TEAMCENTER

Xpedition and PADS Design Management

Teamcenter 2412

SIEMENS

About Siemens Digital Industries Software

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

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

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

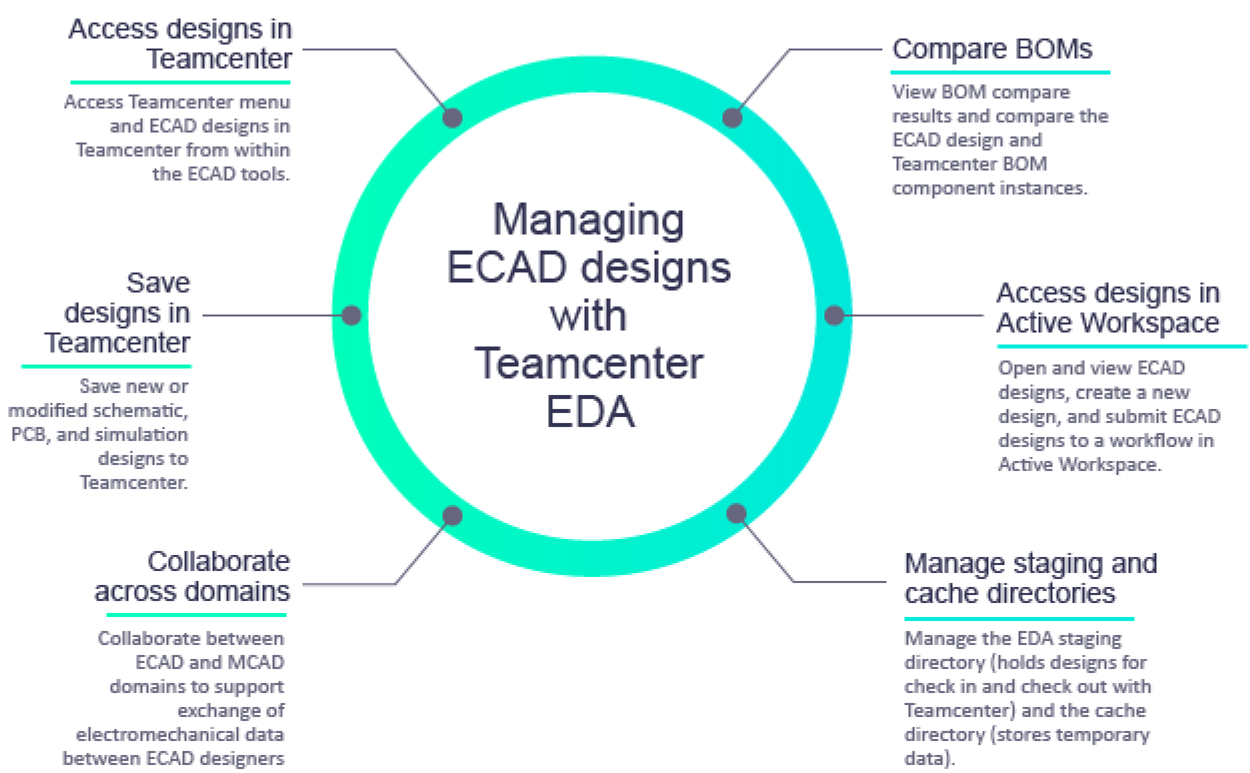
Contents

1. Managing ECAD designs in Teamcenter

Teamcenter Electronic Design Automation (EDA) integrates Siemens EDA electronic computer aided design (ECAD) applications with Teamcenter to allow the ECAD designers to move design data into and out of Teamcenter and manage ECAD part library data in Teamcenter.

Teamcenter EDA supports ECAD design integration and ECAD part library integration. Both these integrations are separate functionalities and run separately. You can install them either together or separately, depending on your requirement.

An ECAD designer can use the design integrations that embed Teamcenter menus into the ECAD design tool 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. In the ECAD tools that do not support embedded EDA integrations, Teamcenter Gateway for EDA is used for integration instead.



Where do I go from here?

Administrator	See the Xpedition and PADS Integration With Teamcenter documentation for instructions on

	how to install and configure Teamcenter EDA for Siemens EDA ECAD applications.
Administrator	See the Installing and Configuring EDA Gateway for (Non-Siemens EDA) ECAD Applications documentation for instructions on how to install and configure EDA Gateway for (non-Siemens EDA) ECAD applications.
Library Administrator	See Managing Xpedition or PADS Part Library in Teamcenter for instructions on how to manage ECAD part library data in Teamcenter.
ECAD Designer	
How do I to access designs in Teamcenter?	Once your Teamcenter client is ready for use and integrated with your ECAD tool, see the information available in the section on **accessing Teamcenter from your ECAD tool**.
Save design files in Teamcenter	You can save schematic, PCB, and simulation designs toTeamcenter. View the list of commands for **saving design files in Teamcenter**.
How can I use the cache and staging directories in EDA?	The EDA staging directory holds designs for checkin and checkout operations with Teamcenter, while the cache stores data temporarily. See the section on **working with the cache and staging directories** for details.
How do I compare BOMs?	You can compare the components in the ECAD design with the components of the same design in Teamcenter. The section on **Comparing ECAD design BOMs** with Teamcenter has information on this.
Access ECAD designs in Active Workspace	Follow the steps provided in the section on how to **open a design in Active Workspace**.
Collaborate on the designs across different domains	EDA allows engineers from different domains to work on the same BOM and collaborate on the design creation. The collaboration functionality is only available for PCB designs. To understand this better, go through the **overview of collaboration across domains**.

2. Accessing Teamcenter menu from your ECAD tool

Prerequisites to enable Teamcenter menu in your ECAD tool

Your Teamcenter client must be up and running and integrated with your ECAD tool before you can use Teamcenter for managing your designs. This is a one-time process.

If your design tool supports embedding the Teamcenter menu in the user interface, you must enable the Teamcenter menu on the EDA client for your design tool.

If your design tool does not support embedding the Teamcenter menu, you must configure Teamcenter Gateway for EDA to perform basic Teamcenter functions, such as opening, checking out, and saving designs, including schematic designs with or without variants.

Before you enable the menu, verify that:

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

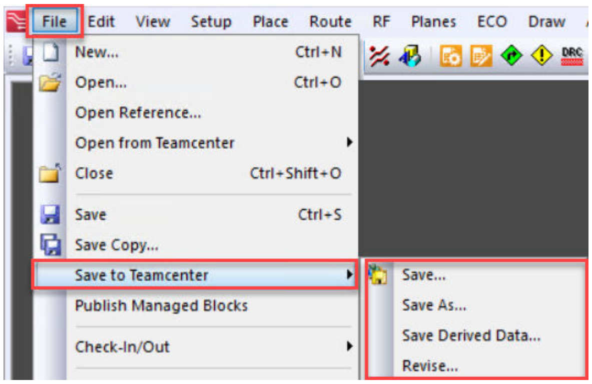
Access Teamcenter menu from your ECAD tool

The steps to access the Teamcenter menu commands differ across ECAD tools. Follow the steps given below to access the Teamcenter menu commands from within your ECAD tool to perform ECAD operations.

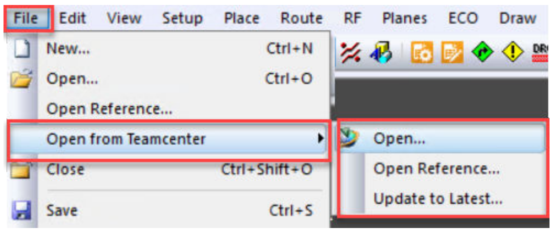
If you are using Xpedition Designer:

Click **File** and choose the applicable Teamcenter operation from the following options:

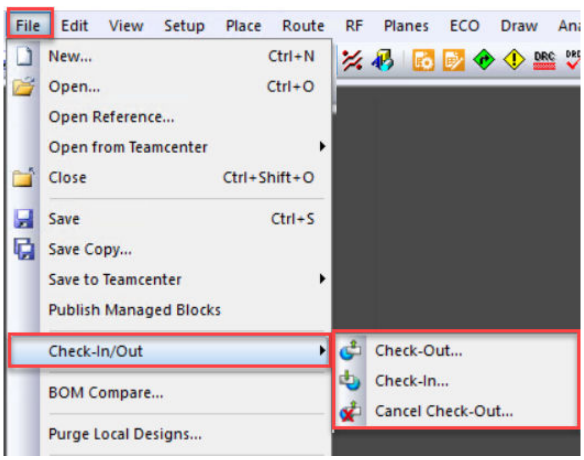
- **Save to Teamcenter**



- **Open from Teamcenter**

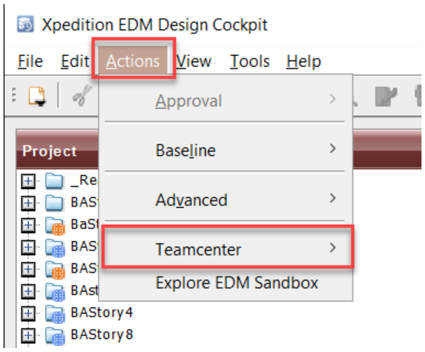


- **Check-In/Out**



If you are using Xpedition EDM Design Cockpit:

Choose **Actions** → **Teamcenter**.

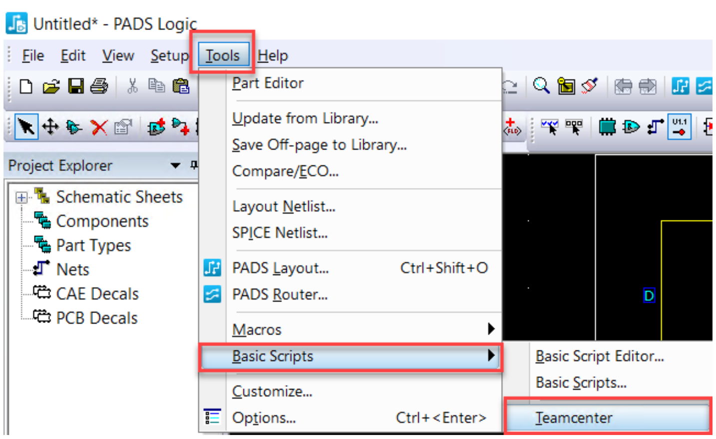


If you are using Mentor PADS Designer:

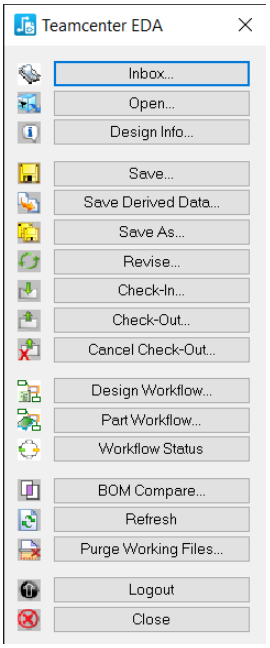
Click **Teamcenter** on the toolbar.

If you are using PADS Layout or PADS Logic

Choose **Tools→Basic Scripts→Teamcenter**.



From the Teamcenter EDA dialog box, select the operations as required.



If the Teamcenter menu is not listed under **Tools**, perform the steps explained in the *Enable Teamcenter menu in PADS Layout or PADS Logic* topic.

Enable Teamcenter menu in PADS Layout or PADS Logic

For Mentor PADS Laout or PADS Logic, if the Teamcenter menu is not listed under **Tools**, perform the following steps:

Procedure

1. Add the Mentor PADS application by completing the following steps. This is a one-time procedure.

    a. Create Windows shortcuts to the launching scripts located in *%TCEDAECAD_ROOT%* **\PADS\wnti\bin**. You can place these shortcuts on your desktop.

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

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

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

    e. Click **Close** and quit the application.

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

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

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

Perform the above steps for the Mentor PADS Layout to activate the Teamcenter menu.

View the EDA task worklist

Choose **Teamcenter**→**Inbox**.

The **My Worklist** view is displayed.

> Note:
>
> Refer to the *Access Teamcenter menu from your ECAD tool* topic for the initial steps to access the Teamcenter menu commands specific to your ECAD tool to perform the ECAD operation.

3. Log on to Teamcenter

When you are not logged on to **Teamcenter** and you choose any **Teamcenter** command in your ECAD design tool, the EDA **Login** dialog box is displayed.

> Note:
>
> The **Login** dialog box may be hidden behind your design tool dialog box. Minimize or move other dialog boxes to reveal the **Login** dialog box.

1. In the Teamcenter Login dialog box, type your user name, password, role, and group (if applicable).

2. Click **Login**.

3. To log off from EDA, choose **Teamcenter→Logout**.

   The next time you choose a command from the **Teamcenter** menu, you are prompted to log on.

It is always a good practice to use the **Logout** command when you complete working in the application.

The **Logout** command only releases the Teamcenter license. To release the ECAD tool license, use the ECAD tools.

4. Change your user group or role

1. Navigate to any EDA wizard such as **Save As** or **Revise**.

2. Click the link with your name, group, and role.

3. In the **User Settings** dialog box, change the **Group** and **Role** as required and click **OK**.

5. Managing ECAD designs in Teamcenter

Open a design in Teamcenter from the ECAD tool

To search for a design dataset file in Teamcenter and open it for reference (read-only) or modification (read and write):

1. In the ECAD design tool, choose **Teamcenter→Open**.

   If prompted, **log on**.

   > Note:
   >
   > If you are using a different ECAD tool, refer to the *Access Teamcenter menu from your ECAD tool* topic for the initial steps to access the Teamcenter menu commands specific to your ECAD tool to perform the ECAD operation.

2. In the **Open Design** dialog box, browse to the design you want to open.

   You can also search for designs based on **Type** or **Criteria**.

3. In the ECAD design tool, choose **Teamcenter→Design Info**.

   The design information is displayed.

4. To open the dataset for modification, select the **Check-out** check box.

5. Click **OK**.

   The dataset opens in the design tool.

Open design related datasets in Teamcenter from the ECAD tool

1. In the ECAD design tool, choose **Teamcenter→Open**.

   If prompted, **log on**.

   > Note:
   >
   > If you are using a different ECAD tool, refer to the *Access Teamcenter menu from your ECAD tool* topic for the initial steps to access the Teamcenter menu commands specific to your ECAD tool to perform the ECAD operation.

2. In the **Open Design** dialog box, browse to the design for which you want to open the related datasets.

3. (Optional) To open related design files, choose one of the following from **Open Related Datasets**:

   - **Blank** (do not download)

     If the related dataset is not in your cache or is checked out by someone else, you can choose the blank space or **Reference**.

   - **Reference** (download only for reference)

     If the dataset is already in your cache for reference, **Reference** is unavailable.

   - **Check-out** (download and check out)

     If the dataset is checked out, **Check-out** is unavailable.

4. Click **OK**.

   The dataset opens in the design tool.

Check out a design

The **Check-Out** command is used to make revisions to the designs in the staging directory that are saved in Teamcenter and are in a checked-in state. When you check out the design, a check-out lock is placed on the design in Teamcenter so that no one else can open it for modification. The **Check-out** check box is disabled if you already have a checked-out version of the design in your staging directory or if the dataset is already checked out to you or someone else.

You can only check out the latest version of the design. You can view earlier versions of a design using the **Open** command.

When a design is checked out, the **CO** (check out) column in Teamcenter displays a **Y**.

1. If you want to revise a design that you have not worked on before, and the design is therefore not in your staging directory, choose **Teamcenter→Open** to search for the design. Then, to open the design for modification, select the **Check-out** check box.

   > Note:
   >
   > If you are using a different ECAD tool, refer to the *Access Teamcenter menu from your ECAD tool* topic for the initial steps to access the Teamcenter menu commands specific to your ECAD tool to perform the ECAD operation.

2. In the ECAD design tool, open the schematic, PCB, or simulation design located in your staging directory that you checked in previously and want to revise now.

3. Choose **Teamcenter→Check-out** .

4. In the **Check-out** dialog box, type an identification number for the check out in the **Change ID** box and type a description in the **Comments** box.

   Click **OK**.

5. Verify that the checkout status of the design in Teamcenter is **Y**.

   The **Check-out** dialog box is not displayed if the **EDA_SkipCheckoutDialog** preference is set to **True**. If this preference is not defined, Teamcenter uses the **Confirm_checkout_suppressed** preference to control whether the **Check-out** dialog box is displayed.

Cancel a checkout

Perform the following steps to cancel an existing checkout lock on the design in Teamcenter. This reverts the opened design to the version prior to the checkout. Only the ECAD designer who originally performed the checkout action can cancel the checkout.

> Warning:
>
> Canceling a checkout overwrites the modified design files in the staging directory with the last checked-in design files.

1. In the ECAD design tool, choose **Teamcenter→Cancel Check-Out**.

   > Note:
   >
   > If you are using a different ECAD tool, refer to the *Access Teamcenter menu from your ECAD tool* topic for the initial steps to access the Teamcenter menu commands specific to your ECAD tool to perform the ECAD operation.

2. In the **Cancel Check out** dialog box, select the **Remove working files from local storage** check box if you want to remove the design files from the staging directory.

   Removing working files from local storage only removes the design files. These are the files that are saved to Teamcenter and then opened from Teamcenter. Other files created while working with the design are not removed when this check box is selected. The design files associated with related designs are not removed either.

3. Click **OK**.

   An **Open Design** dialog box displays the following message:

`Directory` *directory-name* `already exists, do you want to overwrite this directory?`

4. Click **OK** if you want to replace the current version of the design with the previously checked-out version.

   The check-out lock is removed, and the design is reverted to its previous state.

Check in designs

Use the **Check-In** command to save the design changes back in Teamcenter. Ensure that you edit and check in the design from the same workstation where you had initially downloaded and checked out the design. Even if you open the design on another workstation, you must not edit, save, and check in the design from that workstation. The design must be checked in only from the workstation where it was checked out.

1. To save any changes made to a design in Teamcenter, choose **Teamcenter→Check-In**.

   The **Check-In** dialog box appears.

   > Note:
   >
   > If you are using a different ECAD tool, refer to the *Access Teamcenter menu from your ECAD tool* topic for the initial steps to access the Teamcenter menu commands specific to your ECAD tool to perform the ECAD operation.

2. If you see the message `This item is not known to Teamcenter` in your design tool, the design has not been previously saved in Teamcenter. To save the design, choose **Teamcenter→Save As**.

3. If you see the message `The design has not been checked out for this operation` in your design tool, the design has not been checked out from Teamcenter. Check out the design, make modifications, and then choose the **Check-In** command.

4. Select the **Save BOM** check box to indicate whether to save a BOM with the design.

5. Select the **Remove working files from local storage** check box to indicate if you want to remove the design files from the staging directory after check in.

6. Click **Finish**.

   The design is checked in to Teamcenter and replaces the previous version.

6. Working with EDM projects

Create an EDM project from a Teamcenter design

To create an EDM project from a Teamcenter design:

1. Click the **http://<EDM_SERVER_HOST>:<PORT>/xdm** URL to open the **EDM portal** web application, where:

    - **EDM_SERVER_HOST** is the host name of the EDM server where it is installed.

    - **PORT** is the port number on which the EDM server is running, for example, **http://** *edm_server_machine_name***.net.plm.eds.com:31000/xdm**.

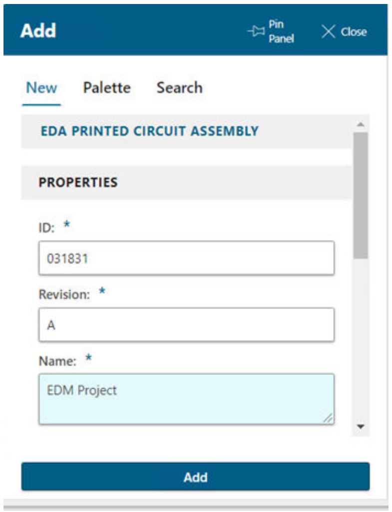
2. Navigate to the folder where you want to create a new EDM project for a Teamcenter design.

3. Select **More Commands** ••• > **New** > **Add** ⊕.

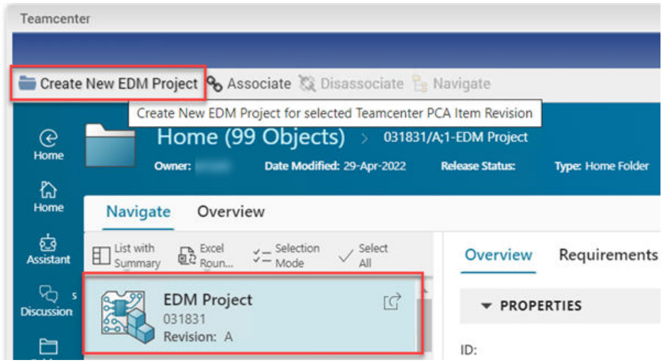
    The **Add** panel opens.

4. In the **Add** panel, select the Teamcenter design that you want to associate to the project, for example, **EDA Printed Circuit Assembly**.

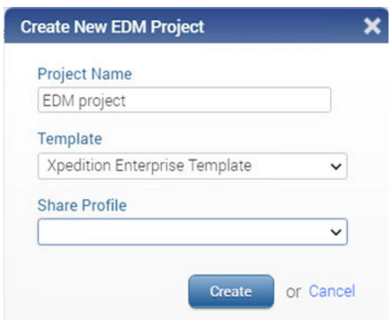
5. Specify the details for the new EDM project in the **Add** panel and click **Add**.



6. Select the project and click **Create New EDM Project** on the toolbar.

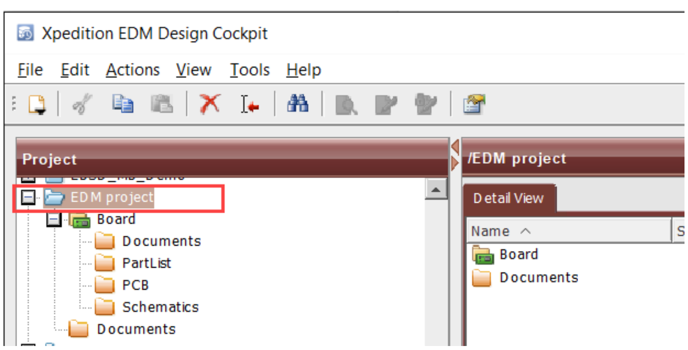


7. In the **Create New EDM Project** dialog box, specify the **Project Name** and select **Xpedition Enterprise Template** for your EDM project.



8. Click **Create**.

A new EDM project associated with the Teamcenter design is created in Xpedition EDM.



Associate an EDM project board to a Teamcenter design

To associate an EDM project to a Teamcenter design:

1. Navigate to the web address of your EDM Portal and open a design in EDM Collaborate from Active Workspace.

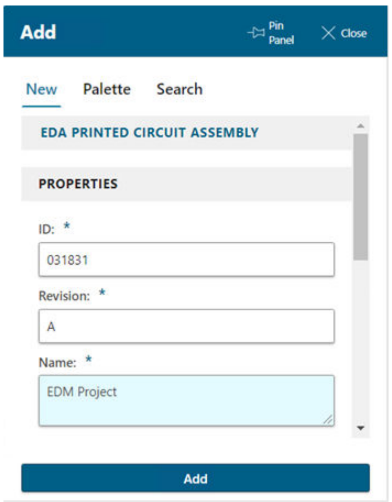
2. Navigate to the folder where you want to create a new EDM project to associated with the existing project in EDM.

3. Select **More Commands** ••• > **New** > **Add** ⊕.

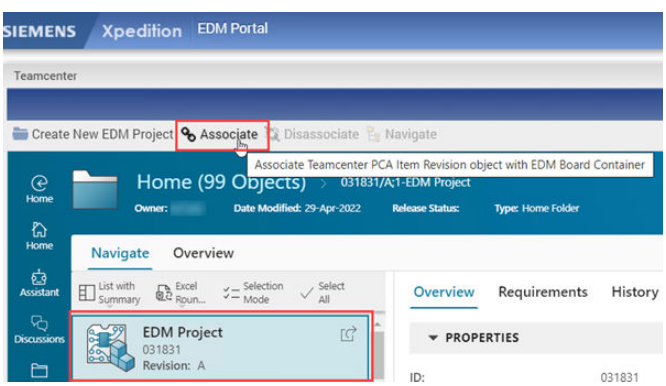
   The **Add** panel opens.

4. In the **Add** panel, select the Teamcenter design that you want to associate to the project, for example, **EDA Printed Circuit Assembly**.

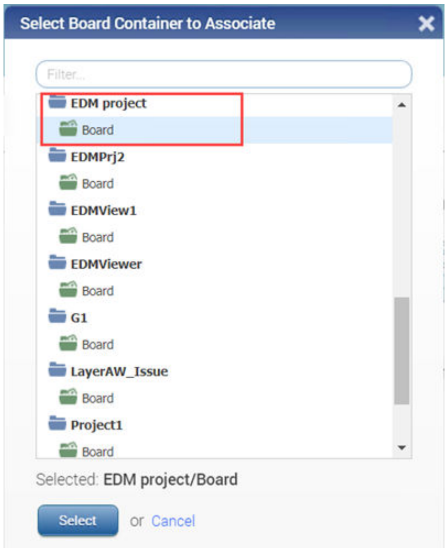
5. Specify the details for the new EDM project in the **Add** panel and click **Add**.



6. Select the project and click **Associate** on the toolbar.



7. In the **Select Board Container to Associate** dialog box, select the EDM project or Board that you want to associate with the design.

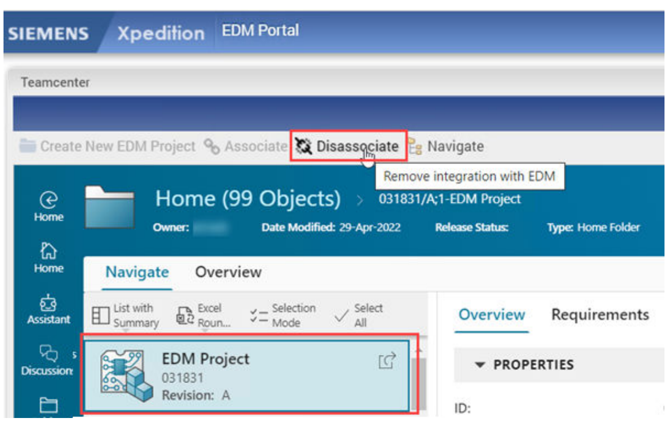


8. Click **Select**.

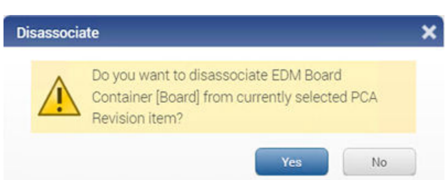
   The EDM project in Xpedition EDM is associated with the Teamcenter design.

To disassociate an EDM project from the Teamcenter design:

1. Select the project and click **Disassociate** on the toolbar.



2. Click **Yes** to disassociate the EDM project from the Teamcenter design.



7. Saving designs

Saving designs to Teamcenter

You can save schematic, PCB, and simulation designs to Teamcenter. The following commands are used for saving designs:

> Note:
>
> If you are using a different ECAD tool, refer to the *Access Teamcenter menu from your ECAD tool* topic for the initial steps to access the Teamcenter menu commands specific to your ECAD tool to perform the ECAD operation.

- Use **Save** to save changes to the current design, without creating a copy or a new revision.
- Use **Save As** to save a new design based on the current design. The design must *not* exist in Teamcenter.
- Use **Save Derived Data** to save updated derived data. If the parent object is released, you can revise the parent object.
- Use **Revise** to create a new revision of the current design. The design must already exist in Teamcenter.
- Use **Check-In** to check the current design into Teamcenter.

Save a new design in Teamcenter

The steps to save a new dual model design or a new combined model design to Teamcenter are similar. To save a dual model design, you must additionally provide information about the schematic items.

1. In the ECAD design tool, open a design (dual or combined model) for modification and make the desired changes to the design.

2. In the ECAD design tool, **navigate** to **Teamcenter→Save As** and, if prompted, **log on**.

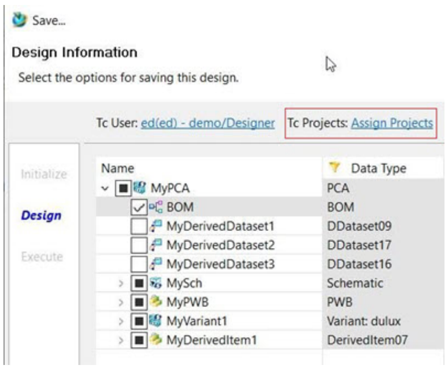
   Teamcenter displays the **Design Information** wizard dialog box.

   > Note:
   >
   > The PWB ID and name default to the values provided by the design tool. Teamcenter appends the value of **DefaultPWBSuffix** to each value.

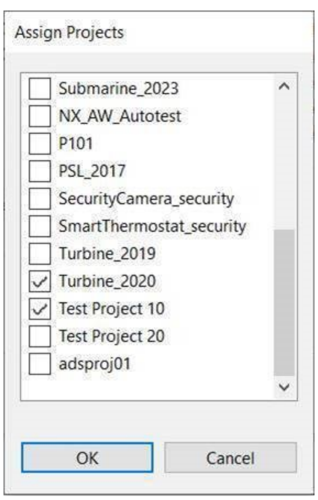
Note:

If you are using a different ECAD tool, refer to the *Access Teamcenter menu from your ECAD tool* topic for the initial steps to access the Teamcenter menu commands specific to your ECAD tool to perform the ECAD operation.

3. On the **Design Information** dialog box, click the **Assign Project** link to assign a Teamcenter project to the design. If a default project is assigned to a user, the default project name is displayed instead of the **Assign Project** link.

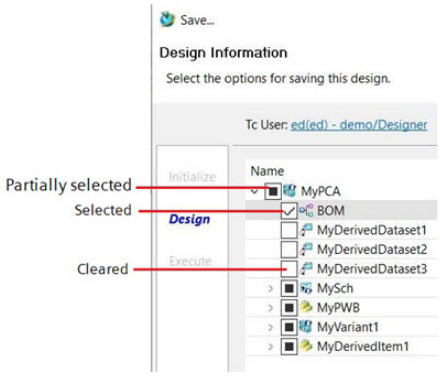


4. From the **Assign Projects** dialog box, select one or more projects that you want to assign to the design and click **OK**. All projects assigned to the user in Teamcenter will be displayed for selection.



5. Enter the information for design items as follows:

   a. Select the check box next to the items you want to save. The **check box** has three states: **selected**, **cleared**, and **partially selected**.



- The **selected** check box indicates that the item and all its child nodes are selected for the update.
- The **cleared** check box indicates that neither the item nor any child node is selected for the update.
- The **partially selected** check box indicates that the item is selected but at least one of its child nodes is not selected.

b. You can double-click a specific field to update the name and ID for any editable item manually. Not all fields are editable.

c. Double-click the relevant cell in the **Select Option** column and choose **Save as new** to create a new PCA item in Teamcenter.

OR

Choose **Select existing** to save it to an existing PCA item in Teamcenter. If you choose **Select existing**, you must browse to the relevant PCA item.

d. Double-click the relevant cell in the **Select Option** column and choose **Save as new** to create a new PWB item in Teamcenter.

OR

Choose **Select existing** to save it to an existing PWB item in Teamcenter. If you choose **Select existing**, you must browse to the relevant PWB item.

e. (For dual model designs only) Double-click the relevant cell in the **Select Option** column and choose **Save as new** to create a new schematic item in Teamcenter.

OR

Choose **Select existing** to save it to an existing schematic item in Teamcenter. If you choose **Select existing**, you must browse to the relevant schematic item.

f. (For dual model designs only) Expand the **Schematic** node to enter or update the information for the **Design** node.

g. Select the **BOM** check box to save a bill of materials (BOM) with the design item. Double-click within the **Select Option** column to specify whether to save a BOM from the PCB or from the schematic or to not save a BOM.

h. Select the **Viewable** check box to save the neutral SCH (XSCH) file with the design.

i. To save variants, specify all the required details the same way.

6. (Optional) Click **Assign ID** to generate IDs for all the items that have a blank ID field and are selected for saving.

7. (Optional) Click **Details** to view or enter additional information for the items.

8. To revert the changes to the design information tree, click **Reset**. All fields are restored to their initial rendering state. Any changes made to the **Details** dialog box are also reverted.

   When a row is selected, the reset action reverts all changes made to the selected row.

   When none of the rows is selected, the reset action reverts all the changes made to the design information tree.

9. (Optional) Select **Check-In** to check in the PCA and PWB when the design is saved.

10. (Optional) Select **Remove working files from local storage** to delete the associated files from your workstation after the design is checked in. This check box is unavailable if you do not select the **Check-In** check box.

11. Click **Finish** to save the design or **Next** to proceed to the BOM compare results.

    If you had earlier selected the option to **Check-In** your designs, your design is saved and checked in to Teamcenter. The save operation may take several minutes.

    To view the BOM compare option, you must configure the **EDA_BOMCompareOnSave** preference. If not configured, the BOM compare option may not be available on your **Save** or **Revise** wizard.

Save or revise updated designs in Teamcenter

You can check out and open a design for modification in the ECAD tool. Once you update the file, you can save the new version of the updated design.

Use the **Save** command to save a new version of an opened and checked-out design to Teamcenter. The revision remains the same.

Use the **Teamcenter→Revise** command to create a new revision. If an item is in the released state, it can be revised but it cannot be saved.

1. In the ECAD design tool, open a design for modification and make the desired changes to the design.

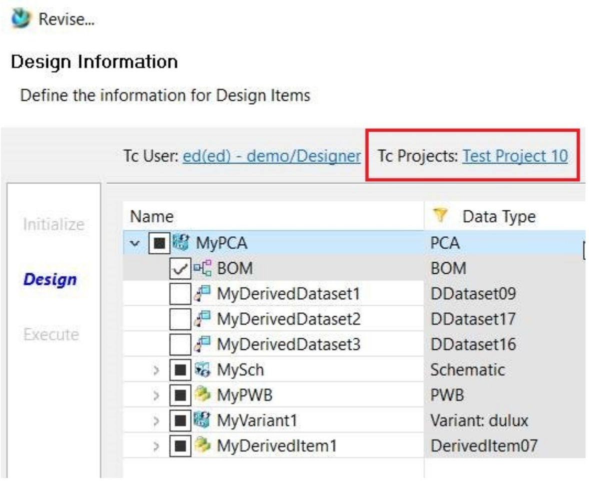
2. In the ECAD design tool, navigate to **Teamcenter→Save** and if prompted, log on.

   Alternatively, select **Teamcenter→Revise**.

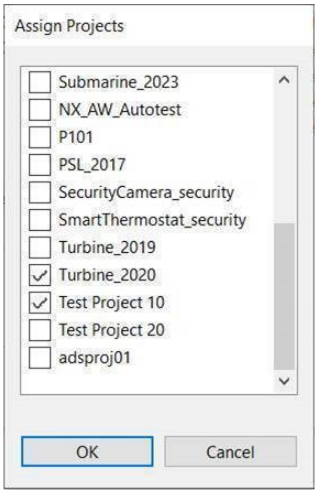
   Teamcenter displays the **Design Information** wizard dialog box.

   > Note:
   >
   > If you are using a different ECAD tool, refer to the *Access Teamcenter menu from your ECAD tool* topic for the initial steps to access the Teamcenter menu commands specific to your ECAD tool to perform the ECAD operation.

3. On the **Design Information** dialog box, click the **Assign Project** link to assign a Teamcenter project to the design. If a default project is assigned to a user, the default project name is displayed instead of the **Assign Project** link.

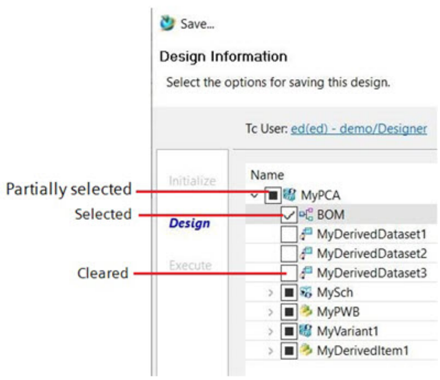


4. From the **Assign Projects** dialog box, select one or more projects that you want to assign to the design and click **OK**. All projects assigned to the user in Teamcenter will be displayed for selection.



5. Enter the information for design items as follows:

   a. Select the check box next to the items you want to save. The **check box** has three states: **selected**, **cleared**, and **partially selected**.

   - The **selected** check box indicates that the item and all its child nodes are selected for the update.
   - The **cleared** check box indicates that neither the item nor any child node is selected for the update.
   - The **partially selected** check box indicates that the item is selected but at least one of its child nodes is not selected.

   b. Double-click the relevant cell in the **Select Option** column and choose **Save** to save the changes.

   OR

Choose **Revise** in the **Select Option** column to save a new revision of a released item. If an item is in the released state, then it is shown with a lock icon. A released item can be revised but it cannot be saved.

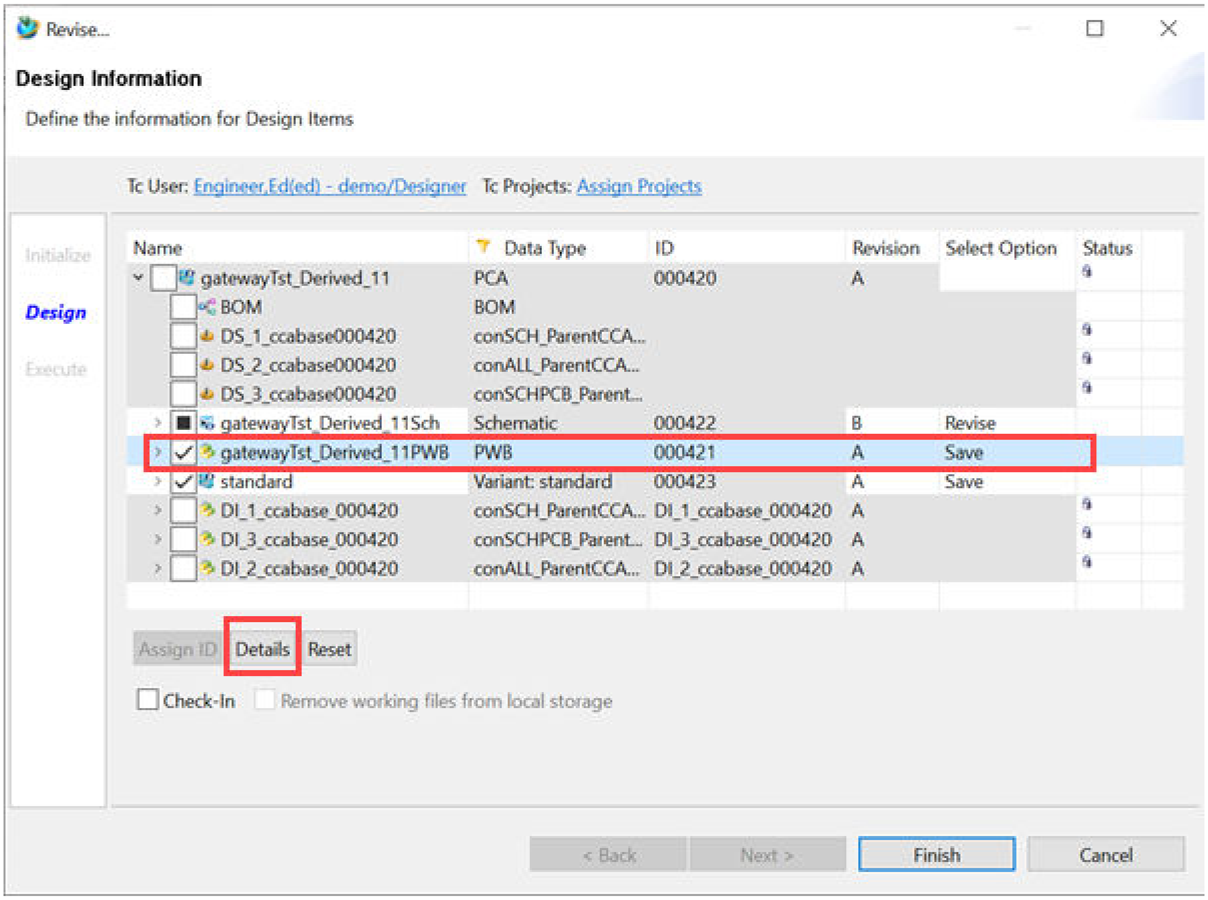
c. Select the **BOM** check box to save a bill of materials (BOM) with the design item. Double-click within the **Select Option** column to specify whether to save a BOM from the PCB or from the schematic or to not save a BOM.

d. Select the **Viewable** check box to save the neutral SCH (XSCH) file with the design.

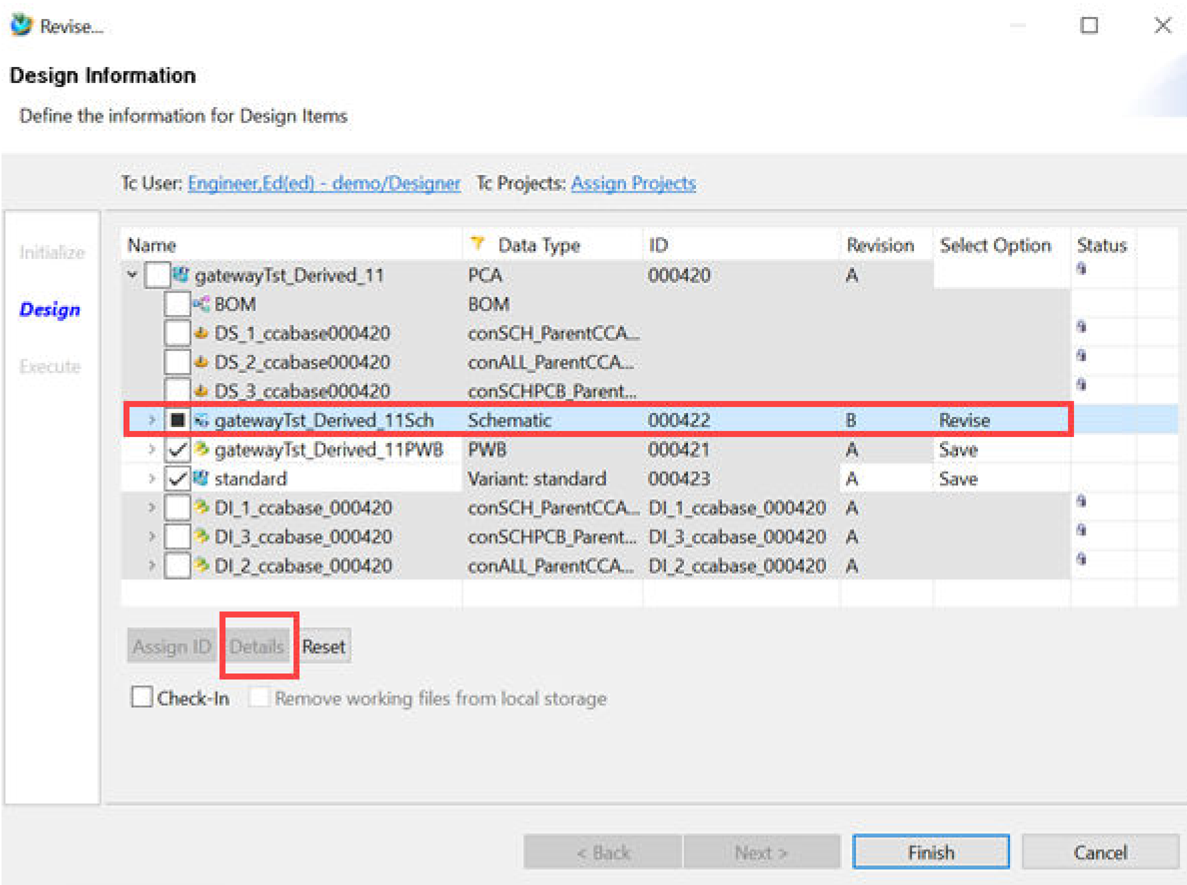
e. To save variants, specify all the required details the same way.

6. (Optional) Click **Assign ID** to generate IDs for all the items that have a blank ID field and are selected for saving.

7. (Optional) Click **Details** to view or enter additional information for the items. **Details** tab is enabled only if you are saving an item.



If you are revising an item, the **Details** tab is disabled. You cannot modify component or attribute values while revising any item.



8. To revert the changes to the design information tree, click **Reset**. All fields are restored to their initial rendering state. Any changes made to the **Details** dialog box are also reverted.

   When a row is selected, the reset action reverts all changes made to the selected row.

   When none of the rows is selected, the reset action reverts all the changes made to the design information tree.

9. (Optional) Select **Check-In** to check in the design items when the design is saved.

10. (Optional) Select **Remove working files from local storage** to delete the associated files from your workstation after the design is checked in. This check box is unavailable if you do not select the **Check-In** check box.

11. Click **Finish** to save the design or **Next** to proceed to the BOM compare results.

    If you had earlier selected the option to **Check-In** your designs, your design is saved and checked in to Teamcenter. The save operation may take several minutes.

    To view the BOM compare option, you must configure the **EDA_BOMCompareOnSave** preference. If not configured, the BOM compare option may not be available on your **Save** or **Revise** wizard.

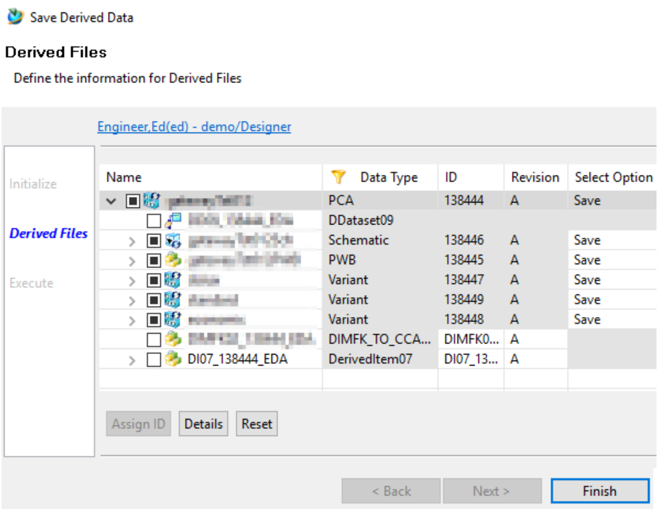
Save derived data

1. In the ECAD design tool, open a design that has been previously saved to Teamcenter.

2. Choose **Teamcenter→Save Derived Data**.

   > Note:
   >
   > If you are using a different ECAD tool, refer to the *Access Teamcenter menu from your ECAD tool* topic for the initial steps to access the Teamcenter menu commands specific to your ECAD tool to perform the ECAD operation.

3. In the **Save Derived Data** dialog box, select the derived files that you want to update.



4. (Optional) Click **Details** to view or enter additional information for the items. Update the information for derived files as required. Not all fields are editable.

5. In the **Select Options** column, select **Save** to update the current revision. If you want to create a new revision, select **Revise**.

6. To revert the changes made to the design information tree, click **Reset**. All fields are restored to their initial rendering state. Any changes made to the **Details** dialog box are also reverted.

   When a row is selected, the reset action reverts all changes made to the selected row.

   When none of the rows is selected, the reset action reverts all the changes made to the design information tree.

7. Click **Finish**.

8. Verify that the new dataset files are saved with the design in Teamcenter by using the rich client to access Teamcenter.

In the Business Modeler IDE specification of a derived item, if the **Add to BOM** property is selected and if the ECAD tool supports a base BOM, the derived item is added to the **CCABaseBOM** object. If the ECAD tool does not support a base BOM, the derived item is added to each **CCAVariantBOM** object.

For more information about working with derived data, see *Business Modeler IDE*.

Save derived data with a revised parent object

In some cases, saving the derived data may not be possible with the existing parent object. For example, if the parent object is in the **Released** state, you may need to save the derived data with a revised parent object.

1. In the ECAD design tool, open a design that has been previously saved to Teamcenter.

2. Choose **Teamcenter→Save Derived Data**.

   > Note:
   >
   > If you are using a different ECAD tool, refer to the *Access Teamcenter menu from your ECAD tool* topic for the initial steps to access the Teamcenter menu commands specific to your ECAD tool to perform the ECAD operation.

3. In the **Save Derived Data** dialog box, select the **Derived Files** section.

4. To save the derived data with a revised parent object, select the parent object in the top row and choose **Revise** in the **Select Option** list.

5. Click **Finish**.

Update only BOM data without changing any design dataset files

1. In the ECAD design tool, open a design for modification and make the required changes to the BOM information.

2. In the ECAD design tool, **navigate** to **Teamcenter→Save As** and, if prompted, **log on**.

   Teamcenter displays the **Design Information** wizard dialog box.

> Note:
>
> If you are using a different ECAD tool, refer to the *Access Teamcenter menu from your ECAD tool* topic for the initial steps to access the Teamcenter menu commands specific to your ECAD tool to perform the ECAD operation.

3. Select the **BOM** check box to save a bill of materials (BOM) with the design item. Double-click within the **Select Option** column to specify whether to save a BOM from the PCB or from the schematic or to not save a BOM.

4. Double-click a specific field to update the name and ID for any editable item manually. Not all fields are editable.

5. Click **Finish** to save the updated BOM.

Filter items in the design information display

The **Filter Data Type** dialog box narrows down the items that are displayed in the **Design Information** view based on the data type. You can apply a filter to display only the data types you are interested in.

- **Select All** displays all data types.
- **Derived Files** displays all derived files irrespective of the status of other check boxes.
- **Derived Items** displays all derived items irrespective of other check boxes.

The following examples explain how filtering works:

- If both check boxes, **PCA** and **Derived Items**, are selected, all derived items and all PCA items are displayed.
- If the **PCA** check box is clear but the **Derived Items** check box is selected, all derived items including the PCA derived items are displayed.
- If the **PCA** check box is selected and the **Derived Items** check box is clear, all PCA items will be displayed but no derived items are displayed.

The data type filter only controls the design information that is displayed on the screen. It does not affect the selection state of the design item, irrespective of whether the design item is displayed or hidden.

Apply filters to a data type

1. Click **Filter** on the **Data Type** column.

2. In the **Filter Data Type** dialog box, select the data type you want to display.

3. Click **OK**.

Remove filters

1. Click **Filter** on the **Data Type** column.

2. On the **Filter Data Type** dialog box, clear the selected check boxes.

Convert a nonvariant design to a variant design

When an ECAD designer modifies an existing nonvariant ECAD design from Teamcenter, adds the first variants, and saves it back into Teamcenter as a variant design, the **Save As** wizard appears with the **PCABase** fields enabled. This allows the system to collect the necessary information to convert the Teamcenter design from the single PCA representation to a multiple PCA representation. The same procedure applies whether you save or revise the nonvariant design.

1. In the ECAD design tool, **navigate** to **Teamcenter→Save As** or **Teamcenter→Revise** and, if prompted, **log on**.

   > Note:
   >
   > If you are using a different ECAD tool, refer to the *Access Teamcenter menu from your ECAD tool* topic for the initial steps to access the Teamcenter menu commands specific to your ECAD tool to perform the ECAD operation.

2. In the **Design Information** dialog box, select the nonvariant design that you want to convert to a variant design.

3. Click **Assign PCA**.

4. Click **Assign ID**.

5. Click **Finish**.

The design is saved and checked in to Teamcenter if you selected **Check-In**. A **Save** progress indicator appears, showing that a save operation is in progress. It may take several minutes for a save operation to complete.

The process for saving Mentor Xpedition EDM designs

The following process describes how to save designs from Xpedition EDM Design to Teamcenter.

Before you can save designs to Teamcenter, the administrator must do the following in EDM Design Client:

Prerequisites

- **Create EDM configuration rules for Teamcenter**

  Configuration rules define what data is allowed in the baseline and defines how the data is stored in EDM. For example, you must define EDM configuration rules for saving different types of data such as BOM, EDIF, ODB++, and derived data.

  You should use the configuration rules provided by Teamcenter.

- **Create EDM Baseline Profiles**

  Baseline Profiles allow you to define a group of files to be processed together in EDM.

Saving designs

1. Create or modify a schematic or a PCB design and save it in an EDM project.
2. Using the EDM Design client, create a project baseline that contains the design and the required (additional) files.
3. Using EDM Design client, export the baseline as an EDX package.
4. Move the exported EDX package to the EDA staging location design directory for the design.
5. Using the Teamcenter Gateway for EDA, select the Mentor EDX Design configuration, and save a new design or save, check in, or revise existing designs in Teamcenter.

Verifying if the design is successfully saved

- A new PCA item is created with an item ID matching the one for the EDM project name or the ID you specified.
- A new PWB item related to the PCA revision is created with the **EDAHasPWB** relation.
- A new design dataset is created and related to the PCA revision with an **IiMAN_specification** relation.
- The EDX package file is attached to the dataset.

The ECAD designer must manage the design check out locks in both EDM and Teamcenter. When a design status is Work in Process (WIP) in EDM, the ECAD designer should not check in the design to Teamcenter. The ECAD designer must provide the necessary BOM output files that Teamcenter requires.

Variant designs require BOM files. These files must be generated with Mentor Partlister by using the configuration file provided in the *TCEDAECAD_ROOT\MentorEDXDesCon* directory. Generating the EDIF and ODB++ files is optional.

> Note:
>
> Design attribute mapping and the *Save As Copy* function of designs that already exist in Teamcenter are not supported.

Save a Mentor Xpedition EDM board design to Teamcenter

1. In the ECAD client, select an EDM design baseline to be saved to Teamcenter.

2. Click **Create EDX...** and enter the required information in the **Create EDX...** dialog box.

3. If prompted, log on to Teamcenter.

   Teamcenter displays the **Save As...** dialog box.

4. Enter an ID, a revision, and a name for the PCA item and the PWB item.

   Alternatively, you can click **Assign All**. Teamcenter generates values for both PCA and PWB.

5. The EDX package is saved and checked in to Teamcenter.

Allow or restrict check in while saving a new file

On the **Save As** wizard, the **Check-in** check box is enabled or disabled based on the **EDA_ForceCheckInOptionDefault** preference value.

Set the **EDA_ForceCheckInOptionDefault** preference to **True** to disable the **Check-in** check box on the **Save As** wizard. Set the preference to **False** to enable the **Check-in** on the **Save As** wizard. Once the check box is enabled, users can change the selection.

The default selection of **Check-in** check box is set using the **EDA_CheckInOptionDefault** preference.

Allow or restrict saving a BOM with the design item

You can select **BOM** check box to indicate whether to save a BOM with the design.

Initially, the check box may be disabled for your site. The availability and default state of the **BOM** check box depends on the applicable preference setting. To control the **BOM** check box behavior during the save operation, set the preferences as following.

- Set the **EDA_SchematicBOMOptionDefault** preference for schematic designs.

- Set the **EDA_PcbBOMOptionDefault** preference for PCB designs.

- Set the **EDA_CombinedBOMOptionDefault** preference for combined model designs.

> Note:
>
> EDA does not support Teamcenter substitute parts specified for ECAD components in a Teamcenter BOM view revision (BVR).
>
> When one or more substitute parts for ECAD component instances are encountered during the EDA update of a Teamcenter BVR as part of a save operation, the condition is reported to the designer and update of that BVR is skipped. The BVR will not be in sync with the ECAD BOM until the substitute parts are removed from the Teamcenter BVR and the design is saved to Teamcenter again.
>
> You can use the **BOM Compare** feature to identify the substitute parts for the ECAD component. You can identify the substitute parts in the **BOM Compare** report by checking any results that contain **PART NUMBERS DIFFER**.

Allow or restrict saving the neutral SCH (XSCH) file with the design item

You can select **Viewable** check box to indicate whether to save the neutral SCH (XSCH) file with the design item.

Initially, the check box may be disabled for your site. The availability and default state of the **Viewable** check box depends on the applicable preference setting. To control the **Viewable** check box behavior during the save operation, set the preferences as following.

- Set the **EDA_SchematicViewableOptionDefault** preference for schematic designs.
- Set the **EDA_PcbViewableOptionDefault** preference for PCB designs.
- Set the **EDA_CombinedViewableOptionDefault** preference for combined model designs.

8. Working with the cache and staging directories

Working with the staging and cache directories

The EDA staging directory holds designs for check in and check out with Teamcenter, and the cache directory is where temporary data is stored.

The cache and staging directories are created automatically when you select a command on the **Teamcenter** menu and log on for the first time. If a user wants the cache directory, or staging directory to be changed, the administrator can do so by editing the **TCEDAClient.properties** file. However, Siemens Digital Industries Software does not recommend this. This can be a one-time change that should be undertaken only if absolutely necessary.

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

Refresh a design

The cache and staging directories temporarily hold data as designs move between your design tool and Teamcenter. Occasionally, you may need to refresh a design to synchronize the local copy of the design with the latest in Teamcenter or clear the cache to remove all local copies of designs that you are not working with.

Refreshing a design updates the staging directory with the latest version of the currently opened design. No refresh is done if the design is checked out for modification and has the Teamcenter check out lock applied.

1. In the ECAD design tool, open a design that has already been checked in.

2. In the ECAD design tool, choose **Teamcenter→Refresh**.

3. If the design is the latest version available, a dialog box appears asking if you want to force a refresh.

   However, if there is a newer design found on Teamcenter, you are notified that the older design is out-of-date and must be refreshed.

4. Click **OK** in the notification dialog box.

   If the design was out-of-date, the newer design replaces it in the cache and staging directories.

Clear the cache

Use the **Purge Working Files** command to clear the client cache of non checked-outch designs of the current configuration type and remove copies of these designs from the client staging directory. The designs that are checked out and/or have related datasets checked out are not affected.

> Caution:
>
> Do not make manual changes to the cache. If you manually delete the cache without using the **Purge Working Files** command, you are blocked from making any further changes to designs.

1. In the ECAD design tool, choose **Teamcenter→Purge Working Files**.

   The **Purge Working Files** dialog box is displayed.

2. Choose the designs you want to purge by selecting the check boxes in the **Purge** column.

   You can also click the **Select All** button to select all designs or click the **Deselect All** button to clear all designs.

3. Click **OK**.

   All data is removed from the cache and staging directories except for those designs with check out locks and those designs that have related datasets checked out.

> Caution:
>
> Related data that has never been saved to Teamcenter is deleted if the primary data is checked in.

9. Compare BOMs

Comparing ECAD design BOMs with Teamcenter

The ECAD design component compare feature compares the components in an ECAD design with the components of the same design in Teamcenter. Changes could have occurred in the Teamcenter BOM to account for component cost, component availability, manufacturing issues, and so on, which are not reflected in the previously saved ECAD design.

This feature displays part number differences in component instances between the ECAD design BOM and a Teamcenter design revision BOM. After you check out an ECAD design, as a first step, you can compare differences between the previously saved ECAD design BOM and the corresponding current Teamcenter BOM.

Component instances, each identified by a reference designator, are compared to determine if the ECAD design part numbers match the corresponding Teamcenter product structure occurrence item IDs. This helps you resolve any differences and perform the necessary component instance additions, removals, or substitutions.

You can compare single PCA designs and multiple PCA designs. An ECAD design BOM can also be compared with another unrelated Teamcenter item revision BOM.

View BOM compare results when saving an updated design

The ECAD user can set the **EDA_BOMCompareOnSave** preference to view the BOM compare results when saving an updated design to Teamcenter.

The valid values for the **EDA_BOMCompareOnSave** preference are:

- Always: The BOM Compare page will always be shown. This is the default value.
- DiffOnly: The BOM Compare page will be shown only if there are differences.
- Never: The BOM Compare option will not be shown with **Save** or **Revise** wizards.

The **EDA_BOMCompareOnSave** preference syntax is as follows:

```
<preference name="EDA_BOMCompareOnSave" type="String" array="false"
disabled="false" protectionScope="User" envEnabled="false">
<preference_description>Controls whether the BOM Compare Result page
 is shown on the EDA save dialog during save operations.
</preference_description>
<context name="Teamcenter">
<value>ALWAYS</value>
```

```
</context>
</preference>
```

1. In the ECAD design tool, open a design for modification and make the desired changes to the design.

2. In the ECAD design tool, choose **Teamcenter→Revise** or **Teamcenter→Save** and, if prompted, **log on**.

   Teamcenter displays the **Design Information** wizard dialog box.

   > Note:
   >
   > If you are using a different ECAD tool, refer to the *Access Teamcenter menu from your ECAD tool* topic for the initial steps to access the Teamcenter menu commands specific to your ECAD tool to perform the ECAD operation.

3. Update the information for design items as required.

4. Click **Next** to proceed to the **ECAD Design Component BOM Compare Results**. The comparison results are displayed based on the **EDA_BOMCompareOnSave** preference. Check the **Result** column to see the BOM comparison results.

5. Click **Back** to go to the **Design Information**.

6. Click **Finish** to save the design.

Compare ECAD design and Teamcenter BOM component instances

1. Check out the design you want to work on. This ECAD design becomes the source for the comparison.

2. In the ECAD design tool or Teamcenter Gateway for EDA, choose **Teamcenter→BOM Compare**.

   If you are working on a single PCA design, the specified ECAD design BOMs are generated, the Teamcenter item revision BOM is retrieved, compared, and the comparison results are displayed.

   If you are comparing a multiple PCA design with two or more variants and/or the ECAD design BOM source must be specified for a combined model design, the **ECAD Design Component Compare** dialog box is displayed.

Note:

If you are using a different ECAD tool, refer to the *Access Teamcenter menu from your ECAD tool* topic for the initial steps to access the Teamcenter menu commands specific to your ECAD tool to perform the ECAD operation.

3. Specify the comparison criteria.

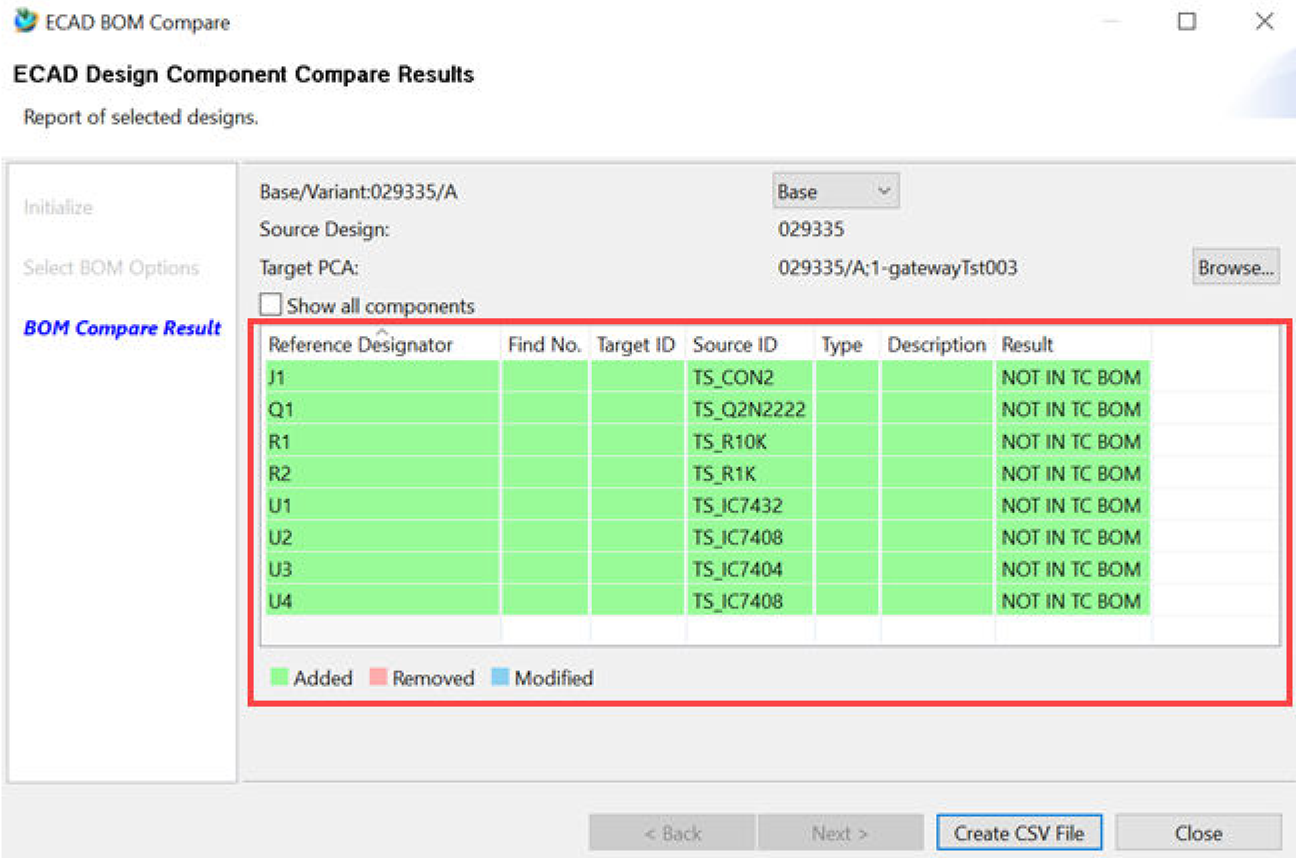
   If you are comparing a multiple PCA design with two or more variants, you must select the variants using the **Base/Variant** list.

   If you are comparing a combined model design, you must specify the BOM option as schematic or PCB.

4. Click **OK**.

   The specified ECAD design BOMs are generated, the Teamcenter item revision BOM is retrieved, compared, and the comparison results are displayed. The Teamcenter item revision is the target of the comparison. The BOM Compare results for selected components are displayed in following colors for different scenarios:

Operation	Color code	Result
No change	No color	MATCH
Added Part	Green	NOT IN TC BOM
Deleted Part	Red	NOT IN DESIGN BOM
Updated Part	Blue	CHANGES REF DES
Updated Part number	Blue	PART NUMBERS DIFFERS



Note:

By default, only the component instances with differences are displayed. You can choose to display all results by selecting the **Show all components** check box.

5. Click **Browse** to select a different Teamcenter target item revision for comparison.

6. Click **Create CSV File** if you want to generate a comma-separated value (CSV) file containing the comparison results.

   You can share this CSV file with others to aid in reconciliation.

Note:

The file is created in the selected location and follows this file naming convention:

BOMCompare_*target item_id target item_revision_id***.csv**

The header line of the CSV file replaces the fixed target ID and source ID headings used in the results tables with the actual target PCA item ID or revision ID and the source design item ID, respectively.

7. Click **Close** to exit the **ECAD Design Component Compare** dialog box, after you have finished viewing all the specified comparison results.

10. Access ECAD designs in Active Workspace

Open a design in Active Workspace

To search for a design dataset file in Teamcenter and open it for reference (read-only) or modification:

1. In the ECAD design tool, choose **Teamcenter→Open**.

   If prompted, **log on**.

   Active Workspace opens the design. If no design is selected or design is not found, the home page is displayed. You must have Active Workspace installed and configured for the **Open** command to work.

   > Note:
   >
   > If you are using a different ECAD tool, refer to the *Access Teamcenter menu from your ECAD tool* topic for the initial steps to access the Teamcenter menu commands specific to your ECAD tool to perform the ECAD operation.

2. In the **Open Design** dialog box, browse to the design you want to open.

   You can also search for designs using the search type and search criteria.

3. To open the design dataset for modification, click **Open For Modification**.

   The file is checked out and no one else can modify it.

   To open a read-only copy, click **Open For Reference**. A design opened only for viewing cannot be modified or revised.

   > Note:
   >
   > If the dataset is already checked out to you or someone else, the **Open for modification** button is disabled and the **CO** (check out) column in Teamcenter displays a **Y**.

4. (Optional) To open related design files, select the **Open Related Design** check box.

   The choices in the list are **Blank** (do not download), **Reference** (download only for reference), and **Check-out** (download and check out).

Navigate to a new design

(Optional) When opening a design for the first time, you can navigate to the Active Workspace home page and then search for the design.

1. In the ECAD design tool, choose **Teamcenter→Design Info**.

   When no design is selected or a design is not found, the home page is displayed.

2. On the home page, browse to the design you want to open. You can also search for designs using the search type and search criteria.

View key design metrics in Active Workspace

When a design is loaded in Active Workspace, the associated design metrics can be viewed in the **Overview** tab. These metrics are read-only and cannot be edited. You can use these to measure design quality and design progress.

> Note:
>
> Viewing key design metrics in Active Workspace is currently not supported for Xpedition Direct and EDM.

Some examples of the design metrics are:

- Component placement
- Connectivity
- Routing
- Online hazards
- Constraints

Open a design in EDM Collaborate from Active Workspace

When you access a PCA EDM board design from Active Workspace, you are provided an option to open the design in the EDM Collaborate tool. To enable the launch of this tool from Active Workspace, the hosting URL for EDM server must be specified using the **EDA_EDMHosting.URL** preference.

In Active Workspace, choose **Open→Open in EDM**. The EDM Collaborate tool is launched in a new tab, and the design is opened in this tool.

View the requirements linked to a circuit card assembly revision

The requirements of your product are stored in Teamcenter. A circuit card assembly or a schematic revision can be linked to the product design requirements using a tracelink. The links may be a direct link, an indirect link, or a mix of both.

1. Select the circuit card assembly in Active Workspace.

2. Click the **Requirement** tab. The linked requirements are displayed. If the requirements are not displayed, check whether the tracelink is created between the product design requirements and the Circuit Card Assembly.

   The **Requirement** tab is not available in the occurrence management sublocation within the **Content** tab.

3. Select a linked requirement to view its description.

View the schematic and PCB designs in Active Workspace

You can view the schematic and PCB designs in Active Workspace. The **Viewer** tab must be enabled in Active Workspace and the hosting URL for EDM server must be specified using the **EDA_EDMHosting.URL** preference.

The **Viewer** tab is displayed only if the design is saved using EDM Design Cockpit and has the necessary information to be viewed in Active Workspace.

If the PCB design exists, it is displayed by default.

> Note:
>
> Viewing the schematic and PCB designs in Active Workspace is supported for Xpedition EDM designs only from with the EDM Collaborate embedded Active Workspace user interface.

1. Search for the ECAD design you want to view, and open the design from the list of results.

2. Click **Viewer** tab to see more information.

   Active Workspace loads data from the EDM database. By default, the PCB design view is displayed.

3. Use the icons in the ECAD tool within the **Viewer** to switch between PCB and Schematic views.

Create a new revision within Active Workspace

You can use the **Save As** or **Revise** commands to create new revisions of EDA objects in Active Workspace.

Create a new revision of an existing design object

1. Navigate to the design item for which you want to create a new revision.

2. Select the design and click **Revise**.

3. Click the **Revision** tab.

4. Enter the properties for the new revision and click **Revise**.

   A new revision is created with the same ID. You cannot enter a different ID when you create a new revision of an existing object.

Create a new object from an existing object

1. Navigate to the design item from which you want to create a new object.

2. Select the design and click **Save As**.

3. Click the **New** tab.

4. Enter the properties for the new object and click **Create**.

Submit a design to a workflow in Active Workspace

1. Select the design that you want to submit to the new workflow and choose **Teamcenter→Design Info**.

   Active Workspace opens the design.

   If no design is selected or design is not found, the home page is displayed. You must navigate to the design in Active Workspace.

   > Note:
   >
   > Refer to the *Access Teamcenter menu from your ECAD tool* topic for the initial steps to access the Teamcenter menu commands specific to your ECAD tool to perform the ECAD operation.

2. Click **Submit to Workflow**.

3. Accept the default workflow name or type your own.

4. From the **Workflow Template** list, select a template for the new workflow.

If a default workflow exists for the content type, it is automatically specified as the workflow template.

5. Enter a description for the new workflow.

6. Click **Submit**.

7. Click the **Workflow** tab to view the workflow status of the design. In Active Workspace, the **Workflow** tab is visible only for the design items that are part of a workflow process.

11. Collaborating across domains

Manage ECAD design collaboration files

Overview of collaboration across domains

Any electrical design, for example, a PCB, is collaboration between two domains, namely, electrical and mechanical.

A typical PCB design is as follows:

- A mechanical engineer creates a board outline and places several components that can have an impact on the mechanical geometry, such as connectors, LEDs, or potentiometers.

- The electrical engineer places the electrical components on the PCB.

- If there are changes in the board components, the engineers change the components from their domain. For example, the mechanical engineer can change a component in case the design has changed, or the electrical engineer can change a component due to an update in the schematic.

- The placement locations of components are changed when there are issues with the placement. For instance, when there are collisions in the housing or there are zones with thermal, vibration or moisture issues.

EDA allows engineers from different domains to work on the same BOM and collaborate on the design creation.

The collaboration is accomplished using the Teamcenter Issue Management functionality. Active Workspace is used to create collaboration objects. A simplified collaboration process works as follows:

- When there are issues that need to be addressed, for example, a proposal to change the board components, an engineer creates a *collaboration issue*.

  Based on the organization's issue management process, the issue is assigned to participants or to the relevant stakeholders.

- To share design files with participants from the other domain, the engineer publishes the design file. The mechanical engineer then receives a notification.

  An IDF or IDX file is attached to the collaboration issue. IDF and IDX files are data neutral formats that can be read by most ECAD and MCAD tools. Use of IDX files is recommended.

- The engineer from the other domain then downloads the design files, updates it, and publishes the changes.

- When the design is ready for review or release, the engineers can create a collaboration issue to review or release the design.

> Note:
>
> The collaboration functionality is only available for PCB designs.
>
> In the embedded mode, this capability is available to all connectors. In Gateway mode, library configuration do not support Collaboration or Issue Manager action. In non-library configuration, some are setup to not support Collaboration but all support Issue Manager actions.

Initiate a design or a part workflow

1. Select the design you want to submit to the new workflow and choose **Teamcenter→Design Workflow**.

   Alternatively, to initiate a part workflow, select the design you want to submit to the new workflow and choose **Teamcenter→Part Workflow**.

2. In the **Initiate Design Workflow** dialog, select the design you want to submit to the new workflow, and click **Submit to Workflow**.

3. Accept the default workflow name or type your own.

4. From the **Workflow Template** list, select a template for the new workflow.

   If a default workflow exists for the content type, it is automatically specified as the workflow template.

5. Enter a description for the new workflow.

6. Click **Submit**.

Create a collaboration issue

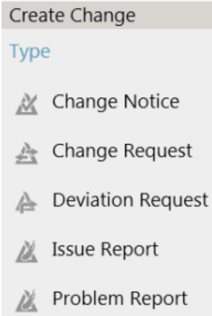
The Issue Management functionality in Active Workspace is used to manage collaboration issues. A collaboration issue can only be an issue report object.

1. Select or open the design for which you want to create a collaboration issue and choose **Teamcenter→Design info**.

   The Active Workspace opens the design If it is saved in Teamcenter. If not, you must navigate to the design in the Active Workspace.

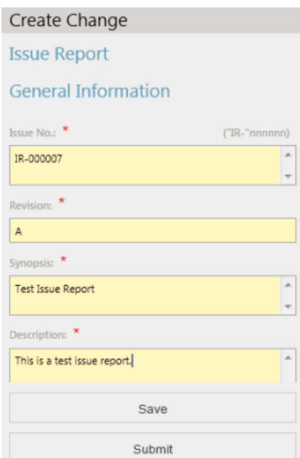
2. In Active Workspace, click **Create Change**.

3. Select the type of change you want to create.



4. Enter the properties for the object.

   For example, specify the item ID, revision, synopsis, and description of the collaboration issue.



5. (Optional) To add supporting objects and reference documents, to the right of **Attachments**, click .

6. Perform one of the following:

   - If you want to continue editing later, click **Save**.
   - If you want to send it for resolution now, click **Submit**.

     The collaboration issue is sent through the default workflow for resolution. Check your Inbox for any tasks. Depending on how your organization's change process is configured, reviewers may be automatically assigned. If a workflow is not configured for issues, you can manually create one in Active Workspace.

Publish collaboration files

Active Workspace is used to publish IDF or IDX files (for collaboration) to Teamcenter.

1. Select or open the design for which you want to publish a collaboration file and choose **Teamcenter→Design info**.

   The Active Workspace opens the design If it is saved in Teamcenter. If not, you must navigate to the design in the Active Workspace.

2. In the Active Workspace, click the **Attachments** tab and to the right of **Files**, click .

3. Browse to the file that you want to publish, and specify the name, description, file type, and relation type for the file that you want to publish.

   We recommend choosing the *EDAIdf* dataset to upload the IDF or IDX file.

4. Click **Add**.

Download collaboration files

Active Workspace is used to download the collaboration files from Teamcenter.

1. Choose **Teamcenter→Design info**.

   The Active Workspace opens the design If it is saved in Teamcenter. If not, you must navigate to the design in the Active Workspace.

2. Click the **Attachments tab.**

3. Select the file you want to download and click .

Create issues

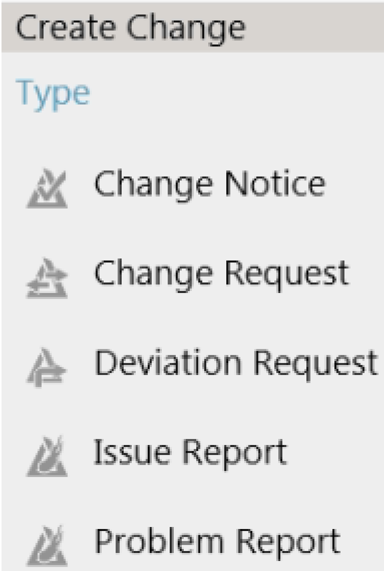
The Issue Management functionality in the Active Workspace is used to create and manage issues.

You can use Issue Manager for schematic and PCB designs.

1. Select a design for which you want to create a collaboration issue and choose **Teamcenter→Design Info**.

2. In the Active Workspace, click **Create Change** .

3. Select the type of issue you want to create.



4. Enter the properties for the object.

   For example, specify the item ID, revision, synopsis, and description of the collaboration issue.

5. (Optional) To add supporting objects and reference documents, to the right of **Attachments**, click .

6. Do one of the following:

   - If you want to continue editing later, click **Save**.
   - If you want to send the issue for resolution instantly, click **Submit**.

     The issue is sent through the default workflow for resolution. Check your Inbox for any tasks. Depending on how your organization's change process is configured, reviewers may also be automatically assigned. If the workflow is not configured for issues, you can manually create a workflow in the Active Workspace.

Manage IDX exchange through Teamcenter Co-design

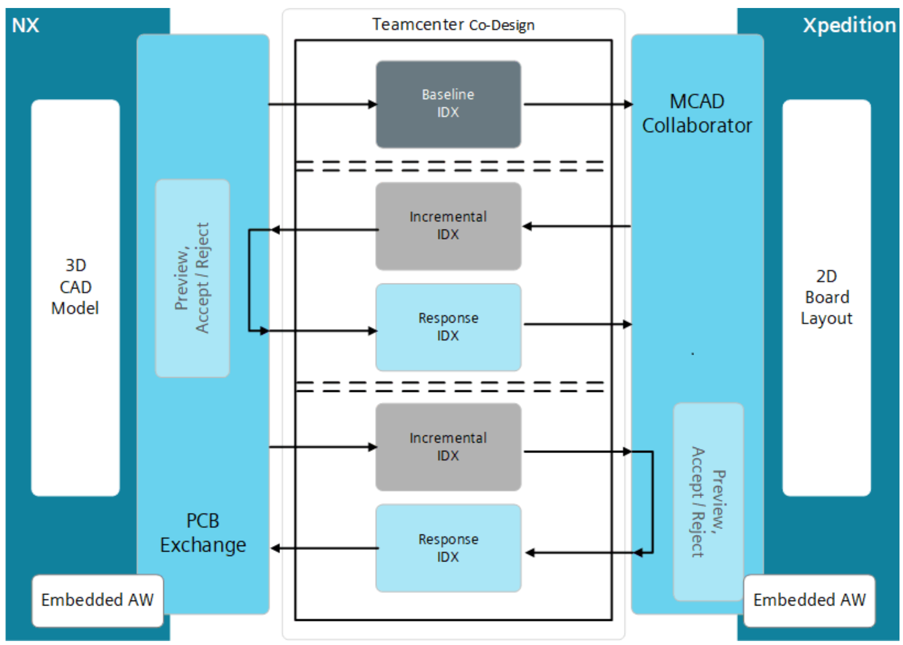
Overview of IDX exchange through Teamcenter Co-design

ECAD and MCAD designers collaborating on the electromechanical products must work in an integrated design development environment that allows close collaboration between the ECAD and MCAD tools during the design development process.

Teamcenter EDA supports this ECAD and MCAD collaboration between Xpedition and NX using the IDX format to easily exchange electromechanical data. The collaboration allows you, as a design engineer, to communicate incremental design changes throughout the design process. The IDX file captures and tracks the proposed incremental design changes initiated by the designers in the ECAD-MCAD collaboration.

ECAD MCAD collaboration using IDX file format requires:

- An IDX baseline file that contains a base PCB reference design for further incremental changes. Both ECAD and MCAD designers must start from the same IDX baseline to synchronize the design process.
- IDX incremental files that contain incremental changes from the last synchronization.
- IDX response files that contain information about the incremental changes incorporated or rejected by the designers.



The incremental exchange of data between ECAD and MCAD designers is done multiple times until the design is ready for use in manufacturing.

Exchange design change proposals and 3D model data using Co-Design

Co-Design in Teamcenter provides a collaborative environment for the ECAD and MCAD designers to create and respond to the design change proposals effectively and accurately. You, as an ECAD designer, can associate collaborative designs, participants working on those designs, and the incremental design changes of the PCB design to a single Co-Design object to ensure synchronization between the electrical and the mechanical board design projects. The design changes may include adding, removing, modifying, and relocating components or editing the board outline.

Co-Design facilitates the exchange of product assembly 3D models between NX and Xpedition using Teamcenter services. The MCAD designers can export the 3D models and assembly files from NX to Co-Design in Teamcenter. The ECAD designer can import these assembly files from Co-Design to Xpedition. In addition, and ECAD designer can also initiate an exchange of 3D files from Xpedition to NX. This happens when new components are added to the PCB design in Xpedition and the 3D representation of those components is not available in NX.

Create a new Co-Design for a design object

A project leader creates a Co-Design object for a project and adds ECAD and MCAD designs to it. The design engineers from the ECAD and MCAD domain can collaborate using this Co-Design object.

1. In Active Workspace, search for a design the ECAD-MCAD design engineers want to collaborate on.

2. Select the design and click **More Commands** ••• > **New** > **Create Co-Design**.

   Teamcenter creates a Co-Design for the selected design.

3. Click any of the below tabs as required to continue working with the Co-Design object.

   a. **Overview** – Shows the details about the object creation date, last modifying user, and so on.

   b. **Proposal** – Lists all ongoing and completed proposals related to the Co-Design.

   c. **Participants** – Lists the design owner, proposed reviewers, and other users responsible for the designs. This tab is configurable.

4. You can also discuss issues and exchange ideas while collaborating on the ECAD-MCAD designs.

After creating a Co-Design object for an ECAD or MCAD design, you can add other designs for collaboration to this **Co-Design** object.

Add designs to a Co-Design object

1. Search for and open the Co-Design object to which you want to add a design.

2. Select **More Commands** ••• > **New** > **Add Design**.

3. In the **Add** panel, enter keywords to **Search** for the required design, or select a design from the **CLIPBOARD**, **FAVORITES**, or **RECENT** list.

4. Select the design and click **Add**.

5. To remove a design from the Co-Design object, select the design and click **Edit** > **Remove Design**.

All the users and participants associated with the ECAD and MCAD designs are now listed under the **Participants** tab. You can also add more participants and users to the Co-Design.

Add participants to a Co-Design object

When a design is added to a Co-Design object, the owner of the design, the reviewers, and the users assigned as *responsible party* and associated with the designs are automatically added as participants. If you are the owner of a design, you can add other participants to the design.

To add participants to the list of **Approver** and **Proposed Responsible Party** for a design:

1. Select the design to which you want to add participants.
2. In the **Approver** or **Proposed Responsible Party** list, click the **Add** ⊕ button.
3. Search for and add participants as required.

To remove a participant, select the participant in the list and then click **Edit** > **Remove Participants**.

Discuss issues and exchange ideas for the designs

You can perform the following tasks to discuss issues and ideas for your ECAD designs with multiple users.

Create a new discussion

You can create a discussion on the design object in Active Workspace. However, if you create a discussion on one of the children of a product structure, your discussion is attached to the revision and not to the occurrence of the revision.

1. Select the design object you want to discuss and click **Discuss** on the toolbar to display the **Discuss** panel.
2. Click **New Discussion** ⊕ to begin a new discussion on the selected object.
3. (Optional) Click **Add Participant** ⊕ to add a participant to the discussion. Select the participant and click **Add**.
4. (Optional) Click the **Add Source** ⊕ to add an additional source object to the discussion.
5. (Optional) Click the **Private Message** box to limit the participants in the discussion. Click **Add Participant** to add a participant to the discussion. Only the selected participants are able to participate in the discussion. In addition, participants in a private message are automatically set to follow the discussion.
6. Compose the message.

From within the **Message** area, you can select:

- Bold
- Font name
- Text color
- Background color
- Insert image (GIF, JPEG, JPG, and PNG)

7. Click **Create** to create the discussion on the object.

   The new discussion appears on the **Discuss** panel. Each discussion has a timestamp, and new discussions appear at the top of the list. As the creator of the discussion, you are automatically set to follow all replies to the new discussion.

View and reply to a discussion

1. To view the discussion on an object, click **Discuss** on the toolbar to display the **Discuss** panel.

2. (Optional) If the discussion is a private message, you can hover over the people icon to determine the recipients of the discussion message.

3. You can **Follow** or **Unfollow** all of the discussions on the selected object or just the selected discussion, create a new discussion, or click either **Reply** or **Advanced** to reply to a discussion.

Follow a discussion

Follow allows you to receive notifications when new discussions or replies are made in relation to the source object. This is particularly useful if you want to see responses to an object you own.

> Note:
>
> You can receive notifications on *all discussions* or *a particular discussion*.

1. Select the object and click **Discuss** on the toolbar to display the **Discuss** panel.

2. To receive notifications:

   - On *all discussions* about the object, click **Follow** next to the name of the selected object.
   - On *a particular discussion*, click **Follow** within the selected replies.

   When you follow a discussion, you are notified of replies via the **Alerts** button on the global navigation toolbar.

Unfollow a discussion

To unfollow a particular discussion or all discussions on a source object at any time to stop receiving notifications, click **Unfollow** next to the selected discussion or discussions.

Delete a discussion

> Note:
>
> You can only delete a discussion if your administrator has given you the necessary privileges to perform this action.

1. Select the object and click **Discuss** on the toolbar to display the **Discuss** panel and its discussions.

2. Select the discussion you wish to delete and click **Delete**.

   A confirmation message appears. Click **Delete** to delete the selected discussion, including all replies.

Design change proposal process using Co-Design in Teamcenter

A high-level process to exchange design change proposals across domains using the **Co-Design** object along with the Teamcenter ECAD and MCAD published services is provided below:

Consider that the ECAD designer is initiating the design changes proposal and that the MCAD designer will review and then accept or reject the changes. However, these tasks can be initiated either by the ECAD designer or the MCAD designer.

1. The ECAD designer submits a design change proposal through the ECAD design tool Xpedition.

   - In the **Co-Design** > **Proposal** section in Active Workspace, a new proposal with the associated IDX file is added to the **Ongoing** proposals.

   - The task maturity of this proposal is set as **Submitted**.

   - A new **Review Proposal** task is assigned to the MCAD designer. The owner of the MCAD design receives a notification for this task in the **Inbox** of the embedded Active Workspace menu in the MCAD tool NX.

2. On receiving the notification about the task, the MCAD designer uses the Teamcenter services to download the proposal.

   - The **Review Proposal** task remains in the MCAD designer's Inbox in the embedded Active Workspace.

   - The task maturity is updated to **In Review**.

3. The MCAD designer reviews the changes in the design tool and accepts or rejects the changes as required. The MCAD designer uses the Teamcenter services to update this information in the **Co-Design**.

   - The task maturity is updated to **Reviewed**.
   - The **Review Proposal** task notification in the MCAD designer's **Inbox** is closed automatically.
   - A new **Process Response** task is assigned to the ECAD designer. A notification is generated in the **Inbox** of the embedded Active Workspace menu Xpedition.

4. On receiving the **Process Response** task notification in the **Inbox**, the ECAD designer uses Teamcenter services to download the reviewed information from Teamcenter. The rejected design changes are rolled back on the ECAD design and the designs in both domains are in sync.

   - The task maturity is updated to **Processed**.
   - In the **Co-Design**>**Proposal** section, the proposal is moved from the **Ongoing** list to the **Completed** list.
   - The **Process Response** task notification in the ECAD designer's **Inbox** is closed automatically.

12. Setting debug levels to identify issues arising during an EDA operation

Teamcenter log files assist you in troubleshooting your setup. Sometimes, you may want more information about the errors in your log files to trouble shoot an issue during an EDA operation. To gather this additional error information, you must get the TcEDA and SOA logs in debug mode. You can define the debug level for logs in the **log4j2.properties** file located in the *TC_ROOT\eda* directory.

To change the debug level, change the values of the **rootLogger.level** and **logger.soa.level** properties to the *DEBUG* level.

```
# All Rights Reserved.
# ================================================================================
# Set the Logging for the entire TcEDA Application.
# The Level can be DEBUG, INFO, WARN, ERROR, FATAL, or OFF
# There are four predefined Appenders. All are defined to start a new log file
# every day, renaming and keeping the last 7 logs.
#  - edaAppender
#  - theConsole
#  - soaAppender
#  - httpAppender
# ================================================================================

# ================================================================================
# The "edaAppender" is for all logging except SOA and HTTP.
# The default logging level is WARN, change it if desired.
# ================================================================================

rootLogger.level                                  = DEBUG
rootLogger.appenderRef.edaAppender.ref            = edaAppender

appender.edaAppender.name                         = edaAppender
appender.edaAppender.layout.type                  = PatternLayout
appender.edaAppender.layout.pattern               = %-5p %d{ISO 8601-1:2019} %m%n
appender.edaAppender.type                         = RollingFile

appender.edaAppender.append                       = true
appender.edaAppender.fileName                     = ${sys:java.io.tmpdir}/eda/${sys:user.name}_TcEDA.log
appender.edaAppender.filePattern                  = ${sys:java.io.tmpdir}/eda/${sys:user.name}_TcEDA-%d{yyyy-MM-dd}-%i.log.zip
appender.edaAppender.policies.time.interval       = 1
appender.edaAppender.policies.time.type           = TimeBasedTriggeringPolicy
appender.edaAppender.policies.type                = Policies
appender.edaAppender.strategy.max                 = 7
appender.edaAppender.strategy.type                = DefaultRolloverStrategy

# ================================================================================
# The "theConsole" writes log messages to the console.
# This is off by default, uncomment the first line if desired.
# ================================================================================

# rootLogger.appenderRef.theConsole.ref             = theConsole

appender.theConsole.name                          = theConsole
appender.theConsole.layout.type                   = PatternLayout
appender.theConsole.layout.pattern                = %-5p %d{ISO 8601-1:2019} %m%n
appender.theConsole.type                          = Console

# ================================================================================
# The "soaAppender" writes SOA log messages to a separate file.
# This is OFF by default, change level if desired.
# ================================================================================

logger.soa.name                                   = com.teamcenter.soa
logger.soa.level                                  = DEBUG
logger.soa.additivity                             = false
logger.soa.appenderRef.soaAppender.ref            = soaAppender
```

After changing the debug levels for these properties, you can perform the EDA operation that is failing. The detailed TcEDA and SOA logs will be generated at the *%Temp%\eda* folder.

You can share these logs with your Teamcenter administrator for further actions.