8. In the **Property Renderer Attachments** box, attach the *business-object*.**object_string**, in this case, **B4_RiskReqRevision.object_string**.

9. Create a compound property pointing to the **B4_RiskReqRevision.b4_RiskPriority** property called **RiskPriorityBOM** on the **Fnd0RequirementBOMLine** business object.

10. Save and deploy.

11. Perform the following in Teamcenter:

    a. Add the **B4_RiskReq** business object to the **TcAllowedChildTypes_RequirementSpec** preference.

    b. Create a requirement specification and add three **B4_RiskReq** requirements.

    c. Add the **RiskPriorityBOM** property to the requirement BOM view.

Teamcenter EDA

Configure Teamcenter EDA using the Business Modeler IDE

Teamcenter EDA integrates Teamcenter with electronic CAD applications that are used to design electronic components, such as circuit boards.

> Note:
>
> Before working with Teamcenter EDA objects in the Business Modeler IDE, you must **install the EDA features in Teamcenter Environment Manager (TEM)**. From the **Features** panel, choose **Extensions→Mechatronics Process Management→EDA for Business Modeler IDE**.
>
> You must also **install the EDA Server Support template (edaserver_template.zip file)** to your project.

Working with derived data

Use the **Extensions\EDA Derived Data** folder in the Business Modeler IDE to create derived data configurations used by the Teamcenter EDA application.

> Note:
>
> Before you can configure derived data in the Business Modeler IDE, you must install the **EDA for Business Modeler IDE** feature and the **EDA Server Support** feature to the server. You must also install the **EDA Server Support** template to the Business Modeler IDE.

Designers use ECAD (electronic CAD) applications to create electronic parts such as circuit boards. The managing of ECAD designs is known in the industry as *electronic design automation (EDA)*. The

Teamcenter EDA application integrates Teamcenter with ECAD applications such as Mentor Graphics and Cadence.

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.

Administrators can configure how the derived data is created in Teamcenter EDA by using the **EDA Derived Data** folder in the Business Modeler IDE and providing the name of the configuration in the **EDA_DerivedDataConfigDefault** preference. After configurations are created, users in Teamcenter EDA can create derived data by selecting the **Save Derived Data** menu command or by selecting the **Generate Derived Data** check box in the **Save As**, **Save**, or **Check In** dialog box. 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.

> Note:
>
> To see all the EDA data model, you must install the **EDA Server Support** template to the Business Modeler IDE.

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

- **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:

- **EDAHasDerivedDataset**
  Identifies the associated dataset as a derived dataset.

- **EDAHasDerivedItem**
  Identifies the associated item as a derived item.

Create an EDA derived data configuration

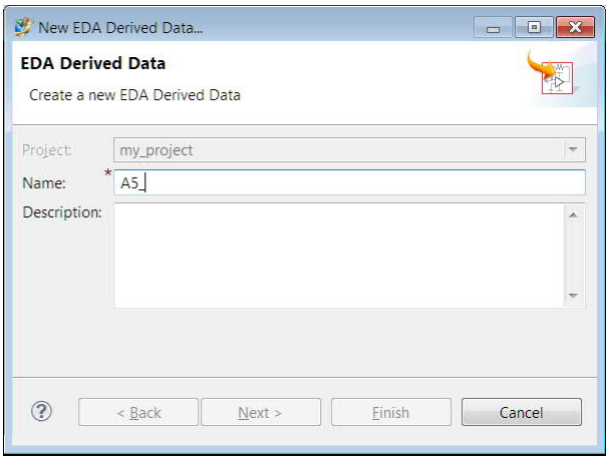
Administrators can configure how derived data is created in Teamcenter EDA by using the **EDA Derived Data** folder in the Business Modeler IDE. After configuration, an administrator provides the name of the configuration in the **EDA_DerivedDataConfigDefault** preference.

1. Set up the Business Modeler IDE.

   a. Ensure that you have installed the **EDA for Business Modeler IDE** feature and the **EDA Server Support** template to the Business Modeler IDE.

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

   The New EDA Derived Data wizard runs.



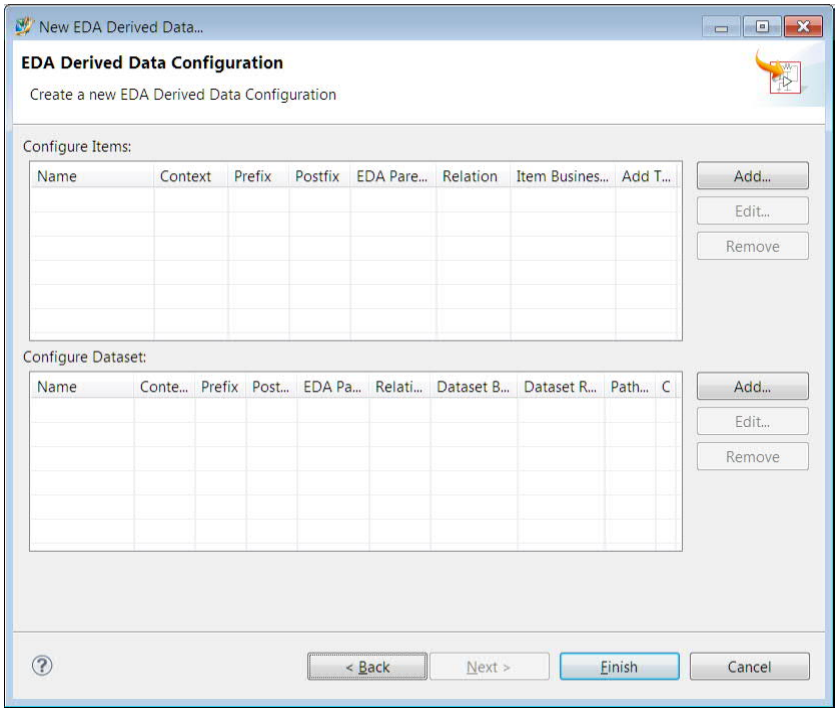
3. In the **EDA Derived Data** dialog box, enter the following information:

   a. In the **Name** box, type the name you want to assign to the new derived data configuration. This is the name used in the **EDA_DerivedDataConfigDefault** preference.

   b. In the **Description** box, type a description for the new configuration.

   c. Click **Next**.

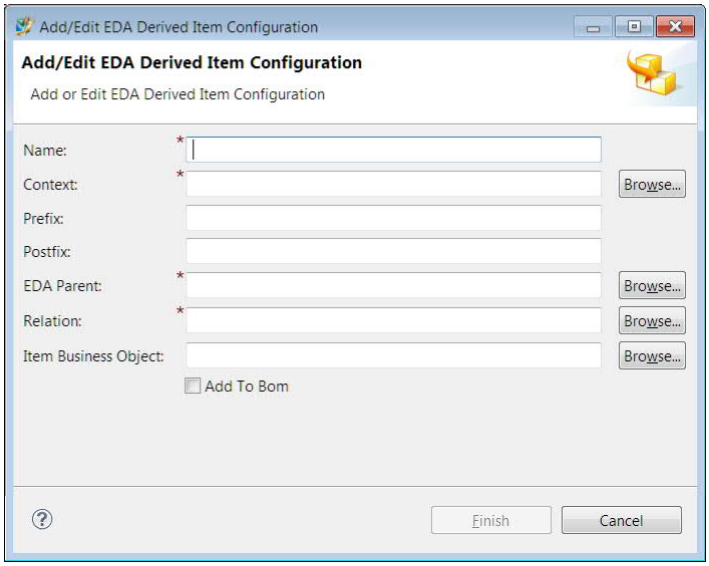
      The next dialog box in the wizard is displayed.



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 to the right of the **Configure Items** table.

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



      Use this dialog box to 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:

A. In the **Name** box, type the name that you want to assign to the derived item configuration. The value in this box is displayed to the user on the Teamcenter EDA **Derived Item** dialog box during save operations.

B. Click the **Browse** button to the right of the **Context** box to select the Teamcenter EDA application contexts that support generation of this derived dataset. (In other words, when users in Teamcenter EDA save derived data for the following specified data type, derived items are generated according to this configuration.)

- **all**
  All types (printed circuit boards, schematic diagrams, and simulations).
- **pcb**
  Printed circuit boards.
- **pcb/simulation**
  Printed circuit boards or simulations.
- **schematic**
  Schematic diagrams.
- **schematic/pcb**
  Schematic diagrams or printed circuit boards.
- **schematic/simulation**
  Schematic diagrams or simulations.
- **simulation**
  Simulations.

C. (Optional) In the **Prefix** box, type a file name string to be attached to the beginning of the parent item ID to distinguish it as being generated by this configuration.

The resulting string, including the prefix and postfix, is used in the derived item user interface in Teamcenter EDA as initial values for the **Derived Item ID** box and **Name** box and can be overridden by the user.

D. (Optional) In the **Postfix** box, type a file name string to be attached to the end of the item ID to distinguish it as being generated by this configuration.

The resulting string, including the prefix and postfix, is used in the derived item user interface in Teamcenter EDA as initial values for the **Derived Item ID** box and **Name** box, and can be overridden by the user.

E. Click the **Browse** button to the right of the **EDA Parent** box to select the derived parent EDA business object to which the derived item is related. (Teamcenter EDA does not support attaching derived items under other derived items.)

- **CCA**
  Represents a circuit card assembly (CCA).
- **CCABase**
  Represents the common design data that is shared between variant circuit card assemblies (CCAs). It is used only for multiple CCA representations.
- **CCAVariant**
  Represents the variant design data for a circuit card assembly (CCA). This is the data that is used on top of the **CCABase** business object.
- **PWB**
  Represents a printed wire board (PWB). A PWB is the product of a schematic design and printed circuit board (PCB) layout design and holds various printed wire board production data created by those designs.
- **Schematic**
  Represents the electrical schematic item.

F. Click the **Browse** button to the right of the **Relation** box to select the relationship between the derived item and the parent item revision.

The **EDAHasDerivedItem** business object and its children are displayed in the selection dialog box.

G. Click the **Browse** button to the right of the **Item Business Object** box to select the business object type name for the derived item, for example, **EDA**.

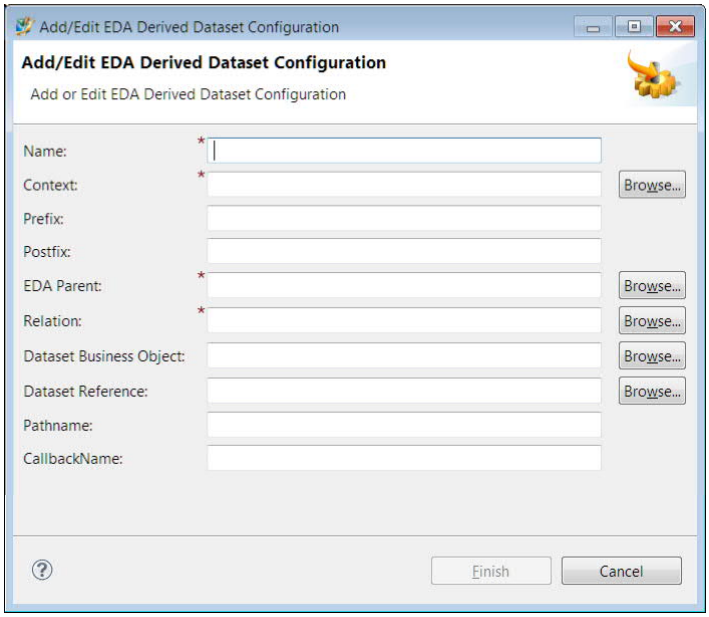
H. Select the **Add to Bom** check box to add the derived item to the bill of materials of the Teamcenter EDA parent. This check box is disabled if the Teamcenter EDA parent is schematic or PWB.

I. Click **Finish**.

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.



Use this dialog box to configure the derived datasets 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 datasets.

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

A. In the **Name** box, type the name you want to assign to the derived dataset configuration.

   The value in this box is displayed to the user on the Teamcenter EDA **Derived Dataset** dialog box during save operations.

B. Click the **Browse** button to the right of the **Context** box to select the Teamcenter EDA application contexts that support generation of this derived dataset. (In other words, when users in Teamcenter EDA save derived data for the following specified data types, derived datasets are generated according to the configuration.)

   - **all**
     All types (printed circuit boards, schematic diagrams, and simulations).
   - **pcb**
     Printed circuit boards.
   - **pcb/simulation**
     Printed circuit boards or simulations.
   - **schematic**

Schematic diagrams.

- **schematic/pcb**
  Schematic diagrams or printed circuit boards.

- **schematic/simulation**
  Schematic diagrams or simulations.

- **simulation**
  Simulations.

C. (Optional) In the **Prefix** box, type a file name string to be attached to the beginning of the parent item ID to distinguish it as being generated by this configuration.

   The resulting string, including the configured prefix and postfix, is used in the derived dataset user interface in Teamcenter EDA as the initial value for the **Dataset Name** box and can be overridden by the user.

D. (Optional) In the **Postfix** box, type a file name string to be attached to the end of the parent item ID to distinguish it as being generated by this configuration.

   The resulting string, including the configured prefix and postfix, is used in the derived dataset user interface in Teamcenter EDA as the initial value for the **Dataset Name** box and can be overridden by the user.

E. Click the **Browse** button to the right of the **EDA Parent** box to select the derived item type to which the derived dataset will be related. In addition to the following item types, the list also includes item configurations you already created.

   - **CCA**
     Represents a circuit card assembly (CCA).

   - **CCABase**
     Represents the common electrical CAD (ECAD) design data that is shared between variant circuit card assemblies. 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 various printed wire board production data created by those designs.

   - **Schematic**
     Represents the electrical schematic item.

F. Click the **Browse** button to the right of the **Relation** box to select the relationship between the derived dataset and the parent item revision.

The **EDAHasDerivedDataset** business object and its children are displayed in the selection dialog box.

G. Click the **Browse** button to the right of the **Dataset Business Object** box to select the parent dataset business object type in Teamcenter to represent the derived item, for example, **PDF**.

H. Click the **Browse** button to the right of the **Dataset Reference** box to select the kind of file reference to use for the derived dataset.

If the derived data instance comprises more than one file, this field must either be specified as a **ZIPFILE** type or must be specified using a separate derived dataset configuration entry with the same derived data name.

I. In the **Pathname** box, type the path where the derived dataset is to be saved on the user's machine.

Path names are evaluated at run time and must be the fully qualified path of the dataset that is to be saved. Path names can be explicitly specified (for example, **D:\EDA\Datasets\readme.txt**) or formed using the variables or file name filters. Derived datasets can contain multiple files. Path names are case sensitive, and the directory delimiters of **/** or **** are used interchangeably.

J. (Optional) In the **Callback Name** box, type the EDA callback name to execute.

This name is used to identify the configured callback in the EDA configuration file to determine what script to execute. The script is responsible for creating or placing the corresponding derived files to be uploaded as specified by the configured source path name.

K. Click **Finish**.

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

c. Click **Finish**.

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

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

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

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

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

   For example, in your ECAD design tool such as Mentor Graphics or Cadence, 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.

Using variables and wildcards in the derived data configuration path names

Several variables are available to be used in path names. This permits the sharing of configurations between users and workstations. This table provides the available variables, with descriptions and examples of each.

Variable	Description
$STAGE	EDA configured staging directory with full path formatted as *drive*:**\EDA\Staging***OS-user-name*\tc-user-ID_teamcenter-site-ID*\ *ECAD-family*, for example, **D:\EDA\Staging\joeb\joe_533845652\zuken**. The staging location supports multiple users on a single client machine and multiple Teamcenter installations for each user.
$TEMP	User system temporary directory with full path, for example, **C:\Temp**
$DESIGN	Current application design directory with full path formatted as *drive*:**\EDA\Staging***OS-user-name*\tc-user-ID_teamcenter-site-ID*\ *ECAD-family*, for example, **D:\EDA\Staging\joeb\joe_533845652\zuken\latest\A5E00444333-01\bd**
$APPNAME	Current running application name, for example, **zukenSchematic**
$FAMNAME	Current running family name, for example, **zuken**

Variable	Description
$USER	User name of the logged-on user, for example, **jsmith**
$ITEMID	Item ID of the current design, for example, **008723**
$VARIANT	Variant name, for example, **A5E01601836**

The following table provides a list of the file name filters that can be used in the path name.

Filter	Description	Example
Asterisk (*)	Matches any number of characters including none.	**$DESIGN**.doc** Selects all **.doc** files from the child folder under the **$DESIGN** folder. **$DESIGN**PWB*.doc** Selects all **.doc** files with **PWB** in their file names from the child folders under the **$DESIGN** folder. **$DESIGN\PWB**.doc** Selects all **.doc** files that start with **PWB** in their file names from the child folder under the **$DESIGN** folder.
Two asterisks (**)	Works like one asterisk (*) but crosses directory boundaries. This syntax is generally used to match complete paths.	**$DESIGN**\MyFile.doc** Selects the **Myfile.doc** file from all child folders under the **$DESIGN** folder.
Question mark (?)	Matches exactly one character.	**$DESIGN\Help\Help?.doc** Selects files with one character after their file name like **Help1.doc**, **Help2.doc** and **HelpA1.doc** but not **HelpMe.doc**.
Braces ({ })	Specifies a collection of subpatterns.	**$DESIGN\Help{Me,You}*.pdf**

Filter	Description	Example
		Selects all **.pdf** files from either the **HelpMe** or **HelpYou** folder.
Square brackets ([ ])	Conveys a set of single characters or when a hyphen (-) is used a range of characters.	**$DESIGN\Help[123]*.doc** or **$DESIGN\Help[1-3]*.doc** Selects all **.doc** files from any one of the folders **Help1**, **Help2**, **Help3**.

Multiple files in a derived dataset must follow certain rules:

- Multiple fully qualified files can be saved to any derived dataset using semicolons, for example, **$STAGE\Board.pdf; $DESIGN\Board.dwg; C:\Temp\Logfile.txt**. The following is not permitted:

  $STAGE*.pdf; $STAGE\Logfile.txt

- Derived datasets that are configured with a **ZIPFILE** dataset reference can contain multiple files, selected using the filters in the table. Files must be in a common directory or in subdirectories of a common directory. You can specify only one path for a **ZIPFILE**; that is, you cannot use semicolons:

  $STAGE*.pdf
  $STAGE\Board.pdf
  D:\EDA\Help[345].pdf

  The following is not permitted:

  $STAGE*.pdf; $DESIGN*.txt

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

Example:

If your folder structure is like this:

$DESIGN\File1.doc

$DESIGN\Help1\File2.doc

$DESIGN\Help1\File3.pdf

$DESIGN\Help2\File4.doc

$DESIGN\Help2\File5.pdf

$DESIGN\Help2\File6.xls

The wildcards give the following results:

$DESIGN*.doc** fails.

$DESIGN.pdf** fails.

$DESIGN.xls** succeeds.

$DESIGN\Help1*.doc succeeds.

$DESIGN\File4.doc** succeeds.

$DESIGN*\File3.pdf succeeds.

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

Example:

$DESIGN/manufacturingData/**

Validation Manager

Configure Validation Manager using the Business Modeler IDE

Use the Business Modeler IDE to create custom objects used by the Validation Manager application. Validation Manager objects are provided by the Foundation template. No additional templates are needed.

You can add new business objects that provide extended properties and specialized behaviors in support of the associated validation agent. The **NXCheckMate** and **NXCMValData** business objects support Validation Manager interaction with the Check-Mate application in NX Integration. The **NXRDDV** and **NXRDDVValData** business objects support the Validation Manager interaction with the Requirement-Driven Design Validation application in NX Integration.