



Library and Part Management in OrCAD X Capture

**Product Version 23.1
September 2023**

© 2023 Cadence Design Systems, Inc.
Printed in the United States of America.

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

Open SystemC, Open SystemC Initiative, OSCI, SystemC, and SystemC Initiative are trademarks or registered trademarks of Open SystemC Initiative, Inc. in the United States and other countries and are used with permission.

Trademarks: Trademarks and service marks of Cadence Design Systems, Inc. (Cadence) contained in this document are attributed to Cadence with the appropriate symbol. For queries regarding Cadence's trademarks, contact the corporate legal department at the address shown above or call 800.862.4522.

All other trademarks are the property of their respective holders.

Restricted Permission: This publication is protected by copyright law and international treaties and contains trade secrets and proprietary information owned by Cadence. Unauthorized reproduction or distribution of this publication, or any portion of it, may result in civil and criminal penalties. Except as specified in this permission statement, this publication may not be copied, reproduced, modified, published, uploaded, posted, transmitted, or distributed in any way, without prior written permission from Cadence. Unless otherwise agreed to by Cadence in writing, this statement grants Cadence customers permission to print one (1) hard copy of this publication subject to the following conditions:

1. The publication may be used only in accordance with a written agreement between Cadence and its customer.
2. The publication may not be modified in any way.
3. Any authorized copy of the publication or portion thereof must include all original copyright, trademark, and other proprietary notices and this permission statement.
4. The information contained in this document cannot be used in the development of like products or software, whether for internal or external use, and shall not be used for the benefit of any other party, whether or not for consideration.

Disclaimer: Information in this publication is subject to change without notice and does not represent a commitment on the part of Cadence. Except as may be explicitly set forth in such agreement, Cadence does not make, and expressly disclaims, any representations or warranties as to the completeness, accuracy or usefulness of the information contained in this document. Cadence does not warrant that use of such information will not infringe any third party rights, nor does Cadence assume any liability for damages or costs of any kind that may result from use of such information. Cadence is committed to using respectful language in our code and communications. We are also active in the removal and replacement of inappropriate language from existing content. This product documentation may however contain material that is no longer considered appropriate but still reflects long-standing industry terminology. Such content will be addressed at a time when the related software can be updated without end-user impact.

Restricted Rights: Use, duplication, or disclosure by the Government is subject to restrictions as set forth in FAR52.227-14 and DFAR252.227-7013 et seq. or its successor.

Contents

1	6
Libraries in OrCAD X Capture	6
2	8
Managing Part Libraries	8
Creating a Library	8
Editing a Library	10
Copying a Part from the Design Cache to a Library	10
Copying a Schematic Page to a Library	11
Copying a Schematic Folder to or from a Library	12
Moving Parts or Symbols between Libraries	15
Renaming a Library	16
Closing and Saving a Library	16
3	18
Using Part Libraries	18
Adding a Library to a Project	19
Configuring Preinstalled Libraries	20
Exporting and Importing a Capture Library	30
Opening a Library	32
Using the Library Correction Utility	33
Library Correction Utility	33
4	40
Managing Libraries on OrCAD X Cloud	40
Placing Components on Schematic	40
Searching for Components	45
Related Topics	47
Managing a Local Component Library	48
Creating Local Components From External Sources in Workspace	49
Assigning Manufacturer Part Number	51
Linking an MPN to a Component in Database	52
Editing Components	56
Copying Components	56

Deleting a Component	57
Managing Database Components	57
Linking Database Components	57
Viewing Database Components	59
Managing Shared Workspaces	62
5	63
Managing Parts	63
Heterogeneous and Homogeneous Parts	63
Creating a Heterogeneous Part	63
Split Parts	65
Placing Parts on a Schematic Page	65
Searching for a Part	65
Placing a Part	67
Creating Hierarchical Blocks	70
Attaching a schematic folder to a hierarchical block	70
Creating a hierarchical block from a Verilog model	72
Creating a hierarchical block from a VHDL model	73
Creating Parts	73
Creating a Part Body	74
Creating a Part Convert	85
Creating a Part Alias	86
Creating a Part from a Spreadsheet	87
Editing and Renaming a Part	90
Part Properties	92
Assigning Properties to a Part	93
Renaming and Deleting Part Properties	96
Update Properties using an Update File	97
Replacing and Updating Cache	98
Part Packages	101
Creating a Package	102
Editing, Deleting, and Viewing a Package	105
Synchronizing Parts	106
Part Instances and Occurrences	108
Removing Part Reference Assignments	109
Generating Library Parts	110
Creating a Split Part	112

Deleting a Part	118
Creating Components	118
Annotating Properties on Schematic	130
Sharing Components	131
Component Locking	133
Editing Shared Components	134
Managing Categories	136
Adding a New Category	136
Modifying a Category	140
Managing Mechanical Assemblies	141
Creating a Mechanical Category	142
Creating a New Mechanical Part	143
Creating a New Mechanical Assembly	143

Libraries in OrCAD X Capture

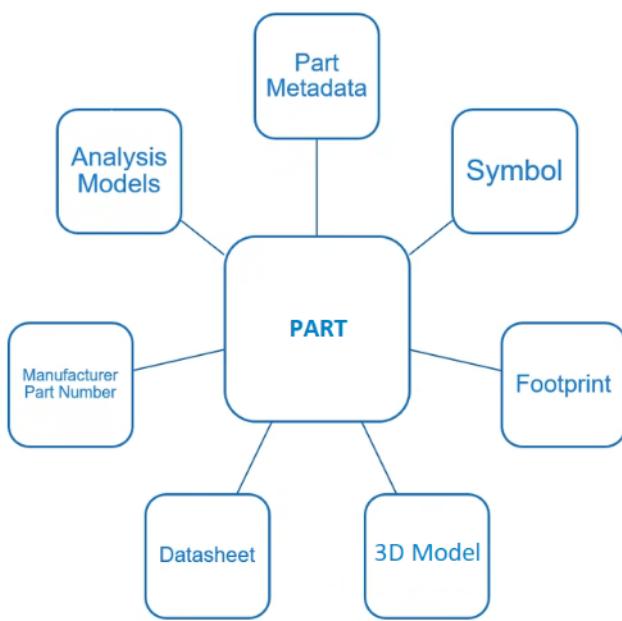
A library is a file that stores parts, symbols, title blocks, schematic folders or schematic pages. OrCAD® X Capture provides more than 80 libraries; in addition, you can create custom libraries. You can, for example, create a library to hold all your programmable logic devices, or hold schematic folders that you use often. There is no need to create a library for a particular project, because the design cache holds all the parts and symbols used in the project.

 If you edit a library provided by OrCAD X Capture, you need to provide a custom name to avoid copying over your changes when you receive updated libraries.

In OrCAD X Capture, you can create custom libraries and:

- Import data from existing library data into these libraries
- Create parts and associated existing data, such as symbols, footprints, and models

A library may consist of multiple parts (components). A part can be associated with one or more symbols, a datasheet, footprint details, analysis models, 3D models, and so on. Multiple parts can also refer to the same symbol. The following chart depicts the part and data relationship:



Part and Data Relationship Model

Managing Part Libraries

The topics covered in this section are:

- [Creating a Library](#)
- [Opening a Library](#)
- [Editing a Library](#)
- [Renaming a Library](#)
- [Closing and Saving a Library](#)

Creating a Library

In Capture, you can create as many libraries as required by specifying a name and storage location for each library. Each library is available to each [project](#). The library size is limited only by the amount of space on your system's hard disk; however larger libraries take longer to load. If speed becomes an issue, consider creating several smaller libraries.

 Schematic folders and [schematic pages](#) cannot be created in a library, but can be copied or moved to a library from a project. Schematic folders and schematic pages can also be edited in a library.

When you create a new library, project manager adds an empty library to the project. To populate the library, you can create your own parts, or you can move or copy parts from another library. See [Moving parts or symbols between libraries](#) for details.

To create a new library from project manager, do the following:

1. Right-click the new library icon and save the new library with a file name and location of your

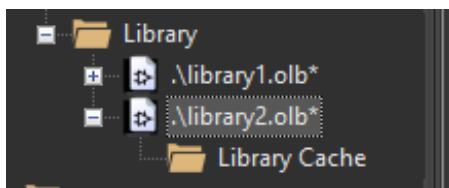
choice.

2. Populate the library by moving or copying files from other libraries. For more information, see [Moving parts or symbols between libraries](#).

Alternatively, do one of the following:

1. Choose *File – New – Library*.

Capture adds a Library folder and a *Library Cache* folder to the project.



If you have an instance of project manager currently open at the time, Capture adds the library to the project. Otherwise, Capture creates a new project for the library.

You can create new parts and symbols in the library.

To rename a library:

1. In project manager, select the library.
2. Choose *File - Save As*.
3. Specify a new name in the *File Name* text box.
4. Click *OK* to return to project manager.

To specify the storage location for a library:

1. In project manager, select the library.
2. Choose *File - Save As command*.
3. Select the drive and directory in which you want to store the library.
4. Click *OK* to return to project manager.

Editing a Library

In this section:

- [Copying a Part from the Design Cache to a Library](#)
- [Copying a Schematic Page to a Library](#)
- [Copying a Schematic Folder to or from a Library](#)
- [Moving Parts or Symbols between Libraries](#)

Copying a Part from the Design Cache to a Library

You can copy parts from the design cache to a library. This is useful if you have modified a part through the schematic page editor, and want a permanent copy of the part.

To save a design cache part in a library:

1. In project manager, open the design cache, and select the part to save.
2. Choose *Edit - Copy*.
3. Open the library where you want to store the part.
4. Choose *Edit - Paste*.

Alternatively, you can open two instances of project manager, one for the design and one for the library, and drag-and-drop the part from the design cache to the library.

-  • If you edit an OrCAD-provided library, it is important to assign a new name to the library so that the changes you make are not overwritten when you upgrade or update your software.

Copying a Schematic Page to a Library

If you have a small circuit that is used in many projects, you can put the circuit on a separate schematic page, save it in a library, and attach it to a part that you can place in any design. It is recommended that you keep the attached schematic folder and the part in the same library.

To save a schematic page in a library, you must first move the page into a schematic folder. When you save a schematic page in a library, Capture puts the circuit's parts in the library cache. A schematic folder or schematic page that is open in an editor cannot be moved or copied.

 Before editing a schematic page stored in a library, you should find out which projects call the schematic folder. Editing a library-stored schematic folder may create problems for other projects that use the library-stored schematic folder.

To save a schematic page in a library:

1. Verify that the page is not open in the schematic page editor.
2. In project manager, create a schematic folder to store the page.
3. Select the schematic page you want to save, and copy or move the page into the new schematic folder.

Keep the new schematic folder selected.

4. From the *Edit* menu, choose *Copy*.
5. Open project manager for the library in which you want to store the schematic folder.
6. From the *Edit* menu, choose *Paste*.

OR

1. Verify that the page is not open in the schematic page editor.
2. Open project manager for the project and create a schematic folder to store the page.
3. Open project manager for the library in which you want to store the schematic folder.

4. Drag and resize the project manager windows for the project and the library so that each is visible.
5. In the project manager window for the project, select the schematic page you want to save, and copy or move the page into the new schematic folder. Keep the new schematic folder selected.
6. Press and hold Ctrl while you drag the schematic folder to the project manager window for the library.

 If you copy or move a document from one design or library to another, you must save the destination design or library immediately. Else, you may lose data if you open the moved document in the schematic page editor or part editor and close the editor without saving the document.

Copying a Schematic Folder to or from a Library

The behavior of schematic folders and schematic pages is almost identical in libraries and designs.

The primary differences are:

- Schematic folders and schematic pages cannot be created in libraries. If you want to add a schematic folder or a schematic page to a library, you must create it in a design and then move it to the library.

In addition to this topic, see [Copying a Schematic Page to a Library](#) for more information.

- Schematic folders and schematic pages are limited to the library tool set, namely updating properties, exporting properties, and importing properties.
- The *Update Cache* and *Replace Cache* commands are not available when parts are selected in the library cache.

- The *Annotate* command is unavailable for parts in schematic folders contained in libraries. You should use the [Annotate command](#) in a schematic folder prior to moving it to a library.
- You can open a library-stored schematic page in a schematic page editor, and edit it exactly the same way as if you had opened it from a design. However, it is recommended that you edit a schematic page in a design.
- If a schematic folder stored in a library is the child of another schematic folder in a design, the [Descend Hierarchy command](#) in the parent schematic folder opens the library containing the child schematic folder and a schematic page editor window containing the schematic page.

If you have a circuit that you use in many [projects](#), you can put that circuit in a separate [schematic folder](#), save it in a [library](#), and attach it to a part that you can then place in any design. It is a good design practice to keep the part and the attached schematic folder in the same library. When you save a schematic folder in a library, Capture puts the circuit's parts in the library cache.

A schematic folder or [schematic page](#) that is opened in an editor cannot be moved or copied.

To save a copy of a schematic folder as a library object:

1. Verify that no part is opened in the schematic folder.
2. In project manager, select the schematic folder to save.
3. Choose *Edit - Copy*.
4. Open the project manager window for the library in which you want to store the schematic folder.
5. Select the library (.OLB) in the project manager window.
6. Choose *Edit - Paste*.

Alternatively, you can open two project manager windows for the library and the design and drag-and-drop the schematic folder from the design to the library.

When using the drag-and-drop procedure, you need to keep the *Ctrl* key pressed.

7. Choose *File* - [Save](#) command.

 If you copy or move a [document](#) from one design or library to another, you must save the destination design or library immediately. Else, you may lose data if you open the moved document in the [schematic page editor](#) or [part editor](#) and then close the editor without saving the document.

To copy a schematic folder from a library to a design:

1. In project manager, open the library containing the schematic folder that you want to copy and select the schematic folder.
2. Choose *Edit* - [Copy](#).
3. Open the project in which you want to store the schematic folder.
4. Select the design (.DSN) in the project manager window.
5. Choose *Edit* - [Paste](#).

Alternatively, you can open two project manager windows for the design and the library and drag-and-drop the schematic folder from the library to the design.

When using the drag-and-drop procedure, you need to keep the Ctrl key pressed.

6. Choose *File* - [Save](#).

You can also attach a schematic folder to a part or hierarchical block without copying the schematic folder into the project or library, but this method affects design portability. For more information, see [Attaching a schematic folder](#).

Moving Parts or Symbols between Libraries

You can store parts or symbols in any library. You can create libraries that serve a specific purpose. You can transfer some parts or symbols from one library to another and store some parts or symbols in multiple libraries. If you move a part with part alias from one library to another, the part alias is also moved to the library.

 A part that is open in an editor cannot be moved or copied.

To move or copy parts between libraries:

1. Verify that the parts are not open in the part editor or the spreadsheet editor.
2. In project manager, select the parts to move.
3. To move the part, choose *Edit - Cut*. To copy the part, choose *Edit - Copy*.
4. In project manager window, select the library (.OLB).
5. Choose *Edit - Paste*.

Alternatively, you can open two instances of project manager for the two libraries and drag-and-drop the parts between the libraries. To copy the parts, keep the Ctrl key pressed while you drag-and-drop the parts.

 If you edit a library provided by OrCAD, it is important that you assign a new library name so that the changes are not overwritten when you upgrade or update your software.

To copy or create a part using an existing part:

1. Follow the procedure for moving parts between libraries to create a copy of the part in another library.
2. If you want to move the new part (created from an existing part) to the original library:
 - a. Use the [Rename command](#) and specify a different name for the part.

- b. Follow the above procedure to drag the part to the original library.

Renaming a Library

To rename a library:

1. In project manager, select the library (.OLB) file.
2. Choose *File - Save As*.

The *Save As* dialog box displays. In this dialog, you specify an alternative name for the library. You can also specify an alternative directory location for the library.

3. Specify a name for the library and click *OK*.

⚠ While using the *Save As* dialog box to rename the library, do not specify a dot (.) in the library file name. Renaming a library in this manner breaks the links between the library and the parts selected from it, and the *Update Cache* command does not work. Such libraries also are not listed in the *CAPTURE.INI* file for configuration.

Closing and Saving a Library

The changes you make to a part are temporary until you save the part or the library using one of the commands on the *File* menu. When you save a library, you save all the parts and symbols residing in the library. If there are several parts or symbols opened in the part editor, the changes you make to any of them are saved. If the library is new and has not yet been saved, the *Save As* dialog box appears, giving you the opportunity to specify a drive and replace the system-generated name.

To save a library:

- From the *File* menu in project manager, choose the *Save* command.

The active library or the library that holds the active part is saved.

To save a library with a different file name:

1. Open the project manager window for the design or the library.
2. Choose *File – Save As*.
3. Specify a new file name in the *File Name* text box, and click *OK*.

For compatibility with future versions of Windows, Capture preserves the case of the path and filename as you specify them in the *Library* text box.

 When you save a project, Capture automatically creates a backup with a *.DBK* file extension. When you save a library, Capture automatically creates a backup with a *.OBK* file extension. If you save only a schematic page or a part, no backup is generated.

To close a library:

- From the *File* menu in project manager, choose the *Close* command.

Using Part Libraries

You can open and edit a library file from the (Windows) File Explorer as well as in Capture.

Instead of editing parts in libraries provided by OrCAD X, copy the part and save the changes in a custom library. In case you edit a library provided by OrCAD X, ensure that you assign a new library name using the

File – Save As command from the main menu, so that your changes are not overwritten when you update or upgrade the software.

If you move a library after you place a part, the connection between the part and its library is broken. In this case, the *Update Cache* command cannot locate the library and you need to use the *Replace Cache* command and specify the new path to the library.

The topics covered in this section are:

- [Adding a Library to a Project](#)
- [Creating a Library](#)
- [Opening a Library](#)
- [Editing a Library](#)
- [Renaming a Library](#)
- [Closing and Saving a Library](#)
- [Using the Library Correction Utility](#)
- [Exporting and Importing a Capture Library](#)
- [Configuring Preinstalled Libraries](#)
- [Adding a Library to a Project](#)
- [Creating a Library](#)

- [Opening a Library](#)
- [Editing a Library](#)
- [Renaming a Library](#)
- [Closing and Saving a Library](#)
- [Using the Library Correction Utility](#)
- [Exporting and Importing a Capture Library](#)

Adding a Library to a Project

When you add a library to a project, all the parts contained within the library become available for placement on the schematic pages.

To add an existing library to your project:

1. From the *File* menu, choose *Open - Library*.
2. Select the library you want to open.

If the library file is not listed:

- In the *Look in* drop-list box, select a new drive, a new directory, or both.
 - In the *Files of type* box, select the type of file to open.
3. Choose the *Open* button. The project manager window opens and the library parts appear in project manager.

 If you open an SDT library, Capture prompts you to save the library.

To add a recently used library to your project:

- From the *File* menu, choose the name of the library you want to add.

 If you move a library after you place a part, the connection between the part and the library is broken. In this case, the *Update Cache* command will not find the library and you need to use the *Replace Cache* command and specify the new path to the library.

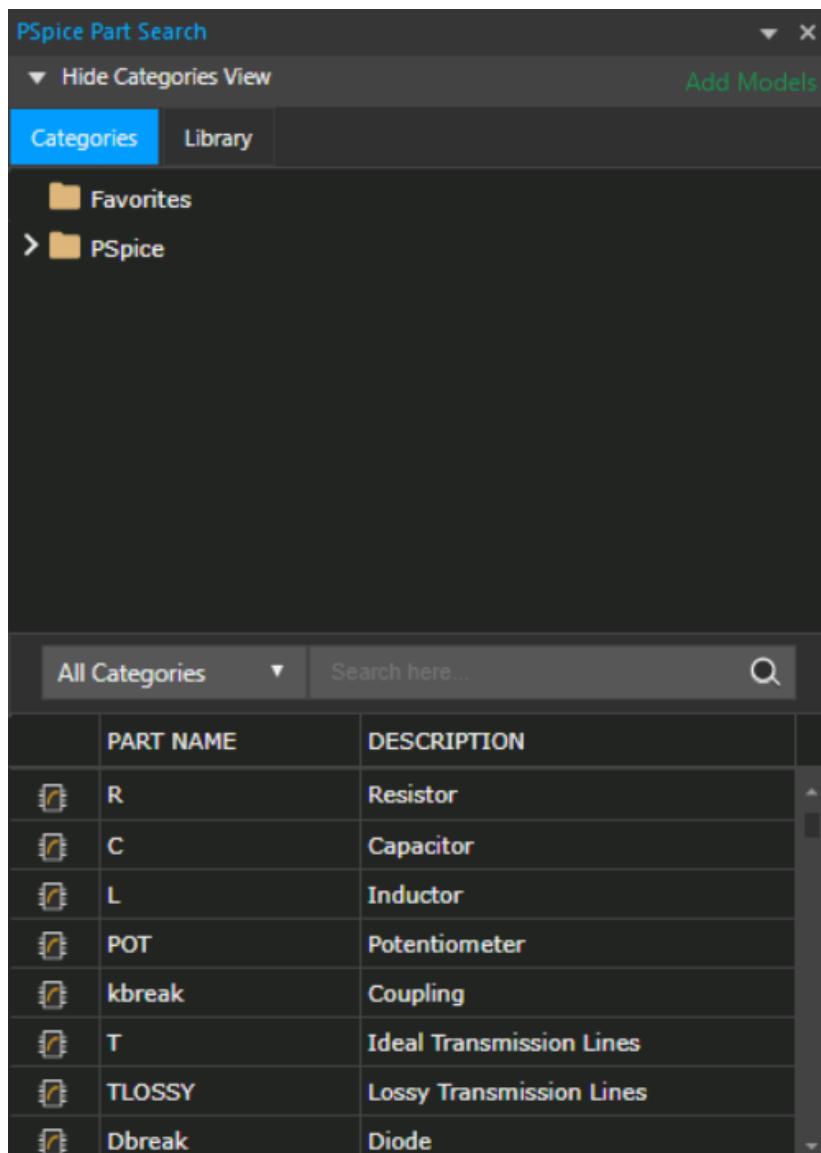
Configuring Preinstalled Libraries

If you have *PSpice for TI* (S006 or above) installation on your system, you can configure libraries from that installation in OrCAD Capture 17.4 (hotfix 017 or above). Click the *Add Models* option in the PSpice Part Search pane to configure the *PSpice for TI* libraries. To do so, perform the following tasks:

1. Open a project.
2. In the PSpice Part Search pane that opens, click the *Add Models* option.

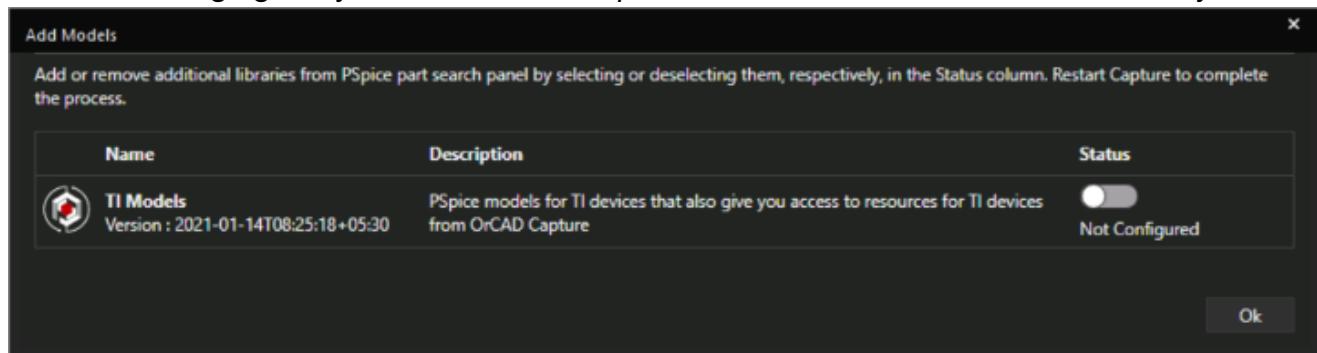
Library and Part Management in OrCAD X Capture

Using Part Libraries--Configuring Preinstalled Libraries

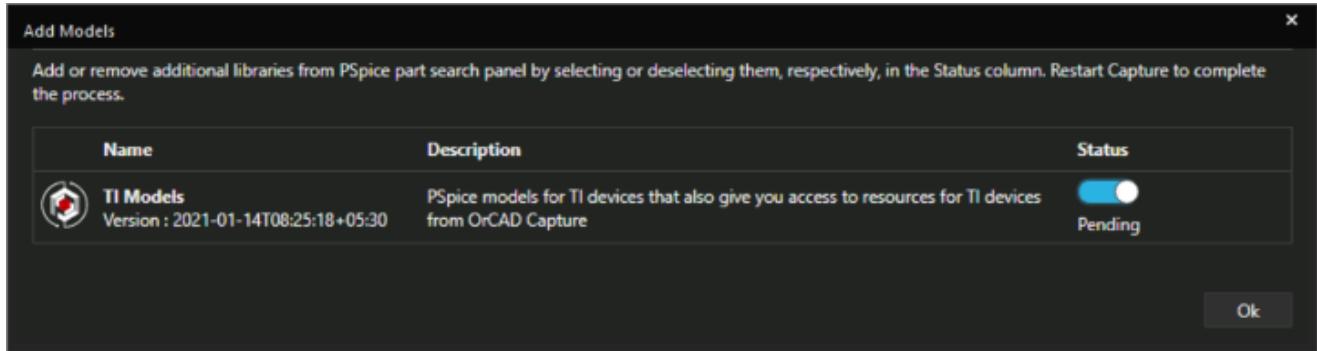


The *Add Models* window opens. This shows the list of preinstalled libraries on your system and also allows you to add or remove any of these libraries.

In the following figure, you see that the *PSpice for TI* libraries are installed on the system.

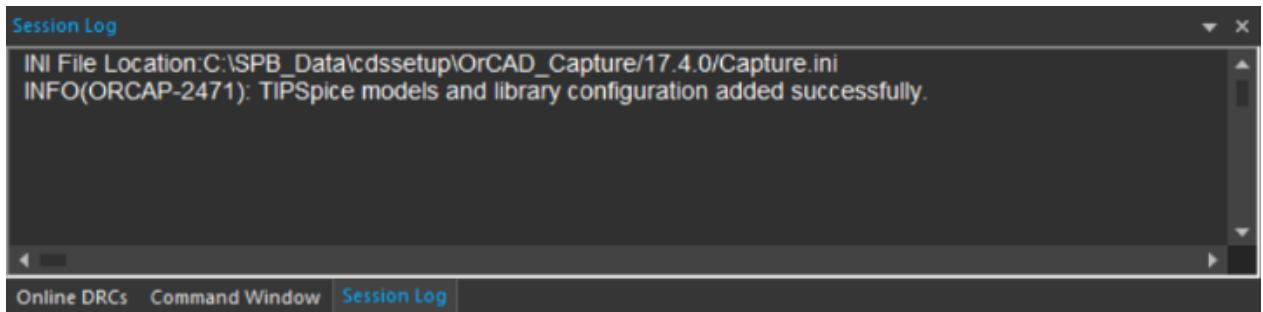


3. Toggle the *Status* in the *Add Models* window. The status changes to *Pending*.



4. Once you restart Capture, you will observe the following changes:

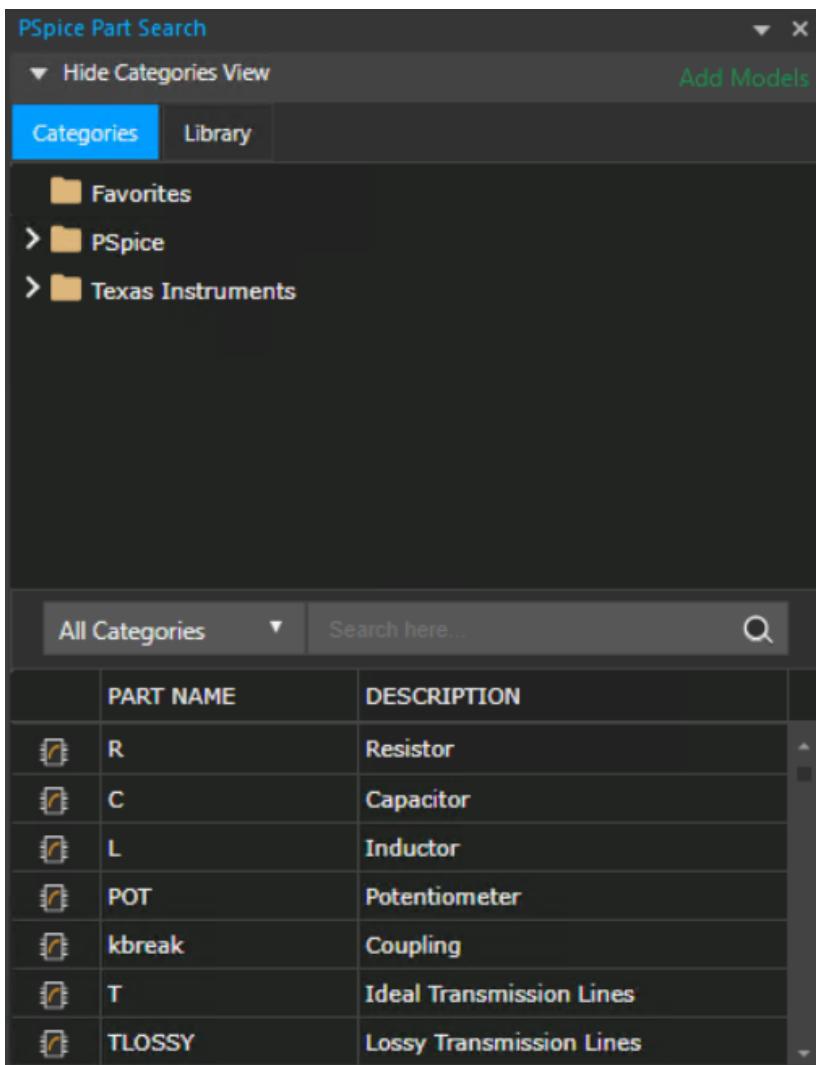
- The following message appears in the *Session Log* tab.



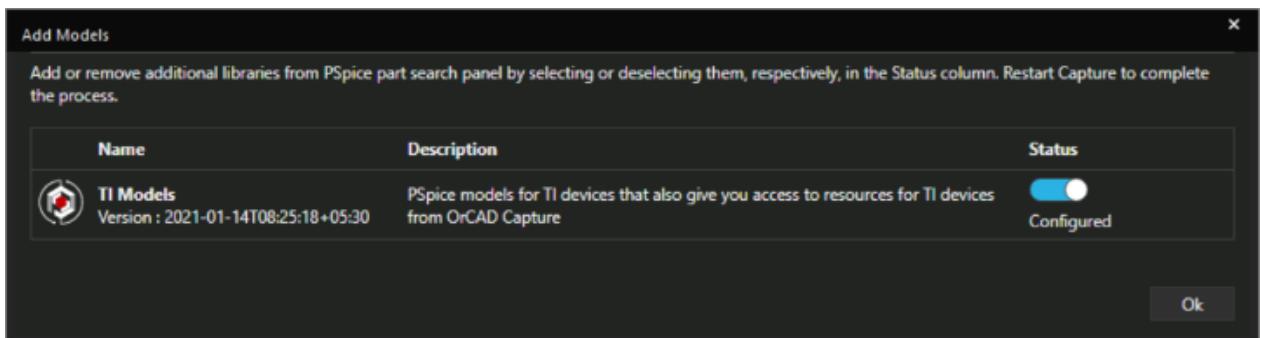
- *Texas Instruments* (TI) libraries are configured and available in PSpice Part Search.

Library and Part Management in OrCAD X Capture

Using Part Libraries--Configuring Preinstalled Libraries



- Click *Add Models* in PSpice Part Search, and you see that the status in the *Add Models* window has changed to *Configured*.



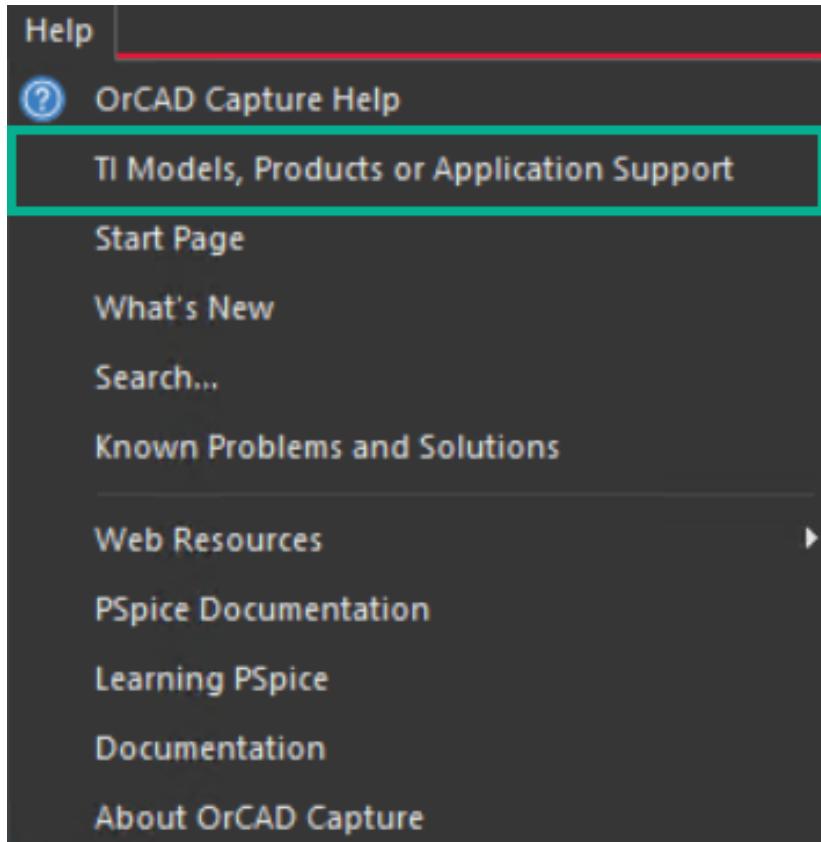
⚠ Important

Once the *PSpice for TI* libraries are configured, the check box, *Prioritize models for same .lib for all circuits* is selected in the Simulation Settings dialog box. You cannot modify this selection.

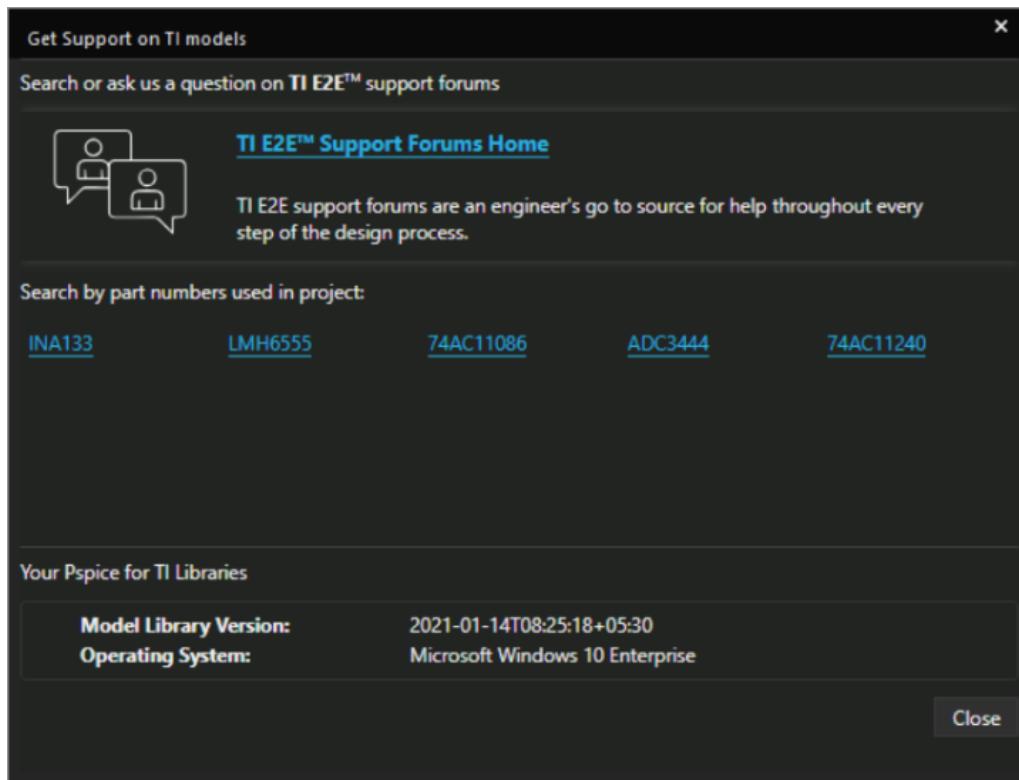
You will also observe the following new features with after configuring preinstalled libraries.

Help Menu Option: TI Models, Products or Application Support

After you configure the preinstalled libraries, you will observe a new *Help* menu option, *TI Models, Products or Application Support*.



If you click this option, the *Get Support on TI models* dialog box opens. This window displays all the TI parts used in the current active project. This window also shows the installed version of the model library. When you click a part link, the Web page for E2E™ support forums opens showing information related to this part.



Product Page for Selected Part

After you configure the preinstalled libraries, you can open the product page at TI.com for a part selected from these libraries.

Library and Part Management in OrCAD X Capture
Using Part Libraries--Configuring Preinstalled Libraries

The screenshot shows the PSpice Part Search window. At the top, there's a toolbar with 'Hide Categories View' and 'Add Models'. Below it, a navigation bar has 'Categories' selected (highlighted in blue) and 'Library' as the other option. The main area is a tree view of component categories:

- Favorites
- PSpice
- Texas Instruments
 - Amplifiers
 - Comparators (28)
 - Current sense amplifiers
 - Difference amplifiers (13)
 - Fully differential amplifiers (13) **(selected)**
 - Instrumentation amplifiers (23)
 - Operational amplifiers (op amps)
 - Programmable & variable gain amplifiers (PGA) (6)

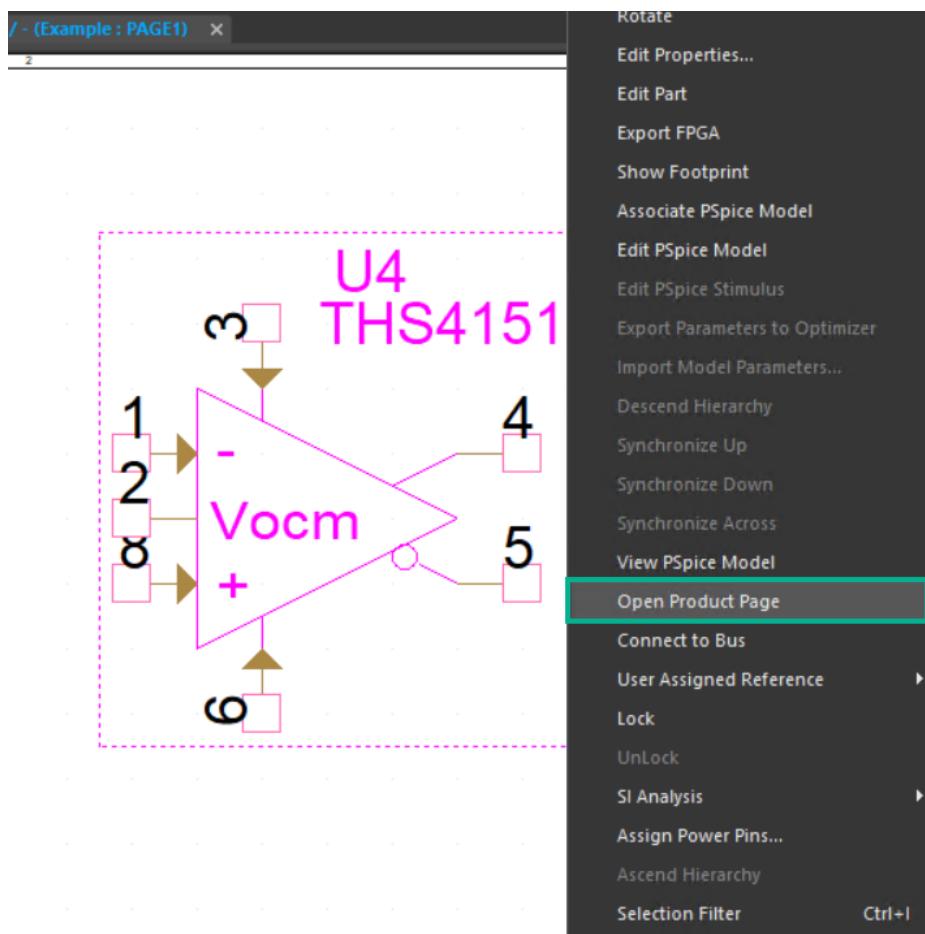
Below the tree is a search bar with 'All Categories' and a search input field. A magnifying glass icon is next to the search input.

At the bottom, there's a table with columns 'PART NAME' and 'DESCRIPTION'. The table lists several parts from the 'Fully differential amplifiers' category:

PART NAME	DESCRIPTION
THS4121	3.3 V, 100 MHz, 43 V/us, Fully Diffe...
THS4130	150MHz, Fully Differential Input/Ou...
THS4131	Fully Differential Input/Output Low ...
THS4141	Place Symbol
THS4151	Fully Differential Input/Ou...
THS4503	View Symbol
THS4505	Parameters
THS4522	Add to Favorites
THS4524	Open Product Page
THS4524	Open TI E2E™ Page
THS4531	Ultra low power 0.25mA, RRO, fully...

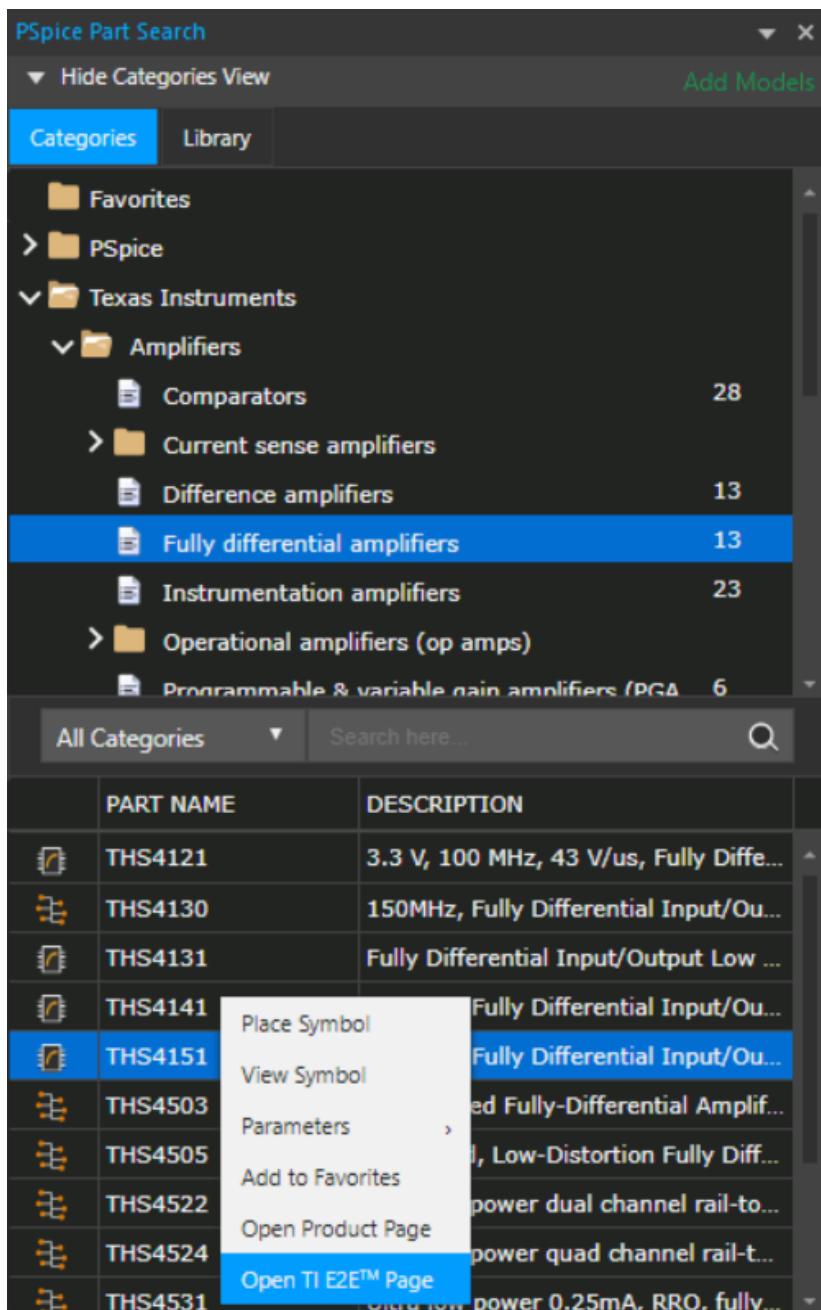
A context menu is open for the THS4151 row, listing options: Place Symbol, View Symbol, Parameters, Add to Favorites, Open Product Page, and Open TI E2E™ Page. The 'Open Product Page' option is highlighted with a blue background.

You can also open this product page after the part is placed on the schematic.



TI E2E™ Page for Selected Part

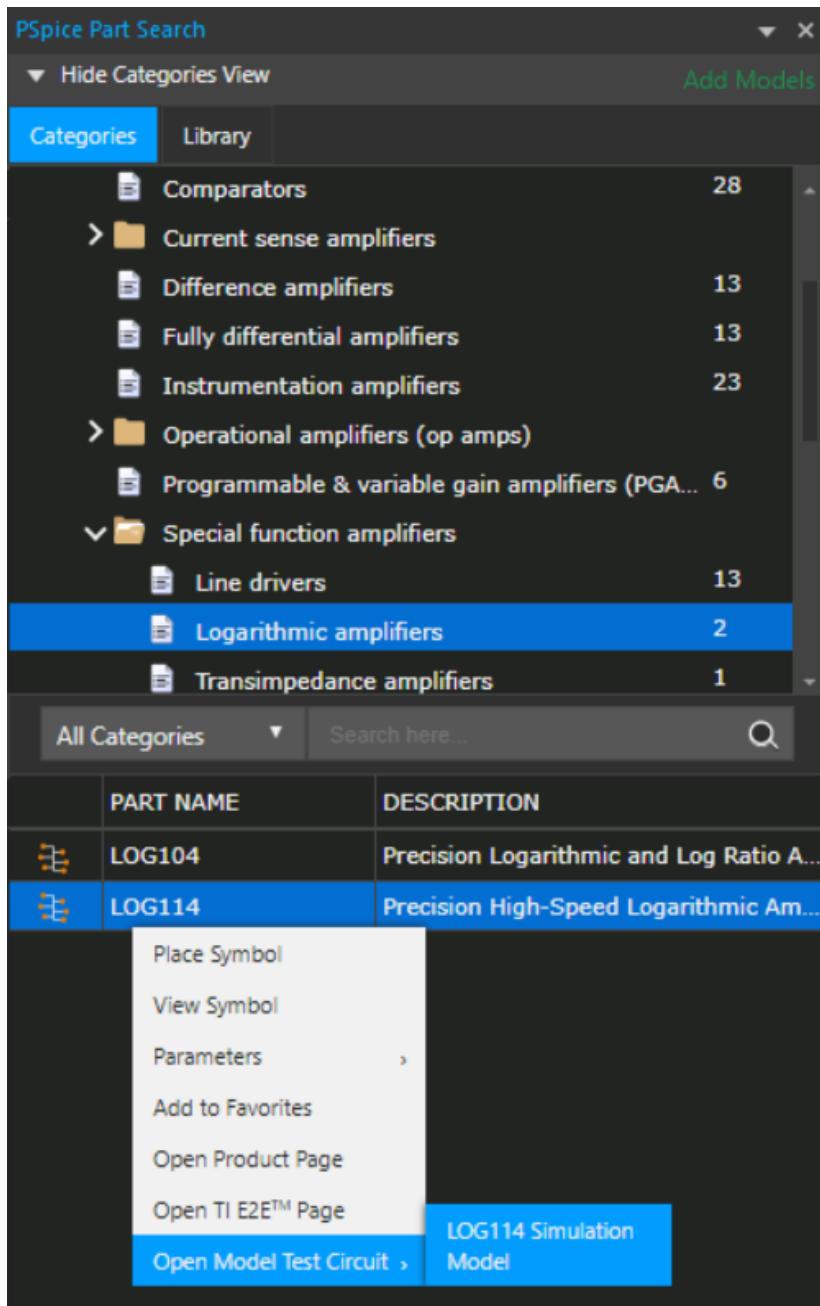
The *Open TI E2E™ Page* option is available to view the Web page for E2E™ support forums that shows information related to the selected part.



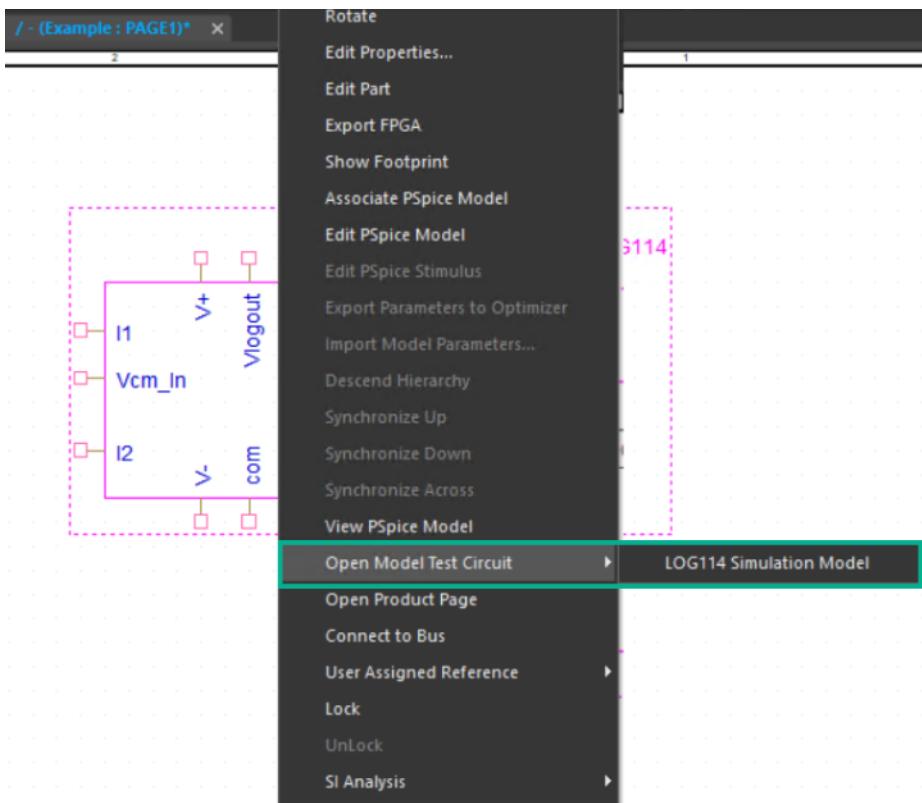
Model Test Circuit for Selected Part

After you configure the preinstalled libraries, you can directly download (from TI.com), and simulate a related model test circuit (if available) for a part selected from these libraries.

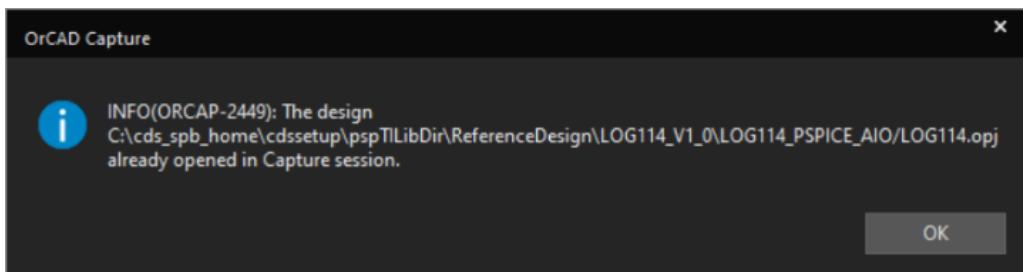
Library and Part Management in OrCAD X Capture
Using Part Libraries--Configuring Preinstalled Libraries



You can also open the model test circuit after you have placed this part on the schematic.



If you have already downloaded a model test circuit and select *Open Model Test Circuit* again for the same part, the tool detects and indicates the same, and shows the following message:



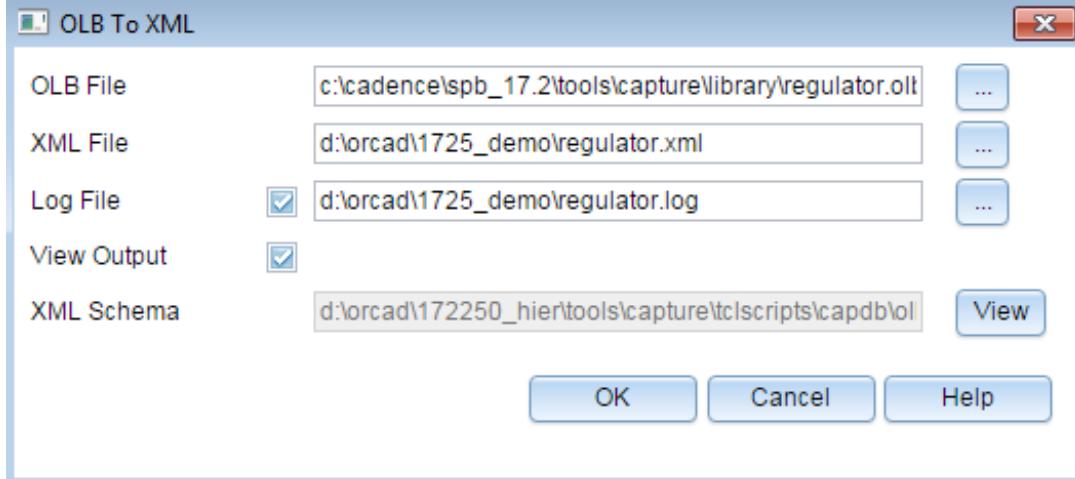
Exporting and Importing a Capture Library

You can convert a Capture library (**.olb**) to an XML (**.xml**) file. To do so, perform the following tasks:

1. Launch Capture.
2. Select *File – Export – Library XML*.

The OLB to XML dialog box opens.

3. Specify the library file path.
The XML File field gets updated.
4. Change the XML file name and location, if needed.



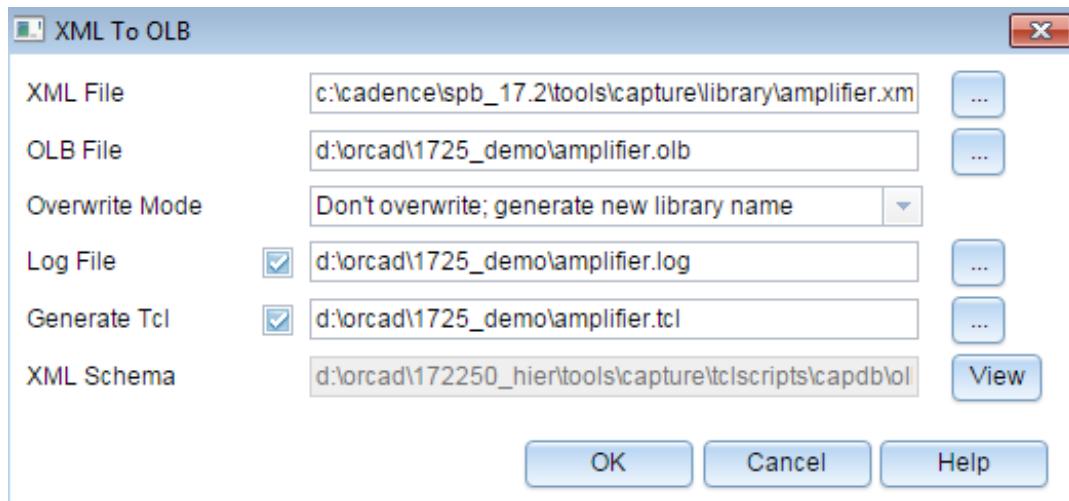
To convert a Capture library (.olb) file to an XML file, Capture uses an XML schema that is located at the following

location: <installation_directory>\tools\capture\tclscripts\capdb\olb.xsd

5. Click *OK*.
The XML file is displayed if you had selected the *View Output* option.

To import an XML to create a Capture library, do the following:

1. Launch Capture.
2. Select *File – Import – Library XML*.
The XML to OLB dialog box opens.
3. Specify the input XML file path.
The *OLB File* field gets updated.
4. Click the Browse button corresponding to the *OLB File* field if you want to save the generated library file at a different location. This is optional.
5. Select the *Overwrite Mode* from the drop-down list.



6. Click *OK*.

Opening a Library

When you open a library, you can edit it in part editor.

To open a library:

1. Choose *File - Open*.

The Open dialog box appears.

2. Select the library to open.

If the library file is not listed:

- In the *Look in* drop list box, select a new drive, a new directory, or both.
- In the *Files of type* box, select the type of file to open.

3. Select the *Open* button.

The Project Manager window opens; the library parts appear in Project Manager.

 If you open an SDT library, Capture prompts you to save the library.

To open a recently used library:

- From the *File* menu, choose the name of the library you want to open.

Using the Library Correction Utility

The library correction utility automatically verifies/corrects missing pin numbers and duplicate pin names in the Capture symbol library by scanning through all the parts in the library, finding out, and correcting the components that have missing pin numbers and duplicate logical pin names. This utility also converts components with lowercase pin names to uppercase and makes the Power pins visible for all the components in the library.

Library Correction Utility

Use the library correction utility to fix missing pin numbers and duplicate pin names issues in a design.

Starting the Library Correction Utility

To run the Library correction utility:

1. Choose *Accessories - LibCorrectionUtil - Library Verification/Correction*.
2. In the Library Correction Utility dialog box, browse to the library for correction in the *Select Library for Correction* field.

Verifying/Correcting Library Components With

Missing Pin Numbers

- Check the *Missing Pin Numbers* option to correct the components with missing pin numbers.

The library correction utility scans through all the parts in the library and finds out the components with missing pin numbers. All the missing pin numbers are updated by their corresponding pin names. At the end of updating process, the following message pops-up:

"Corrected <number of parts corrected> of Parts in <lib_path>/<lib_name>.olb for missing pin numbers" or "No Part in this library has missing pin numbers".

The log file is generated at the location where the selected library resides. The naming convention used for log files is <Library_name>Miss.log. The log file lists all the parts and also the pin numbers for each corrected part in the library. If you choose the Verify option, only the log file is generated but the library is not updated.

Missing pins Log File Example:

List of verified / corrected packages

CONNECTOR DB25

No missing Pin numbers

CAPACITOR NON-POL

CAPACITOR NON-POL.Normal 1

CAPACITOR NON-POL.Normal 2

1 parts in library C:\TEMP\MISSINGPIN_NUMBERS.OLB have missing pin numbers.

The last line of the log file indicates that one part in the MISSINGPIN_NUMBERS.OLB library has missing pin numbers. The pin names for the missing pin numbers are 1, 2.

Duplicate Pin Names

- Check the *Duplicate Pin Names* option to correct the components with missing pin numbers. The library correction utility scans through all the parts in the library and finds out the components that have duplicate logical pin names.
 - If a component has duplicate power pin names, those pins are not considered as duplicate pin names.
 - To remove duplicate pin names, the library correction utility changes the duplicate pin names by appending "#" followed by the pin number to the duplicate pin names.
 - The combination of the pin name and the pin number makes the pin name unique.

 The first pin that this utility encounters does not get appended with # followed by its pin number.

At the end of updating process, the following message pops-up:

"Corrected <number of parts corrected> of Parts in <lib_path>/<lib_name>.olb for duplicate pin names" or "No Part in this library has duplicate pin names".

The log file is generated at the location where the selected library resides. The naming convention used for log files is <Library_name>Dup.log. The log file lists all the parts and also the corrected pin names for each of the corrected part in the library. If you choose the *Verify* option in the next step, only the log file is generated but the library is not updated.

Duplicate Logical pins Log File Example:

List of verified / corrected packages

ASSYMETRICAL

ASSYMETRICALAA.Normal 2 A\#2

ASSYMETRICALCC.Normal 8 A\\#8

74LS00

No duplicate Pin names

```
1 parts in library C:\TEMP\DUPLICATE\_PIN\_NAMES.OLB have duplicate pin names
```

The last line of the log file shown above indicates that one part in the DUPLICATE_PIN_NAMES.OLB library has duplicate pin names. The corrected pin names are A#2 and A\#8.

Verifying vs. Correcting

- Select *Verify*.

The library is verified for the missing pin numbers and/or duplicate pin names and a log file is generated.

Or

- Select *Correct*

The library is corrected for missing pin numbers and/or duplicate pin names and a log file is generated.

 When you opt for correction, the library is corrected. Therefore, if you want to have a copy of the old library, you need to back-up the library. In fact, it would be always safe to make a backup of the old library before correcting the library.

Changing Casing of Pin Names and Numbers

- Check the *Change the Pin name and number to uppercase* option.

The utility scans through all the parts in the library and converts all the components that have pin names appearing in lowercase to uppercase.

At the end of updating process, the following message pops-up:

"Changed all the Pin name to upper case. Please close the library and reopen to see the change."

Making Power Pins Visible

When you check this option, the utility will scan through all the parts in the library and change the Power pins settings for all the non-zero length pins in the library to *Visible*. Additionally, you can also make all zero length Power pins in the library visible. To do this, check the *Change Zero Length Pins* to check box and choose an appropriate pin shape option (Line/Short) from the list box.

 The *Change Zero Length Pins* to check box is available only, if the *Make All Power Pins Visible* check box is checked.

At the end of updating process this utility will pop up message: " Made all the Power Pins visible. Please close the library and reopen to see the change."

Advantages of this Utility in the Flow

In the Capture-PCB Editor flow, new Capture-PCB Editor interface throws error, "Error ALG0031", if any of the components on your design have missing pin numbers. Old third party interface used to netlist the null pin number as pin name and take the component through flow. however, in that case the component was not updated at all.

In Capture-PCB Editor Flow, new Capture-PCB Editor interface throws error "Error ALG0050", if any of the components on your design have duplicate pin names.

Use Model for Capture-PCB Editor Flow

For New Designs:

Run this utility on library files, for correcting missing pin numbers and duplicate pin names, and use these libraries to create new designs. After correcting the libraries, you will not encounter errors, ALG0031 and ALG0050.

Correcting Existing Designs:

Run this utility on library files for correcting missing pin numbers and duplicate pin names, and then use the *Update Cache* command from the *Design* menu.

What if you do not have libraries for Existing Design?

Though this is not the preferred way, you can copy all the design cache components to a new library and then run this utility on that library. Now use the *Replace Cache* command from the *Design* menu. In this process, design cache points to a single library after replacing the cache. This affects the canonical path in the .pst files, as the source library name is embedded in the canonical path.

If you do not have all the libraries for the existing design, perform the followings steps:

1. Copy all design cache components to a new library.
2. Run the library correction utility on the new library, created in step 1.
3. Use Replace Cache command from Design menu.

Assumption:

This utility will put a pin name in place of an empty pin number. Matching footprints is your responsibility as a user because in some cases footprint may not match with pin numbers as shown in the following example:

Pin number	Pin name
------------	----------

1	A
	B
3	C
4	D

The Library Correction utility will copy B in place of the missing pin number between 1 and 3. In this case, footprint should contain pin numbers as 1,B,3,4 instead of 1,2,3,4.

Known Limitation:

If all the pin types other than Power have duplicate pin names and missing pin numbers, the missing pin numbers functionality copies the pin names in place of missing pin numbers. This is followed by the duplicate pin names functionality which appends pin names to pin numbers followed by #. Even then pin names are not unique. This results in error "Error ALG0050". For these parts, you need to edit each part and create unique pin names for each pin type other than the Power pin type.

Managing Libraries on OrCAD X Cloud

With the OrCAD X Professional (POX200 Pro) and OrCAD X Standard (POX100 Standard) licenses, you can create component libraries for your teams by creating new components from scratch or adding them from content providers.

This complete part authoring solution provides the following functions:

- Place components from content providers in OrCAD X Capture designs
- Search for components in content provider libraries
- Create local libraries by adding components from external libraries
- Create new components in a local workspace
- Link a manufacturer part number with an existing library component
- Create shared workspaces
- Collaborate with other team members on projects by publishing components to shared workspaces

Placing Components on Schematic

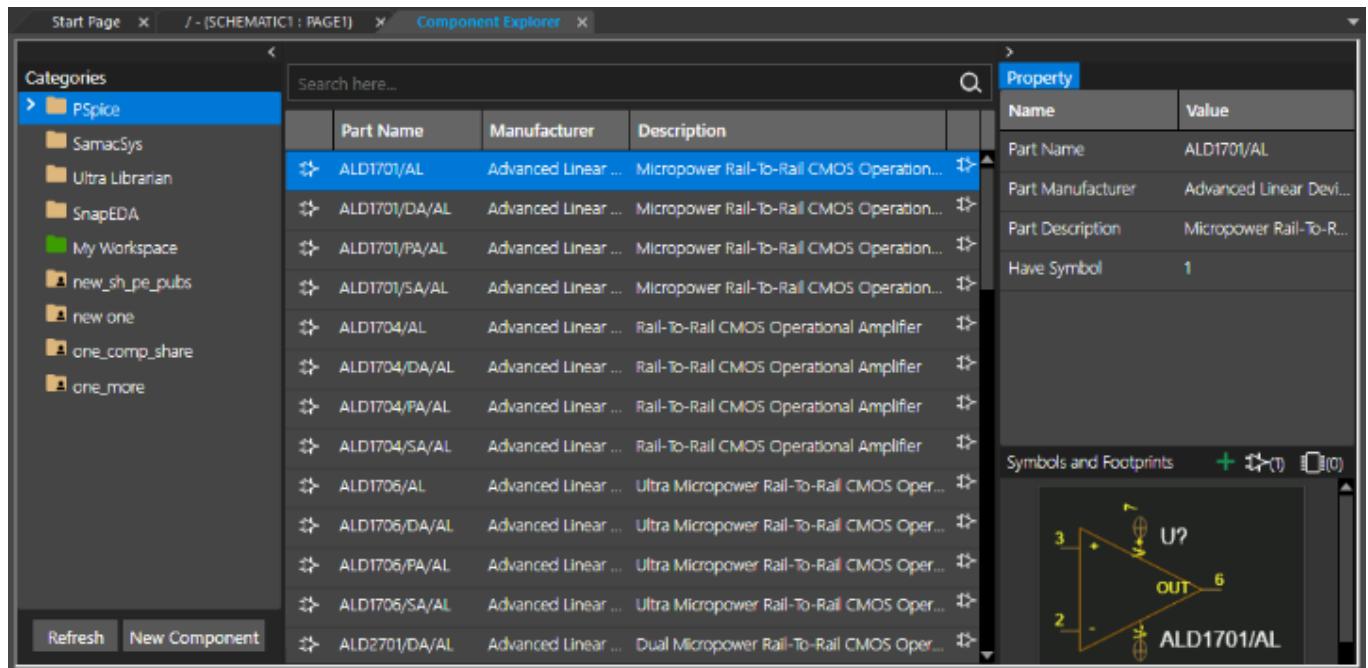
You can place components on a schematic page from the Cadence-supplied PSpice library, Cadence-supported content providers— SamacSys, Ultra Librarian, SnapEDA, or a workspace with previously-added parts from these sources.

To place a component on your schematic, do the following steps:

1. From the main menu of Capture, choose *Place – Component*.
The Component Explorer interface opens in a new tab.

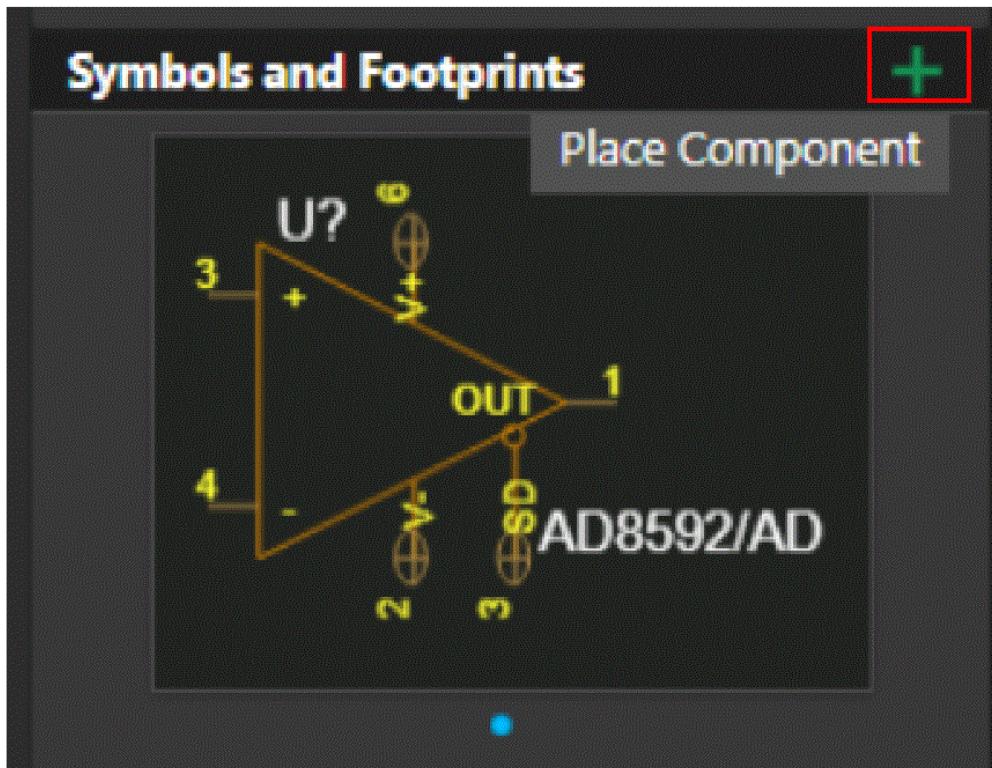
Library and Part Management in OrCAD X Capture

Managing Libraries on OrcAD X Cloud--Placing Components on Schematic



2. Double-click a component row in the part browser to place it on the schematic canvas.
Alternatively, use one of the following ways to place a component:

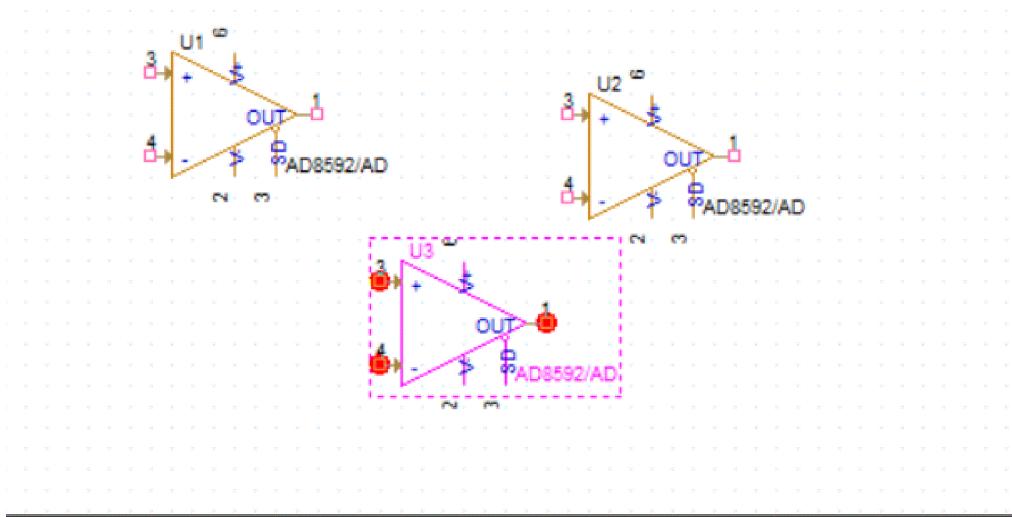
- Right-click a component and choose *Place*.
- Select the component and click the *Place Component* button on the *Symbols and Footprints* bar in the properties browser.



The schematic page tab opens and the selected component is attached to the cursor.

⚠ If a part has multiple symbols, the symbol that appears in the Symbols and Footprints view at the time of selection, is placed.

3. Click anywhere on the schematic to place an instance of the component.
4. Continue clicking to place as many instances of the component as required and press **ESC** when done.

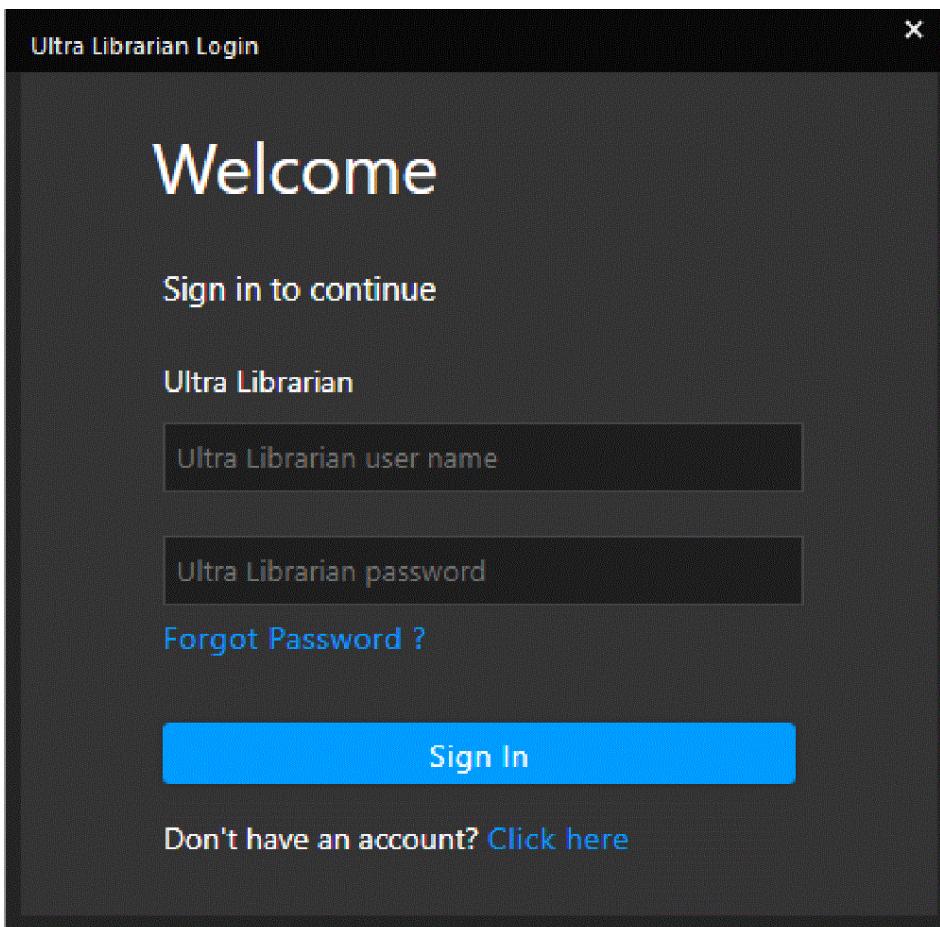


Similarly, you can add components from Cadence-supported content providers—SamacSys, Ultra Librarian, and SnapEDA.

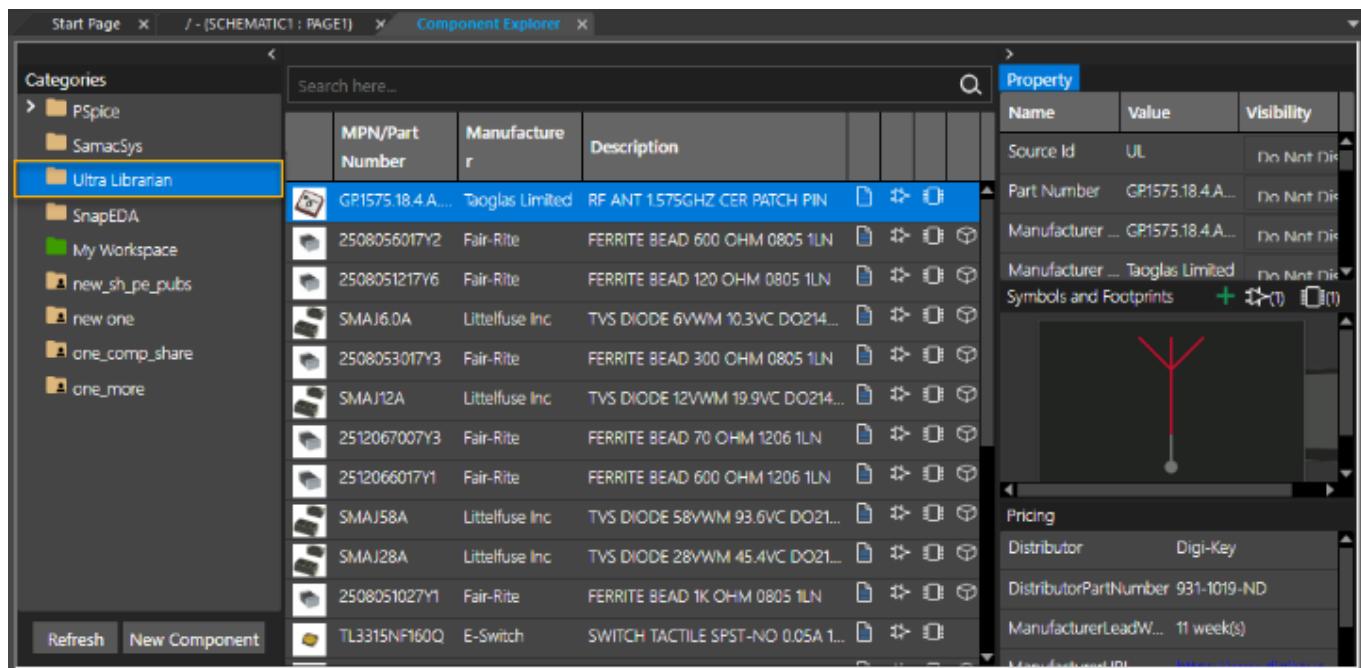
To add components to a Capture design from a Cadence-supported content provider, do the following steps:

1. Select *SamacSys*, *Ultra Librarian*, or *SnapEDA* in the left pane of the [Component Explorer](#) interface.
2. For Ultra Librarian, provide your login credentials and click *Sign In* or create an account, if you do not have one.

⚠ The components from *SamacSys* can be accessed with your cadence.com account. Access to the *SnapEDA* content requires a one-time registration. After you have registered and logged in, you are not prompted to supply login credentials for SamacSys and SnapEDA. For Ultra Librarian, you need to specify exclusive Ultra Librarian account credentials. The valid credentials are good for a session of Capture.



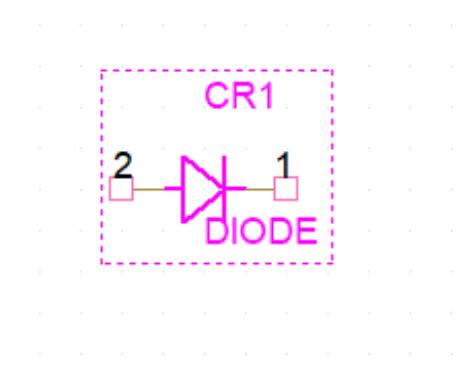
All the components in the selected source are displayed. Note that the availability status of the symbol, footprint, and a 3D image is available for each component along with the link to the datasheet associated with the component. In the *Symbols and Footprints* bar in the right pane, you can view the symbols and footprints associated with the component.



3. Double-click the required component in the part browser to download it and place it on the canvas.

The component is downloaded and attached to the cursor for placement on the schematic.

4. Click anywhere on the schematic to place an instance of the component.



Searching for Components

The Search feature provides a quick and convenient way to find and place parts from external libraries, and local and [shared workspaces](#).

The Search feature supports the following ways to find required components:

- Free text Search

- Field-based search
- Using SQL Query Syntax - Only supported for searching components in *My Workspace*.

To search components, do the following:

1. Select the source to look for a specific component in the *Categories* pane.
2. Specify the name or a search string in the *Search here* text box and press `Enter`, or click the magnifying glass icon.

All the fields displayed in the part browser grid, such as Part Name and Description, are searched for the specified search string.

The following table lists various search string combinations:

Search Type	Keyword	Examples
Free text search	Any information known about the component	An exact match: STPS1045B A substring: LDC1612
<i>Search all parts with specific field name</i>	Name of the field	part_description:Pulse Width Modulator
<i>Search all parts with OR Clause</i>	OR	part_description:N Channel MOSFET OR part_description:Pulse Width Modulator
<i>Search all parts with AND Clause</i>	AND	part_description:140mA part_description:Signal MOSFET
<i>search all part with NOT clause</i>	NOT	part_description:MOSFET NOT Small Signal MOSFET

<i>Combine FTS Query with SQL Query</i>	SQL Query to be enclosed in square brackets	<pre>part_description:MOSFET AND [part_description NOT LIKE 'N Channel MOSFET'] generic_part_number:cdn* AND [part_description LIKE "%cer%"]</pre>
---	---	---

A filtered list of components matching the specified search string or query is displayed in the part browser.

part_description:MOSFET AND [part_description NOT LIKE 'N Channel MOSFET']			
Part Number	Description	Lifecycle Status	
cdn-nmos-001	20-V, N channel single MOSFET, 19.2 mOhm		Active

generic_part_number:cdn* AND [part_description LIKE '%cer%']			
Part Number	Description	Lifecycle Status	
cdn-cap-001	1uF Ceramic Capacitor, GPN 1		Pre-Release
cdn-cap-003	2.2uF Ceramic Capacitor, GPN 100		
cdn-cap-004	4.7uF Ceramic Capacitor		
cdn-cap-005	10uF Ceramic Capacitor		
cdn-cap-009	1uF Ceramic Capacitor		

Related Topics

[Component Explorer](#)

Managing a Local Component Library

The Component Explorer interface provides the following functions to create and manage local component libraries in a workspace:

- [Creating Local Components from External Sources](#)
- [Assigning Manufacturer Part Number](#)
- [Linking an MPN to a Component in Database](#)
- [Editing Components](#)

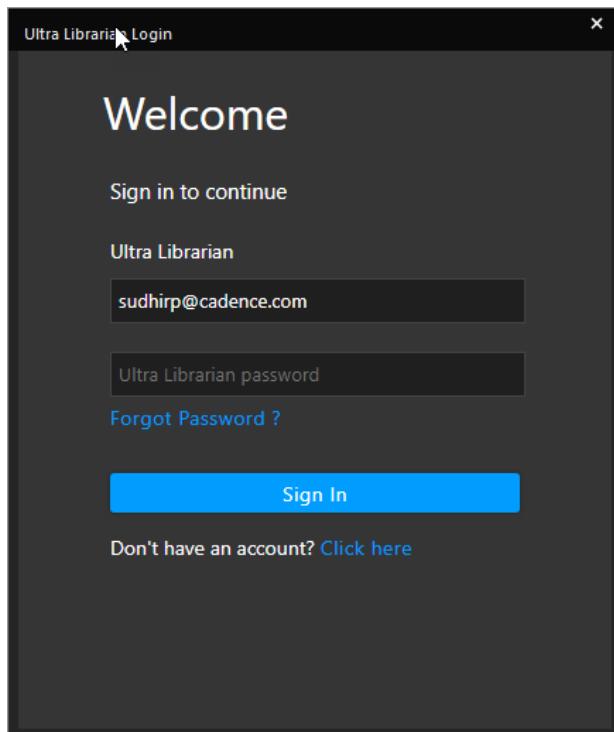
- [Copying Components](#)
- [Managing Database Components](#)

Creating Local Components From External Sources in Workspace

You can add components from Cadence-supported content search providers— SamacSys, Ultra Librarian, and SnapEDA to your local workspace and create your own local libraries with accurate and up to date part information.

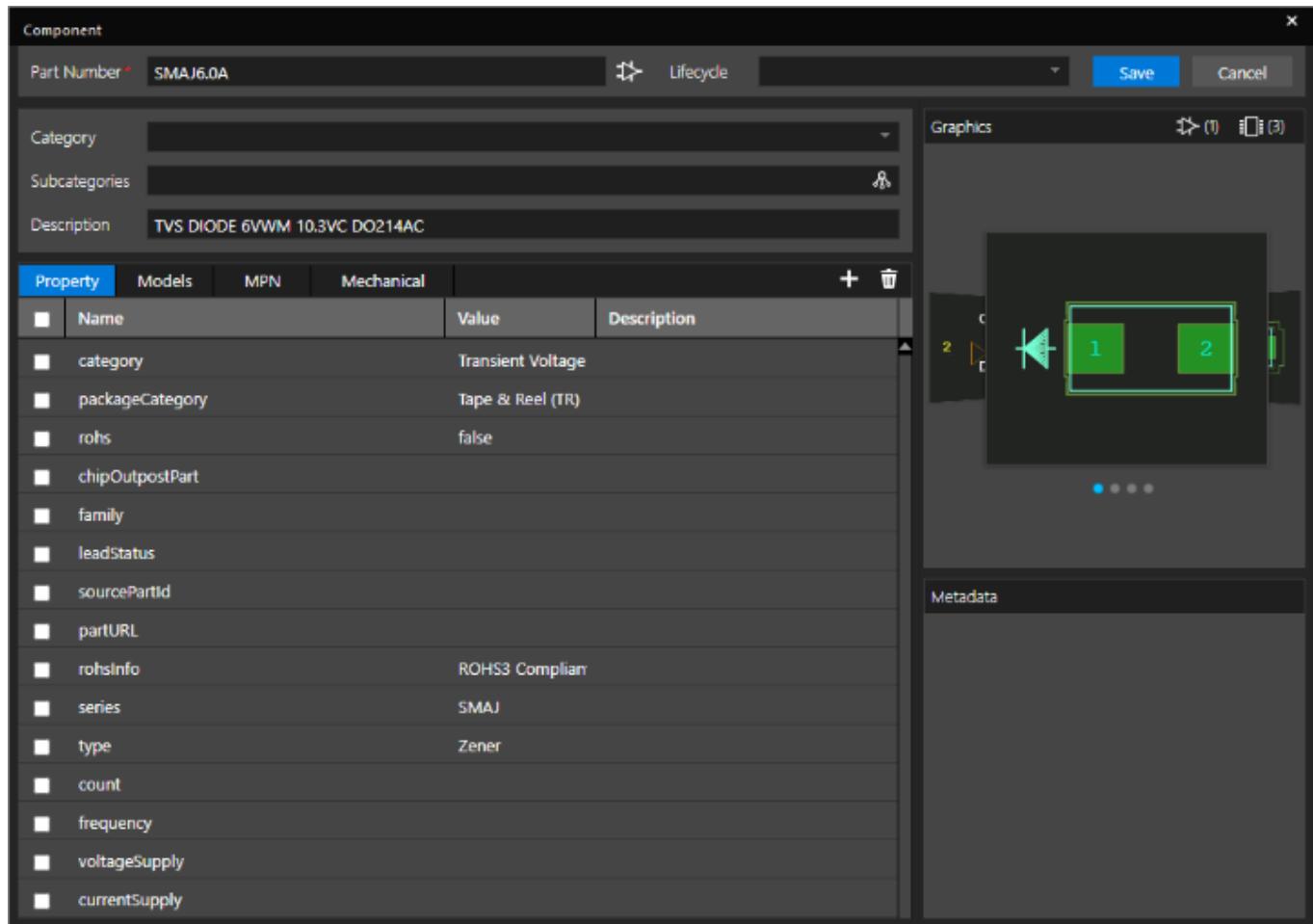
To add a component to the local workspace, *My Workspace*, do the following:

1. Click the node of the desired [content search provider](#) in the *Categories* pane and provide the login credentials when prompted:
 - a. When you select SamacSys or SnapEDA, your Cadence credentials are used to access the libraries and you are not prompted for any additional credentials.
 - b. When you select, Ultra Librarian, you need to sign in with a valid user name and password to access the libraries.

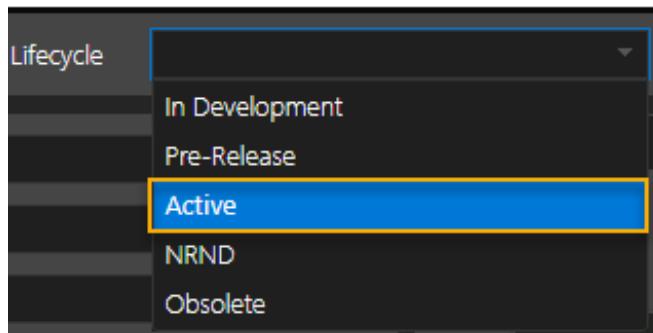


2. Right-click the desired component and choose *Add to Library*.
The component is downloaded and the *Component* dialog box opens. Before adding the

component to your workspace, you can customize its attributes or modify other [part details](#).

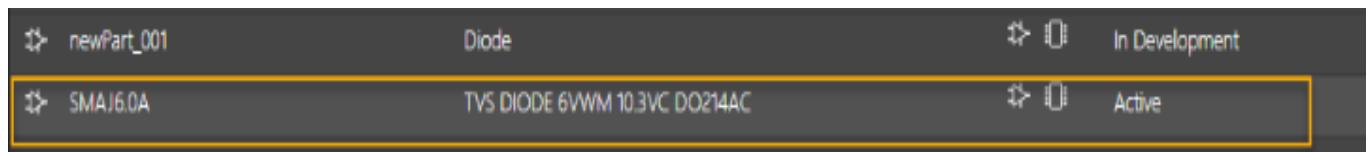


3. Select *Active* from the *Lifecycle* drop-down list to update the lifecycle stage of the component.



4. Click **Save**.
The component is added to the *My Workspace* node.
5. Click *My Workspace*.

The newly-added component is listed in the local workspace.

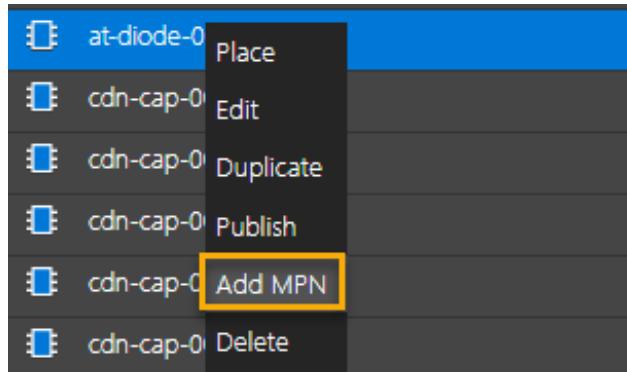


Assigning Manufacturer Part Number

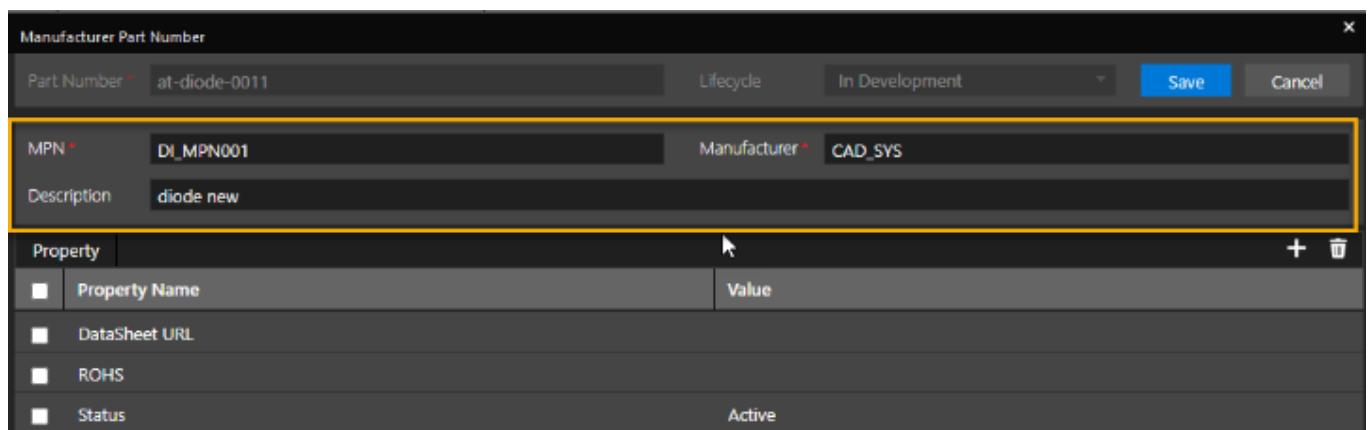
A manufacturer part number (MPN) is assigned to a component as a unique identifier from a specific manufacturer.

To assign an MPN to a component, do the following:

1. Open *My Workspace*.
2. Right-click a component in the part browser and choose *Add MPN*.



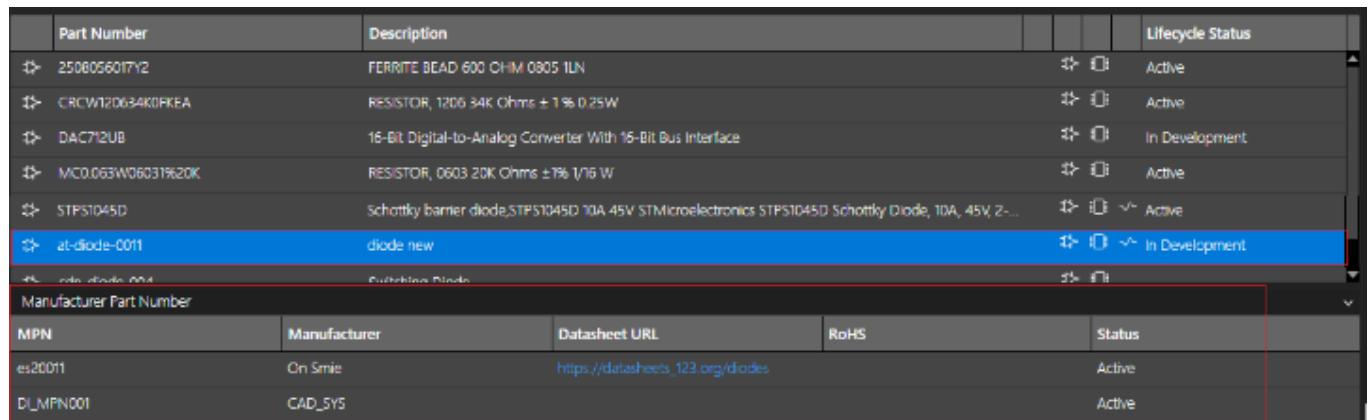
3. In the resultant *MPN Part Number* dialog box, specify an MPN and manufacturer name for the part.



You can add or modify properties as required.

4. Click Save.

The MPN details corresponding to the component display in the *Manufacturer Part Number* section.



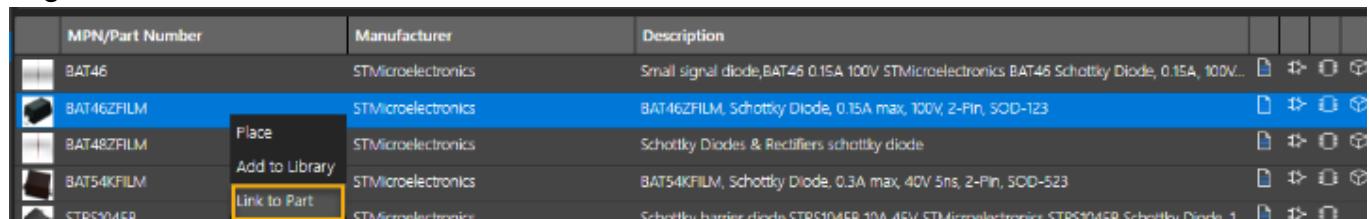
	Part Number	Description		Lifecycle Status
2508056017Y2	FERRITE BEAD 600 OHM 0805 1LN		Active	Active
CRCW120634K0FKEA	RESISTOR, 1206 34 Ohms ±1% 0.25W		Active	Active
DAC712UB	16-Bit Digital-to-Analog Converter With 16-Bit Bus Interface		In Development	In Development
MCO.063W06031R20K	RESISTOR, 0603 20K Ohms ±1% 1/16 W		Active	Active
STPS1045D	Schottky barrier diode,STPS1045D 10A 45V STMicroelectronics STPS1045D Schottky Diode, 10A, 45V, 2...		Active	Active
at-diode-001	diode new		In Development	In Development

Manufacturer Part Number				
MPN	Manufacturer	Datasheet URL	RoHS	Status
es2001	On Semiconductor	https://datasheets.onsemi.com/datasheets/diodes		Active
DLM_PN001	CAD_SYS			Active

Linking an MPN to a Component in Database

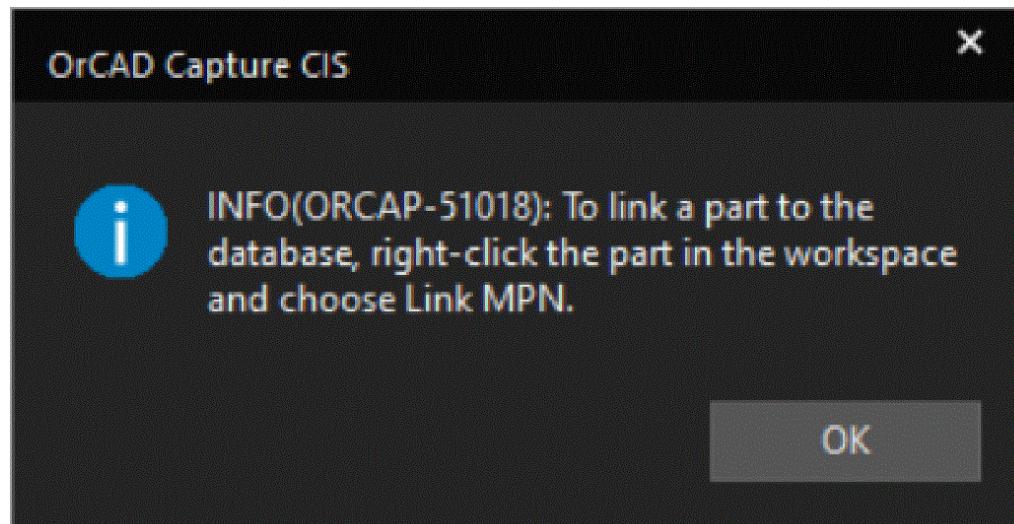
To link the existing MPN in a content provider's database to any component in your local workspace, do the following:

1. Click the required content provider node in the *Categories* pane.
2. Right-click the desired MPN and choose *Link to Part*.



MPN/Part Number	Manufacturer	Description	
BAT46	STMicroelectronics	Small signal diode,BAT46 0.15A 100V STMicroelectronics BAT46 Schottky Diode, 0.15A, 100V...	
BAT46ZFILM	STMicroelectronics	BAT46ZFILM, Schottky Diode, 0.15A max, 100V, 2-Pin, SOD-123	
BAT48ZFILM	STMicroelectronics	Schottky Diodes & Rectifiers schottky diode	
BAT54KFILM	STMicroelectronics	BAT54KFILM, Schottky Diode, 0.3A max, 40V 5ns, 2-Pin, SOD-523	
STPS1045B	STMicroelectronics	Schottky barrier diode,STPS1045B 10A 45V STMicroelectronics STPS1045B Schottky Diode, 1...	

A message pops up informing about the next step.



3. Click *My Workspace*.
4. Right-click the component to be linked and choose *Link MPN*.

	Part Number	Description
	at-diode-0011	diode new
	cdn-diode-001	Switching Diode
	cdn-diode-002	Switching Diode
	cdn-diode-003	Switching Diode
	cdn-diode-004	Switching Diode
	cdn-diode-005	Switching Diode
	mech2	This is a mechanical category

A context menu is displayed over the row for "cdn-diode-005". The menu items are: Place, Edit, Duplicate, Publish, Add MPN, Link MPN, and Delete. The "Link MPN" option is highlighted with a yellow box.

The *Manufacturer Part Number* dialog box opens. You can use the data as is or modify the details, including the MPN, manufacturer name, and property values.

Property	Value
Property Name	
bodyWidth	0
bodyLength	0
height	0
imageLarge	https://componentsearchengine.com/Images/3/BAT46ZFILM.jpg
pinCount	2
confidence	C4 - In house written
releasedDate	2018-01-16 19:45:40
sourceVersion	3.3
packageCategory	Other
category	Zener Diode
DataSheet URL	https://datasheet.datasheetarchive.com/originals/distributors/SFDatasheet-5/sf-0
Status	
Image URL	https://g.componentsearchengine.com/Thumbnails/3/BAT46ZFILM.jpg

5. Click Save.

The MPN from the content provider gets linked to the selected workspace component.

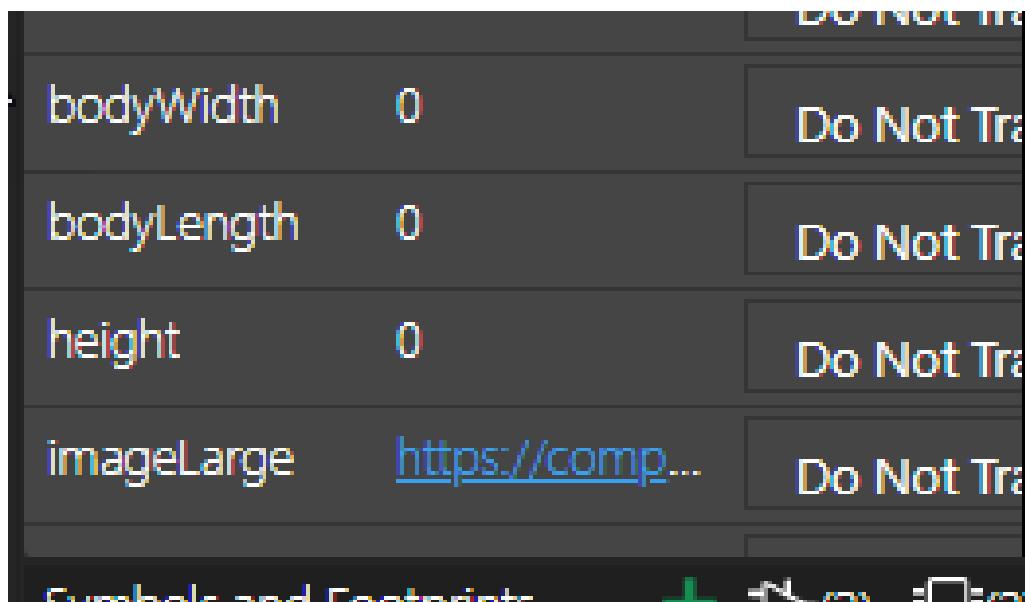
Manufacturer Part Number				
MPN	Manufacturer	Datasheet URL	RoHS	Status
es20011	On Smile	https://datasheets_123.org/diodes		Active
DI_MPNN001	CAD_SYS			Active
BAT46ZFILM	STMicroelectronics	https://datasheet.datasheetarchive.co...		

Similarly, you can link multiple MPNs to a component. The shortcut menu on a linked MPN provides options to place, edit, duplicate, and delete the MPN.

Manufacturer Part Number				
MPN	Manufacturer	Datasheet URL	RoHS	Status
es20011	On Semiconductor	https://datasheets.123.org/diodes		Active
DL_MPNO01	CAD_SYS			Active
BAT46ZFILM	STMicroelectronics	https://datasheet.datasheetarchive.co...		

When you click a linked MPN, the properties browser is populated with the properties of the MPN in addition to the part properties.

Property		
Name	Value	Visibility
Part Number	at-diode-0011	Do Not Display
PCB Footprint	do27a	Do Not Display
Manufacturer	BAT46ZFILM	Do Not Display
Manufacturer	STMicroelectronics	Do Not Display
Part Description	diode new	Do Not Display
Have Symbol	1	Do Not Display
Have Footprint	1	Do Not Display
DataSheetURL	https://datasheets.123.org/diodes	Do Not Display
Status		Do Not Display



Editing Components

You can edit a component in the *Component* dialog box using the following steps:

1. Click *My Workspace*.
2. In the part browser, right-click the component and choose *Edit*.
The *Component* dialog box opens.
3. Modify **part details** as required.
You can also edit the part number in the *Component* dialog box.
4. Click *Save*.

Copying Components

To create a copy of an existing component in the workspace, you can duplicate the component.

1. With the *My Workspace* node displayed, right-click the component in the part browser and choose *Duplicate*.
The *Component* dialog box opens an exact replica of the selected component.
2. Specify a new part number.
3. Modify **part details** as required and click *Save*.
A new component is added to the workspace.

Deleting a Component

You can remove a component from the workspace.

- To delete a component from the workspace, right-click the component and choose *Delete*.

The component is removed from the workspace.

Managing Database Components

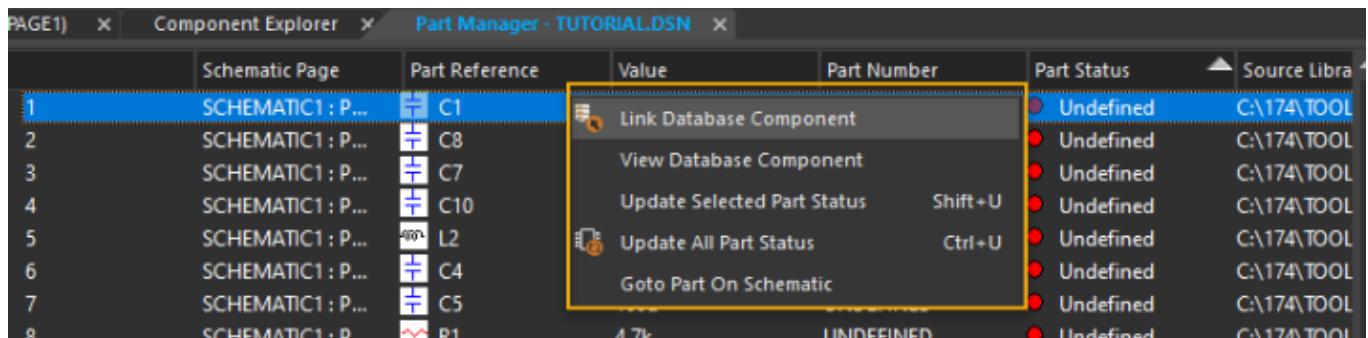
You can replace a placed schematic part and its properties with a workspace component by linking the schematic part to a matching workspace component. You can also view the placed part against the linked workspace component and sync the two if any differences are found.

Linking Database Components

To link a schematic part to a component in the component library (My Workspace), do the following steps:

1. On the schematic canvas, select a part.
2. Right-click and choose *Link Database Component*.

Alternatively, right-click a part in Part Manager and choose *Link Database Component*.



- (i) When you run this command, the component is accessed from the default workspace database.

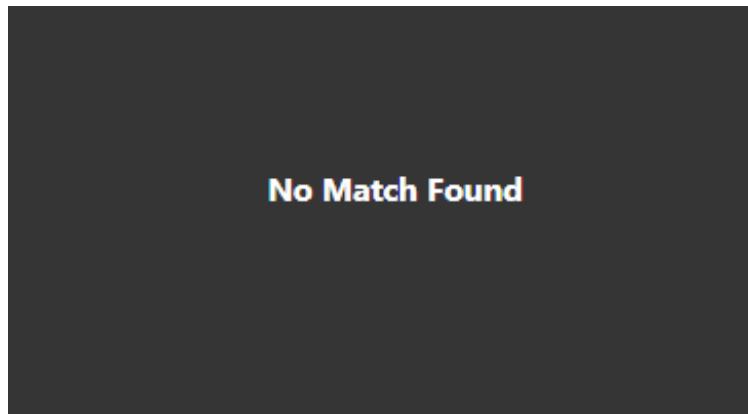
The Component Explorer tab opens and matching components, if found in the local

component library, are listed in the part browser.

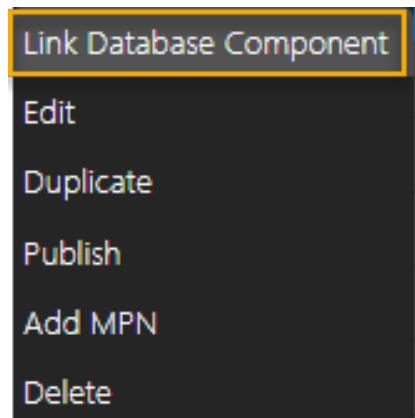
The screenshot shows a component library search results table. The columns are 'Part Number' and 'Description'. The rows list various components from a local library:

Part Number	Description
cdn-cap-005	10uF Ceramic Capacitor
cdn-cap-007	10uF polypropylene film Capacitor
cdn-ind-001	10uH Fixed Inductor
cdn-ind-005	10uH Fixed Inductor
cdn-res-010	10k Through Hole Resistor

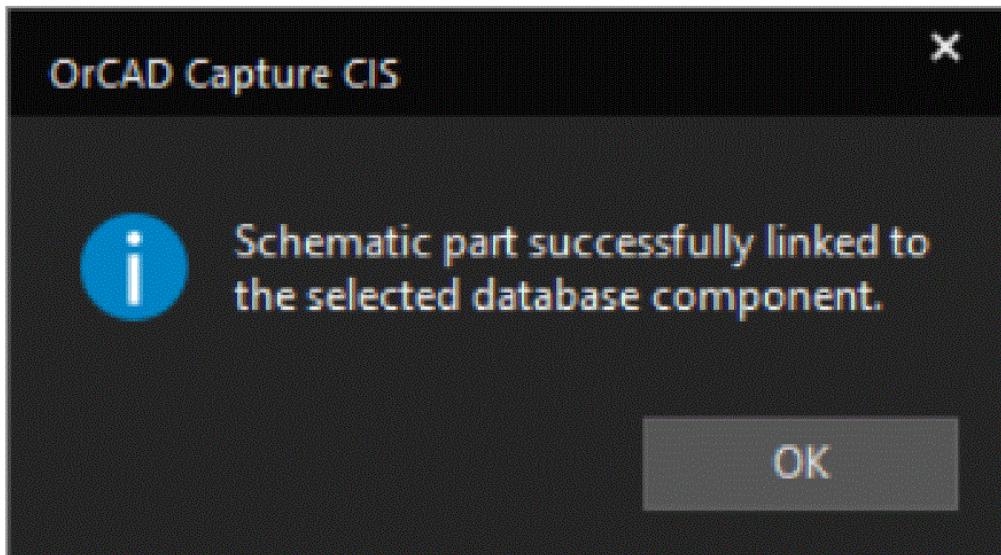
If no match is found in the local library, a No Match Found message is displayed in the middle pane.



3. Locate a suitable component to associate with the schematic part.
4. Double-click the component.
Alternatively, right-click the component and choose *Link Database Component*.



A successful linking message is displayed:



The part is replaced by the component selected in the part browser.

#	Schematic Page	Part Reference	Value	Part Number
1	SCHEMATIC1 : P...	C1	10uF	cdn-cap-005
2	SCHEMATIC1 : P...	C8	0.06u	UNDEFINED
3	SCHEMATIC1 : P...	C7	3.3u	UNDEFINED
4	SCHEMATIC1 : P...	C10	0.22u	UNDEFINED
5	SCHEMATIC1 : P...	L2	10u	UNDEFINED

5. Click *OK*.

Viewing Database Components

If any changes are made to the library component, you can view the changes and update the schematic part instance.

To view the differences between a schematic part and the library component it is linked to, do the following steps:

1. On the schematic canvas, select the part linked to the database component.
2. Right-click and choose *View Database Component*.
Alternatively, right-click a part in Part Manager and choose *View Database Component*.
The *View Database Component* dialog box displays a table comparing the property values of

the database component with the property values of the part instance on the schematic. The changes are highlighted in the table.

The screenshot shows a modal dialog titled "View Database Component-My Workspace". It contains a table with three columns: "Name", "Value In Database", and "Value On Schematic". The table lists various component properties. The rows for "VOLTAGE" and "POWER" are highlighted in red, indicating they are being updated. The "Update" button at the bottom right is also highlighted in red. The "Close" button is greyed out.

Name	Value In Database	Value On Schematic
VALUE	10k	10k
CLASS	DISCRETE	DISCRETE
TC	5ppm	5ppm
MIN_TEMPERATURE	-55C	-55C
MAX_TEMPERATURE	+150C	+150C
PACKAGE	Axial	Axial
VOLTAGE	70V	60V
POWER	10W	7W
TOL	10	10
PCB FOOTPRINT	axrc05	axrc05
PART NUMBER	cdn-res-010	cdn-res-010
PART DESCRIPTION	10k Through Hole Resistor	
HAVE SYMBOL	1	

3. To update the schematic part with the database properties, click *Update*.

The updated property values are displayed on the schematic canvas.

4. Select the part on the canvas, right-click and choose *View Database Component*.
Both the schematic part and the database component appear in sync.

The screenshot shows a modal dialog box titled "View Database Component-My Workspace". The dialog contains a table with two columns: "Name" and "Value In Database" (with a secondary column "Value On Schematic" visible). The table lists various component parameters:

Name	Value In Database	Value On Schematic
VALUE	10k	10k
CLASS	DISCRETE	DISCRETE
TC	5ppm	5ppm
MIN_TEMPERATURE	-55C	-55C
MAX_TEMPERATURE	+150C	+150C
PACKAGE	Axial	Axial
VOLTAGE	70V	70V
POWER	10W	10W
TOL	10	10
PCB FOOTPRINT	axrc05	axrc05
PART NUMBER	cdn-res-010	cdn-res-010
PART DESCRIPTION	10k Through Hole Resistor	
HAVE SYMBOL	1	

Update Close

5. Click *OK* to close the dialog box.

Related Topics

- [Link Database Parts](#)
- [View Database Parts](#)

Managing Shared Workspaces

With the OrCAD X Professional license, OrCAD X Capture provides a comprehensive part development environment where you can create shared workspaces containing work-in-progress components, designs, libraries, board files, and all the other project related files. You can manage the files in workspaces from [File Manager](#) and components from [Component Explorer](#).

- [Configuring Workspaces](#)
- [Sharing Workspaces](#)

Managing Parts

A package consists of one or more parts. Depending on the type of parts in a package, a package is classified as a Heterogeneous or Homogeneous part.

Heterogeneous and Homogeneous Parts

Parts usually correspond to physical objects—gates, chips, connectors, and so on—that come in packages. Think of these packages as physical parts and the parts you place on a schematic page as logical parts. Physical parts that comprise more than one logical part are sometimes referred to as multiple-part packages. To keep it simple, both are referred to as parts in Capture.

Logical parts in a package may have different pin assignments, graphics, and user properties. If all the logical parts in a package are identical except for the pin names and numbers, the package is homogeneous. If the logical parts in a package have different graphics, numbers of pins, or properties, the package is heterogeneous. For example, a hex inverter is homogeneous--the six inverters are identical, except for their pin numbers. A relay--that has a normally opened switch, a normally closed switch, and a coil--is heterogeneous: the three physical parts differ in graphics, number of pins, and properties.

Creating a Heterogeneous Part

To create a heterogeneous part in a library:

1. Open the library and choose *New Part* from the *Design* menu.
2. Specify the part name.
3. Set the number of parts in the package.
4. Select *Heterogeneous* in the *Package Type* group box.
5. Enter the PCB Footprint if you want to assign one to the part at this time.
6. Click the *OK* button.

Capture creates a the logical part with the reference designator U?A.

1. Draw the part body and add the pins.

2. To get to the B package, choose *Next Part* from the *View* menu, or press **CTRL+N**.

Capture displays the U?B part for you to edit.

1. Again, draw the part body and add the pins.

2. Repeat this process until all the logical parts are created.

After creating the logical parts of your heterogeneous part in the library, you need to assign a unique property to each one. For example, create a property named **PACKAGE**.

Note:

Do not use **GROUP** as the name for this property. This may cause problems while annotating your design for a PCB Editor tool, such as Allegro PCB Editor. The **GROUP** property is used in PCB Editor for a specific purpose.

To create this property, add a user property in the *Part Properties* section of the *Property Sheet* pane.

Open your schematic and place the A, B, C (and so on) logical parts of your heterogeneous parts appropriately in your design. After you place each logical part, double-click on the part to get the property editor. Edit the value of the **PACKAGE** property shown in the spreadsheet. Leave the Value of 1 on that property for each logical part of the first set you place; assign a Value of 2 to each logical part in the second set; assign a Value of 3 to each logical part in the third set, and so on. Capture uses these values to group the heterogeneous parts correctly when assigning reference designators.

When you get ready to annotate the design, you add that property name to the combined property string in the Annotate dialog box. Capture will use this property and the assigned values to annotate the parts correctly in the design. To do this, go to the project manager window, select the design name, and choose **Annotate** from the Tools menu. Select **Update entire design**; select **Unconditional reference update** (select **Incremental** if you have already partially annotated your design); and type in **{PACKAGE}** into the Combined property string box. This gives you a combined property string like **{Value}{Source Package}{PACKAGE}**. When you click **OK**, Capture then assigns the appropriate reference designators to all your parts in the design including the heterogeneous parts.

Note:

Do not change the reference designators of heterogeneous parts for a complex hierarchical design manually. In case you want to change the reference designator for a part placed in the schematic page, delete it, and add it again. This way all the occurrences will get updated correctly.

Split Parts

A split part is a multi-sectioned package. You may need to section a part for different reasons:

- Your design may include parts that have thousands of pins. Such large-sized parts may not fit in a single schematic page. To handle such parts, you can split them into multiple sections based on your specification and can place different sections in different schematic pages. This will ease designing.
- You want to partition a large part based on its functionality and use sections individually. For example, you may like to create different sections for pins with the same voltage rating.

Placing Parts on a Schematic Page

Capture ships with more than 400 part libraries. To place a part on your schematic, you will first need to search for the part, and then place the part on the schematic page.

In this section:

- [Searching for a Part](#)
- [Placing a Part](#)
- [Searching and Placing PSpice Parts](#)
- [Creating Hierarchical Blocks](#)

Searching for a Part

To search for a part you can go to the Place Part window. In this window, you can either select one or more libraries to search. This method is useful if you know the library (or group of libraries) in which the part exists. Alternatively, you can search for a part from the libraries contained in a Windows directory.

To search for a part in the selected libraries, do the following:

1. From the *Place* menu, choose *Part*.
In the Place Part window, add the libraries in which you want to search for the part.
2. To add a library, click *Add Library*.
3. In the Browse File dialog, select <the library to add>.

To add multiple libraries:

- a. Use the Ctrl + click or the Shift + click combinations.
 - b. Choose one library at a time. Use this method if the libraries to add are in different directories.
4. In the Libraries list, choose the library (or libraries) within which you would search for the part.
To select multiple libraries, use the Ctrl +click or the Shift + click combinations.
5. To use the *Filter* command, click the *Filter* button.
6. In the *Part* text box, enter <the name of the Part>. You can use the * (asterisk) or ? (question mark) wildcard characters to search. Notice that as you type, Capture auto-completes the Part name.
7. Press Enter.

To search for a part in all the libraries in a directory, do the following:

1. From the *Place* menu, choose *Part*.
In the *Place Part* window, add the libraries in which you want to search for the part.
2. Click the Plus sign to the left of *Search for Part* at the bottom of the *Place Part* window.
3. In the *Search For* text box, enter the name of the part to search.
You can use the * (asterisk) or ? (question mark) wildcard characters to search.
4. Click the *Search* button.
Capture scans the libraries in the selected directory, and lists all the parts that match the name or wildcard.
5. In the *Libraries* list, select the library containing the part you want, and click *Add*. If you need to know a part's library of origin, you can select the part in the [project manager](#), then select *Replace Cache* from the *Edit* menu. The part name and the library and path are listed in the dialog box that appears.
6. Click *Cancel* to return to the project manager.

You can identify the library of origin for multiple parts by [creating a cross reference report](#).

Placing a Part

Parts are stored in libraries. In addition, you can [create your own parts](#) in custom libraries. Some library parts have a [convert](#) as well as the normal graphical representation. Many [packages](#) contain more than one part, in which case you may need to specify which of the parts to place.

To place a part, do the following:

1. From the *Place* menu, choose the [Part command](#).
The *Place Part* window is displayed. For more information, see [Searching for a part in the libraries](#).
2. From the *Parts* list, select the part you wish to place.

 When you select a part, the PSpice symbol () is displayed for a part that can be simulated using PSpice and the layout symbol () is displayed for a part that is supported for PCB Editor flow. The symbols are displayed below the *Packaging* box.

Remember the following points, when you are placing a part:

- To place a convert version of the part, select *Convert* in the *Graphic* group box.
- If the package contains more than one part, select one of the parts from the drop-down list in the *Packaging* group box.
The part appears in the preview box.
- In case of a [heterogeneous part](#), you need to decide which part in the package you want at this time, because you will not be able to change the part in the package after the part is placed.

 Whether the part is homogeneous or heterogeneous, if you are placing multiple copies of the same part in the package, you do not need to specify the part number at the time of placing the part. Capture will auto-increment the part numbers as you keep placing the parts on the schematic.

3. Click the *Place part* button.

OR

Press [Enter](#).

An image of the part is attached to the pointer.

Press [F6](#) to change the cursor to a crosshairs to place the part at a specific location.

4. Move the part image and click to place the part.
5. Press the `ESC` key or select another tool to dismiss the part that is attached to the pointer.

The first time a part is placed, a copy of the part is created in the design cache.

When you place a part off-grid, it remains off-grid throughout any cut-and-paste and drag-and-drop operations. If you place parts so that two pins meet end to end, the pins are connected.

It is recommended that you connect the pins of the parts using a wire, and avoid placing parts in a way that two pins meet end to end. This is because, parts with direct pin-to-pin connections produce a system-generated net name to establish the connection and:

- Capture will not allow you to assign your own net name in the place of the system-generated net name.
- Searching for the system-generated net name can be difficult if you are not aware of the pin to pin connection.
- If you move the parts after creating the netlist, the system-generated net name might change. This may cause net name conflicts when you run back-annotation.

It is recommended that you do not connect a power symbol directly to a power pin. Connect the power symbol to the power pin using a wire.

You can place a part in the middle of a wire segment without redrawing the wire by placing the part over the wire such that two pins on the part connect with the wire segment. Then click over the part with the `TAB` key pressed until just the overlapping wire segment is selected. Finally, delete the wire segment.

Do not change the reference designators of heterogeneous parts for a complex hierarchical design manually. In case you want to change the reference designator for a part placed in the schematic page, delete the part and add it again. This way all the occurrences will get updated correctly.

When you place a part, make sure that the **Automatically reference placed parts** check box is selected in the **Miscellaneous** tab of the [Preferences dialog box](#). This will ensure that the part references for the newly placed part are unique.

Shortcut

Tool palette:



Uniquely Identifying Parts

1. In the project manager, select schematic folders or schematic pages, if you want to process only a portion of the design. If you want to process the entire design, leave the schematic folders or schematic pages unselected.
2. From the Tools menu, choose [Annotate](#).
The [Annotate dialog box](#) appears.
Verify that the dialog box options are set the way you want them. For example, you specify whether to update the entire design or only the schematic folders or pages selected in the project manager, whether to assign part references to all parts or to only those that have not been previously updated, or whether to return all the part references to the unassigned state (such as C? or U?A). Note that if you choose the Reset reference numbers to begin at 1 in each schematic option, it is possible that part references will be duplicated within a schematic folder that contains multiple pages.
3. Click OK.

If you copy a part into the Clipboard and then paste it onto a schematic page, Capture will automatically assign a unique reference designator to the pasted part when the following two conditions are met:

1. The Auto Reference option on the [Miscellaneous tab](#) of the Preferences dialog box is selected.

 For hierarchical PCB designs, you can choose to perform design level auto-referencing by selecting the Design Level (Only PCB Designs) option in the Miscellaneous tab of the Preferences dialog box. This option is not selected by default.
2. The pasted part has a reference designator assigned to it when it is copied to the Clipboard.

Capture assigns the reference designator, updated to the next available value (one greater than the highest value used on the schematic at that point). If the pasted part has a default reference (for example, R?) Capture does not assign a unique reference designator to it.

Shortcut

Toolbar: 

Creating Hierarchical Blocks

Topics covered in this section:

- [Attaching a schematic folder to a hierarchical block](#)
- [Creating a hierarchical block from a Verilog model](#)
- [Creating a hierarchical block from a VHDL model](#)

[Attaching a schematic folder to a hierarchical block](#)

Attaching a schematic folder to a hierarchical block

A hierarchical block is a representation of a schematic folder, which is attached to the hierarchical block. It provides vertical (downward-pointing) connection only. The hierarchical pins in a hierarchical block act as points of attachment for electrical connections between the hierarchical block and other connectivity objects in the attached schematic folder. A hierarchical block functions just like a part with an attached schematic folder.

Before you create or re-size a hierarchical block, make sure the Snap to grid option is turned on (from the schematic page editor's Options menu, choose [Preferences](#)). If the hierarchical block is on Fine grid, then hierarchical pins inside it are also on Fine grid—even if you change the Snap to grid setting before you place them—and it may be difficult to connect to these off-grid hierarchical pins. A part with an attached schematic folder functions exactly as described for hierarchical blocks, and pins on such a part function exactly as described for hierarchical pins within a hierarchical block. You can use the same attached schematic folder for either method of defining a hierarchy. The only difference between the two methods is that a part with an attached schematic folder is easier to reuse.

If you choose the [Descend Hierarchy command](#) on a non-primitive part or hierarchical block, and Capture cannot find the attached schematic folder, Capture creates a schematic folder in the active design.

 When you descend into an object that does not yet have a schematic folder or page associated with it, Capture creates the new schematic page and folder and gives it the same name as the hierarchical block. Because schematic names, schematic page names, part names, and symbol names are all limited to 31 characters, it is best to limit hierarchical block names to 31 characters.

When you place a hierarchical block, you must specify a reference. However, the reference need not be updated if the hierarchical block is primitive. For example, you could specify the reference to be "Halfadd?" when you place a hierarchical block. Then, when you run Annotate, the hierarchical block's reference is updated along with other parts.

If you attach an existing schematic folder to a hierarchical block, Capture automatically creates the

hierarchical pins that correspond with the schematic folder's hierarchical ports. If you descend hierarchy on a hierarchical block whose schematic folder doesn't yet exist, then Capture automatically creates the hierarchical ports that correspond with the hierarchical pins of the hierarchical block.

 If you attach external schematic folders or other files to hierarchical blocks in a design or parts in a library, be sure to include the attachments when you pass the design or library to a board fabrication house or to another engineer. Attached schematic folders and other files are not carried along automatically when you copy or move a part, schematic folder, or schematic page to another library, design, or schematic folder. Only the "pointers" to the attached schematic folders and files—that is, their names and the names of the designs or libraries that contain them—are carried along.

Attached files work much like their counterparts in email—they do not provide an alternative definition of the part (as do attached schematic folders).

 When you attach a schematic folder to a part or hierarchical block, you can specify a full path and filename in the Library text box. So, although you can specify a library that hasn't been saved, you should not try to descend into the attached schematic folder until the library that contains the schematic folder has been saved.

If you don't specify a full path and file name in the Library text box, Capture expects to find the attached schematic folder in the same design as the part or hierarchical block to which it is attached. If the specified schematic folder doesn't exist in either the design or library, Capture creates the schematic folder when you descend hierarchy on the part or hierarchical block. For compatibility with future versions of Windows, Capture preserves the case of the path and filename as you specify them in the Library text box.

 The [Select Entire Net command](#) is restricted to the active schematic page—it doesn't follow hierarchical blocks, hierarchical ports, or off-page connectors across schematic folders or schematic pages. For more information, see [Tracing a net](#).

Remember that nets on a schematic page are electrically connected by name, by alias, or by connection to a named hierarchical port or off-page connector.

To attach a schematic folder to a hierarchical block

1. From the *Place* menu, choose *Hierarchical Block*.
2. Enter a name for the hierarchical block in the *Reference* text box.
3. Choose *Schematic View* as the implementation type, in the *Implementation Type* list box.

4. Type the name of the schematic folder in the *Implementation Name* text box.
5. If the schematic folder is not part of the current project, specify the path to the schematic folder in the *Path and Filename* text box.
6. Click *OK*.
7. Use the cursor to draw the boundaries of the hierarchical block on the schematic page.
Capture creates the new hierarchical block and automatically places the hierarchical pins according to the ports that exist in the attached schematic.

 Hierarchical Block or Hierarchical Part referring to the same schematic design should have implementation path as empty.



- Be careful not to create **recursion** in your design. Capture cannot prevent recursion, and the **Design Rules Check command** does not report it.
- Recursion causes Capture to process infinitely as it tries to expand the design, resulting in the loss of any changes you've made to your design since it was last saved.

Creating a hierarchical block from a Verilog model

In Capture, you can create hierarchical blocks from Verilog models for inclusion on your schematic page. Creating hierarchical blocks in this manner is generally termed "bottom-up" design.

To create a hierarchical block from a Verilog model

1. Open the parent schematic page in the schematic page editor.
2. From the *Place* menu, choose **Hierarchical Block** (ALT, P, H).
3. Enter a name for the hierarchical block in the *Reference* text box.
4. Select *Verilog* as the implementation type in the *Implementation Type* drop-down list box.
5. Type the module name for the model in the *Implementation name* text box.
6. Specify the Verilog file for which you want to create a hierarchical block in the *Path and filename* text box. Make sure the file is a Verilog type file (*.V).
7. Click *OK*.
8. Use the cursor to draw the boundaries of the hierarchical block on the schematic page.

Capture creates the new hierarchical block and automatically places the hierarchical pins according to the port list specified in the module section of the Verilog file.

-  If the port names in the Verilog model have both upper and lower case characters in their identifiers, the property Vlog_Uppercase is attached to the resulting hierarchical block. For more information about Vlog_Uppercase, see the discussion in [Verilog tab](#).

At this point, the hierarchical block is defined and ready to be "wired in" to the rest of the schematic.

Creating a hierarchical block from a VHDL model

In Capture, you can create hierarchical blocks from VHDL models for inclusion on your schematic page. Creating hierarchical blocks in this manner is generally termed "bottom-up" design.

To create a hierarchical block from a VHDL model:

1. Open the parent schematic page in the schematic page editor.
 2. From the *Place* menu, choose [Hierarchical Block](#).
 3. Enter a name for the hierarchical block in the *Reference* text box.
 4. Select *VHDL* as the implementation type in the *Implementation Type* drop-down list box.
 5. Type the entity name for the model in the *Implementation name* text box.
 6. Specify the VHDL file for which you want to create a hierarchical block in the *Path and filename* text box. Make sure the file is a VHDL type file (*.VHD).
 7. Click *OK*.
 8. Use the cursor to draw the boundaries of the hierarchical block on the schematic page.
- Capture creates the new hierarchical block and automatically places the hierarchical pins according to the port list specified in the VHDL entity.

At this point, the hierarchical block is defined and ready to be "wired in" to the rest of the schematic.

Creating Parts

In Capture, you can create a part and add it to a new or existing library. The part may be a single part or a multiple-part package. It can contain graphics and IEEE symbols, which must be inside the part-body border, as well as text, which can be either inside or outside the part-body border. For any part that you create, you can also create a part convert.

Creating a part involves three processes:

1. Define the part properties in the Property Sheet pane.
2. Define the part body.
3. Place pins on the part body.

In this section:

- [Creating a Part Body](#)
- [Creating a Part Convert](#)
- [Creating a Part Alias](#)
- [Creating a Part from a Spreadsheet](#)

 You can use an existing part as a model for a new part by moving a copy of the part to a second library and then editing the copy. If you want to have the new part in the original library, rename the new part, then move it to the original library.

Creating a Part Body

When you create a new part, the part editor opens with an empty, rectangular visible part outline (the part-body border) visible. The part body border expands to accommodate the graphic elements of the part body, and pins are constrained to the part-body border.

If you want to change the size or shape of the part-body border, you can select the border and drag the selection handles until the part-body border appears as you want it.

Pins, when you place them, are constrained to the part-body border. If the edge of the part-body coincides with this border, the pins are directly attached to the part-body, but if the part-body is inside this border, you must draw a line from the pin to the part-body. You may place individual pins, or you may place an array of pins.

You define the part-body using the tools available on the tool palette. All of these tools are also available on the *Place* menu. Using the selection tool, you can select a placed object for editing.

 You can draw part bodies thicker than pins and the rest of the part by adjusting the line style for the graphic objects you want to be thicker.

When you place a pin, you can describe it completely. To place pins, you need to be in the Part view of the part editor.

If you want to place several identical pins that are not sequentially numbered on the part-body border, the Pin tool is ideal. See the *To place an array of pins* section, if you wish to create multiple

identical pins that are numbered sequentially on the part body border.

When you place a pin in one view (normal or convert), Capture places an identical pin in the other view to prevent the parts from getting out of sync. The same is true about deleting pins. Changing the name of a pin in one view doesn't cause the name to change in the other. However, if you change a pin number in one view, Capture changes the pin number in the other view so the two views stay in sync.

If the part you are creating includes a series of pins that vary only in pin number, placing a pin array is very convenient. A pin array is defined by a single set of electrical characteristics. This tool is ideal if you wish to create multiple pins with identical properties and place them so that the pin numbers and names are sequentially arranged on the part body border, this tool is ideal.

Both homogeneous and heterogeneous parts may have shared pins. A common use of shared pins is for supply (power or ground) pins, which are referred to in Capture as 'power pins'.

On heterogeneous parts, power pins can be visible on every part in the package. If the pins are visible, they must be placed on at least one part in the package, and that part must be placed in the design for the power connections to appear in the netlist. Invisible power pin types must also be in a part that is placed in the design for them to appear in a netlist.

On homogeneous parts, power pins appear on every part in the package. The pin names are filled in automatically, but you must specify the pin numbers. For the pins to be shared, verify that both the pin names and pin numbers are the same for every part in the package.



- If you place the same pin on multiple parts in a package, you can inadvertently short two nets. Use caution to avoid this problem, and always run Design Rules Check before creating a netlist.
- Pin names are shared, but pin numbers are not.

You may have comment text, in the font of your choice, on a schematic page or a part. Use the text tool to document your schematic folder or to place the logic definition for a programmable logic device.

At certain zoom scales, Capture substitutes text that is too small to appear with filled rectangles.. These placeholders are only for display—the text prints correctly.

Before you begin drawing, you may want to specify default line and fill styles because all lines and shapes you draw adopt the current line style, and closed shapes adopt the current fill style. You can use a variety of line types or fill styles for any schematic page or part.

To change the snap-to-grid option

- From the *Options* menu, choose the *Preferences* command, then choose the [Grid Reference tab](#).

You can set the option separately for the schematic page editor and the part editor.

To set a default line style

1. From the *Options* menu, choose the *Preferences* command and then choose the [Miscellaneous tab](#).
2. Click the *Line Style and Line Width* drop-down list to view the options.
Note that you can specify separate options for the [schematic page editor](#) and the [part editor](#).
3. Select one of the options and click *OK*.
Any lines or shapes you draw will have this line style.

To define a default fill

1. From the *Options* menu, choose the *Preferences* command and then choose the [Miscellaneous tab](#).
2. Click the *Fill Style* drop-down list to view the options.
Note that you can specify separate options for the [schematic page editor](#) and the [part editor](#).
3. Select one of the options and click *OK*.
Any closed shapes you draw will have this fill style.

To create the part body you use the drawing objects in the *Place* menu (or the *Draw* toolbar). These include line, polyline, rectangle, and arcs.

Notice that when you are in the part editor, most of the options on the *Place* menu are disabled.

To draw an object

1. From the *Place* menu, choose the appropriate drawing command or select the appropriate drawing command from the *Draw* toolbar.
2. Use the mouse to draw the object.
To constrain the object by the orthogonality rules, press and hold the **SHIFT** key while you draw.

To edit line style or fill style of a placed object

1. Select an object.
2. The *Basic Attribute* section appears in the *Property Sheet* pane.
3. Select another line style or fill style and save the object.

To add comment text to a schematic page

1. From the *Place* menu, choose *Text*.

The Place Text dialog box appears.

2. Enter the text.
3. Complete the dialog box selections; you can specify the font, color, or rotation.
4. Click *OK*.
A rectangle representing the text is attached to the pointer.
5. Use the mouse to move the text.
6. Click to place the text at the desired location.



- You can place multiple copies of the text. Just click at each location where you would want the text. When you are through placing text, select the selection tool or press **ESC**.
- You can create multiple lines within a text object by pressing **Ctrl+Enter** to create the new line. This is useful for creating piped PLD commands without having to place multiple lines of text. Piped SPICE commands must be placed as separately placed lines of text.
- A comment starting with `@PSpice:` is netlisted to PSpice during netlist creation, only if it is placed in the root schematic.

Examples of single-line comment and multi-line comment starting with `@PSpice:` are:

- **Single-line comment**

```
@PSpice: R1 1 0 1k will be netlisted as R1 1 0 1k
```

- **Multi-line comment**

```
@PSpice: .autoconverge ITL1=1000 ITL2=1000 ITL4=1000 RELTOL=0.05
```

```
ABSTOL=1.0E-6 VNTOL=.001 PIVTOL=1.0E-10 .TEMP 125
```

will be netlisted as

```
.autoconverge ITL1=1000 ITL2=1000 ITL4=1000 RELTOL=0.05 ABSTOL=1.0E-6
```

```
VNTOL=.001 PIVTOL=1.0E-10 .TEMP 125
```

To add comment text in part editor

To add comment text in part editor, do the following

1. From the *Place* menu, choose the *Text* command.
The Edit Comment Text dialog box appears.

2. Specify the text to place on the part page.
3. Click *OK*.
The text is immediately attached to the cursor.
4. Click where you want to place the comment text.
5. Select the selection tool or press `ESC` to complete placing the text.

Shortcut

Tool palette: 

To edit text display properties in a schematic page

1. Select the text.
2. From the *Edit* menu, choose the *Properties* command.
3. In the dialog box that appears, change the font, color, or rotation, then click *OK*.

Shortcut

Mouse: Double-click the text to edit.

To edit text display properties in part editor

1. Select the text.
2. The *Text Properties* section appears in the Property Sheet pane.
3. Modify the font, color, or justification.
4. Save the part.

To place a pin

1. From the *Place* menu, choose *Pin*. The *Place Pin* dialog box appears.
2. Edit the values as required.

Name	The name can be up to 128 characters long and may include any character. If you place multiple copies of the pin and the name ends with a numeric component, that final numeric component increments by one with each successive pin you place. Note: If you are using Capture design with PCB Editor, make sure that the pin names do not exceed 255 characters.
-------------	--

Number	The pin number can be up to 32 characters long and may include any character. If you place multiple copies of the pin and the number ends with a numeric component, the final numeric component increments by one with each successive pin you place.
Shape	Select one; the choices are CLOCK, DOT, DOT CLOCK, LINE, SHORT, SHORT CLOCK, SHORT DOT, SHORT DOT CLOCK and ZERO LENGTH. If you select a pin type of POWER, the pin shape is set to ZERO LENGTH automatically. Note: You can also specify a user-defined pin shape if the pin shape is available in the CAPSYM.OLB library.
Type	Select one; the choices are 3STATE, BIDIRECTIONAL, INPUT, OPEN COLLECTOR, OPEN EMITTER, OUTPUT, PASSIVE, and POWER. The Design Rules Check tool uses pin type to check electrical rules.
Width	Select Scalar or Bus. If you choose Bus, the pin name must be of the form basename[m..n] where m..n specifies a range of decimal integers representing the number of bus members. For more information, see Naming Conventions .
Visibility	If you are placing a power pin, you can select the <i>Pin Visible</i> check box to cause the pin to cancel the pin's global attribute. This is useful if you want to create an isolated power net. If a power pin is visible, it must be connected to a wire. For more information see About power and ground pins or Isolating power or ground . Some netlist formats do not accept certain characters in pin names. See the description for the netlist format you want to use.

3. Define the pin and click *OK*.

The pin appears attached at the periphery of the part.

4. Use the mouse to move the pin to its intended location and click to place it.

The pin appears in the selection color until you move the pointer.

5. If you want to place additional pins, repeat step 1-4.

As you place successive pins, any final numeric component of the pin name or pin number increases by one.

6. If you need to edit pin properties,

a. Select the pin.

b. The *Pin Properties* section appears in the Property Sheet pane.

- c. Modify the properties and save the part.
7. When the pins are placed, select the selection tool, or press **ESC** to dismiss the pin tool.
8. If the part body does not coincide with the part-body border, draw a line from the pin's connection point to the part body. You may need to temporarily turn off the *Pointer snap to grid* option (*Options – Preferences – Grid Display*) while you draw the line.

If you want an overbar over a signal name, follow each character in the name with a backslash (\).

You can edit every pin in the package using the *Edit All Pins* dialog box. You can also use the *Pin Array* command for placing large numbers of pins even though the properties or pin numbers vary.

Shortcut

Tool palette: 

To place an array of pins

1. From the *Place* menu, choose *Pin Array*.
The *Place Pin Array* dialog box appears.
2. Modify the values as required.

Starting Name	The starting name can be up to 128 characters long and may include any character. The leftmost or upper pin of the array is assigned the starting name. If the starting name ends with a numeric component, that component increments by the increment amount from top to bottom and from left to right.
Starting Number	The starting number can be up to 128 characters long and may include any character. The leftmost or upper pin of the array is assigned the starting number. If the final component of the starting number is numeric, the pin numbers change by the increment amount from top to bottom and from left to right.
Number of Pins	An integer
Pin Spacing	A positive value. The distance between the pins is measured in grid units.
Shape	Select one; the choices are CLOCK, DOT, DOT CLOCK, LINE, SHORT, ZERO LENGTH. If you select a pin type of POWER, the pin shape automatically is set to ZERO LENGTH.

Type	Select one; the choices are 3STATE, BIDIRECTIONAL, INPUT, OPEN COLLECTOR, OPEN EMITTER, OUTPUT, PASSIVE, POWER. Pin type is used by the Design Rules Check tool to check electrical rules. Note: Some netlist formats do not accept certain characters in pin names. See the description for the netlist format you want to use.
Pin Visible	Specify the pin visibility when the part is placed on the schematic page. Only power pins can be set to not visible.
Pin# Increment for Next Pin	Specify the increment for the next pin number in the pin array.
Pin# Increment for Next Section	Specify the increment between pin numbers for the next section. This is valid only for homogeneous parts.

3. When you have completely defined the array, click *OK*. The array is attached to the periphery of the part; the part body border automatically increases in size if necessary.
4. Use the mouse to move the array to its intended location and click to place it. The array appears in the selection color until you move the pointer.
5. Select the selection tool to dismiss the pin tool.
6. If the part body does not coincide with the part body border, draw a line from the pins' connection points to the part body. You may need to temporarily turn off the *Pointer snap to grid* option (*Options – Preferences – Grid Display*) while you draw the lines.

Shortcut



Tool palette:

To connect a pin to a non-rectangular part body

1. Place the pin on the part body border.
2. From the *Options* menu, choose *Preferences*, then choose the *Grid Display* tab.
3. In the *Part and Symbol Editor* group box, clear the *Pointer snap to grid* option, then click *OK*.
4. Draw a line between the pin and the part body.
5. If the line does not look like the pin, edit the line's style and width.

6. From the *Options* menu, choose *Preferences*, then choose the *Grid Display* tab.
7. In the *Part and Symbol Editor* group box, select the *Pointer snap to grid* option, then click *OK*.

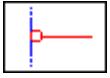
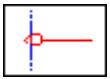
 The size of a part or a symbol is limited to 32 by 32 inches.

Pin Shapes

When you place pins on a part body, you can specify the shapes of the pins. You can use the predefined System-Defined Pin shapes or you can create your own Pin Shape.

System-Defined Pin Shapes

Capture provides a list of system-defined pin shapes that you can use when you create a new part or edit the pins shapes on an existing part.

	Clock Clock symbol
	Dot Inversion bubble
	Dot-Clock Clock symbol with inversion bubble.
	Line Normal pin with lead three grid units in length.
	Short Normal pin with lead one grid unit in length.
	Short Clock Clock symbol with lead one grid unit in length.
	Short Dot Inversion bubble with lead one grid unit in length.
	Short Dot-Clock Clock symbol with inversion bubble with lead one grid unit in length.
	Zero length Normal pin with lead zero grid units in length.

User-Defined Pin Shapes

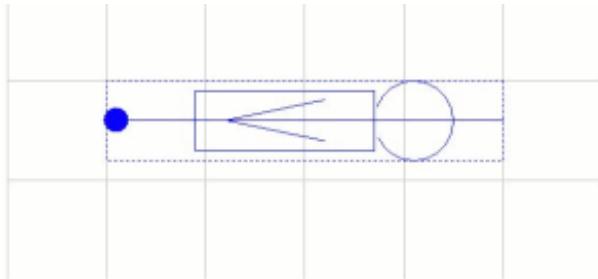
You can create your own pin shapes in Capture. You can then use these pin shapes on new or existing parts.

To create a pin shape

1. Open the CAPSYM.OLB library from the installation path: <Installation Directory>\tools\capture\library\capsym.olb.
(see [Opening a library](#))
2. Select the library (capsym.olb) in the project manager and choose *Design – New Symbol*.
OR
Right-click the library (.olb) and choose *New Symbol* from the shortcut menu.
3. Enter a name for the new pin shape.
A symbol name length cannot exceed 31 characters.
4. Choose the *Pin Shape* option in the *Symbol* type group and click *OK*.
The *Part Editor* page opens with an empty rectangle defining the boundary of the pin shape.
5. Draw the pin shape using the available shapes on the *Draw* toolbar.
The new pin shape is now available in the selected library.
You can now use this pin shape by placing it on a part (see *To place a pin* section).

When **creating user-defined pin shapes**:

- You can choose Line, Arc, Polyline, Bezier curve, Rectangle, Ellipse, Elliptical arc shapes to create a user-defined pin shape around a dot connection point.



- The bounding box of the pin shape determines the length of the pin. If the bounding box does not completely enclose the graphics then you may see your pin offset from the boundary of the symbol. For a correct adjustment, ensure the bounding box completely encloses the graphics.
- The Capture Part and Schematic Page editors require a dot connection point to be on the grid so for the pin start point to touch the body of the symbol, you must keep the end points on the grid.

- You should not draw objects enclosing the dot connection point. That may not show the pin shape correctly on the Capture Part and Schematic Page editor.
- IEEE symbols and pictures are not supported in user-defined pin shapes.

When **adding user-defined pin shapes** to parts in your designs:

- To place a user-defined pin shape on a part, the pin shape must be available in the CAPSYM.OLB library in the path %CDSROOT%\tools\capture\library\capsym.olb. If you create your pin shape in a user-defined library, you need to copy that symbol into the CAPSYM.OLB library. Capture reads the user-defined pin shapes from this location and populates the pin shape names in the *Shape* cell of the *Pin Properties* dialog in Part Editor.
- If you place a new part in a design that has pin shapes assigned to pins, Capture will search for the pin shapes in the CAPSYM.OLB library. When the part is placed on the design and the pin shapes are found in the CAPSYM.OLB library, the pin shapes are cached to the design cache. Once you save the design, the pin-shapes will be read from the design cache.
- If a pin shape is not available in the CAPSYM.OLB library, the default pin shape, which is a line, will display on the part.
- If you downgrade a design containing user-defined pin shapes from any Capture version greater than 16.2 to 16.2 or any previous version, the user-defined pin shapes will be removed from the design cache. You can still open the design in 16.2 but the pins will show with default pin shape, line. However, if you reopen this 16.2 design in any version greater than 16.2, Capture will search the CAPSYM.OLB library for the user-defined pin shapes used in the design. The pin shapes found in the library will display on the schematic. For pin shapes not found in the library, the default, line, and shape, will display.
- If you run the Replace cache command on a user-defined pin shape, say, A with B in the design, all instances of the user-defined pin shape A will be replaced with B in the design, and the Pin Shape property on the pins will be updated to the new pin shape value. This property is an instance override. At any time, if you want to revert to the library-level pin shape value, you can use the Delete property in the Property editor and it will delete the instance override and this will be reflected on the schematic. However, if you do an Edit part, then it will still show the part level user-defined pin shape and not the instance override that exists in the schematic.

Pin Types

3 State	A 3-state pin has three possible states: low, high, and high impedance. When it is in its high impedance state, a 3-state pin looks like an open circuit. For example, the 74LS373 latch has 3-state pins.
Bidirectional	A bidirectional pin is either an input or an output pin. For example, pin 2 on the 74LS245 bus transceiver is a bidirectional pin. The value at pin 1 (an input) determines the active type of pin 2, as well as others.
Input	An input pin is one to which you apply a signal. For example, pins 1 and 2 on the 74LS00 NAND gate are input pins.
Open Collector	An open collector gate omits the collector pull-up. Use an open collector to make "wired-OR" connections between the collectors of several gates and to connect with a single pull-up resistor. For example, pin 1 on the 74LS01 NAND gate is an open collector gate.
Open Emitter	An open emitter gate omits the emitter pull-down. The proper resistance is added externally. ECL logic uses an open emitter gate and is analogous to an open collector gate. For example, the MC10100 has an open emitter gate.
Output	An output pin is one to which the part applies a signal. For example, pin 3 on the 74LS00 NAND gate is an output.
Passive	A passive pin is typically connected to a passive device. A passive device does not have a source of energy. For example, a resistor lead is a passive pin.
Power	A power pin expects either a supply voltage or ground. For example, on the 74LS00 NAND gate, pin 14 is VCC and pin 7 is GND. It is not a good idea to use overbars above power pin names; if you do, any netlists that you create will have invalid power pin names. Power pins are invisible.

Creating a Part Convert

You can store a part convert with a library part, then place either the normal view of the part or its convert.

To add a convert while you are creating a part

1. From the *Design* menu of the library's project manager window, choose *New Part*. The *New Part Properties* dialog box appears.
2. Select the *Create Convert View* check box.
3. Modify the remaining properties and click *OK*.

4. The *Convert* tab appears next to the *Normal* tab in the part editor.

Creating a Part Alias

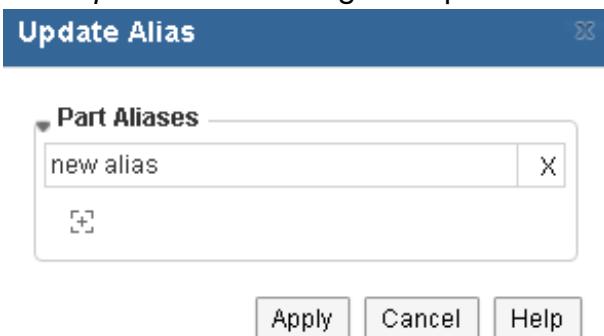
A part may come with several speed ratings or be made by several manufacturers. If all the variations have a common graphic and PCB footprint, you may not have the time and space to create and store a different library part for each variation. Instead, create a single library part and assign it multiple aliases.

To create a part alias

1. In the project manager, select the design (.OLB) file.
2. From the *Design* menu, choose *New Part*.
The *New Part Properties* dialog box opens.
3. Click the *Part Aliases* button.
The *Part Aliases* dialog box opens.
4. Click the *New* button.
The *New Aliases* dialog box opens.
5. Specify the name of the alias and click *OK*.
6. Click *OK*, as needed, to dismiss the remaining open dialog boxes.

To add or update a part alias

1. Select a part.
2. In the *Package Properties* section of the Property Sheet pane, click the *Update* button next to the *Part Aliases* field.
The *Update Alias* dialog box opens.



3. Click the plus icon.
A new row appears.
4. Specify a new alias and click the green check mark button to add this alias.
5. Click the *Apply* button.

Creating a Part from a Spreadsheet

You can use the New Part Creation Spreadsheet dialog box to create new parts (multi-section/single-section). This dialog box has an interface similar to a spreadsheet. It allows you to paste contents copied from a part data sheet to the spreadsheet.

Each row in *New Part Creation Spreadsheet* corresponds to a pin while each column corresponds to properties associated with the pin. The property names are listed as the column header.

 Before you copy and paste part information (*pin number*, *pin name*, *pin type*, and so on) from a data sheet, ensure that you arrange the part information in the same column header format/sequence as it appears in the New Part Creation Spreadsheet.

To create a new part from spreadsheet

1. Select a library (.OLB) file that will contain the new part in the project manager.
 2. Select the *Design* menu and choose the [New Part from Spreadsheet](#) command.
OR
Right-click the library file and select *New Part From Spreadsheet* from the pop-up menu.
- The [New Part Creation Spreadsheet](#) is displayed.
3. Specify a name for the new part in the *Part Name* text field.
 4. If you want to create a multi-section part, specify the number of sections you want to have in your new part in the *No. of Sections* text field. The *New Part Creation Spreadsheet* creates single-section parts, by default.

 The *Section* property column changes to a list box displaying the number of sections you specified in the *No. of Sections* text field.

 **Number of sections cannot be less than one.**

5. Specify a part reference prefix for the part in the *Part Ref Prefix* text box.

6. Select *Numeric* or *Alphabetic* in the *Part Numbering* group box.

If you select *Alphabetic*, an alphabet (between A to Z) will be added as a suffix to the current part reference for each of the new parts. If you select *Numeric*, a number (between 1 and 1024) will be added as a suffix to the current part reference for each of the new parts.

The *Section* property column changes based on your selection in the *Part Numbering* group. For example, if *Alphabetic* is selected, the Section property column displays ^A.

7. Specify a pin number in the *Number* property column.

To sort on any property, double-click its name in the column header.

8. Specify a name for the pin in the *Name* property column.

9. Select the type of pin from the *Type* property drop-down list.

You can select the *Type* cells for multiple pins simultaneously using the Shift+down arrow keys and then enter the pin type. The selected *Type* cells get populated with the pin type of your choice.

Alternatively, you can:

- a. Select the *Type* cells for multiple pins simultaneously using the Shift+click, then press the Ctrl key, and then select a pin type of your choice from the list box.

The selected Type cells get populated with the pin type of your choice.

- b. Click the first cell of the range, and then drag to the last cell, and then enter the pin type of your choice.

The selected Type cells get populated with the pin type of your choice.

(You can use these methods to make selections in the *Shape*, *Position*, and *Section* property drop-down lists also).

10. Select a shape for the pin from the *Shape* property drop-down list.

You can hide or unhide a property column in the *New Part Creation Spreadsheet*. To do this, right-click the property column header you want to hide and select *Hide* from the pop-up menu. The selected property column will now not appear. To unhide a property column, right-click the property column header next on the right-hand side of the hidden property column and select *Unhide* from the pop-up menu. The hidden property column appears in the [New Part Creation Spreadsheet](#).

Alternatively, you can unhide a property column by:

- Double-clicking the column handle () of the property column header.
- Dragging the column handle of the property column header.

(only the last two methods can be used to unhide a property column, which is the last column in the *New Part Creation Spreadsheet*)

You can change the order in which the property columns and rows appear in the *New Part Creation Spreadsheet*. To do this, select the property column/row header you want to move and drag and drop it to the location where you want it in the New Part Creation Spreadsheet.

11. Specify a value for a swappable (input) pin of the part in *PinGroup* text field.
12. Select the position where you want this pin to appear in the part for the *Position* property drop-down list.
13. Select a section number you want to associate with the pin from the *Section* property drop-down list.
You can select Section cells for multiple pins simultaneously using the Shift+down arrow keys and enter the section number. Alternatively, you can:
 - Select the Section cells for multiple pins simultaneously using the Shift+click, then press the Ctrl key, and then select a section number of your choice from the list box. The selected Section cells get populated with the section number of your choice.
 - Click the first cell of the range, and then drag to the last cell, and then enter the section number of your choice. The selected Section cells get populated with the section number of your choice.

You can select alternate Section cells for multiple pins simultaneously using the Ctrl+Left mouse button click and enter the section number.

 The pins will be added in the same order in the new part as the order in which the pins appear in the spreadsheet.

14. Click the *Save* button to save the new part.

If any warnings are generated during the save operation, a message appears asking you whether you want to view the warnings.

To view the warnings, click the *View Warnings* button. The *New Part Creation Spreadsheet* expands and displays a grid showing warnings messages. If you select the *Continue* button, the part is saved as is.

 Click the *Hide Warnings* button to hide the warning messages or the *Show Warnings* button to display the warning messages again.

 If the new part you created contains one part per package then the part created would be homogeneous otherwise the part will be heterogeneous.

To add a pin

1. In the *New Part Creation Spreadsheet* dialog box, click the *Add Pins* button.
The *Add Pins* dialog box appears.
2. Specify the number of pins you want to add in the *Number of Pins* text field.
3. Click *OK*.
The desired number of blank rows gets added at the end of the current row set in the *New Part Creation Spreadsheet* dialog box.

To delete a pin

1. Select a row in the *New Part Creation Spreadsheet* dialog box.
2. Click the *Delete Pins* button.
A message appears asking you to confirm the deletion.
3. Click *OK* to confirm deletion.
The selected row is deleted from the *New Part Creation Spreadsheet* dialog box.

Editing and Renaming a Part

If you have a library part that is nearly perfect, you can tailor the original part so that it suits your project. You can edit the part properties and you can change its graphical representation or its pins.

To edit a library part

1. Open the library containing the part.
2. In the project manager, double-click the part.
The part editor opens with the part displayed.
3. Make changes to the part body definition (such as graphics, text, and images) using the Property Sheet pane.
4. To make any changes to pins, select the required pin, and modify its properties using the Property Sheet pane.
You can move pins after you select them. You can also move pin names or pin number text when you are editing a library part.
5. From the *File* menu, choose *Save*.

To edit a part instance on a schematic page

1. Select the part in the schematic page editor.
2. From the *Edit* menu, choose the [Part command](#).
The part editor opens with the selected part displayed.
3. Edit the part as required.
4. From the *File* menu, choose the *Close* command.
The *Save Part Instance* dialog displays:
Update All replaces the old part in the design cache with the newly edited part and breaks the link with the original library.
Update Current creates a new part in the design cache. The new part has no link to the original library.

The *Part Editor* window closes and the updated part appears in the schematic page editor.

The edited part does not exist in a library, so the only way to place a copy of it is to use the *Copy* and *Paste* commands on the schematic page editor *Edit* menu.

The edited part has no link with the original library, so it is not affected by the *Update Cache* command. To restore its link with the original library, choose the *Replace Cache* command from the project manager *Design* menu. For more information, see [Replacing and Updating Cache](#).

When you open the [part editor](#) from the [schematic page editor](#), the part you are editing cannot be selected on the [schematic page](#). After you close the part editor window, the part can be selected.

You can move pin names and pin number text when you are editing a part instance on a schematic page. For more information, see the *Moving pin name and pin number text* section.

When you edit a part's graphic representation on a schematic page, you break the connection between the part and the [library](#); if you want to reverse your edits, you use the [Replace Cache command](#) of the *Design* menu.

Moving pin name and pin number

You can move pin names and pin numbers only when you are editing a part instance on a schematic page.

- To move a pin name or pin number, select the pin name or number text and drag it to the desired location.
- To reset a pin name movement, select the pin name text you had moved and from the *Edit* menu, choose *Reset Location*, then *Pin Name*.
- To reset a pin number movement, select the pin number text you had moved and from the *Edit* menu, choose *Reset Location*, then *Pin Number*.

To rename a part

1. In Project Manager, select the part.
2. From the *Design* menu, choose *Rename*.
The Rename dialog box appears.
3. Specify the new name and click *OK*.

Part Properties

When you need to edit properties for several parts, you can save time by using the spreadsheet editor, the *Update Properties* command or the [Export Properties command](#).

After you have added properties to a part on a schematic page, its properties no longer match the properties of the same part residing in the library. This part is unique in that it has properties assigned specifically to it that are not inherited from the library part definition.

If you add a user-defined property to one part in a homogeneous multiple-part package, all parts in the package inherit the property and its value. If you add a user-defined property to one part in a heterogeneous multiple-part package, the other parts in the package are not affected.

You can also edit properties on part packages, in which case the changes appear on every part in the package, and on every part instance.

When you are editing the properties of a library part, you use the *Property Sheet* pane. You can add or remove user-defined properties or change the values for the following properties:

- Part Value
- Part Reference
- PCB Footprint
- Power Pin Visibility
- User Properties

In this section:

- [Assigning Properties to a Part](#)
- [Renaming and Deleting Part Properties](#)
- [Update Properties using an Update File](#)

Assigning Properties to a Part

Assigning a reference designator

When you place parts on a [schematic page](#), all parts of the same type are assigned the same part reference. For example, C? is assigned to all capacitors. Regardless of the ultimate purpose of your design, each part needs a unique identifier. You can assign part references by editing individual parts in the [part editor](#), or, for PCB designs, by creating a swap file or packages for swapping to use with the Back Annotate tool. For uniquely identifying parts, it is more convenient to use the Annotate command on the Tools menu.

The Annotate tool assigns unique alphanumeric part references. For PCB designs, it assigns individual parts to a package and assigns unique pin numbers to each part in a multiple-part package. References are assigned in order from top to bottom and left to right; parts located at the top of the page have the lowest numerical designation. If two parts share a vertical coordinate, the part further to the left has the lower numerical designation. If you add parts after you have assigned part references, you can easily [remove part reference assignment](#) and run [Annotate](#) again.

In general, you use Annotate after you have placed all parts and before you use other Capture tools. See [Design Annotation](#) for an overview of the design processing tools.

You can update references incrementally, so that previously assigned part references are not changed, or you can update unconditionally.

Capture automatically updates either instances or occurrences depending upon the type of design you are working with. In general, you should update instances for FPGA and PSpice projects, and update occurrences for PCB and Schematic projects.

 If you want to preserve a reference designator during annotation, choose *User Assigned Flag - Set* from the pop-up menu of the part on the schematic page. You can also open the part in *Properties Editor* and from the pop-up menu of *Reference*, choose *User Assigned Flag - Set*. The User Assigned Flag is set by the tool if the reference designator is changed using the *Property Editor* or *Schematic Editor* or through *Backannotation*.

Note that if the property does not exist on the occurrence, the *User Assigned Flag* is set for the instance and thus will be used for all occurrences of that part unless the property is explicitly overridden on the occurrence. If you want to set the *User Assigned Flag* for the occurrence value, set the property on the occurrence from the property editor and then set the flag.

Assigning other properties

You can assign other properties to your library parts. See [Defining Properties](#) for more information.

To globally change the visibility of pin names or pin numbers

1. In part editor, select a part.

The *Part Properties* section in the Property Sheet pane appears.

2. Select the *Pin Name Visible* property to control pin name visibility.
3. Select the *Pin Number Visible* property to control pin number visibility.
4. Save the part to apply the changes.

Property tables

Following are some of the properties used in schematic to layout flow. You can assign any of these properties to a part.

Part Property Name	Example Value	Description
COMPFIXED	YES	If the value is YES, the part (such as an edge connector) is permanently fixed to the board.
COMPGROUP	2	An integer value that assigns the part to a group for placement. The value must be numeric, and between 0 and 100.
COMPKEY	YES	Used to designate a component as the key component in a given group. The key component is placed first, with all the other components in the group placed in proximity to it.
COMPLOC	[1000, 1000]	Part location on the board as X and Y coordinates. Use the following format [X, Y], where X and Y represent the coordinates. Both must be integers in mils or microns.
COMPLOCKED	YES	If the value is YES, the part is temporarily locked in position.
COMPROT	270.00	Part rotation in degrees and minutes counterclockwise from the orientation defined in the Layout library. Use a period (.) to separate degrees and minutes.
COMPSIDE	BOT	Determines which side of a board a part will reside on, TOP or BOT.
FOOTPRINT	DIP24	An explicit definition of the footprint name to attach to the component.

FPLIST	DIP24\400	Comma-delimited list of alternate footprints to attach to components, to ease switching between footprints.
GATEGROUP	1	Identifies gate swapping restrictions within a component. To be swapped, two gates must belong to the same gate group.
PARTNUM	489746	A customer part number that is generally unique for each customer and identifies the exact part, including the manufacturer and case type.
PARTSHAPE	74LS04	A generic part number (such as 74LS04 or CK05) that represents a certain part throughout the industry, but may not identify the manufacturer or case type. If no footprint is defined, or the correct footprint is not found, PARTSHAPE value is compared to the data in SYSTEM.PRT (in ORCADWIN/LAYOUT/DATA) and the footprint listed in SYSTEM.PRT is used.
POWERPIN	YES	Defines non-wired pins (such as unusual voltages) as belonging to a particular net. POWERPIN is typically used to override the standard GND or VCC attachments to particular pins of an IC.
CONNWIDTH	10	Sets the track width, leaving MINWIDTH and MAXWIDTH at their defaults.
HIGHLIGHT	YES	If the value is YES, the net is highlighted.
MAXWIDTH	12	Sets the maximum track width.
MINWIDTH	8	Sets the minimum track width.
NETGROUP	2	Identifies grouped nets. Use this to select or color nets as a group in Layout (for editing or routing).
NETWEIGHT	60	Integer between 1 and 100 assigning relative priority to the net. The default value is 50.
PLAINLAYERS	GND	Assigns a net to a plane layer. Values are GND and POWER.

RECONNTYPE	ECL	Specifies the reconnect rules for each type of reconnect. Values are STD, HORZ, VERT, NONE, or ECL.
ROUTELAYERS	GND,PWR	Restricts the layers on which this net can route.
SPACINGBYLAYER	TOP=13,BOT=8	Net spacing for one or more layers.
TESTPOINT	YES	If the value is YES, a test point is automatically assigned to the net.
VIAPERTNET	VIA1	Via types allowed for net.
WIDTH	12	Track width value assigned to the MINWIDTH, MAXWIDTH, and CONNWIDHT properties unless overridden.
WIDTHBYLAYER	TOP=6,BOT=12	Net width for one or more layers.

Renaming and Deleting Part Properties

Capture allows you to rename and delete a [part property](#) in every part [instance](#) that contains the property for an entire design.

To rename a part property

1. In the project manager, select the design file or the schematic page.
2. Select the [Rename Part Property command](#) from the *Edit* menu.

The [Rename Part Property](#) dialog box is displayed.
3. In the *Find User Property* text field, specify the current property name.
4. In the *Replace with User Property* text field, specify the new property name.
5. Click *OK*.

To delete a part property

1. In the project manager, select the design file or the schematic page.
2. Select the [Delete Part Property command](#) from the *Edit* menu.

The [Delete Part Property](#) dialog box appears.
3. In the *Property Name* text box, specify the property name.

4. Click *OK*.

Update Properties using an Update File

If you need specific information, like stock number or supplier, on your parts, you can add a property and set the property value in the library. Later, when you place the part in a project, the information is present; there is no need to add information to the placed part.

You can edit parts individually in the part editor, but when you wish to update properties of a number of parts, the Update Properties tool is very convenient. You can use the Update Properties tool to edit any properties except part value and part reference.

Before you run the Update Properties tool, you create an update file as described in [Update File](#). To identify the parts you wish to update, you specify an identifying property or combination of properties called a combined property string. For each combined property string you use, you must create a separate update file and run the Update Properties tool.

To update part properties

1. In the Project manager, select schematic folders or schematic pages if you want to process only a portion of the design.

If you want to process the entire design, leave the schematic folders or schematic pages unselected.

2. Choose *Tools - Update Properties* command.

The Update Properties dialog box appears. Verify that the dialog box options are set as required.

For example, you specify, among other things, whether you are updating parts or nets, whether you want to overwrite existing property values, and the name and location for the update file.

Unless you select the button to unconditionally update the property, properties that currently hold a property value are not updated.

3. In the *Property Update File* box, enter the name and location of the update file.

OR

Use the *Browse* button to navigate to the file.

4. Click *OK*.

If you edit a library provided by OrCAD, it is important that you assign a new library name so that your changes are not overwritten when you upgrade or update your software.

Capture preserves the case of part names and net names, but ignores the case when comparing names for electrical connection. That means you may use upper-case or lower-case letters as you wish and you need not remember the case.

Capture report files are text files and can be opened in any text editor. You may want to use the tab alignment capability of your word processor to line up reports correctly. Spreadsheets will automatically align the columns of Capture-generated report files.

Replacing and Updating Cache

When you place the first [instance property](#) of a part in a project, a copy of the part is created in the design cache. The design cache stores a copy of every part used in the design. (You can think of it as an embedded library.) All instances of the part normally refer to this copy in the design cache. A cache part also retains a link to the [library](#) part on which it is based, so you can update or replace all of the parts in the design cache to synchronize them with the parts in the libraries. The [project manager](#) updates the display of the design cache every time you open the cache. Just click on the design cache icon to close or open the design cache.

Using the Replace Cache and Update Cache commands

When you use the [Update Cache command](#) or the [Replace Cache command](#) with the option to preserve schematic part properties, the part retains all instance and occurrence properties. This means you will not lose any changes made to pin properties after the part was placed, including those made by the *Back Annotate* or the *Annotate* tools.

The *Replace Cache* and *Update Cache* commands are quite similar. However, there are a few significant differences between the two commands.

You can modify a part's link to the library (part name, path, and library) with *Replace Cache*, but not with *Update Cache*. *Update Cache* only brings in new data when the path has not changed and part is modified in the library after placing it on schematic.

Another difference is that if the path and library names do not change, *Replace Cache* reloads the part definition into the design. However, if *Update Cache* finds that the part name and the library names are the same, it does not bring in part changes.

Replacing parts in a design

If you need to replace a part in your project, you could open the schematic page editor to find and delete each instance of the part, then place the replacement part. If your design includes many instances of this part, you can more easily achieve the same end with the [Replace Cache command](#).

Restoring parts in a design

When you delete a part, you also delete all of its properties. When you use the *Replace Cache* command to restore the part, the properties of the part instance are attached to the replacement part, but the pin properties are not restored.

You can also use the *Replace Cache* command if you want to undo edits to a part on a schematic page and restore the part's link to its library.

You would also use the *Replace Cache* command after you edit a part on a schematic page if you

want to undo the edits and restore the part's link to its [library](#).

When you delete a part, all of its [properties](#) are also deleted. When you use the *Replace Cache* command, properties of the part instance are attached to the replacement part; however, the pin properties are lost.

 When you use the *Update Cache* command or the *Replace Cache* command with the option to preserve schematic part properties, all instance and occurrence properties of the schematic part are retained. This means you will not lose any changes made to pin properties after the part was placed, including those made by the *Back Annotate* or the *Annotate* tools.

To replace a part throughout a design

1. From the design cache, select the part you want to replace.
2. Choose *Design – Replace Cache*.
3. In the dialog box that appears, enter the name of the replacement part and the library that contains it.
4. Select the *Replace schematic part properties* action if you want to completely replace the part and its properties. Otherwise, use the default action to preserve schematic part properties.
5. Click *OK*. When the project manager appears again, double-click on the design cache to verify that the replacement part is listed instead of the original part.
If you need to know a part's library of origin, you can select the part in the project manager, then select *Replace Cache* from the *Design* menu. The part name and the library and path are listed in the dialog box that appears.
6. Click *Cancel* to return to the project manager.

You can discover the library of origin for multiple parts by [Creating a Cross Reference Report](#).

Following table provides an overview on Update Cache and Replace Cache commands:

	Update Cache	Replace Cache command

		Preserve schematic properties	Replace schematic properties without preserve Refdes	Replace schematic properties with preserve Refdes
Part (Symbol) Graphics and Pins	Updated	Updated	Updated	Updated
Part's library level property values	Updated if property value is not edited on schematic instance. Preserved if value is edited on schematic.	Updated if property value is not edited on schematic instance. Preserved if value is edited on schematic.	Updated	Updated
Part's schematic level properties values	Preserved	Preserved	Updated	Updated
Part's reference designator reference values	Preserved	Preserved	Updated	Preserved
Pin properties values	Updated if property value is not edited on schematic instance. Preserved if value is edited on schematic.	Updated if property value is not edited on schematic instance. Preserved if value is edited on schematic.	Updated	Updated

Package Level properties values (such as PCB Footprint)	Preserved	Preserved	Updated	Updated
Display location settings & coordinates for pin properties	Updated	Updated	Updated	Updated
Display location settings & coordinates for part properties	Preserved	Preserved	Updated	Updated

 When you use the Update Cache command or the Replace Cache command with the preserve schematic properties option on a part, same actions are performed.

Part Packages

Parts are the basic building blocks of a design. For PCB designs, part may represent one or more physical components; or it may represent a function, a simulation model, or a text description for use by an external application. The part's behavior is described somehow, whether by a SPICE model, an attached schematic folder, HDL statements, or other means.

Parts in PCB designs usually correspond to physical objects—gates, chips, connectors, and so on—that come in packages of one or more parts. "Multiple-part packages" are physical objects that comprise more than one part.

Each logical part has graphics, pins, and properties that describe it. As you place the parts in a package to suit your design requirements, Capture maintains the identity of the part package for back annotation, netlisting, bills of materials, and processes that require it. Parts inherit this information from the package.

You specify packaging information when you create a part. You can also change it in the part editor (using the *Package Properties* section in the Property Sheet pane).

The parts in a package may have different pin assignments, graphics, and user properties. If all the parts in a package are identical except for the pin names and numbers, the package is homogeneous. If the parts in a package have different graphics, numbers of pins, or properties, the package is heterogeneous.

For example, a hex inverter is homogeneous: the six inverters are identical, except for their pin numbers. A relay, which has a normally opened switch, a normally closed switch, and a coil, is heterogeneous: the three physical parts differ in graphics, number of pins, and properties.

When you place a part on a schematic page, you actually create an instance of the part. A part instance is like a "snapshot" of the part in the library; that is, it inherits all the properties of the library part. Once a part instance is on the schematic page, you can edit the properties of that instance without changing the properties of any other instance. The instance values of those properties supercede the values of any identical properties that exist on the library part. For more information, see [Instances and occurrences](#).

When using multiple-part packages in your design, you must either make all shared pins visible and connect them to power and/or ground nets, or you must make them all invisible, in which case they are connected according to their pin names.

In this section:

- [Creating a Package](#)
- [Editing, Deleting, and Viewing a Package](#)

Creating a Package

Creating a package

A part or hierarchical block may have an underlying hierarchical description, such as an attached schematic folder. If it does, it is called a non-primitive. A part or hierarchical block that has no underlying hierarchical description is called a primitive. In Capture, this characteristic is defined in a property, called Primitive, on every part instance. You can change the Primitive property as often as you like during the design process. When a part or hierarchical block is marked as primitive, all of Capture's tools treat it as such. You cannot descend into a part or hierarchical block that is marked as primitive, even if it has an attached schematic folder.

For example, you might create a part and attach a schematic folder that describes its gates and wiring, and then attach schematic folders to some of those parts to describe their transistors. Before you create a netlist for simulation, you should specify those parts as non-primitive, so Create Netlist can descend far enough to find the transistor-level descriptions. Before you create a netlist for board layout, you should specify the parts as primitive, so Create Netlist stops at the gate-level descriptions. Bill of Materials and Cross Reference work similarly.

For part instances that have their Primitive property set to Default, you can configure Capture to treat them as either primitive or non-primitive on a design-wide basis, using the Design Template and Design Properties commands on the Options menu. This is useful when you are describing and simulating your design at varying levels of abstraction (as in a top-down design).

If you attach a schematic folder to a homogeneous part, it is attached to each part in the package, not the package itself. You cannot attach a schematic folder to a heterogeneous part.

When you attach a schematic folder to a part or hierarchical block, you can specify a full path and file name in the Library text box. So, although you can specify a library that has not been saved, you should not try to descend into the attached schematic folder until the library that contains the schematic folder has been saved.

If you do not specify a full path and file name in the Library text box, Capture expects to find the attached schematic folder in the same design as the part or hierarchical block to which it is attached. If the specified schematic folder does not exist in either the design or library, Capture creates the schematic folder when you descend the hierarchy on the part or hierarchical block. For compatibility with future versions of Windows, Capture preserves the case of the path and filename as you specify them in the Library text box.

Using the package view

In the part editor, to view and edit all sections of a part, click the *Edit Pins of All Sections* button in the *Property Sheet* pane. The *Edit All Sections* dialog box opens.

Forcing multiple parts into a single package

For PCB designs, if you need to make sure that two or more parts in your design are in the same package, you use the Update Part Reference tool. First, choose a property that all the parts share and verify that the parts all have the same value for that property, then use the Update Part Reference tool.

For example, you might have several NANDs in a schematic folder and four that are in close proximity in the final product. For each of the four NANDs, create a user-defined property named `COMPGROUP` and set the property's value to `1`. In the `Combined Property String` text box in the *Update Part Reference* dialog box, enter `{COMPGROUP}`.

To specify ignored package pins

The `IGNORE` property, available for package pins, provides a method for you to specify that certain pins on a part are ignored when the part is placed on a schematic page. Pins that have the `IGNORE` property assigned to them do not appear on the schematic page. These pins will also not be included on the part footprint for any downstream PCB layout tools. Also, note that ignored pins will not be included in any back annotation from a layout tool.

1. Open a part in part editor.
The *Package Properties* section appears in the *Property Sheet* pane.
2. In the *Section Pins* section of the *Property Sheet* pane, select the *Pin Ignore* check box for all the pins that you want to ignore.
3. Click the *Apply Pin Changes* button.

4. Save the part.

To force multiple parts into a single package

1. Choose one property that the parts share and assign the same value to that property for each part. You may want to add a user-defined property to each part.
2. In the project manager, select schematic folders or schematic pages if you want to process only a portion of the design. If you want to process the entire design, leave the schematic folders or schematic pages unselected.
3. Choose *Tools - Annotate*.
The *Annotate* dialog box appears.
4. In the *Combined Property String* text box, enter the property name. The name must be enclosed in braces: "{" and "}".
5. Verify that the remaining dialog box options are set the way you want them. For example, specify, among other things, whether you are unconditionally updating all references or only those that are set to the unassigned (?) reference.
6. Click *OK*.

Shortcut

Toolbar: 

To create a multiple-part package

1. Open the library that will contain the part.
2. Choose *Design – New Part*.
The *New Part Properties* dialog box appears.
3. In the *Parts per Pkg* field, specify the number of parts in the package, and also specify whether they are all the same (homogeneous) or different (heterogeneous).
4. Specify the other properties and click *OK*.
The part editor opens with an empty part outline.
5. Edit the part properties as required in the *Property Sheet* pane.
6. Choose *File – Save*.
If you are creating the part in a new library that has not yet been saved, the *Save As* dialog box appears to name the library file.



- If you edit a library provided by OrCAD X, it is important that you assign a new library name so that your changes are not overwritten when you upgrade or update your software.
- After a part is created, you can add to or decrease the number of parts in the package, even if the part starts out as a package of one.

Editing, Deleting, and Viewing a Package

Editing package properties

As with part properties, you can edit package properties in the library or on the schematic page. If you edit package properties on the schematic page, the changes affect only the parts in the project; you are, in effect, creating a new part that is not stored in a library.

Package properties are inherited by every part in the package and by every part placed on a schematic page. Packages do not support user-defined properties.

1. Select and open a part in the part editor.

The *Package Properties* section in the Property Sheet pane appears.

2. Make your changes in this section and save the part.

You can change the number of parts in the package or the *Section Count*, *PCB Footprint*, and *Part Reference Prefix*. The changes are reflected in the part editor, but they are not permanent until you save the part.

To edit pin-specific package properties

1. Select and open a part in part editor.

2. In the Property Sheet pane, make the required changes in the *Package Properties* section and the *Section Pins* section.

3. Click *Apply Pin Changes* button.

Capture updates pin property information and checks for duplicate pin numbers.

Using part or net properties in a package

Using properties, you can conveniently store part and net information. To change part properties on a single part (or on every instance of the part in a design), edit the part in the schematic page editor. To set part properties on every instance of the part that you place, edit the part in the part editor. If you want to make changes to a number of parts or nets, the *Update Properties* command on the

Tools menu is a convenient method. You can use *Update Properties* to edit any properties except part value, part reference, and netname, and you can update the properties of parts in a design or in a library.

Before you run *Update Properties*, you create an update file. To identify the parts or nets you want to update, you specify an identifying property or combination of properties. For each identifier you use, you must create a separate update file and run *Update Properties*.

You can update references incrementally, so that previously assigned part references are not changed, or you can annotate unconditionally, changing all the parts across all the schematic pages processed.

Capture automatically selects to update either instances or occurrences depending upon the type of design. In general, you should update instances for FPGA and PSpice projects, and update occurrences for PCB and Schematic projects.

To delete a part in a package of a heterogeneous part

You can decrease the value of the section count to delete a section.

-  You cannot undo the deletion of part from a package. Once the section is deleted from a package, all information regarding the part is lost.

Viewing a Package

To view a section of the part, select the section number from the drop-down list in the lower left corner of the part editor window.

To switch to a different part in the package

Choose *View – Next Part* or *View – Previous Part* to switch to the next section of the part.

Alternatively, select the section number from the drop-down list in the lower left corner of the part editor window

Synchronizing Parts

Capture enables you to place parts in a design, modify the parts in the [library](#), then update the design so that all of the modified parts reflect the changes you have made in the library. You can use the *Update Cache* command on the Design menu to do this.

When you use the *Update Cache* command, properties of the part are retained; however, the pin properties of the part instance are lost.

To update a part in the design cache so it matches a part in the library

1. Using the part editor, edit the parts in the library as required.
2. Open the project manager window for the design.
3. Select the parts in the project manager.
4. Choose *Design – Update Cache* command.

The design cache and all part instances are updated to match the library parts.

When you copy pages from one design or library to another, parts displayed on the copied pages may appear different due to differences in each design or library cache. If a part is not already in the destination design cache, Capture will copy it from the source design's cache. Otherwise, it will use the part already present in the destination design's cache.

When you use the *Replace Cache* or *Update Cache* command, all properties of the part are retained, but the pin reflects the properties of the library part. This means you lose any changes made to pin properties after the part was placed, including those made by the *Back Annotate* or the *Annotate* tools.

The *Replace Cache* and *Update Cache* commands are quite similar, but they have the following differences:

- You can modify the part's link to the library (part name, path, and library) with *Replace Cache*; you cannot modify the part's link with *Update Cache*.

If you move a library after you place a part, the connection between the part and its library is broken. In this case, the *Update Cache* command will not find the library; you will need to use the *Replace Cache* command and specify the new path to the library.



- Using Replace Cache, you can replace multiple parts selected in the design cache with another single part.
- Using Update Cache, you can update multiple parts at a single instance.

Part Instances and Occurrences

A [part instance](#) is a part you have placed on a schematic page. A part [occurrence](#) is created each time the part instance occurs in a schematic that is within the design hierarchy. So, for example, placing a part on a schematic page in your design creates both an instance and an occurrence. You can assign properties to the part instance, in which case the property (and its associated value) will "shine through" to each part occurrence unless the value is specifically replaced by a value on an individual part occurrence. [The Property Editor window](#) graphically illustrates the state of instance and occurrence properties.

The instance property values shines through to the occurrence as long as the occurrence property values have not been edited in any way. When you explicitly edit an occurrence property value or when Capture modifies it via one of its tools, the occurrence values overrides the instance value. Only the occurrence value will be placed in the netlist.

If you...	The result will be...
place a part on an unused page in your design	only instance properties on that part.
place one part in the root schematic	one set of instance properties and one set of occurrence properties on that part.
do not modify the occurrence properties on an object	instance properties will "shine through" on the occurrence. The instance properties themselves may be shining through from the library definition (or design cache if the two differ)
edit a part in a library	no effect on the part in any project. Use the Update Cache command or Replace Cache command to bring library changes into a design.

How Capture uses instance and occurrence properties

The type of property you update or use in Capture depends on the type of project in which you are working. If you are working with an FPGA project or a PSpice project, Capture allows you a choice, but defaults to update instances when you use the Annotate, Update Properties or Export Properties commands. It is best that you use instances to create Cross Reference and Bill of Materials reports, as well.

When you work with a PCB or schematic project, it is best to update occurrences when you use the Annotate, Update Properties, and Export Property commands. In these projects, Capture also uses occurrences to create reports with Cross Reference and Bill of Materials.

 While modifying occurrence properties, open only one occurrence at a time.

The EDIF 2 0 0, VHDL, and Verilog netlist formats generate true hierarchical netlists. Capture uses the instance property values on nets and parts when it generates a netlist for a design with one of these formats. All other netlist formats in Capture produce flat netlists and use occurrence property values.

Preferred modes for design processing

Capture determines and recommends a preferred mode for processing your design based on the type of project flow, type of design with respect to complex or simple hierarchy, and whether occurrence properties already exist on the design.

The [Cross Reference Parts dialog box](#) is an example of where you can choose to use instances or use occurrences as the preferred mode.

Use instances	If you are in an FPGA, PSpice, or digital simulation project, or if your design does not have occurrence properties.
Use occurrences	If your design has at least one schematic used multiple times or a schematic from an external library. Using occurrences is also the preferred mode if the design already has some occurrence properties.

Removing Part Reference Assignments

If you want to incrementally update a design in which some of the schematic pages have already been updated, you can use the *Annotate* command to remove part references from those schematic pages.

To remove part references

1. In the project manager, select schematic folders or schematic pages if you want to process only a portion of the design. If you want to process the entire design, leave the schematic folders or schematic pages unselected.
2. From the *Tools* menu, choose *Annotate*. The *Annotate* dialog box appears.
3. In the Action group box, select Reset part references to "?" ;select other options as appropriate; then click *OK*.

Shortcut

Toolbar: 

Generating Library Parts

Generating a part from a PSpice model library

You can use the Generate Part dialog box to create a part that has pins matching all the hierarchical ports on the root schematic of the design for design reuse. If a destination library file does not exist, Capture creates a library part file (.OLB) that contains the newly generated part and a copy of the top-level schematic. The new .OLB appears in the project manager's Outputs directory along with the new part and schematic. The new part has a reference back to the design where the source schematic resides so that when you descend the hierarchy from the placed part, it will open the source schematic from the destination library.

-  To create a pin on a symbol using the Generate Part utility, the pin must have a pin to port mapping in the pin file.

To generate a part from a PSpice model library

1. From the Tools menu, choose Generate Part to open the Generate Part dialog box.
2. In the Netlist/source file text field, enter the path and name of the file you want to use to generate the part, or click the Browse button to browse for it. Ensure that you change the type of files in the Browse File dialog box to PSpice Model Library Files (*.cir, *.net, *.lib).
3. Select the Implementation name from the drop-down list box in the Implementation options. If more than one model is available, you can choose one from the drop-down list, or you can keep the default setting of <ALL> that appears in the Implementation name and the Part name text boxes. Choosing <ALL> will create a part from all available models.
4. In the Destination part library text field, you can choose to accept the default path and file name, enter the path and name of the file you want to use for the new library, or click the Browse button to search for another one.
5. Click *OK*.

Generating a part from a schematic or library

You can use the Generate Part dialog box to create a part that has pins matching all the hierarchical ports on the root schematic of a design and use the design as a reuse module. If a destination library file does not exist, Capture creates a library part file (.OLB) that contains the newly generated part and a copy of the top-level schematic (if the Copy schematic to library option is selected). The new .OLB appears in the project manager Outputs directory along with the new

part and schematic. The new part has a reference back to the design where the source schematic resides so that when you descend the hierarchy from the placed part, it will open the source schematic from the destination library.

To generate a part from a schematic design for reuse

1. In the project manager, click to select a schematic folder from which you want to create a part. The schematic can be in a design (.DSN) file or a library (.OLB) file.
Note: The design file you use for the source must contain **ports** or the Generate Part operation will fail.
2. From the Tools menu, choose Generate Part. Capture fills in default settings in the Generate Part dialog box.
3. Browse to change the Netlist/source file or keep the default settings. You can generate a part from an **external design** or from the current design in the active project manager window.
4. Select the Netlist/source file type. For example, choose Capture/Schematic Design if you are generating a design reuse module.
5. Keep the default Part name or enter another name from the design.
 - If the source design has more than one schematic and no schematic is selected, or if more than one schematic is selected, the Part name setting defaults to the root schematic.
 - If the source design is not opened in the current project manager window, its root schematic will be the default Part name.
6. Browse to change the Destination part library or keep the default settings. The default setting creates a new .OLB file that matches the name of the source .DSN file. If the source design is a library (.OLB), the default part library name will be the same name as the source file.
7. Set Primitive property of the new part using the Primitive radio buttons. The default value is No. You can **descend** from the placed part to its schematic if the Primitive is set to No. For a design reuse module check Default.
8. Select the Copy schematic to library check box if you want to include a copy of the schematic in the new .OLB file along with the new part. This option is unavailable if the source and destination files are the same.

- ✓ If you are generating a design reuse module, do not check the Copy schematic to library option. If you do, you will lose occurrence properties that are critical in the design reuse module. Also, checking this option will overwrite the custom library, so make sure to specify an used library name unless you intend to overwrite.

9. Click OK.

- ⚠ If the source schematic is copied and becomes locally available in the library, the [implementation path](#) is the same as the destination library. If the source schematic is not copied, the implementation path is the same as the source design or library.

To reuse an existing hierarchical design

1. Create a new project or open a working design.
2. In the project manager from the Tools menu, choose Generate Part.
3. Use the Browse button to pick a source design file that has a reusable design and is useful in the working design.
4. Select the schematic to reuse.
5. Enter a resulting library name or keep the default.
6. Select the Copy Schematic to Library Check box and click OK to create a part.
7. Select the schematic page editor window to place instances of the part to reuse its schematic design.
8. Click OK.

Creating a Split Part

Capture allows you to split a part using the [Split Part Section Input Spreadsheet](#).

To split a part

1. Select the part that you want to split from the Library folder in the project manager.
You need to select a single-sectioned part from a library. You can split a multi-sectioned part only when it has already been split using the Split Part Section Input Spreadsheet.
2. Select the Tools menu and choose the Split Part command.
OR
Right-click on the selection and choose Split Part from the pop-up menu.

The Split Part Section Input Spreadsheet appears displaying all the pins and their corresponding properties for the selected part.

 The Part Name and Part Ref Prefix fields display the name of the selected part and its part reference. These fields are view-only.

3. Select Numeric or Alphabetic in the Part Numbering group. If you select Alphabetic, an alphabet (between A to Z) will be added as a suffix to the current part reference for each of the split parts. If you select Numeric, a number (between 1 and 1024) will be added as a suffix to the current part reference for each of the split parts.

The Section property column changes based on your selection in the Part Numbering group. For example, if Alphabetic is selected, the Section property column displays “A”. Resize the Split Part window by dragging the borders as per your requirement.

4. Specify the number of sections you want to have in the split part in the No. of Sections text box. The Section property column changes to a list box displaying the number of sections you specified in the No. of Sections text box.

 Number of sections cannot be less than one.

 If you select alphabetic numbering, then you can create up to a maximum of 26 sections only. If you select numeric numbering, then you can create up to a maximum of 1024 sections.

5. Click on a Section cell for a specific pin and select a section number from the list box.

OR

Click on a Section cell and enter the new section number.

The selected Section cell displays the new section number.

- ✓ You can select Section cells for multiple pins simultaneously using the Shift+Down Arrow keys and enter the section number.

OR

Select the Section cells for multiple pins simultaneously using the Shift+Click, then press the Ctrl key, and then select a section number of your choice from the list box. The selected Section cells get populated with the section number of your choice.

Click the first cell of the range, and then drag to the last cell, and then enter the section number of your choice. The selected Section cells get populated with the section number of your choice.

- ✓ You can select alternate Section cells for multiple pins simultaneously using the Ctrl+Click and enter the section number.

- ✓ To sort on any property, double-click its name in the column header.

⚠ The pins will be added in the same order in the split part as the order in which the pins appear in the spreadsheet.

6. Click the Save button to save the changes to the current part. If any warnings are generated during the save operation, a message box appears asking you whether you want to view the warnings.

To view the warnings, click the [View Warnings button](#).

The Split Part Section Input Spreadsheet expands and displays a grid showing warnings messages.

If you select the Continue button, the split part is saved as is.

- ⚠
- Click the Hide Warnings button to hide the warning messages or the Show Warnings button to display the warning messages again.
 - Use Save As to retain the original part and save the changed part as a new part in the same library.

- ✓ You can hide or show a property column in the Split Part Section Input Spreadsheet. To do this, right-click the property column header you want to hide and select Hide from the pop-up menu. The selected property column will not appear now. To show a property column, right-click the property column header next on the right-hand side of the hidden property column and select Unhide from the pop-up menu. The hidden property column appears in the Split Part Section Input Spreadsheet. Alternatively, you can show a property column by:
- Double-clicking the column handle () of the property column header.
 - Dragging the column handle of the property column header.
 - (only the last two methods can be used to show a property column, which is the last column in the Split Part Section Input Spreadsheet).

If the number of Sections specified in the Split Part Section Input Spreadsheet is greater than one, then the resulting part will be of heterogeneous package type.

You can change the order in which the property columns and rows appear in the Split Part Section Input Spreadsheet. To do this, select the property column/row header you want to move and drag and drop it to the location where you want it in the Split Part Section Input Spreadsheet.

You can use standard copy and paste features to copy all the data from the Split Part Section Input Spreadsheet to MS Excel. You can later use the MS Excel file for archiving or documentation. It is recommended that you avoid using MS Excel to paste information into the Split Part Section Input Spreadsheet.

Adding pins to Split Part Section Input Spreadsheet

You can add more pins to the Split Part Section Input Spreadsheet. The pins are added at the end of the current set of pins. The pins numbers for the added pins is generated automatically.

To add a pin

1. From the Split Part Section Input Spreadsheet, Click the Add Pins button. The Add Pins dialog box appears.
2. Specify the number of pins you want to add in the Number of Pins text box.
3. Click OK. The required pins are added at the end of the current pin set in the Split Part Section Input Spreadsheet.

 All the pins added are populated with default property values. The default value for the Section property column is one.

Deleting pins from Split Part Section Input Spreadsheet

You can delete pins along with their corresponding properties in the Split Part Section Input Spreadsheet.

To delete a pin

1. Select a row in the Split Part Section Input Spreadsheet.
2. Click the Delete Pins button. A message box appears asking you to confirm the deletion.
3. Click OK to confirm deletion. The selected row containing the pin information is deleted from the Split Part Section Input Spreadsheet.

! Once you delete a pin from the Split Part Section Input Spreadsheet, you cannot retrieve it later.

Viewing split part properties

When you split a part, Capture assigns the following properties to it:

- SWAP_INFO—This property is assigned to each section and its value is determined by the number of sections you specified in the No. of Sections text field in the Split Part Input Spreadsheet. For example, if you split a part into 3 sections then all the 3 sections will be assigned the SWAP_INFO property with value (S1+S2+S3).
- SPLIT_INST=TRUE

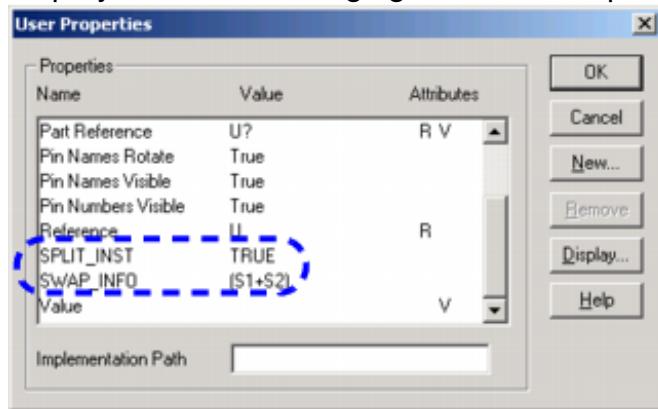
! Saving a multi-section split part will remove any existing SPLIT property. Instead, it will add the SPLIT_INST property.

Saving a multi-section split part will overwrite any existing SWAP_INFO property value with the value (S1+S2+S3+...Sn), where n is the number of sections specified.

⚠ Ensure that you use ‘(’ and ‘)’ brackets for defining single sections. For example, If you have a split part having 6 sections (S1 to S6) and wish to add the SWAP_INFO property such that swapping happens only between sections S5 and S6, you need to add the SWAP_INFO value as (S1), (S2), (S3), (S4), (S5+S6).

To view split part properties

1. Double-click a split part from the Library folder in the Project manager. The [Part editor window](#) displays.
2. Select the Options menu and choose Part Properties. The [User Properties dialog box](#) is displayed. The following figure shows a split part with the two properties assigned to it.



⚠ The SWAP_INFO and SPLIT_INST properties appear in the User Properties dialog box for only those parts that are heterogeneous.

Viewing split part package and its properties

A split part is a multi-sectioned package. You can view a split part in package view.

To view split part package

- From the part editor, select the View menu and choose the Package command. An image of all the sections in the package appears.
OR
- You can also double-click a split part from the Library folder in the project manager to open the part editor window.

⚠ The part reference for each of the sections contain a suffix entry. For example, J?A, J?B, and J?C, where A, B, C are the section numbers.

To view split part package properties

1. From the part editor, select the View menu and choose the Package command.
2. Select the Options menu and choose the Package Properties command.

The Edit Part Properties dialog box displays.

The Multi-Part Package group displays the properties for your split part package, like parts per package, package type, and part numbering.

Deleting a Part

To delete a part

1. Open the library containing the part to delete.
2. In the project manager, select the part.
3. Right-click the part and choose *Delete*.
4. To save the changes to the library, choose *Save* from the File menu.

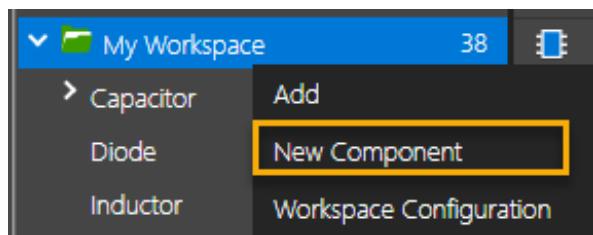
Creating Components

You can create components in the local library workspace using existing symbols, models, footprints properties. You can add existing CAD models of the following types into the workspace at %HOME%\workspace\ and build parts using the models in the Cloud workspace:

- Symbols – (.olb)
- Footprints – (.dra, .psm, .pad)
- PSpice models – (.lib)
- 3D – (.stp)
- Technology file – techfile

To create a new component, follow these steps:

1. Right-click the *My Workspace* node in the *Categories* pane and choose *New Component*.



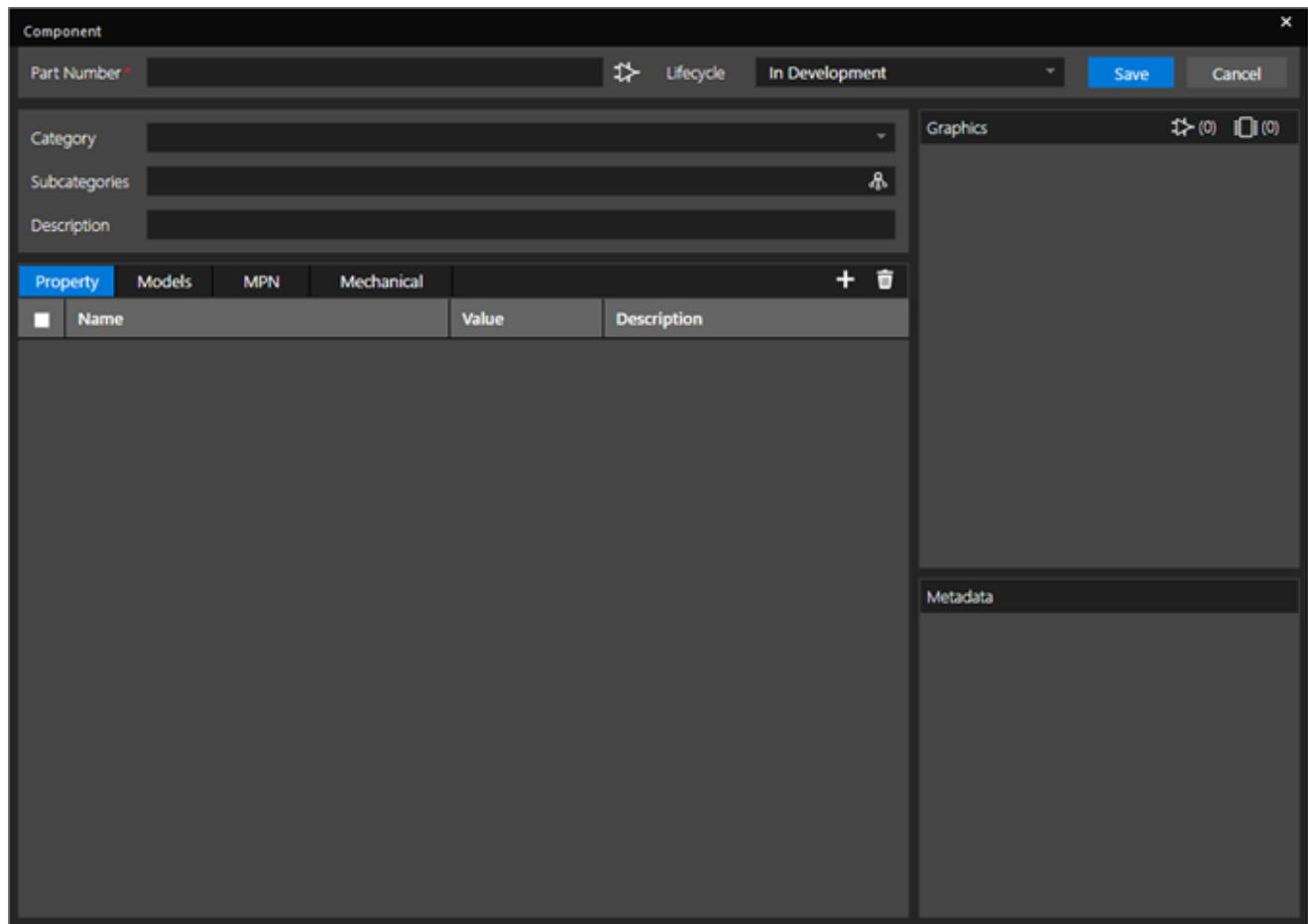
Alternatively, use one of the following ways to create a new component:

- Click the *New Component* button in the Component Explorer interface.
- In the project manager:
 1. Open a new .olb file.
 2. Select the *Library* node in the project manager tree.
 3. Right-click a library name and choose *New Component* from the shortcut menu.

The *Component* dialog box is displayed providing you an interface to create a new component using existing symbols and footprint information.

Library and Part Management in OrCAD X Capture

Managing Parts--Deleting a Part



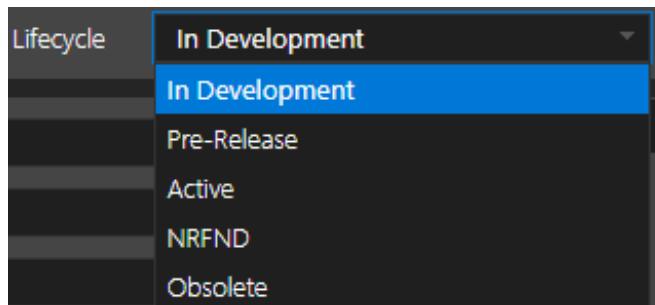
2. Specify a part number in the *Part Number* field.

The image next to the Part Number field indicates the type of the part. You can create the following types of components:

- Electrical
- [Mechanical](#)

The type is determined by the seeded category. The default category is *Electrical*. To create a *Mechanical* component, you first need to create a [category](#) of the type and then create the component.

3. Select the lifecycle status of the component from the *Lifecycle* field.



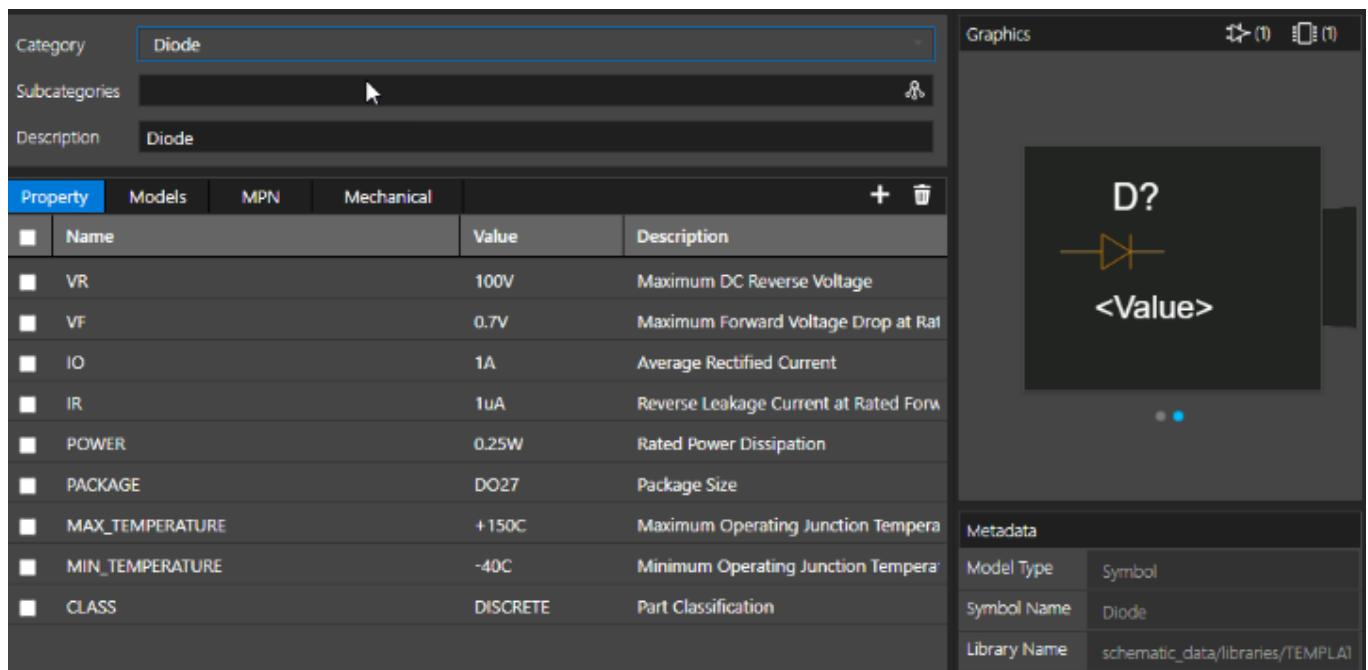
This field helps in understanding the availability status of a component:

Lifecycle Stage	Component Availability Status
In Development	The component is being developed.
Pre-Release	The component is developed but not released yet.
Active	The component is actively available for use in the designs
NRFND	The component is <i>Not Recommended for New Designs</i> .
Obsolete	The component is no longer available for use.

You can create a component from scratch or use an existing [category](#).

4. Optionally, select a category from the *Category* list.

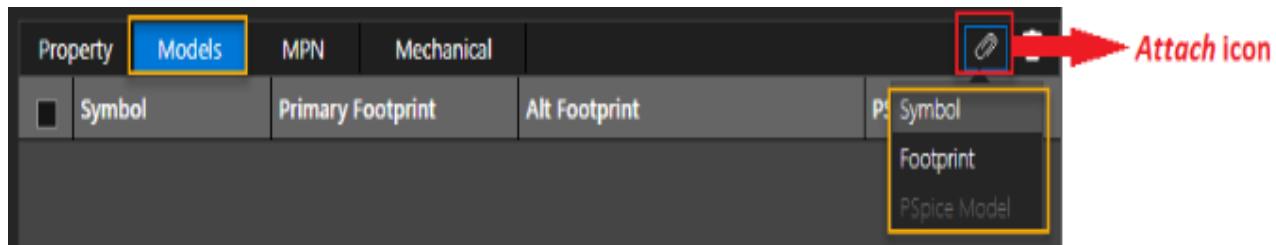
You can choose an existing [category](#) from the *Category* list to use as a template to create the component. When you select a category, the symbol, footprint, description, and part-level properties are automatically assigned to the component as illustrated in the following image:



5. Specify a brief description of the component or update the entry if you selected a category.
6. In the *Property* tab, click the *Add* icon (plus sign) to add a new property.
7. Specify a property name, value, and description for the property.
You can add more properties by clicking the plus sign.

 To remove a property from the component, select the check box to the left of the property name, and click the *Delete* icon.

8. Click the *Models* tab.
You can add a symbol, a PSpice model, and a footprint to the component in the Models tab.

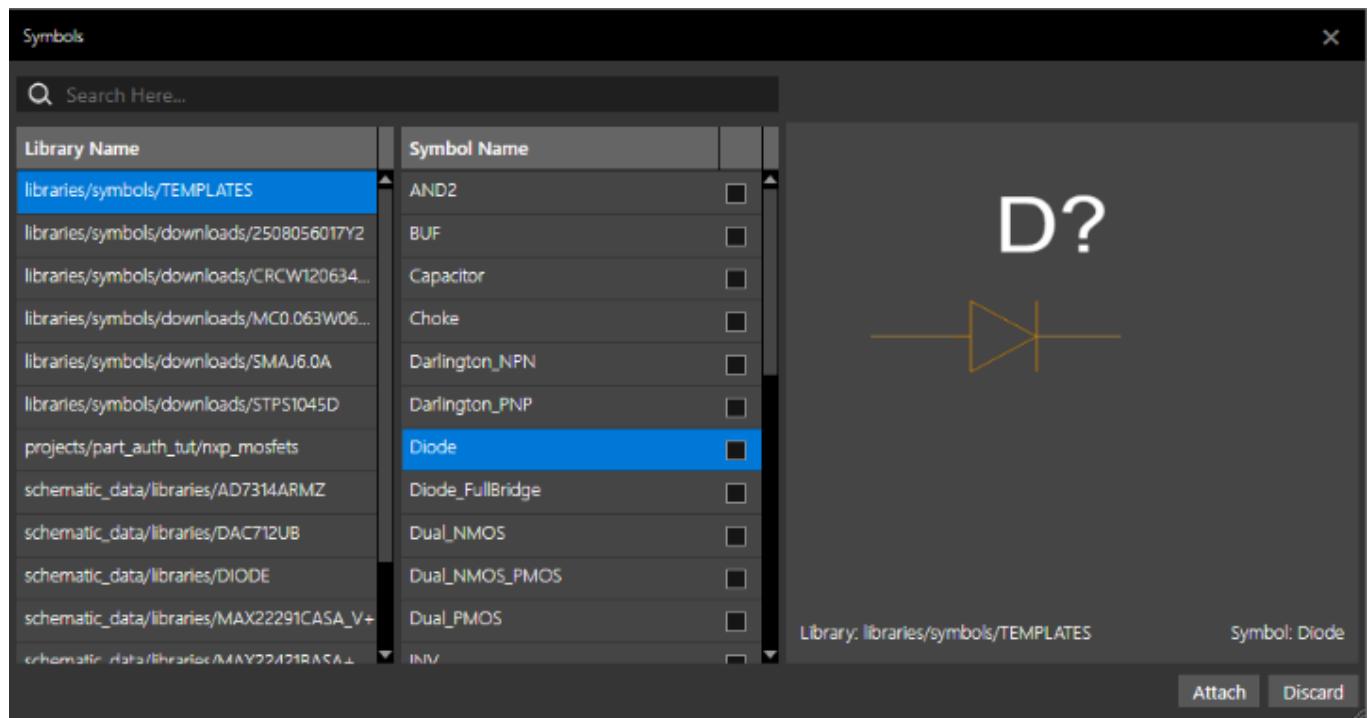


9. Click the *Attach* icon and choose *Symbol*.
The *Symbols* dialog box displays a list of symbols from the available libraries.

For the components based on a category, symbols and footprint are pre-selected. You can select any other symbol or footprint from the list of available symbols and footprints.

10. To search for symbols across libraries, specify a search string in the Search bar and click the magnifying glass icon or press **Enter**.

The string is searched in both the library list and the symbol list sections. The symbol library list displays the library files that include the string and the symbol list displays all the symbols in the selected library containing the string.

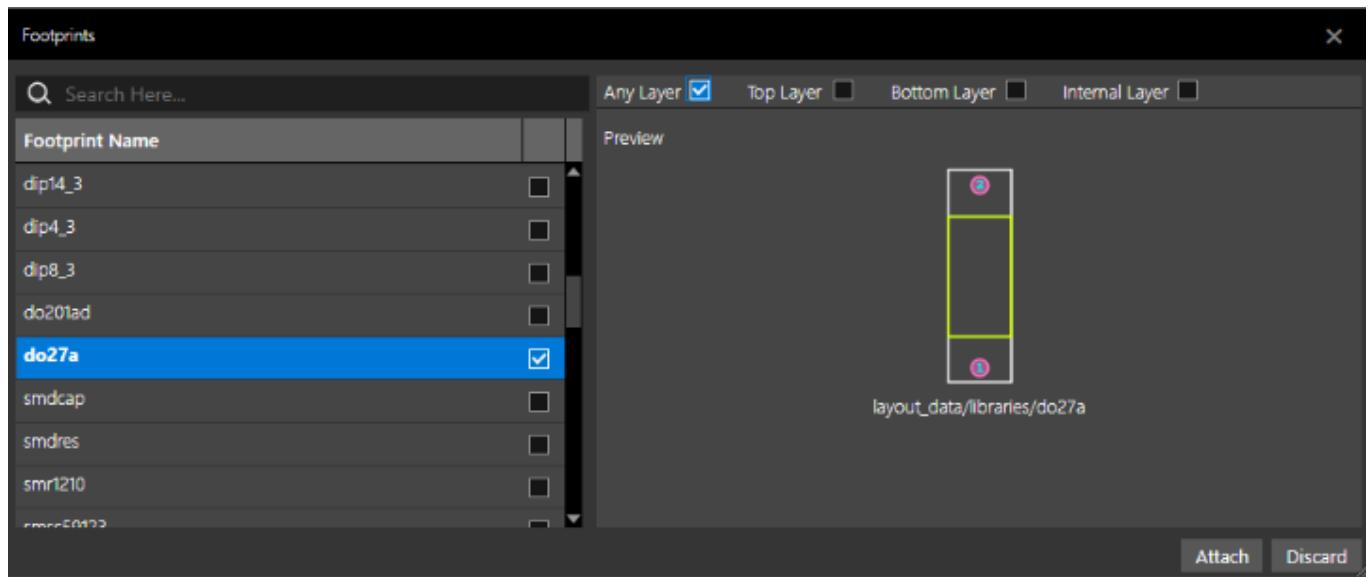


11. To associate a symbol with the component, select the check box next to the symbol.

The name of the first symbol you select starts appearing in bold and is set as primary. This indicates that the selected symbol is set as the primary symbol for the component. You can change this by selecting another symbol and choosing *Set as Primary* from the shortcut menu on the symbol.

Symbol Name	
AND2	<input type="checkbox"/>
BUF	<input type="checkbox"/>
Capacitor	<input type="checkbox"/>
Choke	<input type="checkbox"/>
Darlington_NPN	<input type="checkbox"/>
Darlington_PNP	<input type="checkbox"/>
Diode	<input checked="" type="checkbox"/>
Diode_FullBridge	<input type="checkbox"/>
	Set as Primary
Dual_NMOS	<input type="checkbox"/>
Dual_NMOS_PMOS	<input type="checkbox"/>
Dual_PMOS	<input type="checkbox"/>
INIV	<input type="checkbox"/>

12. Click the *Attach icon* and choose *Footprint* to associate the footprint information. The *Footprints* dialog box displays a list of available footprints for you to choose.



13. To associate a footprint with the component, select the check box next to the footprint.
14. Choose the layers to which the footprint is to be associated: *Any Layer*, *Top Layer*, *Bottom Layer*, and *Internal Layer*.

The first footprint you select appears in bold text. It is the default footprint and the `PCB_FOOTPRINT` property gets assigned to this footprint on symbol. You can also select one or more alternative footprints. These alternative footprints are associated with the `ALT_SYMBOLS` property.
15. Click *Attach*.
16. To associate a PSpice model with the symbol, click the *Attach* icon and choose *PSpice Model*.

The *Associate PSpice Model* dialog box opens.
17. Select the PSpice model library containing the model to be associated with the component in the *Model Library* drop-down list.

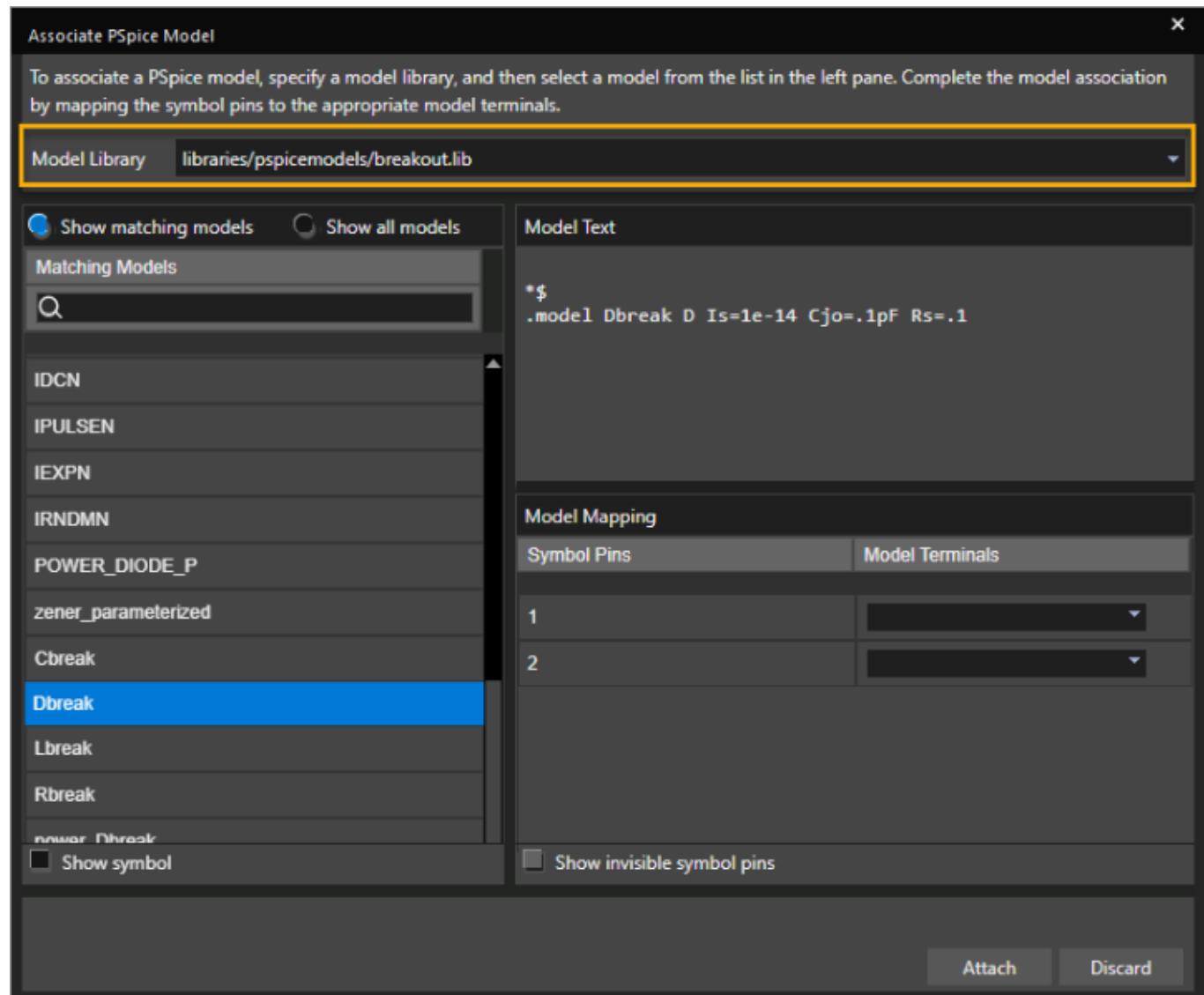
All the PSpice model libraries available in the Cloud database can be accessed from the *Model Library* list. You need to ensure that the required PSpice model library is present at the following location: `$HOME\cdssetup\workspace\schematic_data\libraries`. You can access Cloud data from the *File Manager* user interface.

- ! You need to add the path to the PSpice models library (`LIBPATH`) in `pspice.ini` to run the PSpice simulation. Else, the simulation will fail.

```
LIBPATH="$HOME\cdssetup\workspace\schematic_data\libraries"
```

The Matching Models section is populated with the list of models that can be associated with the selected component.

The Model Text section displays the model definition of the model selected in the list.



18. Select the required model from the list or search for a specific model.

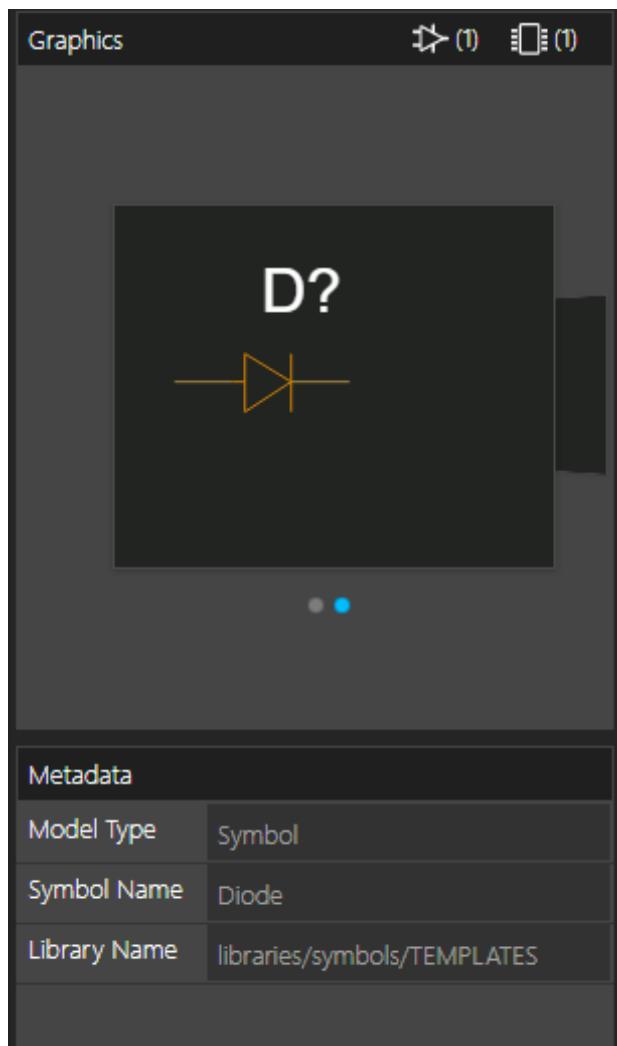
19. In the Model Mapping section, map the symbol pins to the correct model terminal. For this, select a port from the Model Terminals for each symbol pin.

20. Click *Attach*.

The associated symbol, footprint, alternative footprint, and the PSpice model display in the *Models* tab.

Property	Models	MPN	Mechanical	
<input type="checkbox"/>	Symbol	Primary Footprint	Alt Footprint	PSpice
<input checked="" type="checkbox"/>	Diode	do27a	(do27a)	Dbreak

Associated symbol and footprint graphics and the metadata for the selected symbol and footprint also display in the right pane:

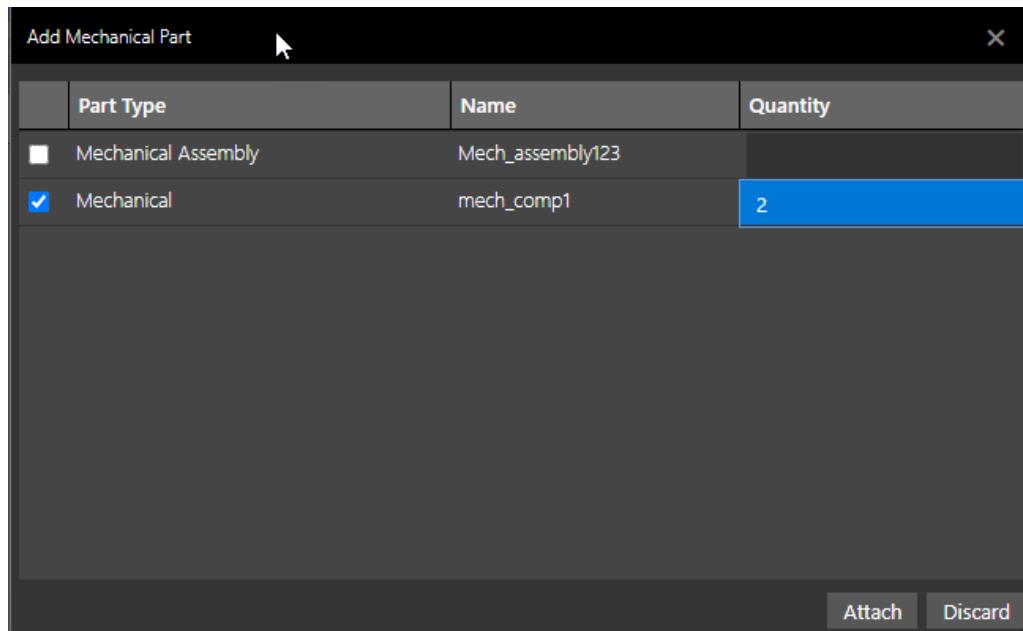


21. Click the *MPN* tab.
22. Click the *Add* icon to add a **Manufacturer Part Number** (MPN) to the component.
23. Specify required values for the MPN, manufacturer name, datasheet URL, and RoHS (Restriction of Hazardous Substances) compliance.

Property	Models	MPN	Mechanical	+ Delete	
<input type="checkbox"/>	MPN	Manufacturer	Datasheet URL	RoHS	Status
<input checked="" type="checkbox"/>	es20011	On Smie	https://diodes.com/datasheet		Active

24. Click the *Mechanical* tab and then click the plus sign to add a Mechanical part.

The *Add Mechanical Part* dialog box opens. A list of available mechanical parts is displayed if the mechanical parts exist in the workspace.



25. Select the check box to the left of *Mechanical Assembly* or *Mechanical* part types.
26. Specify the quantity of the mechanical parts to be included in the assembly.
27. Click *Attach*.

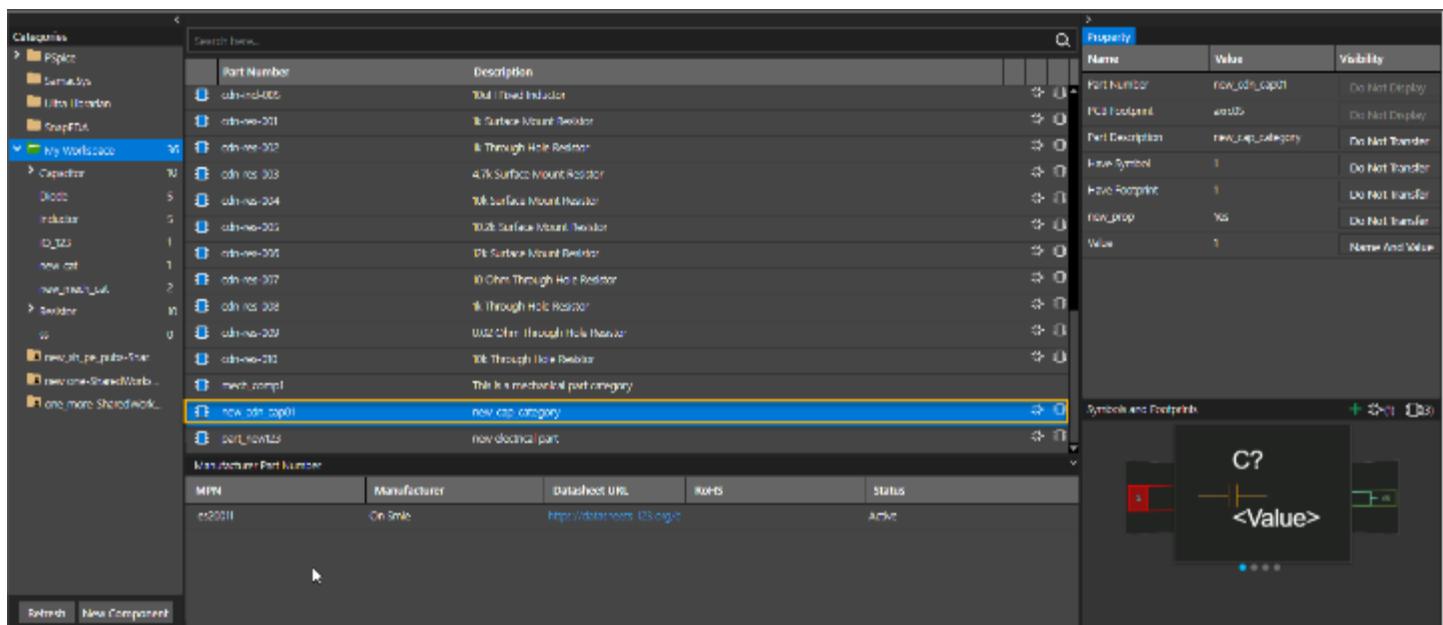
The selected part is added in the *Mechanical* tab.

Click *Save*.

The component appears in the list of available components in the local workspace, *My Workspace*. Note the component properties display in the properties browser on the right and the associated symbol and footprint appear in the *Symbols and Footprints* section.

Library and Part Management in OrCAD X Capture

Managing Parts--Deleting a Part



Annotating Properties on Schematic

Component property annotation and visibility are set to default values defined in the category on which the component is based. For a component created from scratch, the default value of properties is set to *Do Not Transfer*. You can override this behavior by changing the value of the *Visibility* column for a property in the property browser of the Component Explorer interface.

To annotate a property on schematic canvas with visibility settings different from those defined in the category, do the following:

1. In the property browser (the right pane) of the Component Explorer interface, click the *Visibility* column for a property.
2. Select one of the available options to change the visibility of the property:

Name	Value	Visibility
Part Number	new_cdn_cap01	Do Not Display
PCB Footprint	axrc05	Do Not Display
Part Description	new_cap_category	Do Not Transfer
Have Symbol	1	Do Not Transfer
Have Footprint	1	Do Not Display
new_prop	Yes	Use Library Setting
Value	1	Value
		Name And Value
		Name Only
		Both if Value Exists
		Value if Value Exists

When you place this component on the schematic, the properties display as selected in the *Visibility* column

Related Topics

- Managing Mechanical Assemblies
- Managing a Local Component Library

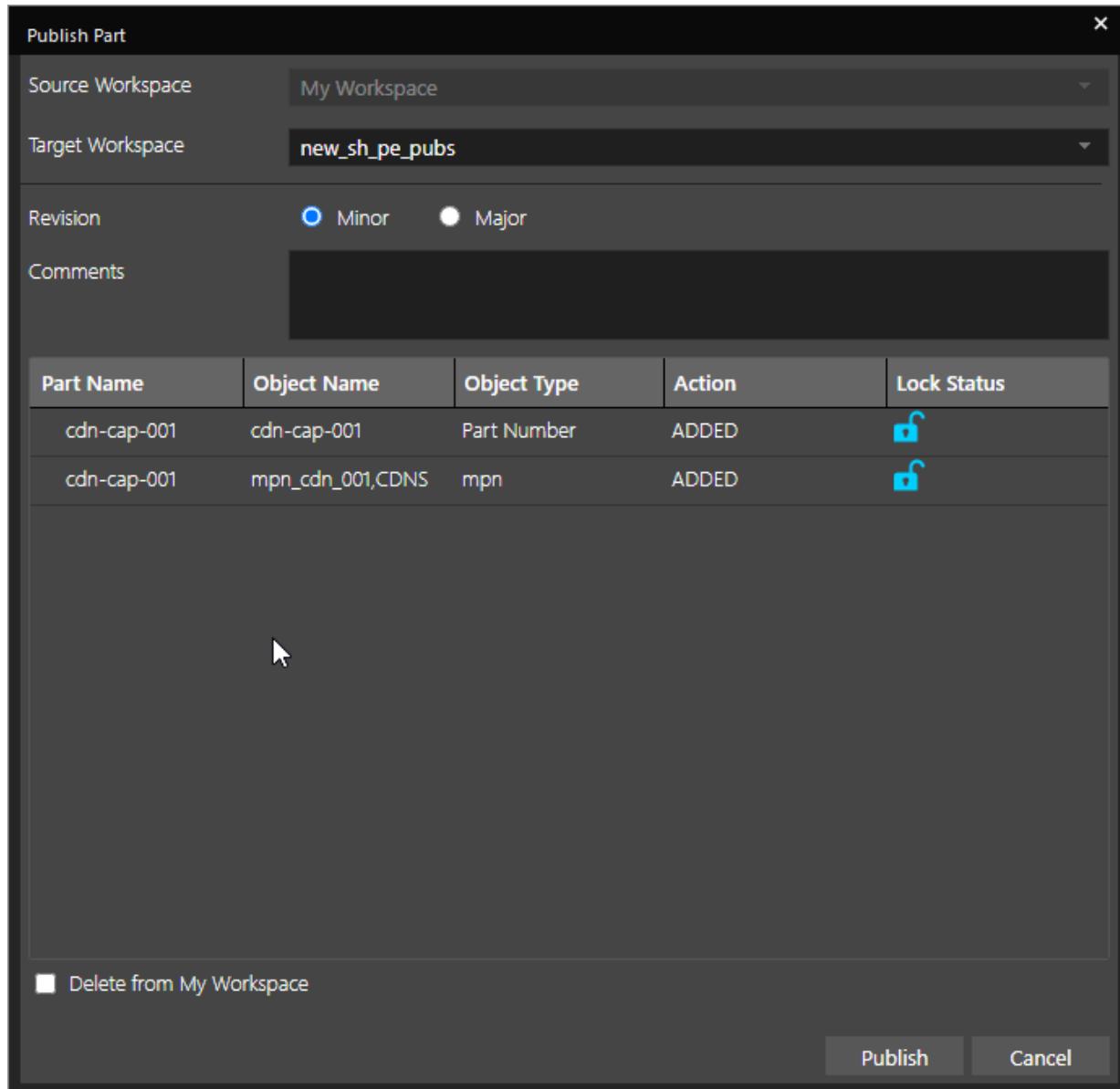
Sharing Components

You can publish a component in your local workspace to create its copy in the [shared](#)

[workspace](#). All the team members with access to this workspace can work on the component.

To share a copy of a component in your local workspace, do the following:

1. Right-click the component to be shared and choose *Publish*.
2. In the resultant *Publish Part* dialog box, select the target workspace where the components are to be published for sharing.



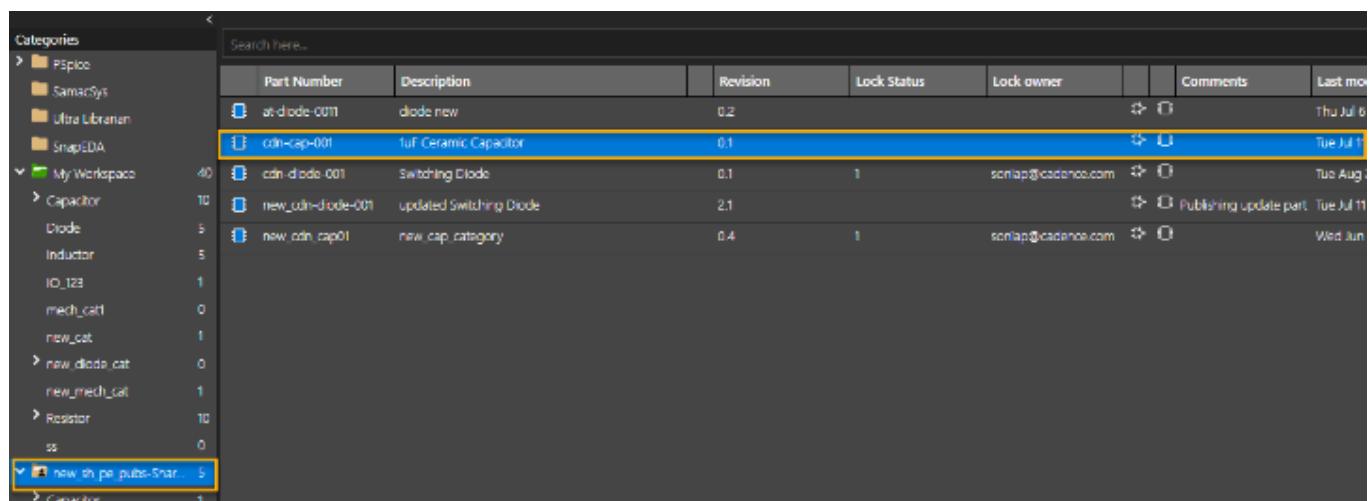
Note that My Workspace is selected by default in the *Source Workspace* field. The component and any MPN associated with the component is also listed. The *Action* column lists the most

recent action done on the object since the last time it was published.

Added indicates the component is being published for the first time.

3. Select either *Minor* or *Major* depending on how you want to save the version.
By default, the published part is saved with a *Minor* version.
4. Select the object to be checked out.
5. Select *Delete from My Workspace*, if you want to delete the component from the local workspace.
6. Click *Publish*.
7. Click the shared workspace in the *Categories* pane.

The published component appears in the shared workspace.



The screenshot shows the 'Categories' pane on the left with various workspace categories listed. The 'My Workspace' category is expanded, showing sub-categories like 'Capacitor', 'Diode', 'Inductor', etc. A specific workspace named 'new_thin_pnape_publis-Shar...' is selected, indicated by a yellow highlight. The main area displays a table of components with columns: Part Number, Description, Revision, Lock Status, Lock owner, Comments, and Last modified. The table contains several entries, with one row highlighted in blue.

Part Number	Description	Revision	Lock Status	Lock owner	Comments	Last modified
at-diode-001	diode new	0.2	Unlocked			Thu Jul 6
cdn-cap-001	1uF Ceramic Capacitor	0.1	Locked			Tue Jul 11
cdn-diode-001	Switching Diode	0.1	Unlocked	sonlap@cadence.com		Tue Aug 1
new_udn-diode-001	updated Switching Diode	2.1	Unlocked			
new_cdn_cap01	new_cap_category	0.4	Locked	sonlap@cadence.com	Publishing update part	Tue Jul 11

Component Locking

When a component is checked out, it is locked by the current user (lock owner). The *Lock Status* and *Lock Owner* columns are updated to reflect the status. This component will be available for editing by other users only after the lock owner checks it back into the shared workspace by publishing it or unlocks the file explicitly using the *Unlock* command. Any user trying to edit a component locked by another user is notified of the status.

When publishing a component from a source shared workspace to a target shared workspace, it is checked whether the component is locked in the destination workspace. The check is not done on lock status of the component in the source workspace. This is done because the components in

the two shared workspaces are independent of each other. A component can be published to a shared workspace and then copied to *My workspace* for editing. From My Workspace, the component can be published back into the shared workspace.

As the components in the two shared workspaces are independent of each other, they can have the same revision, but different content.

Editing Shared Components

All the members with access to this shared workspace, can use this component depending on the assigned [roles](#). To edit a shared component, the users first need to move it to their local workspaces (*My Workspace*).

To edit the component, do the following:

1. Right-click the component and choose *Edit*.

In the *Publish Part* dialog box, *My Workspace* is now the target workspace.

2. Click *OK*.

The *Lock Status* and *Lock owner* columns are updated to reflect the status of the component.

	Part Number	Description	Revision	Lock Status	Lock owner	Comments	Last modified
	at-diode-001	diode new	0.2				Thu Jul 6 02:54:52 20...
	cdn-cap-001	1uF Ceramic Capacitor	0.1	1	sonlap@cadence.com		Tue Jul 11 03:58:51 2023
	cdn-diode-001	Switching Diode	0.1	1	sonlap@cadence.com		Tue Aug 23 08:35:10 ...
	new_cdn-diode-001	updated Switching Diode	2.1			Publishing update part	Tue Jul 11 03:45:12 2023
	new_cdn_cap01	new_cap_category	0.4	1	sonlap@cadence.com		Wed Jun 21 07:16:45 ...

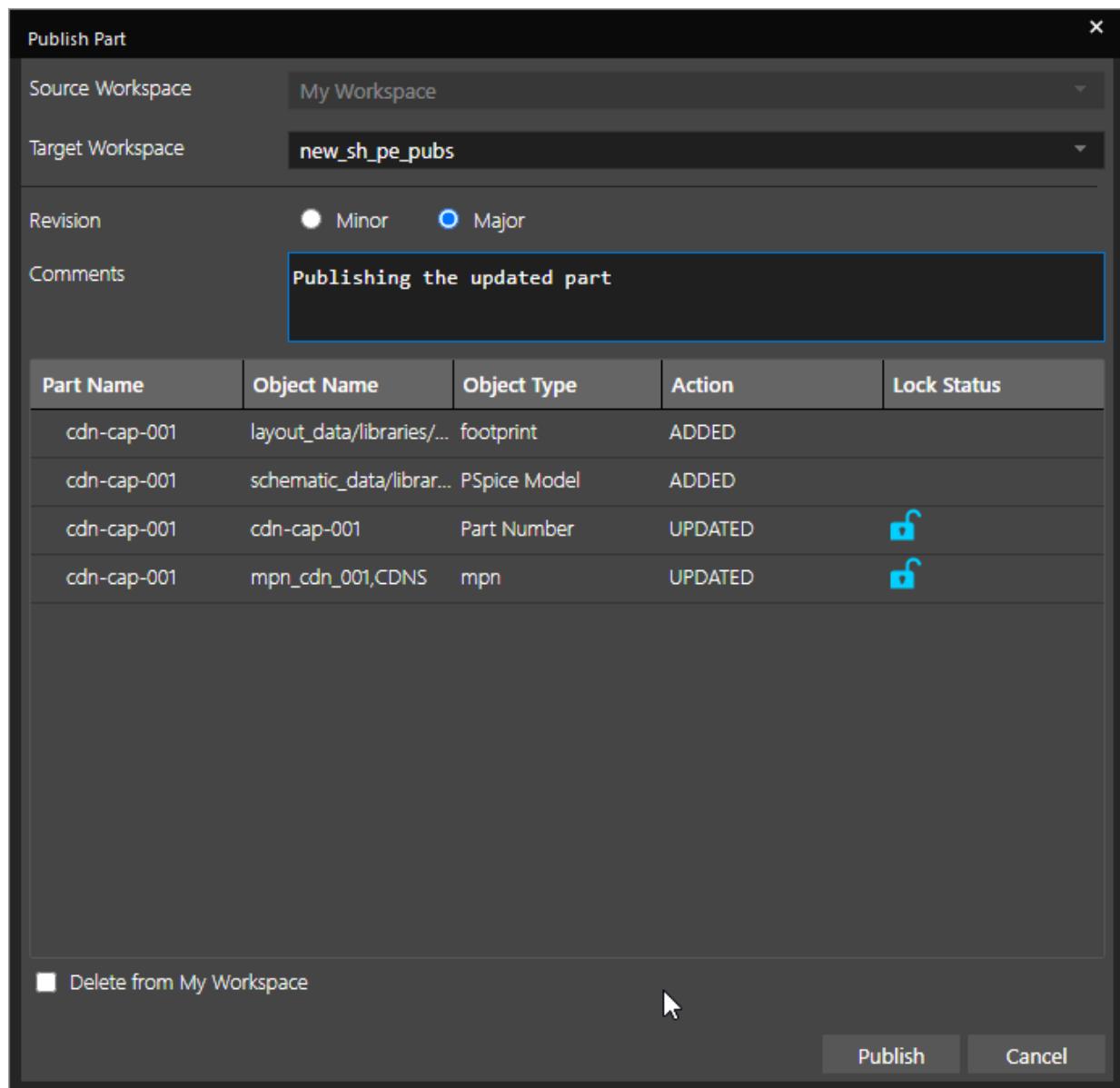
The component is copied to the local workspace where you can edit it.

After making the required changes to the component, you need to check it back in to the shared workspace.

3. Right-click the component in *My Workspace* and choose *Publish*.
4. Select the *Minor* or *Major* options as required and add any tags or comments.

Library and Part Management in OrCAD X Capture

Managing Parts--Sharing Components

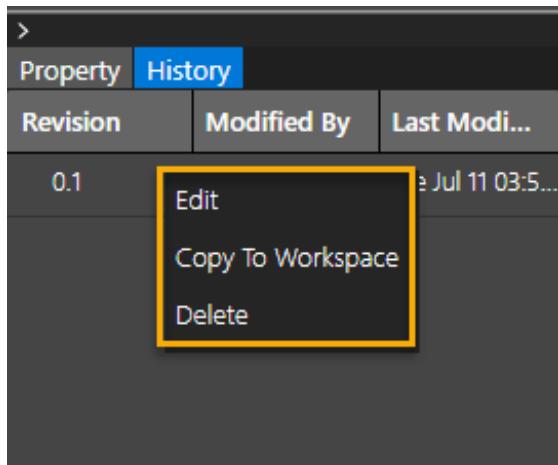


5. Click *Publish..*

The component is updated in the shared workspace. The version number of the component is modified as specified and it is unlocked as well.

	Part Number	Description	Revision	Lock Status	Lock owner	Comments	Last modified
	at-diode-0011	diode new	0.2				Thu Jul 6 02:54:52 20...
	cdn-cap-001	1uF Ceramic Capacitor	1.0			Publishing the update...	Tue Jul 11 04:20:54 20...
	cdn-diode-001	Switching Diode	0.1	1	soniap@cadence.com		Tue Aug 23 08:35:10 ...
	new_cdn-diode-001	updated Switching Diode	2.1				Tue Jul 11 03:45:12 2023
	new_cdn_cap01	new_cap_category	0.4	1	soniap@cadence.com		Wed Jun 21 07:16:45 ...

The *History* tab lists the previous version of the component. Each time this component is modified and checked in, a new row is added for the previous version. You can [restore or roll back](#) to a previous version of the component using the *Edit* command from the shortcut menu. You can also copy a read-only version to the local workspace or delete it from the workspace.



Related Topics

- [Configuring Workspaces](#)
- [Sharing Workspaces](#)

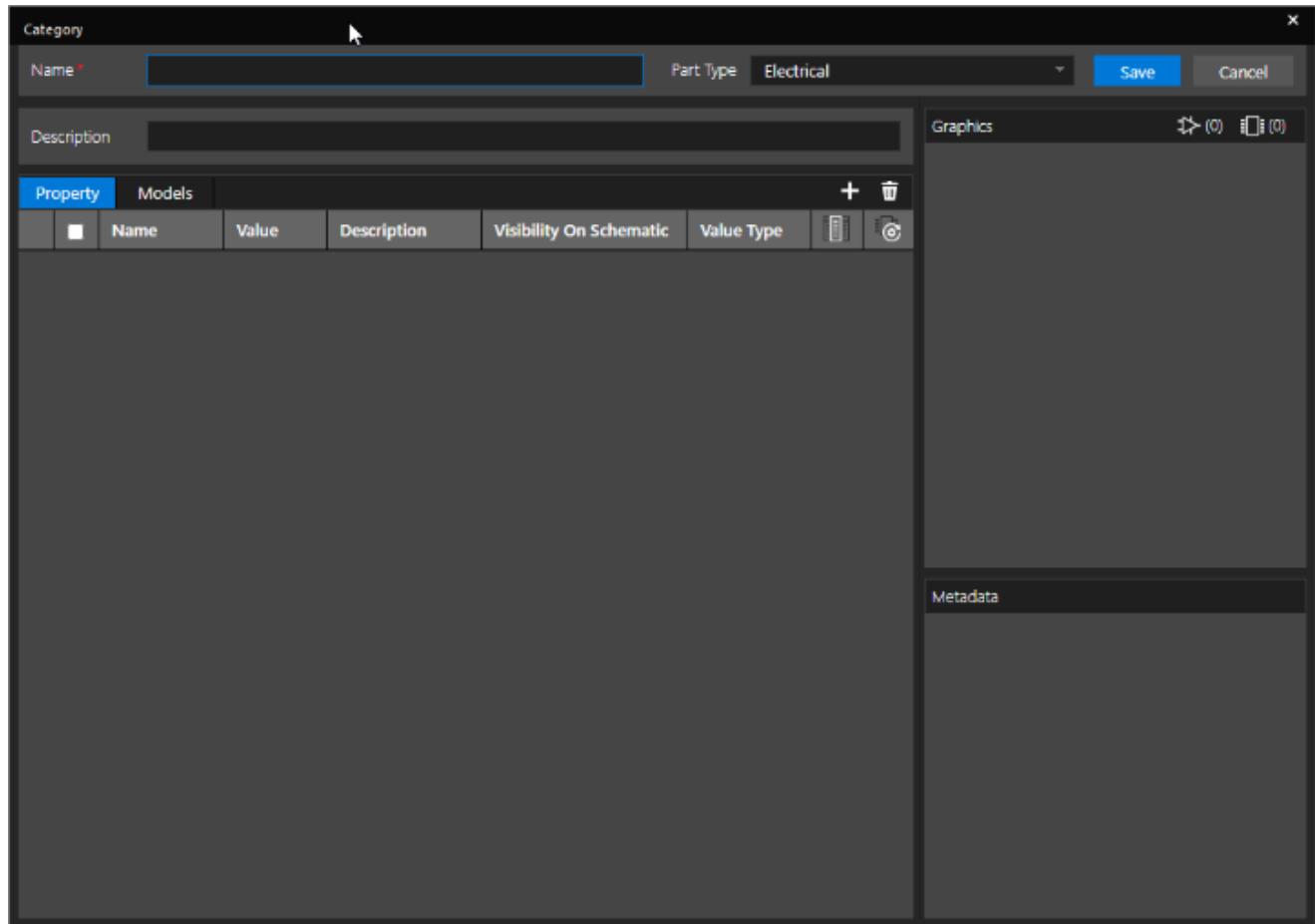
Managing Categories

A category is a template that simplifies and speeds up the creation of components. You can create a category with pre-defined symbols, footprints, and properties, and then create components based on the category. A category not only helps in categorizing parts in the category tree, but also acts as a template at the time of creating a new component.

Adding a New Category

To create a new category, follow these steps:

1. Right-click *My Workspace* and choose *Add Category*.
The *Category* dialog box opens. This is where you create a new category.
2. Specify a name and description for the new category.

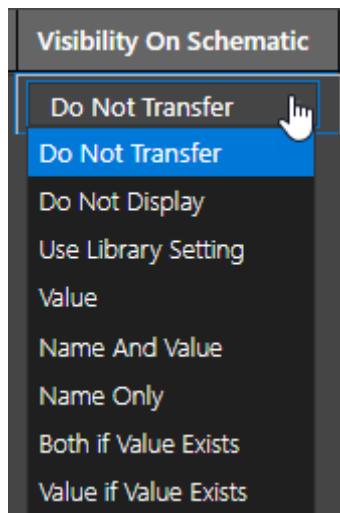


3. Select a part type from the Part Type List. The default part type, *Electrical*, is pre-selected. You can choose from the following options:
 - Electrical
 - Mechanical
4. In the *Property* tab, click the *Add* icon (plus sign) to add a new property to a category. Any component created based on this category inherits the properties defined for the category.
A blank row is added for a new property where you specify the property details

Property		Models						
	Name	Value	Description	Visibility On Schematic	Value Type			
=	<input type="text"/>	<input type="text"/>	<input type="text"/>	Do Not Transfer	String	<input type="checkbox"/>	<input checked="" type="checkbox"/>	

5. Specify a name, value, and description of the property.

A category controls how a property is transferred to the schematic. You can specify the default behavior of property transfer while defining the category in the *Visibility On Schematic* column.

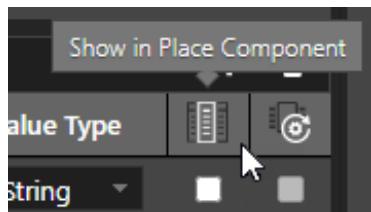


The following table provides a description of the options the *Visibility On Schematic* column provides:

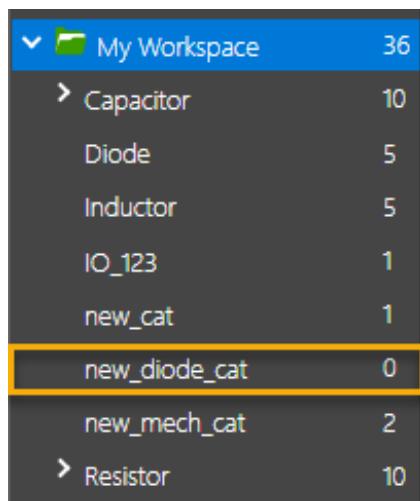
Option	Description
Do Not Transfer	The property is not annotated to the schematic design.
Do Not Display	The name or the value of the annotated property is not displayed on the schematic canvas.
Use Library Setting	With this option selected, the property is annotated to the schematic design. If the same property is also defined at the library level, its visibility is set by the display option set in the library.

Value	The value of the annotated property is displayed on the schematic canvas.
Name and Value	The name and value of the annotated property is displayed on the schematic canvas.
Name Only	Only the name of the annotated property is displayed on the schematic canvas.
Both if Value Exists	Both the name and the value of the annotated property are displayed on the schematic canvas, if the value exists.
Value if Value Exists	The value of the annotated property is displayed on the schematic canvas, if it exists.

6. To display this column alongside the component in the part browser after a component is placed, select the checkbox in the *Show in Place Component* field.



7. Select the checkbox in the *Update Part Property* field to update the part property with this property when you choose the *Link Database Component* menu command or click the *Update* button in the View Database dialog box.,
8. Click the *Models* tab to associate symbol, footprints, and PSpice models with the category.
For details see, [Creating Components](#).
9. Click *Save*.
The category is displayed under the *My Workspace* node.



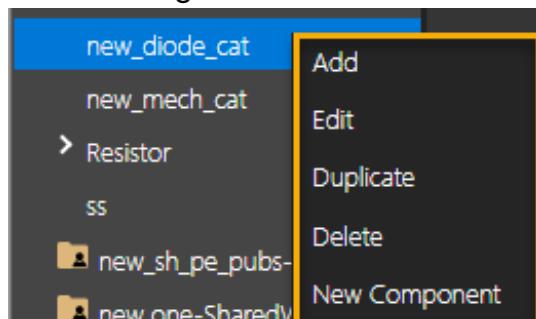
The numeral next to the category indicates the number of components created using this category.

Modifying a Category

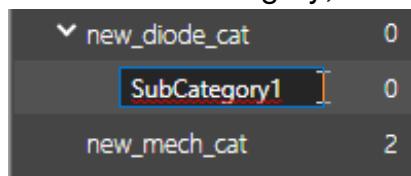
You can edit, duplicate, or delete an existing category. You can also add a subcategory to a category.

You can modify a category in the following ways:

1. Right-click the category under the *My Workspace* node and choose the desired option from the following list:



2. To add a subcategory, choose *Add* and specify a name for the subcategory.



3. To edit a category, choose *Edit*.

The *Category* dialog box opens where you can change the category details.

4. To create a copy of the category, choose *Duplicate*.

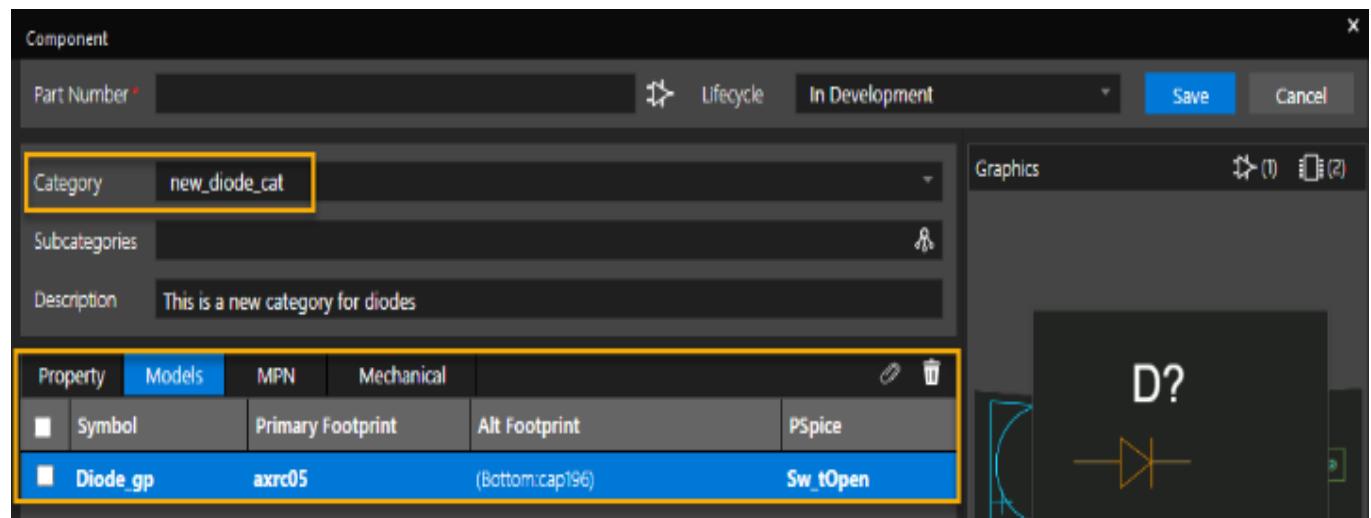
The *Category* dialog box opens where you can specify a new name for the category and save it as a new category.

5. To delete a category, choose *Delete*.

The selected category is deleted.

6. To create a new component based on the category, click *New Component*.

The *Component* dialog box opens with pre-seeded values from the category. You can specify a part number for this component and save it.



Related Topics

- [Creating Components](#)

Managing Mechanical Assemblies

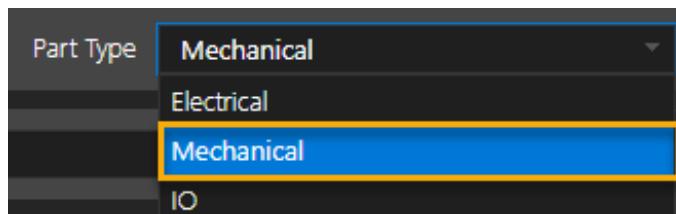
In addition to electrical components, some non-electrical components on a PCB are used for

mechanical support, such as screws and connectors. You can create categories or templates for mechanical assemblies and parts and then associate them with electrical components.

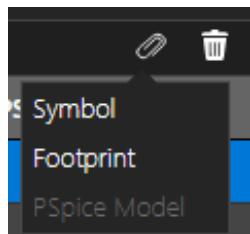
Creating a Mechanical Category

To create a mechanical category, do the following:

1. In Component Explorer, right-click the *My Workspace* node and choose *Add*.
2. In the *Part Type* field, select *Mechanical*.

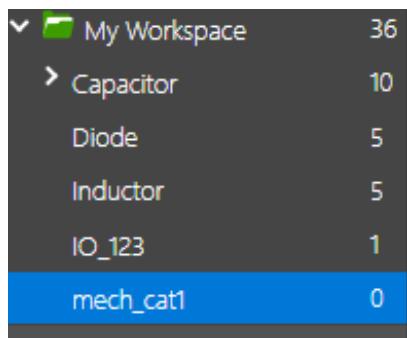


3. Specify details, such as name and description.
4. Associate mechanical symbols and footprints from the available libraries. See [Managing Categories](#).



Note that the *PSpice Model* option is deactivated for a mechanical symbol.

5. Click *Save*.
The new mechanical category is created.



Creating a New Mechanical Part

To create a new mechanical part, do the following:

1. Right-click the mechanical category and choose *New Component*.
The *Component* dialog box opens. Note the icon next to the Part Number field. It indicates that the component type is mechanical.

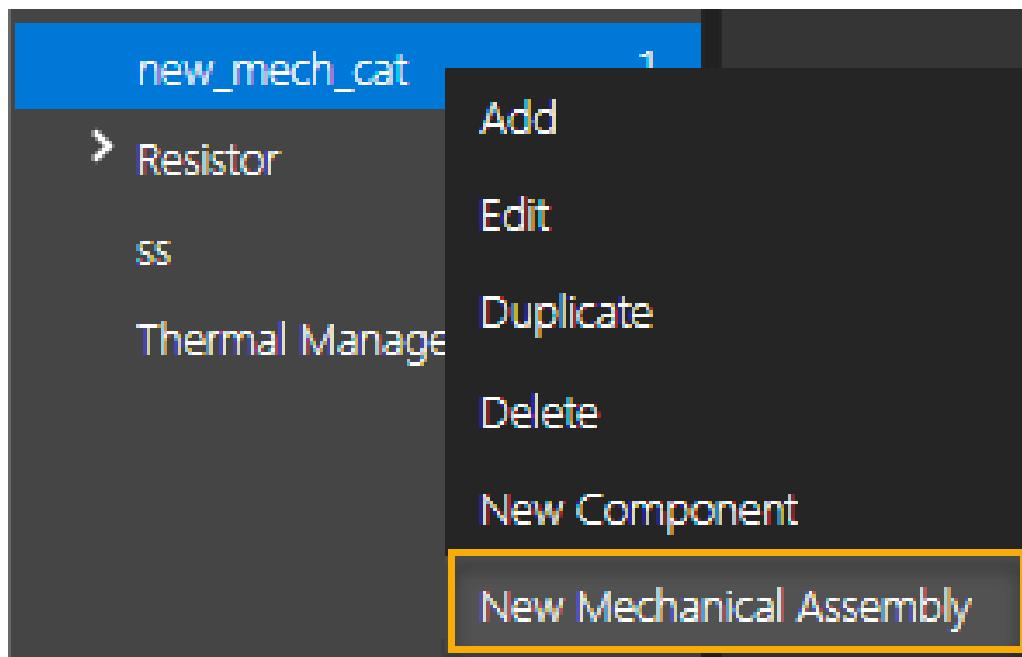


2. Specify the details to create the component. See [Creating Components](#) for details.
3. Click *Save*.

Creating a New Mechanical Assembly

To create a new mechanical assembly, do the following:

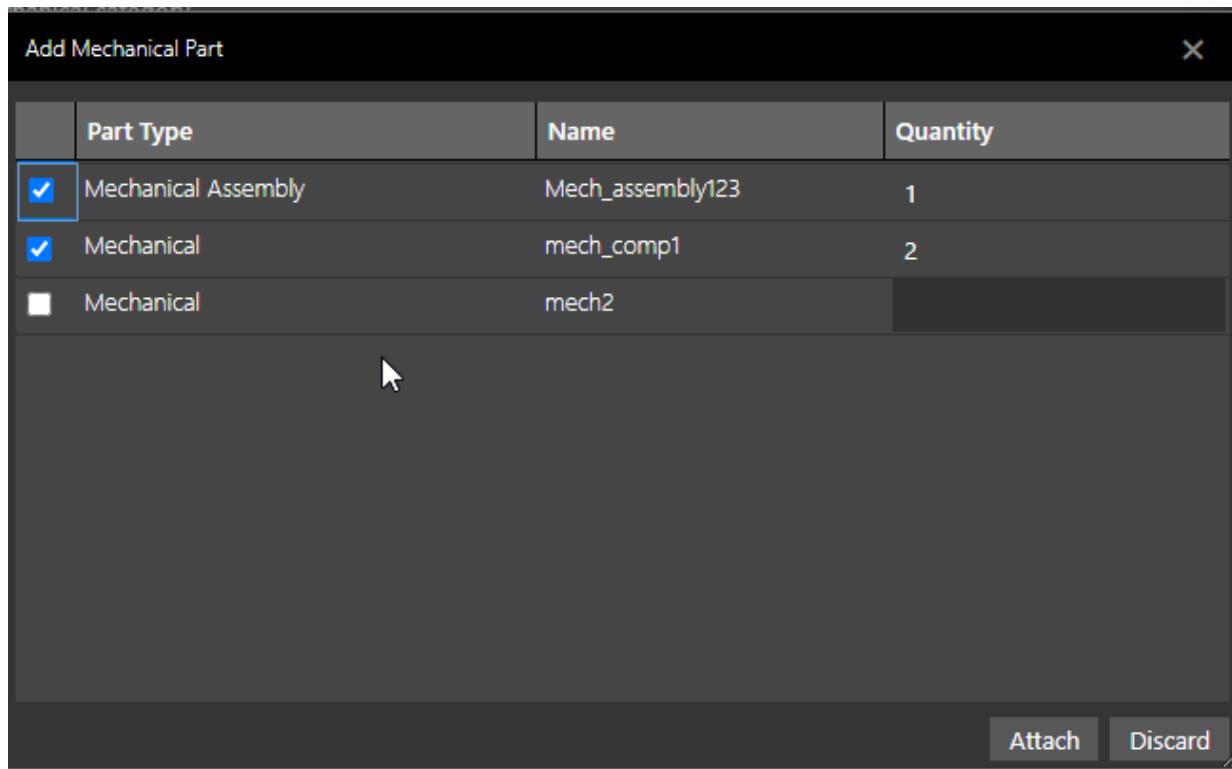
1. Right-click the mechanical category, choose *New Mechanical Assembly* to create a new mechanical assembly.



The *Component* dialog box appears where you specify the required details.

2. Click the *Mechanical* tab.
3. Click the *Add* (plus) icon.

The *Add Mechanical Part* dialog box opens. A list of available mechanical parts is displayed.



4. Select the check box to the left of *Mechanical Assembly* or *Mechanical*.
5. Specify the quantity of the mechanical parts to be included in the assembly.
6. Click *Attach*.
Selected part added in the *Mechanical* tab.
7. Click *Save*.

Related Topics

- [Creating Components](#)