cadence®

Allegro® EDM
Database Editor User Guide

Product Version 23.1
September 2023

Cadence Design Systems, Inc. (Cadence), 2655 Seely Ave., San Jose, CA 95134, USA.

Contents

Preface

About This Guide

The *Allegro® EDM Database Editor User Guide* explains the methodology and the procedures with which you can manage the library data for your component database.

Related Documentation

You can also refer the following documentation to know more about related tools and methodologies:

- For information on new features, see *Allegro EDM: What's New in Release*.
- To learn how to use Database Administrator, see *Allegro EDM Database Administrator User Guide.*
- To learn how to use Library Import, see *Allegro EDM Library Import User Guide*.
- To learn how to use Library Distribution, see *Allegro EDM Library Distribution User Guide*.

Related Tools and Flows

- For information on various PCB design working environments such as a team of designers working on a Design Entry HDL project, implementing FPGAs in designs, working with high-speed constraints, importing IFF files for radio-frequency designs, and reusing existing modules, see *Allegro PCB Design Flows*.
- For learning how to create and configure Design Entry HDL projects, see the *Allegro Project Manager User Guide*.

Typographic and Syntax Conventions

This list describes the syntax conventions used for this user guide:

`literal`	Nonitalic words indicate keywords that you must enter literally. These keywords represent command (function, routine) or option names.
`argument`	Words in italics indicate user-defined arguments for which you must substitute a name or a value.
|	Vertical bars (OR-bars) separate possible choices for a single argument. They take precedence over any other character.
`[ ]`	Brackets denote optional arguments. When used with OR-bars, they enclose a list of choices. You can choose one argument from the list.
`{ }`	Braces are used with OR-bars and enclose a list of choices. You must choose one argument from the list.

1

Getting Started with Database Editor

Understanding Database Editor

The Allegro EDM Component Database contains part and model data, which is used by designers in their designs. ECAD librarians keep this database up-to-date. For any changes in the database, due either to new parts and models or a revision of existing parts and models, librarians have the difficult job of making the changes, and making them readily available to the design teams.

Allegro EDM Database Editor not only helps librarians exercise complete control over authoring and managing component libraries but also work closely with the enterprise lifecycle management methodology.

After you have defined the administrator data (using Database Administrator), which acts as a template for your component database, you can create, edit, and delete library-specific data with the Database Editor application.

For information on how to work with administration data using the Database Administrator application, see *Allegro EDM Database Administrator User Guide*.

Parts and Model Types Supported

To understand models in EDM, let us first understand what EDM considers a part. Off-the-shelf physical components, such as resistors, diodes, capacitors, integrated circuits, and so

on, are called *parts* in Allegro EDM. From the perspective of Cadence applications, the part number in a front-end `.ptf` file (illustrated below) is defined as a part in Allegro EDM.

```
FILE_TYPE = MULTI_PHYS_TABLE;

PART 'VGA'

{=============================================================
:PART_NUMBER     | PACK_TYPE     = JEDEC_TYPE | HEIGHT     | ROHS
{=============================================================
 'CDN-CON-0015' | 'VGA'(!)      = 'DSUB15'    | '12.55MM' | 'COMPL

END_PART
```

From this point on, the term *part* will be used to refer to off-the-shelf physical components available in the industry.

Models in EDM

Applications such as Design Entry HDL, Allegro System Capture, OrCAD Capture, each have different abstract representations of a part, depending on the way the application is structured.

To ensure that Allegro EDM understands the authoring tool of a part as well as the use of the part, Allegro EDM categorizes these abstract representations as database models. Each part can be linked to one or more models. For example, EDM can categorize an electrical part as a Schematic Model-Footprint Model-Datasheet Model. A part is constructed by first creating the various models and then referencing/linking those models to define the part.

For more details about models, refer to the model-related section of the Allegro EDM Frequently Asked Questions document.

Database Editor supports the following models:

- Standard
- Schematic
- Mechanical
- Block
- Capture
- Footprint

- Padstack
- Flash
- Shape
- Board
- Format
- Module
- Datasheet
- SI DML

Collaborative Library Authoring and Management

Database Editor helps you work in an integrated and concurrent library development environment. This has been made possible with the help of working sets. Working sets are sharable workspaces that can contain work-in-progress components. Librarians can concurrently work on the components for which they are responsible.

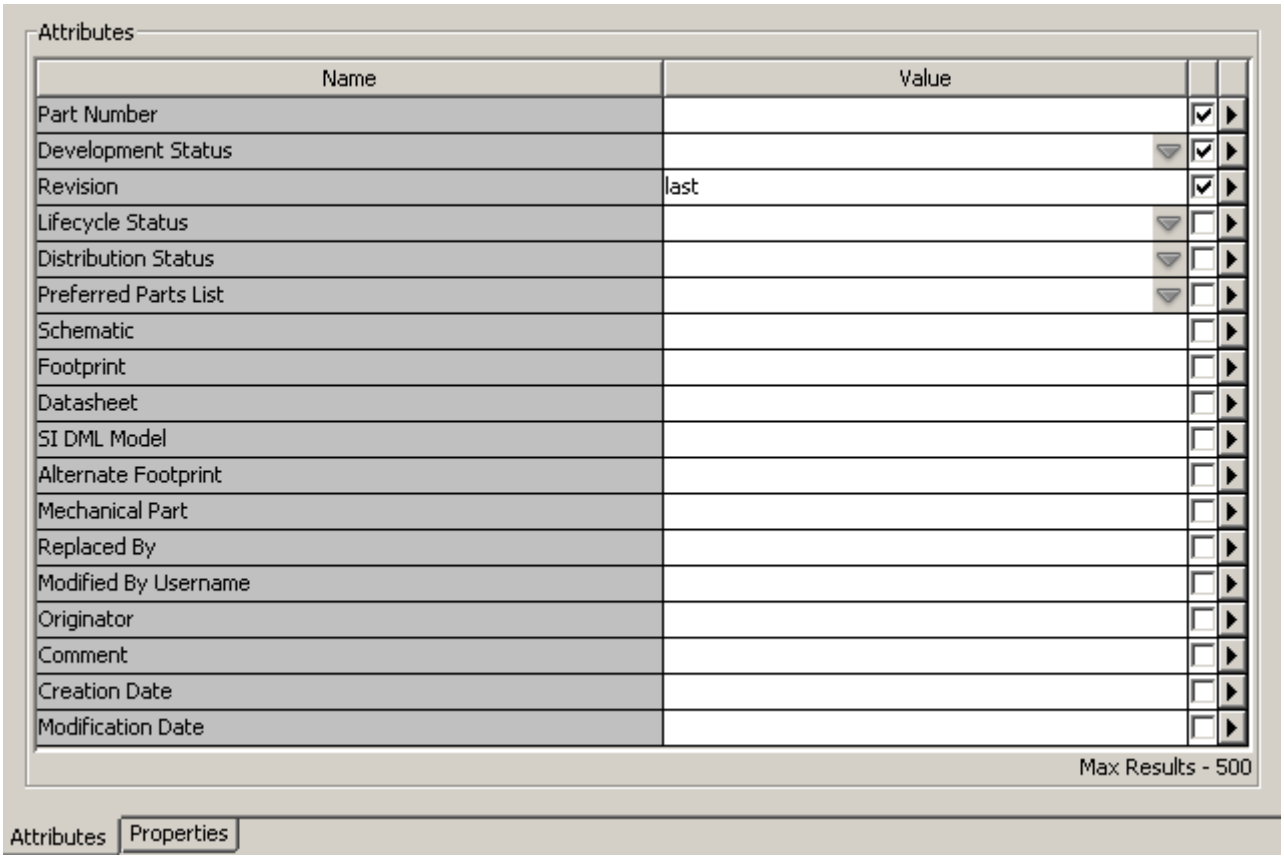
Library Development and Lifecycle States

Enterprise library components follow a well-defined development process, which can include many stages such as component creation or modification, component verification, component flow-verification, component release, and component distribution.

Database Editor helps you synchronize all these activities by enabling you to work on all these activities from a unified interface. The following fields define the various states of parts and models in the Allegro EDM component database:

- Development status
- Lifecycle status

- Distribution status



Development Status

A library object's *Development Status* attribute holds the values that correspond to the states that the object goes through during its library development/ECO cycle.

The Development Status has the following values:

- *Preliminary*
- *Checkout*
- *Checked-In & Verified*
- *Released*
- *Pre Released*
- *Deleted*

Lifecycle Status

Library parts can also have additional associated business lifecycle states, which are configurable by the library administrator. For example:

- Approaching EOL
- EOL
- Approved

Such states are specific to companies. The site administrator can also specify actions to be taken by Part Information Manager when designers add parts with certain business lifecycle states to their designs.

Distribution Status

The distribution status of an object is displayed as an attribute. This field can have any of the following values:

- **Pending Distribution**: For new or checked-out models
- **Distributed**: After library distribution is done for the latest version of the model or part
- **Previously Distributed**: Applies to older versions of distributed parts and models
- **Pending Delete**: For models and parts that have been marked as deleted after which the library has not yet been distributed
- **Pending Purge**: For models and parts that have been marked for permanent deletion from the database after which the library has not yet been distributed

Launching Database Editor

You can launch the application from the following:

- Command Line
- Flow Manager

Command Line

1. In the Allegro EDM system console, type `dbeditor,` and press the Enter key.

   The Login dialog box appears.

2. Enter the login ID and password, and click *Login*.

   The *Allegro EDM Database Editor* window opens.

Note: You can change the size or location of the Database Editor window. This information is saved and used the next time you launch the application.

Flow Manager

Depending on the library flow steps configured in the Flow Manager, you can launch Database Editor directly from the flows.

Important

You can access Database Editor if the login name you use has privileges related to the following roles: ECAD Librarian, Senior ECAD Librarian, or Library Administrator.

For information on the interface of Database Editor, see Appendix C, “Database Editor User Interface,” or press F1 to launch the online help for the product.

2

Working with Database Editor

Overview

This chapter provides a basic understanding of the tasks you can perform in Allegro EDM Database Editor. The major tasks you can perform are as follows:

- Working with Parts
- Working with Models
- Working with Attributes and Relations
- Searching Parts and Models
- Using Working Sets

For information on the user interface elements of Database Editor, see Appendix C, “Database Editor User Interface,”.

Working with Parts

This section explains the following tasks:

- Creating Parts
- Editing Parts
- Data and Development Status Tasks

Creating Parts

To create a part, do the following:

1. In the Allegro EDM Database Editor window, choose *File – New – Part*.

   The New Part dialog box appears.

   - To create a block part, choose *File – New – Block Part*.
   - To create a mechanical part, choose *File – New – Mechanical – Mechanical Part*.
   - To create a mechanical kit, choose *File – New – Mechanical – Mechanical Kit*.

2. Enter a part number in the *Part Number* field.

   If you are creating a mechanical kit, enter a kit number.

   Note: Database Editor writes all electrical part numbers in uppercase letters if `extension0007`, using the `adwschema` utility, has been enabled. Mechanical kit and part numbers are automatically written in uppercase letters in Database Editor.

3. Click *Create*.

   The *Part - <Part_Detail>* tab appears.

   If you are creating a mechanical kit, the *Mechanical Kit - <Kit_Detail>* tab appears.

4. In the tab view, you can do the following:

   a. use the left pane (also known as explorer pane) to specify relations to the part. For information on how to do this, see Working with Attributes and Relations.

   b. view the key and mandatory attributes in the *Informational Attributes* area of the right pane. You cannot edit these attributes.

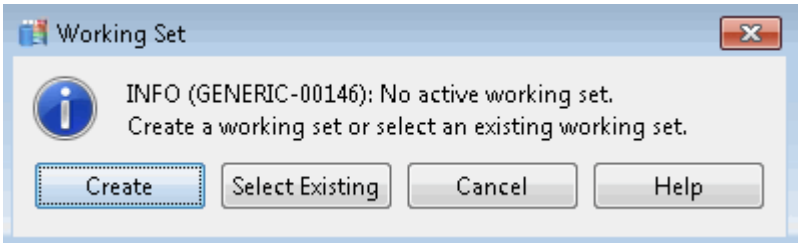
c. specify other mandatory and optional attributes using the *Attributes* area of the right pane. For information on how to do this, see Working with Attributes and Relations.

The attributes in **bold** are key and mandatory. For a detailed list of default attributes and their types, see Appendix A, “Attributes and Models,”.

5. Choose *File – Save* to save the part information in the database.

When you save a new part, it is saved with an initial version of 1.0. This version information supports the revision management features in the Allegro EDM component database.

When you save a new part, it is by default added to your current working set. If you do not have any working set, an alert appears prompting you to create or select an existing working set. For information on how to do this, see Using Working Sets.



By default, all new parts are assigned `Preliminary` as the *Development Status*. For information on database and lifecycle management tasks in Database Editor, see Data and Development Status Tasks.

Editing Parts

You can edit parts using any one of the following ways:

- Editing a Single Part or Model
- Editing Multiple Parts or Models
- Using the Group By Feature
- Working with Footprint Sets

Important

The steps for editing parts, block parts, mechanical kits, and models are similar.

Editing a Single Part or Model

To edit a single part or model, do the following:

1. In the *Database Editor* window, choose *File – Open – Part*.

   The Open Part dialog box appears.

   To edit a block part, choose *File – Open – Block Part*.

   To edit a mechanical part or kit, choose *File – Open – Mechanical – Mechanical Part or File – Open – Mechanical – Mechanical Kit*.

   To edit a model, choose *File – Open – Model – <Model_Type>*.

2. Enter a part number in the *Part Number* field.

   If you are opening a mechanical kit, then enter a kit number.

3. Click *Open*.

   The *Part - <Part_Detail>* tab appears.

   If you are opening a mechanical kit, the *Mechanical - <Part_Detail>* tab appears.

Important

You can edit a part, block part, mechanical kit, or model only when it has been checked out. For checked-in parts, block parts, mechanical kits, or models you can only view their details.

4. In the tab view, you can do the following:

   a. Use the explorer pane to edit relations to the checked-out part. For information on how to do this, see Working with Attributes and Relations.

   b. View the key and mandatory attributes in the *Informational Attributes* area of the right pane. You cannot edit these attributes.

   c. Edit available mandatory and optional attributes using the *Attributes* area of the right pane. For information on how to do this, see Working with Attributes and Relations.

   Attributes in **boldface** are key and mandatory. For a detailed list of default attributes and their types, see Appendix A, “Attributes and Models,”.

5. Choose *File – Save* to save the changes made in the part.

Editing Multiple Parts or Models

You can edit multiple parts, block parts, or models within a single tab.

Important

You can edit either multiple parts or models within a single tab but **not** both.

To edit multiple parts, do the following:

1. In the *Search Results* tab or *Working Set* tab, select multiple parts using `Shift+Click`.

   For information on how to search for parts and models, see Searching Parts and Models and for information about working sets, see Using Working Sets.

2. Right-click and choose one of the following pop-up menu commands:

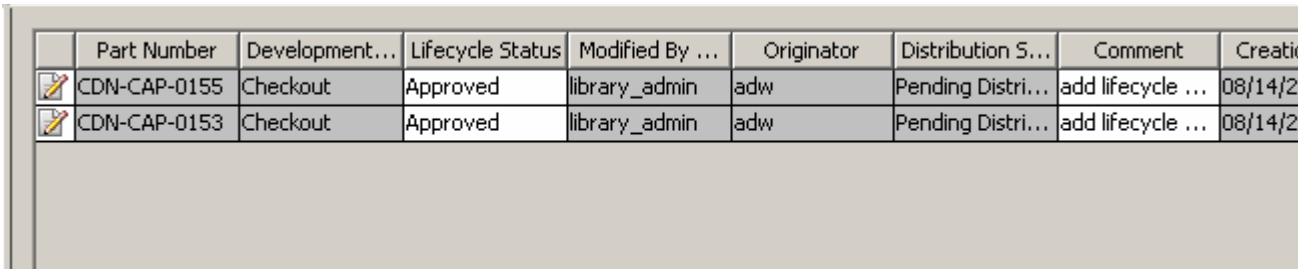
   - *Edit All*: The *Edit: Parts* tab appears listing all the parts. Choose *Edit – Check-out*.
   - *Check-out*: The *Edit: Parts* tab appears listing all the parts with `Checkout` as the Development Status.

   If you are editing models, the *Edit: Model* tab appears.

3. Edit the properties (non-grayed cells) in the grid.

   You can also use the *Copy* and *Paste* pop-up menu commands to copy and paste values in the grid.

4. Choose *File – Save*.



	Part Number	Development...	Lifecycle Status	Modified By ...	Originator	Distribution S...	Comment	Creati
	CDN-CAP-0155	Checkout	Approved	library_admin	adw	Pending Distri...	add lifecycle ...	08/14/2
	CDN-CAP-0153	Checkout	Approved	library_admin	adw	Pending Distri...	add lifecycle ...	08/14/2

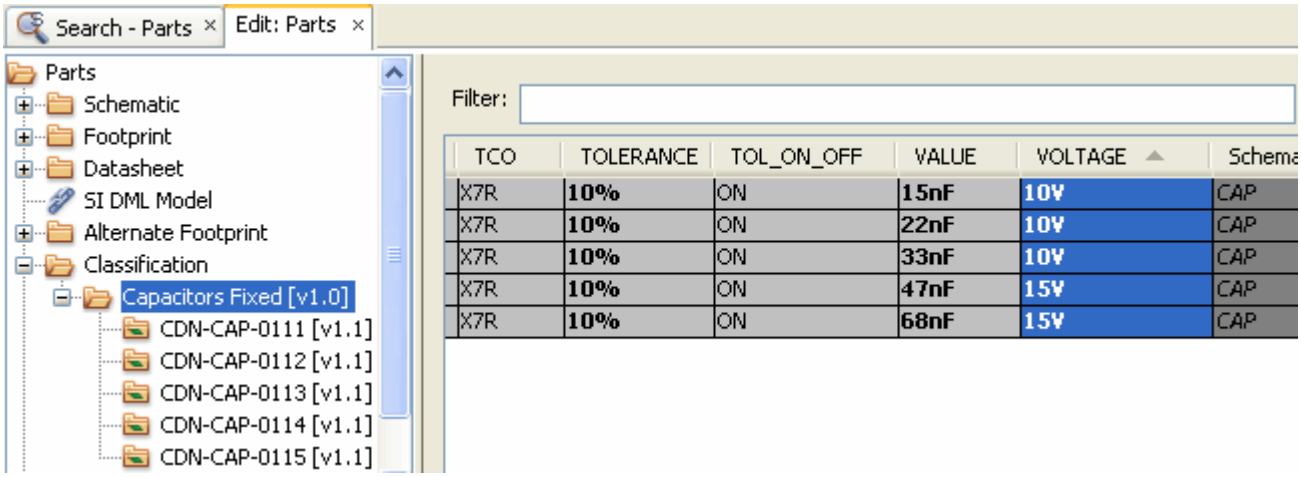
5. Choose *Edit – Check-in*.

Filtering Parts or Models for Multi-Editing

To filter part rows, do the following:

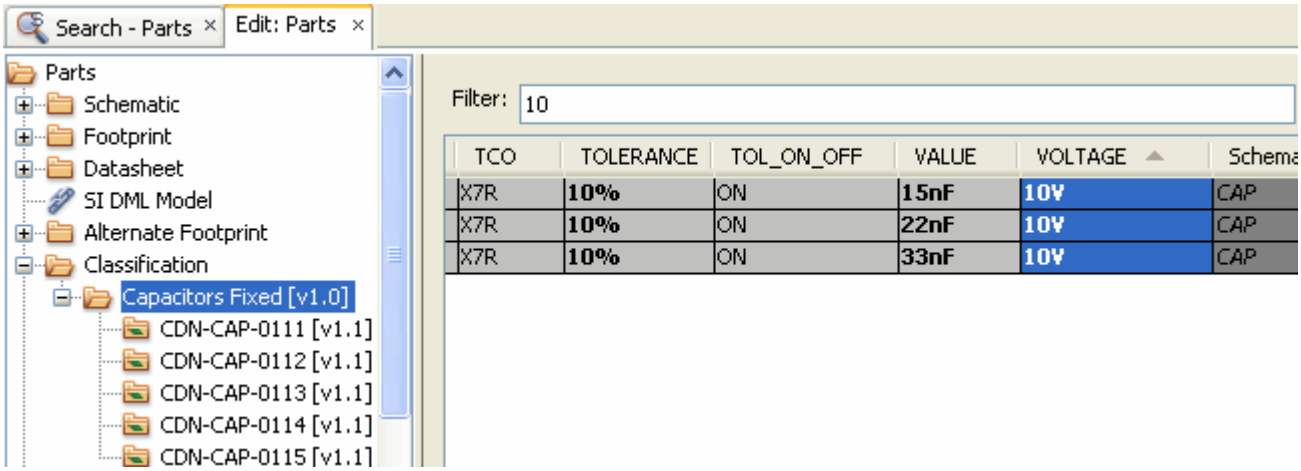
1. In the *Search Results* tab or *Working Set* tab, select multiple parts using `Shift+Click`.
2. Right-click and choose *Edit All.*

   The *Edit: Parts* tab appears listing all the parts. If you are filtering models, the *Edit: Model* tab appears.
3. In the explorer pane, choose *Classification* – `<Part_Classification_Name>`.
4. In the right pane, choose the property header based on which you want to filter the parts.



5. Specify the value of the property in the *Filter* field.

   The filtered part rows appear in the right pane.

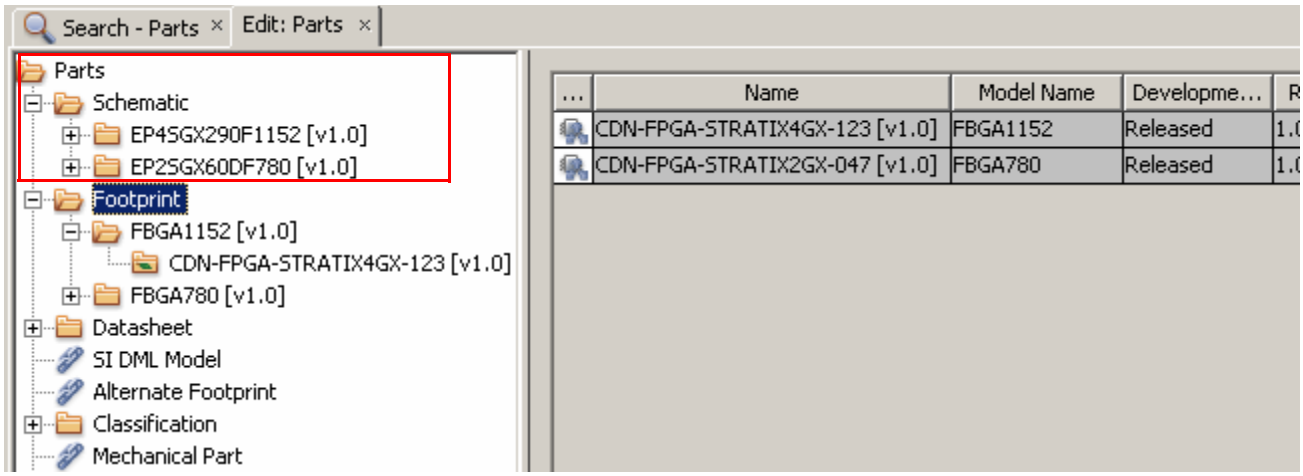


Note: Filtering works only on the basis of the columns that you select. If you do not select any column, the filter will act as a free-text filter.

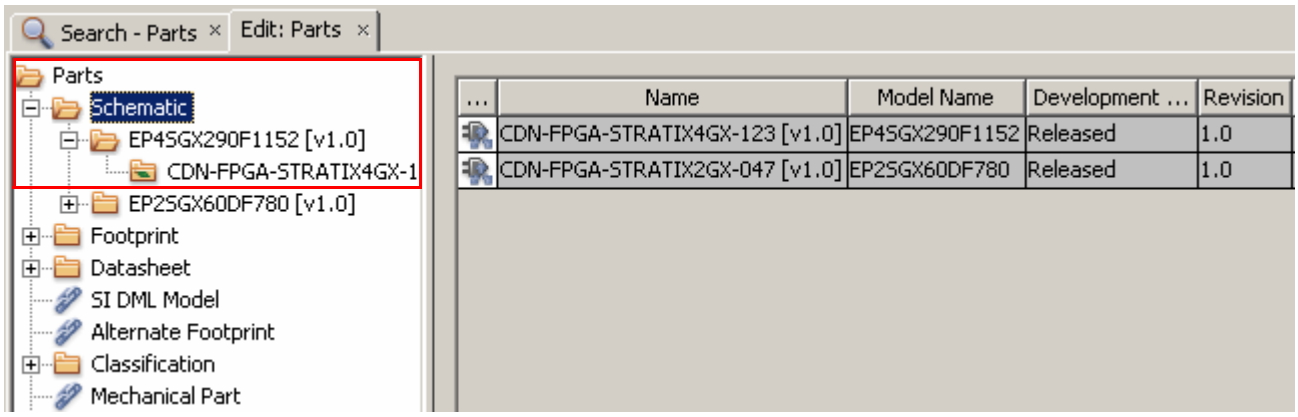
Using the Group By Feature

The Group By feature in the multi-part and multi-model editing mode allows you to easily view and edit the library data. You can group the library data by:

- Library Data Type (part or model): In this group mode, all the relation instances (for all the parts or models) are listed by the part or model number. For example, in the following figure, the `Footprint` relation contains all the part numbers, which display the footprints linked to them.



- Relation Entities: In this group mode, all the relations instances (for all the parts or models) are listed by the relation entities. For example, in the following figure, the `Schematic` relation contains the schematic models, which display the parts linked to them.



To group the parts in the *Edit: Parts* tab, do the following:

1. Choose a relation node.
2. Right-click the relation node.
3. Choose *Group By Part* or *Group By* `<Relation_Name>` command.

Note: By default, all the relations are grouped on the basis of the data model.

If you are working with models in the *Edit:* `<Model_Type>` tab, do the following:

1. Choose *Classification* or *Linked Parts* node.
2. Right-click the node.
3. Choose any of the following active options:
   - *Group By Classification*
   - *Group By* `<Model_Type>`
   - *Group By Linked Parts*

Working With Relations

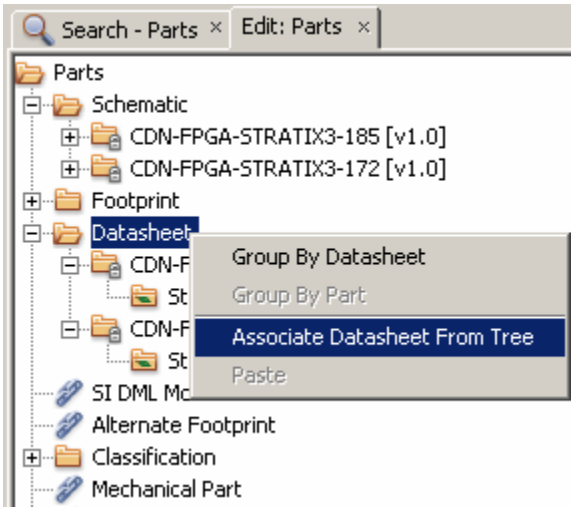
Using the explorer pane in the multi-part or multi-model editing mode, you can create, edit, and delete relations.

To create and edit a relation instance associated with the part or model, do the following:

1. Click the *Edit: Parts* or *Edit:* `<Model_Type>` tab.
2. Choose *File – Check-out*.
3. Right-click the required part or model.
4. Choose *Associate* `<Relation_Name>` *From Tree*.

   Note: If this option is disabled, it signifies that the maximum allowed relation instances

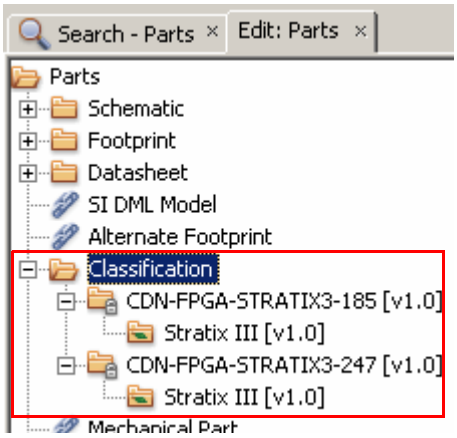
have already been created for this part or model.



You can also delete relation instances. For more information on how to do this, see *Deleting Relation Instances*.

Copy-Paste Operation

Multi-component editing also allows you to effectively use copy-paste operations on the relations. Assume you want to associate an additional part classification with all of the parts you are editing.

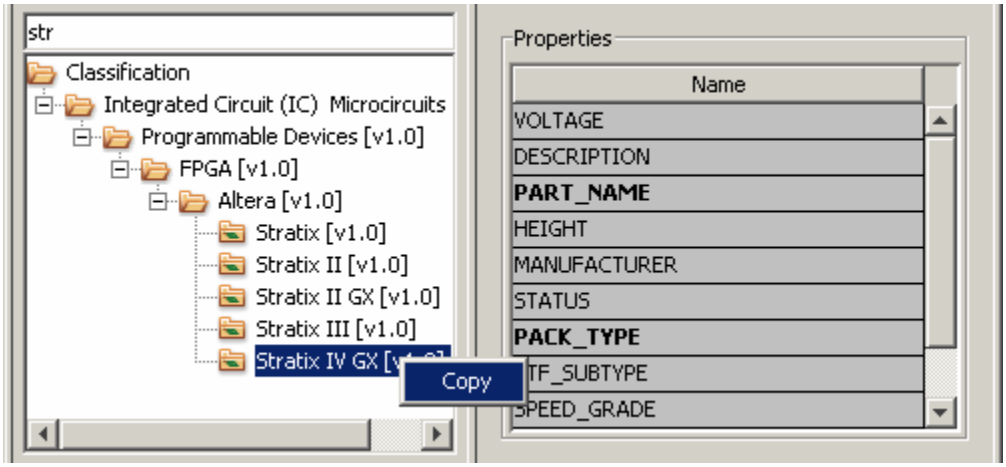


To do this, perform the following tasks:

1. Click the *Edit: Parts* tab.
2. Choose *File – Check-out*.

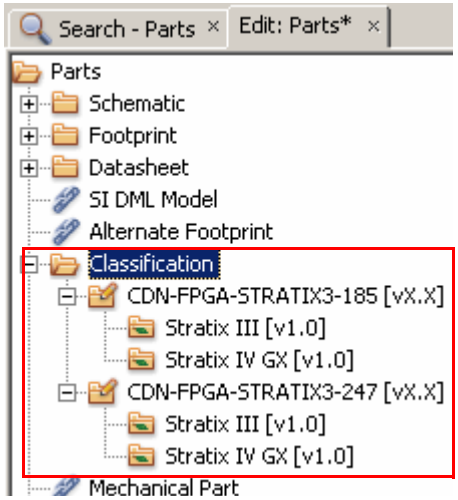
3. Click the `Classification` node.
4. Right-click the node and choose *Associate Classification From Tree*.

   The Associate Classification dialog box appears.
5. Choose the required classification.
6. Right-click this classification and choose *Copy*.



7. Click *Cancel* in the Associate Classification dialog box.
8. Choose the `Classification` node in the explorer pane of the *Edit: Parts* tab, and right-click it to choose *Paste.*

   The new classification is added to all the parts listed under it as shown in the following figure.



Working with Footprint Sets

In Allegro EDM, the combination of footprint and alternate footprint associated with a released part is defined as a Footprint Set. Database Editor automatically detects and displays unique footprint sets from the component database.

You can now associate these footprint sets to parts while creating or modifying parts. You can also add or delete alternate footprints in a footprint set and apply the changes to other existing parts that use the same footprint set.

- Viewing Footprint Sets
- Creating New Footprint Set
- Modifying Existing Footprint Set
- Associating New or Modified Footprint Sets with Other Parts
- Replacing Footprint Set

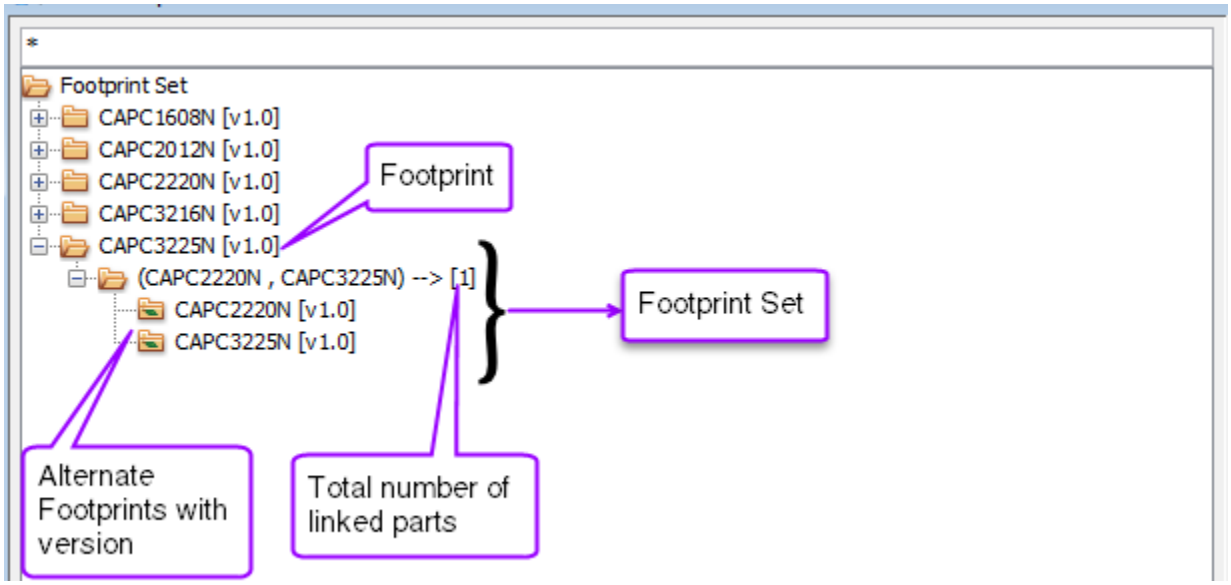
Viewing Footprint Sets

To view the footprint sets in the database, do the following:

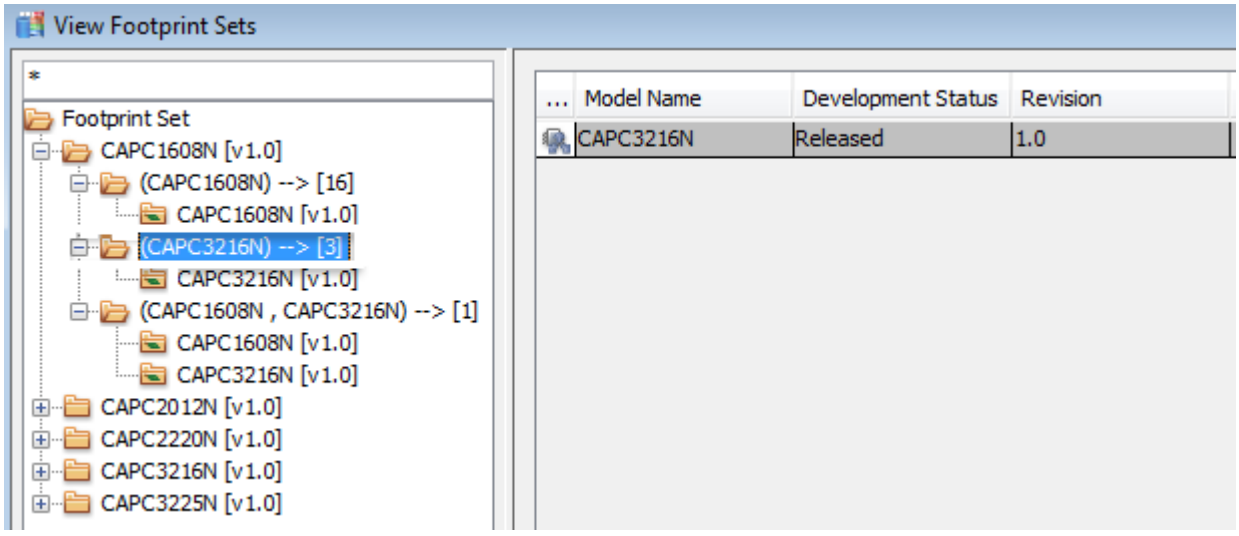
1. Open Database Editor.
2. Choose *View – Footprint Sets*.

   The View Footprint Sets window opens.
3. Select and expand any node in the left panel.

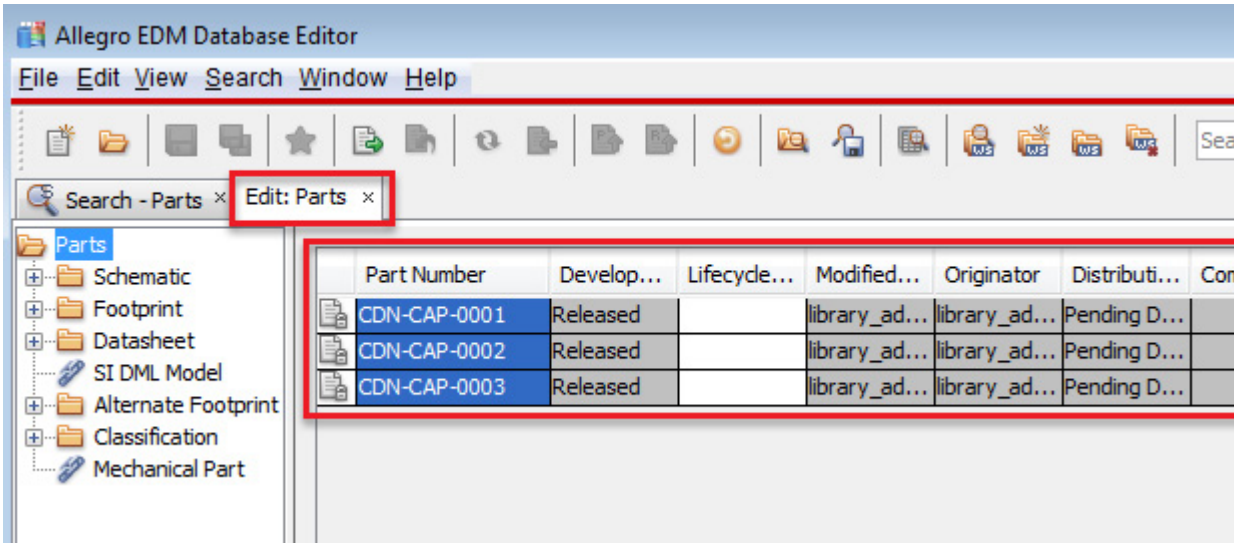
For example, in the following figure, you can identify a footprint set along with the number of released parts linked to it.



4. To view the released parts linked to a selected footprint set, select the alternate footprint node, which is followed by a square bracket containing a numeral.
5. Click the *Show Linked Parts* button.



The *Edit: Parts* tab appears with a list of linked parts.

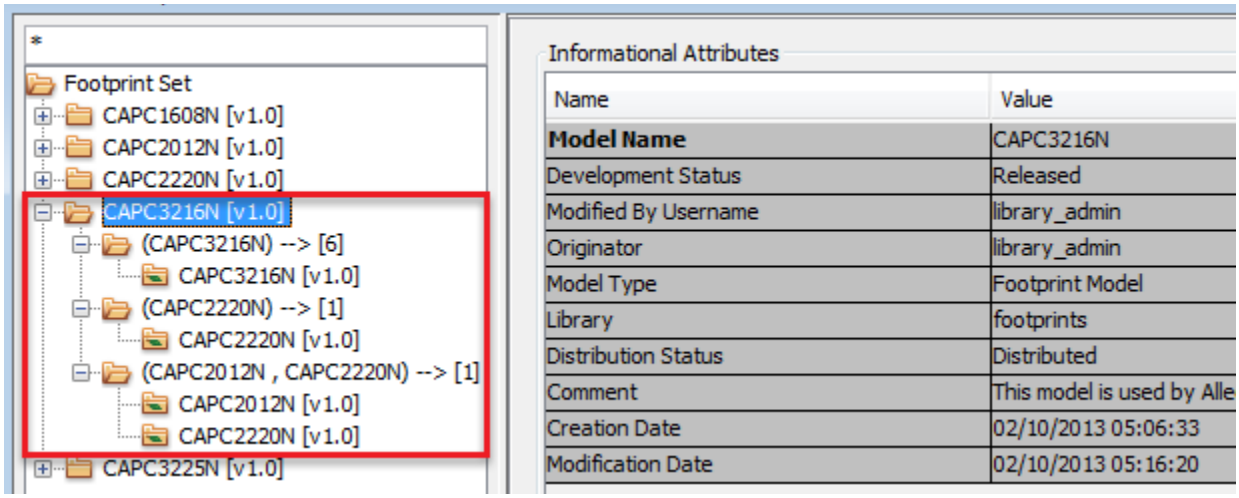


Creating New Footprint Set

Any change in any footprint set (addition or deletion of alternate footprints) creates a new footprint set. To create a new footprint set, do the following:

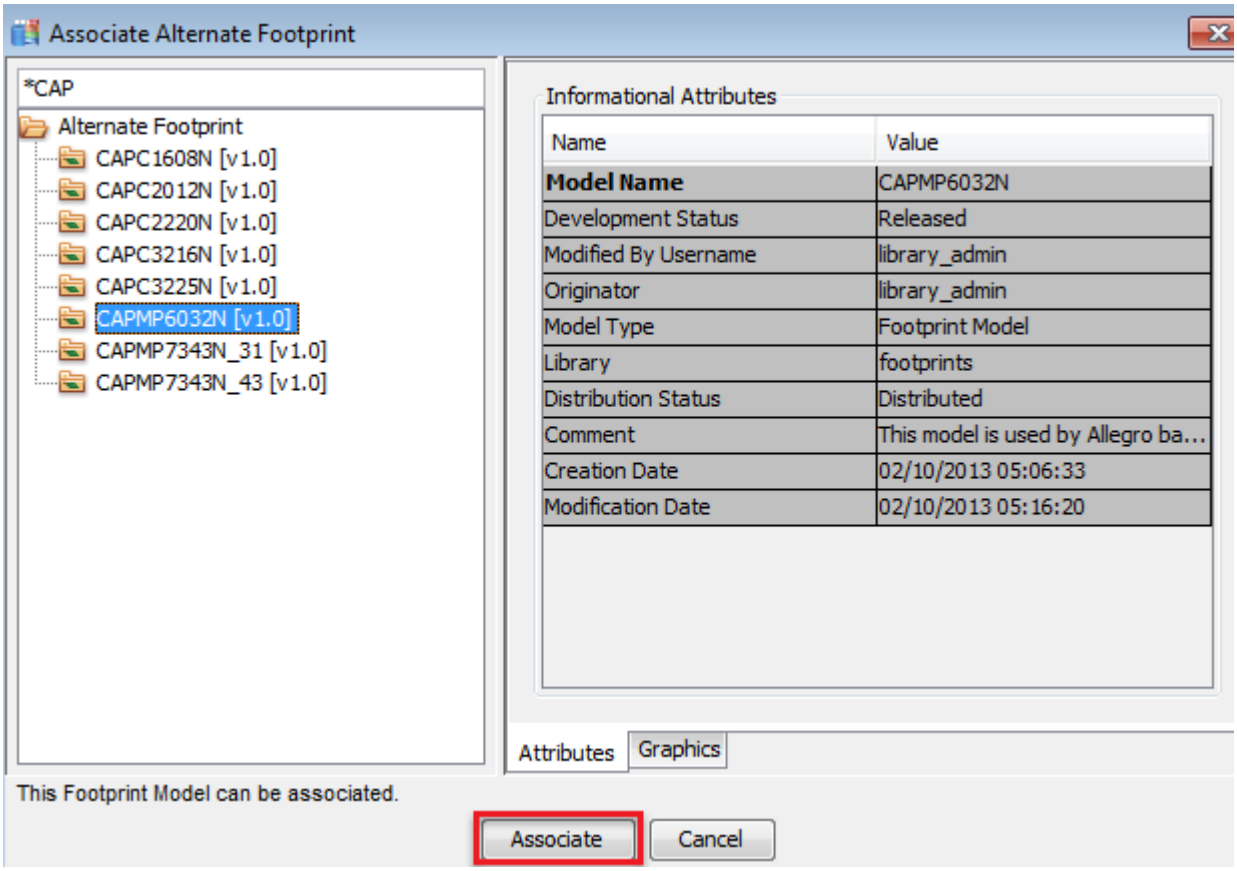
1. Open Database Editor.
2. Choose *View – Footprint Sets*, and review the existing footprint sets. This step is optional.

   For example, a new footprint set is to be added for the footprint, `CAPC3216N [V1.0]`.



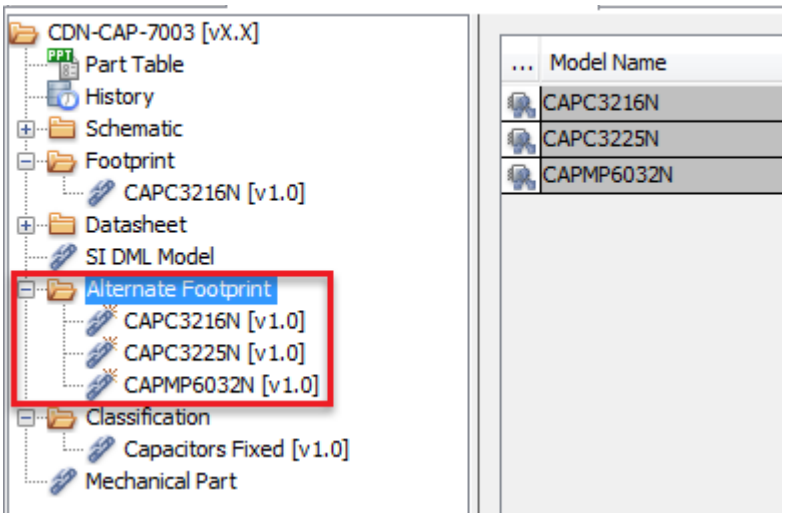
3. Search and select a part from the *Search Results* tab.
4. Check out the selected part.
5. Right-click *Alternate Footprint* and select *Associate Alternate Footprint*.

   The Associate Alternate Footprint dialog box opens.
6. Select a footprint and click the *Associate* button.



A new relation instance for *Alternate Footprint* is added.

7. Repeat step 6 if you want to add more alternate footprints.

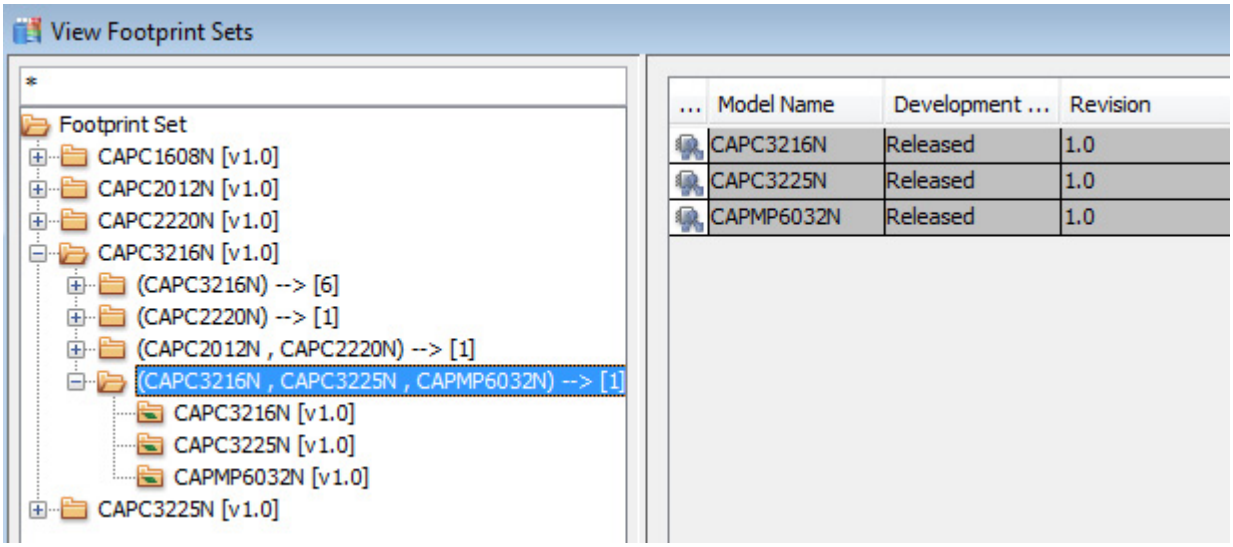


8. Save the part.
9. Check in and release the part.

   Note: For the new footprint set to be displayed in the View Footprint Set window, the part must be in the `Released` state.

10. Open the View Footprint Set window.
11. Expand the footprint node for which the alternate footprint relation was modified.

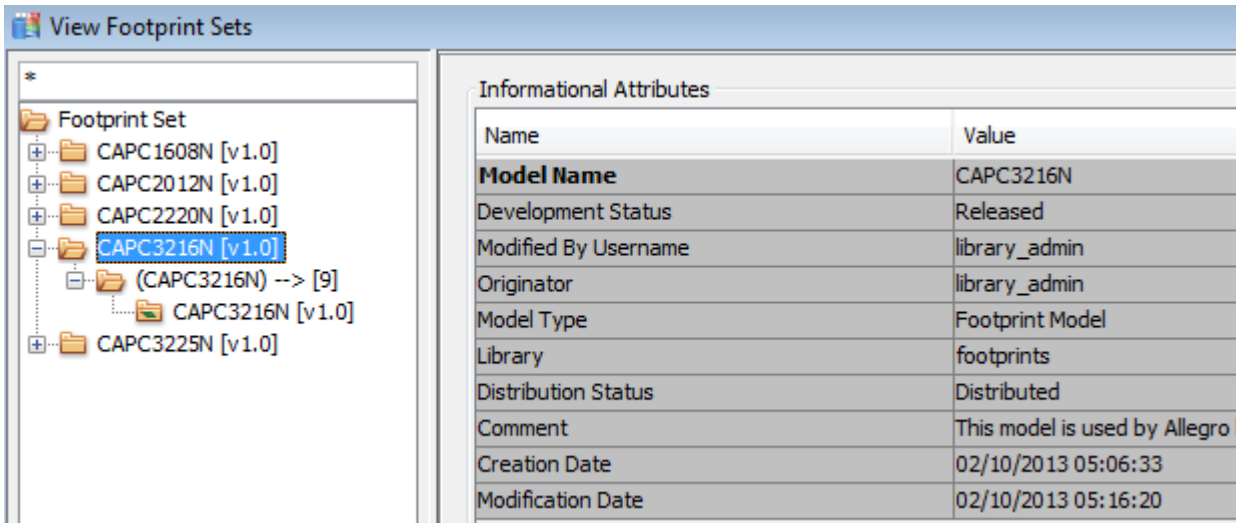
    In this example, a new footprint set is created which has three alternate footprints.



Modifying Existing Footprint Set

To modify a footprint set, do the following:

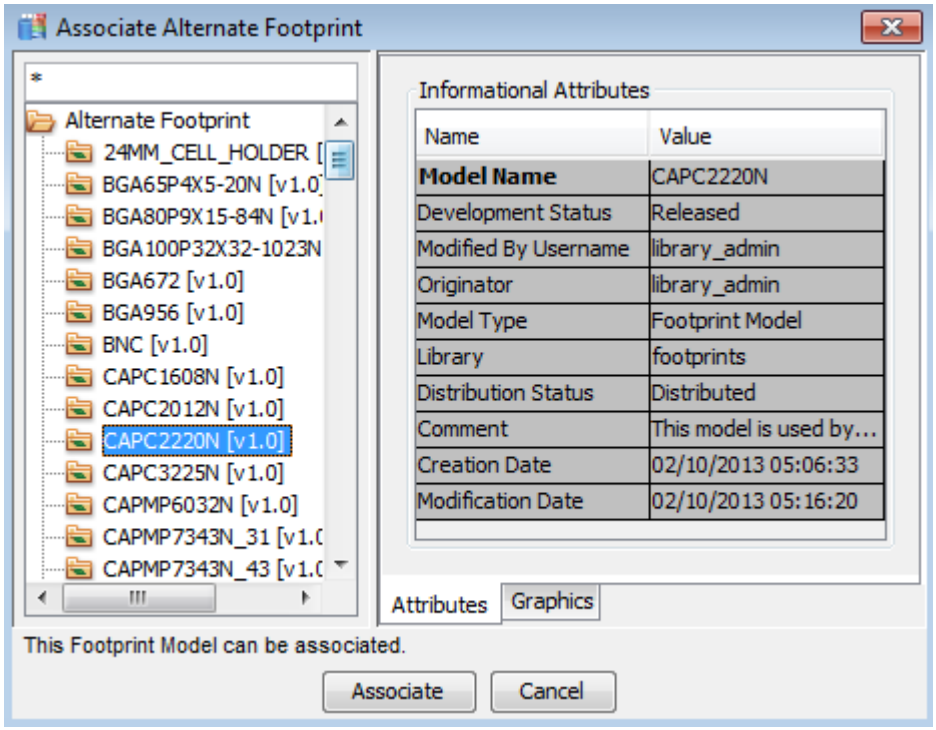
1. Open Database Editor.
2. Choose *View – Footprint Sets*, and review the existing footprint sets. This step is optional.



3. Search and select a part from the *Search Results* tab.
4. Check out the selected part.
5. Right-click *Alternate Footprint* and select *Associate Alternate Footprint*.

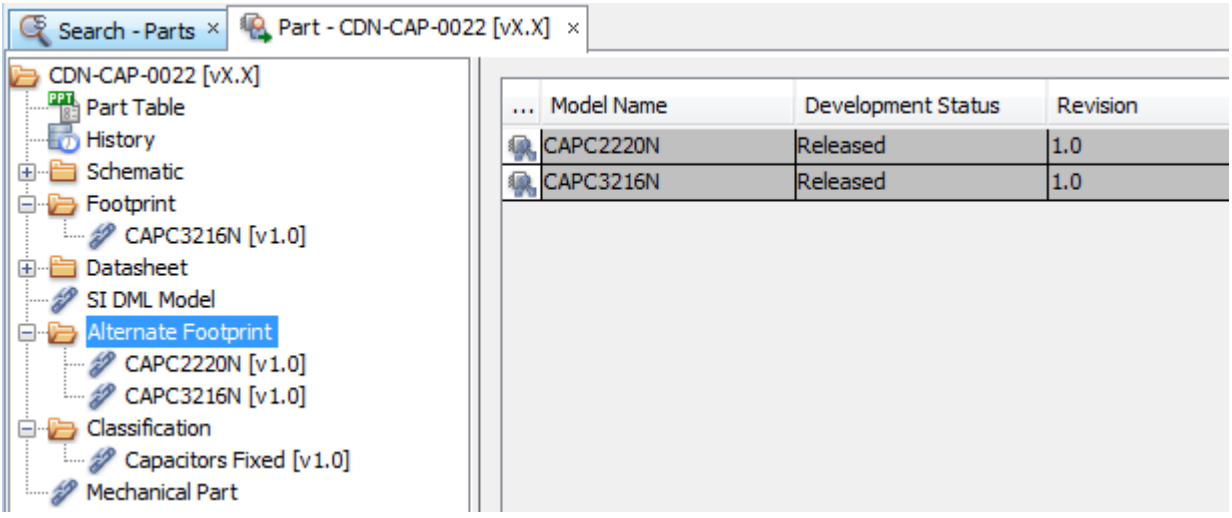
   The Associate Alternate Footprint dialog box opens.

6. Select a footprint and click the *Associate* button.



In this example, another relation instance for the alternate footprint is added. You can also the replace an existing alternate footprint with a different one.

7. Save the part.

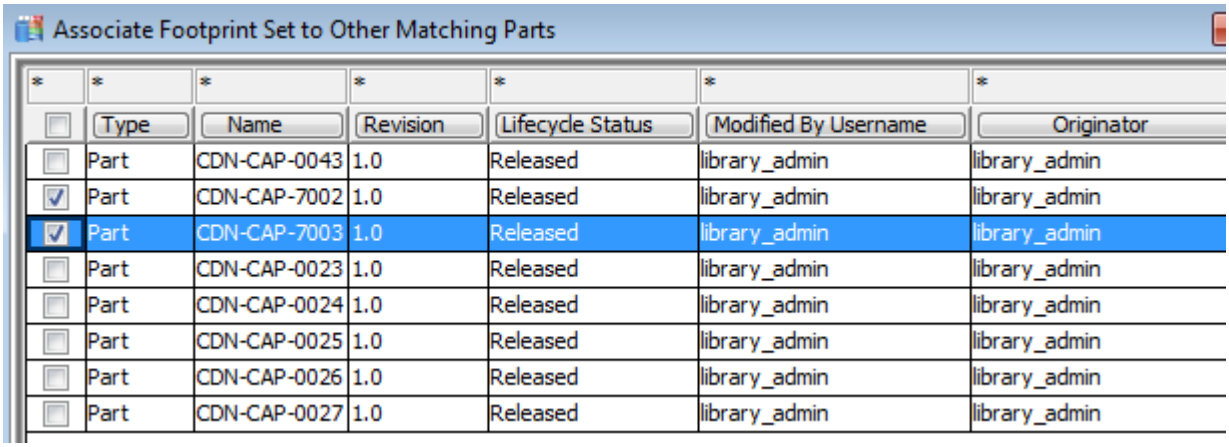


8. Check in and release the part.

Note: For a modified footprint set to be displayed in the View Footprint Set window, the part must be in the `Released` state.

9. If this footprint set was associated with other parts, a message appears to confirm if you want to associate this modified set with those parts.
10. Click *Yes* if you want to replace footprint sets in other parts with this modified footprint set.

    The Associate Footprint Set to Other Matching Parts window opens.
11. Select the parts that need to be updated with this modified footprint set.
12. Click the *Associate Footprint Set to Other Matching Parts* button.



Type	Name	Revision	Lifecycle Status	Modified By Username	Originator
Part	CDN-CAP-0043	1.0	Released	library_admin	library_admin
Part	CDN-CAP-7002	1.0	Released	library_admin	library_admin
Part	CDN-CAP-7003	1.0	Released	library_admin	library_admin
Part	CDN-CAP-0023	1.0	Released	library_admin	library_admin
Part	CDN-CAP-0024	1.0	Released	library_admin	library_admin
Part	CDN-CAP-0025	1.0	Released	library_admin	library_admin
Part	CDN-CAP-0026	1.0	Released	library_admin	library_admin
Part	CDN-CAP-0027	1.0	Released	library_admin	library_admin

13. Open the View Footprint Set window.
14. Expand the footprint node for which the alternate footprint relation was modified.

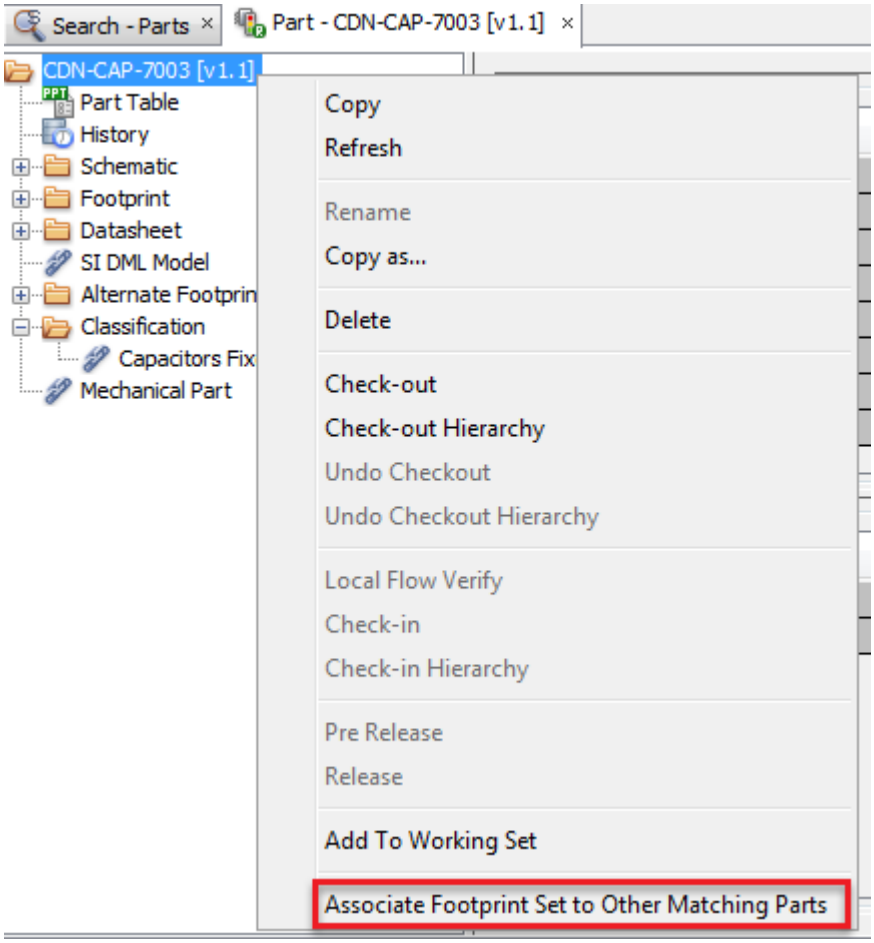
    In this example, a new alternate footprint is added to the set and this set is associated with two other parts.

Associating New or Modified Footprint Sets with Other Parts

To associate modified footprint sets with other parts, do the following:

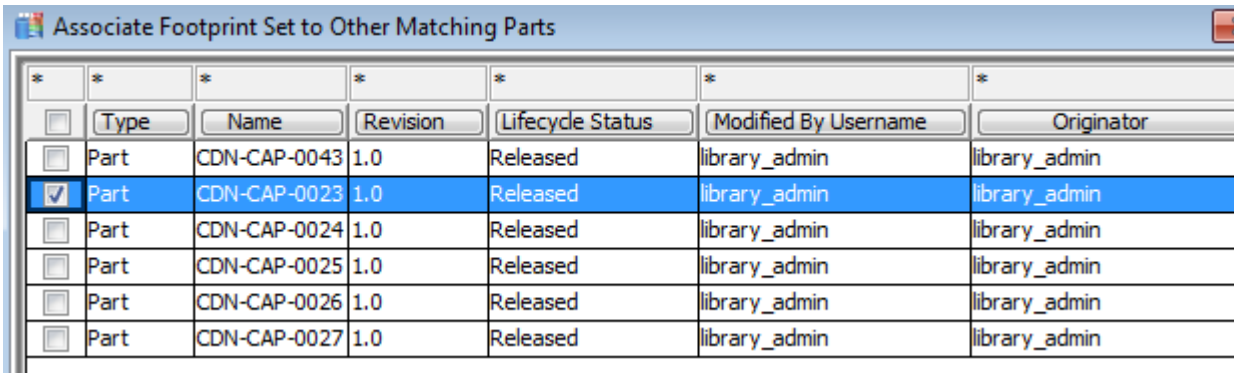
1. Open Database Editor.
2. Search, select, and open a part from the *Search Results* tab.

3. Right-click the part node and select *Associate Footprint Set to Other Matching Parts.*



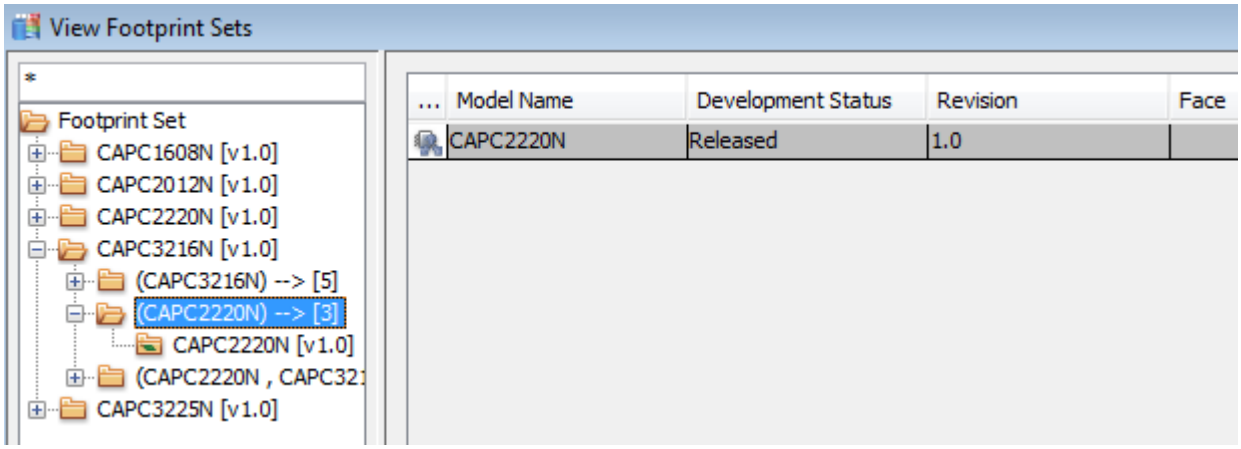
The Associate Footprint Set to Other Matching Parts window opens.

4. Select the parts that need to be updated with the footprint set associated with the selected part.

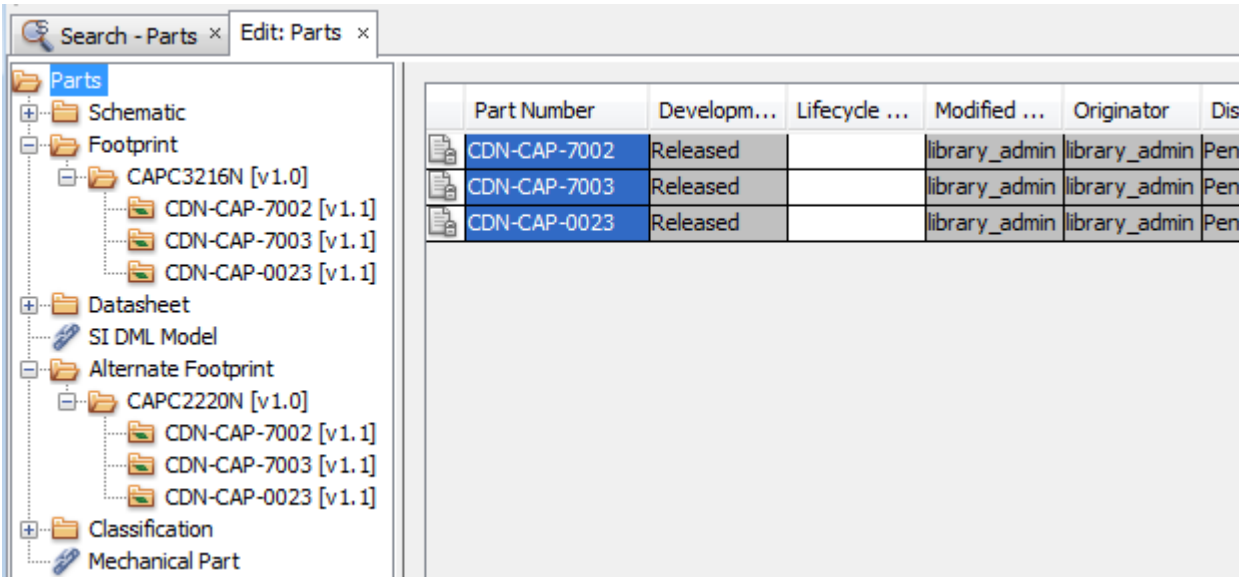


5. Click the *Associate Footprint Set to Other Matching Parts* button.
6. Open the View Footprint Set window.
7. Expand and observe the footprint set node that was associated with other parts.

   In this example, the number of released parts adjacent to the alternate footprint has increased by one.



8. Select the alternate footprint subnode and click *Show Linked Parts* button.
   In this example, the part that was selected for updating its footprint set is seen in the right pane in the *Edit - Parts* tab.



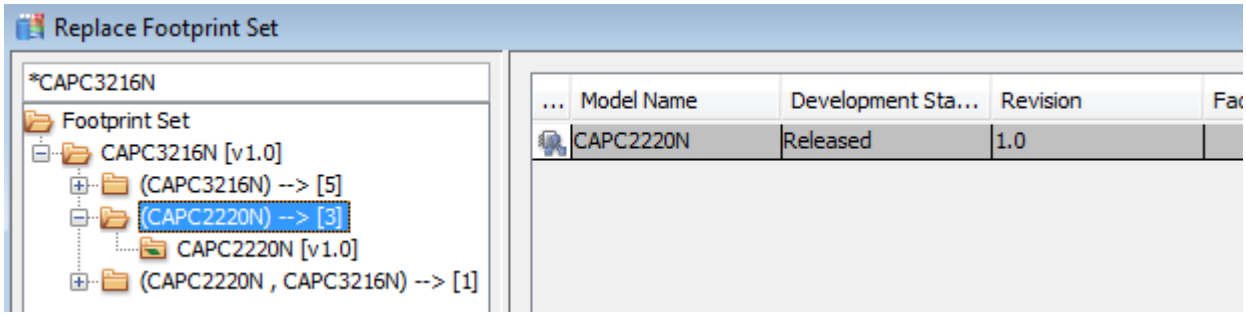
Replacing Footprint Set

To replace a footprint set, do the following:

1. Open Database Editor.
2. Search, select, and check out a part from the *Search Results* tab.
3. Right-click the *Footprint* relation node.
4. Select *Replace Footprint Set*.

   The Replace Footprint Set window opens.

5. Select the required footprint set and click *Associate*.



The selected footprint set replaces the footprint set in the checked out part.

6. Save the part.
7. Check in and release the part.
8. If the footprint set that was associated with the selected part, was also associated with other parts, a message appears to confirm if you want to replace the footprint set in those parts.

Data and Development Status Tasks

Database Editor is a application from where you can perform the component database transactions such as check in, check out, and undo checkout on the parts and block parts. These tasks, in turn, relate to lifecycle management transactions (`Verify`, `Flow Verify`, `Pre Release`, `Release`), which you can also manage using Database Editor. This section includes the following tasks:

- Configuring Data and Lifecycle Management Options
- Checking in Parts
- Verifying Parts Locally
- Prereleasing a Part
- Releasing a Part
- Checking Out Part
- Cancelling Checkout
- Running Library Flow on a Part

Note: Lifecycle management transactions can also be performed using Flow Manager.

Configuring Data and Lifecycle Management Options

Before you start working with the component database, you can configure the behavior of the commands such as check in and check out. To do so:

1. Choose *Edit – Options*.

   The Options dialog box appears.

2. Select the *Commands* tab.

3. The five configurable commands are: Check-out, Get Copy, Copy As, Check-in, and Open Part/Model Details.

4. In the *Checkout command* section:

   a. Select the *Overwrite local work area* check box to overwrite the local work area on running a checkout operation.

   b. Select the *Uprev to current tool version* check box to update model to the current application version installed on your computer on a checkout operation.

5. In the *Get Copy command* section:

a. Select the *Overwrite local work area* check box to overwrite the local work area on running a *Get Copy* command.

b. Select the *Uprev to current tool version* check box to update model to the current application version installed on your computer on running a *Get Copy* command.

c. Select the *Set selected model active* check box to set the selected model as active on running a *Get Copy* command.

Note: If multiple models are selected, the last selected model is set active.

6. In the *Copy As command* section:

a. Select the *Overwrite local work area* check box to overwrite the local work area on running a *Copy As New* command.

b. Select *the Uprev to current tool version* check box to update model to the current application version installed on your computer on running a *Copy As* command.

c. Select the *Set selected model active* check box to set the selected model as active on running a *Copy As* command.

Note: If multiple models are selected, the last selected model is set active.

7. In the *Check-in command* section:

a. Select the *Remove local copy* check box to remove the local work area copy on a successful check-in operation.

b. Select the *Check in local submodels linked to model* check box to always check in model along with its newly linked submodels on a check-in operation.

c. Select the *Remove links to unused submodels* check box to remove all the submodels from a Model which are no longer used on a check-in operation.

d. Select *Release models on successful check in* check box to release the model as soon as it is checked in. The model is automatically released if the check-in is successful. If the checkin or release operation fails, the model remains in the checked-out state.

8. In the *Release command* section, select *Release linked prereleased objects*, if you need to release the prereleased objects that are linked to the part or the model being released.

9. In the *Open part/model details* section, select *Expand classification node*. If you select this option, the classification node will be automatically expanded in the *<Part/Model_Detail>* tab.

10. In the *Search options* section:

a. Select the *Show search toolbar* check box to display the search toolbar.

b. Select the *Show only part properties selected by administrator* check box to see only the part properties selected by the database administrator in the *Search Results* tab.

c. Select the *Show only model properties selected by administrator* check box to see only the model properties selected by the database administrator in the *Search Results* tab.

11. Click the *General* tab to configure the table column order using the *Move Up* and *Move Down* buttons.

12. Click *OK* to close the Options dialog box.

Checking in Parts

When you create a part or block part, you can check it into the component database so that it becomes a part of your repository, and is available to other librarians.

To check in a part, do the following:

1. Open a *Part* - `<Part_Detail>` tab.

2. Do one of the following:

- Choose *Edit – Check-in*.
- Click  on the toolbar.
- Right-click the part node and select *Check-in*.

The Check-in window opens. The part is checked in and verified for the check-in rules specified, if any, by Rule Manager.

If the part, block part, or model is modified, depending on the magnitude of change, the librarian decides if the check in is major or minor. After the check in, the Check-in Log for `<Part, Block Part, or Model Type>` window appears that allows you to select the change. Thus, after the check-in operation is successful, you need to:

1. Choose *Major* or *Minor.*

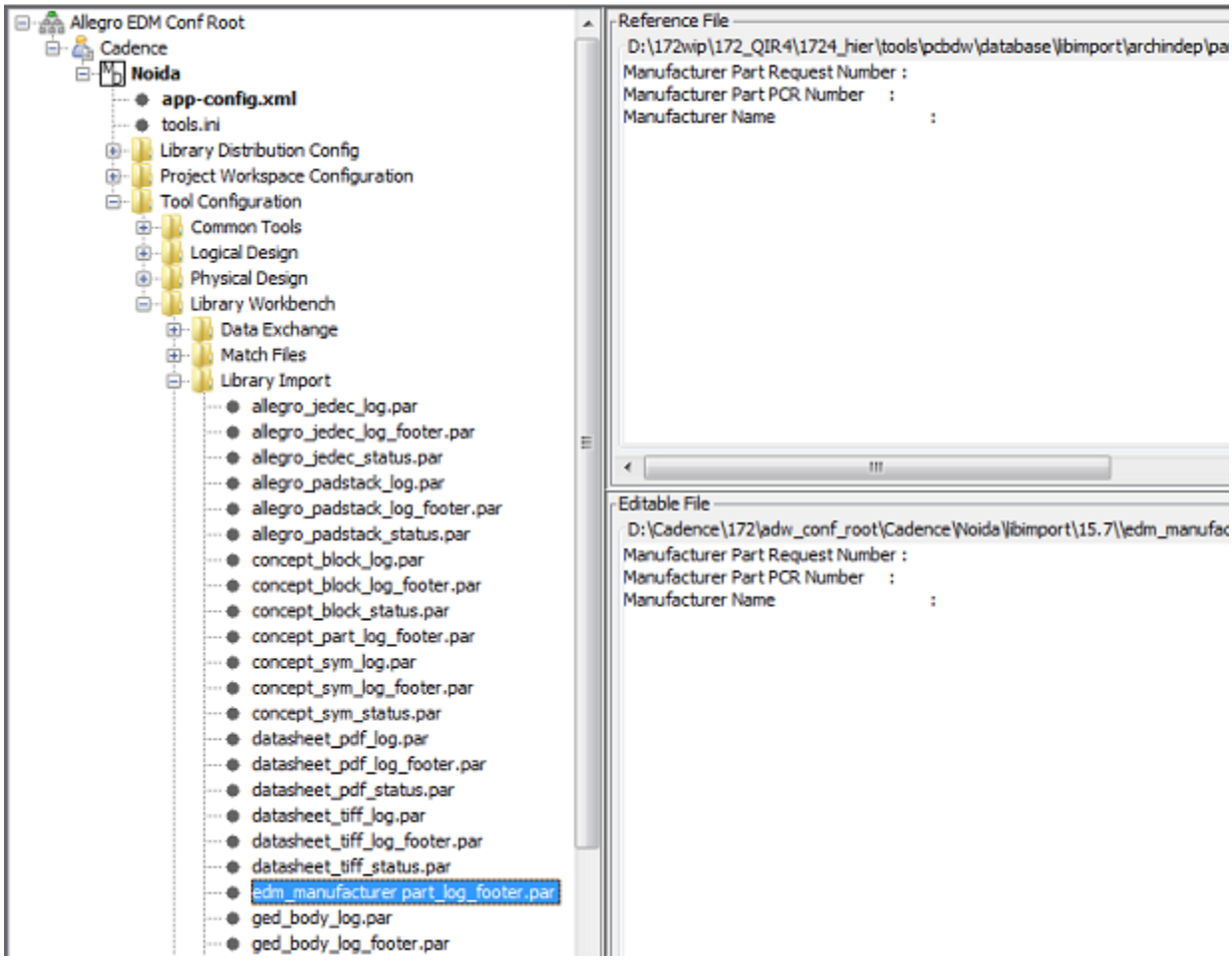
2. Update the log for the part, block part, or model.

The `Checked-In & Verified` state is symbolized by the  , and this icon appears in the part, block part, or model tab. The *Development Status* of the part changes to `Checked-In & Verified`.

You can also customize the Check-in Log window. To do so:

- Copy the required `.par` file from *<SPB_installation_directory>*`\tools\pcbdw\database\libimport\archindep\par` to *<adw_conf_root>*`\`*<company>*`\`*<site>*`\libimport\15.7`, modify it as required, and save it.
- Open Configuration Manager and perform the following tasks:
  - **a.** Select *Set up or Manage Company Site*.
  - **b.** Navigate to *Allegro EDM Conf Root – <company> – <site> – Tool Configuration – Library Workbench – Library Import*.
  - **c.** Select the required `.par` file.

d. Modify the *Editable File* section in the right pane as required.



e. Click the *Save* button.

Verifying Parts Locally

If you need to locally verify a part or block part without changing its state for the purpose of quick testing or to verify the development of the part or block part, you can use the Local Flow Verify option.

1. Open the *Part* - `<Part_Detail>` tab for a checked-in part.
2. Check out this part.
3. Choose *Edit – Local Flow Verify* to verify the part.

   An alert appears asking if you want to verify the part.

4. Click *Yes*.

   The Local Flow Verification window shows the verification rules and their status.

Note: The Local Flow Verify command is available only when the part or block part is in a preliminary or checked-out state.

Prereleasing a Part

After you have completed the design and verification of a part or block part, you can prerelease it into the component database. `Pre Release` is an intermediate lifecycle state of a part or block part, where it has been verified but a front-to-back verification has not been performed.

This is specially helpful when your part is not completely ready and it has no impact on the front-to-back design flow. For example, you can prerelease a part when front-end models are ready but the back-end models are still being worked upon. This helps you reduce the time-to-release and distribute the part to designers.

To prerelease a part, do the following:

1. Open the *Part - `<Part_Detail>`* tab for a checked-in part.
2. Do one of the following:
   - Choose *Edit – Pre Release*.
   - Click  on the toolbar.
   - Right-click the part node and select *Pre Release*.

Note: The *Pre Release* command is available only when the part is in `Checked-In & Verified` state.

3. Click *Yes.*

The `Pre Released` Development Status is symbolized by  and this icon appears in the the part tab.

Releasing a Part

After you have completed the design and verification of parts and block parts, you can release them to the component database. This process runs the front-to-back verification rules.

Ensure that the *Part - `<Part_Detail>`* tab is open and the part has already been checked in. If you are not using flow verification, use any one of the following ways to release the part:

- Choose *Edit – Release*.
- Click  on the toolbar.
- Right-click the part node and select *Release*.

The `Released` Development Status is symbolized by an icon (), which appears in the part tab.

Important

Prereleased and released parts are valid states for distribution. During the library distribution process, all parts belonging to either of the states, are distributed. The Distribution lifecycle stage is achieved when you run the library distribution process for the components, successfully. For more information on this, see *Allegro EDM Library Distribution User Guide*.

You can also use the Flow Manager.

- On clicking *Pre-Release* in Flow Manager, all valid parts and models that need to be prereleased are displayed.
- On clicking *Release* in Flow Manager, all the parts and models that are ready to be released are displayed.

Checking Out Part

To check out a part, do the following:

1. Search for a part. For information on searching, see Searching Parts and Models.
2. Open the *Part - `<Part_Detail>`* tab.
3. Do one of the following:
   - Choose *Edit – Check-out*.
   - Click  on the toolbar.
   - Right-click the part node and select *Check-out*.

   On successful checkout,  icon appears in the part tab.
4. Edit the part according to your requirements.

Note: After being checked out, the part is locked in the component database for other librarians to use. However, designers are not impacted by this checkout.

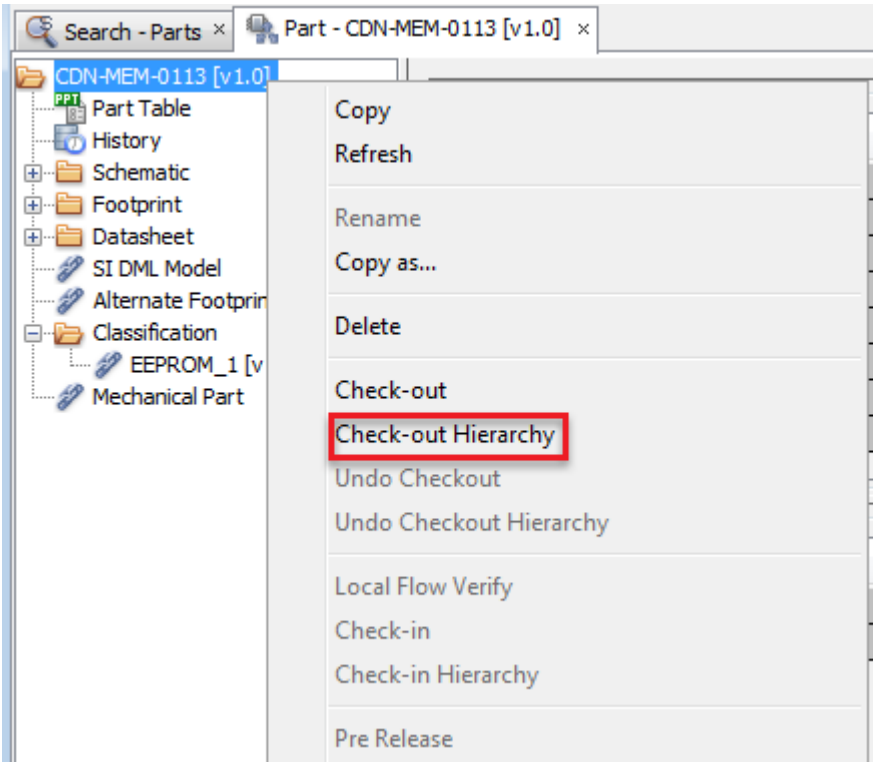
Important

By default, checked-out parts, block parts, and models are available in your current working set. If there is no current working set, then an alert appears asking you to create or select an existing working set before the checkout operation.

Checking Out Part with Linked Models

Use the *Check-out Hierarchy* option to check out the part with its associated models. To do so:

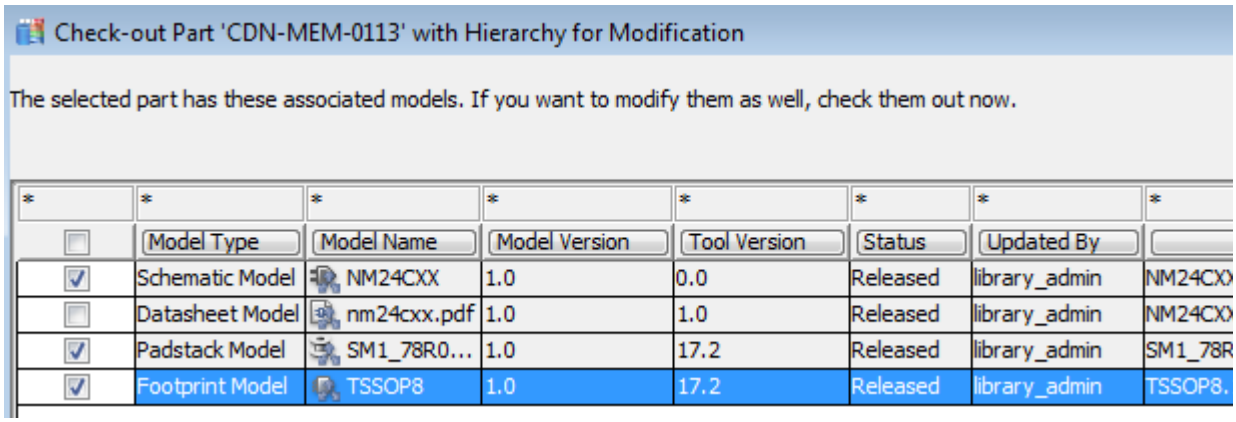
1. Search for a part.
2. Open the *Part - <Part_Detail>* tab.
3. Use any of the following ways to check out a part with its linked models:
   - Select *Edit – Check-out Hierarchy*.
   - Right-click the part node and select *Check-out Hierarchy*.



On successful checkout, the following icon ( ) icon appears in the part tab.

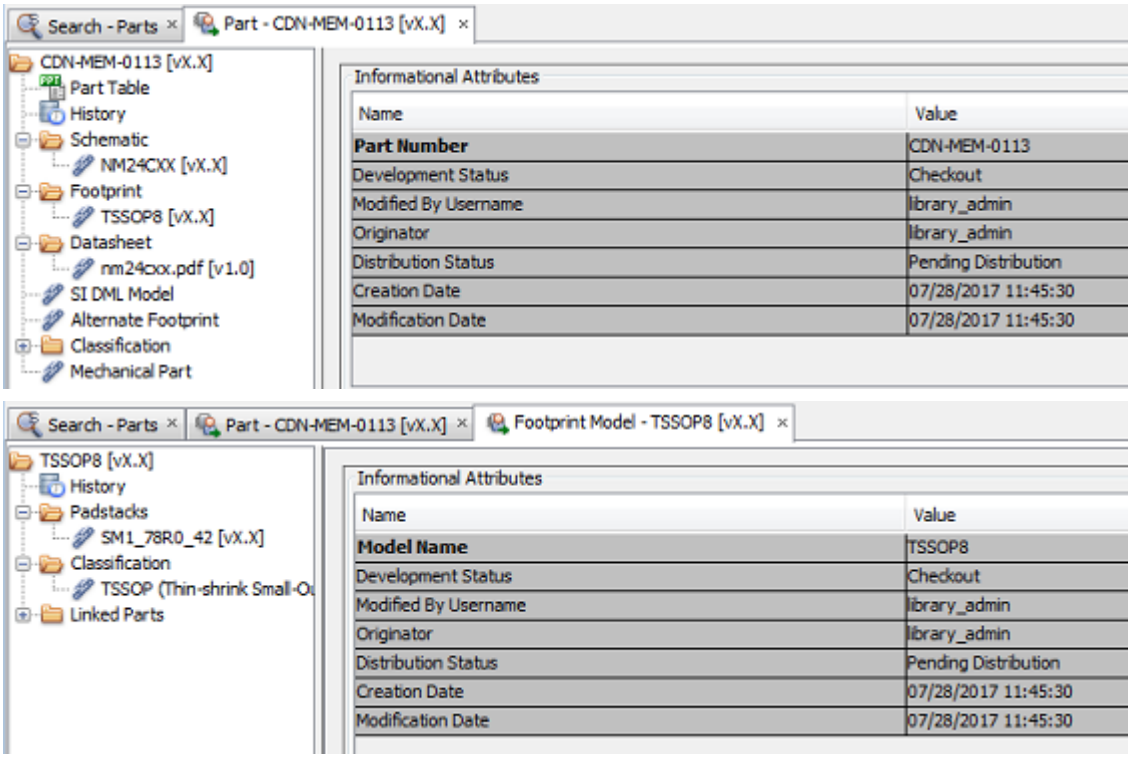
The Check-out Part '*<part number>*' with Hierarchy for Modification window opens.

4. Select the associated models that you want to check out.



5. Click *OK*.

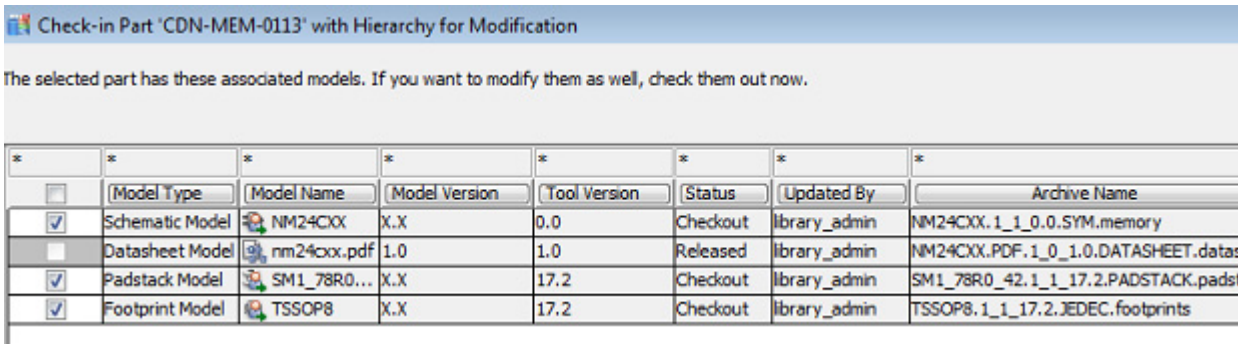
6. Observe that the models you selected for modification are checked out.



7. After doing the changes, right-click the part node and select *Check-in Hierarchy*.

   The Check-in Part '`<part number>`' with Hierarchy for Modification window opens.

8. Select the associated models to be checked in and click *OK*.



The Check-in Log for Model window appears followed by the Check-in Log for Part window.

Cancelling Checkout

To undo or cancel a checkout, do one of the following:

- Choose *Edit – Undo Checkout*.
- Click the *Undo Checkout* button ( ) on the toolbar.
- Right-click the part node and select *Undo Checkout*.

  You can use the *Undo Checkout Hierarchy* option if you have used the *Check-out Hierarchy* option for the checkout operation.

The checkout is canceled, and the part is restored in its earlier version (before the checkout).

Important

The undo checkout operation deletes the latest revision (that is version X.X) of the part or model because revisions are created in the checkout operation.

Running Library Flow on a Part

In a typical library development flow, part-specific modifications are many and librarians are expected to work on many parts simultaneously. In such cases, it is cumbersome and time consuming to perform lifecycle tasks such as verification, flow verification, and release individually for each part.

To help you perform these tasks as a single operation, Database Editor provides you with the library flow support for parts. This involves the following tasks:

- Checking in Parts
- Prereleasing Parts
- Releasing Parts
- Distributing Parts

Checking in Parts

1. Choose *Edit – Library Flow – Check-in Parts.*

   The *Check-in Parts* tab appears listing all the parts that are to be checked in and verified.

2. Select the parts to check in and verify.
3. Click *Check-in*.

   The Check-in window opens. The selected parts are checked in and verified for the check-in rules specified, if any, by Rule Manager.

   If the check in is successful, the Check-in Log for `<Part>` window appears. You need to:

   a. Choose *Major* or *Minor*.

   b. Update the log for the part.

   Note: You can customize the fields in the log information section.

4. Click *OK*.

Prereleasing Parts

This involves running the front-to-back verification process on parts.

1. Choose *Edit – Library Flow – Pre Release*.

   The *Pre-Release Working Set* tab appears listing all the checked-in and verified parts.

2. Click *Pre Release*.

   If the parts satisfy the prerelease rules, they move to the next stage, *Pre Released*.

Releasing Parts

1. Choose *Edit – Library Flow – Release*.

The *Release Working Set* tab appears listing all the `Pre Released` or `Checked-In & Verified` parts to be released.

2. Select the parts to release.
3. Click *Release*.

The parts are checked for the release rules specified, if any, by Rule Manager. If the parts satisfy the release rules, they move to the next stage, *Released*.

Distributing Parts

Choose *Edit – Library Flow – Run Library Distribution*. The library distribution process starts.

Deleting Parts and Models

There are two ways in which you can delete parts, block parts, mechanical parts, and models from the Allegro EDM Component Database:

- Delete: If you select this delete operation, the selected part/model is only marked for deletion and becomes invisible to the designer after library distribution. You can undo this deletion by checking out this part/model and then releasing it again.
- Permanent Delete (Purge): If you select this delete operation, the selected part/model is marked for permanent deletion from the Allegro EDM Component Database, and is then not available for use by librarian or designer. When you run library distribution after selecting this type of delete operation, all the versions of this part/model are permanently removed from database and all its associated archives are deleted from the vault area.

  After you permanently delete a part/model:

  - You cannot recover it from the component database.
  - All versions of the part/model are removed from the database.
  - The part/model is removed from all working sets.
  - The associated replacement part does not appear for any permanently deleted part in the replacement part report launched from LRM.
  - The archives containing model files are removed from vault area for all versions of the model.
  - The part/model is created again when imported from an external system using Data Exchange.

In addition, for any type of delete operation, the following results also hold true:

- The part/model is not available to the designers and the designs using this part/model will be unusable.
- Any locally cached copies of the part/model in the designs used by designers is not removed. LRM will show such a part/model as missing in reference when launched on design.
- The part is not removed from the shopping cart.

Deleting Single Objects

To delete a part, do the following:

1. Open the part.

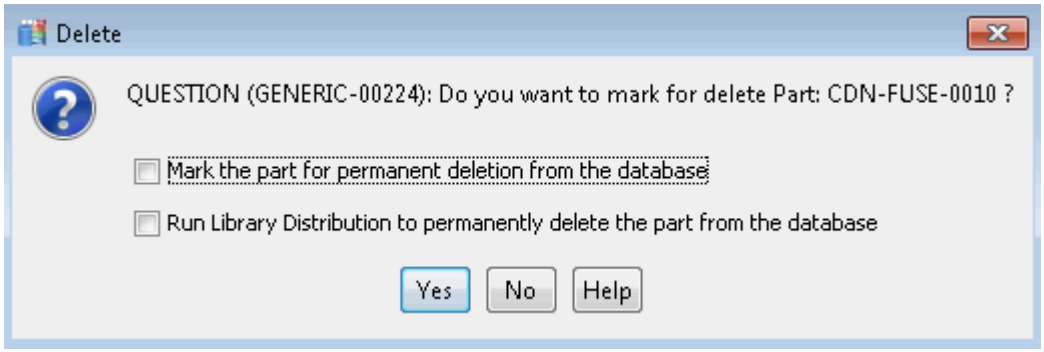
   To open a block part, choose *File – Open – Block Part*.

   To open a mechanical part or kit, choose *File – Open – Mechanical – Mechanical Part or File – Open – Mechanical – Mechanical Kit*.

   To open a model, choose *File – Open – Model –* `<Model_Type>`.

2. Choose *Edit – Delete*.

3. If the selected part is in the `Released`, `Pre Released`, or `Deleted` state, the following message appears:



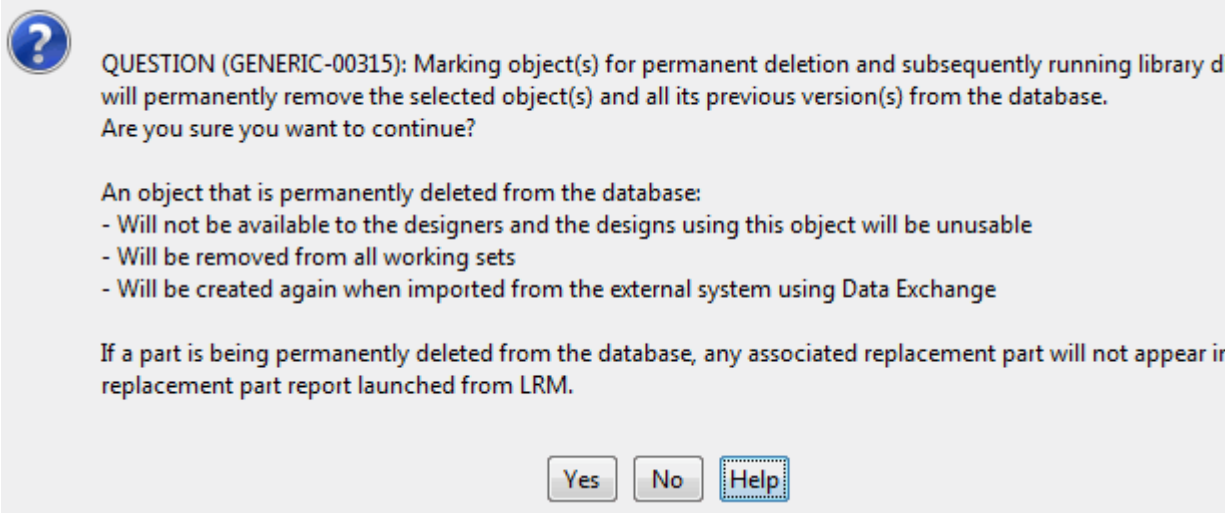
- ❑ If you want to just mark the part for deletion, click *Yes* without selecting any of the options that follow.
- ❑ If you want to permanently delete the part, do the following:

   a. Select *Mark the part for permanent deletion from the database*, if you want to mark the selected part for permanent deletion. The selected part and all its versions will be permanently deleted from the Allegro EDM Component Database on the next run of library distribution.

   b. Select *Run Library Distribution to permanently delete the part from the database*, if you want to run library distribution immediately after marking the part for permanent deletion.

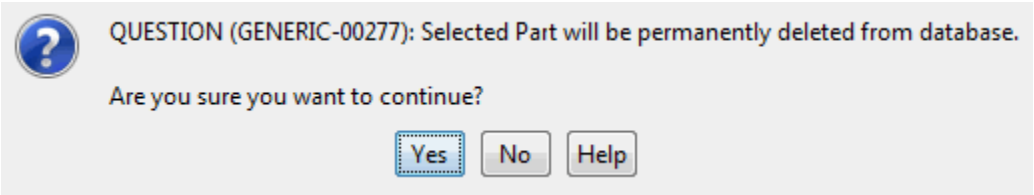
   c. Click *Yes*.

Another message appears to explain the consequences of deleting the part or model permanently. Click Help to see the detailed message.



d. Click *Yes* to confirm that you want to permanently delete the part or model.

4. If the selected part is in the `Preliminary` or `Checked-In & Verified` state, the following message appears:



Click *Yes* to permanently delete the part from the database.

Note: Mechanical part cannot be deleted if it is linked to any electrical part or a mechanical kit. Schematic models and Footprint models can only be deleted if they are not linked to any part.

The outcome of deletion depends on the Development Status as explained in the following table:

Table 2-1 Development Status of Object and Outcome of Deletion

Development Status of the object	When the object is deleted...	Status of the object after library distribution is run...
Preliminary	It is permanently deleted from the Allegro EDM Component Database. If you are deleting a model, any related model archive is deleted from the work area.	The object is deleted even before running library distribution.
Checked-In & Verified	The selected version of the object is permanently deleted from the Allegro EDM Component Database. If a model is deleted, then its related model archive is deleted from the integration area.	The object is deleted even before running library distribution.
Pre Released and Released	If you do not select the *Mark the part for permanent deletion from the database* option, the selected object is only marked as deleted in the database. This means it is still in the database with its *Development Status* as `Pre Released or Released` and *Distribution Status* is changed to `Pending Delete`.	After you run library distribution: The deleted part is removed from the Part Table File (PTF). The deleted model is removed from the reference area. However, the deleted model archive remains in the vault area. The *Development Status* is now changed to `Deleted` and the *Distribution Status* is changed to `Distributed`. To undo this delete operation, you can again check out and release the deleted object.

Development Status of the object	When the object is deleted...	Status of the object after library distribution is run...
Pre Released, Released or Deleted	If you select the *Mark the part for permanent deletion from the database* option, the selected object is marked for permanent deletion from the database. This means it is still in the database with its *Development Status* as `Pre Released`, `Released`, or `Deleted`. The Distribution Status of the selected object is changed to `Pending Purge`.	After you run library distribution: The deleted part is removed from the Part Table File (PTF) and all its versions are removed from the Allegro EDM Component Database. You will no longer be able to see this part in the database. The deleted model is removed from the reference area and all its versions are removed from the Allegro EDM Component Database. You will no longer be able to see this model in the database. In addition, the model archives of all versions of the model are deleted from the vault area. When you select *Refresh* from the pop-up menu of the object detail node, a message appears to indicate that the object does not exist in the database. You can undo this permanent delete operation only if the next library distribution process has not been run. To do so, check out the deleted-marked object and release it again. If library distribution is completed after marking any object for permanent deletion, you can never recover the object.

Deleting Multiple Objects

To delete multiple parts:

1. In the *Search Results* tab or *Working Set* tab, select multiple parts using `Shift+Click`.

2. Right-click and choose *Edit All*.

   The *Edit: Parts* tab appears listing all the parts.

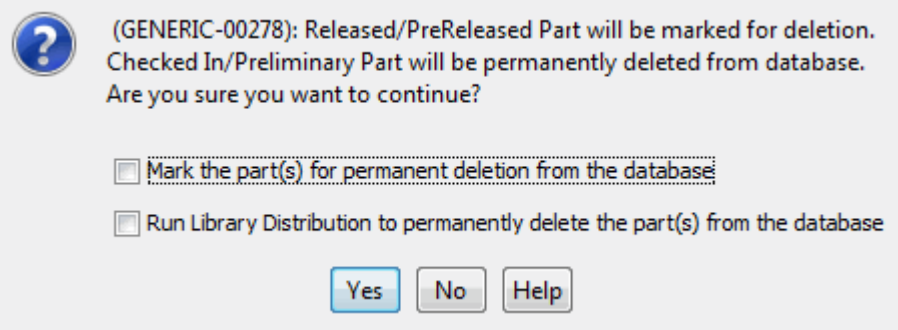
   If you are deleting models, the *Edit: Model* tab appears.

3. Choose the parts to be deleted from the grid in the *Edit: Parts* tab.

4. Choose *Edit – Delete*.

5. Depending on the Development Status of the selected parts, the message to confirm deletion appears.

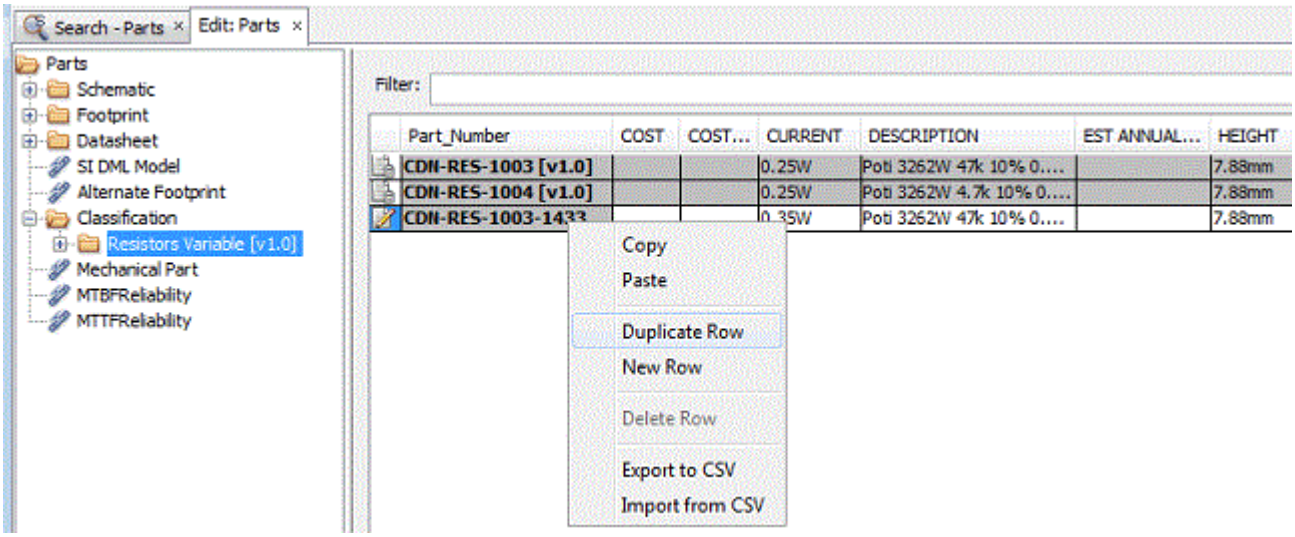
   - If the selected parts are only in the `Released`, `Pre Released`, or `Deleted` state, see step 3 in Deleting Single Objects.
   - If the selected parts are only in the `Preliminary` or `Checked-In & Verified` state, see step 4 in Deleting Single Objects.
   - If the Development Status of one of the selected parts is `Preliminary` or `Checked-In & Verified,` and that of any other part is `Released`, `Pre Released`, or `Deleted`, the following message appears.



   Choose the desired option and click *Yes*. The outcome of deletion depends on the Development Status of the selected part and is described in the table, Development Status of Object and Outcome of Deletion.

Deleting Duplicate Object Rows in Multi-Edit Mode

If you create a duplicate object row, modify and save it, a new preliminary object is created in the database.



If however, after creating a duplicate object row, you modify this row and right-click it and choose *Delete Row*, this row is deleted.

Specifying Replacement Parts

You can associate a *Replaced By* relation with a deleted part or block part. A replacement part or block part can be added for a part with either *Distribution Status* as `Pending Delete` or *Development Status* as `Deleted`.

To add a replacement part:

1. Open a library project.
2. Start Database Editor.
3. Select a part with *Distribution Status* `Pending Delete` or *Development Status* `Deleted`.
4. Choose *Edit – Add Replacement Part*.

   The *Associate Replaced By* dialog box opens.
5. Specify a replacement part.
6. Click *Associate*.

   The *Replaced By* relation is shown in the deleted part and the *Replaces* relation in the part that replaces the deleted part.
7. Run `lib_dist`.

   The deleted parts get removed from the reference PTF.
8. Open a board design project that used the deleted part.
9. Library Revision Manager (LRM) runs and detects the missing part.

   You can get a *Replaced Part* report from LRM.

Working with Models

The use model of operations in Database Editor is similar for parts and models in the Allegro EDM component database. Basic tasks you can perform with models are:

- Creating models
- Editing models
- Deleting relation instances
- Performing data and lifecycle management tasks

Important

The steps to perform these tasks are similar to the tasks performed for parts. See, Working with Parts.

The following tasks are specific to models:

- Setting a Model as Active
- Working with the Explorer Tab
- Working with Schematic Models
- Working with Footprint Models
- Working with Padstack Models
- Working with Models Marked as “Update(s) Required”
- Working with Datasheet Models
- Working with Capture Models

Setting a Model as Active

Setting a model as active ensures that all library development operations in the Flow Manager interface (using the library flow tree) are performed on the active model only.

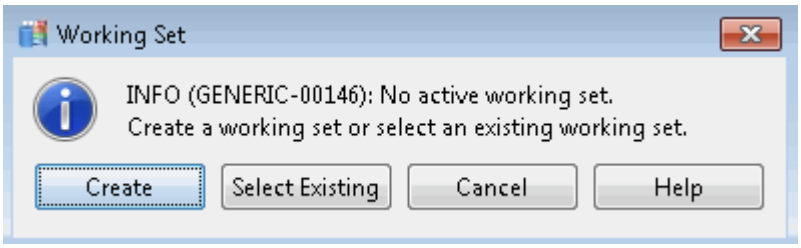
To set a model as the active model, do the following:

1. In the Database Editor window, select a model.

   You can select a model from the *Search Results* or *Working Set* tabs.

2. Choose *Edit – Set Active*.

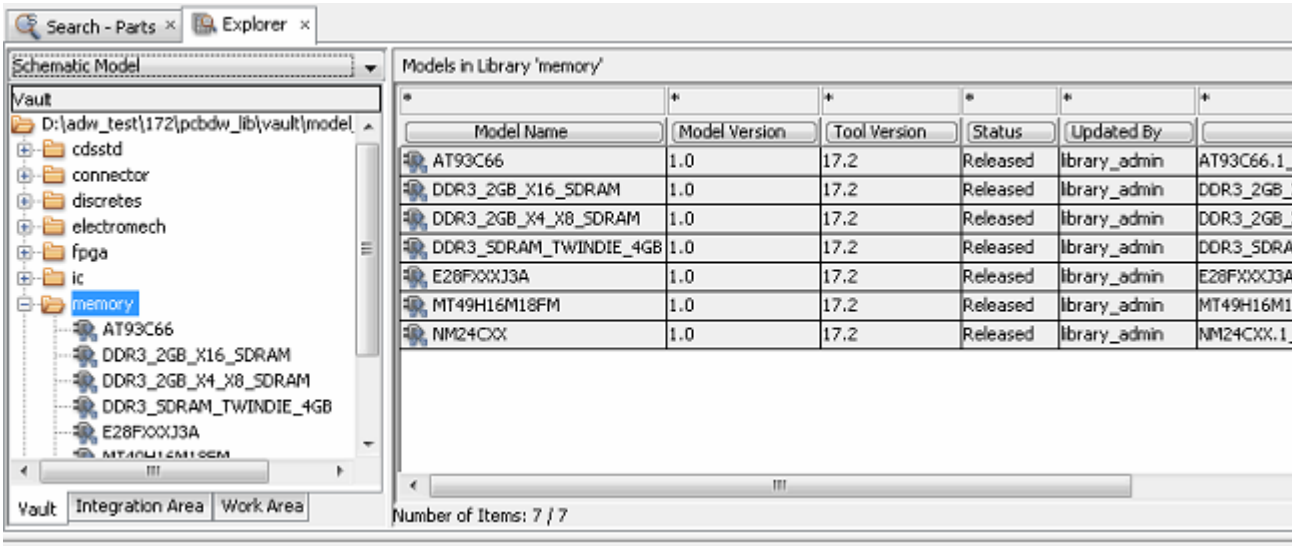
If you try and set a model as the active model without checking it out, that is, the model is not in your Working Set, the following message appears:



If you click Check-out in such cases, the model is checked out. If you click Get Copy, the model is copied to your work area. When the model is copied to your work area, you cannot check in this version of the model and it is only for local use.

Working with the Explorer Tab

The *Explorer* tab helps you manage all library development tasks using a single interface. To launch the *Explorer* tab, click  on the toolbar.



The *Explorer* tab gives you a view of the Vault, Integration Area, and Work Area, which represent different views of the library development area. The details of these areas are:

- **Vault**: A tree-structured file folder that contains the archived models. In the Vault, there are archived files that contain Allegro models, schematic models, and other models supported by Allegro EDM.

- **Integration Area**: An intermediate area where the archived models are stored until they pass through the verification process. After they are verified and released, the model archive files are moved to the Vault.
- **Work Area**: A local area where the checked-out models are stored until they go through the edit process and are checked in. After models are edited, verified, and checked in, Allegro Library Manager creates and moves the model archive files to the Integration area.

For more information about the physical location of these library-specific areas, see *Allegro EDM Library Import User Guide*.

All the area tabs (*Vault*, *Integration Area*, and *Work Area*) display the libraries in a tree view in the left pane. The display of model libraries depend on the model type, selected in the drop down list above the left pane.

The right pane in the *Explorer* tab contains the view-only details of the libraries and models selected in the left pane. These details include: Model Name, Model Version, Tool Version, Status, Updated By, Archive Name, Library Name, and Model Type.

Important

The *Explorer* tab uses many model-specific icons to communicate the state of models in the component database. For more information on the icons and what they represent, choose *Help – Help on Icons* in the Database Editor window.

Using the Explorer Tab

You can perform tasks using the context-sensitive pop-up menu in the *Explorer* tab. The pop-up menu commands available depend on the library area (Vault, Integration Area, or Work Area) you have selected. The components of the *Explorer* tab are:

- Vault Tab
- Integration Area Tab
- Work Area Tab

Vault Tab

The following table lists the pop-up menu commands and the tasks you can perform in the Vault area.

Command	Allows you
Set Active	To set a model as active in Flow Manager. This command is available in all the views.
Check-out	To check out an existing model for modification.
Check-out Hierarchy	To check out an existing model along with its submodels for modification. This command is not available for schematic model.
Copy as New	To copy an existing model as new in the Work area.
Get Copy with Hierarchy	To get a copy of the model in the Work area along with its submodels. This does not change the status of the model in the database.
Get Copy	To get a copy of the model in the Work area. This does not change the status of the model in the database.
Show Errors	To view errors associated with a model. This command is available in all the views.
Show Linked Parts	To view the linked parts associated with the selected model in a separate window. This command is available in all the views.
Show Details	To view the details of the model in a new tab. This command is available in all the views.
Refresh	To refresh the model libraries.

Integration Area Tab

The following table lists the additional pop-up menu commands (other than those available in the Vault area) and the related tasks you can perform in the Integration area.

Command	Allows you
Check-out	To check out a model that was previously checked out and checked into the Integration area for updates.
Release	To release the model to the Vault area.
Release Hierarchy	To release the model along with its submodels to the Vault area.

Work Area Tab

The following table lists the additional pop-up menu items and the related tasks you can perform in the Work area.

Command	Allows you
New Sub Model	To create a new model by copying a template in the Work area
Check-in	To check in the model that was created or modified in the Work area.
Check-in Hierarchy	To check in model along with the submodels that were checked out simultaneously.
Undo Checkout	To undo the checkout of the model.
Undo Check-out Hierarchy	To undo the checkout of the model, along with the submodels (that are part of the checked out hierarchy).

The right pane in the *Work Area* tab contains the following two tabs:

- *Model Data*: This tab displays the details of the model in the Work area
- *Log Data*: This tab helps you enter a summary of ECO details for the model.

Working with Schematic Models

To create a schematic model, do the following:

1. Choose *File – New – Model – Schematic Model*.

   When a new schematic model is created, the New Schematic Model dialog box shows two tabs, *Basic* and *Advanced*.

2. Do one of the following:

   - In the *Basic* tab, choose the library and enter the name of the new model. Set the model as active and click *Create*. From Flow Manager, launch Part Developer and create the required cell.

Important

When you click *Create & Verify Test Schematic* in the Flow Specific Tools pane in the Flow Manager, if there are two symbols for a part created in Part Developer, these are instantiated on two different pages in Design Entry HDL.

   - In the *Advanced* tab, choose the library. Click *Browse* to select an existing cell. The Model Name field is populated. Set the model as active and click *Create*.

3. Check in this newly created schematic model to the database.
4. Using Database Editor, you can add parts and PTF data to the schematic model.
5. Verify the model.
6. Release the model.

Working with Footprint Models

To create a Footprint model, do the following:

1. Choose *File – New – Model – Footprint Model*.

   The New Footprint Model dialog box appears.

2. Choose the library.
3. Do one of the following:

   - Enter the model name.
   - Click *Browse* to add an already existing model to this library.

4. Ensure that you have selected the Set Model Active check box.
5. Click *Create*.
6. From Flow Manager, launch Allegro PCB Designer to work on the model.

7. Verify the model.
8. Check in and release the model.

Working with Padstack Models

To create a Padstack model, do the following:

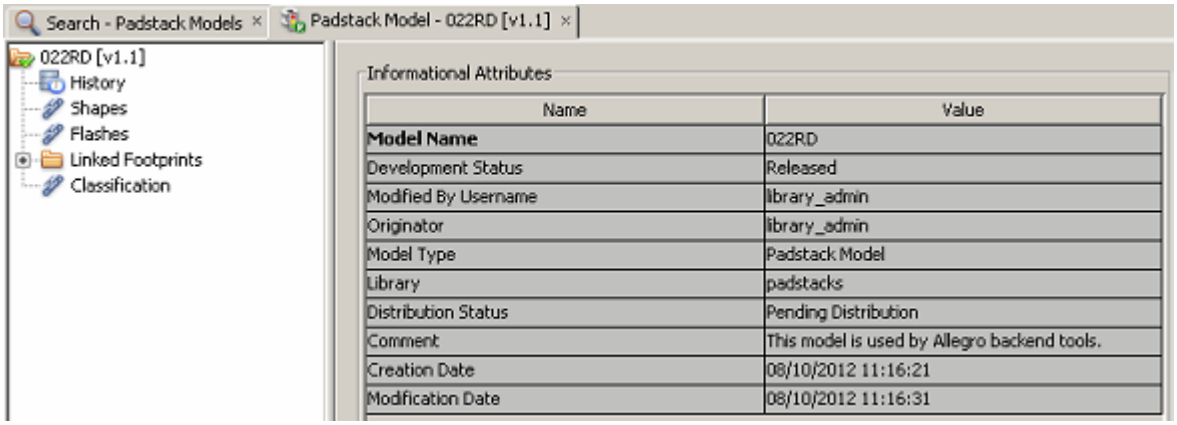
1. Choose *File – New – Model – Padstack Model*.

   When a new padstack model is created, the New Padstack Model dialog box appears.
2. Choose the library.
3. Enter the model name or click *Browse* to add an existing model that is not in the component database.
4. Ensure that you have selected the Set Model Active check box.
5. Click *Create*.
6. From Flow Manager, launch Pad_Designer to work on the model.
7. Check in the model.

Creating Duplicate Padstack Models

You can now create multiple padstack models with the same name with the condition that they are saved in different libraries.

For example, if there is an already existing padstack model with the name `022RD` in the library `padstacks`.



You can now create another padstack model with the same name in a different library. To do so:

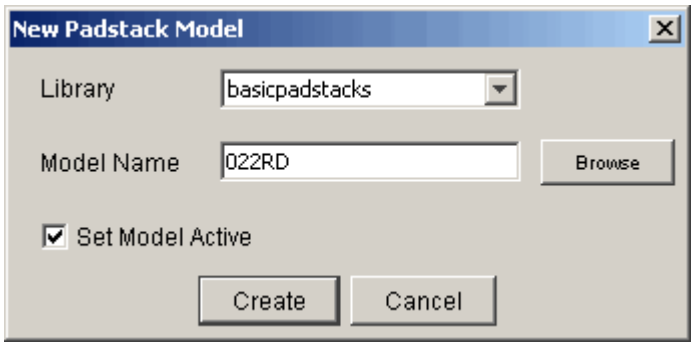
1. Choose *File – New – Model – Padstack Model*.

   The New Padstack Model dialog box appears.

2. Choose another library.

   In this example, the library `basicpadstacks` is selected.

3. Enter the same model name.



4. Ensure that you have selected the *Set Model Active* check box.
5. Click *Create*.
6. Work on the new model and check it in.

Viewing Duplicate Padstack Models

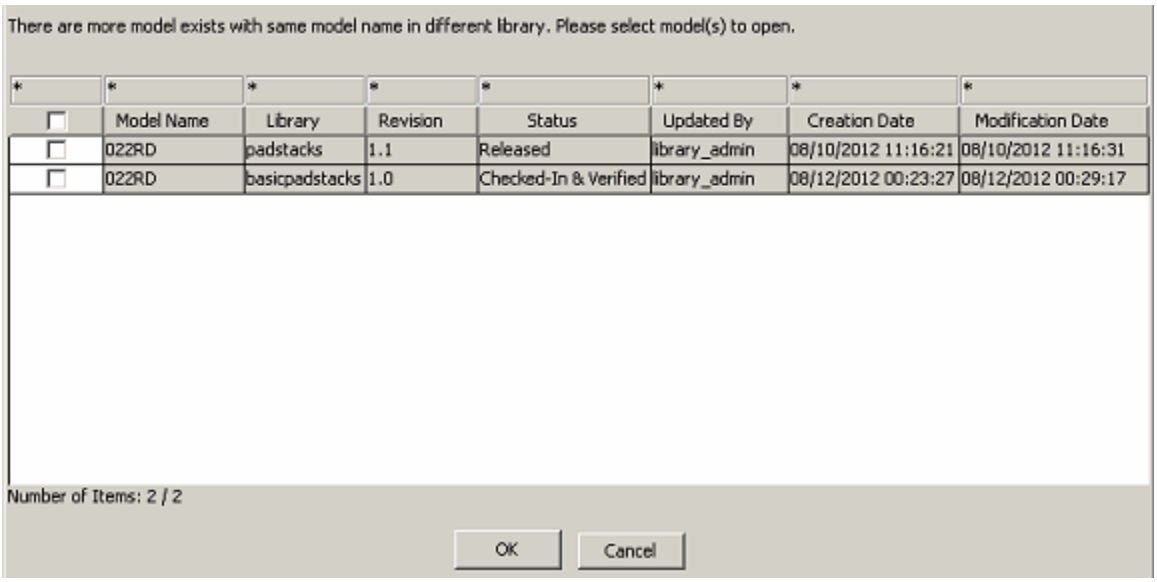
To see all the padstacks with the same name, do the following:

1. Choose *File – Open – Model – Padstack Model*.

   The Open Padstack Model dialog box appears.

2. Enter the name of the duplicate padstack model.
3. Click *Open*.

The Select Model dialog box appears.

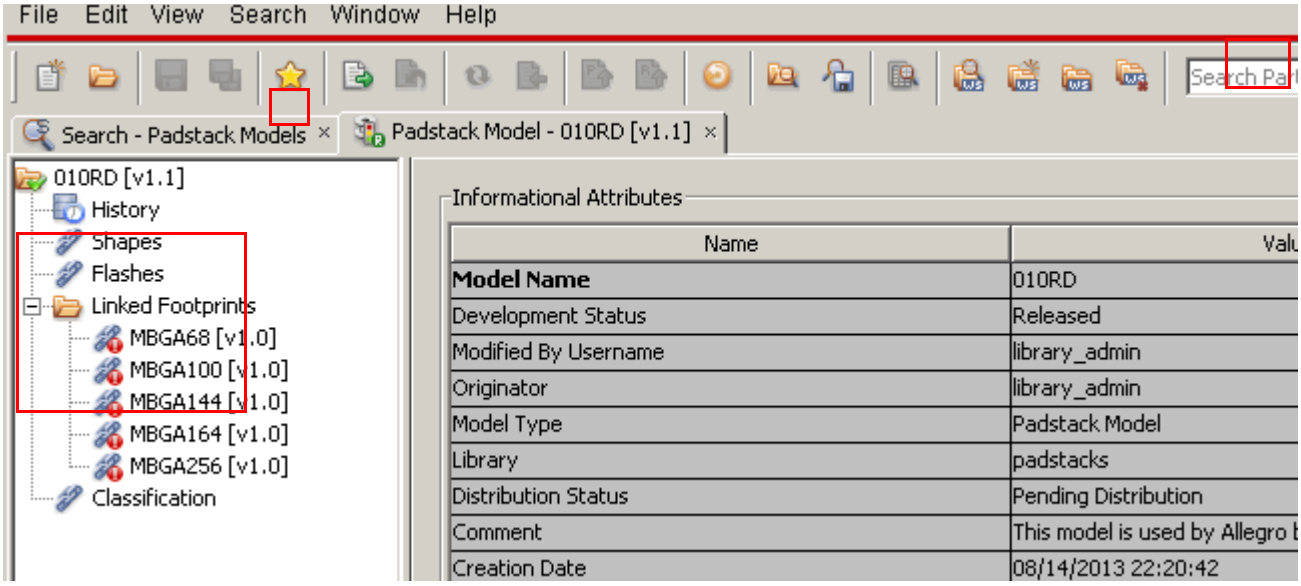


4. Choose the padstack model to open.
5. Click *OK*.

Editing Padstack Models

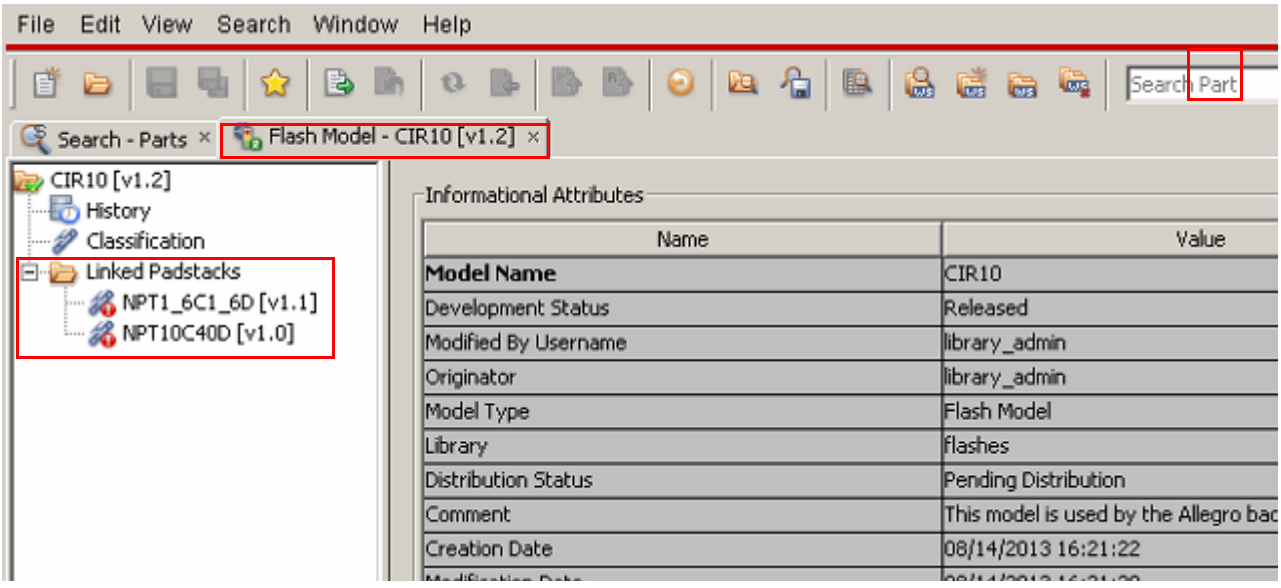
After you edit, check in, and release a padstack model, all the footprint models associated with it are marked as *Update(s) Required*. For example, `010RD` is a padstack model that has been modified, checked in, and released. All its associated footprints are now marked as

Update Required. In addition, the Show Notifications area shows the number of footprint models that need an update.



Editing Flash or Shape Models Linked to Padstacks

After you edit, check in, and release a flash or shape model, all the padstack models associated with it are marked as Update(s) Required. For example, `CIR10[v1.1]` is a Flash model, which has been modified, checked in, and released. All its associated padstack models are now marked as *Update(s) Required*. In addition, the Show Notifications area shows the number of footprint models that need an update.



Working with Models Marked as “Update(s) Required”

When a model is released or pre-released, it could have an impact on related models. For example, if a padstack is edited and a new version is released, it may have an impact on a footprint model. In such cases, when the padstack is released or pre-released, Allegro EDM Database Editor marks the linked models as requiring an update - *Update(s) Needed*.

Similarly, when you edit, check in, and release a Flash or Shape model, all the padstack models associated with it are marked as *Update(s) Required*.

The number of items that require an update is displayed as a notification in the top, right corner of the Database Editor window.

As a designer, you now have three options:

- *Accept Update(s)*: Choose this option only if you are sure that the changes to the padstack have not impacted the footprint. If you choose this option, the notification flag for the footprint is removed.
- If you choose *Edit All*, the models that are impacted by changes to the padstack are opened for editing in the *Edit: Footprint Models* tab. You can then carry out the necessary flows, such as checking the models out, checking them back in, and releasing them so that they are updated.
- If you choose *Check-out*, the models that are impacted will be checked out and displayed in the *Edit: Footprint Models* tab. You need to check the models in and release them to accept the updates. The difference between option 2 and this is that in this case, all the models that are impacted are automatically checked out.

The following table shows the relation between impacted models:

Linked model that is modified...	Model that is marked for Update(s) Required...
Padstack	Footprint, Module
Footprint	Module
Format	Module
Shape or Flash	Padstack
Mechanical	Module

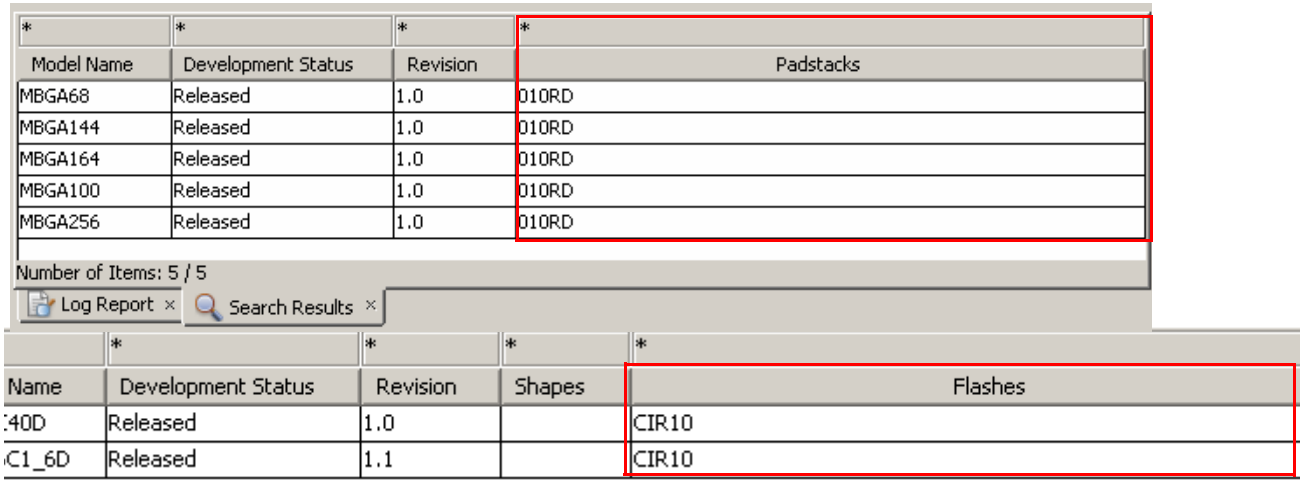
Linked model that is modified...	Model that is marked for Update(s) Required...
Schematic	Block The block model is also marked as needing an update if any linked part is modified, checked in, and released.

The following procedure considers padstacks and footprint models as an example.

To view the models that need to be updated, do the following:

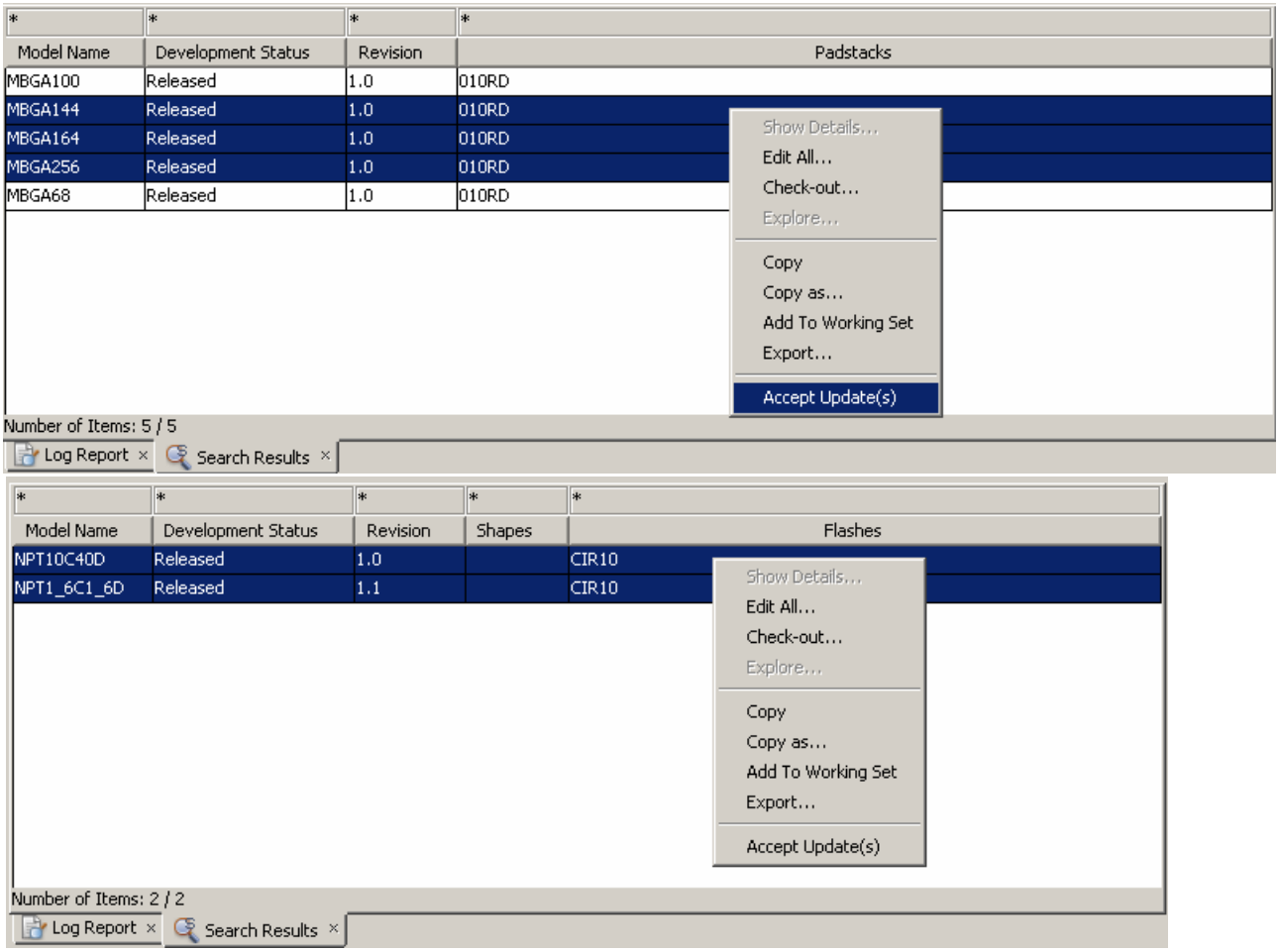
1. Right-click in the Show Notification area.
2. Choose *Update(s) Required – Show -* `<number of footprint models that need to be updated>` *Footprint Models*.

   The Search Results tab shows all the footprint models that need an update. The results also show the padstack model that was modified as a result of which the corresponding footprint models need an update.



3. Choose the footprint models to be updated.

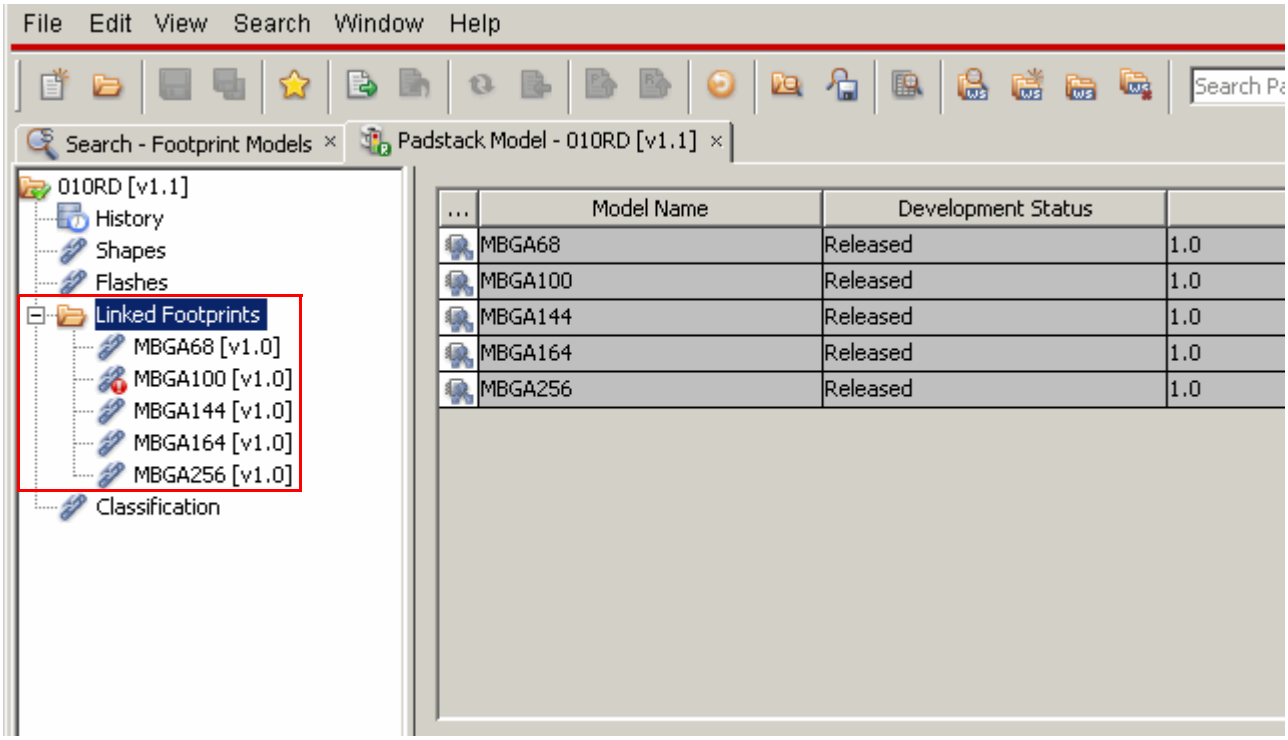
4. Right-click the selected models.



5. Choose one of the following options from the context menu:

- ❑ *Accept Update(s)*
- ❑ *Edit All*

- *Check-out*



Working with Datasheet Models

To create a Datasheet model, do the following:

1. Choose *File – New – Model – Datasheet Model*.

   The New Datasheet Model dialog box appears.

2. Choose the library.
3. Do one of the following:
   - Enter the datasheet model name with the extension.
   - Click *Browse* to add an existing datasheet model that is not in the component database.
4. Ensure that you have selected the Set Model Active check box.
5. Click *Create*.

Important

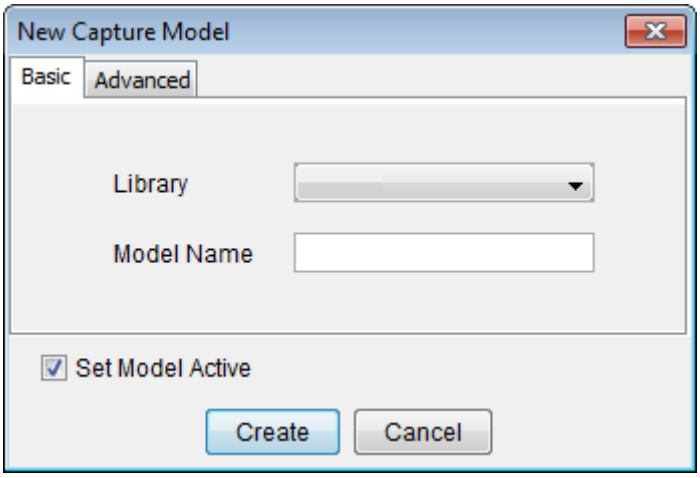
By specifying the extension name in the datasheet model name itself, you can now have multiple datasheet models with the same name but different extensions.

Working with Capture Models

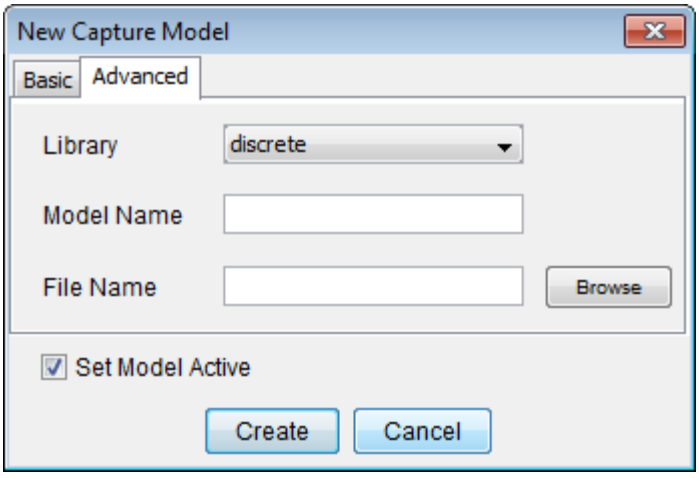
To create a Capture model, do the following:

1. Choose *File – New – Model – Capture Model*.

   The New Capture Model dialog box appears.



2. Choose the library.



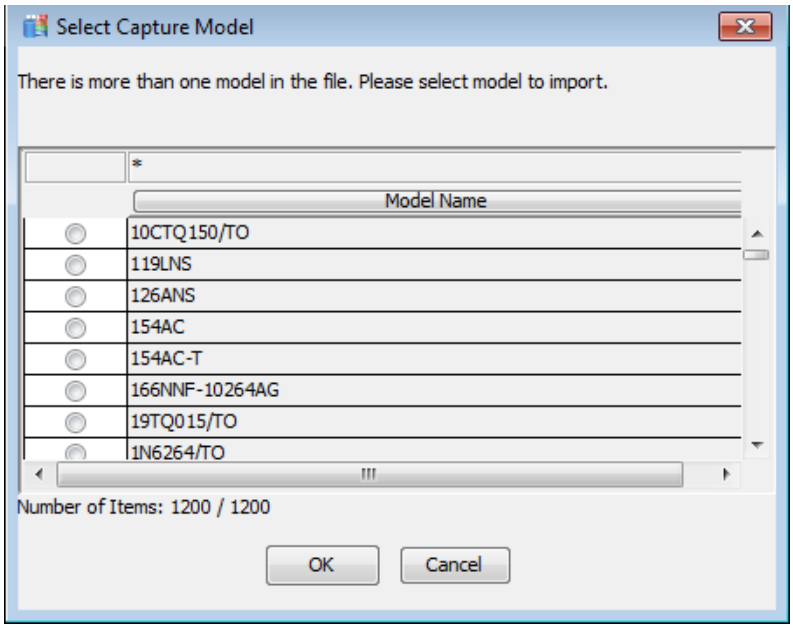
3. In the *Advanced* tab, do one of the following:

   - Enter the Capture model name with the extension.

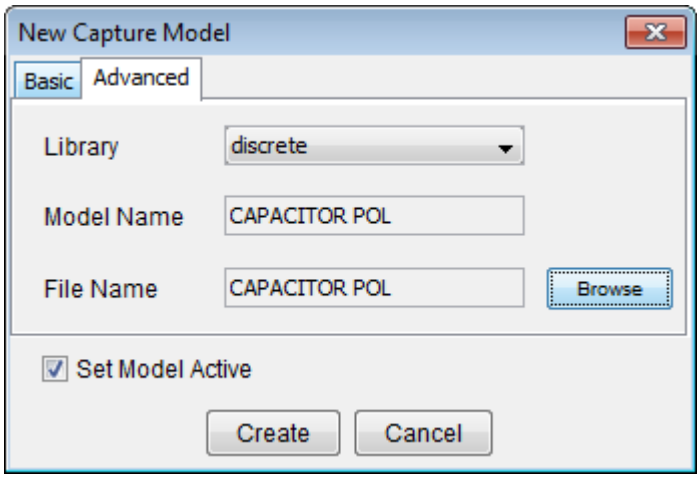
     Note: Parts in OrCAD Capture are referred to as Capture models in Allegro EDM.

- Click *Browse* to add an existing Capture model that is not in the component database.

  When you click Browse, the Select Capture Model dialog appears.

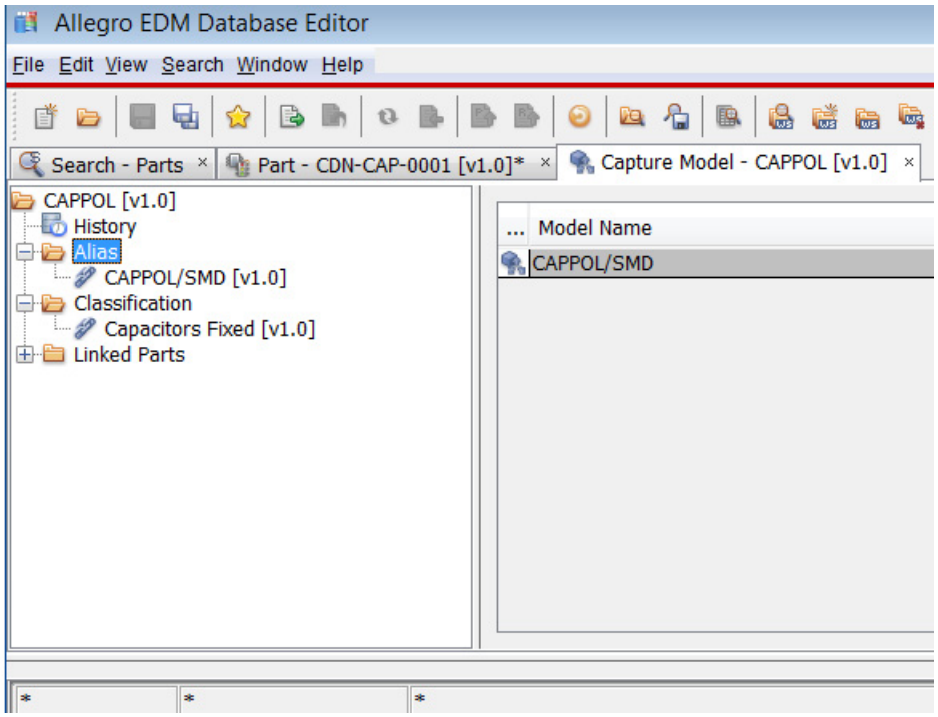


4. Select the Capture model that you want to add to the Allegro EDM component database.



5. Ensure that you have selected the *Set Model Active* check box.
6. Click *Create*.

If you assigned multiple aliases to the Capture model/part that you are adding to Allegro EDM, these aliases are also imported into the component database. They can now be viewed in Allegro EDM as illustrated in the following example:



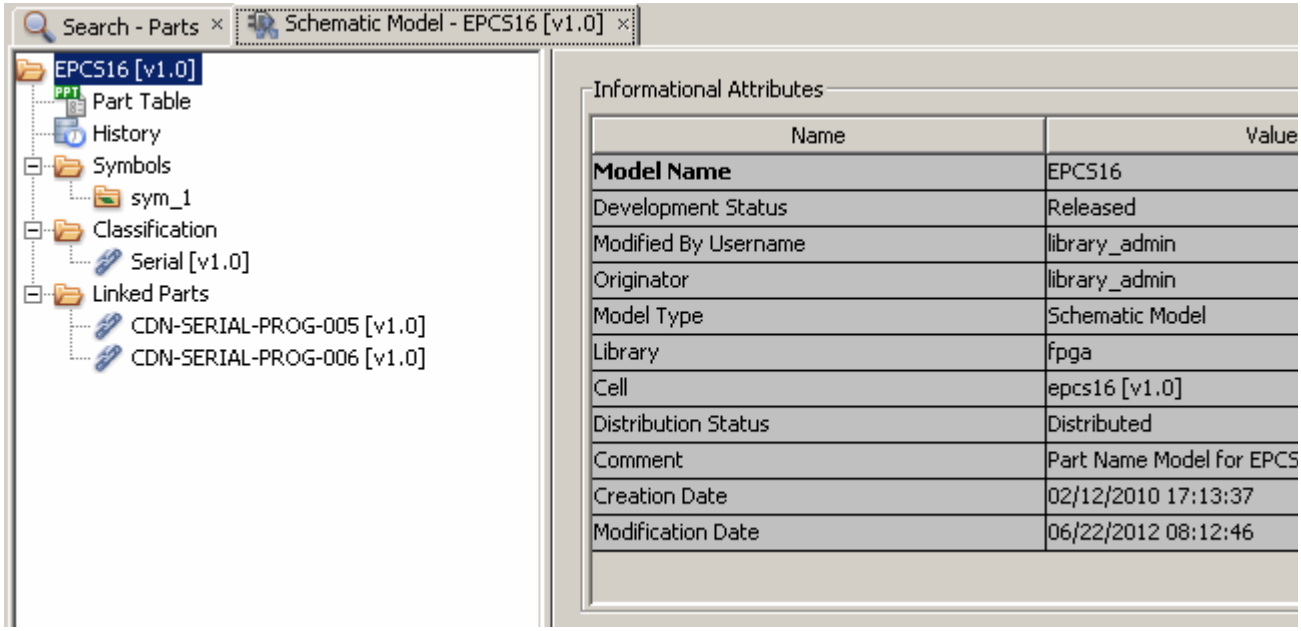
Working with Attributes and Relations

Attributes and relations help you specify properties and linkages for library data (such as parts and models). When you open a *Part - `<Part_Detail>`* or a *Model - `<Model_Detail>`* tab, the explorer pane (left pane) contains the relations of the part or the model. These appear in form of a tree-like structure, where all the available relations appear as nodes.

The right pane of the *Part - `<Part_Detail>`* or *Model - `<Model_Detail>`* tab contains two areas: *Informational Attributes* and *Attributes*. The former displays all the non-editable properties, while the latter contains the editable properties, if any.

Working with Attributes

Attributes are the name-value pair properties defined for part and model data. For example, a model can have attributes such as `Model Name`, `Development Status`, `Originator`, `Creation Date`, and so on.



Some attributes are derived from classifications and component-specific entities defined in the Database Administrator, while some attributes are library-data-specific properties defined when you created the part or model. These properties can be key, mandatory, or optional, based on your database requirements.

For information on default attributes, relations, and their types, see Appendix A, “Attributes and Models,”.

The tasks you can perform on attributes using Database Editor are:

- Adding and Modifying Attribute Values
- Deleting Attribute Values

Adding and Modifying Attribute Values

To add or modify the attribute values, do the following:

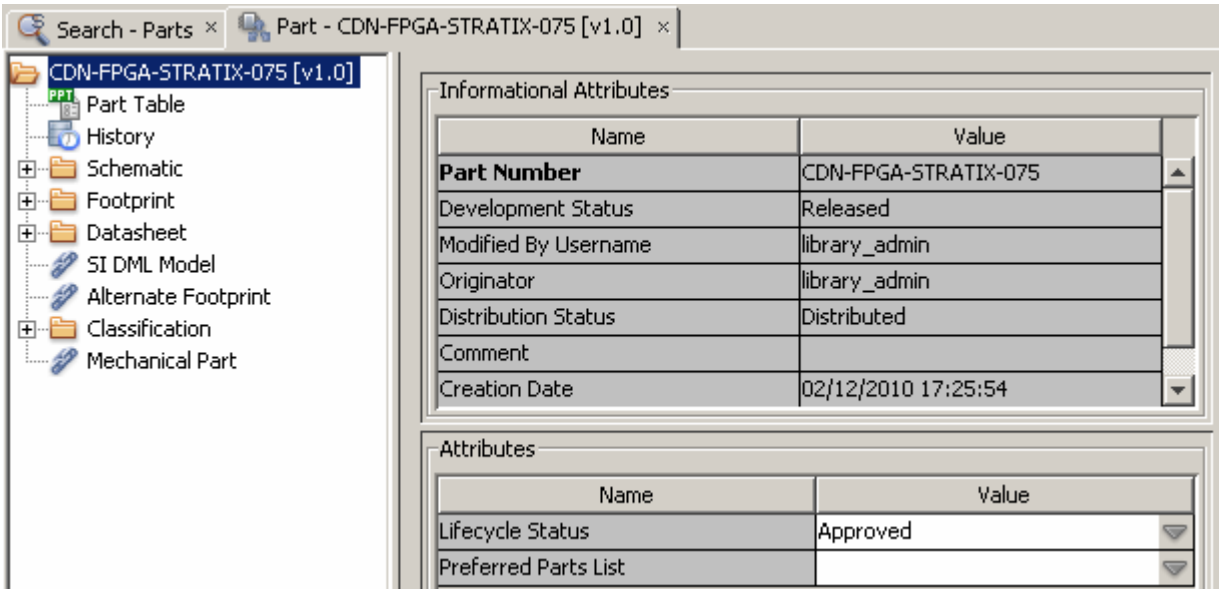
1. Open the *Part - `<Part_Detail>` or Model - `<Model_Detail>`* tab.

Important

Before you add attributes, you need to either create the part or the model, or search for the part or the model. For information on how to do so, see Working with Parts, Working with Models, or Searching Parts and Models.

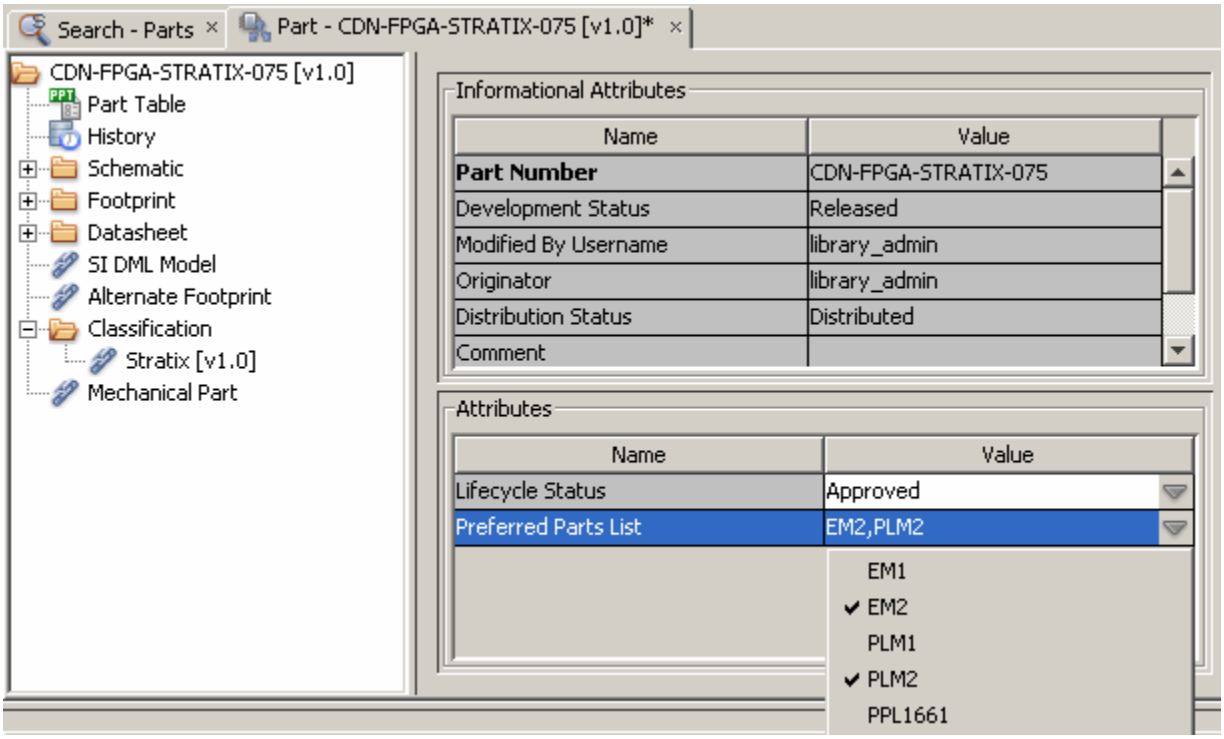
2. Check out the part or model.
3. Select the root node in the explorer pane.

   The *Informational Attributes* and *Attributes* areas appear in the right pane.



4. In the *Attributes* area, select the row (that lists the attributes) under the *Name* column.

5. Add or edit the value in the field next to the attribute (under the *Value* column).



6. Choose *File – Save* or click the *Save* button to save changes.
7. Check in the part or model.

Important

You can modify the attributes *Lifecycle Status* and *Preferred Parts List*, even without checking out the part or block part. You can also assign multiple PPLs to a part or block part. Therefore, for *Preferred Parts List*, you can select more than one value.

Deleting Attribute Values

To delete an attribute value, you have to remove the value against the attribute name. In such cases, the value field remains blank.

Important

You cannot delete attributes that are key or mandatory.

Working with Relations

Relations are associations that you can create between library data entities (such as parts/ block parts and models) that are known as relations. For example, a model can have a relation, such as *Linked Parts*, which allows you to associate all the related parts with it.

Important

Relations, by default, are defined by the administrator. You can only specify instances of the default relations.

For information on the default attributes and relations available for parts and models, see Appendix A, “Attributes and Models,”.

Adding and Editing Relation Instances

To add or edit a relation instance, do the following:

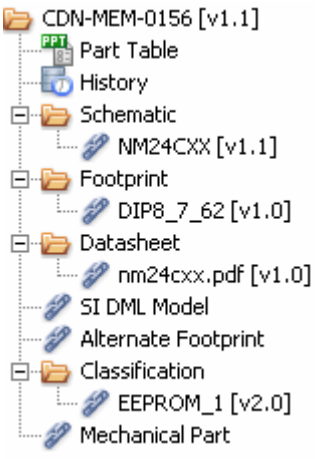
1. Open the *Part -* `<Part_Detail>` *or Model -* `<Model_Detail>` tab.

Important

Before you create relations, you need to either create the part or the model or search for the part or model. For information on how to do so, see Working with Parts, Working with Models, or see Searching Parts and Models.

2. Check out the part or model.
3. In the explorer pane, click a node (*except* the root node).

All the second-level folders ( ) in this pane specify the predefined relations. A chain icon ( ) represents an existing instance of the relation.



When you click a relation or a relation instance, the right pane displays the corresponding information in a table.

4. To specify a relation instance, do one of the following:

   - If you already know the exact instance to specify, choose *Associate* `<Relation_Name>` from the pop-up menu command.

     In the blank row that is added to the table in the right pane, specify the values under each column.

   - Choose *Associate* `<Relation_Name>` *From Tree*.

     The Associate `<Relation_Name>` dialog box appears.

     a. Specify a value from the dialog box.

     b. Click *Associate*.

     A row with the values specified is added to the table in the right pane.

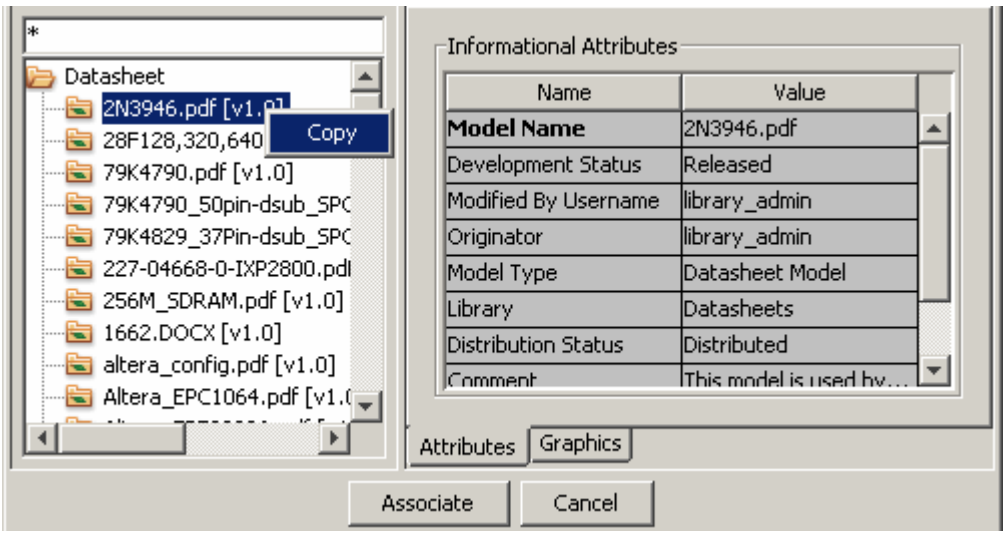
   Note: The number of relation instances you can associate depend on the selected relation. A new and unsaved instance is symbolized by an icon ( ).

5. Choose *File – Save* or click the *Save* button on the toolbar.

Using Copy-Paste Operation

You can also copy a relation instance (using the *Copy* pop-up menu command) in the Associate `<Relation_Name>` dialog box and can paste it on a relation name node. For

example, you can copy a datasheet model, and can paste it on the `Datasheet` node in the explorer pane.



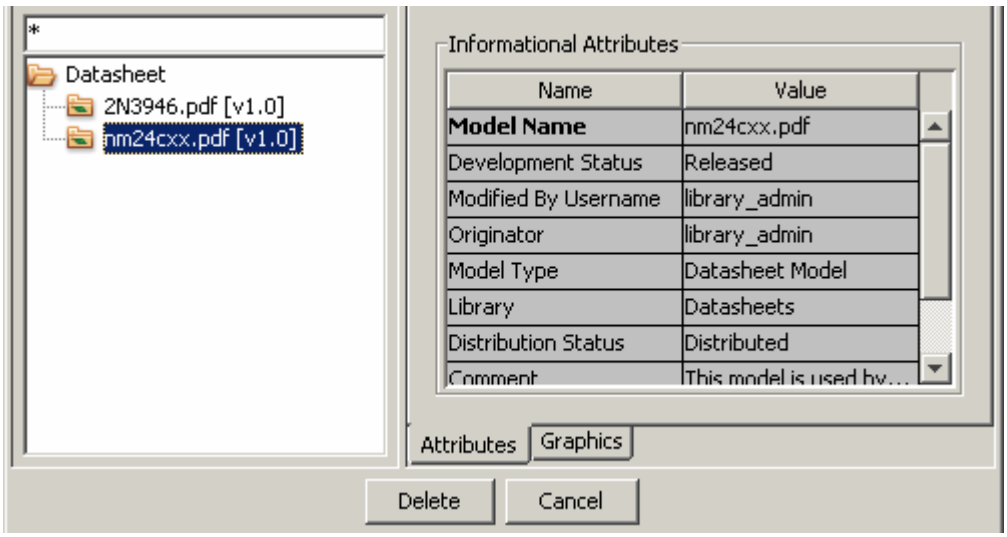
Deleting Relation Instances

To delete a relation instance, do the following:

1. Open the *Part - `<Part_Detail>` or Model - `<Model_Detail>`* tab.
2. Check out the part or model.
3. Right-click the relation instance corresponding to the desired relation node.
4. Do one of the following:
   - Click *Delete*.
   - Right-click a relation node ( ) in the explorer pane.
   - Choose *Delete* from the pop-up menu.

     In the Delete `<Relation_Name>` dialog box that appears, select the relation instance from the list on the left side.

The informational attributes for the instance appear on the right side of the dialog box.



Click *Delete*.

5. Choose *File – Save*.

Searching Parts and Models

Database Editor allows you to search for library data, such as parts (block parts, mechanical parts, and mechanical kits) and models and submodels, as well as administrator data. You can perform a search using the following:

- **Attributes**

  This involves specifying:

  - Informational and other attributes name-value pairs as search parameters.
  - Associations and their values as search parameters.

- **Properties**

  This involves specifying the search parameters of the searchable properties (if defined on parts and models).

How to Search

To search for parts or models, do the following:

1. In the Database Editor window, choose:
   - *Search – Part*
   - *Search – Block Part*
   - *Search – Mechanical –* `<Mechanical Part or Kit>`
   - *Search – Model –* `<Model_Type>`

     For example, choose *Search – Model – Schematic Model* to search for Schematic models. Or for example, choose Search – *Model – Capture Model* to search for Capture models (referred to as Capture parts in OrCAD Capture).

     The appropriate *Search –* `<Parts, Model_Type or Mechanical_Parts/Kits>` tab appears.
2. In the right pane, choose:
   a. The *Attributes* tab to specify attribute-based search parameters and search parameters based on relations.
   b. The *Properties* tab to:
      - Specify the search parameters for searchable properties

- Choose the properties to be displayed in the search results
- Specify the order of properties to be displayed in the search results

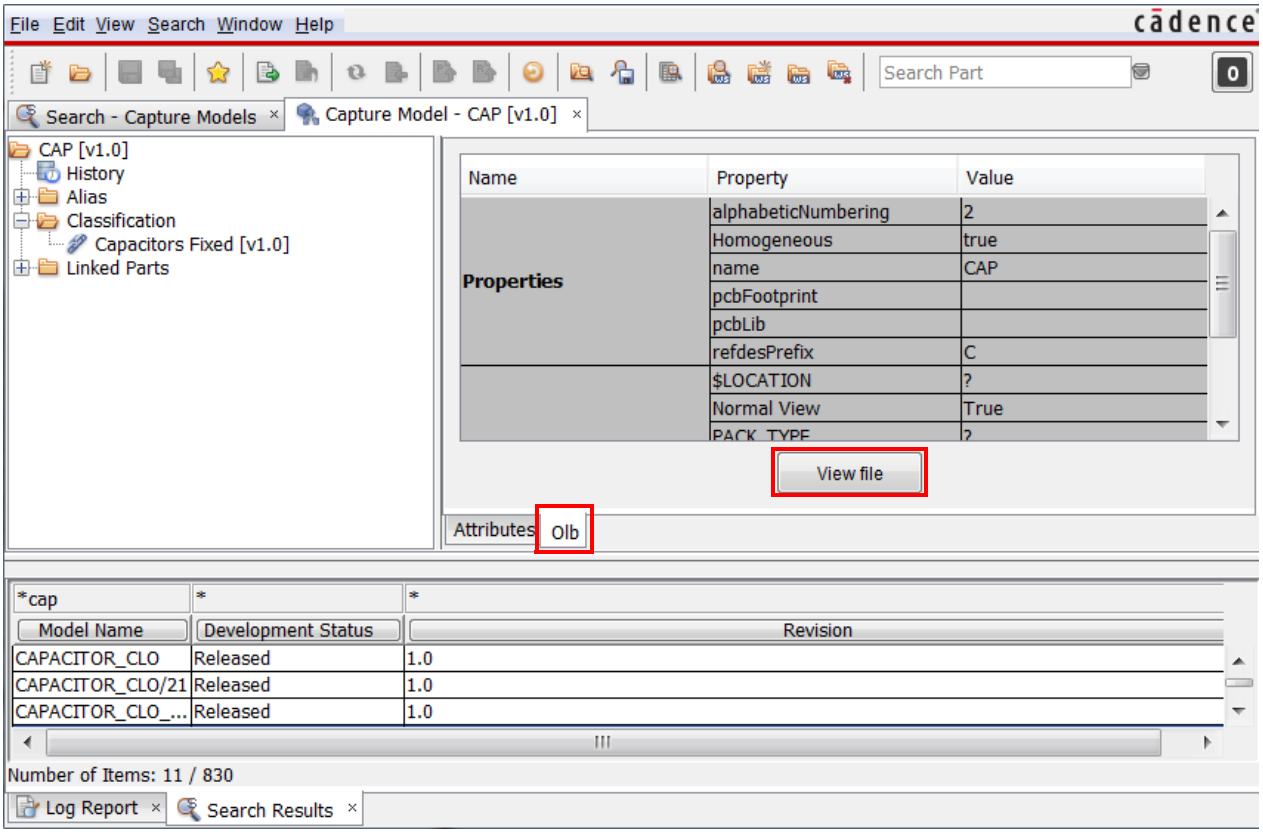
For information on how to specify the search criteria, see Performing Searches.

3. After specifying the search criteria, click the search icon ( ).

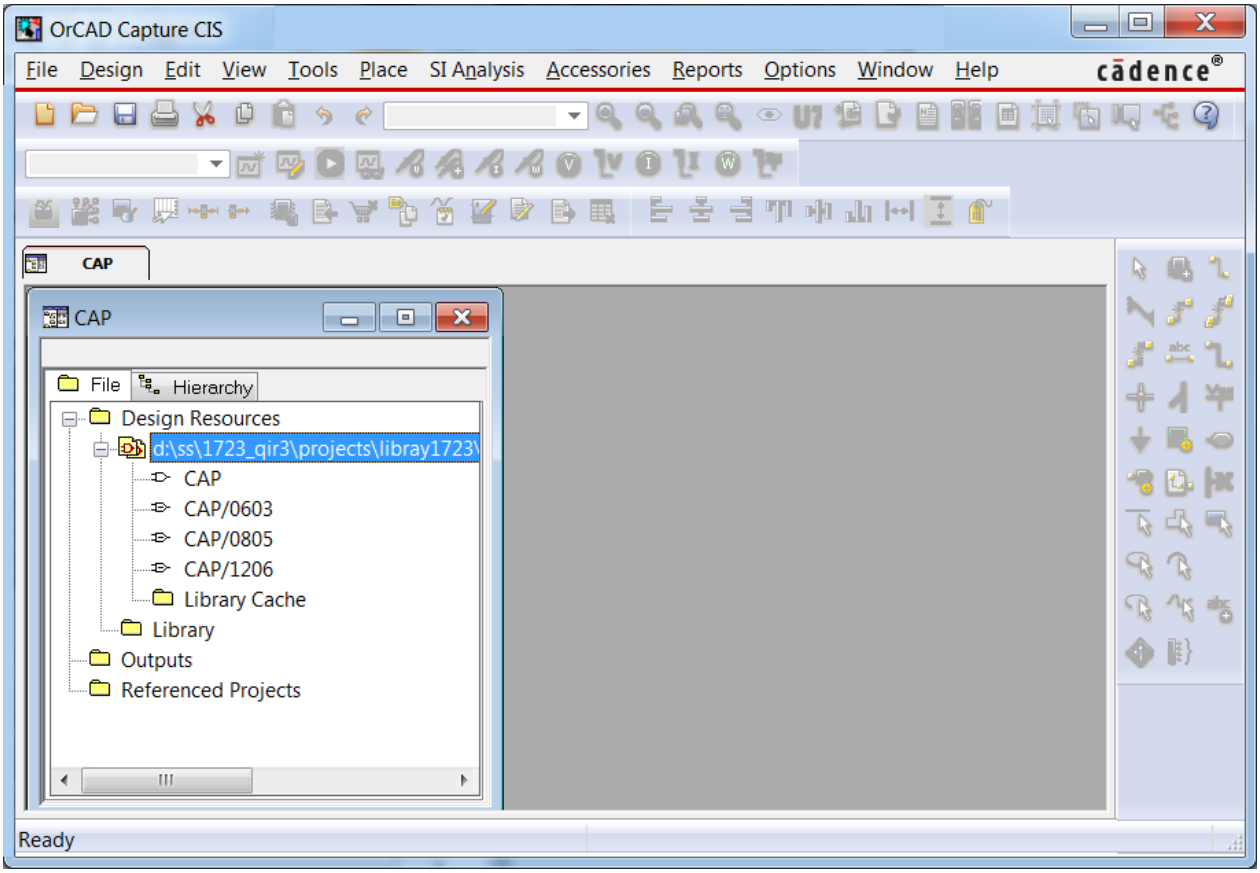
   Search results appear in the *Search Results* tab at the bottom.

4. Double-click a search result row to view its details or right-click and choose *Show Details*.

   In the case of Capture models, you can also open the Capture model in OrCAD Capture to view its details. To open a Capture model in OrCAD Capture, click the *View file* button in the *olb* tab.



The Capture model is opened in OrCAD Capture.



Search-related tasks are as follows:

- Performing Searches
- Resetting the Search Criteria
- Saving the Search Criteria
- Running a Saved Search Criterion
- Refreshing Search Results

Performing Searches

Depending on the complexity of the search expression, you can perform two types of searches.

- Simple Search

- Complex Search

Simple Search

This involves specifying direct and straightforward search parameters. For example, a search on parts with name equal to `CDN-MEM-0094`, or a search on models where *Development Status* is `Preliminary`.

To perform a simple search:

1. Choose *Search – Part, Search – Mechanical –* *<Mechanical Part or Kit>,* or *Search – Model –* *<Model_Type>.*

   Alternatively, you can also use the Search toolbar. For details on using the Search toolbar, see Using Search Toolbar.

2. Choose a classification node in the explorer pane.
3. Choose the *Attributes or Properties* tab.

   The *Attributes* area in the right pane contains a four-column table containing default attributes. The details of the columns are:

Control	Description
Name	Name of the attribute.
Value	Lets you specify the search value for the attribute. Depending on the attribute type, this can be a drop-down list, text field, or a calendar field.
Check box	Lets you specify whether to display the attribute value (as a header) in the search results.
Triangle button ( )	Click this button to form a complex search expression involving logical or boolean operators.

   The *Properties* tab contains the following columns:

Control	Description
Name	Name of the property.

Control	Description
Value	Lets you specify the search value for the property. Depending on the property type, this can be a drop-down list, text field, or a calendar field.
Check box	Lets you specify whether to include the property (as a header) in the search results.
Triangle button ()	Click this to form a complex search expression involving logical or boolean operators.

4. Enter the value next to an attribute or a property (under the *Value* column).

Important

For search parameter values that involve special characters, ensure that you use quotes (‘’) around the value specified.

5. If you have multiple attributes or properties to search:
   - Choose *Search – Search Option – Match All* to specify the AND operator between all the search parameters.
   - Choose *Search – Search Option – Match Any* to specify the OR operator between all the search parameters.
6. To specify the limit for the number of search results:
   a. Choose *Search – Search Option – Search Limit.*
   b. Enter the value between the range given in the Input dialog box.
7. To include or exclude the visibility of attributes or properties in the search results, select the check box corresponding to the attribute or property name.
8. Click the up or down arrow to define the display order in which an attribute or property should appear in the search results. Alternatively, drag and drop the property name to the desired position.

   Note: The revised display order will be available only for the current session. In the next launch of Database Editor, you will see the property display order as predefined by the administrator, if you have configured to see the predefined property display order.
9. Click  .

   The search results appear in the *Search Results* tab.

Using Search Toolbar

By default, the search toolbar is not available. To configure the search toolbar:

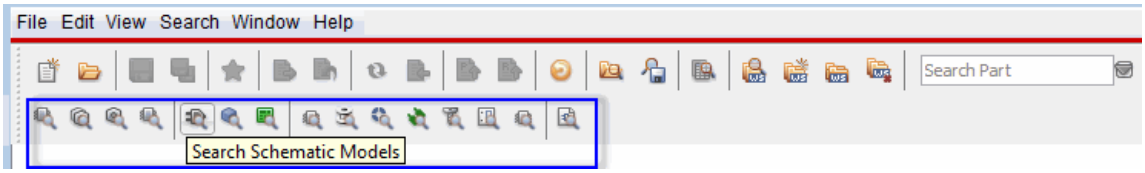
1. Select *Edit – Options*.

   The Options dialog box appears.

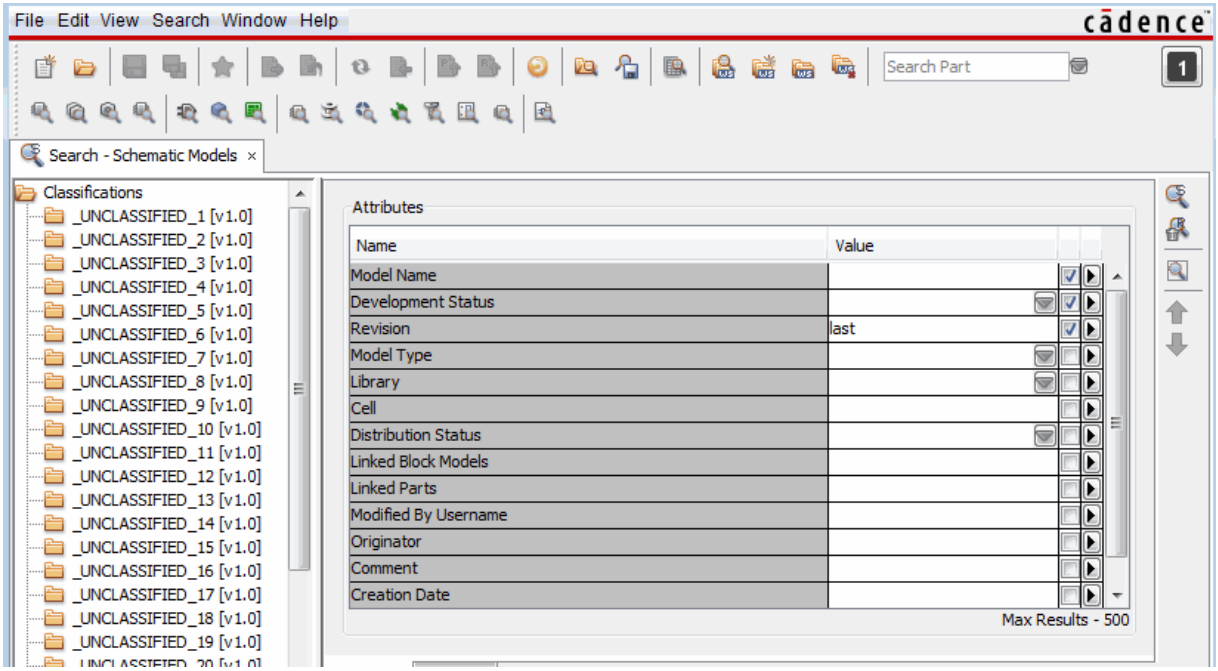
2. Click the *Commands* tab.
3. Select the *Show search toolbar* check box to display the toolbar.
4. Click OK.

   The search toolbar appears.

5. Click the required search icon in the search toolbar.



The search tab is refreshed with selected search option. For example, if you click the icon to search schematic models, the existing search tab, if any is open, will show the search tab for schematic models.



6. Click  in the right pane.

   The search results appear in the *Search Results* tab.

Complex Search

Database Editor allows you to perform complex search queries using a variety of relational operators. The following table lists the operators and their usage.

Operator	Usage
< (Less than)	The value to search must be less than the value specified.
> (Greater than)	The value to search must be greater than the value specified.
!= (Not Equal to)	The value to search must not match the value specified.
== (Equal to)	The value to search must match the exact value specified. This operator is used for case sensitive exact match.
<= (Less than or equal to)	The value to search must be less than or equal to the value specified.
>= (Greater than or equal to)	The value to search must be greater than or equal to the value specified.
~~ (Case insensitive string match)	The pattern of the first value must match the pattern of the second value. The value can be included anywhere in the string. With this operator, character case is ignored so that “library” is considered a match for “LIB*.”
!~~ (Case insensitive string not match)	The pattern of the first value must not match the pattern of the second value. The value can be included anywhere in the string. With this operator, character case is ignored. For example, a first value of “Part Name” and a second value of “pA* nA*” would result in a FALSE comparison since the two are considered a match regardless of the difference in uppercase and lowercase characters.

Operator	Usage
~= (Case sensitive string match)	The pattern of the first value must match the pattern of the second value. The value can be included anywhere in the string. This includes testing for uppercase and lowercase characters. This operator is used for case sensitive string match. If you use * or ? in the string, they are considered as wildcard operators.
!~= (Case sensitive not match)	The pattern of the first value must not match the pattern of the second value. The value can be included anywhere in the string. For example, if the first value is `Part*` a second value of `part` would produce a true result because the lowercase `p` is not an exact match to the first value's uppercase `P`.
&&	The first value and the second value must be present.
Logical AND	
||	Specifies that either of the values be present.
Logical OR	
*	Specifies any number of characters match.
(Any String of Characters)	
?	Specifies exactly one character match.
(Any Single Character)	

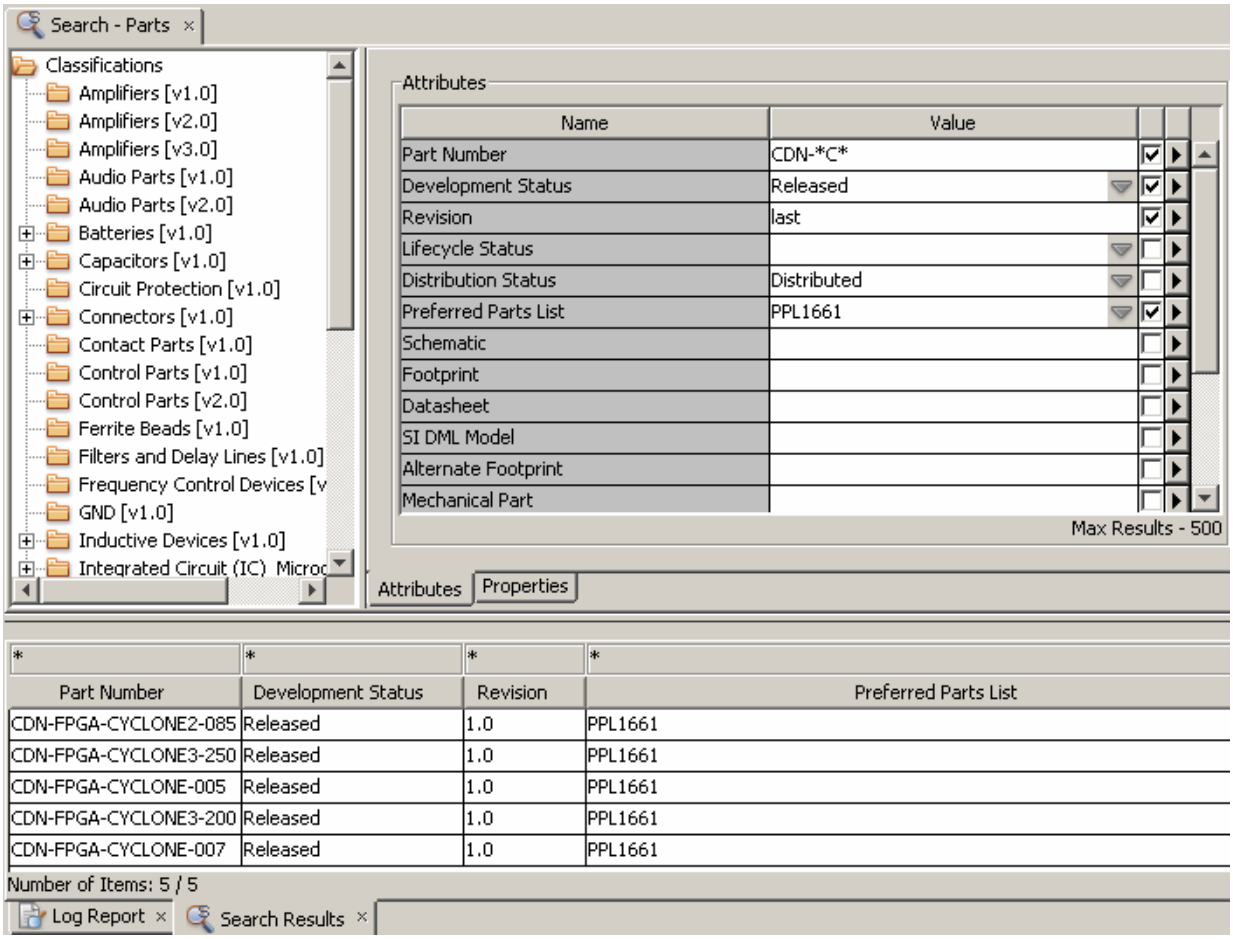
Important

By default, if you do not provide any operator, the ~~ (string match) operator is used. If you specify underscore (_) as a part of search string, then it is treated as the ? character (single character).

To help understand a complex expression search, assume that you want to perform a search with the following parameters:

- Part Number: `CDN-*C*`
- Development Status: `Released`
- Distribution Status: `Distributed`
- Preferred Part List: `PPL1661`

The following figure illustrates how such a complex expression can be defined. You can use the triangle (▶) button to view the operators supported. To run the search, click the Search button.



The selected check boxes signify that these attributes appear as a header in the search results, and the attribute values will appear for the items that meet the search criteria.

Resetting the Search Criteria

To reset a search criteria, do the following:

1. Select a row (in the *Attributes* or *Properties* tab) containing the existing search criterion.
2. Click the *Clear Criteria* icon () in the right pane.

   The search criterion disappears.

Saving the Search Criteria

To save a frequently used search query and its parameters, do the following:

1. In the *Search – <Parts, Block Parts, Model_Type, or Mechanical_Parts/Kits>* tab, enter the search criteria.
2. Choose *Search – Save Search Criteria*.

   The Save search criterion dialog box appears.
3. Enter the name of the search criteria in the *New Search Criterion* field.
4. Click the *Save* button on the toolbar.

All search criteria, by default, are saved in an XML file in the *<Project_Directory>**<Project_Name>*\atdmdir\search folder.

Running a Saved Search Criterion

To run a saved search criterion, do the following:

1. Choose *Search – Load Search Criteria*.

   The Load search criterion dialog box appears listing all the previously saved search criteria.
2. Choose a search criterion from the *Search criteria in current project* list.

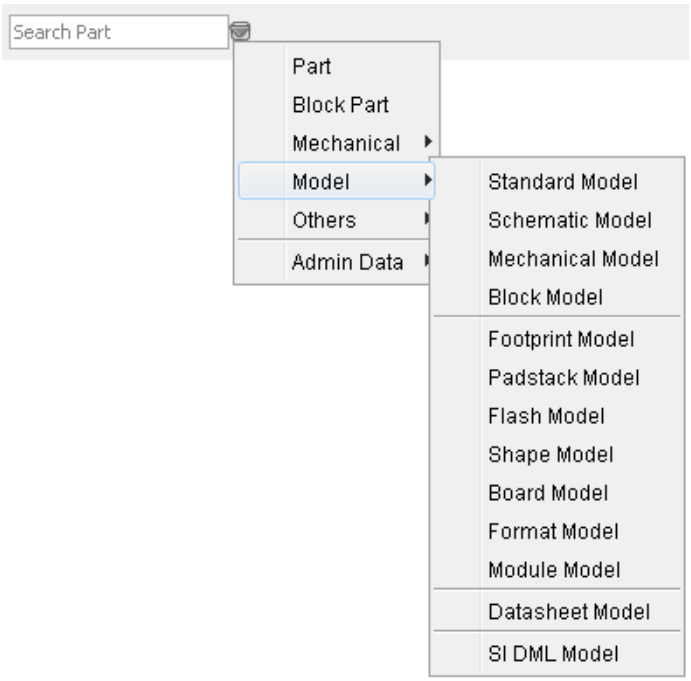
   Alternatively, click *Browse* to navigate to the folder containing saved search criteria files (`.ctr`).
3. Click *Load*.
4. The search criterion appears.
5. Click  to run the search.

Refreshing Search Results

To refresh the *Search Results* tab, choose *View – Refresh* or click  on the toolbar.

Free-Text Search

The toolbar contains a search text box along with a drop down to select the type of search. The free text search works with the attributes, features, and relations under the selected search type.



To search for a part, enter search terms, separated by a space character, in this box. Press the Enter key or click the Search icon. Part Information Manager searches for your search criteria and displays the results in descending order with the best possible match at the top of the results list, and the least relevant result at the bottom of the list.

For example, a search for a criterion such as “res 1k 5%” will be displayed as follows:

Part 1	res 1k 5%
Part 3	res 1000 5%
Part 4	res 1000 5percent
Part 5	1000 5%
Part 6	5%

Using Working Sets

Working sets are the individual workspaces available to a librarian. A working set is a virtual place to keep all your work-in-progress data. You can create, edit, and release the elements of an object as a collection of related objects. Using a working set helps the librarian to validate each operation on an object against all the related objects. This is because all verifications rules run in context of the elements of the working set. This further ensures that when an object is released, all the linked objects are verified so that the front-to-back flow runs successfully.

Librarians can share these working sets among them. This enables librarians to work concurrently in a collaborative environment. If you have checked out a part for editing, the part gets added in your working set. As soon as you check it in, the new version is available to the other librarians. This ensures that you can quickly access your work-related component data.

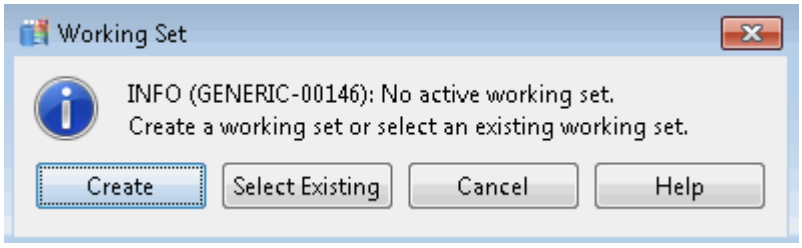
When you check out an object, it is automatically added to the active working set. So there is no need to manually manage a working set.

The tasks you can perform with a working set are:

- Creating a Working Set
- Opening or Changing a Working Set
- Closing a Working Set
- Viewing the Current Working Set
- Adding Objects to Working Set in a Multi-Librarian Environment
- Using Pop-up Menu Commands in the Working Set Tab

Creating a Working Set

The checked-out parts and models are available, by default, in your current working set. If there is no active working set, you are prompted to create a working set or select an existing working set before you check out an object for editing.

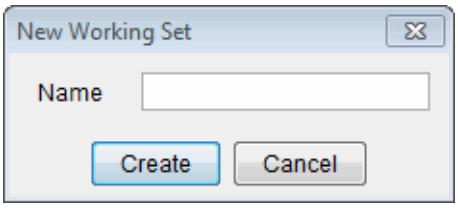


To create a working set:

1. Choose *File – Working Set – Create*.

   Alternatively, click  on the toolbar.

   The New Working Set dialog box appears.



2. Enter a name for the working set.
3. Click *Create*.

   The working set is created.

Opening or Changing a Working Set

If you have an active working set from a previous session, checking out an object will add it to the same working set. You can select and activate a different working set, or create a new one before you check out a part or model for editing.

You can have only one open or active working set at a time. You can create multiple working sets but only one can be active at a time.

To open or change a working set:

1. Choose *File – Working Set – Change*.

   Alternatively, click  on the toolbar.

   The Change Working Set dialog box appears containing a list of all the working sets you have created.

2. Select the required working set.
3. Click *Set Active*.

The working set becomes your current working set, and appears in form of a tab beside the *Search Results* tab. It contains all the components you have created or edited.

Note: By default, you can only see the working sets that you have created.

- *View All*: Select this option to see the working sets created by all the users.
- *View Closed*: Select this option to see the working sets that have been marked closed in the database by all the users. If required, you can again set these working sets active.

Closing a Working Set

You should close a working set after you have released the objects in it, and open a new working set for the next object editing session. When you close a working set, it is not deleted from the database. It is only marked as closed.

You can close a working set at any time. However, if the working set contains objects, which have been edited and not released, such objects will be permanently deleted from the database.

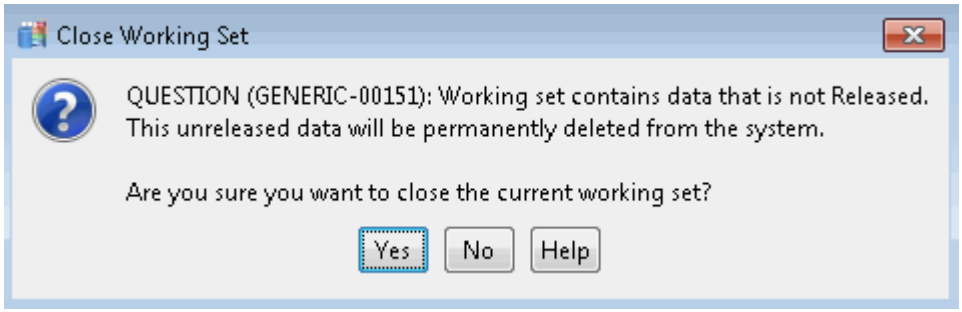
To close a working set:

1. Choose *File – Working Set – Close*.

   Alternatively, click  on the toolbar.

Important

You are prompted to close the working set. When a working set is marked closed in the database, any objects that are not released in this working set are permanently removed from the database.



2. Click *Yes*.

Reopening a Working Set

You can reopen a closed working set. To do so:

1. Click the *Change Working Set* icon () on the toolbar.

The Change Working Set dialog box appears containing a list of all the available working sets.

2. Select the *View Closed* option to see all the working sets that have been marked as closed in the database.
3. Select the required working set.
4. Click *Set Active*.

   A message appears to confirm if you want to set a closed working set active.
5. Click *Yes*.

   The selected working set appears as a new tab. However, it only shows the released objects as the other objects, which were edited and not released were permanently deleted from the database when this working set was closed.

Viewing the Current Working Set

If you have closed the *Working Set* tab, or want to determine your active working set, then choose *View – Current Working Set.*

Alternatively, click  on the toolbar. The *Working Set* tab reappears.

Adding Objects to Working Set in a Multi-Librarian Environment

While working on objects in the current working set, you may need to add to it one or more objects which are being modified (checked in or checked out) by another librarian in a different working set. In such situations, you also need to know the related objects being modified by other librarians. This will enable you to verify the changes you are making to objects in your working set.

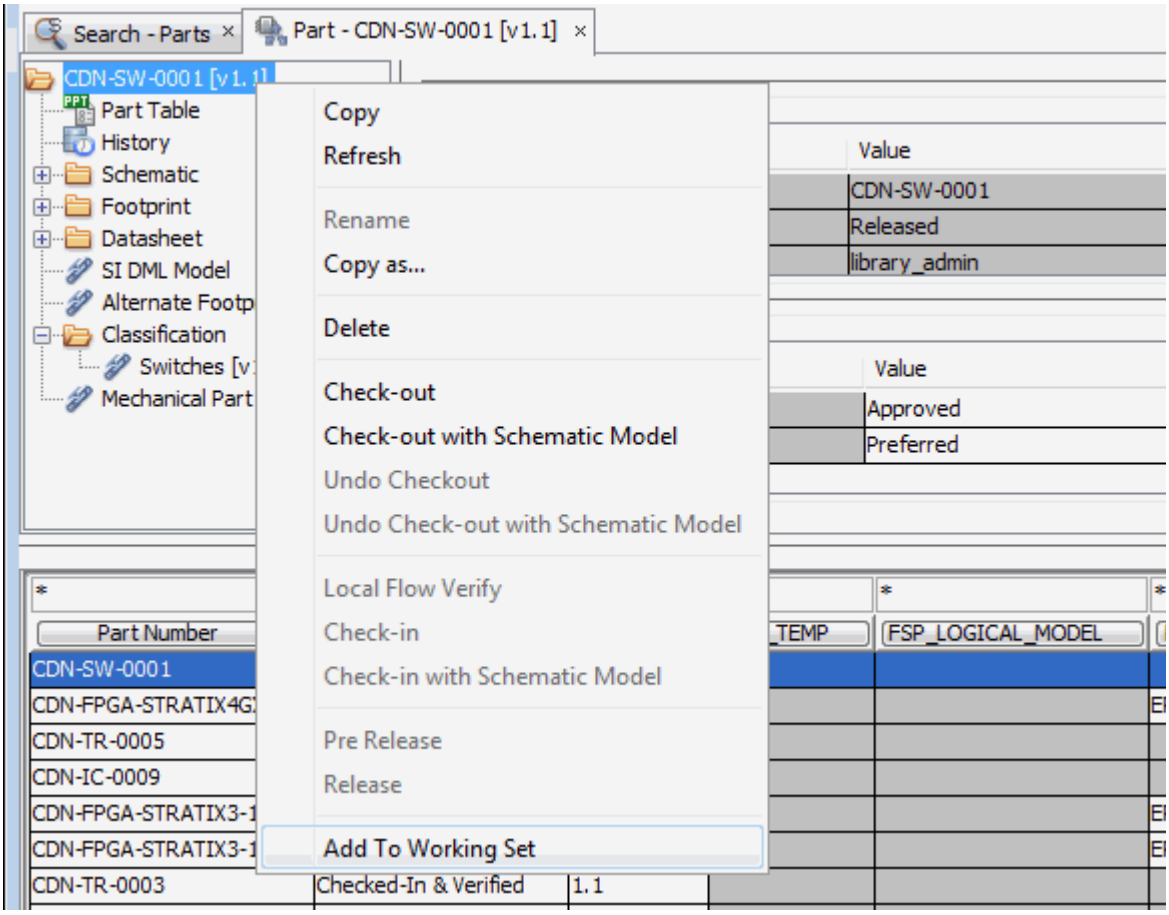
To add objects to the current working set, do the following:

1. Search and select the objects to be added to the current working set by doing the following:

   a. Double-click a search result row to view its details or right-click and choose *Show Details*.

   b. In the `<Object-Detail>` tab, choose the object root node.

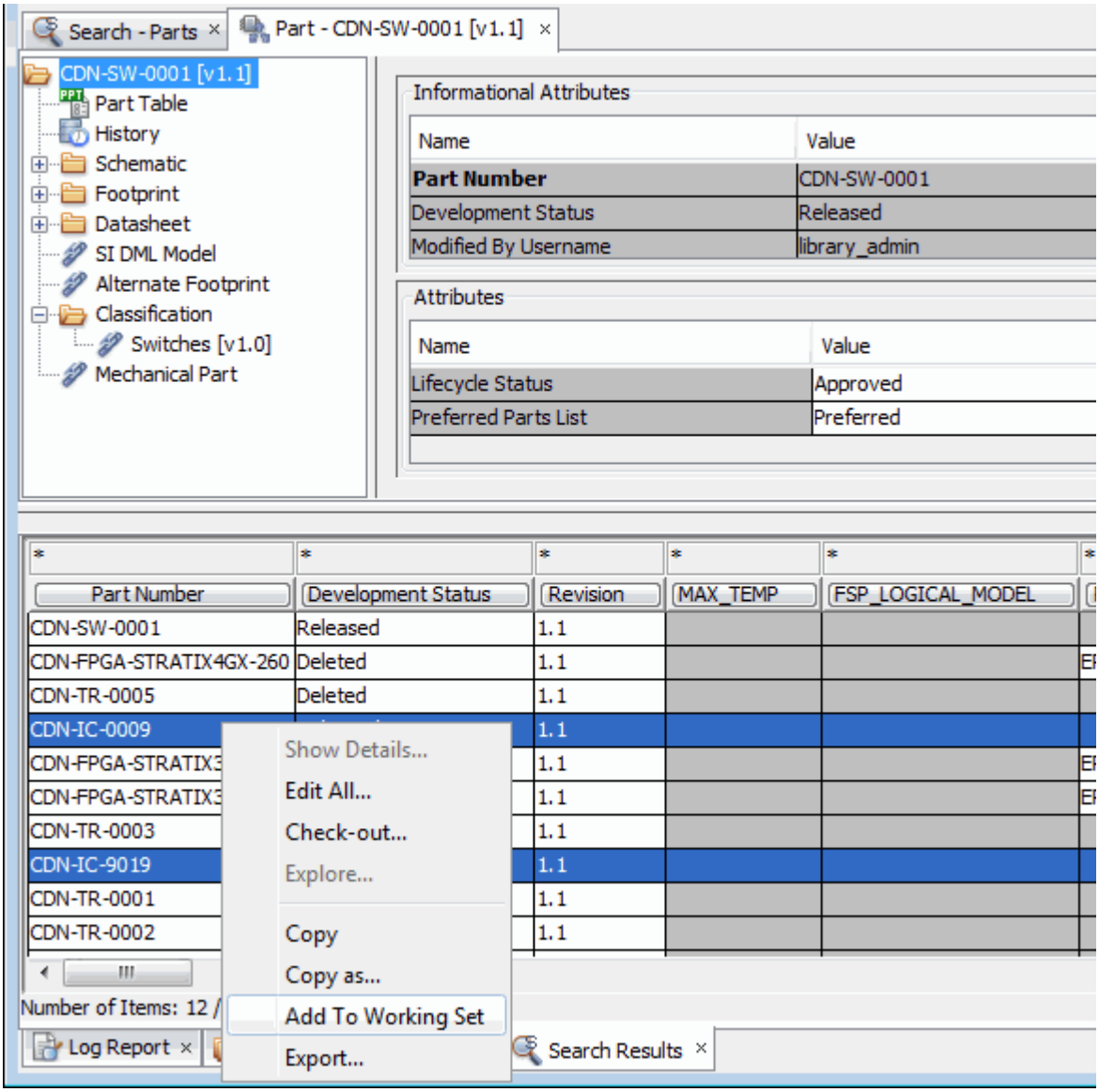
c. Right-click and choose *Add to Working Set*.



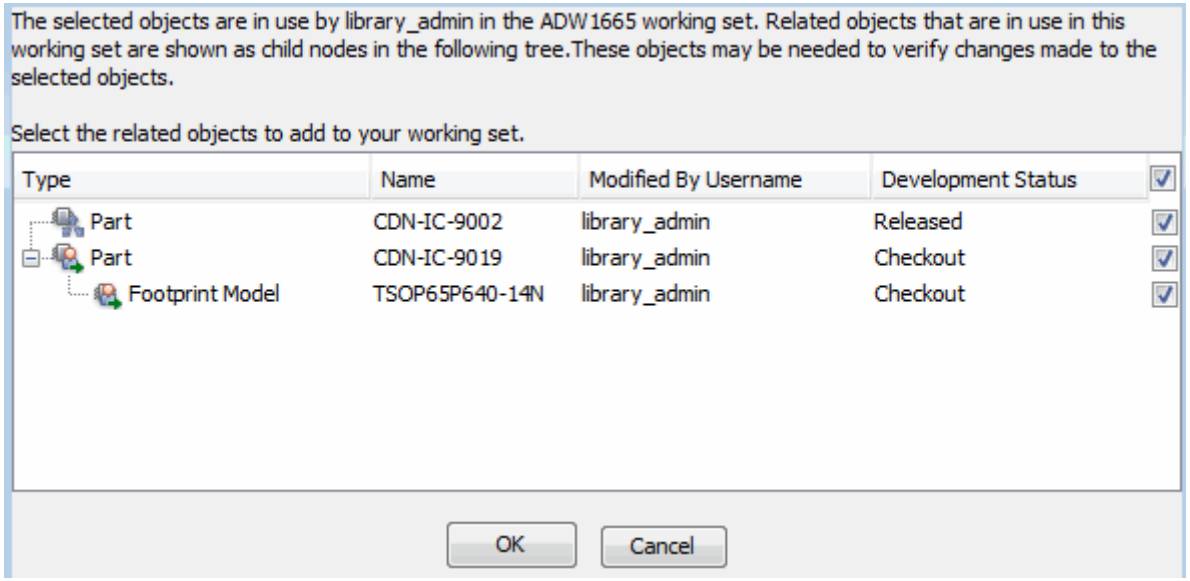
OR

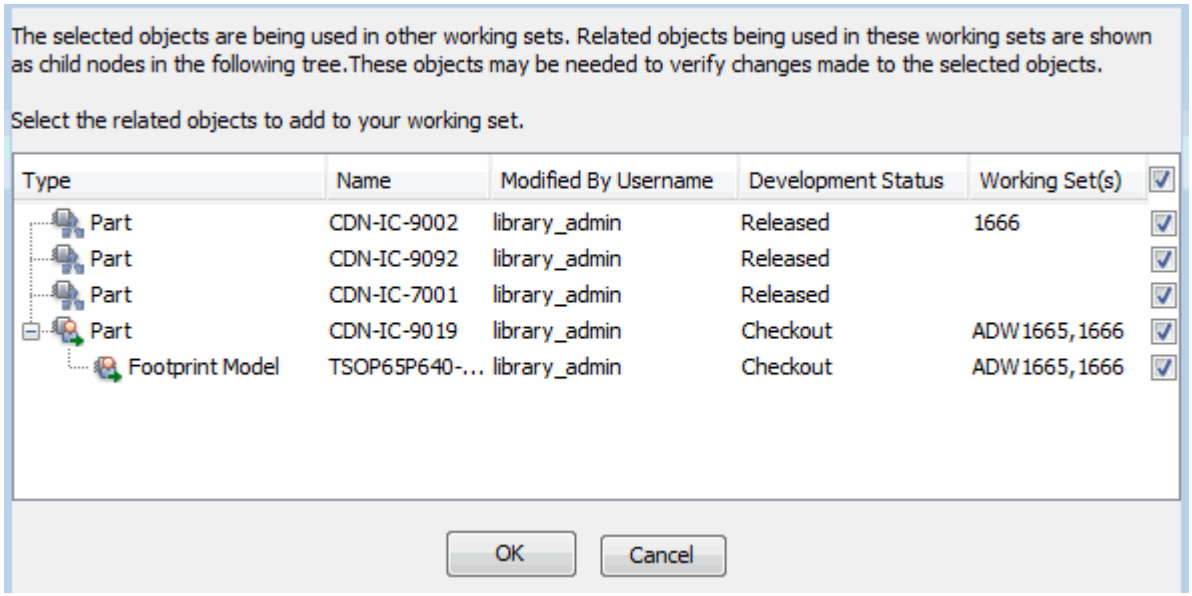
a. In the *Search Results* tab, select one or more objects.

b. Right-click and choose Add to Working Set.



If the selected objects belong to another librarian's working set or if the selected objects belong to more than one working sets, the Add to Working Set dialog box appears.





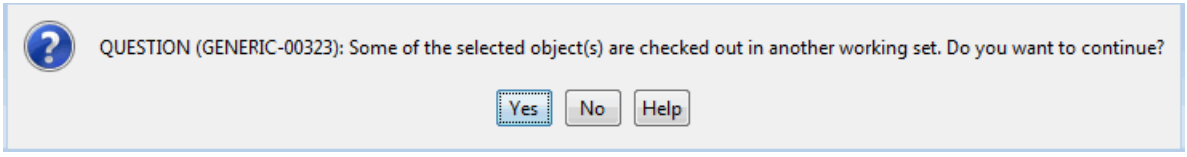
This dialog box shows the following information:

- Object type and the related object type
- Object name and the related object names
- User name of the librarian who has modified the object
- Development Status of the object and the related objects

- Name of the working sets to which the object and related object belong to
- Option to select the object and related objects

2. Expand the object tree.
3. Select the objects and related objects that you want to add to your working set.
4. Click *OK*.

   A message appears if any of the related objects are checked out in another working set. If you add such an object in your working set, the changes may not be verified during the check-in or release process.



5. Click *Yes* to add such objects to your working set.

Using Pop-up Menu Commands in the Working Set Tab

The following table describes the related operations you can perform by selecting a part or model in the *Working Set* tab.

Command	Lets you
Show Details	View the part or model details in the respective tabs.
Edit All	Open the selected parts or models in the right pane for viewing or filtering them.
Export	Create a local compressed copy of the part or model selected.
Copy	Copy the selected part or model.
Remove From Set	Remove a part or model from the working set.
Refresh	Refresh the contents of the working set.

3

Working with Classifications

Working with Administrator Data

Allegro EDM Database Editor lets you do the following:

- View parts and model classifications
- Search parts and models

When you log on to the Database Editor as an administrator, you can view the options to:

- Search model types, libraries, and preferred part lists

  These tasks can be performed using the following menus:

  - *File – Open – Admin Data*
  - *Search – Admin Data*
- Manage classifications

Overview of Classifications

Classification is a hierarchical categorization that helps you organize your Allegro EDM database components (parts and models). Classifications enable faster searching. They also play a critical role in Part Table File (PTF) generation. Schematic Model Classifications define all the PTF Properties: Global, Key, Injected, Additional, and so on. Classifications are defined in the Database Editor (dbeditor) utility.

Database Editor supports the following types of classifications:

- Part Classification (such as transistors, resistors, capacitors, ICs, and so on)
- Block Part Classification
- Mechanical Part Classification

- Model Classification (such as Schematic Model, Flash Model, Capture Model, Shape Model, Footprint Model, Padstack Model, and Datasheet Model)

Classification helps you logically arrange parts and models for your design site. Moreover, common attributes can be defined for a parent classification and all of those can be linked to parts, models, or child classifications that are under the parent classification. For example, you can specify various attributes for the `Resistors` parent classification, which are inherited by the parts and child classifications under the `Resistors` classification. In addition, classifications also help you search parts and models quickly.

Classification Versioning

Revision management is a key feature that helps you subclassify versions of a classification (part and model) that can be used by your designers and librarians. The default version for all the classifications is `1.0`. For example, if you have an existing part classification, say `LOGIC_GATE [1.0]`, and want to update it because of some changes, you need to modify it and save it. An incremental version of this classification, say `LOGIC_GATE [2.0]`, is created and the previous version is marked as obsolete.

Important

Be aware that only the last version of a classification is marked as obsolete. If there are any previous versions of the classification, you need to manually mark these versions of the classification as obsolete.

Understanding Hierarchical Classifications and Properties Behavior

In a hierarchy of classifications, if you try to add, edit, or delete a property inherited by the child classifications, then the changes you make in the parent classifications are not propagated to the child classifications. All such properties are overridden by the property changes in the child classifications and the properties are displayed with the overridden property icon () in the right pane.

Changes to a property in a parent classification are propagated to a child classification only if the child classification does not have that property as overridden.

Managing Classification Tree

The main tasks you perform while working with classifications are as follows:

- Creating Child Classifications

- Revising Classifications
- Using the Refresh Pop-up Menu Command
- Deleting Classifications
- Working with Obsolete Classifications
- Renaming Classifications
- Using the Cut, Copy, and Move Pop-up Menu Commands

Creating Child Classifications

The Library Import application allows you to specify the default classification for your site libraries. When these are available, you can make modifications to the existing classifications:

To create a child classification, do the following:

1. Choose *File – Manage Part Classification*, *File – Manage Block Part Classification*, *File – Manage Mechanical Part Classification*, or *File – Manage Model Classification – <Model_Type> Classification.*

   The appropriate *Classification* tab appears.

2. Select a classification node in the explorer pane and choose *Add Child* from the pop-up menu. Alternatively, use the shortcut key `Ctrl+N`.

   The New dialog box appears.

3. Specify the name for the new classification and click *Create*.

   The new classification appears under the node selected with `[v1.0]` as the suffix and is added to the database.

For new child classifications with inherited properties,  appears against the property row in the right pane.

Note: To create classifications further, follow the aforementioned steps as many times as needed.

Revising Classifications

You can now easily revise a classification. To revise a classification, do the following:

1. Choose *File – Manage Part Classification*, *File – Manage Block Part Classification*, *File – Manage Mechanical Part Classification*, or *File – Manage Model Classification – <Model_Type> Classification*.

   The appropriate *Classification* tab appears.

2. In the explorer pane, select a classification node you want to revise.

3. Modify the classification.

Important

If objects linked to the classification are in:

- `Distributed` state, a revised classification is created
- `Pending Distribution` state, a new revision is not created

4. An asterisk (*) appears next to the revised classification node icon in the explorer pane indicating an unsaved classification.

5. Click the *Save* button.

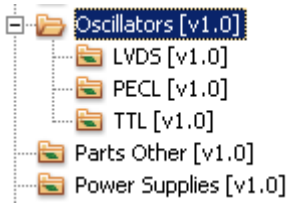
   A new version of the classification is seen with the incremental version number as the suffix. For example, if the previous version is 1.0, the new version will be 2.0.

Important

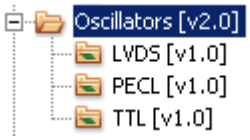
The previous version of the classification disappears because it is marked obsolete. Any obsolete classification cannot be revised.

Revising Hierarchical Classifications

When a classification contains subclassifications and you revise the immediate parent classification, the version of children classifications are not revised. For example, you have a part classification, say `Oscillators` with version `1.0` with child classifications as shown below.



After you revise this, `Oscillators[v2.0]` is created. Note the `[v1.0]` of its child classifications.



Similarly, a change made to a subclassification (of a specific version) does not apply to other subclassification that has the same name but a different parent classification. For example, if you make changes to `TTL[v1.0]` under `Oscillators[v2.0]`, these changes does not affect `TTL[v1.0]` under `Oscillators[v1.0]`.

Using the Refresh Pop-up Menu Command

The pop-up menu items for classifications contain the *Refresh* command. This command helps you refresh the view for the selected nodes in the explorer pane.

Note: In hierarchical classifications this command becomes important for making property changes. To help you understand this, assume you have two hierarchical classification nodes where `part2[v1.0]` is a subclassification of `part1[v1.0]`, and `part2[v1.0]` has parts linked to it, and you want to edit properties for this version. To do so, you revise `part2[v1.0]` to `part2[v2.0]`, and make property changes to it. Before you save the hierarchy, you must select `part2[v1.0]`, choose *Refresh* pop-up command. When done, select `part1[v1.0]` and choose the *Save Hierarchy* pop-up menu command.

Deleting Classifications

You cannot delete a classification if the version of its associated objects has changed, and the latest versions of these objects are linked to a new classification.

This is done to maintain the integrity of the database. For example, assume that you have created and released some parts in classification1, then checked these parts out and associated classification2 with them. If you delete classification1, the history of these parts will be lost.

If you do not want to view certain classifications in the classification tree, you can mark those classifications as *Obsolete*. For more information, see Working with Obsolete Classifications.

You cannot recover a deleted classification.

To delete a classification, do the following:

1. Depending on the classification that you want to delete, select one of the following:
   - *File – Manage Part Classification*
   - *File – Manage Block Part Classification*
   - *File – Manage Mechanical Part Classification*
   - *File – Manage Model Classification – <Model_Type> Classification*.

   The selected *Classification* tab appears.
2. Select the target node in the explorer pane, and choose *Delete* from the pop-up menu.

   You are prompted to delete the classification and all its children.
3. Click *Yes*.

Working with Obsolete Classifications

Obsolete classification are those that are no longer used. This happens when you have incremental versions of classifications and you are confident that they are no longer required by the design groups. In such cases, you can mark the classification *Obsolete*. An obsolete classification is not visible to designers. However, it is available in the database. This ensures that as a data administrator, you have access to it.

A classification name is unique within a parent classification node. As a result, you can create more than one classification with the same name but under different parent classifications. Moreover, a parent and child classification can share the same name. For example, the following hierarchy is valid.

ClassP1 [v1.0]

……..ClassC1 [v1.0]

ClassP2 [v1.0]

……..ClassC1 [v1.0]

In this example, you cannot add or copy another classification with the name ClassC1 (under ClassP1 [v1.0]). But, you can do this when you mark the ClassC1 obsolete.

Note: Database validations are not performed on an obsolete classification.

The procedures related to obsolete classifications are:

- Marking a Classification as Obsolete
- Viewing Obsolete Classifications

Marking a Classification as Obsolete

You cannot set a classification to obsolete if it has any *distributed* objects linked to it. After you have ensured that a classification is not in use, you can mark it as obsolete.

To mark a classification as obsolete, do the following:

1. Depending on the classification that you want to mark as obsolete, select one of the following:
   - *File – Manage Part Classification*
   - *File – Manage Block Part Classification*
   - *File – Manage Mechanical Part Classification*
   - *File – Manage Model Classification – <Model_Type> Classification*.

   The appropriate *Classification* tab appears.
2. Select a node in the explorer pane and choose *Obsolete* from the pop-up menu.

   Note: A classification is always active before it is marked obsolete.

   You are prompted to mark the selected classifications (and its children, if any) as obsolete.

Important

If you mark a classification that contains subclassifications as obsolete, all the subclassifications are also marked obsolete.

Important

After a classification along with its children classifications, if any, is marked obsolete, it becomes a dormant classification. You can view dormant classifications through the *Show Obsolete Classifications* pop-up menu command, but cannot use them for classifying part and model data.

3. Click *Yes*.

   Note: If the classification you want to mark as obsolete is linked to any active database objects, the *Log Report* tab appears displaying the error.

Viewing Obsolete Classifications

To view obsolete classifications, do the following:

1. Depending on the obsolete classification you want to view, select one of the following:
   - *File – Manage Part Classification*
   - *File – Manage Block Part Classification*
   - *File – Manage Mechanical Part Classification*
   - *File – Manage Model Classification – <Model_Type> Classification*.

   The appropriate *Classification* tab appears.
2. Select the highest node (*Part Classification* node for part classification or *<Model_Type> Classification* node for model classification) in the explorer pane and choose *Show Obsolete Classifications* from the pop-up menu.

   All obsolete classifications nodes appear in black in the explorer pane.

Figure 3-1 Obsolete Classifications Appear in Black Italics

Renaming Classifications

To rename a classification node, do the following:

1. Depending on the classification you want to rename, select one of the following:
   - ❑ *File – Manage Part Classification*
   - ❑ *File – Manage Block Part Classification*
   - ❑ *File – Manage Mechanical Part Classification*
   - ❑ *File – Manage Model Classification – `<Model_Type>` Classification*.

   The appropriate *Classification* tab appears.
2. Select the desired node in the explorer pane, and choose *Rename* from the pop-up menu, or use the shortcut key `F2`.

   The Rename dialog box appears.
3. Specify a new name and click *Rename*.

   The new name of the node appears in the explorer pane.

Note: When you rename a parent classification, the children classifications are not affected.

Using the Cut, Copy, and Move Pop-up Menu Commands

You can have the same parts, model classifications or their properties under different classifications by cutting or copying a node to another classification node in Database Editor.

These operations are described in the following table:

Operation Selected	Operations Enabled	Description
Cut	Paste with Linked Objects	Allows you to cut and paste the selected classification node and all its child nodes, along with the linked objects of each node to another classification node.
Copy	Paste	Allows you to copy and paste only the selected classification node and all its child nodes to another classification node.

Operation Selected	Operations Enabled	Description
	Paste with Linked Objects	Allows you to copy and paste the selected classification node all its child nodes, along with the linked objects of each node to another classification node.
Move Linked Objects	The Associate Parent Classification dialog box appears.	Allows you to move linked objects from a source to a target classification. The classification node is not moved. If the selected classification node has child nodes, which have linked objects, the move operation will move all those linked objects as well.

To cut and paste a classification node, do the following:

1. Depending on the classification you want to work with, select one of the following:
   - ❑ *File – Manage Part Classification*
   - ❑ *File – Manage Block Part Classification*
   - ❑ *File – Manage Mechanical Part Classification*
   - ❑ *File – Manage Model Classification –* `<Model_Type>` *Classification*.

   The appropriate *Classification* tab appears.
2. Choose a source node in the explorer pane, and choose *Cut* from the pop-up menu or use the shortcut key `Ctrl+x`.
3. Select a destination node in the explorer pane, and choose *Paste with Linked Objects* from the pop-up menu (or use the shortcut key `Ctrl+H`) to move the classification node, and its child nodes along with their respective linked objects.

To copy and paste a classification node, do the following:

1. Depending on the classification you want to work with, select one of the following:
   - ❑ *File – Manage Part Classification*
   - ❑ *File – Manage Block Part Classification*
   - ❑ *File – Manage Mechanical Part Classification*
   - ❑ *File – Manage Model Classification –* `<Model_Type>` *Classification*.

The appropriate *Classification* tab appears.

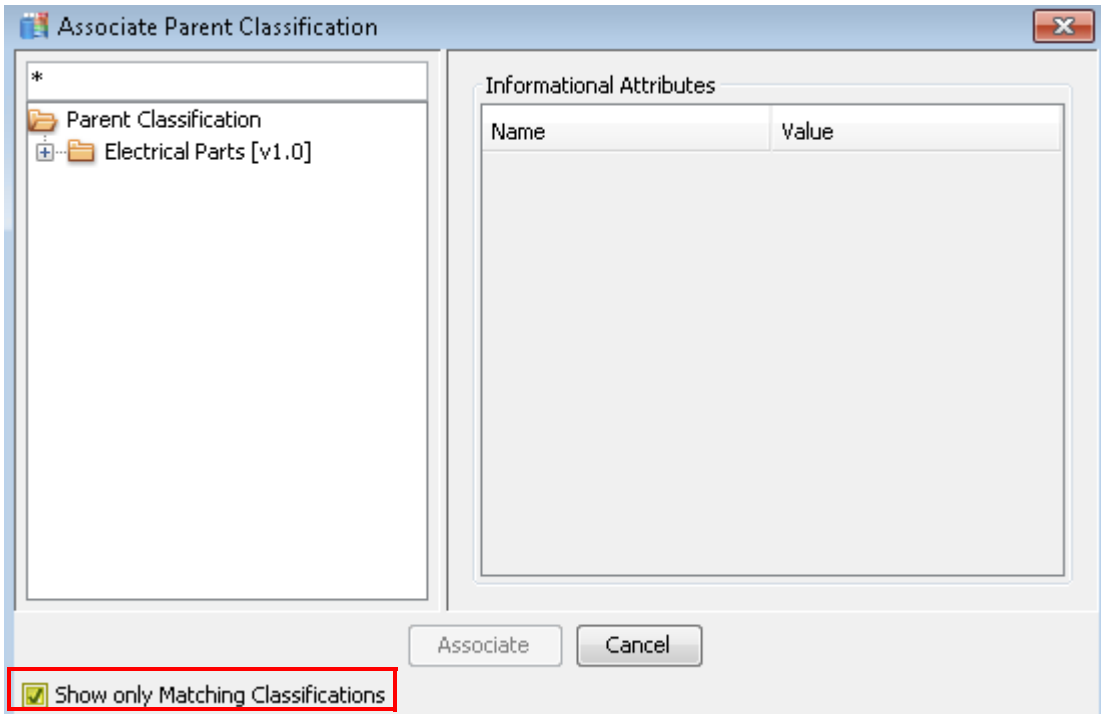
2. Choose a source node in the explorer pane, and choose *Copy* from the pop-up menu or use the shortcut key `Ctrl+C`.
3. Select a destination node in the explorer pane, and choose:
   - *Paste* from the pop-up menu (or use the shortcut key `Ctrl+V`) to copy only the selected classification node, and all its child nodes.
   - *Paste with Linked Objects* from the pop-up menu (or use the shortcut key `Ctrl+H`) to copy the classification node, and its child nodes along with their respective linked objects.

To move linked objects to another classification node, do the following:

1. Depending on the classification you want to work with, select one of the following:
   - *File – Manage Part Classification*
   - *File – Manage Block Part Classification*
   - *File – Manage Mechanical Part Classification*
   - *File – Manage Model Classification – <Model_Type> Classification*.

   The appropriate *Classification* tab appears.
2. Choose a source node in the explorer pane, and choose *Move Linked Objects* from the pop-up menu.

The Associate Parent Classification dialog box appears.



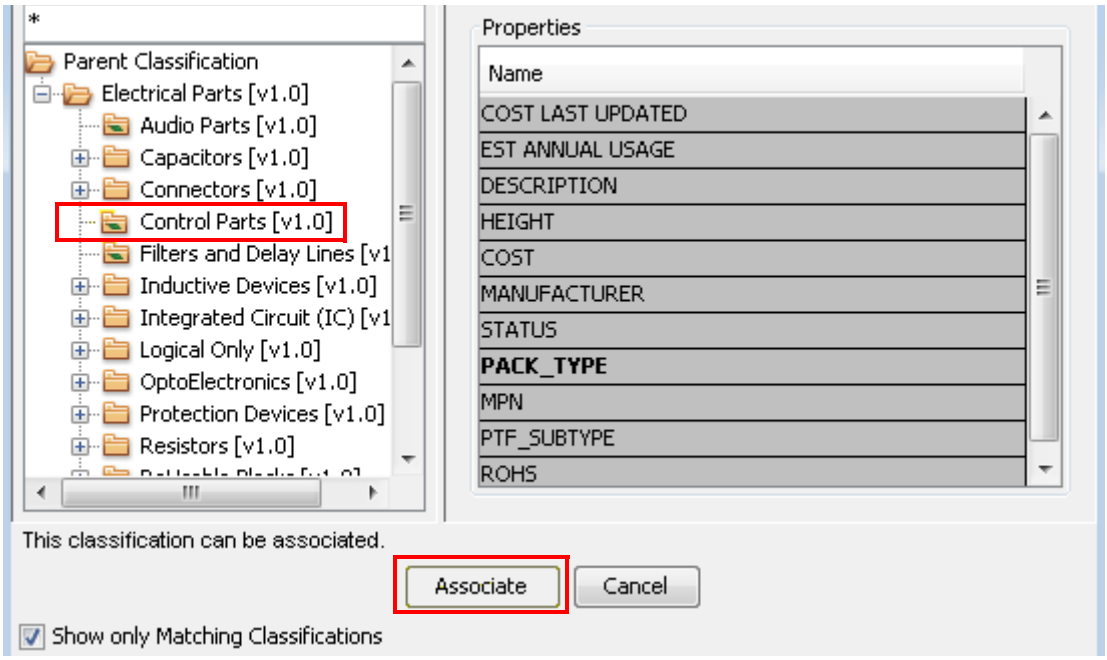
3. As a destination node, select a classification that contains properties matching the selected classification node.

Important

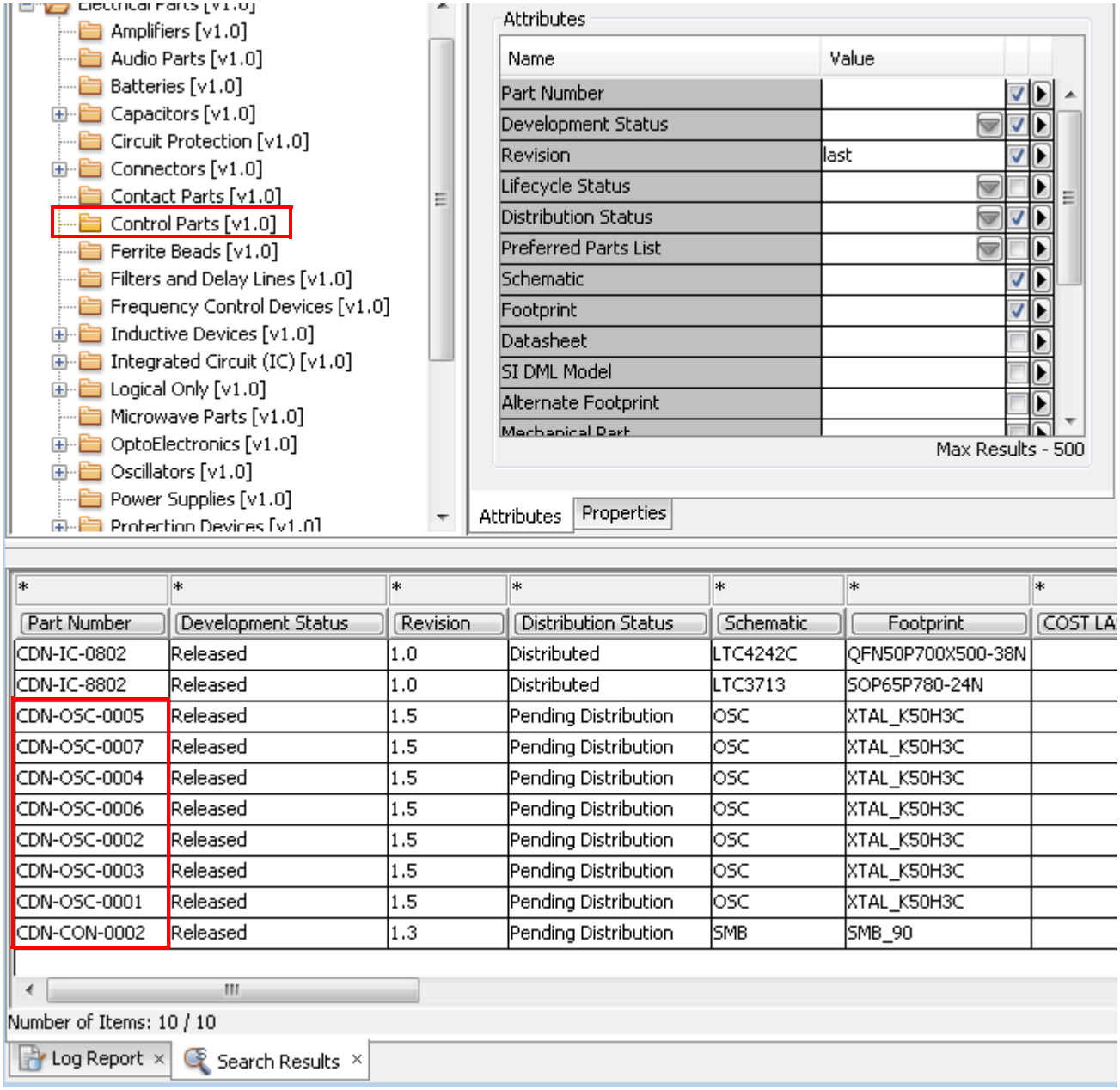
Multiple symbol classifications are only supported when the classification attributes match.

Note: If you clear the *Show only Matching Classifications* check box, you will see all the classifications from the explorer pane.

4. Click *Associate*.



The linked objects from the source node move to the destination node. You can check by searching for those linked objects at the destination node.



Managing Properties Union on Parent Classification

You can now view the union of properties on a parent classification node and make global changes to a set of classifications.

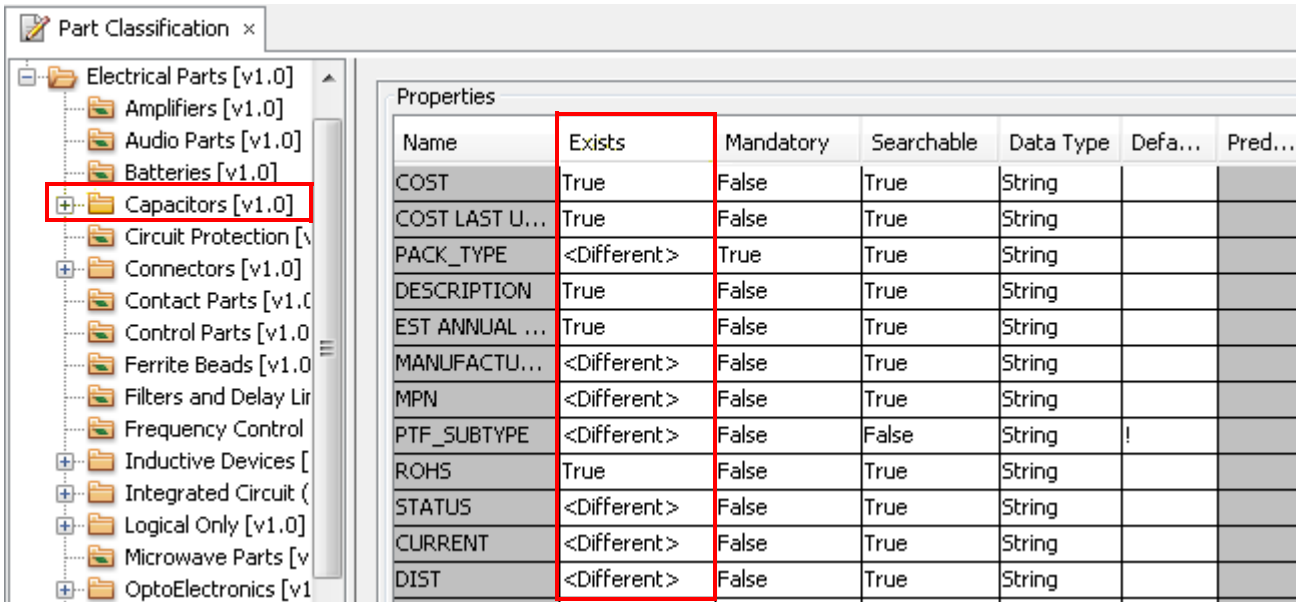
This feature allows you to:

- Identify whether or not a property exists on a selected classification node.

  For each property, the *Exists* column value signifies the following:

  - `True`: Indicates that the property exists on the parent and all its child classifications
  - `<Different>`: Indicates that the property exists on some of the classifications

    Note: For a classification property, when you place the mouse over the column value, `<Different>`, the tooltip displays the list of classifications which contains this property.



- Add a property to the parent and/or to all its child classifications.
- Delete a property from the parent and all its child classifications.
- Make global changes across all child classification in one go. For one or more properties on a parent and all its child classifications, you can:
  - Set the value of the mandatory field
  - Specify whether it is searchable

- Define the data type
- Set the default value
- Set predefined values
- Specify the display and order

Working with Values of Exists Column

Case 1: Adding a property from parent classification to all child classifications

Parent Classification - P

➡ P - Property does not exist. Value is `<Different>`.

Child Classifications - C1, C2, C3, and C4

- C1 - Property does not exist.
- C2 - Property does not exist.
- C3 - Property exists. Value is `True`.
- C4 - Property exists. Value is `True`.

Procedure:

1. Select P.
2. Set value of *Exists* to `True`.
3. Save the classification hierarchy.
4. Property is now added on:
   - P
   - C1
   - C2

Case 2: Removing a property from all child classifications after performing an operation on the parent classification

Parent Classification - P

➡ P - Property does not exist. Value is `<Different>`.

Child Classifications - C1, C2, C3, and C4

- C1 - Property does not exist.
- C2 - Property does not exist
- C3 - Property does not exist
- C4 - Property exists. Value is `True`.

Procedure:

1. Select P.
2. Set value of *Exists* to `False`.
3. Save the classification hierarchy.
4. Property is now deleted from C4.

Case 3: Removing a property from a parent classification

Parent Classification - P

➡ P - Property does not exist.

Child Classifications - C1 and C2

- C1 - Property does not exist
- C2 - Property does not exist

Procedure:

1. Select P.
2. Add a property. *Exists* is set to `True`.
3. Save the classification hierarchy. Property is added to P, C1, and C2.
4. Select P.
5. Set value of *Exists* to `False`.
6. Save the classification hierarchy.
7. Property now gets deleted from P, C1, and C2.

Managing Classification Properties

The main tasks you can perform while working with classification properties are as follows:

- Working with Predefined Values
- Modifying Classification Property Order for Display in Search Results
- Working with Part Classification Properties
- Working with Schematic, Mechanical, and Block Model Classification Properties

Working with Predefined Values

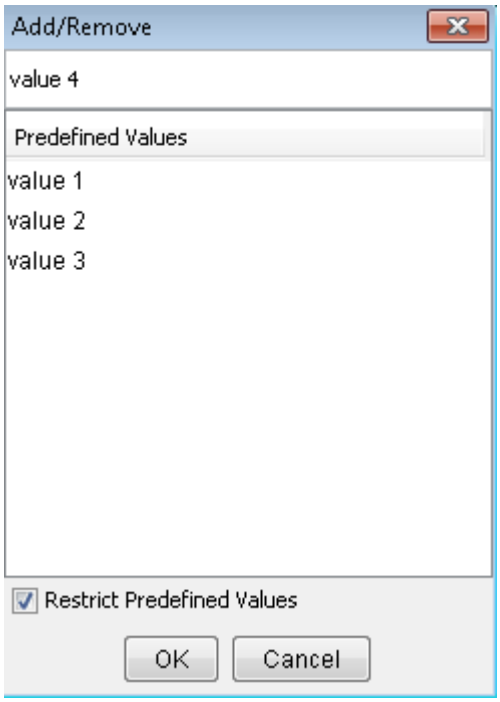
For each new property or an existing property, you can add predefined values. This allows you to select values from a given list. To add these values, do the following:

1. Choose a property row in the right pane.
2. Click next to *Predefined Values*.

   The Add/Remove dialog box appears.

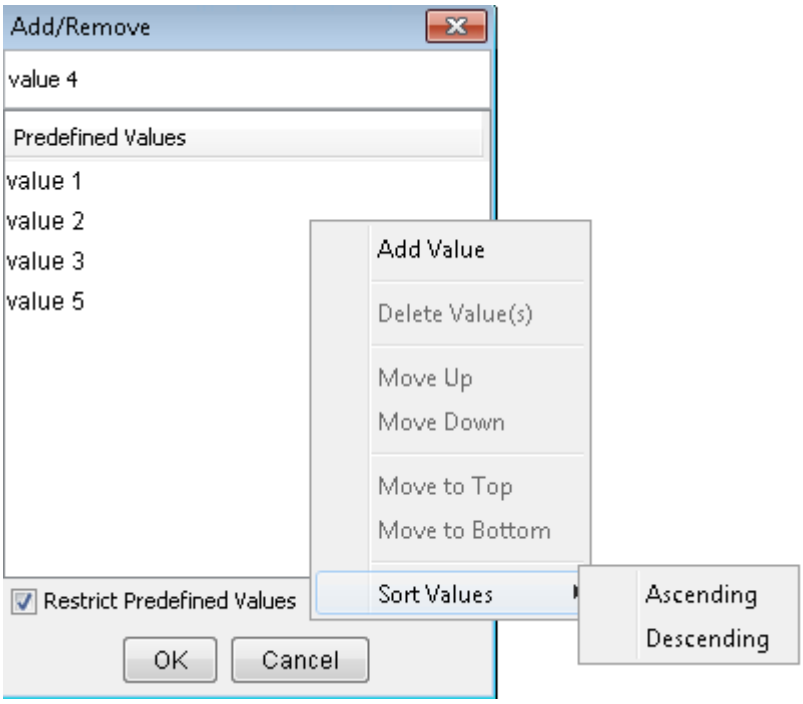
   Note: If there are any existing values for this property, the database retrieves and populates them in the Add/Remove dialog box. If you want to use these values, click *OK*.

3. Add values and press the Enter key. Alternatively, right-click the *Predefined Values* area and choose *Add Value* from the pop-up menu.



4. Choose any value and right-click it to perform any of the following operations, if required:
   - *Delete Value(s)*
   - *Move Up*
   - *Move Down*
   - *Move to Top*
   - *Move to Bottom*
   - *Sort Values – Ascending*

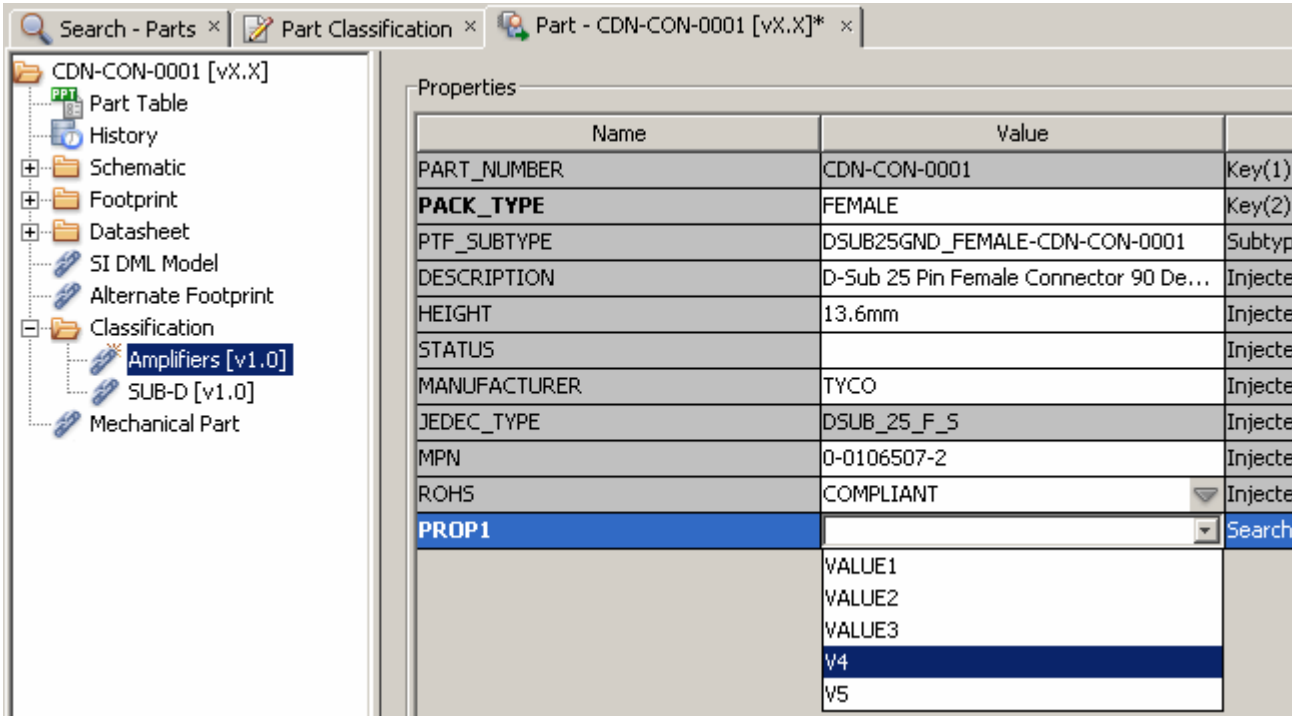
- *Sort Values – Descending*



Note: You cannot delete a value if it is being used in the latest version of part or model.

5. Select the *Restrict Predefined Values* check box if you do not want the librarian to add value other than the ones in the predefined list.
6. Click *OK*.
7. Choose *Save* from the classification node pop-up menu to save changes in the database.

After you associate this classification with a part or block part, you will be able to select from the list of predefined values as shown in the following figure.



Modifying Classification Property Order for Display in Search Results

You can predefine which classification properties should be displayed and in which order when you search library data.

As an administrator, you can:

- Set the order in which properties should be displayed in the search results
- Control whether a certain property should be displayed in the search results

The order defined by the administrator becomes the default property display order that appears in search results in Allegro Library Manager. This ensures that all librarians see a consistent view of the search results.

As a librarian, you can:

- Override the default order and define your own display order. However, the revised display order will be available only for the current session and will not be retained for when Database Editor is launched again.
- Configure an option that allows you to view the object properties in the search results as selected by the administrator.

Administrator Tasks

Administrator tasks are as follows:

- Setting Property Display Order
- Copying Property Display Order
- Exporting Property Display Order
- Importing Property Display Order
- Resetting Property Display Order
- Highlighting Classifications with Modified Display Order

Setting Property Display Order

At a classification node, you can view the union of:

- Properties on the selected parent classification
- Properties inherited from the parent classification
- Properties from its child classifications

You can also identify whether or not a property exists on a selected parent classification node.

As an administrator, this feature enables you to set the display and also order the properties at the highest classification node in the explorer pane. This order becomes the default order of properties for display in the *Search Results* tab.

To predefine the classification property order, do the following:

1. Choose *File – Manage Part Classification*, *File – Manage Block Part Classification*, *File – Manage Mechanical Part Classification*, or *File – Manage Model Classification – `<Model_Type>` Classification*.
2. Click any classification node in the explorer pane.

For example, click the part classification, `Capacitors [v1.0]`. Next, click the subclassification, `Capacitors Fixed[v1.0]`.

On this child classification node, you will see that:

- It shows both the properties inherited from the parent classification and properties at this subclassification.
- When you add a new property to this child classification, it is displayed in the parent classification.
- The *Display in Search* field is set to `True` for all the properties. As a result, all the properties will be visible in the search results.

  Note: The *Display in Search* column value is set to `False` for all the schematic model classification properties. The *Display in Search* setting is `True` for ECAD properties that have the *PTF Mapping* attribute as `Global`, and non-ECAD properties that are searchable. You can change the display settings for other properties, if required.

3. Set the *Display in Search* column value to `False` for the properties that should not be visible in the search results.
4. Select the property name and click the up or down arrow to define the order in which it should appear in the search results. Or, drag and drop the property name to the desired position.
5. Choose *Save* from the classification node pop-up menu to save changes in the database or click the *Save* button.

Important

The display and order that you specify at a classification node is inherited by all its child classifications. However, if you modify the display order at a child classification, and then modify the display order of its parent classification, any change in the order at the parent classification is not inherited by any of its child classifications.

6. Search for objects related to the updated classification. For example, search for parts associated with the part classification `Capacitors Fixed[v1.0]`.
   - The specified property display order is visible in the search results.
   - In addition, all nonsearchable properties will be shown as disabled. You cannot search using any of the nonsearchable properties but can view them in the search results.

Copying Property Display Order

As an administrator, you can copy the display order of properties specified for one classification on to another classification. To do so:

1. Choose a classification node in the explorer pane.
2. Choose *Property Display Order – Copy* from the pop-up menu.

   Note: The *Copy* option is disabled if no display order is specified for the selected classification.
3. Choose the classification node on which you want to copy the display order.
4. Choose *Property Display Order – Paste* from the pop-up menu.

Important

The display order of the properties common to both the classifications is applied to this classification and they appear before the properties that are not common to both classifications.

Exporting Property Display Order

As an administrator, you can export and import the display order of properties on all the classifications.

To export the display order of properties, do the following:

1. Choose the root node.

Important

The Export and Import options are enabled only at the root node.

2. Choose *Property Display Order – Export* from the pop-up menu.

   The Export dialog box opens.
3. Specify the name of the `<property_display_order>.csv` file.
4. Click *Export*.

   The CSV file will be saved in the project directory and will contain the property display order for all the classifications in the explorer tree.

   A message appears to indicate successful completion of export.

5. Click *OK*.

Importing Property Display Order

While importing the property display order, the existing display order for properties is deleted from the database and the display order being imported is applied.

To import the display order of properties, do the following:

1. Choose the root node.

Important

The Export and Import options are enabled only at the root node.

2. Choose *Property Display Order – Import* from the pop-up menu.

   The Import dialog box appears.

3. Choose the `<property_display_order>.csv` file.

4. Click *Import*.

   A message appears to confirm the import because it will delete the existing display order for properties and then apply the display order being imported.

5. If you click *Yes*, a message appears to indicate successful completion of import and the classification node icon changes.

6. Click *OK*.

7. Click the *Save All* button to save the display order changes that have been imported in the database.

Format of Property Display Order File

The property display order file is a comma separated file with a `.csv` file extension. The header contains three columns:

- `Classification Name`: Name of the classification on which the property display order exists.
- `Property Name`: Property name which is ordered.
- `Display In Search`: Indicates if the property is selected for display in search results.

Each entry in this file contains the information for a property on the classification based on these three headers.

The order of the appearance of properties for a classification in this file is the same as the property order on the classification. You can change the ordering and import this file to change the property order on a classification.

Example of a property display order file:

```
"Classification Name","Property Name","Display in Search"
"CAD Component Classification.Electrical Parts [v1.0].Contact Parts [v1.0]","COST LAST UPDATED","False"
"CAD Component Classification.Electrical Parts [v1.0].Contact Parts [v1.0]","EST ANNUAL USAGE","False"
"CAD Component Classification.Electrical Parts [v1.0].Contact Parts [v1.0]","DESCRIPTION","True"
"CAD Component Classification.Electrical Parts [v1.0].Contact Parts [v1.0]","COST","True"
"CAD Component Classification.Electrical Parts [v1.0].Contact Parts [v1.0]","ROHS","True"
```

When you export the property display order for the complete classification tree, the classification entries in the exported `.csv` file have the same property order as seen in the *Properties* tab.

Resetting Property Display Order

As an administrator, you can reset the display order of properties on the selected classification and its children.

To reset the property display order, do the following:

1. Choose a classification node in the explorer pane.
2. Choose *Property Display Order – Reset* from the pop-up menu.

   A message appears to confirm the reset operation because it will permanently remove the existing property display order on this classification and its children.
3. Click *Yes*.

   The display order preferences are reset and the classification node icon changes.
4. Click the *Save All* button.

Important

After this operation, if the parent classification of the selected classification node has an order defined, that order will be inherited by the selected classification and its children.

Highlighting Classifications with Modified Display Order

As an administrator, you can highlight the child classifications for which you have specified a display order.

To identify the child classifications which have a property display order set, do the following:

1. Choose a classification node in the explorer pane.
2. Choose *Property Display Order – Highlight Modified Child Classifications* from the pop-up menu.

   A message prompts you that the selected classification node has been expanded and all the child classifications under this node have been highlighted to indicate that the property display order has been modified for them.
3. Click *OK*.

Sorting, Filtering, and Defining Display Order of Classifications

You can define the property order of classifications by sorting columns as required. For example, you can sort on any column in the Properties or Information Attributes panels. After modifying the property order by moving columns up or down, save the changes.

Important

If you define the property order for a schematic classification, the order is propagated to the part classification as well. If the part classification already had an order defined, the schematic classification order overwrites the part classification order.

If you select more than one classification and define a property order, then save, the order you define is applied to all the selected classifications.

If you have modified data in the Properties or Information Attributes panels, then sorted the columns but only want to save the data changes without saving the sort, click the *Discard Order* button (to the right of the Properties panel).

If you select more than one classification, Allegro EDM displays the properties in the property order you defined.

You can also filter the attributes or properties of a classification.

Managing Librarian Tasks

As a librarian, at a selected classification node, you can see the union of properties on the selected classification, properties inherited from its parent classification, and properties from its child classifications. This enables you to search on properties from any top-level classification node in the explorer pane.

In addition, in the Search Results tab, you can also see the property display order specified by the administrator. However to be able to see the properties as selected by the administrator, you need to do the following:

1. Choose *Edit – Options.*

   The Options dialog box appears.

2. Select the *Commands* tab.

3. In the *Search options* section:

   - Select the *Show only part properties selected by administrator* check box to see only the part properties selected by the database administrator.
   - Select the *Show only model properties selected by administrator* check box to see only the model properties selected by the database administrator.

4. Click *OK*.

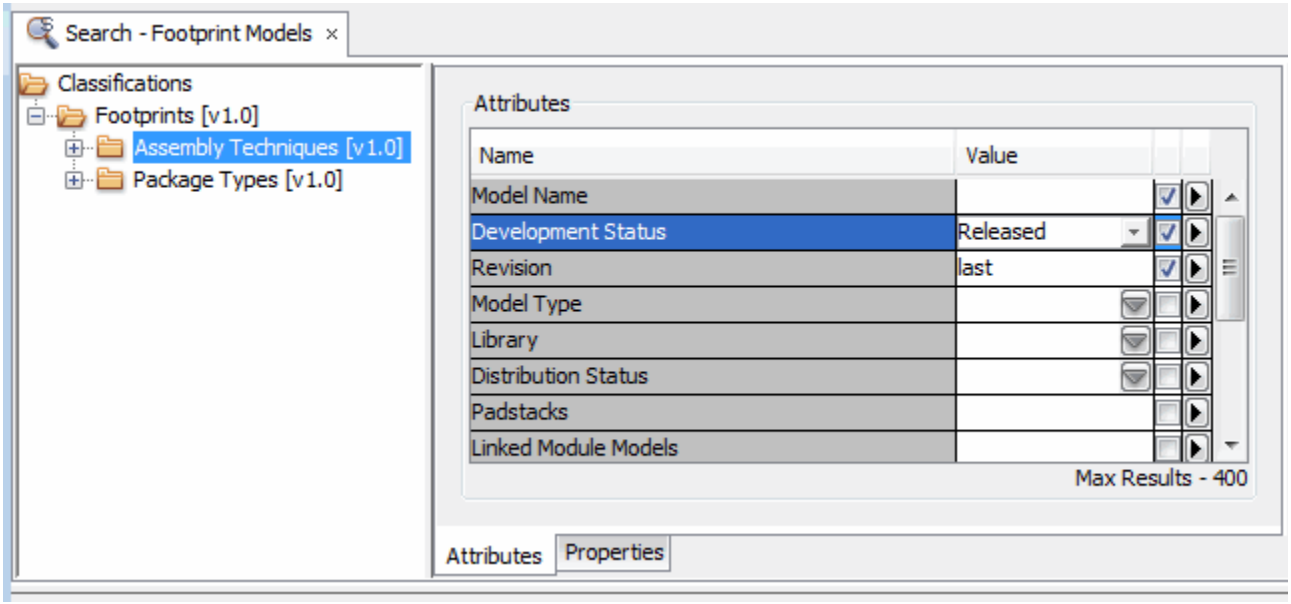
You can also override the predefined property display order and define your own order. To do so:

1. Choose *Search – Part, Search – Block Part, Search – Mechanical –* `<Mechanical Part or Kit>,` or *Search – Model –* `<Model_Type>.`

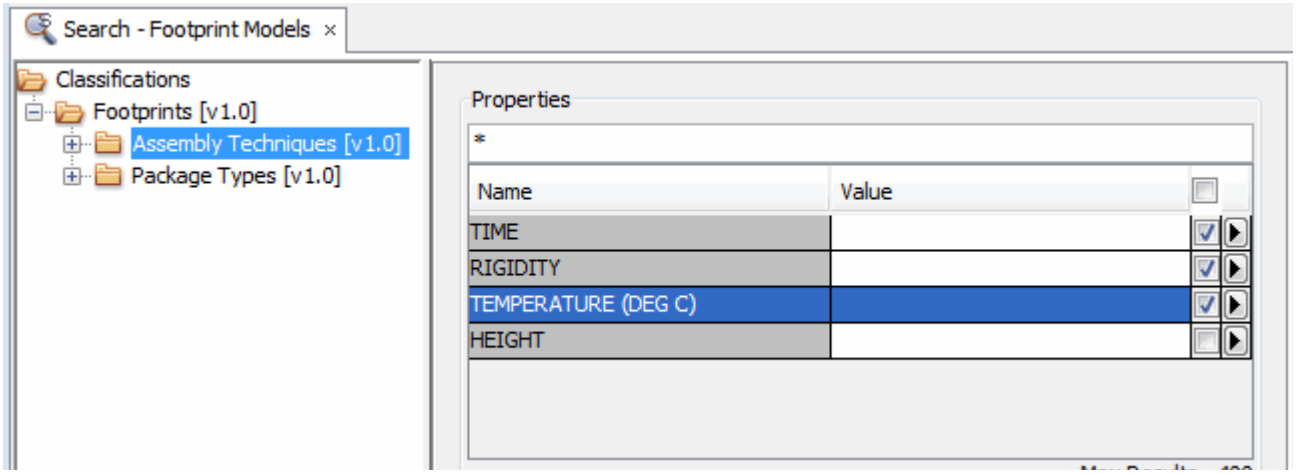
   For example, choose *Search – Model – Footprint Model*.

2. Choose a classification node.

3. Enter the value next to the an attribute under the *Value* column in the *Attributes* tab. In addition, you can also specify the order in which the attributes should appear in the search results.

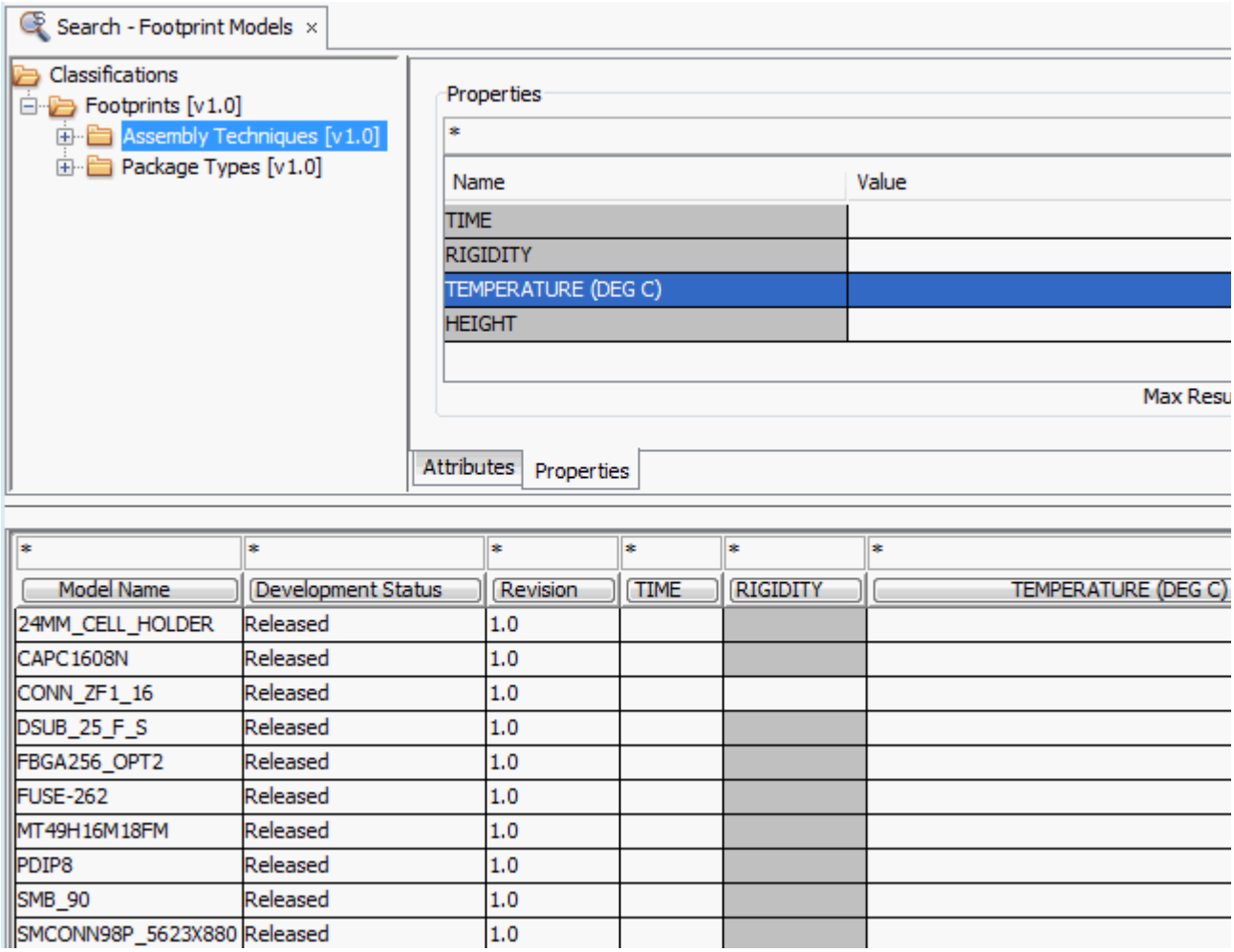


4. Click the *Properties* tab.
5. Enter the value next to a property (under the *Value* column).
6. Select and/or clear the check box to set the display of properties in the Search Results tab.
7. Select a property name and then click the up or down arrow to define the order in which it should appear in the search results. Or, drag and drop the property name to the desired position.



8. Click  .

   The property names appear in the *Search Results* tab according to the modified display order.



Restoring Default Property Order

To restore the default property order, you need to close this session and launch Database Editor again. The property display order preferences as defined by the administrator will be restored if you have configured the use of the predefined display order as set by the administrator.

Working with Part Classification Properties

For classifications, you can specify properties, which in turn, are transferred to the parts, block parts, or models specified under them.

The tasks you can perform with classification properties are:

- Adding Properties
- Editing Properties
- Deleting Properties

However, if you are working with a Schematic Model, see Working with Schematic, Mechanical, and Block Model Classification Properties.

Important

You can add, edit, and delete properties for classifications not linked to a part, block part, or model.

Adding Properties

To add a property, do the following:

1. Choose a classification node in the explorer pane.

   The default properties, if available, appear in the right pane.

2. Right-click the right pane, choose *Add* from the pop-up menu. Alternatively, click the *Add* () button.

   The Add dialog box appears.

3. Enter the name of the new property, and click *Create*.

   The property is added and appears in a new row in the right pane, and a New interface property icon () appears against the property row.

4. Enter appropriate values in the row.

   You can provide the following information for a property:

   - Mandatory
   - Searchable
   - Data Type

- ❑ Default Value
- ❑ Predefined Values

To learn how to add or enable predefined values, see Working with Predefined Values.

5. Click up or down arrow to define the order in which new property (which appears as the last property) should appear in the search results.

   Note: If you change the display order of properties, the part classification is updated and not revised.

6. Choose *Save* from the classification node pop-up menu to save changes in the database or click the *Save* button ( ).

   If you have made more than one change to a hierarchy, then select the parent classification and choose *Save Hierarchy* from the pop-up menu to save all the changes.

Editing Properties

To edit a property, do the following:

1. Select a classification node in the explorer pane.

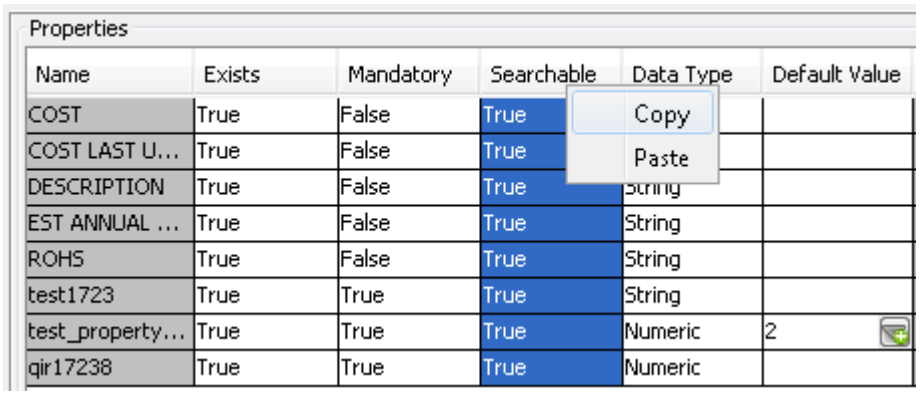
   The default properties, if available, appear in the right pane.

2. Select a property row in the right pane and edit the values.

   You can also copy-paste column values from one column to another. To do so:

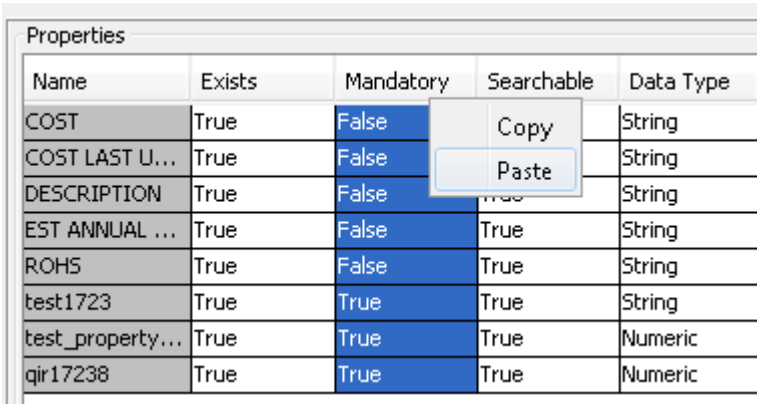
   a. Select a column header.

   b. Right-click and choose *Copy*.

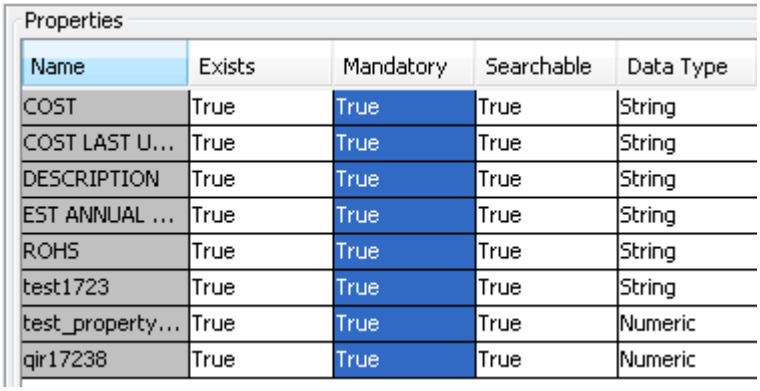


   c. Select the column header to which you want to copy the selected values.

d. Right-click and choose *Paste*.



You can see values in all the property rows get modified using this operation.



3. Select a property row and right-click to choose *Rename* from the pop-up menu. Alternatively, click the *Rename* ( ) button.

Note: You cannot rename a property if it is a PTF Property (that is, *ECADType* is set to `True`). Such a property can only be renamed from the schematic, block, or mechanical model classification it is associated with.

Important

For each property you can enable *Predefined Values*. To learn how to add or edit predefined values, see Working with Predefined Values.

4. Click up or down arrow ( ) to define the order in which a property should appear in the search results.

5. Choose *Save* from the classification node pop-up menu or click the *Save* button ( ) to save changes in the database.

If you have made more than one change to a hierarchy, then select the parent classification and choose *Save Hierarchy* from the pop-up menu to save all the changes.

Note: An asterisk (*) next to a node in the explorer pane signifies unsaved classification.

Deleting Properties

To delete a property, do the following:

1. Select a classification node in the explorer pane.

   The default properties, if available, appear in the right pane. If there are no values, the right pane is grayed out.

2. Select a property by clicking the corresponding row in the right pane.

3. Choose *Delete* from the pop-up menu or click the *Delete* () button in the right pane.

   As soon as you delete a property row, it is grayed out and a delete mark icon () appears against the property row.

4. Choose *Save* from the classification node pop-up menu to save changes in the database or click the *Save* button.

   The property row disappears from the right pane.

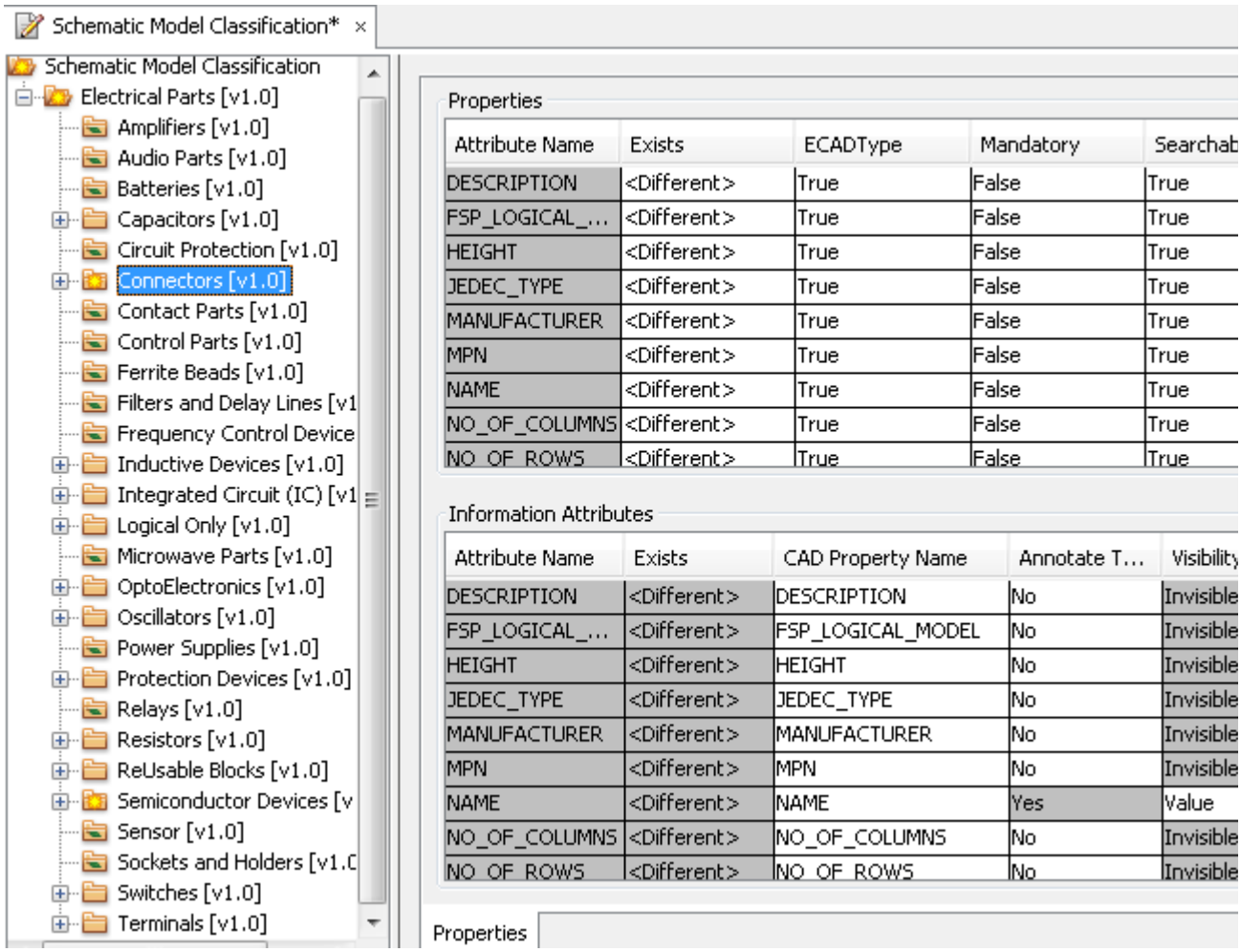
   If you have made more than one change to a hierarchy, then select the parent classification and choose *Save Hierarchy* from the pop-up menu to save all the changes.

Working with Schematic, Mechanical, and Block Model Classification Properties

A schematic, mechanical, and block model is an ECAD model, and hence plays a special role in the generation of PTF files for your library data, which is why schematic, mechanical, and block model classification properties are defined in a slightly different manner. The Capture model classification is also defined similarly. However, it is does not impact the PTF generation process.

Note: Because Value is a key property in Capture-CIS, for classifications associated to parts which will be written to the Capture CIS database, the Value property in the Schematic and Part Classifications is recommended. It is also recommended that you make this a mandatory property.

For the purpose of explanation, only managing schematic model classification properties is covered in this section.



The right pane in the *Schematic Model Classification* tab contains two panes. The upper pane displays the properties for the model, and the lower pane, *Information Attributes* contains the ECAD properties.

You can specify the following information for each schematic model classification property:

- *Exists*: Identifies whether or not a property exists on a selected classification node. For each property, the *Exists* column value signifies the following:
  - `True`: Indicates that the property exists on the parent and all its child classifications
  - `<Different>`: Indicates that the property exists on some of the classifications

    Note: For a classification property, when you place the mouse over the column value, `<Different>`, the tooltip displays the list of classifications which contains this property.
- *ECADType*: Specifies if the property is of the type ECAD. Only an ECAD property can appear in the PTF.
- *Mandatory*: Specifies whether or not a property is mandatory. If the value of a property is `True`, you need to specify a value in the *Default Value* field.
- *Searchable*: Specifies if you can search objects on the basis of this property.
- *Data Type*: Specifies if the property value is string or numeric.
- Default Value: Specifies the default value of the property.
- Predefined Values: Specifies the permissible set of values for the property.

  For each property, you can enable *Predefined Values*. To learn how to add or edit predefined values, see Working with Predefined Values.
- *Display in Search*: By default, this field is set to `False` for all the schematic model classification properties. The *Display in Search* setting is `True` only for:
  - ECAD properties that have the *PTF Mapping* attribute set as `Global`
  - Non-ECAD properties which are searchable

  You can change the display settings for other properties, if required.

Information Attributes Pane

- *Exists*: Identifies whether or not a property exists on a selected classification node.
- *CAD Property Name*: If *ECADType* is `True`, then only this property:
  - is visible in the *Informational Attributes* section of the right pane under the *Attribute Name* column.
  - appears in the PTF

The name of this property that appears in the PTF is the one you specify in the *CAD Property Name* field. For example, the property name in the *Attribute Name* column is `DESCRIPTION` and in the *CAD Property Name* column, it is `DESC`, then the property name that will appear in the PTF will be `DESC`.

- *Annotate to Design*: Indicates whether or not the property is annotated to the design.
  - Key properties are annotated to the design by default. You cannot modify this field.
  - Injected properties are not annotated to the design by default. You can modify this field.
- *Visibility*: Indicates whether or not a property that is annotated to the design will be visible in the design.
  - Key properties are editable by default.
  - Injected properties are invisible in the design by default. You can edit this.
- *PTF Mapping*: Specifies if the property is of the following type:
  - Key
  - Injected
  - Optional Key
  - Key & Injected
  - Optional Key & Injected
  - Global
  - Added
  - Subtype
- *Key Property Order*: If you selected a property as `Key` in the *PTF Mapping* field, then use this field to specify its order in the PTF.
- *Optional Property Value*: If you selected a property as `Optional Key` in the *PTF Mapping* field, then this field is enabled to specify its optional value.
- *Link To*: Allows you to link the ECAD property to another classification's property (one that is associated with another part or model) or an attribute of a part or model. This ECAD property will then use the value of property that you have specified in the *Link To* field while generating PTF table. The format to be used is any combination of:

```
<SampleText>, $prop(<Model Type>.<Property Name>); <SampleText>
$prop(<Model Type>.<Property Name>)
```

The tasks you can perform with Schematic Model Classification properties are:

- Adding Schematic Model Classification Properties
- Editing Schematic Model Classification Properties
- Deleting Schematic Model Classification Properties
- Working with Global Property
- Working with Link To Property

Important

You can add, edit, and delete properties for schematic model classifications that are not linked to a part or model.

Adding Schematic Model Classification Properties

To add a property, do the following:

1. Choose *File – Manage Model Classification – Schematic Model Classification*.

   The *Schematic Model Classification* tab appears.

2. Select a classification node in the explorer pane.

   The default properties and the information attributes appear in the respective areas of the right pane.

3. Right-click the right pane and choose *Add* from the pop-up menu or click the *Add* () button.

   The Add dialog box appears.

4. Enter the name of the property to add and click *Create*.

   The property is added and appears in a new row in the *Properties* area, and a new interface property icon () appears beside the property row.

5. Enter appropriate values in the row.

   Note: When you add an *ECADType* property to a schematic model classification, this property will be added to the part classifications associated with all the parts linked to the schematic models associated with this schematic model classification.

6. If you set the *ECADType* property to `True`, the newly added property appears in the *Informational Attributes* area. Specify values for the ECAD properties that get listed in the *Informational Attributes* area.

Important

Depending on the *PTF Mapping* column value you specify, the *Key Property Order* and *Optional Property Value* columns may or may not be editable.

Note: For Schematic Model Classification, the *CAD Property Name* is used as it is in the PTF (generated by the PTF Generator utility). Therefore, ensure that the property name specified is valid according to PTF property naming convention. For more information on part table files, see *Part Table Editor User Guide*.

7. Click up or down arrow to define the order in which the newly added property (which appears as the last property) should appear in the search results.

   Note: If you change the display order of properties, the schematic model classification is updated and not revised.

8. When you have entered all the property values, choose *Save* from the pop-up menu to save changes into the database. Alternatively, if you have made more than one change to a hierarchy, select the parent classification and choose *Save Hierarchy* from the pop-up menu to save all the changes.

Note: An asterisk (*) next to a node in the explorer pane signifies unsaved classification.

Editing Schematic Model Classification Properties

The editing process is similar to adding properties where you can not only add new properties but can also modify existing properties. For more information on how to do this, see the Adding Schematic Model Classification Properties and editing part classification properties (Editing Properties) sections.

Important

While editing Schematic Model Classification, ensure that no duplicate Key Property Orders exist between a classification and any of its subclassifications. In such cases, using the *Save Hierarchy* and the *Save* commands display an error asking you to specify unique key property order values.

Deleting Schematic Model Classification Properties

To delete an existing property, do the following:

1. With the *Schematic Model Classification* open, select a classification node in the explorer pane.

The default properties and the information attributes appear in the respective areas of the right pane.

2. Select a property in the *Properties* or the *Informational Attributes* area.
3. Choose *Delete* from the pop-up menu or click the *Delete* button in the right pane.

   As soon as you delete a property row, it is grayed out and a delete mark icon () appears against the property row.
4. Choose *Save* from the pop-up menu to update the database or click the *Save* button.

   If you have made more than one change to a hierarchy, select the parent classification and choose *Save Hierarchy* from the pop-up menu to save all the changes.

Working with Global Property

To specify or modify the value of a global ECAD property, do the following:

1. With the *Schematic Model Classification* tab open, select a classification node in the explorer pane.

   The default properties and the information attributes appear in the right pane.
2. Select an ECAD property in the *Informational Attributes* area.
3. Set the column value of *PTF Mapping* for the selected property to `Global`.

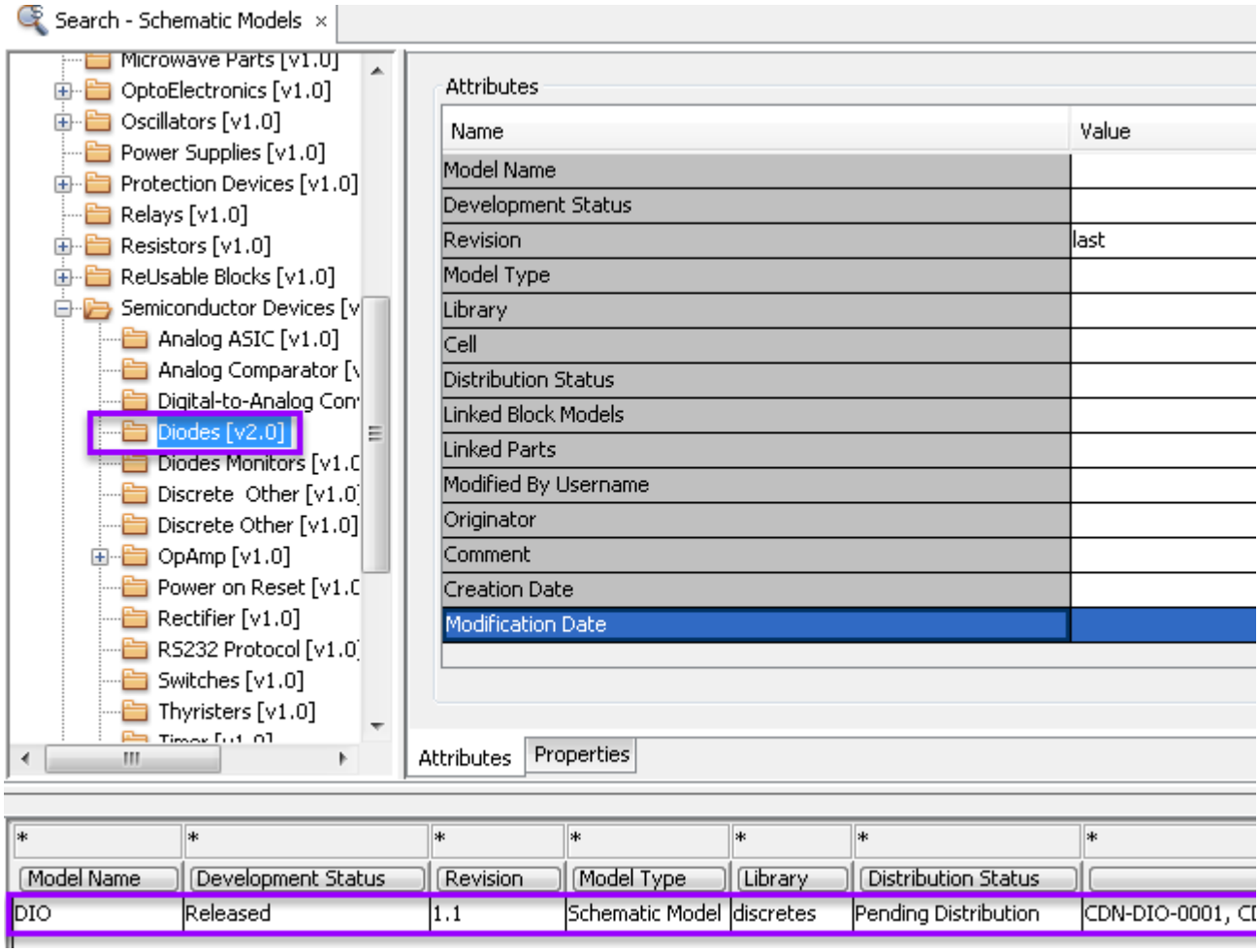
Important

Note that as the property value is changed to `Global`, the *Link To* value is automatically changed to `$prop(Schematic Model.<Selected_Property_Name>)` and becomes non-editable.

Information Attributes

Attribute Name	Exists	CAD Property Name	Annotat...	Visibility	PTF Map...	Key Pro...	Optional ...	Link To
CURRENT	True	CURRENT	No	Invisible	Injected			
DESCRIPTION	True	DESCRIPTION	No	Invisible	Injected			
HEIGHT	True	HEIGHT	No	Invisible	Injected			
JEDEC_TYPE	True	JEDEC_TYPE	No	Invisible	Injected			$prop(F
MANUFACTURER	True	MANUFACTURER	Yes	Value	Global			$prop(S
MPN	True	MPN	No	Invisible	Injected			
PACK_TYPE	True	PACK_TYPE	Yes	Value	Key	2		
PART_NUMBER	True	PART_NUMBER	Yes	Value	Key	1		$prop(F
PTF_SUBTYPE	True	PTF_SUBTYPE	No	Invisible	Subtype			

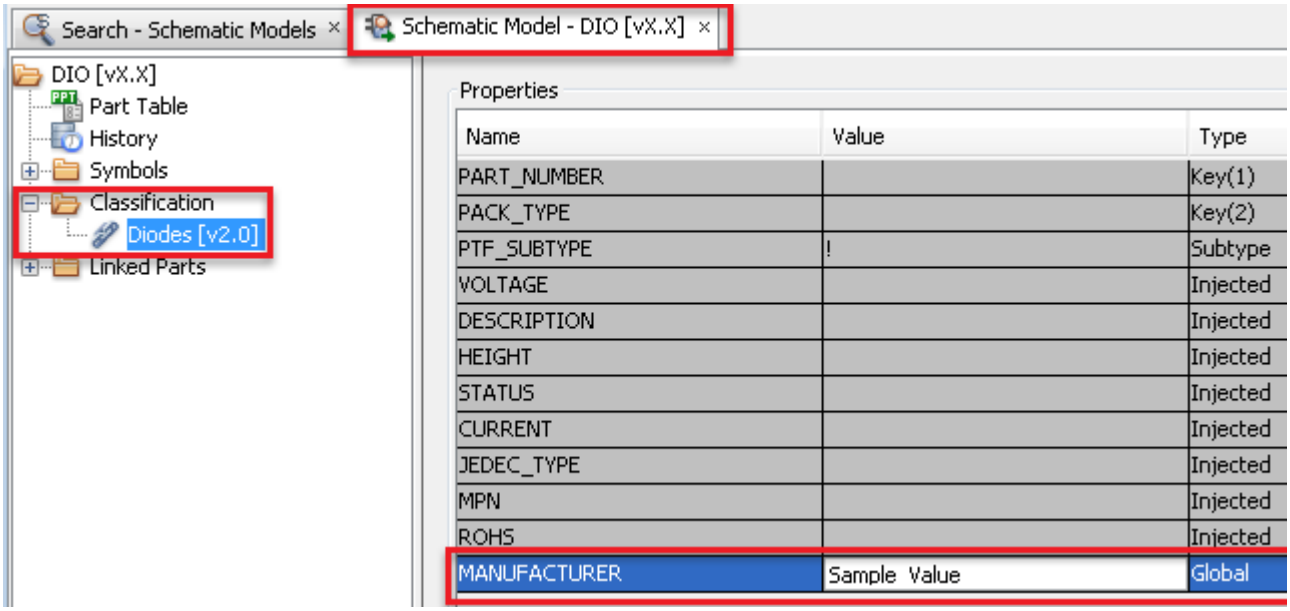
4. Search for schematic models associated with the selected schematic model classification.



5. Select and check out a schematic model from the search results.

6. Select the schematic model classification associated with the checked-out schematic model.

7. Specify a value for the ECAD property.



8. Click the *Save* button.
9. Check in and release the schematic model.

Alternatively, perform the following tasks:

1. With the *Schematic Model Classification* tab open, select a classification node in the explorer pane.
2. Select an ECAD property in the *Informational Attributes* area.
3. Set the column value of *PTF Mapping* for the selected property to `Global`.
4. In the *Properties* area, specify the required value in the *Default Value* column for the property selected in step 2.
5. Click the *Save* button.

Working with Link To Property

You can define an ECAD property value based on the values of other properties. The *Link-To* property allows you to define the value of a particular property as a combination of other properties or attributes of a part or model. You can map that property to a relation.

The format that can be used is any combination of:

`<SampleText>, $prop(<Model Type>.<Property Name>); <SampleText> $prop(<Model Type>.<Property Name>)`

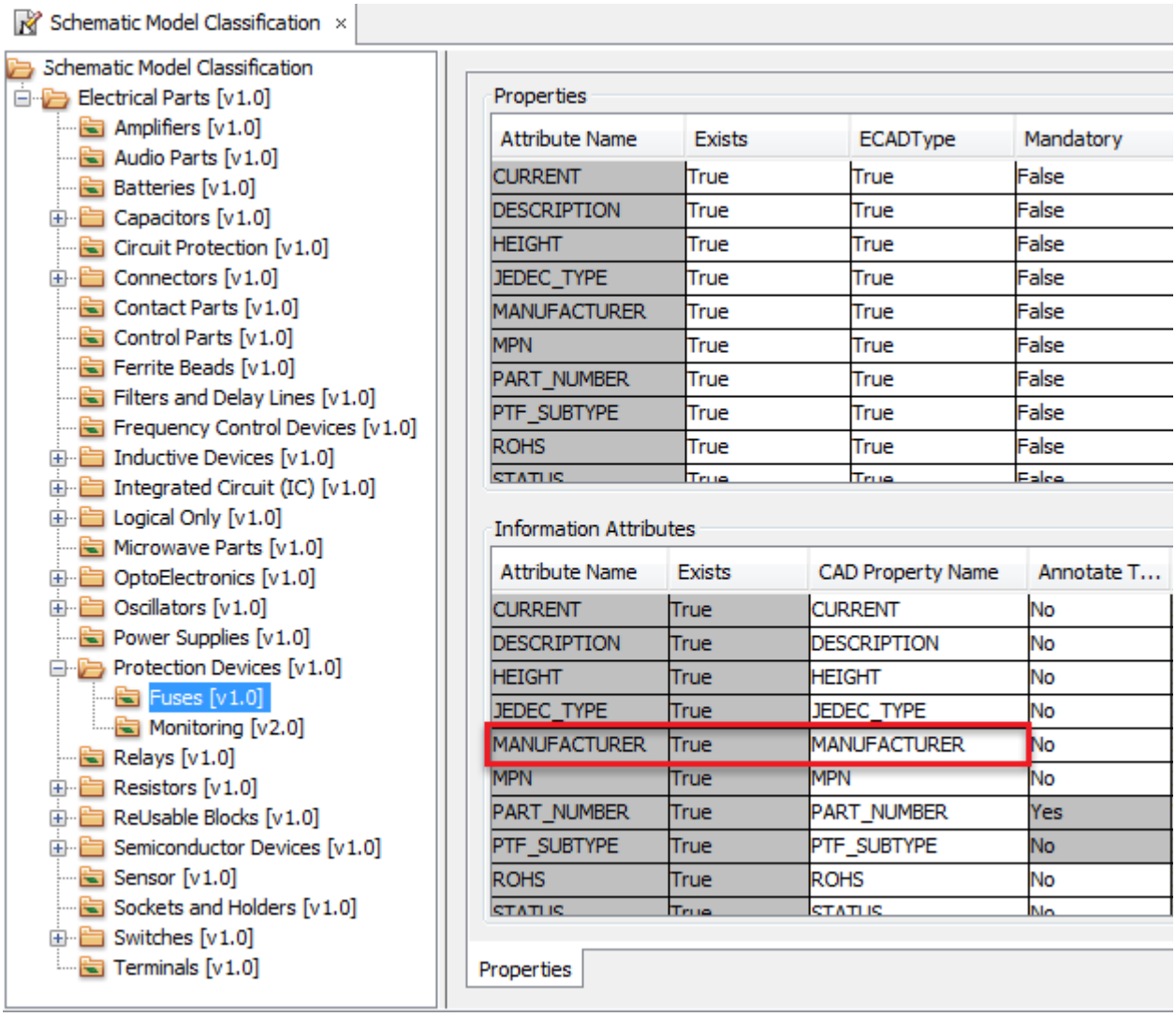
To specify or modify the value of the *Link To* field, do the following:

1. With the *Schematic Model Classification* tab open, select a classification node in the explorer pane.

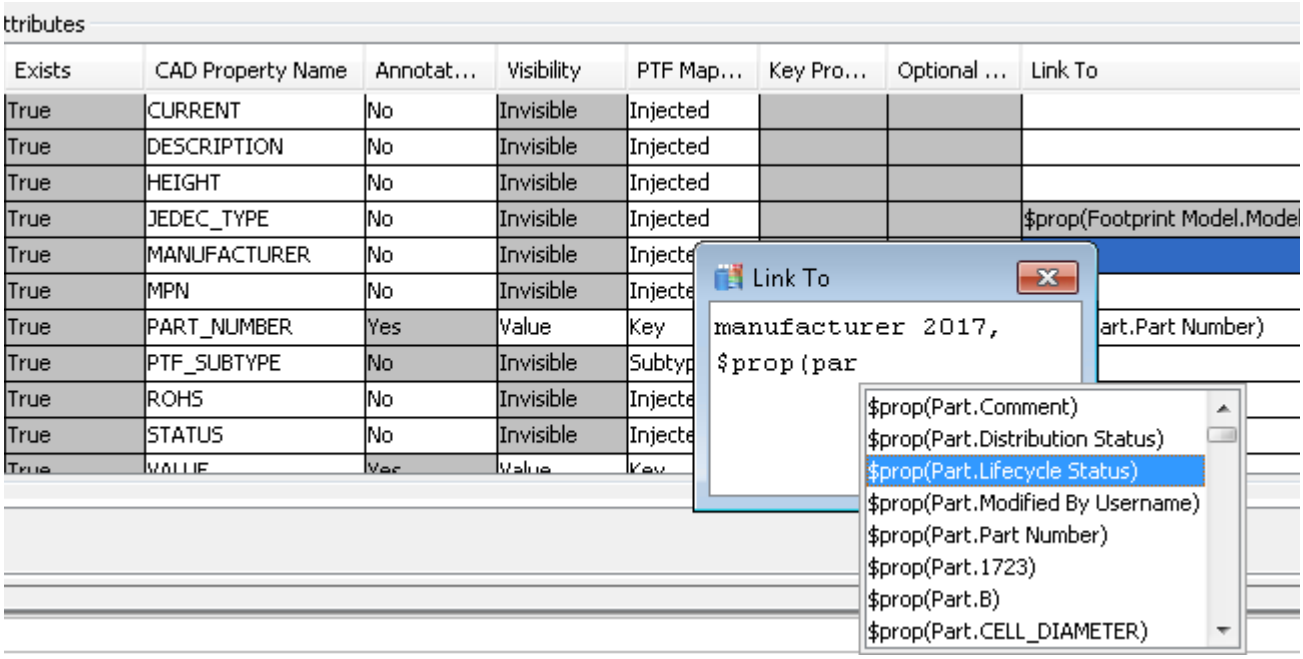
   The default properties and the information attributes appear in the right pane.

2. Select an ECAD property in the *Informational Attributes* area.

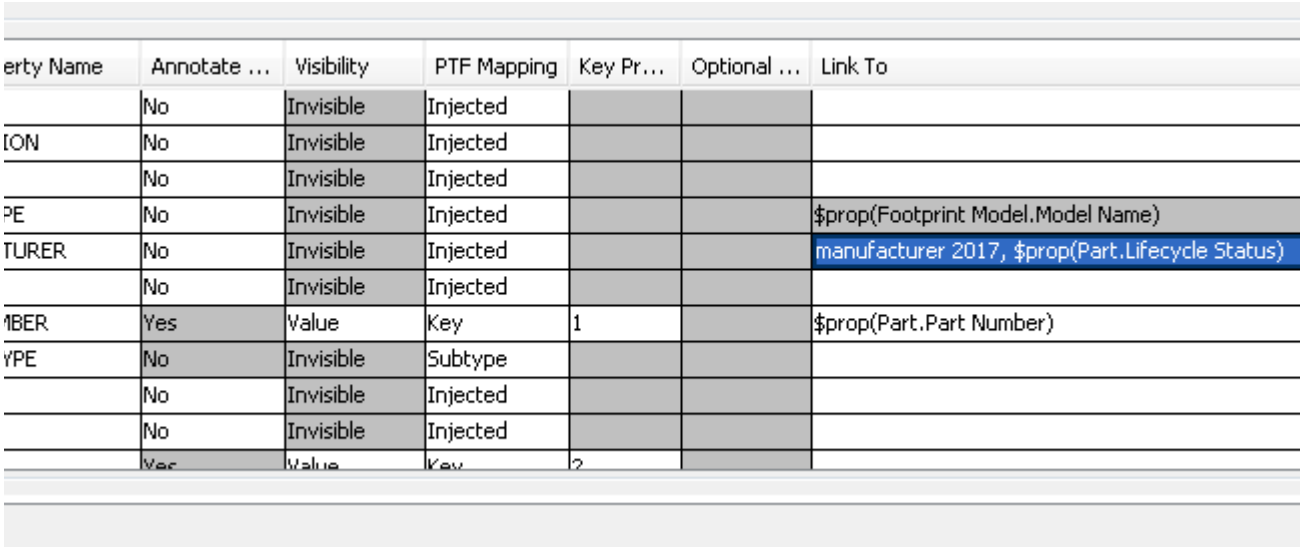
3. Click the *Link To* field corresponding to the selected ECAD property. The *Link To* dialog box opens with an example that can be used to specify the combination of values.



As you start typing `$`, you can see an auto-suggestion list box.

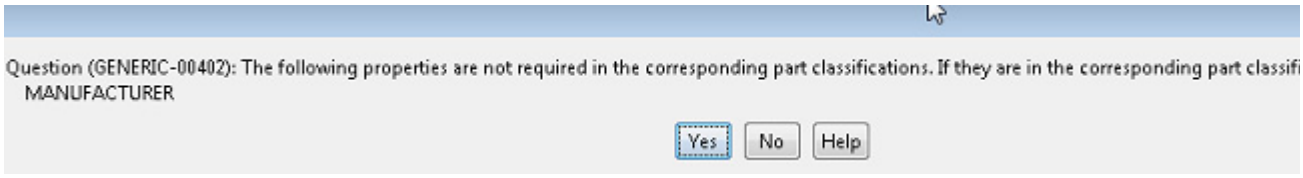


4. Select an item from this list box and double click it. The *Link To* dialog box is updated. Press Enter after you have completed specifying the various values, in any combination of: `$prop(<Model Type>.<Property Name>), <SampleText>`



5. Click the *Save* button.

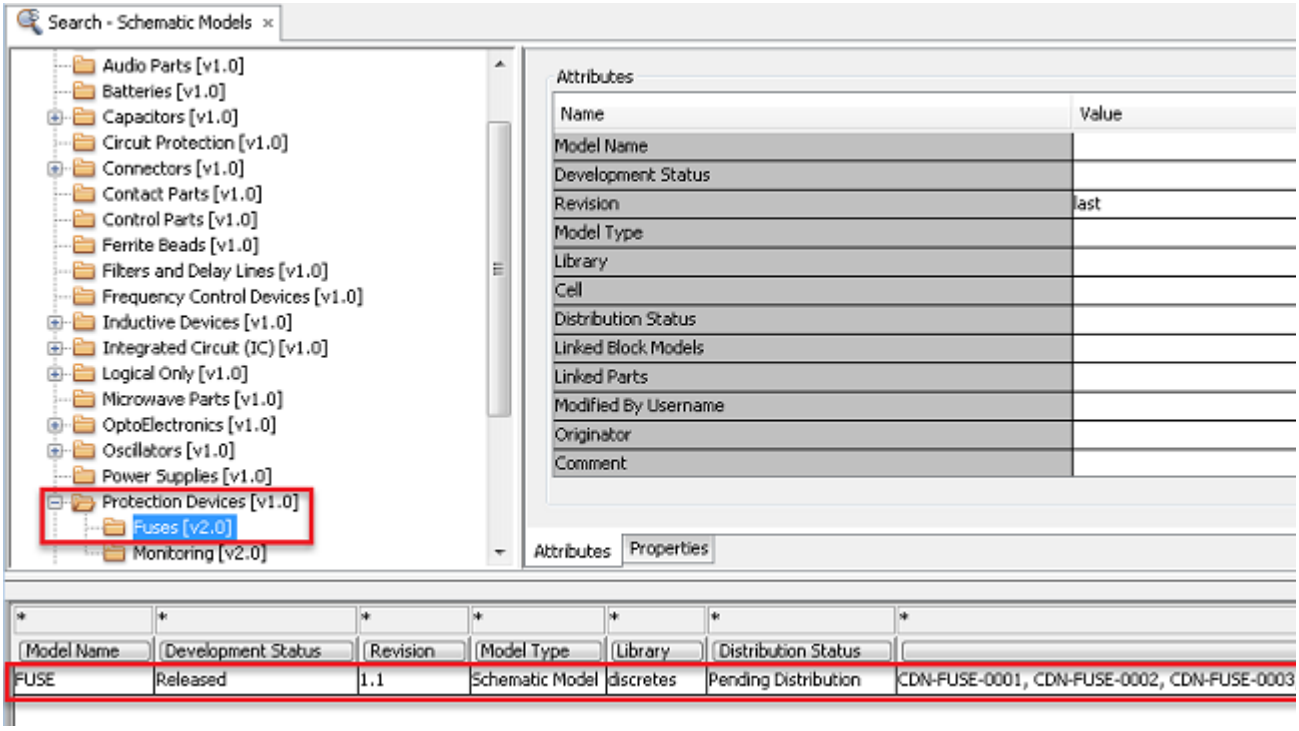
A message prompts you to check if the value of this property (if existing) in the part classification of the linked parts is to be overridden.



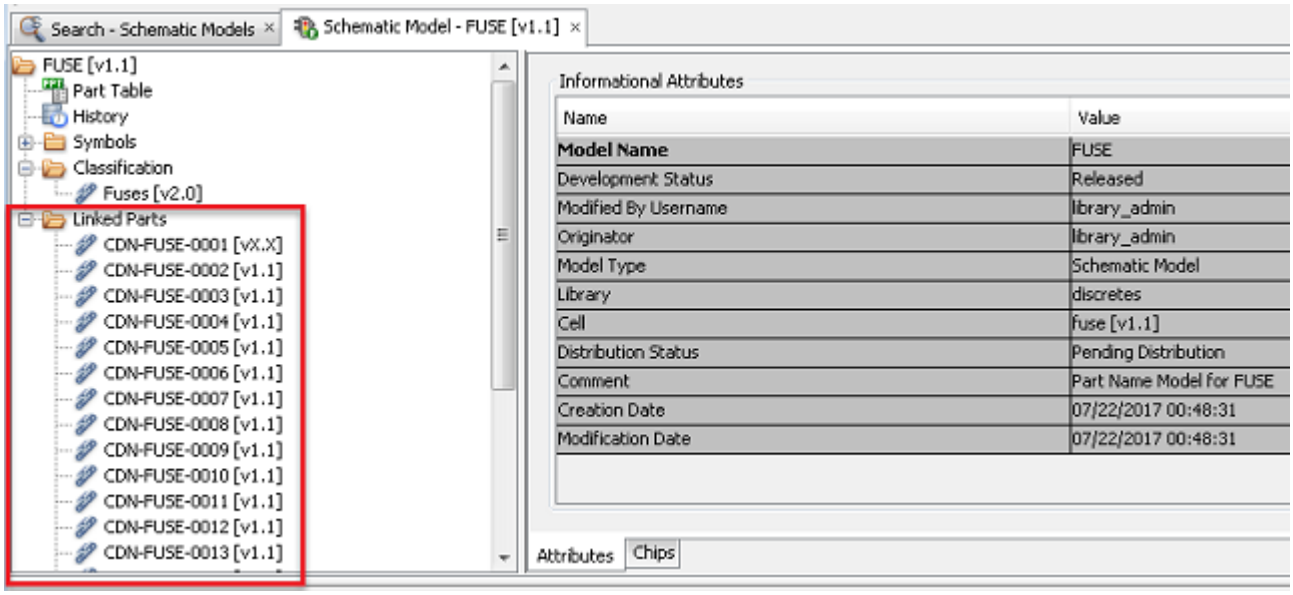
6. Click *Yes* to ensure that the value specified in *Link To* is reflected in the PTF.

To check if the property value in the part classification has been updated, do the following:

1. Search for the schematic models associated with the schematic model classification for which the *Link To* value was specified corresponding to an ECAD property.

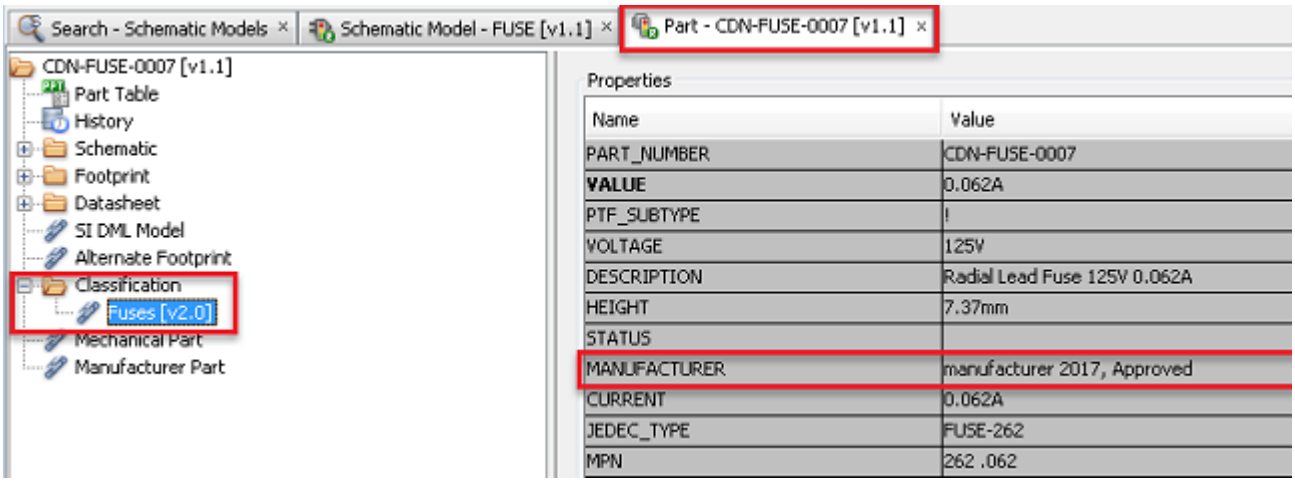


2. Select and open a schematic model from the search results.



3. Select and open a linked part.

4. Click the associated part classification to observe the updated value of the property.



4

Updating Data in Allegro EDM Component Database

Librarians can update (for example, apply a property to all parts) existing data in the Allegro EDM component database using:

- Allegro EDM Data Exchange - this is done with processing rules and in batch mode. As a result, it does not require the manual intervention of a librarian.
- Database Editor - allows librarians to visually compare updated data with the data already in the component database and decide whether they want to merge the differences

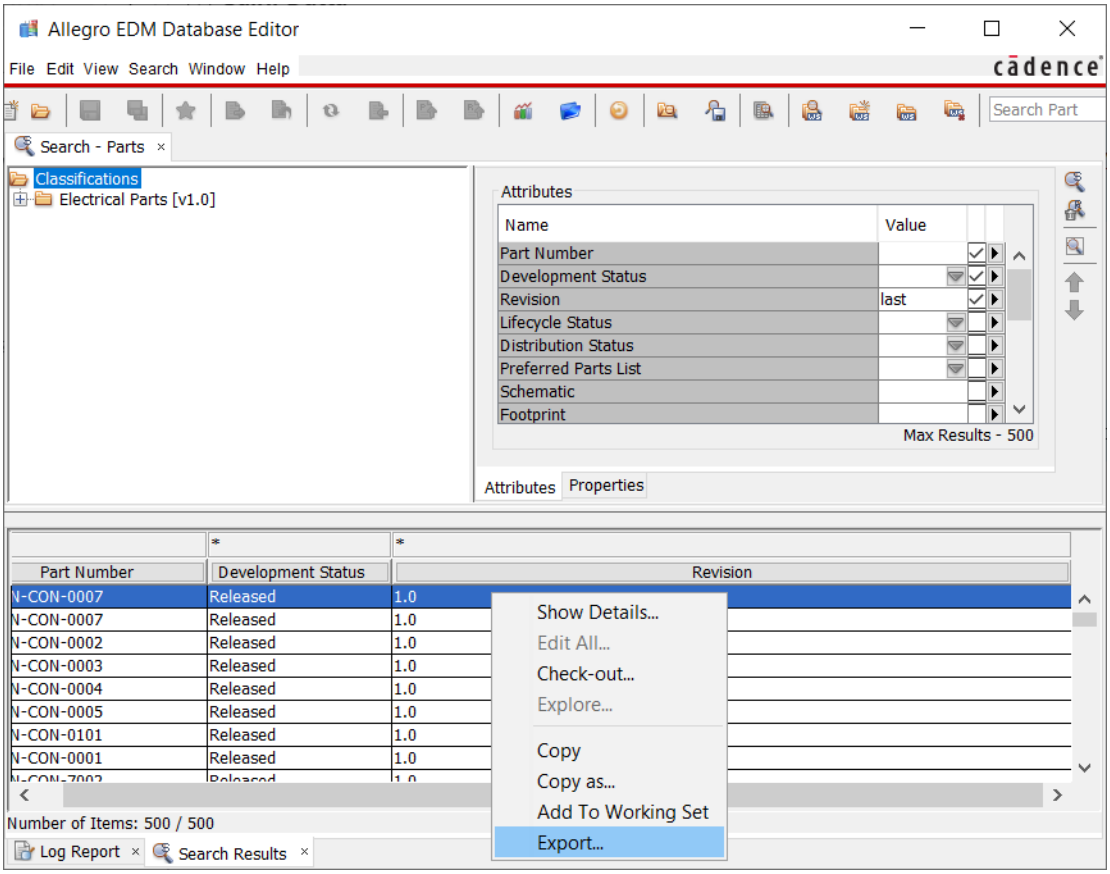
Comparing Incoming and Existing Data in Component Database

To compare new or updated data (that is, the input data) with the data already in the component database, you need the following:

- An XML file that contains the data that needs to be bought into the EDM component database

Caution

To ensure that the XML file has the correct structure and is in the Allegro EDM format, do not create the XML file from scratch. Instead, export the required data type and follow the same syntax. To export data, right-click on the object in the working set and select Export.



- Proper setup of the sync system, available at `<PCBDW_LIB>/exchange/sync`. This system has the configuration options defined for each data source.
- Appropriate rules and conditions in a file, `sync.xml`, which contains processing rules to synchronize the incoming data with the data already in the database

For details on XML files for the synchronization of data and the `exchange` area, refer to *Allegro EDM Data Exchange Reference Guide*.

Importing Updated Parts or Part Data with Database Editor

After you ensure that your data in XML format is available, and that you have set up the `exchange/sync` area as well as the `sync.xml` file, import the data by doing the following:

1. Ensure that the XML has the following entry:

   `<AllData source="<value>">`

   Where `<value>` is the name for a folder in `../exchange/sync/<value>`.

   This `<value>` is the sync system that Database Editor will use to import the XML into the database.

   For example, a file called `export.xml` is to be imported with Database Editor. This file has the following data:

   ```
   <?xml version='1.0' encoding="UTF-8" ?>
   <!-- Generated on Wed Aug 08 10:21:30 IST 2012 -->
   <AllData source="adw">
   ```

   The logic for mapping the incoming XML will be looked for in:

   `<PCBDW_LIB>\exchange\sync\ADW\configuration\sync.xml.`

2. Check that a corresponding `sync.xml` exists.

   Look for `<PCBDW_LIB>\exchange\sync\ADW\configuration\sync.xml`.

3. Check the rules in the `sync.xml` file. For details, see *Allegro EDM Data Exchange Reference Guide*.

4. Start Database Editor.

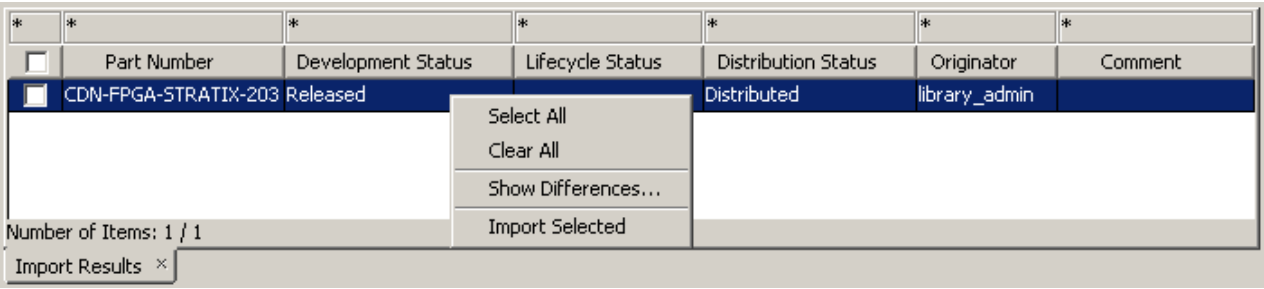
5. Choose *File – Import*.

6. Choose the file to import. You can select a CSV or XML file, or a zip file, which contains a CSV or XML file with the data you want to import.

7. Click *Open*.

   Parts with different data in the XML and in the database are displayed.

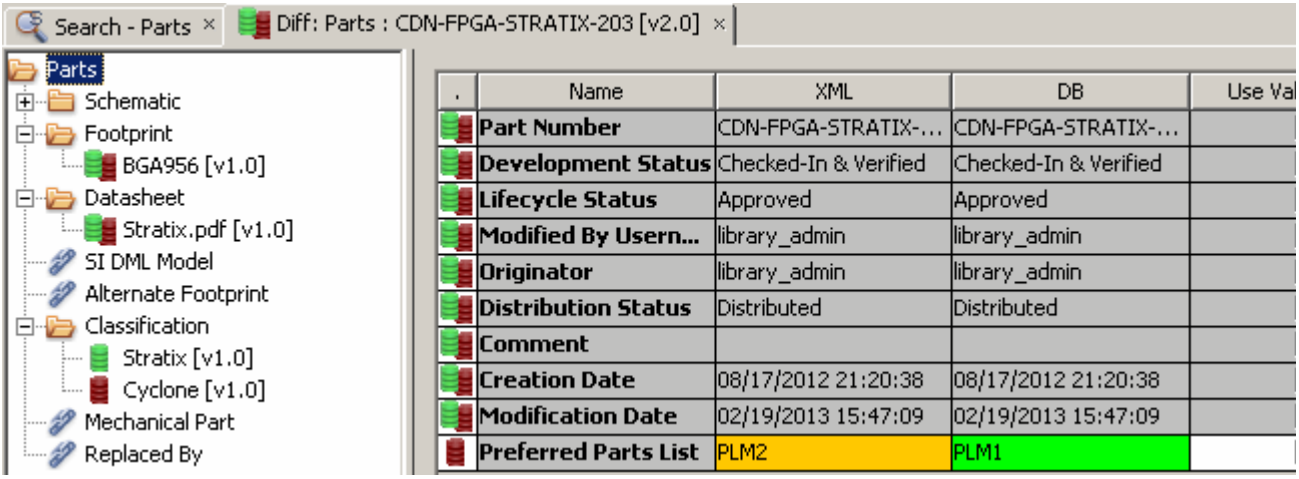
8. Right-click any row.



From here, you can:

- Select all the parts that have conflicting information. See Synchronization Conflicts for details of what is considered a conflict.
- Remove the selections made.
- See the differences between the values in the XML and the database. You can view part and model differences.
- Import the selected rows from the XML file into the database.

9. Select *Show Differences* from the context menu. The differences are displayed.



There are four columns in this tab:

- Name
- XML
- DB

- Use Value in XML

Values from the XML are shown in red and values from the database are in green.

- Green: Indicates that this value will be updated in the database
- Red: Indicates that this value will be ignored

Select the rows where you want to add the XML value to the database. If you do not select the check box, the value from the database is retained.

The left panel shows the source of the information:

- If data is found in the XML and the database, and their values are different, you will see two rows.

- If the data is from just one source, you will see one row, and the icon depicts the source of the data.

10. Select the check boxes for the rows where the values from the XML will override the database values.
11. Right-click in the list of parts in the *Import Results* tab.
12. Choose *Import Selected*.

This imports the data from the XML into the database.

Synchronization Conflicts

A difference in the value of **any** attribute of a part or model is a conflict. Any change in relationship is also a conflict. For example, a different footprint model is now attached to a part, or the classification attached to a particular model has been revised.

There can be the following scenarios in terms of the relations while importing models or parts:

- There is a new relation in the XML.
- A relation is missing from the XML but is in the database.

- A relation has been modified for the existing part/model.

  In this case:

  - The same version of the related part/model is displayed as *modified*.
  - The modified XML has a different version than the existing version in the database of the related object.

Similarly, other differences, such as those between attributes and searchable properties, will be displayed.

Joint Library Partner (JLP)

Allegro EDM allows two or more companies to work as joint library development partners. This allows a joint library partner (JLP) to develop library elements and send the resulting elements back to the master library, where they will be verified and released. This allows for:

- The support of an outsourced library development partner
- A local library development server for multiple geographies

This solution is based on the fact that the JLP will get regular updates from the master library for any administration-data related changes. Librarians at the JLP site use the replicated component database of the master library site. They can work on adding, deleting, modifying of parts and/or models data using their instance of the Allegro Library Manager. The modified data is exported and sent to the master library server as data packets. The master librarians then import this data into the master library, verify, and release it.

This solution uses the Allegro EDM sync system called *jlp* to support this process.

Setting up Joint Library Development Environment

Before you start using the JLP flow, do the following:

- Ensure that all joint library development partners have access to the same component database as that of the master library site and it is synchronized with the master library database at regular intervals.
- If you have an existing *<PCBDW_LIB>*, copy the `jlp` folder from *<installation_hierarchy>*`\pcbdw_lib\exchange\sync` to *<partner_site_Allegro EDM>*`\pcbdw_lib\exchange\sync`.
- Librarians at the JLP site need to be created as part of the user list in the master library database.
- JLP users must only have the librarian role and must not have the library administration password.
- Any administration data changes made by JLP users cannot be exported, and thus cannot be synchronized with the master library database.
- Ensure that the Allegro EDM Server at both the master and the JLP site are at the same Allegro EDM version and also at the same ISR release.
- Allegro EDM Conf Root at both the sites needs to be synchronized using the OS-level synchronization commands. While replicating the Allegro EDM Conf Root, ensure that the administrator user name and password are not specified in `workbench.ini`.

- The Vault folder needs to be synchronized by users at the JLP site using the OS-level synchronization commands.
- If required, configure `sync.xml` to get updates from the JLP site for changes that can be done without revising the objects. For example, a change in the Lifecycle Status value.
- Copying exported tarballs from the JLP site to the master library site needs to be done using OS-level synchronization commands.
- Ensure that all the required related objects are also exported by the JLP site to the master library site.

Recommendations for Using JLP

- You should export the complete working set and not parts of the working-set contents to the master library site.
- The master librarian should ensure that the modified/added objects are in the Released state.
- It is recommended that JLP sites not use Data Exchange to modify the data without revising the objects.

Conditions for Using JLP

- The JLP flow cannot work for OEM/ODM setup.
- Data sent by the JLP site will not get imported if there are any administration-data related modifications at master library server. The JLP site may have to again perform the tasks after replicating the updated master library database.
- Errors related to data import in the JLP site are the same as the Data Exchange errors.
- The modified user name will not be retained when data is imported from JLP. The modified user name will be the same as the user name who imports the tarball. Effectively, the master library database will never have any object modified by the JLP user. Further, when the JLP site is synchronized with the master library database, JLP will also lose the actual author or name.
- For front-end cells, the user name in the library metadata will be that of the JLP user. For any cell model, the user name will be that of the master library database. This inconsistency will be seen in LRM too.
- The JLP flow cannot work in the MLR environment.

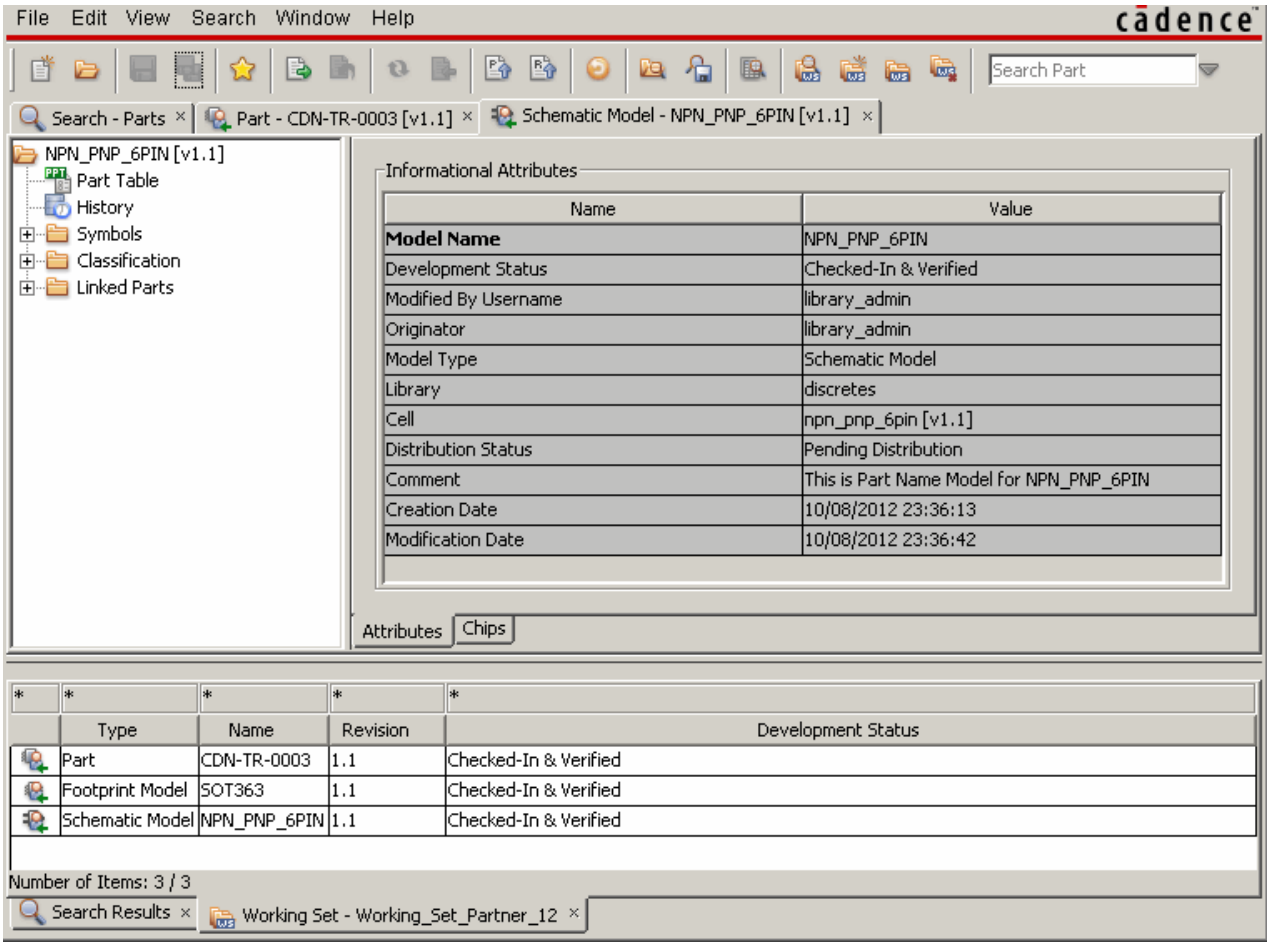
- While merging imported data, the master site needs to ensure that data is not overwritten for the changes done by the master site itself for the same data. For example, if the master server has updated the RoHS status from the ERP system after data is imported from the JLP, then, while merging data from the JLP, the master librarian needs to ensure that the RoHS is not overwritten again. Note that when RoHS is updated using Data Exchange, it can be updated without revising the part. As a result, there will be no way to find out whether the value was removed by the JLP site or added after the last database dump was given to JLP.
- If a JLP librarian works with an object multiple times (for example, from v1.0 to v8.0), then while exporting, only v8.0 is exported and the previous versions of the data is lost when the JLP site is next synchronized.

Exporting Library Data

As a joint library partner, to export your library development work, you need to do the following:

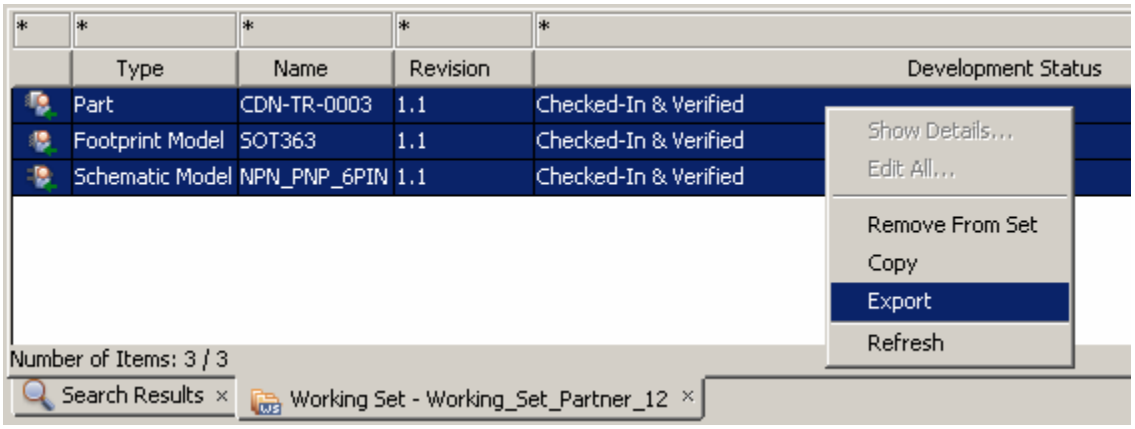
1. Modify the site-level `workbench.ini` file to specify the default sync system for the JLP site. To do so:
   a. Navigate to *<adw_conf_root>*`\`*<company>*`\`*<site>*`\workbench.ini`.
   b. Open this file and in the `[general_options]` section, add:
   ```
   default_sync_system = jlp
   ```
   Note: This is an optional step. If you do not specify the default sync system, you be prompted to select a sync system while exporting your data.
2. Launch Database Editor.
3. Create a new working set for the library development or modification task.
4. Complete the library development or modification task using this working set.

5. Choose *View – Current Working Set*. The working set used by the partner for library development opens in a separate tab.



6. Select all the required objects to be exported to the master library site.

7. Choose *Export*.



   A message indicating successful export appears. A zip file with the name of the working set is created at: `<partner_site_Allegro EDM>\pcbdw_lib\exchange\sync\jlp\export`

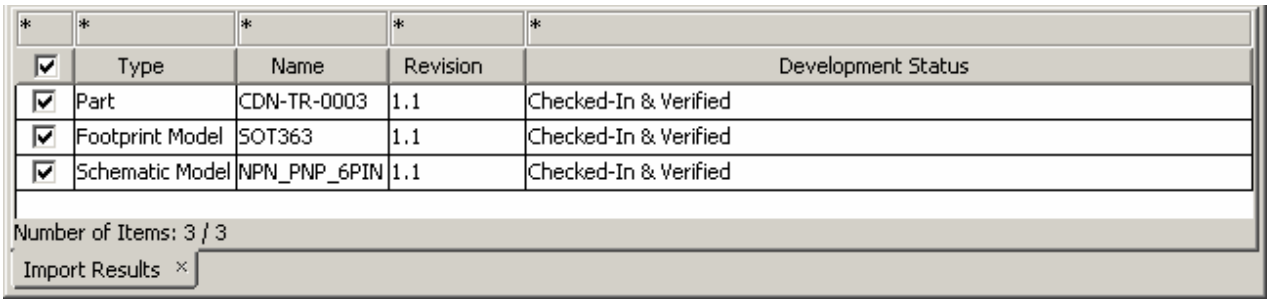
8. Send this zip file to the master library site.

Importing Library Data

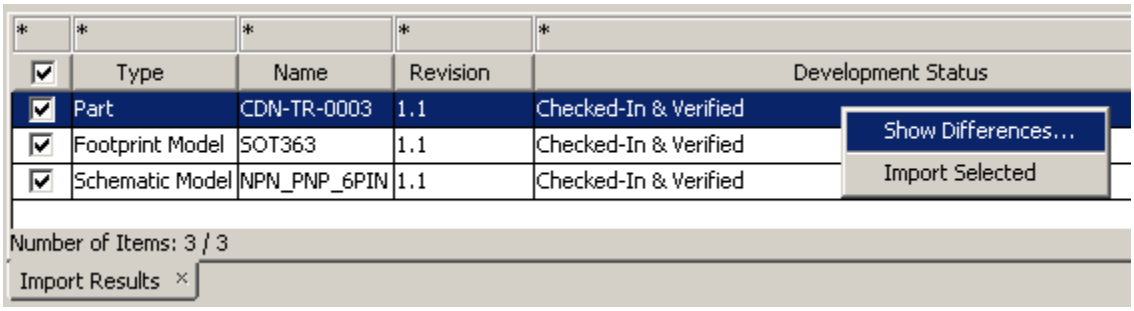
As a librarian at the master library site, you need do the following:

1. Launch Database Editor.
2. Choose *File – Import*.
3. Choose the zip file to import.
4. Click *Open*.

   All the objects in the partner's working set are displayed in the *Import Results* tab.



5. Right-click any object row and choose *Show Differences.*



   This option allows you to visually compare the parts and models between the zip file (working set sent by the partner site) being imported and the component database at the master library site.

6. Right-click and choose *Import Selected*.

   This imports the data into the component database at the parent site and a message appears to indicate the successful import.

   Note: Database Editor, while exporting the working set at the partner site, specifies the sync system being used in the zip file. Thus, at the master library site, the same sync system is used to import that zip file.

7. Click *OK*.

   The current working set is set to the imported working set.

8. Choose *View – Current Working Set*.

   The working set exported by the partner site opens in a separate tab with the same name as the zip file. It contains all the objects exported by the partner site in *Checked-In & Verified* state. As a librarian from the master library site, you can validate and release them.

5

Auto Generation of SI DML Models for Discrete Parts

Overview

The core Allegro tools allow you to associate the SIGNAL_MODEL property on the discrete parts (2-pin components, resistors and capacitors pack discrete parts) with the auto generated SI DML model names.

Allegro EDM also provides the capability of assigning the SIGNAL_MODEL property value on the discrete parts with the auto generated SI DML model name. This helps the designer who is using the discrete parts with the task of auto-generation of SI DML models.

There are three ways that can enable you to generate SI DML model name:

- Using Library Import

  If you have large number of discrete parts in the component database, do the following:

  a. Associate the SIGNAL_MODEL property to the required schematic model classification using Database Editor (as an administrator).

  b. Configure Library Import setup to import SI DML models and choose the option to auto generate SI DML model name.

  c. Generate XML and upload the data using library import process.

  d. Run the library distribution command.

  For detailed information, see *Allegro EDM Library Import User Guide*.

- Using Library Flow

  If you have only a selected number of discrete parts, then perform the following tasks using Database Editor:

  a. Associate the SIGNAL_MODEL property to the required schematic model classification.

b. Associate the SIGNAL_MODEL property to the related part classification.

c. Associate the discrete parts with the revised part classification.

d. Auto generate the SI DML model for the selected discrete parts.

e. Run the library distribution command.

- Importing Library Data for the First Time

  If you are importing library data first time, do the following:

  a. Add the SIGNAL_MODEL property to the PTF file of the discrete parts using PTF Editor.

  b. Configure Library Import setup to import SI DML models and choose the option to auto generate SI DML model.

  c. Generate XML and upload the data using the library import process.

  d. Run the library distribution command.

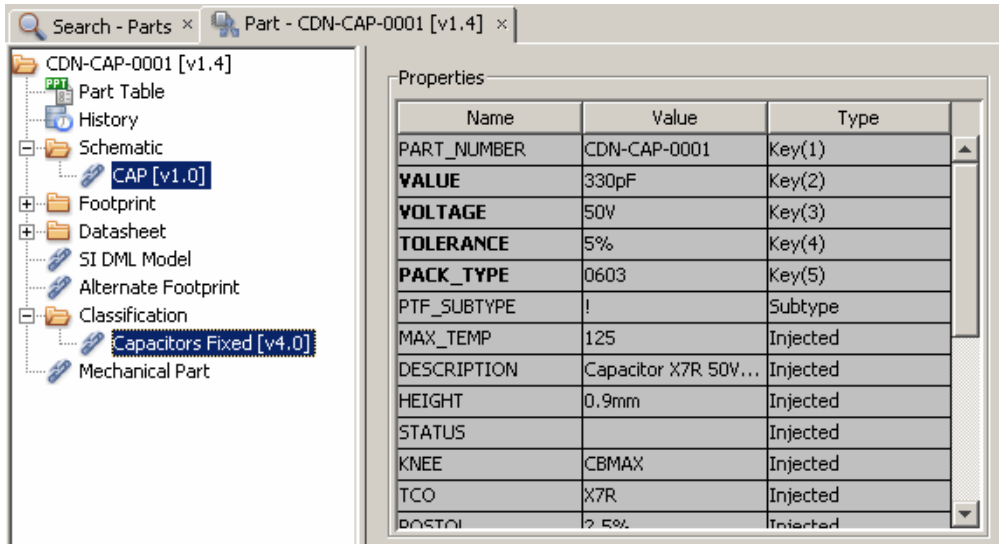
Auto Generation of SI DML Models Using Database Editor

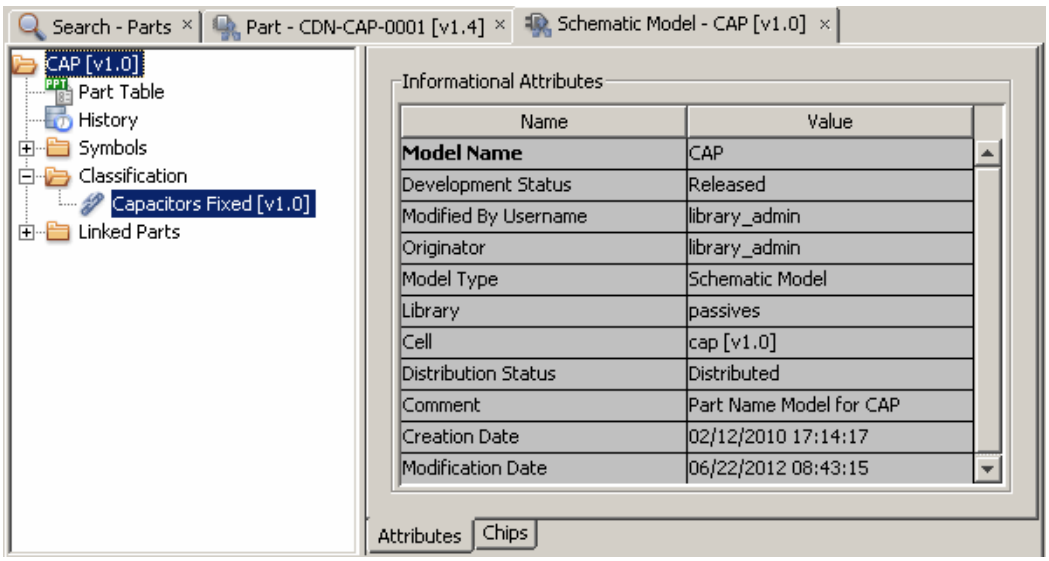
You can use Database Editor when you have a selected few parts for which you need to auto generate the SI DML model name. Consider the following example to understand the procedure.

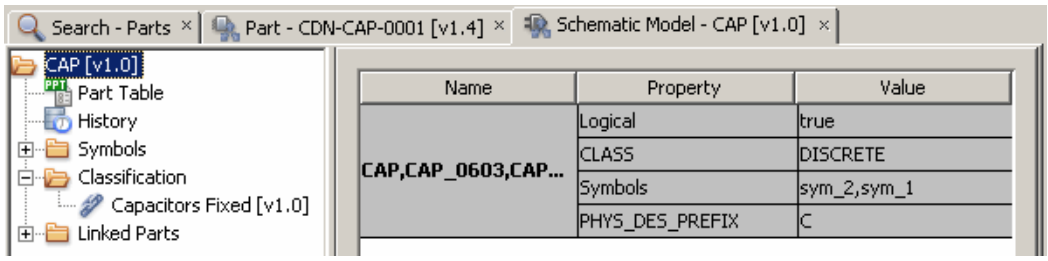
The discrete part, `CDN-CAP-0001[v1.4]` has the following associations:

- Part Classification: `Capacitors Fixed[v4.0]`
- Schematic Model: `CAP[v1.0]`

- Schematic Model Classification: `Capacitors Fixed[v1.0]`







Perform the following steps to auto generate SI DML Model name on `CDN-CAP-0001[v1.4]`:

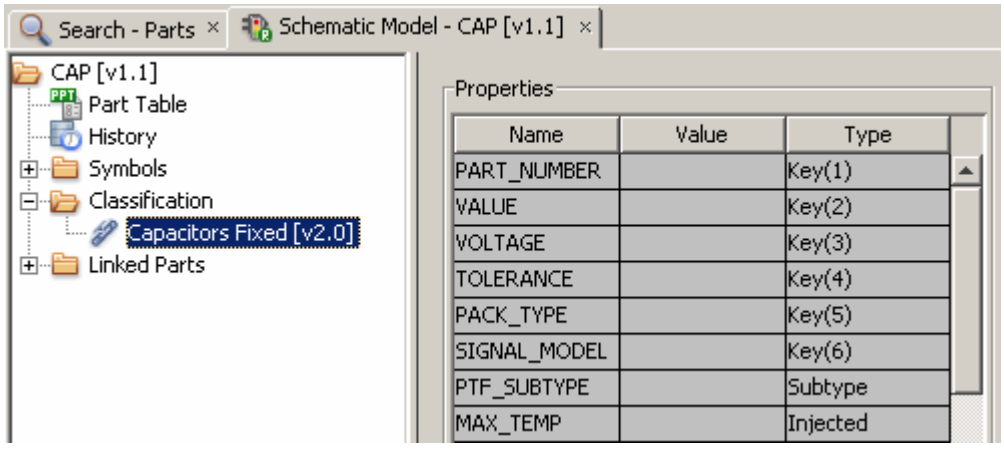
1. Launch Database Editor.
2. Choose *File – Manage Model Classification – Schematic Model Classification*.
3. Choose `Capacitors Fixed[v1.0]` and click the Add button in the right pane to add to new property.
4. Enter the property name `SIGNAL_MODEL` and click *Create*.
5. Add the *Link To* value for *SIGNAL_MODEL* property as `SI DML Model.Model Name`.
6. Save the classification.

   The Update message appears asking you to select from the three options, Preserve, Check-out, or Release.
7. Click the required option. For this example, click *Preserve*.

   The Check-in Log for Model: `CAP` dialog box appears.
8. Enter the log details and click *Apply to All*.

   You will see the creation of revised:

   - Schematic Model Classification: `Capacitors Fixed[v2.0]`
   - Schematic Model: `CAP[v1.1]`



Next, you need to add the SIGNAL_MODEL property on the part classification `Capacitors Fixed[v4.0]`. To do so:

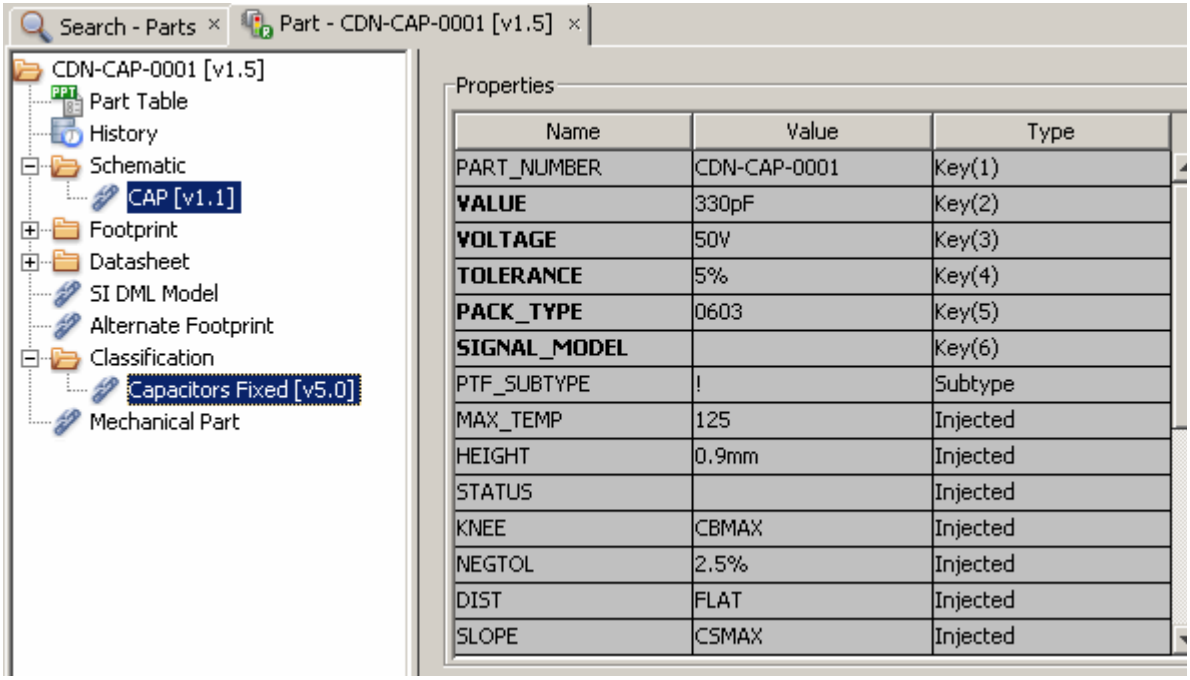
1. Choose *File – Manage Part Classification*.
2. Choose `Capacitors Fixed[v4.0]` in the explorer pane.

3. Click the Add button in the right pane to add a new property.
4. Enter the property name `SIGNAL_MODEL` and click *Create*.
5. Save the part classification.

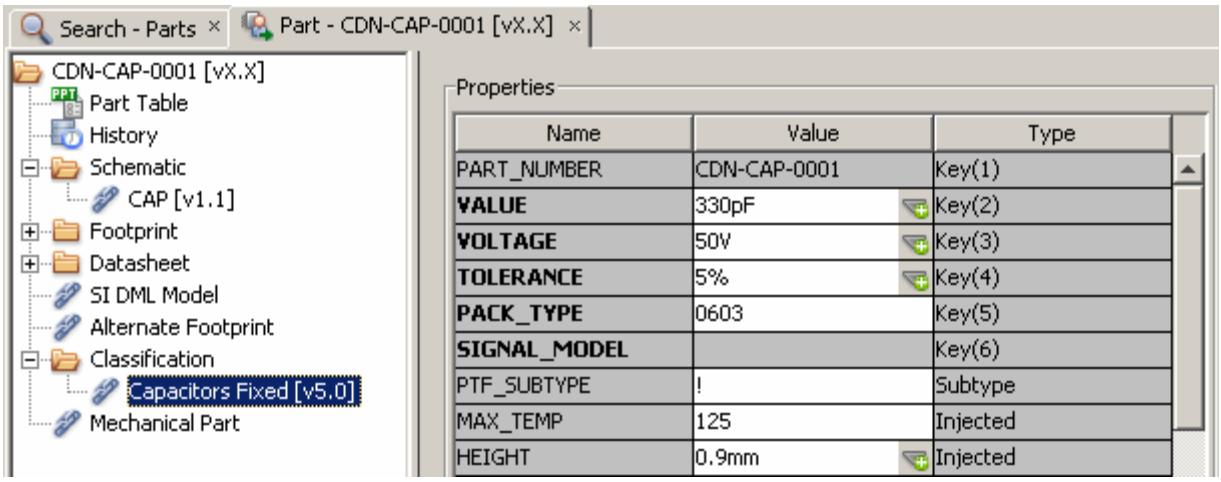
   The Update message appears asking you to select from the three options, Preserve, Check-out, or Release.
6. Click the required option. For this example, click *Preserve*.
7. The Check-in Log for Part: `<part_number>` dialog appears.
8. Enter the log details and click *Apply to All*.

   This creates a revised:

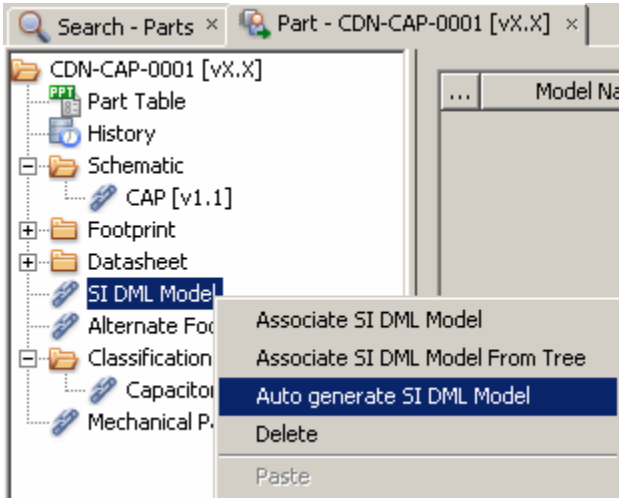
   - Part Classification: `Capacitors Fixed[v5.0]`
   - Part: `CDN-CAP-0001[v1.5]`



9. Check out `CDN-CAP-0001[v1.5]` that is now associated with schematic model `CAP[v1.1]` and part classification `Capacitors Fixed[v5.0]`.



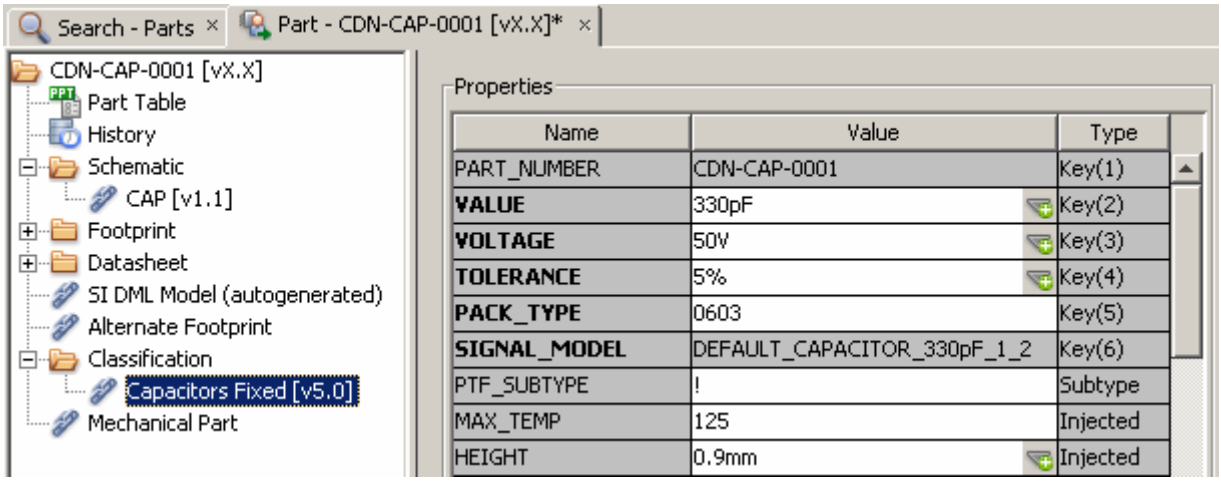
10. Right-click the relation *SI DML Model* and choose *Auto generate SI DML Model*.



A message appears to indicate that SI DML model name has been auto generated.

11. Click *OK*.

The value of the *SIGNAL_MODEL* property has been automatically populated in the right pane.



12. Check in `CDN-CAP-0001[v1.5]`.

13. Run the library distribution command.

A

Attributes and Models

About Attributes

All the library data attributes display in the following two panes:

1. **Informational Attributes**: These are attributes that are derived from the schema or can be the key attributes. These attributes are non-editable.
2. **Attributes**: These attributes are editable, and can be modified by the administrator.

Furthermore, depending on their nature and the way they appear in Database Administrator and Database Editor interfaces, these attributes have been classified into the following categories.

Interface Appearance	Signifies	Notation (in Documentation)
Bold	key attribute or mandatory attribute	K
Roman Normal	■ system-generated and non-editable attribute or ■ optional attribute	S

Note: This notation has been used in the documentation to explain the nature of the attributes.

Default Attributes

The following table lists the default informational attributes that are available in the out-of-the-box implementation of the Allegro EDM Component Database.

Data Object	Attribute Name	Type
Part		
	Part Number	K
	Development Status	S
	Distribution Status	S
	Originator	S
	Creation Date	S
	Modification Date	S
	Comment	S
	Modified by Username	S
Block Part		
	Part Number	K
	Development Status	S
	Modified by Username	S
	Originator	S
	Distribution Status	S
	Comment	S
	Creation Date	S
	Modification Date	S
Mechanical Part		
	Part Number	K
	Development Status	S
	Distribution Status	S
	Originator	S

Data Object	Attribute Name	Type
	Creation Date	S
	Modification Date	S
	Comment	S
	Modified by Username	S
Mechanical Kit		
	Kit Number	K
	Development Status	S
	Distribution Status	S
	Originator	S
	Creation Date	S
	Modification Date	S
	Comment	S
	Modified by Username	S
Schematic **Model**		
	Model Name	K
	Development Status	S
	Modified by Username	S
	Originator	S
	Model Type	S
	Library	S
	Cell	S
	Distribution Status	S
	Comment	S
	Creation Date	S
	Modification Date	S
Block Model		
	Model Name	K

Data Object	Attribute Name	Type
	Development Status	S
	Modified by Username	S
	Originator	S
	Model Type	S
	Library	S
	Distribution Status	S
	Comment	S
	Creation Date	S
	Modification Date	S
	Design Author	S
	Update Required	S
Standard Model	Model Name	K
	Development Status	S
	Modified by Username	S
	Originator	S
	Model Type	S
	Library	S
	Distribution Status	S
	Comment	S
	Creation Date	S
	Modification Date	S
Mechanical Model, Footprint Model, Padstack Model, Flash Model, Shape Model, Board Model, Format Model, Datasheet Model		
	Model Name	K

Data Object	Attribute Name	Type
	Development Status	S
	Modified by Username	S
	Originator	S
	Model Type	S
	Library	S
	Distribution Status	S
	Comment	S
	Creation Date	S
	Modification Date	S
Module Model		
	Model Name	K
	Development Status	S
	Revision	S
	Originator	S
	Model Type	S
	Library	S
	Distribution Status	S
	Comment	S
	Creation Date	S
	Modification Date	S
	Design Author	S
	Update Required	S
SI DML Model		
	Model Name	K
	Development Status	S
	Modified by Username	S
	Originator	S

Data Object	Attribute Name	Type
	Model Type	S
	Library	S
	Distribution Status	S
	Comment	S
	Creation Date	S
	Modification Date	S
	File Name	S

Default Relations

Following are the default relations that are available in the out-of-the-box component database entities.

Component Data	Relation Name
Part	Part Table
	History
	Schematic
	Footprint
	Datasheet
	SI DML Model
	Alternate Footprint
	Classification
	Mechanical Part
Block Part	Part Table
	History
	Block
	Module
	Datasheet
	Classification
Mechanical Part	Part Table
	History
	Mechanical
	Linked Kits
	Classification

Component Data	Relation Name
Mechanical Kit	Part Table
	History
	Mechanical
	Mechanical Parts
	Classification
Standard Model	History
	Symbols
	Classification
	Linked Block Models
Schematic Model	Part Table
	History
	Symbols
	Linked Schematic Models
	Classification
	Linked Parts
Mechanical Model	Part Table
	History
	Classification
	Linked Parts

Component Data	Relation Name
Block Model	Part Table
	History
	Symbols
	Parts
	Schematics
	Standard Models
	Linked Block Parts
	Classification
Footprint Model	History
	Padstacks
	Classification
	Linked Parts
Padstack Model	History
	Shapes
	Flashes
	Linked Footprints
	Classification
Flash Model	History
	Classification
	Linked Padstacks
Shape Model	History
	Classification
	Linked Padstacks
Board Model	History
	Padstacks
	Classification

Component Data	Relation Name
Format Model	History
	Classification
Module Model	History
	Footprints
	Board
	Format
	Vias
	Classification
	Linked Block Parts
Datasheet Model	History
	Classification
	Linked Parts
SI DML Model	History
	Classification

B

Setting Up Complex Searches

Database Editor allows you to specify complex search expressions using a variety of relational, logical, and special operators. The search expression supports the following operators.

Operator	Usage
< (Less than)	The value to search must be less than the value specified.
> (Greater than)	The value to search must be greater than the value specified.
!= (Not Equal to)	The value to search must not match the value specified.
== (Equal to)	The value to search must match the exact value specified. This operator is used for case sensitive exact match.
<= (Less than or equal to)	The value to search must be less than or equal to the value specified.
>= (Greater than or equal to)	The value to search must be greater than or equal to the value specified.
~~ (Case insensitive string match)	The pattern of the first value must match the pattern of the second value. The value can be included anywhere in the string. With this operator, character case is ignored so that `library` is considered a match for `LIB*`.

Operator	Usage
!~~ (Case insensitive string not match)	The pattern of the first value must not match the pattern of the second value. The value can be included anywhere in the string. With this operator, character case is ignored. For example, a first value of “Part Name” and a second value of “pA* nA*” would result in a FALSE comparison since the two are considered a match regardless of the difference in uppercase and lowercase characters.
~= (Case sensitive string match)	The pattern of the first value must match the pattern of the second value. The value can be included anywhere in the string. This includes testing for uppercase and lowercase characters. This operator is used for case sensitive string match. If you use * or ? in the string, they are considered as wildcard operators.
!~= (Case sensitive not match)	The pattern of the first value must not match the pattern of the second value. The value can be included anywhere in the string. For example, if the first value is `Part*` a second value of `part` would produce a true result because the lowercase `p` is not an exact match to the first value’s uppercase `P`.
&& Logical AND	The first value and the second value must be present.
|| Logical OR	Specifies that either of the values be present.
* (Any String of Characters)	Specifies any number of characters match.
? (Any Single Character)	Specifies exactly one character match.

Important

By default, if you do not provide any operator, the ~~ (string match) operator is used. If you specify _ (underscore) as a part of the search string, it is treated as a ? (question mark) character, a single character.

C

Database Editor User Interface

This chapter details the following user interface components of Database Editor:

- Menus
- Database Editor Window
  - Panes
  - Tabs
- Dialog Box Help

Menus

File Menu

Command	Lets you
New – Part	Create a part in the component database.
New – Block Part	Create a block part in the component database.
New – Mechanical – Mechanical Part	Create a mechanical part in the component database.
New – Mechanical – Mechanical Kit	Create a mechanical kit in the component database.

Command	Lets you
New – Model – <Model_Name>	Create a model in the component database. You can create the following supported models, which appear as submenus: ■ Standard Model ■ Schematic Model ■ Mechanical Model ■ Block Model ■ Capture Model ■ Footprint Model ■ Padstack Model ■ Flash Model ■ Shape Model ■ Board Model ■ Format Model ■ Module Model ■ Datasheet Model ■ SI DML Model
Open – Part	View an existing part in the component database.
Open – Block Part	View an existing block part in the component database.
Open – Mechanical – Mechanical Part	View an existing mechanical part in the component database.
Open – Mechanical – Mechanical Kit	View an existing mechanical kit in the component database.
Open – Model – <Model_Name>	View existing models in the component database. Supported models appear as submenus.

Command	Lets you
Open – Admin Data	Access administrator data in the form of the following submenus. ■ Model Type ■ Library ■ Preferred Part List
Manage Part Classification	Manage part classification on the basis of a set of common attributes.
Manage Block Part Classification	Manage block part classification on the basis of a set of common attributes.
Manage Mechanical Part Classification	Manage mechanical part classification for your design site on the basis of a set of common attributes.
Manage Model Classification	Manage model classification for your design site on the basis of a set of common attributes.
Save	Save the changes you make into the component database.
Save All	Save all the changes into the component database.  *Important* *Save* and *Save All* commands are disabled in the Search tabs.
Working Set – Create	Create a Working Set.
Working Set – Change	Change an active Working Set. By default, you can see the working sets you created. Select *View All* to see working sets created by all the users on the database you are connected to.
Working Set – Close	Delete a Working Set.
Import	View the library data to be imported into the component database.
Exit	Close the Database Editor.

Edit Menu

Command	Lets you
Set Active	Open or close the Search Results tab.
Rename	Rename a part or model
Check-out	Check out a part or model.
Check-out Hierarchy	Check out a part or model for making minor changes.
Undo Checkout	Cancel a check-out for a part or model.
Undo Checkout Hierarchy	Undo the checkout of a model, along with the submodels that are part of the checked-out hierarchy.
Synchronize Models	Assign the submodels associated with a model before you check in a model. For example, for a schematic model, you can assign symbols and part name submodels.
Local Flow Verify	Run front-to-back verification rules for a part, block part, or model.
Check-In	Check in a component into the database.
Check-In Hierarchy	Check in a model along with the submodels that were checked out simultaneously.
Pre Release	Prerelease a component to the database. Prereleased components are not flow-verified, but can be distributed.
Release	Release a component into the database. Released components can be distributed.
Add Replacement Part	Add a replacement part for a deleted part.
Remove Replacement Part	Remove any replacement part already associated with a part.
Delete	Delete a part or model.

Command	Lets you
Library Flow	Run verification rules on a batch of parts. The submenu commands are: ■ Check in Parts ■ Pre Release ■ Release ■ Run Library Distribution
Options	Configure the behavior of database management commands.

View Menu

Command	Lets you
Search Results	Open or close the Search Results tab.
Current Working Set	Open the current Working Set tab.
Show Logs	View the Log Report tab.
Toolbar	Control the visibility of the toolbar.
Environment	View environment parameters, such as database server details, users, and the location of library project and reference libraries.
Refresh	Refresh the Search Results tab.

Search Menu

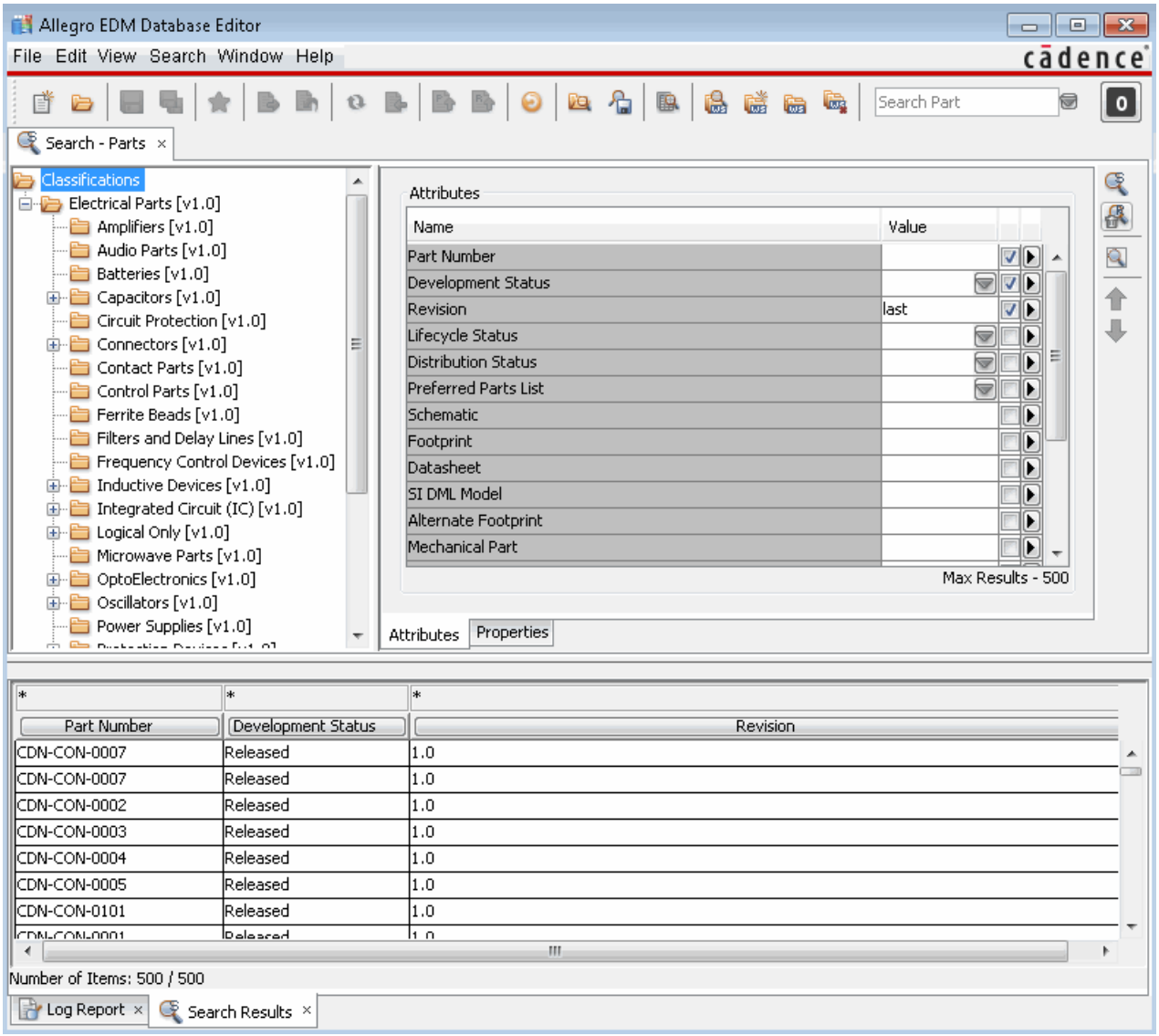
Command	Lets you
Search – Part	Search existing part in the component database.
Search – Block Part	Search existing block part in the component database.
Search – Mechanical – Mechanical Part	Search existing mechanical part in the component database.
Search – Mechanical – Mechanical Kit	Search existing mechanical kit in the component database.

Command	Lets you
Search – Model – <Model_Name>	Search existing models in the component database. Supported models appear as submenus.
Search – Admin Data	Lets you search for administrator data in the form of the following submenus: ■ Model Type ■ Library ■ Preferred Part List
Search Option	Search Limit: Specify the number of search results to be returned for a search. The maximum number of results is 32767. Select a search mode for performing search. The options are Match All and Match Any
Load Search Criteria	Load existing search criteria.
Save Search Criteria	Save new search criteria.

Window Menu

Command	Lets you
Single Detail Window	Control tab-based views in the Database Editor window.
Recently Viewed	View the history of the tabs you have recently opened in the main window.

Database Editor Window



The interface of Database Editor is similar to Database Administrator and Library Import.

Panes

By default, there are two panes in the window:

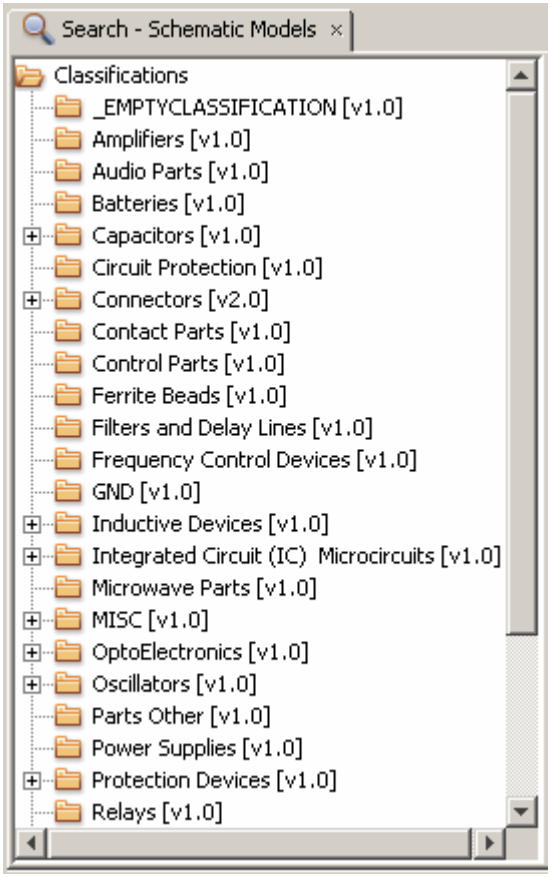
- Explorer pane
- Right pane

These panes are context-specific, and their content and visibility depend on the component database entity selected (part or model) and the type of operation being performed. For example, when you search for Models, the *Search - <Model_Name>* classifications hierarchy appears in the Explorer pane. When you choose a node in the classification hierarchy, its attributes and properties appear in the Right pane.

Similarly, when you view any part detail or model detail tab, the explorer pane contains the existing relations for the part or model selected.

Explorer Pane

This pane provides a hierarchical view of part and model classifications. This hierarchical view might also represent relations associated with data entities (for tools, libraries, PPLs, and users).



Context-sensitive Pop-up Menu Options

Depending on the level of node you select in the explorer pane, the pop-up menu can help you:

- Create and remove relations. It contains commands such as *Associate <Relation_Name>* and *Associate <Relation_Name> from Tree*, *Delete*, and *Paste*.
- View the details of a classification using the *Show Details* command.

Note: Parent nodes contain the *Refresh* pop-up menu command.

Right Pane

This context-sensitive pane displays different views depending on the type of operation you perform with Database Editor. The active tab determines the view of the pane, which includes:

- Editable and non-editable views of the part and model data selected.
- When you perform a search, this pane allows you to specify search criteria for attributes and properties.

Note: You can use the Tab key to navigate to various editable fields in the right pane.

Tabs

The Database Editor window allows you to open as many tabs as you want to. A tab can be of the following types:

- Search Tab
- Details Tab
- Search Results Tab
- Error Tab
- Log Report Tab
- Working Set Tab

Note: The Search Results, Log Report, and Error tabs appear at the bottom of the window. The Error tab appears only if there are any errors during the various operations you work on in Database Editor.

Important

By default, the maximum number of results per search is 500. You can specify the limit for the number of search results using the *Search Limit* button. It is recommended that you keep this number low for a faster search.

Search Tab

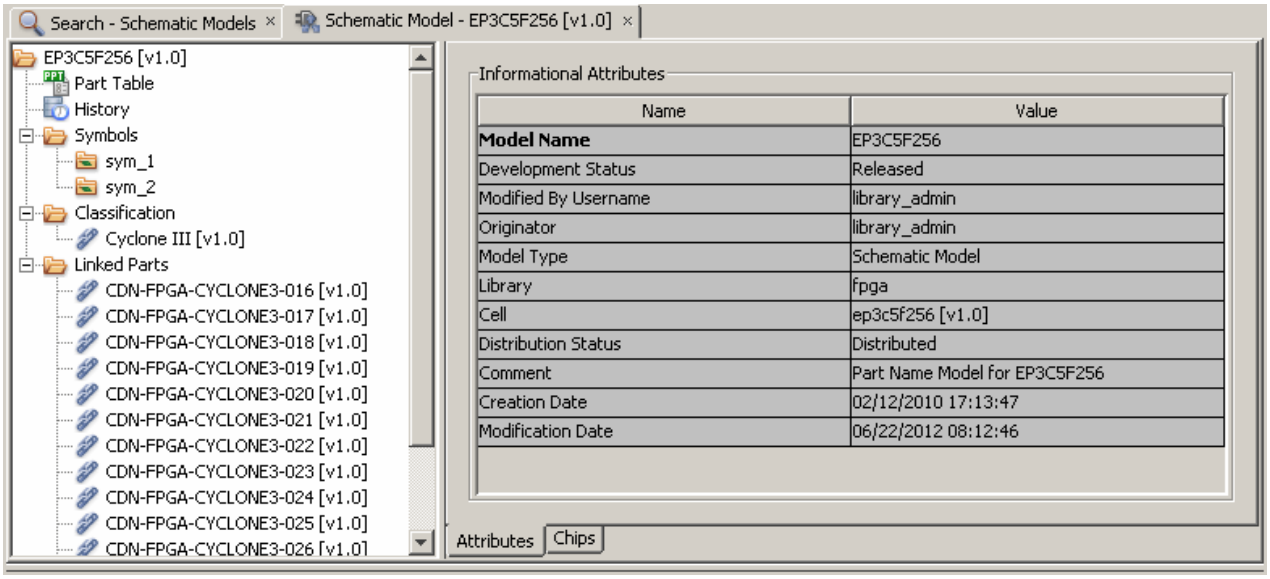
This tab appears below the main menu when you search for library data or administrator data, such as model type, library, and PPL. For library data search, two tabs appear at the bottom of the right pane:

- Attributes: When selected, the default search attributes and relations for the selected part or model appear.
- Properties: When selected, the default searchable properties for the selected part or model appear.

Details Tab

Part or model details and related component data can be viewed in the Detail tab. Whenever you open a part or a model, its details appear in form of a *Part - <Part_Detail>* tab or *<Model_Type> - <Model_Detail>* tab, respectively.

.



Note: There are also tabs to view *Admin Data* and *Library Flow*.

Search Results Tab

All the search results appear at the bottom of the Database Editor window. This pane also shows the number of results that match your search criteria at the bottom left of the Search Results tab.

Important

The Search Results tab always displays the last search performed, irrespective of which tab is active.

Error Tab

This tab appears at the bottom, beside the Search Results tab, and shows all the errors encountered, if any, during the various operations you work on in Database Editor.

Log Report Tab

This tab appears before the *Search Results* tab, at the bottom, and shows all the log details for each operation performed in Database Editor.

Working Set Tab

Parts and models, which are part of your Working Set, appear in this tab.

Dialog Box Help

New Dialog Box

This dialog box appears when you create library data, such as parts and models. You will need to specify a name for the part or model you want to create.

Note: For a list of available parts and models, see File Menu.

Open Dialog Box

This dialog box appears when you view existing library data. To view a part or model, provide its name.

Note: For a list of available Parts and Models, see File Menu.

Associate <Relation_Name> Dialog Box

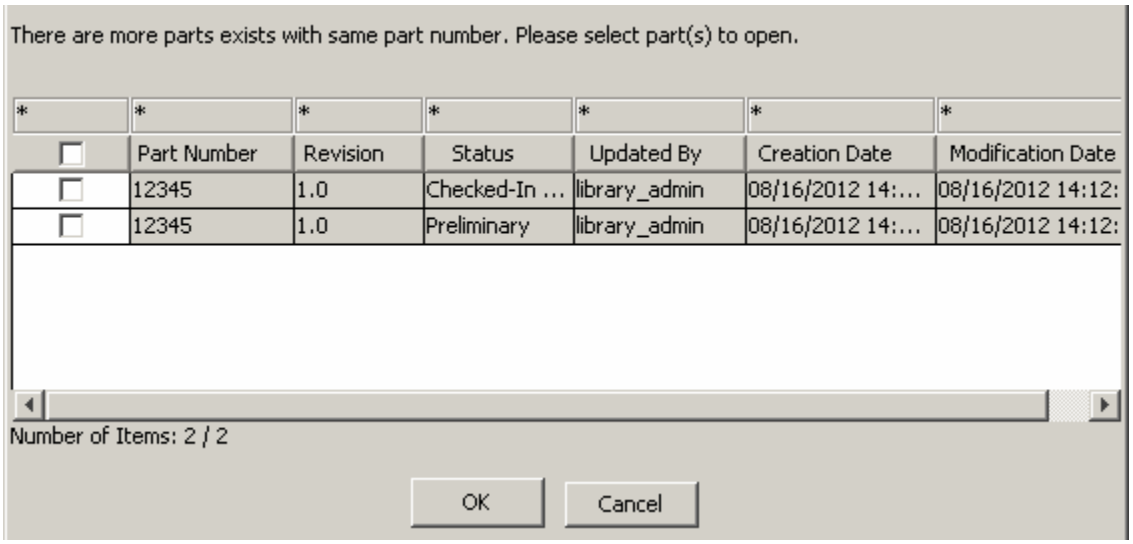
This dialog box allows you to choose a relation instance that you want to associate with the selected relation. For details, see Adding and Editing Relation Instances.

Delete <Relation_Name> Dialog Box

This dialog box allows you to choose a relation instance to be deleted for the selected relation. For details, see Deleting Relation Instances.

Select Part Dialog Box

This dialog box appears when you try to open a duplicate part (using the *File – Open – Part* command), which means that two or more parts exist in the database with the same name. This dialog box contains a table with all the parts with the same name. Select a part, and click *OK* to view its details.

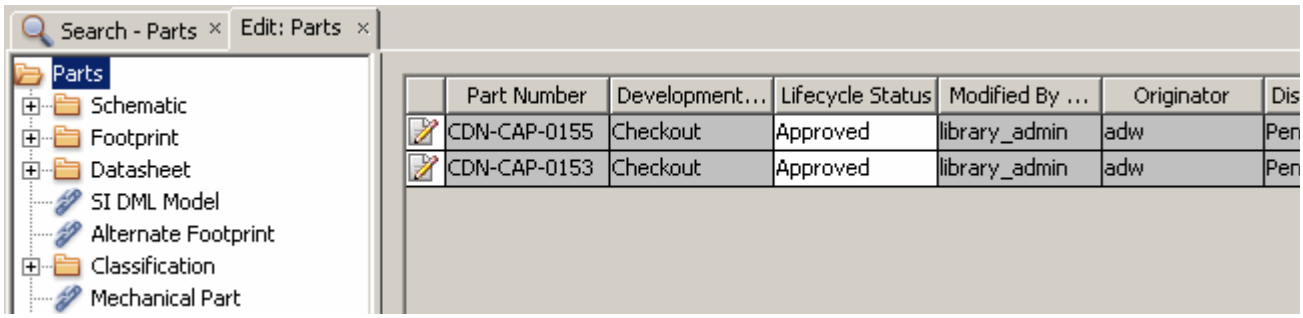


Tip

Use the *Select All* or *Clear All* pop-up menu commands to select or deselect all the parts.

Edit All Parts or Models

This tab appears when you select library data (parts and models) in the *Search Results* tab, and then choose the *Edit All* pop-up menu option. The tab allows you to edit multiple parts or models.



Index

Symbols

A

B

C

D

E

I

K

L

M

O

P

R

S

V

W