or false	Name list of variants that

Callback type	Argument 1	Argument 2	Argument 3
			require BOM to be generated.

The arguments are defined as follows:

- **designFolder**
Specifies the path (directory/folder) containing the design.
- **edaDesignFile**
Specifies the **edaDesign** XML file to be returned to the gateway. This file must be created by the callback to contain any data to be returned. It must follow the **EDADesignSchema.xsd** schema.
- **cadFolder**
Specifies the folder where intermediate CAD files are placed.

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 (**edadef.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 files in your installation and modify it to enable RDN attribute mapping for your ECAD authoring tools.

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 XML file contains additional subelements to enable and configure RDN attribute mapping. You must add this set of elements to your **edadef.xml* file in the required order.

The required 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 PSOccurrence 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.
- 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 `*edadef.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 is created under `USER_HOME` and its value written to the user property file.

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 is 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.

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 data 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.

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 is 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 `USER_HOME` and its value is written to this property file.

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.

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 paths of the cache and staging directories must contain double backslashes on Windows systems, for example:

```
C:\\ecad\\EdaCache\\
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.

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.

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

Syntax

edaccli -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.

Command	Description
	<div style="border: 1px solid black; padding: 10px;"> <p>Note:</p> <p>The logout command only releases the Teamcenter license. To release ECAD tool license, use the ECAD tools.</p> </div>
purgeCache	<p>Displays the Purge Working Files dialog box.</p> <p>The Purge Working Files command allows you to clear the cache of the designs that are not checked out.</p>

-application

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

-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.

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:

- 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 Installing and Configuring Dispatcher help.

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_RootModule\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 dcproxy 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=dcproxy -p=dcproxy
-changeOwnerToCad=false "-pim=yes" %*
```

If you set up **dcproxy** 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 [Installing and Configuring Dispatcher 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.

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 Server Installation on Windows help](#).

For more information about using the ECAD viewer, see [Using the ECAD Viewer documentation on Support Center](#).

3. Installing and Configuring EDA Gateway for (Non-Mentor) ECAD Applications for part library management

Planning your deployment for part library management

For installing and setting up EDA Gateway 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	<p>Teamcenter server is used to store and track the ECAD designs.</p> <p>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.</p> <p>If you have an existing Teamcenter environment, you can update it to support the latest EDA version.</p> <p>Ensure that you download the compatible versions from Support Center before starting your deployment.</p>
EDA server support and library support	<p>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.</p> <p>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.</p>
BMIDE	<p>You require BMIDE to map ECAD objects with Teamcenter objects.</p>
Teamcenter Classification Admin	<p>You require Teamcenter Classification Admin to create custom classification schema and set up classification mappings for part library management.</p>

Client components	Requirements
EDA client	<p>EDA client moves designs between ECAD design tools and Teamcenter. The EDA client must be installed on the ECAD workstation.</p> <p>Before installing the EDA client, see the Hardware and Software Certifications knowledge base article on Support Center to identify which versions of the ECAD tool are supported on which versions of Teamcenter EDA.</p> <p>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.</p> <p>Prerequisites and requirements before installing the EDA client:</p> <ul style="list-style-type: none"> Supported Operating System: <p>The EDA client framework is supported only on the Windows operating system. Both 32-bit and 64-bit installers are available for Windows systems. The available options vary between the two choices.</p> <p>For the ECAD library mappings using the ODBC connector, the EDA client operating system bit version must match with the ODBC database operating system bit version.</p> Supported Java version: <p>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.</p> Information required for EDA client installation from Teamcenter installation administrator: <ul style="list-style-type: none"> 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 URI, and the connection name for the EDA client

Client components	Requirements
	<ul style="list-style-type: none"> 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 <ul style="list-style-type: none"> 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.
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.

Installing Teamcenter server and the EDA client for part library management through Deployment Center

Prerequisites for installing Teamcenter server and the EDA client through Deployment Center

You can use Deployment Center to perform a new installation of Teamcenter EDA or upgrade from a previous version. Deployment Center is a centralized web application for deploying software to Teamcenter environments. It simplifies the process of installing and updating software and automates the deployment process.

Before you begin, ensure that you review the following prerequisites to deploying Teamcenter EDA through Deployment Center:

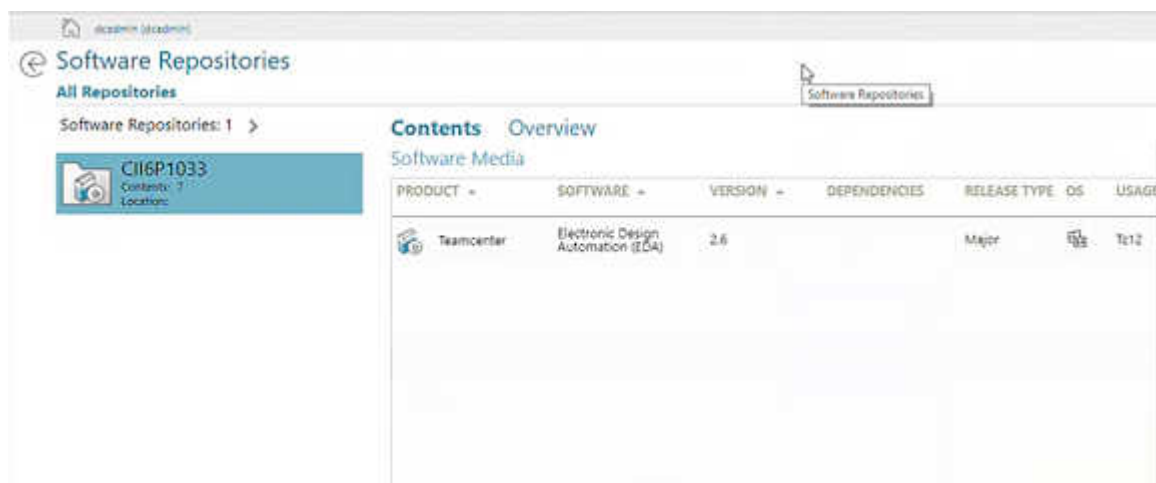
- Install Deployment Center**

You must install and use the latest available Deployment Center to install Teamcenter EDA. This centralized web application for deploying software to your Teamcenter environments simplifies the process of installing software and automates the deployment.

For instructions on installing and using Deployment Center, see the Deployment Center documentation on Support Center.

- **Put required software kits into the repository**

1. Download the EDA client software kit for the software versions that you want to deploy.
2. Unzip the software kit and copy the unzipped directories to the **software** subdirectory in the repository.
3. Log on to Deployment Center, and click **SOFTWARE REPOSITORIES**.
The **Software Repositories** page opens the **Contents** of the repository and displays the **Software Media** table.




4. Check the list of software to verify that it is correct and complete for your planned deployment. Note whether there are missing dependencies as noted. If so, retrieve the missing software and copy it (unzipped) into the repository and check again.

If you experience a problem in adding software to the Deployment Center repository, you can try to troubleshoot the repository service. See the *Troubleshoot the repository service* topic in the Deployment Center help.

Install Teamcenter server and the EDA client through Deployment Center for part library management

This topic documents how to install Teamcenter EDA using Deployment Center. For detailed information about how to use Deployment Center, see the Deployment Center documentation on Support Center.

Using Deployment Center you can install server and client components in the same distributed environment by providing their machine names. Deployment Center generates separate scripts for each machine.

1. Start the Deployment Center web server.
2. Access the Deployment Center web application from a web browser. The format of the Deployment Center URL is `http://host:serverPort/deploymentcenter`. Here, host is the server where Deployment Center is installed, and serverPort is the port number specified by the **-serverPort** argument in the installation script.
3. Enter the user name and password as specified in the **-user** and **-password** arguments of the Deployment Center installation script.
4. From the Deployment Center home page, open the **Environments** page and choose the environment where you want to install or update software. The **Deploy Software** page provides access to the deployment tasks.
5. In the **Software** task, click  **Edit Selected Software** to add software. The **Available Software** panel displays the **Electronic Design Automation (EDA)** software that you have added to the Deployment Center software repository.
6. Select the following software, and click **Go to Options**:
 - Teamcenter Electronic Design Automation (EDA)
 - Teamcenter Foundation
 - Teamcenter MBSE Integration Gateway



7. In the **Options** panel, select the **Environment Type** and **Architecture Type**, and click **Save Environment Options**.

Deploy Software Overview

1 Software 2 Options 3 Applications 4 Components 5 Deploy

Options

Select the options for this environment.

Environment Type

☐ Single Box
Choose this option to create a Teamcenter environment where the architectural components are all installed onto a single machine. This option is most often used for Development, Test, Training and Demonstration type environments where a distributed type environment is not required.



☒ Distributed
Choose this option to create a Teamcenter environment where the architectural components can be spread out onto various machines. This option is most often used for Production type environments where the work performed by the various servers are spread across multiple machines.

Architecture Type

☒ Java EE
Choosing this option indicates that you plan to install software into a Java EE architecture. This results in the Deployment Center only showing you selections that support Java EE.

☐ .NET
Choosing this option indicates that you plan to install software into a .NET architecture. This results in the Deployment Center only showing you selections that support .NET.

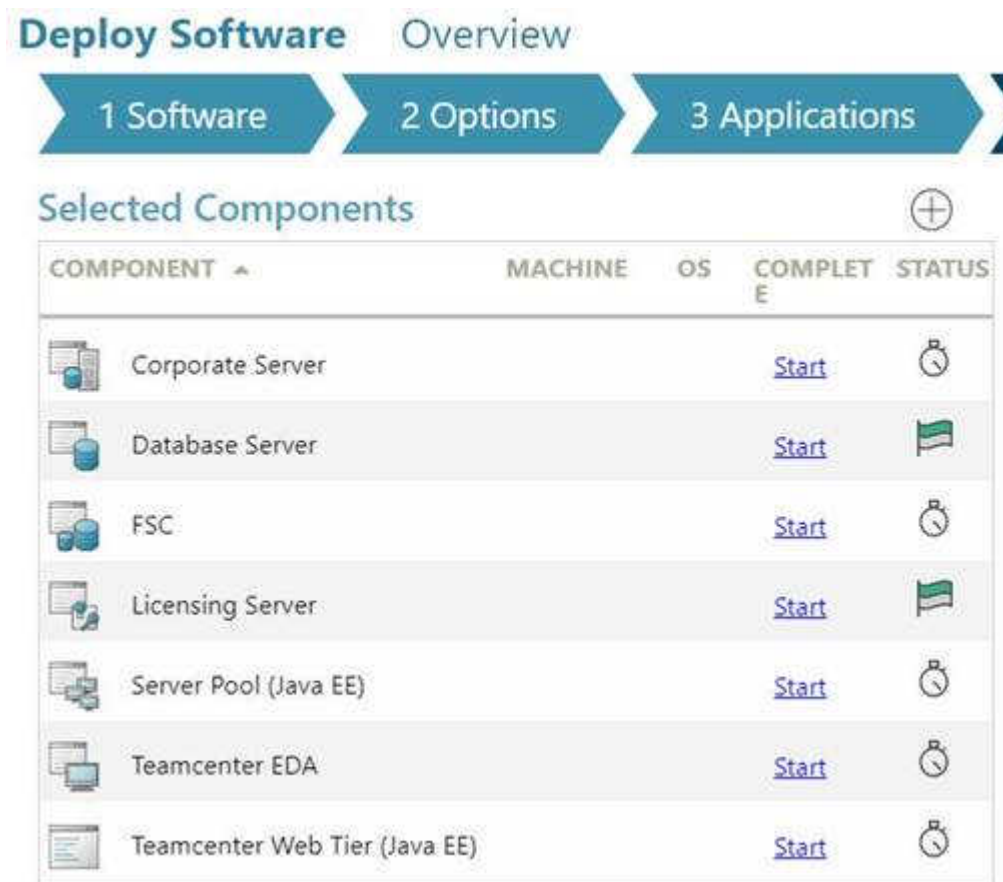
Save Environment Options

8. Go to **Applications** panel and click  **Edit Selected Applications**.
9. In the **Available Applications** panel, select the following if you are installing Teamcenter EDA for design management:
 - **ECAD Library Integrations**
Select the ECAD tool with which you are integrating Teamcenter EDA.
 - **Teamcenter Gateway for EDA Library**
10. Click **Update Selected Applications** to add the applications to the **Selected Applications** list.
11. Click **Go to Components**.
12. In the **Selected Components** panel, click  to add the following components and click **Update Selected Components**. By default, the **Corporate Server**, the **Database Server**, **FSC**, the **Licensing Server**, and the **Teamcenter EDA** options are added to the panel.

- **Server Pool (Java EE)**
- **Teamcenter Web Tier (Java EE)**

For performing derived data configurations, you can add the **Business Modeler IDE 2 tier** or the **Business Modeler IDE 4 tier** component to the **Selected Components** panel.

- In the **Components** panel, below the **Selected Components** list, click **Start Configuration**. Alternatively, you can click the **Start** link against each component to configure the selected component. Components that are not yet installed display the **Pending Install** status. Components that are installed but require updates to support your selected applications display the **Pending Update** status.



- In the right panel, specify the configuration values for each of the selected components. For the **Teamcenter EDA** component, specify the following details:

Parameter	User action
Enable Mass Client Deploy?	Select this option to generate deploy script for installing the Teamcenter EDA client on multiple machines.

Parameter	User action
	Leave this option unchecked to generate a deploy script for installation on a single machine.
Instance Name (in case of multiple client installation) OR Machine Name (in case of single client installation)	Specify a name for the zip file created for client installation.
OS	Select the operating system of the Teamcenter EDA client host computer.
Teamcenter EDA Installation Path	Specify the path where the EDA client will be installed.
Staging Path	Specify the staging directory location on the client machine.



Teamcenter EDA



Status: Pending Install

**Machine**

"Enable Mass Client Deploy?" generates a special deploy script that can be deployed on multiple machines.

☒ Enable Mass Client Deploy?

Mass Client Instance

Instance Name

OS

wnb64 ▼

General Settings

Teamcenter EDA Installation Path

Staging Path

15. Click **Save Component Settings**.
16. After specifying all the required parameters in the **Components** panel, click **Go to Deploy**.
17. In the **Deploy** panel, click **Generate Install Scripts**.

Deployment Center generates installation scripts and submits information about the scripts in the panel on the right.

Deploy Scripts displays the ZIP files that were generated for each server along with the associated component names. Each ZIP file contains the installation scripts for a single server. If all the components were installed on a single machine, there is only one ZIP file. The ZIP file name ends with the target machine name where you run the script.

18. Go to the ZIP file location, copy the Teamcenter EDA deployment script to the host computer where you want to install it, and extract the contents of the file.
19. Open a command prompt window and navigate to the location where you extracted the file contents. Run the **deploy.bat** (Windows) script using the following arguments:

-dcusername	Specify the user name for Deployment Center as defined when installing Deployment Center.
-dcpassword	Specify the password for Deployment Center as defined when installing Deployment Center.
-softwareLocation	Specify the location of the Teamcenter EDA software kit in the -softwareLocation argument, for example: - softwareLocation=D:\deploy_software

When the installation is complete, the command prompt returns the message **Deployment action successfully completed**.

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 part library integration.

- **To integrate ECAD tools with Teamcenter EDA for part library management:**
 - **Set up Teamcenter server for part library management**
 - **Configure the EDA client for part library management**

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 Server Installation on Windows 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 Server Installation on Windows 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**.
 - c. 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.

- d. 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.
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 [Configure your business data model in BMIDE](#) documentation on Support Center.

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 custom classification schema, set up the mappings, map the ECAD part attributes and component instance attributes, and configure the import 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:

- **Import new or modified EDA library preferences** in Teamcenter to support latest EDA version. This is required for the integration to work properly for part library management.
- **Create external applications for your ECAD tool in Teamcenter.**
External applications are part libraries that are based on site objects. To create EDA library gateway application configuration file on the EDA client, you must first setup external applications on your Teamcenter server. You can use 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. 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 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**
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.**

Import new or modified EDA library preferences in Teamcenter to support latest EDA version

Before installing the EDA client, you must import new or updated preferences from the EDA kit in the Teamcenter environment. You must import preferences that are specified in an xml file included in the EDA kit path specified below for the EDA client to work properly. If the preferences are not updated or added since the Teamcenter base install, this file will not be available in the EDA kit.

- `<EDA_KIT>\server_updates\edaserver\install\edalib_preferences.xml`
- `<EDA_KIT>\server_updates\edaserver\install\edalib_preferences_merge.xml`
- `<EDA_KIT>\server_updates\edaserver\install\edalib_preferences_override.xml`

You can import these preferences in Teamcenter from the rich client or using the Teamcenter command prompt.

Importing preferences in Teamcenter from the rich client:

1. Log on to Teamcenter and start the rich client.
2. Choose **Edit**→**Options** to display the **Options** dialog box.
3. Click **Filters** to open the preferences view.

4. Click **Import**.
5. Browse to the EDA kit location where the preferences file exists and select the *edalib_preferences.xml* file for import. Repeat the procedures to import preferences from the *edalib_preferences_merge.xml* and *edalib_preferences_override.xml* files.
6. Select **Site** from the **To Location** list.
7. Choose **Automatic Import**.
8. If the preference exists in Teamcenter, you can select any one of the **Action in case of conflict** options to override, merge, or skip that preference.
9. Click **Import**.

Importing preferences in Teamcenter from the command prompt:

1. Open a Teamcenter command prompt.
2. Run the following command:
preferences_manager -u=<user id> -p=<password> -g=dba -mode=import -scope=SITE -action=SKIP -file=<EDA_KIT>\server_updates\edaserver\install\edalib_preferences.xml

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 setup 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.

Library Integration	Application Name	Application Type (External Applications)	Family Name	Primary Dataset type
Cadence Allegro	cadenceLib	Cadence Allegro	cadence	EDAComAllegro
Altium Designer	altiumDes	Altium Designer	altium	EDAComAltiumDes
Orcad	orcadLib	Orcad	orcad	EDAComOrcad
Cadence ADW	cadenceADWLib	Cadence ADW	cadenceADW	None

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.
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.

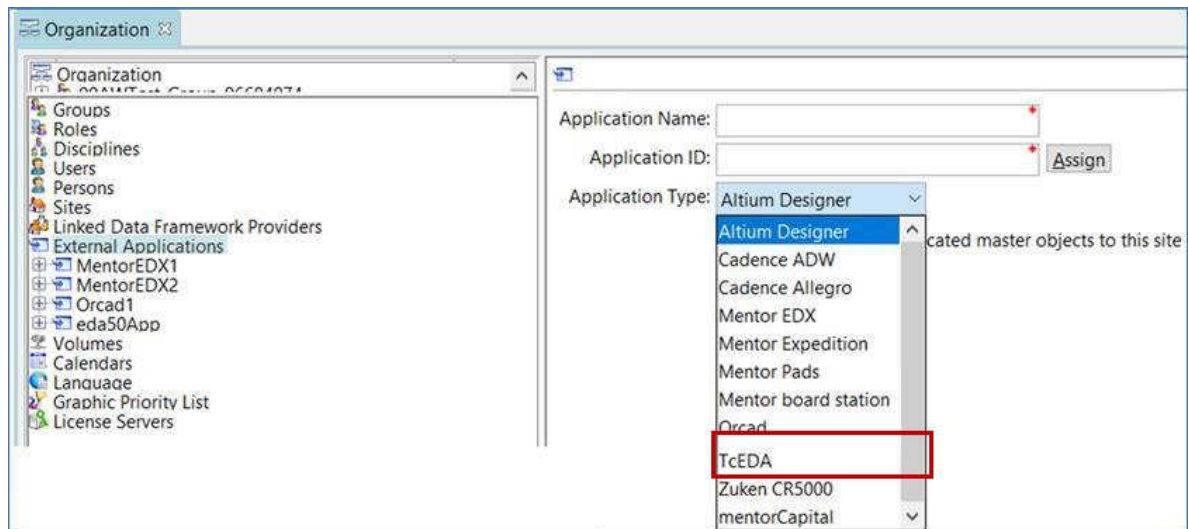
- **Use the OOTB application type from the available options:**

- Cadence Allegro
- Altium Designer
- Orcad
- Cadence ADW

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.



- 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.
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.

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 Classification object. 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 created by default. You cannot override or remove the OOTB mappings, but you can create additional, custom mappings.

You can customize the mappings in Teamcenter by exporting the attribute or object mappings to a file, editing the file, and importing the file back into Teamcenter. The mapping file contains the mapping of attributes from ECAD tool to Teamcenter. It is a text file used as the source for importing the mappings into Teamcenter. Its location (file name and path) is specified when you run the **export_attr_mappings** utility to import the mappings. 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:

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 Classification Admin 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 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 location where Teamcenter is installed.

3. Run the **export_attr_mappings** utility to output the mapping to a file, for example:

```
export_attr_mappings -file=test_attribute_mapping -u=infodba
-p=infodba
```

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

- **attr_from_ECADtool** is the name of the part attribute from the ECAD library tool.
- **attr_from_Tc** is the attribute from Teamcenter.
- **description** is the information used to describe this attribute.

5. Import the edited mapping file back into Teamcenter by running the **import_attr_mappings** utility. For example:

```
import_attr_mappings -file=test_attribute_mapping -u=infodba
-p=infodba
```

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.**

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 Configure your business data model in BMIDE.

2. **Update the object mapping preference.**

Update the **EDALIB_ItemTypesMapping** preference to map the ECAD object to the Teamcenter object.

The **EDALIB_ItemTypesMapping** preference specifies the item types mapping between ECAD system and Teamcenter. Each string value should be in the format **<ECAD_Object>,<Teamcenter_ItemType>,<Is_PrimaryObject>** where:

- * **<ECAD_Object>** is the object name in ECAD tool.
- * **<Teamcenter_ItemType>** is the item or sub type of item in Teamcenter.

- * <Is_PrimaryObject> is the flag if the <ECAD_Object> should be used as primary object to find the candidate of synchronization in synchronize operation in the Teamcenter EDA (Electronic Design Automatic) common client, the provided value is to be "true". Otherwise, the provided value is to be "false". If no value is provided, the default applied value is "false".

For example:

```
EDALIB_ItemTypesMapping=Block,ReusableBlock,false
```

To use this mapping in more than one integration, update the item mapping preference for each integration.

3. 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 Classification Admin help on Support Center.

4. Configure the manage rule.

Update the **EDALib_ManagedObjects** preference with information about what relations should load with the object.

The **EDALib_ManagedObjects** preference specifies how the EDA common client handles objects with Teamcenter. Each string value should be in the format <ECAD_Object>;<Teamcenter Relation>;<Secondary ECAD_Objects>;<Mode>, where:

- * <ECAD_Object> is the object type name in ECAD tool.
- * <Teamcenter Relation> is the Teamcenter relation real name between <ECAD_Object> and <Secondary ECAD_Objects>.
- * <Secondary ECAD_Objects> is the object type name in ECAD tool that needs to be handled from <ECAD_Object>; use comma to separate multiple values.
- * <Mode> only support **Write** or **Read**. If **Read** is specified, the EDA common client only reads <Secondary ECAD_Objects> from Teamcenter but does not update them during the **Save** and **Check-In** operations. If no value is provided, the default value is **Read**.

For example:

```
EDALib_ManagedObjects=Component;view;Symbol, Footprint;Write  
Symbol;view;Symbol;Write  
Footprint;view;Footprint;Write  
Footprint;EDAPadstackRelation;Padstack;Write
```

To support more than one integration, update the preference for each integration.

5. Define the synchronization rule.

For synchronizing the Teamcenter and ECAD object, update the **EDALIB_ItemTypesMapping** preference with the name of the ECAD object.

The **EDALIB_ItemTypesMapping** preference specifies the item types mapping between the ECAD tool and Teamcenter. Each string value should be in the format `<ECAD_Object>,<Teamcenter_ItemType>,<Is_PrimaryObject>` where:

- * `<ECAD_Object>` is the object name in ECAD tool.
- * `<Teamcenter_ItemType>` is the item or sub type of item in Teamcenter.
- * `<Is_PrimaryObject>` is the flag if the `<ECAD_Object>` should be used as primary object to find the candidate of synchronization in synchronize operation in the Teamcenter EDA common client, the value is must be **true** or **false**. If no value is specified, the default value is **false**.

For example:

```
EDALIB_ItemTypesMapping=Component,EDAComPart,true
Symbol,EDASymbol,false
Footprint,EDAFootprint,false
```

To support more than one integration, update the preference for each integration.

Note:

If you set the third parameter of the preference to **true**, the mapped object in Teamcenter is considered the primary object.

The synchronization process starts with the primary object and then the other objects are traversed.

Set the third parameter of the preference to **true** only when required as it may impact performance during the synchronization process.

6. Add the new object to the edalib.xml file.

To maintain the format of the new object between Teamcenter and the ECAD tool, add callbacks to import and export the new object from the ECAD tool.

Add the callbacks to the *ECAD tool name_usrdef.xml* file.

The *ECAD tool name_usrdef.xml* file is located in the *TCEDAECAD_ROOT/gateway* folder.

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 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 prepopulate 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 custom classification schema if you are using custom business objects.

Before you start working with the Teamcenter Classification Admin application:

Prerequisites	<p>Your classification administrator must have already created a classification hierarchy in the Classification Admin application before you can use Classification.</p> <p>For details about how to create a schema, see the <i>Classification Admin</i> help.</p> <p>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.</p>
Enable Classification	<p>Classification does not need to be enabled before you use it.</p> <p>If you have trouble accessing Classification, see your system administrator; it may be a licensing issue.</p>

	<p>Note:</p> <p>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.</p>
Configure Classification	Before you begin using Classification, your Classification administrator must create the classification hierarchy.
Start Classification	<p>Click Classification  in the navigation pane.</p> <p>If you are not licensed to run the application, you see an error message and you are not able to perform any application functions.</p>

Set up Classification mapping

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.

1. Type **edalib_generate_classification_csv_maps -u=infodba -p=infodba -g=dba -classid=ICM -pathroot=d:\my -format=b** to generate the Classification mapping files, **d:\my_eda2tc.csv** and **d:\my_tc2eda.csv**.

Note:

The characters of the class ID are case-sensitive.

2. Create a text dataset named **datasetClassificationMapping_eda2tc** and attach **d:\my_eda2tc.csv** to the dataset.
3. Update the **EDALIB_ClassificationMapping_eda2tc** preference with the value **datasetClassificationMapping_eda2tc**.
 This preference specifies the dataset storing the Classification mappings from the ECAD library to Teamcenter.
 Set the scope of the preference to **Site**. During a **Save** or **Export Library to Teamcenter** operation for a new part, the file is downloaded into the EDA client, and the part is classified according to the mapping.
 Teamcenter EDA first looks for the **EDALIB_ClassificationMapping_eda2tc_library-integration-type** preference. If Teamcenter EDA cannot find the preference, it uses the **EDALIB_ClassificationMapping** preference.
 You can get the value of the *library-integration-type* from the *ecad-tool_usrdef.xml* file that is located in the *TCEDAECAD_ROOT/gateway* directory.
4. Create a text dataset named **datasetClassificationMapping_tc2eda** and attach **d:\my_tc2eda.csv** to the dataset.

5. Update the **EDALIB_ClassificationMapping_tc2eda** preference with the value **datasetClassificationMapping_tc2eda**.
 This preference specifies the dataset storing the Classification mappings from Teamcenter to the ECAD library.
 Set the protection scope of the preference to **Site**. During a checkout or synchronization operation for a new part, the file is downloaded into the EDA client and the part is classified according to the mapping.
 Teamcenter EDA first looks for the **EDALIB_ClassificationMapping_eda2tc_library-integration-type** preference. If Teamcenter EDA cannot find the preference, it uses the **EDALIB_ClassificationMapping** preference.
 You can get the value of the *library-integration-type* from the *ecad-tool_usrdef.xml* file that is located in the *TCEDAECAD_ROOT/gateway* directory.
6. Update the files to define the mapping by replacing *ECAD-library-category* with the ECAD library tool category information.

Example:

my_eda2tc.csv file

```
category,classId
"Electronic Parts,RESISTOR,FIXED,LINEAR",RNC131
"Electronic Parts,DIGITAL CIRCUIT ASSEMBLIES,INTERFACE
CIRCUIT",XJA577
"Electronic Parts,Connector,Card",RNC535
"Resistor",RNC131
"Capacitor",XJA044
"LINEAR",RNC131
"amd",XJA310
"Classification Root,Electronic Parts",id3
"Classification Root,Electronic Parts,RELAYS",XJA245
"Classification Root,Mechanical Parts",id4
"Classification Root,Mechanical Parts,CLAMP",CPN001
"Classification Root,Mechanical Parts,CLAMP,HOSE",RNC382
```

Example:

my_tc2eda.csv file

```
classId,category
RNC131,"Electronic Parts,RESISTOR,FIXED,LINEAR"
XJA577,"Electronic Parts,DIGITAL CIRCUIT ASSEMBLIES,INTERFACE
CIRCUIT"
RNC535,"Electronic Parts,Connector,Card"RNC131,"Resistor"
XJA044,"Capacitor"
```

```

RNC131, "LINEAR"
XJA310, "amd"
id3, "Classification Root,Electronic Parts"
XJA245, "Classification Root,Electronic Parts,RELAYS"
id4, "Classification Root,Mechanical Parts"
CPN001, "Classification Root,Mechanical Parts,CLAMP"
RNC382, "Classification Root,Mechanical Parts,CLAMP,HOSE"

```

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. Set the **EDALIB_PartTypeDefault** preference to the part type you want to use for EDA library integration. The default value of this preference is **EDAComPart**.
2. Set the **EDALIB_ItemTypesMapping** preference to map the ECAD component with the part type set in the **EDALIB_PartTypeDefault** preference.

Example:

For creating a custom part type instead of the default **EDAComPart** part:

- a. Replace the value of the component defined in the **EDALIB_ItemTypesMapping** preference from **Component,EDAComPart,true** to **Component,YourCustomPartType,true**, where *YourCustomPartType* is the part type defined in the step 1.

3. 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**.
4. 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.
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.

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
EDALIB_Classification_Object_Type	This preference is used to define the classification level for EDA library integration.	ItemRevision
ICS_classifiable_types	This preference is used to identify whether a particular type can be	EDAComPart EDAComPart Revision EpIOEDALibPart

Preference name	Description	Default values
	classified or not. It is recommended that the default value should be set as Item/ItemRevision	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.	EDAComPADS EDAComCR5000 EDAComCADStar EDAComPCAD Epl0EDXPackage EDAComPart Epl0EDALibPart EDASymbol EDAFootprint
EDA_ComponentItemTypes	This preference is used to define the set of all Teamcenter base types that should be supported by EDA as components.	EDAComPart
EDALIB_PartTypeDefault	This preference is used to specify the default Item type that EDA uses for Part creation. The value of this preference must be EDAComPart or a valid Teamcenter EDAComPart subtype.	EDAComPart
EDALIB_SymbolTypeDefault	This preference is used to specify the default Item type that EDA uses for Symbol creation. The value of this preference must be EDASymbol or a	EDASymbol

Preference name	Description	Default values
	valid Teamcenter EDASymbol subtype.	
EDALIB_FootprintTypeDefault	This preference is used to specify the default Item type that EDA uses for Foot Print creation. The value of this preference must be EDAFootprint or a valid Teamcenter EDAFootprint subtype.	EDAFootprint
EDALIB_PadstackTypeDefault	This preference is used to specify the default General Design Element type that EDA uses for Pad Stack creation. The value of this preference must be EDAPadstack or a valid Teamcenter EDAPadstack subtype.	EDAPadstack
EDALIB_ItemTypesMapping	<p>This preference is used to specify the item types mapping between ECAD system and Teamcenter.</p> <p>If any of the Teamcenter type (EDAComPart or EDASymbol or EDAFootprint) is changed in this preference, then that type must be updated in the all respective preferences where these types are referenced.</p> <p>Refer the EDALIB_ItemTypesMapping_<application> preferences for the application specific item types mapping between ECAD system and Teamcenter.</p>	Component,EDAComPart,true Symbol,EDASymbol,false Footprint,EDAFootprint,false
EDALIB_GDETypesMapping	This preference is used to specify the General	Padstack,EDAPadstack

Preference name	Description	Default values
	Design Element (GDE) types mapping between ECAD system and Teamcenter.	
EDALIB_ManagedObjects	<p>This preference is used to specify how the EDA common client handles objects with Teamcenter.</p> <p>Refer the EDALIB_ManagedObjects_<application> preferences for the application specific preference values.</p>	<p>Component;view;Symbol,-Footprint;Write</p> <p>Symbol;view;Symbol;Write</p> <p>Footprint;view;Footprint;Write</p> <p>Footprint;EDAPadstackRelation;Padstack;Write</p>

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
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

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.

Following EDA library preferences are used for creating the OOTB Teamcenter relations:

Preference name	Description	Default values
EDAComPart Revision_<DatasetType>-_default_relation	The preference is used to define the default relation between EDAComPart Revision and dataset type specified by <DatasetType>.	IMAN_reference
EDAFootprint Revision_<DatasetType>-_default_relation	This preference is used to define the default relation between EDAFootprint Revision and dataset type specified by <DatasetType>.	IMAN_reference
EDAPackage Revision_<DatasetType>-_default_relation	This preference is used to define the default relation between EDAPackage Revision and	IMAN_reference

Preference name	Description	Default values
	dataset type specified by <DatasetType>.	
EDAPadstack_<DatasetType>_default_relation	This preference is used to define the default relation between EDAPadstack and dataset type specified by <DatasetType>.	IMAN_reference
EDASymbol Revision_<DatasetType>-_default_relation	This preference is used to define the default relation between EDASymbol Revision and dataset type specified by <DatasetType>.	IMAN_reference
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	This preference is used to specify the default Item type that EDA uses for Symbol creation. The value of this preference must be EDASymbol or a valid Teamcenter EDASymbol subtype.	IMAN_master_form_rev bom_view_tags IMAN_requirement 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 vendors vendorparts structure_revisions

Preference name	Description	Default values
		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	The preference is used to list the properties that can be used as children for the type Epl0EDALibPartRevision object in the Rich client.	IMAN_master_form_rev bom_view_tags IMAN_requirement IMAN_manifestation IMAN_reference IMAN_MEView IMAN_vi_sos IMAN_external_object_link IMAN_specification TC_WorkContext_Relation TC_AuditLog TC_Is_Represented_By

Preference name	Description	Default values
		release_status_list structure_revisions view Epl0HasSimulation Epl0HasComponent
Epl0EDALibPartRevision-PseudoFolder	The 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

Based on your connector configurations, specific Teamcenter relations will be used. A list of all supported Teamcenter relations is given below for reference only:

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

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 PLM XML/TC XML Export Import Administration 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.

Name: __EDALIB_find_parts_sync_multiple_sites_SiteA

Description:

Query Type: Local Query

Search Type: EDAComPart ☐ Show Indented Results

Revision Rule:

Property Selection

- EDA Commercial Part
 - IMAN_based_on
 - Fnd0DigitalSignatureRel
 - Fnd0DigitalSignObsoleteRel
 - Trace Link
 - Fnd0ShapeRelation
 - Fnd0DiagramTmpRel
 - Fnd0Diagram_Attaches
 - Fnd0DiagramSnapshot
 - Att0AttrRelation
 - Cls0ClassifiedBy
 - Crt0AnalysisRelation

Search Criteria Order By

	Attribute	User Entry L10N Key	User Entry Name		Default Value
	object type	ObjectType1	ObjectType1	=	EDAComPart
AND	ImanExportRecord<-ixr exported object			IS NULL	
OR	object type	ObjectType2	ObjectType2	=	EDAComPart
AND	ImanExportRecord<-ixr exported objectixr target site name	SiteName1	SiteName1	=	<u>SiteB</u>

- 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.
- Save the query. By default, the type of the query is **Local Query**.
- 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=infodba -p=pw_infodba -g=dba -
query="__EDALIB_find_parts_sync_multiple_sites_SiteA" -run=query
```
- 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 Altium integration, the value can be set to the preference **EDALIB_Sync_FindNewParts_altium**.

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=infodba -p=pw_infodba -g=dba -
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 to launch Teamcenter EDA Environment Manager (TEM) to perform installation and maintenance operations of Teamcenter EDA on an ECAD workstation for library support.

TEM is available with the Teamcenter EDA 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 EDA 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 **Mechatronics Process Management** and select the required options:
 - (Optional) **EDA Translation**
EDA translation allows converting a PCB or a schematic design into the Teamcenter Visualization ECAD format.
 - **ECAD Library Management**
To install ECAD library management, select the library support you require for integration. A separate installation of your design tool is required.
 - **Teamcenter Gateway for EDA Library**
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.
 - 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**.

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.
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 cache and staging directories are created from the settings in the **TCEDAClient.properties** file.

The paths of the cache and staging directories contain double back slashes (\\) on Windows systems. For example:

C:\\ecad\\EdaCache

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 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 kit.

Configuration templates are located in the `TCEDAECAD_ROOT\eda\example\gateway` directory 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 `TCEDAECAD_ROOT/gateway` folder.

For a basic configuration, the minimum required properties are as follows:

- **Family**
This is the subdirectory name under the top-level EDA staging directory.
- **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.
- **PrimaryDataType** and **datasetType**
 - **PrimaryDataType** specifies the design type. Valid types include **schematic**, **pcb**, and **simulation**.
 - **datasetType** represents the Teamcenter dataset type to be created when saving designs using this configuration.
- **RelatedDataType**
The data type that is associated with the primary data type for the concurrent design process.

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.
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="TcEDA"
    className="com.Teamcenter.edabase.lib.DefaultLibraryFactory"
    family="TcEDA">

    <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.

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 Callbacks* section in the callbacks reference document 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	edaLibXmlFileName		
postExportLibrary	applicationName	edaLibXmlFileName	statusFileName	
preSyncLibrary	applicationName	edaLibXmlFileName		

Callback type	Argument 1	Argument 2	Argument 3	Argument 4
postSyncLibrary	applicationName	edaLibXmlFileName	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
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**

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.

- **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 Cadence, Orcad, and Altium library integration.

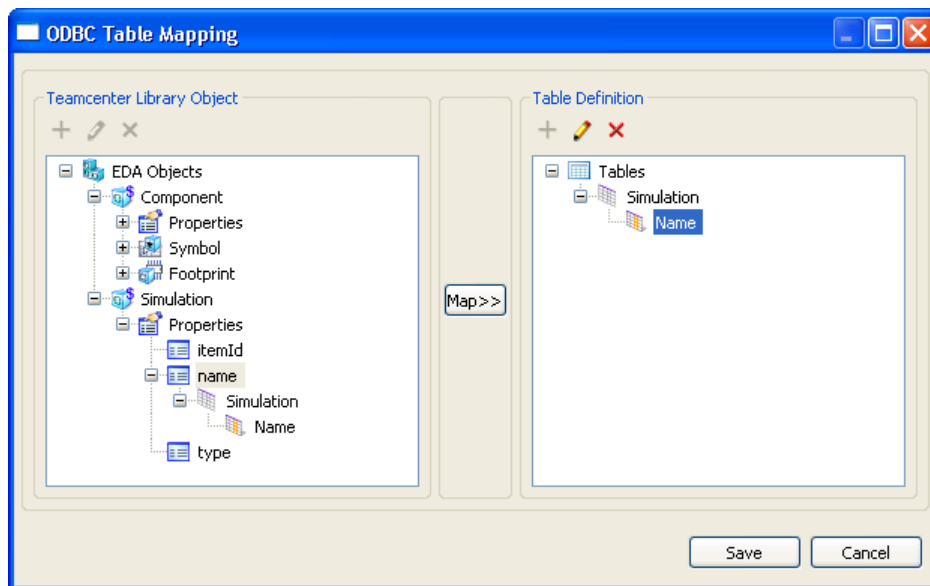
Select the **Configure Part Metadata Database** check box to configure the part metadata database.

Click **Configure** to configure the data source and a mapping file.

- (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.






- (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 .
- C. To add properties or columns to the table, select the table and click the **Add** button .
- 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.

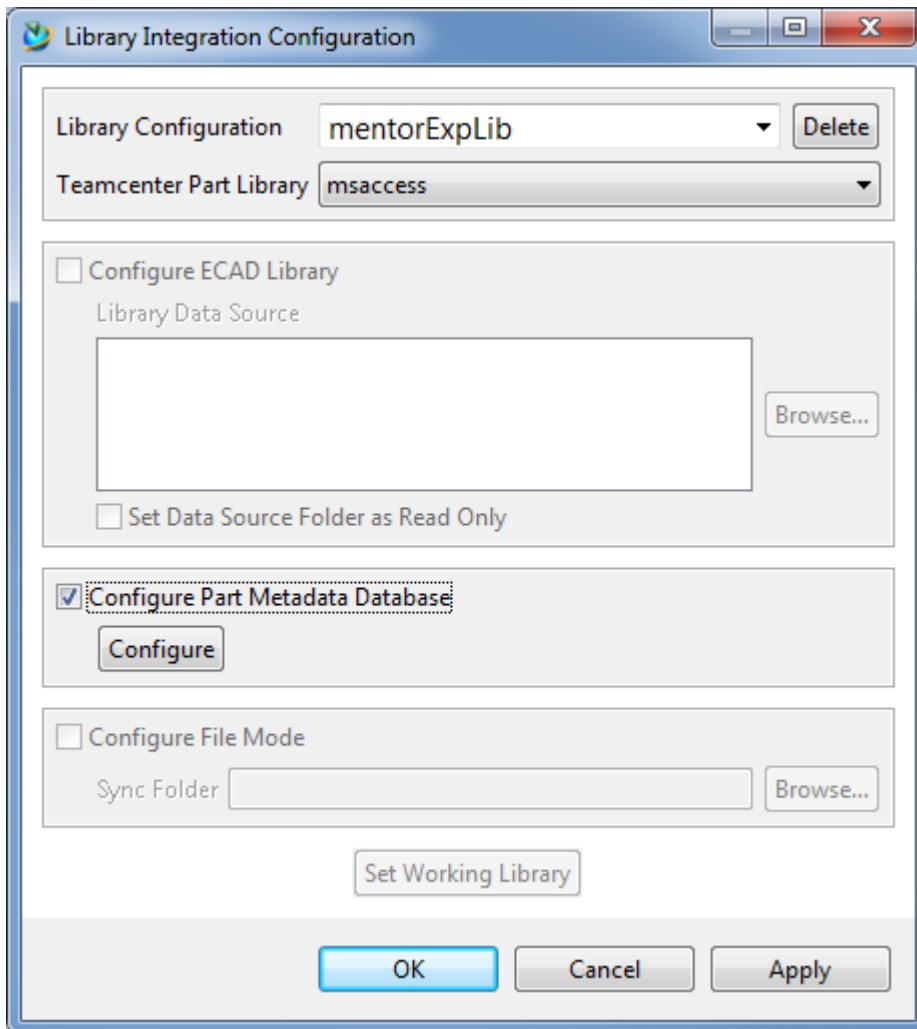
3. Click **OK** to close the dialog boxes and complete the configuration process.

Map ECAD library objects to Teamcenter objects using ODBC mapping

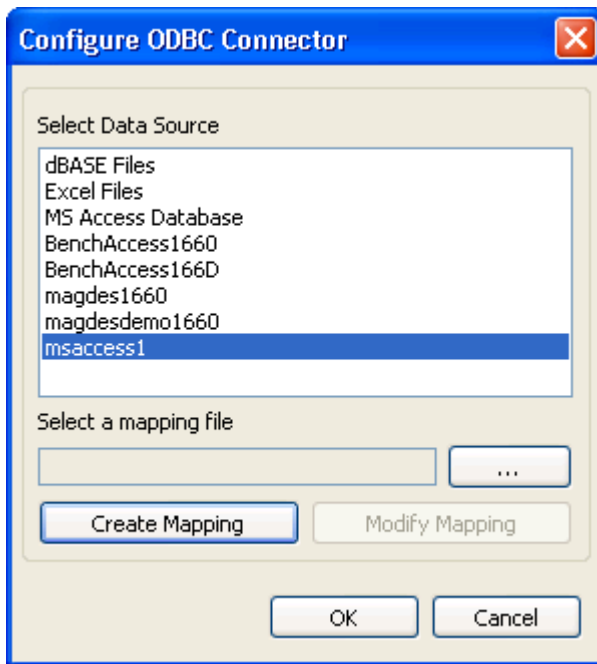
After mapping ECAD library objects to Teamcenter objects, map the properties of ECAD objects to the Teamcenter objects using ODBC mapping. The ODBC mapping feature is only available if your ECAD tool supports ODBC databases.

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.

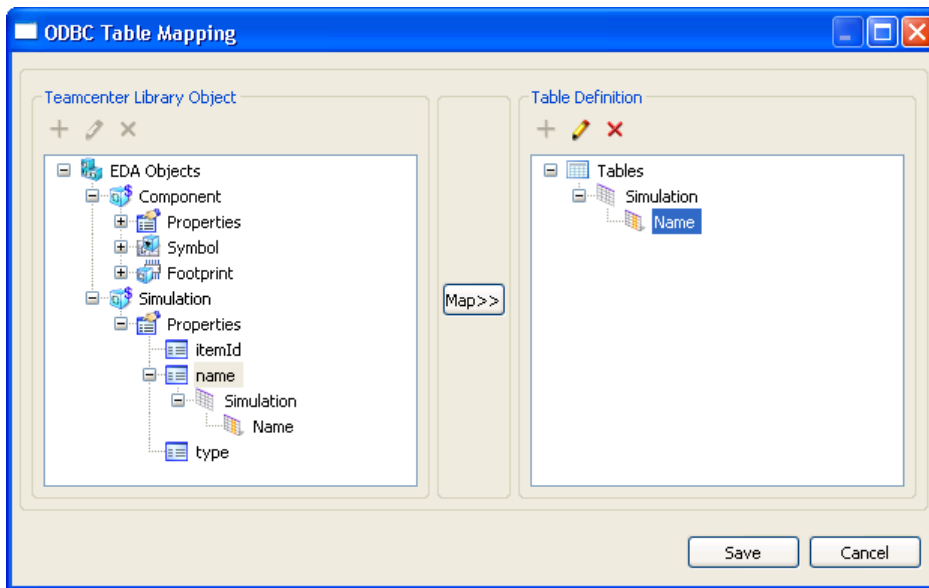
1. In EDA Gateway, choose the **Teamcenter**→**Configure** menu command.
2. In the **Library Integration Configuration** dialog box, click the **Configuration** tab.



3. Select the **Configure Part Metadata Database** option.
Click **Configure** to configure the data source and a mapping file.



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.
5. Browse to an existing 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 to a column of the table.
If the required mapping is not available, click **Create Mapping** to create a new ODBC table mapping or **Modify Mapping** to modify the current ODBC table mapping.



6. In the **ODBC Table Mapping** dialog box, the **Teamcenter Library Object** tree shows all the available EDA objects and its properties.
You can add new objects, for example, **Simulation** as follows:
 - a. Select **EDA objects** and click the **Add** button **+**.
Teamcenter 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 **+**.
Teamcenter displays the **Add a table** dialog box, allowing you to define the name of the new table.
 - c. Click **Save**.

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"/>
```

7. In the **ODBC Table Mapping** dialog box, the **Teamcenter Library Object** tree shows all the available attributes under the EDA objects.
Add a new attribute to the object as follows:

Example:

To add an attribute named **Value**:

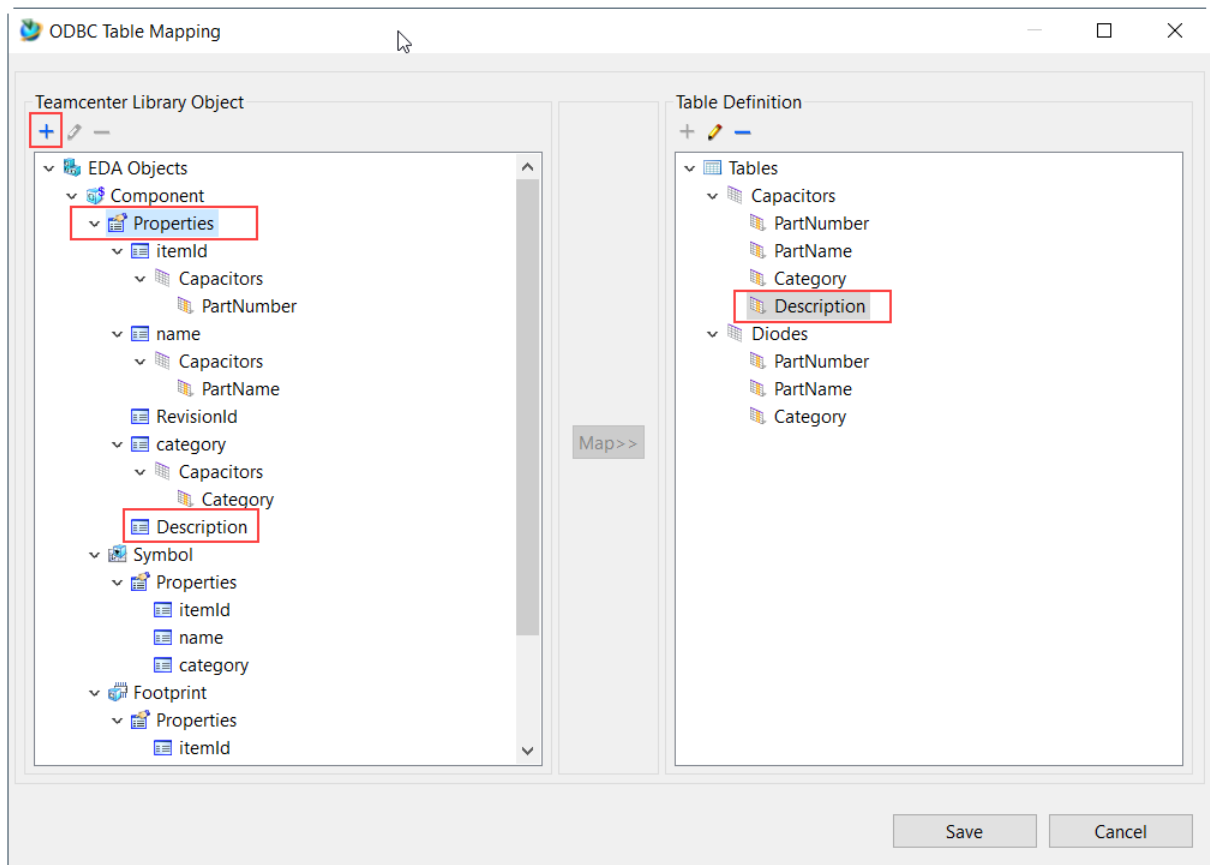
- a. Select a component and click the **Add** button **+**.

Teamcenter displays the **Add a column** dialog box, allowing you to define the name of the new attribute. The new column is added to the table, but no new column is added to the physical table.

The number of characters in the column string must not exceed 8000.

- b. Click **Save**.

8. To add additional properties,
 - a. Select the **Properties** node.
 - b. Click the **Add** button **+** and specify a new property in the **Add Property** dialog box. For example, a new property *Description* is added under *Component*.



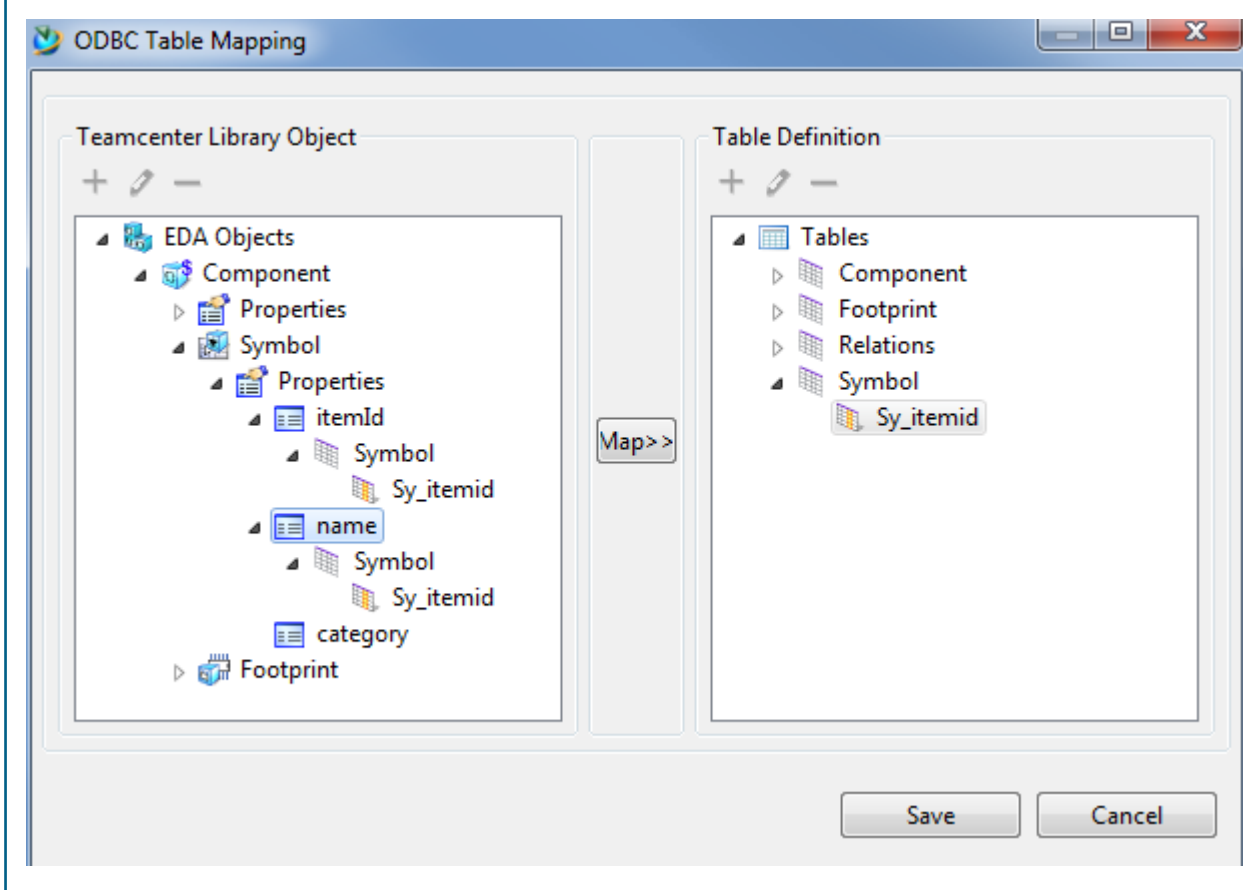
- c. Click **Save**.

9. To map EDA attributes with the attributes of Teamcenter objects, do the following:

- a. 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.
- b. When the list is complete, click **Save** to save the mapping result in a file in a specified folder.

Note:

The **name** attribute is mandatory and some ECAD applications do not have a corresponding attribute. In this case, you can map the **name** attribute to the same ECAD attribute that you used to map the **item ID**.



10. Click **OK** to close the dialog boxes and complete the configuration process.

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"/>
```

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.

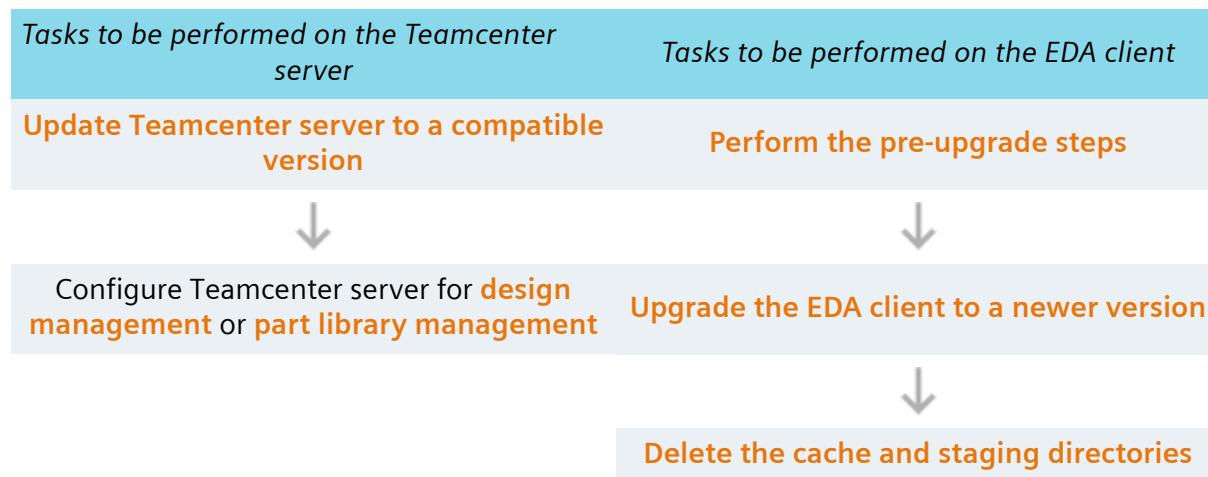
1. Create a Teamcenter proxy user account that will be used to perform the library integration sync operations.
2. In the **TcEDAClient.Properties** file, set the **User** value to match the Teamcenter proxy user ID.
3. Generate a file to hold the Teamcenter proxy user password, using the following command:
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.
5. In a task scheduler, create an entry that invokes the sync script edited in the previous step at the desired time and frequency.

4. 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:



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 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**.

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. You must copy the latest EDA kit on the Teamcenter server machine to perform these configurations.

Configuration	Why is it required?
Import new or updated EDA preferences in Teamcenter to support the latest EDA version	The EDA client requires the latest preferences to be imported in the Teamcenter environment for the client to work correctly.
Import new or updated Active Workspace preferences in Teamcenter to support the latest EDA version	The EDA client requires the latest EDA and Active Workspace preferences to be imported in the Teamcenter environment for the client to work correctly.
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 <i>Awp0ItemRevisionSummary</i> style sheet must be modified.
Add EDA relations and EDA objects in Active Workspace Relation Browser	For Active Workspace Relation Browser to traverse EDA relations and show EDA objects, EDA information must be included in the Active Workspace Relation Browser configuration.
Update the EDA SOA client policy file	You must copy the EDA SOA client policy file from the EDA kit to the Teamcenter server installation folder.
Install xml rendering stylesheet for issue management and collaboration functionality support	For the EDA client Issue Manager and Collaboration functions to work correctly, the <i>IsM1ProblemReportRevisionSummary</i> stylesheet rendering XML file must be installed on the Teamcenter server.

Import new or updated EDA preferences in Teamcenter to support the latest EDA version

The EDA client requires the latest preferences to be imported in the Teamcenter environment for the client to work correctly. Before installing or updating the client, you must import preferences that are specified in an xml file included at the following EDA kit path:

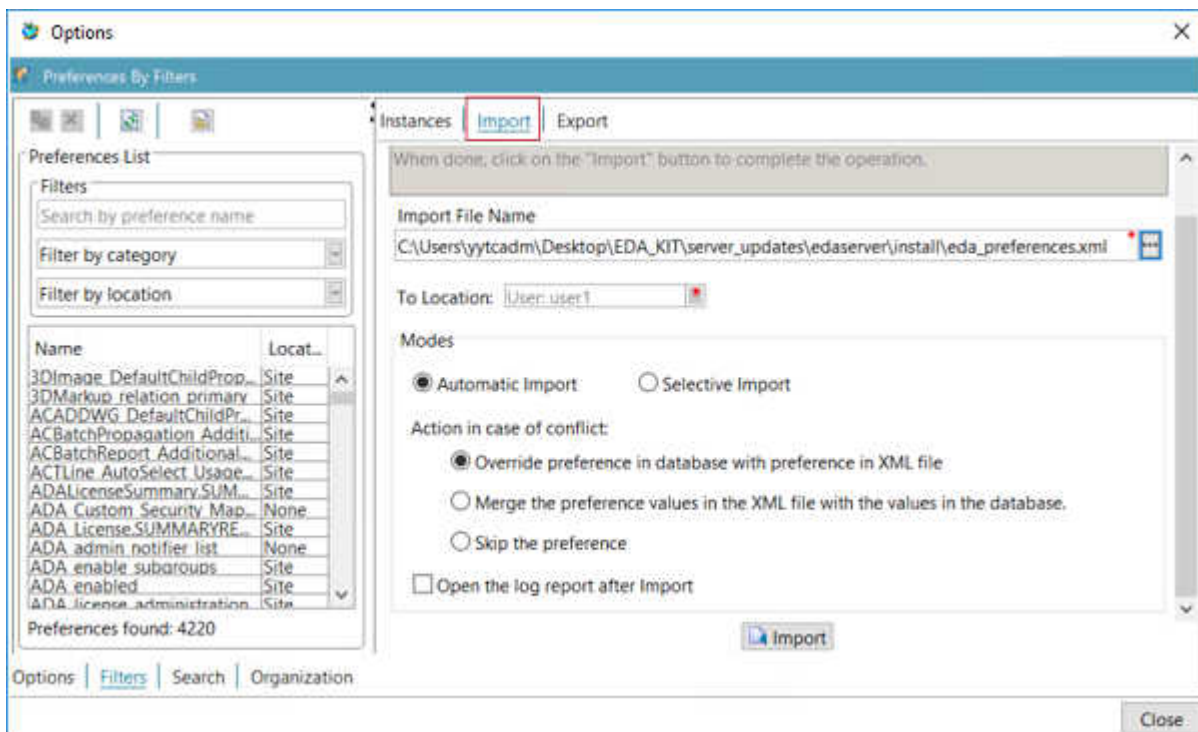
```
<EDA_KIT>\server_updates\edaserver\install\eda_preferences.xml
```


If the preferences are not updated since the Teamcenter base install, this file is not available in the EDA kit.

You can import these preferences in Teamcenter from the rich client or using the Teamcenter command prompt.

Importing preferences in Teamcenter from the rich client:

1. Log on to Teamcenter and start the rich client.
2. Choose **Edit**→**Options** to display the **Options** dialog box.
3. Click **Filters** to open the preferences view.
4. Click **Import**.



5. Browse to the EDA kit location where the preferences file exists and select the *eda_preferences.xml* file for import.
6. Select **Site** from the **To Location** list.
7. Choose **Automatic Import**.
8. If the preference exists in Teamcenter, you can select any one of the **Action in case of conflict** options to override, merge, or skip that preference.

9. Click **Import**.

Importing preferences in Teamcenter from the command prompt:

1. Open a Teamcenter command prompt.
2. Run the following command:

```
preferences_manager -u=<user id> -p=<password> -g=dba -mode=import -scope=SITE -
action=SKIP -file=<EDA_KIT>\server_updates\edaserver\install\eda_preferences.xml
```

Import new or updated Active Workspace preferences in Teamcenter to support latest EDA version

The EDA client requires the latest EDA and Active Workspace preferences to be imported in the Teamcenter environment for the client to work correctly. Before installing or updating the client, you must import preferences that are specified in an xml file included at the following EDA kit path:

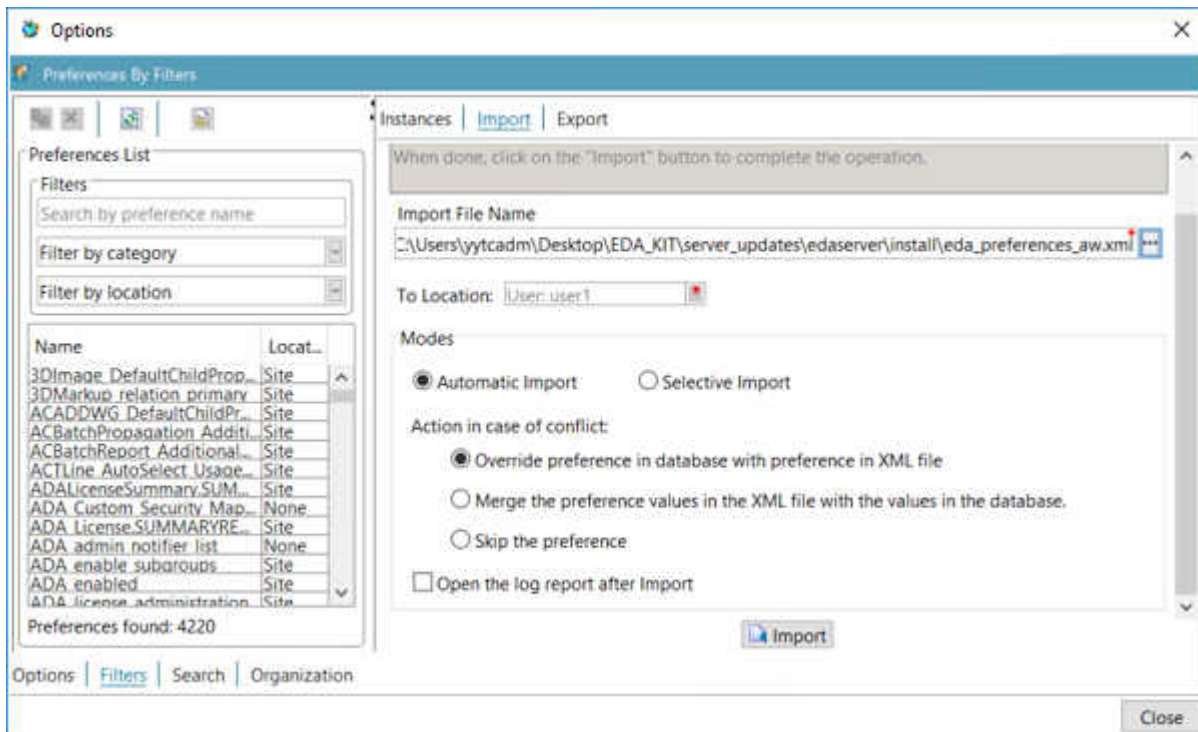
```
<EDA_KIT>\server_updates\edaserver\install\eda_preferences_aw.xml
```

If the Active Workspace preferences are not updated since the Teamcenter base install, this file is not available in the EDA kit.

You can import these preferences in Teamcenter from the rich client or using the Teamcenter command prompt.

Importing preferences in Teamcenter from the rich client:

1. Log on to Teamcenter and start the rich client.
2. Choose **Edit**→**Options** to display the **Options** dialog box.
3. Click **Filters** to open the preferences view.
4. Click **Import**.



5. Browse to the EDA kit location where the preferences file exists and select the `eda_preferences_aw.xml` file for import.
6. Select **Site** from the **To Location** list.
7. Choose **Automatic Import**.
8. If the preference exists in Teamcenter, you can select any one of the **Action in case of conflict** options to override, merge, or skip the preference.
9. Click **Import**.

Importing preferences in Teamcenter from the command prompt:

1. Open a Teamcenter command prompt.
2. Run the following command:
`preferences_manager -u=<user id> -p=<password> -g=dba -mode=import -scope=SITE -action=SKIP -file=<EDA_KIT>\server_updates\edaserver\install\eda_preferences_aw.xml`

Modify the Active Workspace style sheet to include EDA derived data relation

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 `AwpOItemRevisionSummary` style sheet.

To allow EDA derived datasets to be displayed as ItemRevision attachments in Active Workspace, you must modify the *AwpOItemRevisionSummary* 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 *AwpOItemRevisionSummary* dataset.
 - b. Double-click the dataset.
The dataset is automatically checked out and the xml file is launched with the dataset.
3. In the *AwpOItemRevisionSummary* 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
source="IMAN_specification.Dataset, IMAN_reference.Dataset,
+.....IMAN_manifestation.Dataset, IMAN_Rendering.Dataset
, TC_Attaches.Dataset,
+.....IMAN_UG_altrep.Dataset, IMAN_UG_scenario.Dataset, I
MAN_Simulation.Dataset,
+.....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

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

1. Log on to Teamcenter and start the rich client.
2. In the navigation pane, click **My Teamcenter**.
 - a. Search for the *RelationBrowserConf* dataset.
 - b. Double-click the dataset.
The dataset is automatically checked out and the xml file is launched with the dataset.

3. Modify the *RelationBrowserConf* dataset in the *RB_UIConfigure.xml* file to add EDA relations and EDA objects as follows. Modified lines are represented with the plus (+) sign on the extreme left.

```
=====
==
    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>
+.....<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>

```

Update the EDA SOA client policy file

Eclipse provides the framework for interface dialog boxes and provides extension points for communication with the ECAD design tool. All designs are opened and saved in the staging directory whereas the cache holds Teamcenter item data. Communication from client services to Teamcenter is accomplished using SOA services.

You must copy the EDA SOA client policy file from the EDA kit to the Teamcenter server installation folder.

Copy *EDAClientPolicy_1_0.xml* from *<EDA_KIT>\server_updates\tcdata_soa_policies* to *<TC_DATA>\soa_policies*.

This file may not exist in a patch kit if it has not been updated since the base version install.

If a previous version of the EDA SOA client policy file exists, make a backup copy of the file before replacing it with the new file. After copying the new file, you must carry forward all customizations performed in the old policy file into the new file.

Install xml rendering stylesheet for issue management and collaboration functionality support

For the issue management and collaboration functions to work correctly in the EDA client, the xml file rendering the *IsM1ProblemReportRevisionSummary* stylesheet must be installed on the Teamcenter server.

Copy the EDA kit to the Teamcenter server machine and perform the following updates. This file may not exist in a patch kit if it has not been updated since installing the base version.

Open a Teamcenter command prompt and run the following command:

```
install_xml_stylesheet_datasets -u=<user id> -p=<password> -g=dba-input=<EDA_KIT>
\server_updates\edaserver\install\import_eda_stylesheets.txt -filepath=<EDA_KIT>\server_updates
\edaserver\install
```

Configure Teamcenter for part library management

Overview of configurations required for part library management

You must **import new or modified EDA library preferences in Teamcenter to support latest EDA version** for part library management.

Import new or modified EDA library preferences in Teamcenter to support latest EDA version

Before installing the EDA client, you must import new or updated preferences from the EDA kit in the Teamcenter environment. You must import preferences that are specified in an xml file included in the EDA kit path specified below for the EDA client to work properly. If the preferences are not updated or added since the Teamcenter base install, this file will not be available in the EDA kit.

- `<EDA_KIT>\server_updates\edaserver\install\edalib_preferences.xml`
- `<EDA_KIT>\server_updates\edaserver\install\edalib_preferences_merge.xml`
- `<EDA_KIT>\server_updates\edaserver\install\edalib_preferences_override.xml`

You can import these preferences in Teamcenter from the rich client or using the Teamcenter command prompt.

Importing preferences in Teamcenter from the rich client:

1. Log on to Teamcenter and start the rich client.
2. Choose **Edit**→**Options** to display the **Options** dialog box.
3. Click **Filters** to open the preferences view.
4. Click **Import**.

5. Browse to the EDA kit location where the preferences file exists and select the *edalib_preferences.xml* file for import. Repeat the procedures to import preferences from the *edalib_preferences_merge.xml* and *edalib_preferences_override.xml* files.
6. Select **Site** from the **To Location** list.
7. Choose **Automatic Import**.
8. If the preference exists in Teamcenter, you can select any one of the **Action in case of conflict** options to override, merge, or skip that preference.
9. Click **Import**.

Importing preferences in Teamcenter from the command prompt:

1. Open a Teamcenter command prompt.
2. Run the following command:

```
preferences_manager -u=<user id> -p=<password> -g=dba -mode=import -scope=SITE -
action=SKIP -file=<EDA_KIT>\server_updates\edaserver\install\edalib_preferences.xml
```

Steps to perform before you upgrade the EDA client

Before upgrading the EDA client on your ECAD workstation:

- 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>\leda` 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.

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. 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:

1. Extract the Teamcenter EDA software distribution image to a temporary location.
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 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.
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.
 - 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.

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.**
 - You can find the cache directory of the ECAD designer from the value of the **CacheDir** property in the **TCEDAClient.properties** file. The default value of the cache directory is `USERPROFILE\Teamcenter\EDA\Cache`.
 - If you cannot find the location of the cache directory in the `USERPROFILE\Teamcenter\EDA\TCEDAClient.properties` file, find the location of the cache directory in the value of the **CacheDir** property, in the `TCEDAECAD_ROOT\TCEDAClient.properties` file.
- **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.

5. 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.

6. 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.

Siemens Digital Industries Software

Headquarters

Granite Park One
5800 Granite Parkway
Suite 600
Plano, TX 75024
USA
+1 972 987 3000

Americas

Granite Park One
5800 Granite Parkway
Suite 600
Plano, TX 75024
USA
+1 314 264 8499

Europe

Stephenson House
Sir William Siemens Square
Frimley, Camberley
Surrey, GU16 8QD
+44 (0) 1276 413200

Asia-Pacific

Suites 4301-4302, 43/F
AIA Kowloon Tower, Landmark East
100 How Ming Street
Kwun Tong, Kowloon
Hong Kong
+852 2230 3308

About Siemens Digital Industries Software

Siemens Digital Industries Software is a leading global provider of product life cycle management (PLM) software and services with 7 million licensed seats and 71,000 customers worldwide. Headquartered in Plano, Texas, Siemens Digital Industries Software works collaboratively with companies to deliver open solutions that help them turn more ideas into successful products. For more information on Siemens Digital Industries Software products and services, visit www.siemens.com/plm.

This software and related documentation are proprietary and confidential to Siemens.

© 2020 Siemens. A list of relevant **Siemens trademarks** is available. Other trademarks belong to their respective owners.