

Library Authoring and Editing in Allegro® X System Capture

**Product Version 23.1
September 2023**

© 2023 Cadence Design Systems, Inc. All rights reserved.

Portions © Apache Software Foundation, Sun Microsystems, Free Software Foundation, Inc., Regents of the University of California, Massachusetts Institute of Technology, University of Florida . Used by permission. Printed in the United States of America.

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

Allegro System Capture contains technology licensed from, and copyrighted by: Apache Software Foundation, 1901 Munsey Drive Forest Hill, MD 21050, USA © 2000-2005, Apache Software Foundation. Sun Microsystems, 4150 Network Circle, Santa Clara, CA 95054 USA © 1994-2007, Sun Microsystems, Inc. Free Software Foundation, 59 Temple Place, Suite 330, Boston, MA 02111-1307 USA © 1989, 1991, Free Software Foundation, Inc. Regents of the University of California, Sun Microsystems, Inc., Scriptics Corporation, © 2001, Regents of the University of California. Daniel Stenberg, © 1996 - 2006, Daniel Stenberg. UMFPACK © 2005, Timothy A. Davis, University of Florida, (davis@cise.ulv.edu). Ken Martin, Will Schroeder, Bill Lorensen © 1993-2002, Ken Martin, Will Schroeder, Bill Lorensen. Massachusetts Institute of Technology, 77 Massachusetts Avenue, Cambridge, Massachusetts, USA © 2003, the Board of Trustees of Massachusetts Institute of Technology. All rights reserved.

Trademarks: Trademarks and service marks of Cadence Design Systems, Inc. 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. 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.

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/or 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

<u>Allegro Unified Libraries in System Capture</u>	5
<u>Allegro Unified Library Organization</u>	5
<u>Allegro Unified Library Project</u>	6
<u>Libraries Tree</u>	7
<u>Library Explorer</u>	8
<u>Creating Library in a New Library Container</u>	8
<u>Creating Library in an Existing Library Container</u>	9
<u>Renaming a Library</u>	10
<u>Deleting a Library</u>	11
<u>Adding Existing Library Container to a Project</u>	12

2

<u>Working with Symbols</u>	15
<u>Symbol Defaults</u>	16
<u>Setting Up General Symbol Defaults</u>	17
<u>Setting Up Symbol Grid Defaults</u>	18
<u>Setting Up Symbol Pin Shape and Attribute Defaults</u>	20
<u>Setting Up Pin Style Defaults</u>	21
<u>Creating a New Blank Symbol</u>	21
<u>Adding Pins using Table View</u>	22
<u>Editing Pins in Table View</u>	24
<u>Adding Pins Using Add Pin Dialog Box</u>	27
<u>Adding Low Asserted Pins</u>	32
<u>Adding Symbol Properties</u>	34
<u>Adding Notes</u>	36
<u>Adding Images</u>	36
<u>Drawing Arcs</u>	37
<u>Drawing Lines</u>	38
<u>Reusing Existing Symbols</u>	38

Library Authoring and Editing in Allegro X System Capture

<u>Renaming a Symbol</u>	39
<u>Deleting a Symbol</u>	39
<u>Editing Symbol Objects</u>	39
<u>Creation of Shapes</u>	41
<u>Creating a Custom Shape</u>	41
<u>Creating a Pin Shape</u>	43
 3	
<u>Template Libraries</u>	47
<u>Creating a New Template Library Container</u>	48
<u>Creating a New Part Template</u>	51
<u>Editing an Existing Template</u>	55
<u>Creating a Part Using a Part Template</u>	55
<u>Creating a New Symbol Template</u>	58
<u>Creating a Symbol Using a Symbol Template</u>	59
<u>Creating a Part Using a Symbol Template</u>	61
 4	
<u>Management of Library Parts</u>	63
<u>Creating Electrical Flat Parts</u>	63
<u>Creating Electrical Split Parts</u>	67
<u>Creating Symmetrical Parts</u>	72
<u>Creating Asymmetrical Parts</u>	73
<u>Creating Documentation Parts</u>	77
<u>Associating Existing Symbols</u>	78
<u>Assigning a Footprint</u>	80
<u>Assigning a Datasheet</u>	80
<u>Viewing Details of a Specific Part</u>	82
<u>Viewing All Parts in a Library</u>	85
<u>Removing Symbols from a Part</u>	86
<u>Renaming Parts</u>	87
<u>Deleting Parts</u>	88
<u>Deleting a Section</u>	88
<u>Adding Part Properties</u>	88
<u>Modifying Part Properties</u>	90

Library Authoring and Editing in Allegro X System Capture

<u>Copying a Part Across Libraries</u>	90
<u>Duplicating Parts in the Same Library</u>	92
<u>Adding Global Pins</u>	93
<u>Editing Multiple Parts in a Grid</u>	94
<u>Category Management</u>	96
<u>Creating a Category for a Specific Part</u>	96
<u>Creating a Category within a Library</u>	97
<u>Adding Parts from External Content Providers</u>	100

5

<u>Validation of Models</u>	105
<u>Validating a Part</u>	105
<u>Configuring Library Audit Settings</u>	107
<u>Configuring Library Validation Window</u>	109
<u>Exporting a Validation Report</u>	110
<u>Validating Multiple Parts</u>	110
<u>Validating a Symbol</u>	111
<u>Validating a Library</u>	113

A

<u>Working with Legacy Libraries</u>	115
<u>Importing OrCAD Capture Libraries to Unified Library</u>	115
<u>Import Log</u>	116
<u>Enabling OrCAD Library Editing in Allegro X System Capture</u>	117
<u>Changes in Imported Part Details</u>	119
<u>Changes in Pin Types</u>	119
<u>Changes in Pin Names</u>	120
<u>Duplicate Pin Names</u>	121
<u>Changes in Property Names</u>	121
<u>Properties without Values</u>	121
<u>Reserved Property Names Not Imported</u>	122
<u>Allegro DE-HDL Libraries in System Capture</u>	124
<u>Creating a New Library in DE-HDL</u>	124
<u>Editing DE-HDL Schematic Model within System Capture</u>	125

Library Authoring and Editing in Allegro X System Capture

Allegro Unified Libraries in System Capture

Allegro X System Capture is an integrated design environment where you can seamlessly work on various aspects of the design process, such as library creation and management, and schematic capture, without switching between applications. This enables creating, viewing, and editing of parts and symbols within the same session of System Capture.

The in-tool library authoring and editing functionality of System Capture is accessible through a System Capture project created in the Allegro Unified library format. A single unified library file (`.ldax`) or a *library container* can store one or more libraries. These libraries store all the part details including symbols, properties, and datasheet. With a single `.ldax` file, transferring of libraries becomes a simple task.

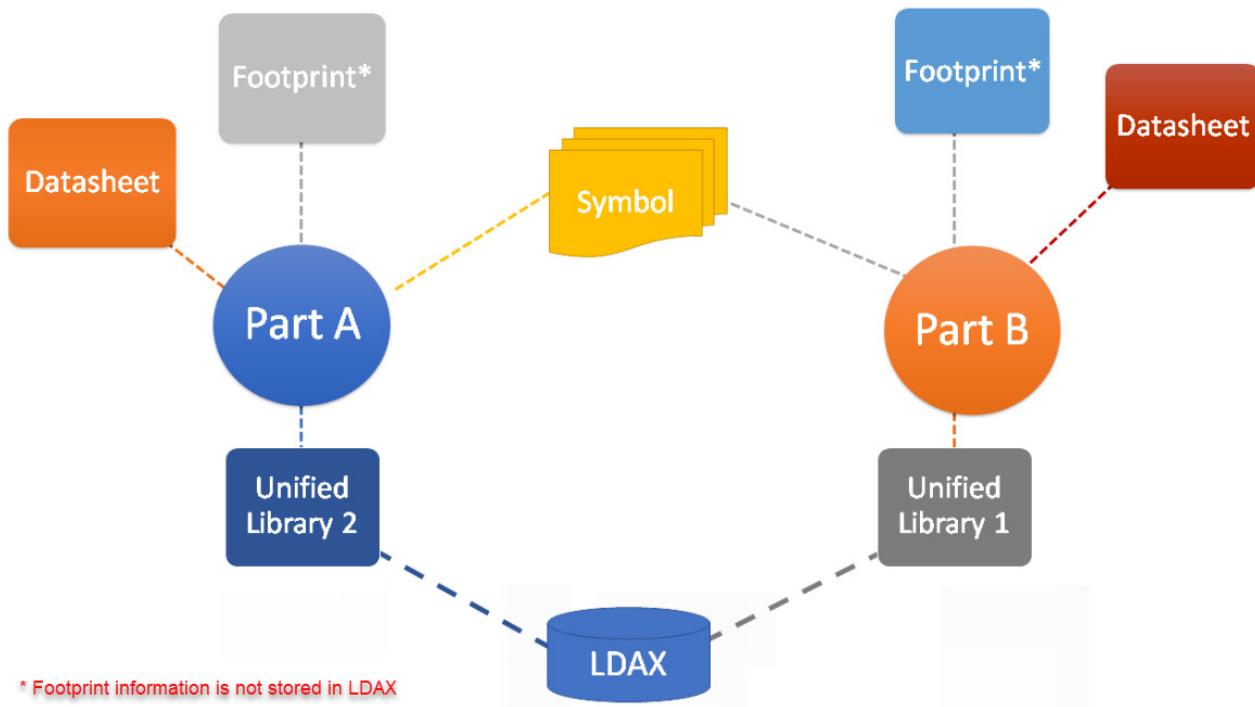
Allegro Unified Library Organization

A System Capture project created with the *Allegro Unified* library format comprises one or more `.ldax` files that can store multiple libraries. Each library in turn can contain multiple parts. A part can be associated with a symbol, a datasheet, and footprint details. Multiple parts can refer to the same symbol. While symbol details and datasheet are stored in the

Library Authoring and Editing in Allegro X System Capture

Allegro Unified Libraries in System Capture

.ldax file, the .ldax points to a .dra file for the footprint information. The following diagram describes how data is organized in a .ldax file.



Allegro Unified Library Project

When you create a project for the first time in Allegro X System Capture with the *Allegro Unified* library format, an empty library container file, `userlib`, is created. This container includes a blank library, `userlib`. On project creation, a read-only library, `standard`, is also available, which has parts for schematic authoring.

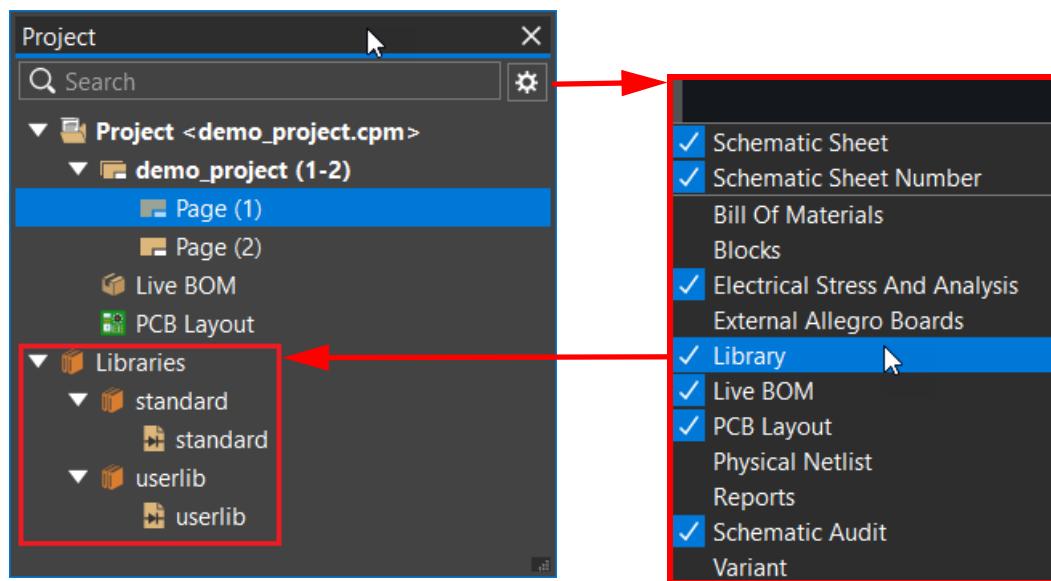
The `userlib` and `standard` libraries are available in all the new *Allegro Unified* library projects. You can create new libraries, parts, and symbols in the `userlib` library container. Additionally, you can create your own library containers and libraries in a System Capture project.

Library Authoring and Editing in Allegro X System Capture

Allegro Unified Libraries in System Capture

Libraries Tree

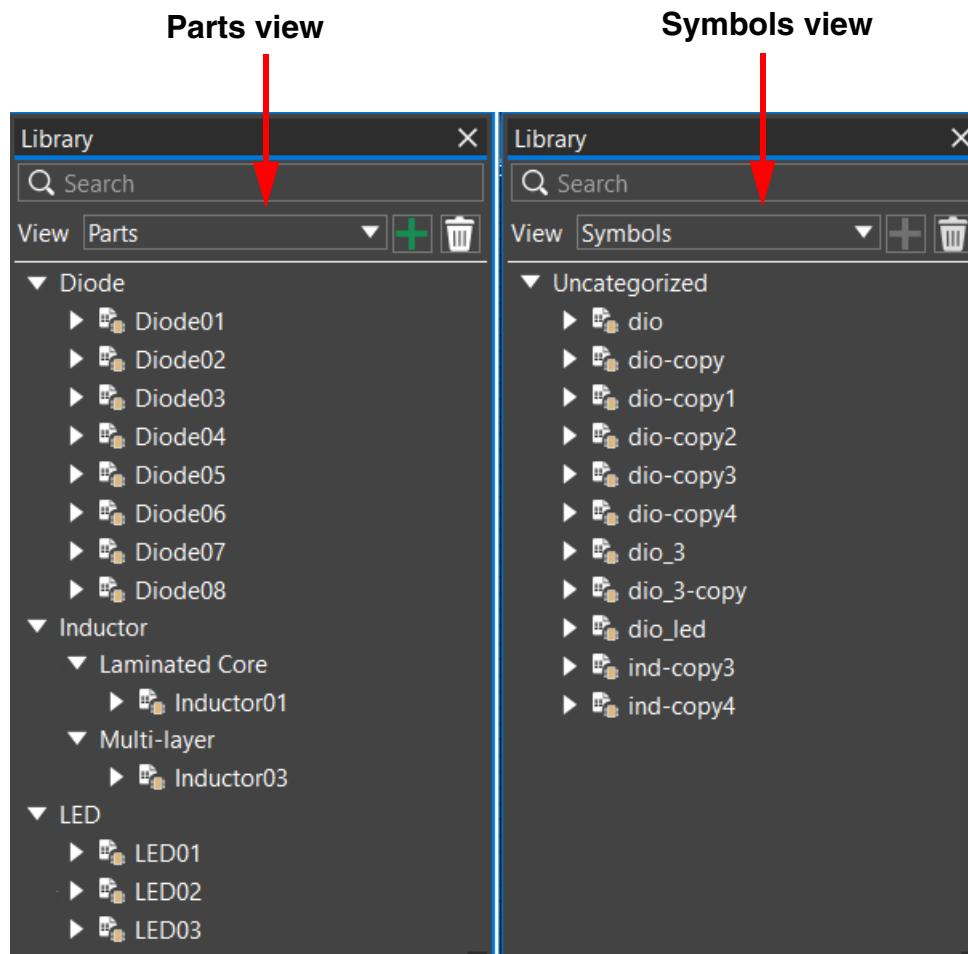
All the library containers and libraries appear under the *Libraries* tree in the *Project Explorer* panel. You can display the *Libraries* tree by selecting *Library* in *Project Explorer Settings*.



Libraries are organized library container-wise. You can expand or collapse a library container to view its libraries.

Library Explorer

When you open a library, a list of all the parts and symbols in the library are displayed in the *Library* explorer. You can switch between the *Parts* and *Symbols* views from the *View* drop-down list in the *Library* explorer.



Related Topic

- [Adding Existing Library Container to a Project](#)

Creating Library in a New Library Container

During the schematic capture phase, you might need to create new parts and symbols. The parts and symbols you create need a library to reside in. You can use the default library,

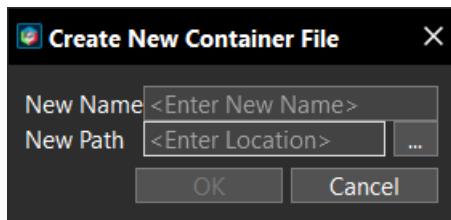
Library Authoring and Editing in Allegro X System Capture

Allegro Unified Libraries in System Capture

userlib, to add new parts and symbols, or create a new library to store them. You can add a new library to the default library container or create a new library container and create libraries in that.

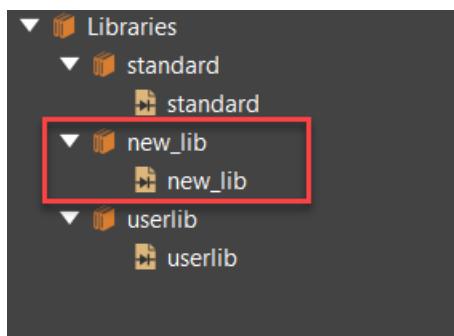
To create a Unified library in a new library container, do the following:

1. Right-click the *Libraries* tree and select *New Library Container* from the context menu.



2. Specify a name and path for the new container file and click *OK*.

A new library container and a blank library with the same name is added to the project.



Related Topic

- [Adding Existing Library Container to a Project](#)

Creating Library in an Existing Library Container

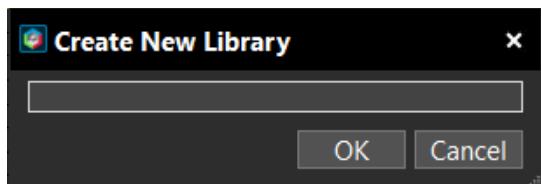
To create a new Unified library in an existing library container, do the following:

1. Right-click an existing library container in the *Libraries* tree and select *New Library*.

Library Authoring and Editing in Allegro X System Capture

Allegro Unified Libraries in System Capture

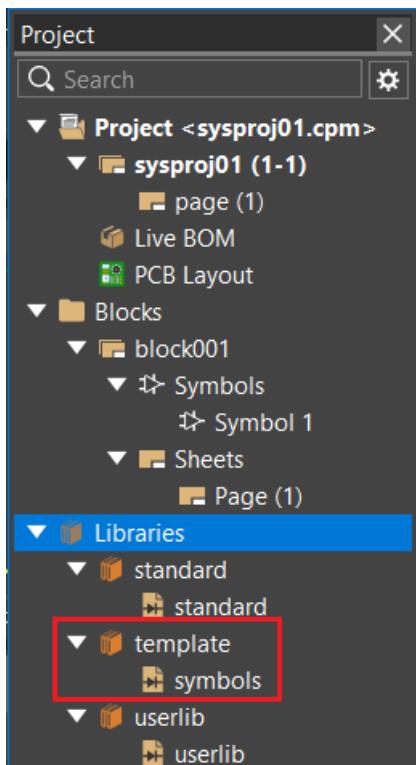
The *Create New Library* dialog box appears.



- Specify a name for the new library to be created and click *OK*.

Note: A user library should not be named `worklib` because a reserved library exists with the same name.

The library is created and displayed in the *Libraries* tree.

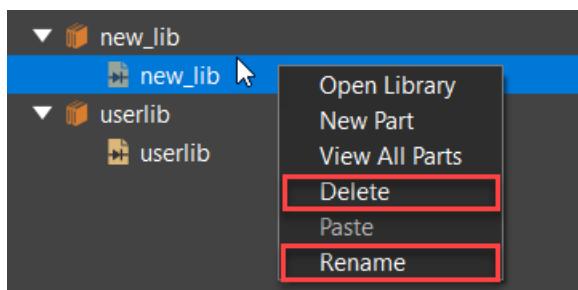


Renaming a Library

At any time, you can rename or delete a library from the *Libraries* tree.

To rename a library, do the following:

1. Right-click the library in the *Libraries* tree and select *Rename*.



Alternatively, you can select the library and either press F2 or click the library name.

2. Specify a new name and press TAB or ENTER.

The library is renamed as specified.

Deleting a Library

To delete a library, do the following:

1. Right-click a library in the *Libraries* tree and select *Delete*.
2. Click Yes in the confirmation box.

The library is deleted from the library container.

Adding Existing Library Container to a Project

To reuse parts from existing Unified libraries in a project, you can add the library container to the project in the *Preferences* dialog box.

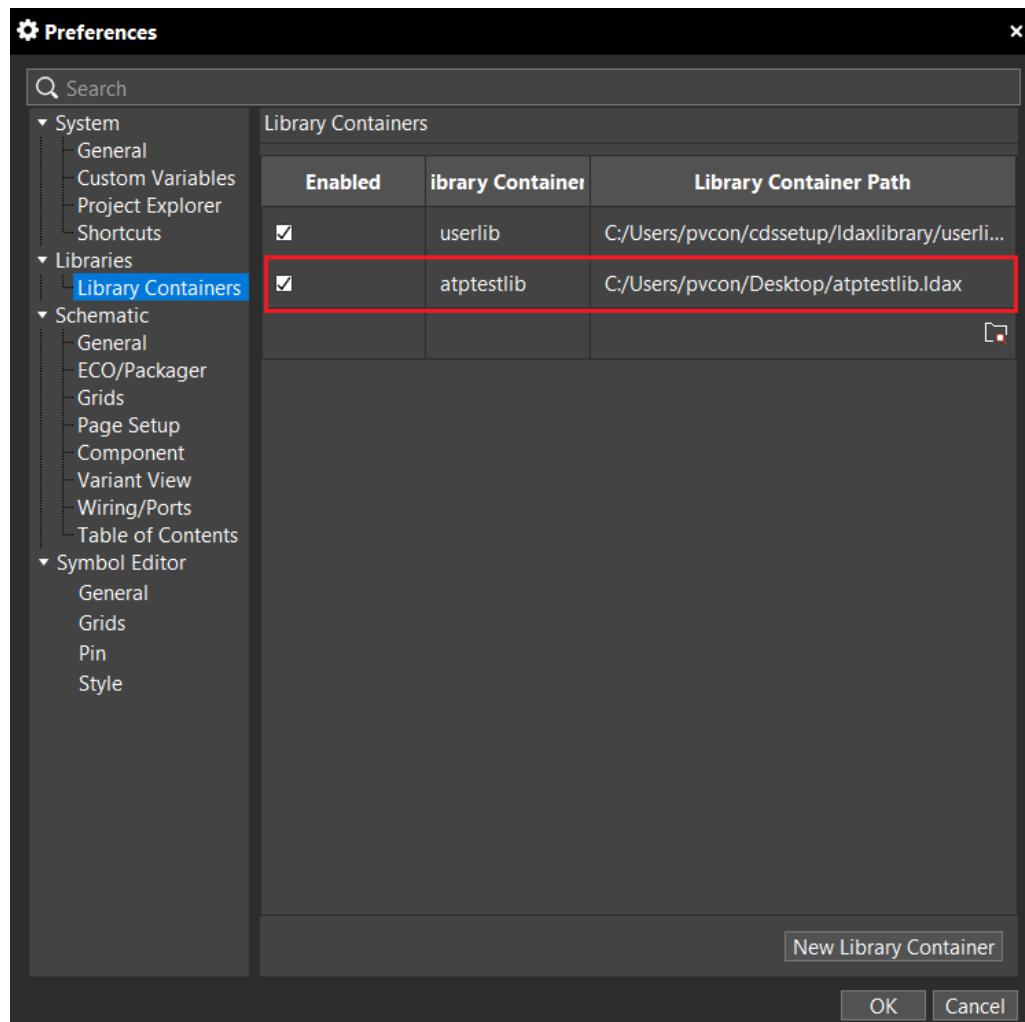
To add an existing library container to the current System Capture project, do the following:

1. Select *Edit – Preferences*.

The *Preferences* dialog is displayed.

2. Click *Libraries – Library Containers*.

3. Click  and select one or more library container files (.ldax).

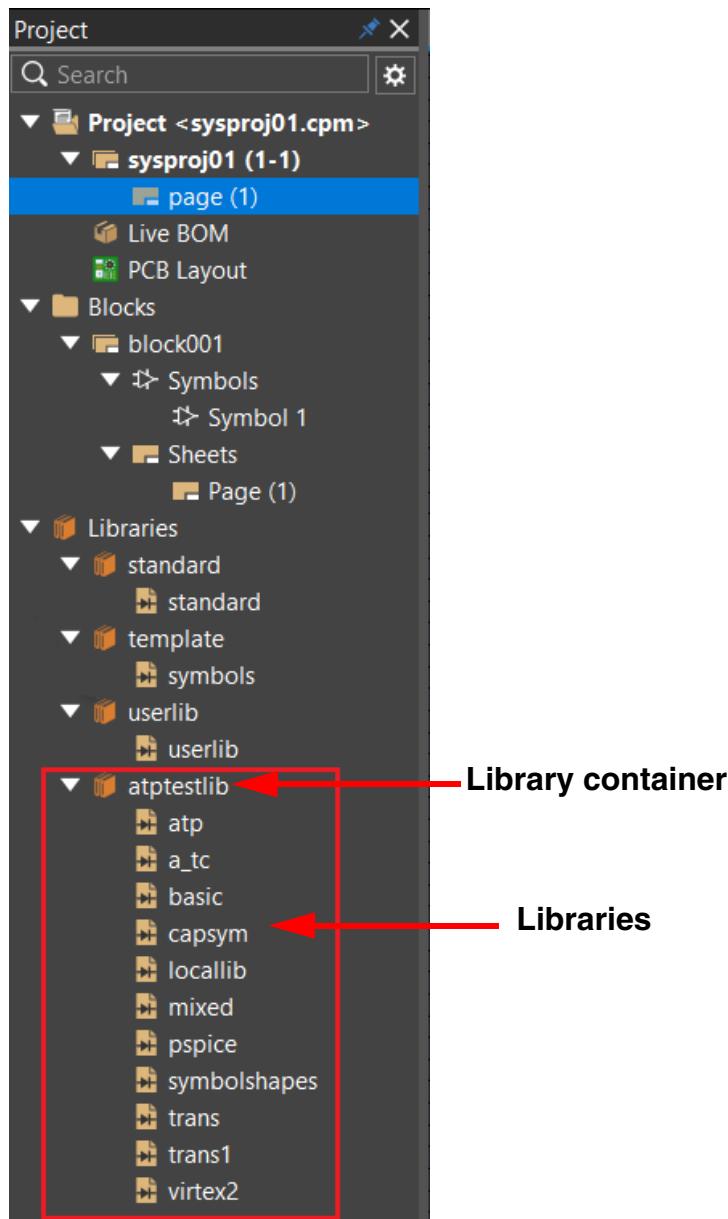


4. Click *OK*.

Library Authoring and Editing in Allegro X System Capture

Allegro Unified Libraries in System Capture

The selected .ldax file is added to the *Libraries* tree as a library container.



All the libraries and the parts and symbols contained in them are now available for placement on the schematic from *Unified Search*.

Library Authoring and Editing in Allegro X System Capture

Allegro Unified Libraries in System Capture

Working with Symbols

Logical symbols are graphical representation of parts that include the shape of a part and one or more pins. Allegro X System Capture provides a graphical user interface to create new symbols and edit existing symbols by adding pins, properties, and custom shapes.

You can create a new symbol while creating a part or independently, from the *Symbols* view of the *Library* explorer.

Creating symbols independently comes in handy when you want to create symbols and associate them with new or existing parts later. Regardless of how you initiate symbol creation, the tasks and steps involved remain the same.

Creating new symbols involves the following tasks:

- [Creating a New Blank Symbol](#)
- Adding Pins
 - [Adding Pins using Table View](#)
 - [Adding Pins Using Add Pin Dialog Box](#)
- [Adding Low Asserted Pins](#)
- [Adding Symbol Properties](#)
- [Adding Notes](#)
- [Adding Images](#)
- [Drawing Arcs](#)
- [Drawing Lines](#)

You can also create a duplicate view of a symbol using an existing symbol.

Related Topics

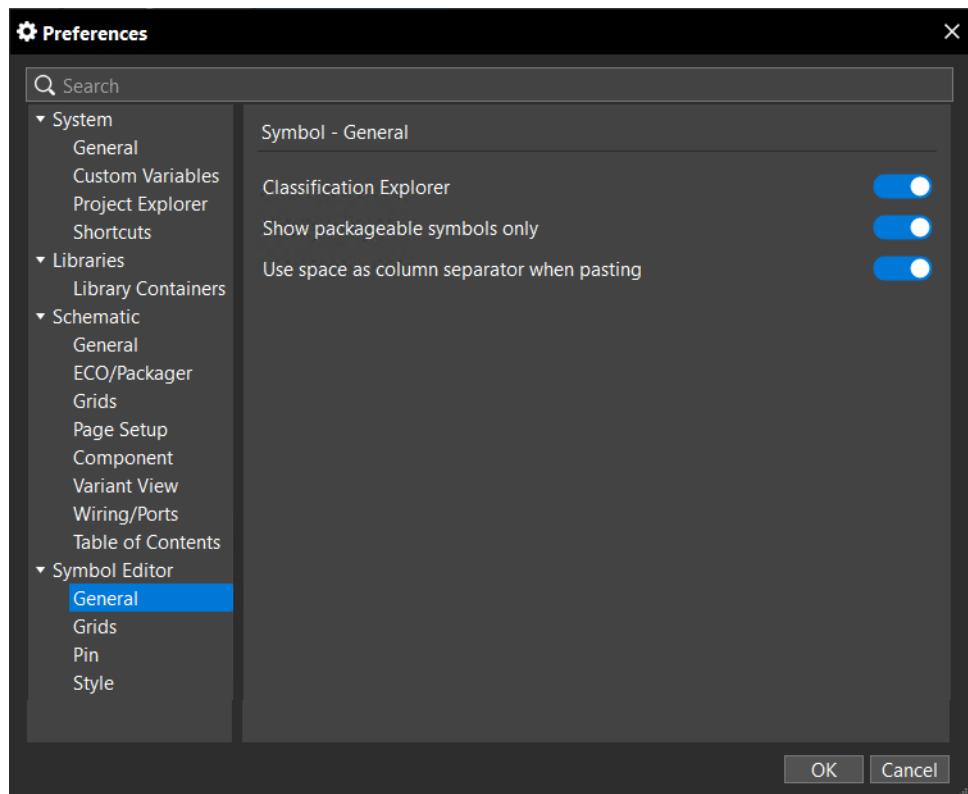
- [Reusing Existing Symbols](#)

■ Creation of Shapes

Symbol Defaults

Before creating or associating existing symbols with parts, you need to configure default values for symbols, such as pin shapes and grid settings. You can also configure the default pin and text style settings to change the color and pattern of pin, text, and graphics.

You set up these settings in the Symbol Editor section of the *Preferences* dialog box.



Library Authoring and Editing in Allegro X System Capture

Working with Symbols

You can configure symbol pin and properties settings, such as `Symbol_Units` as `Metrics`, and so on from `site.cpm` file available in `<your_inst_dir>/share/cdssetup/projmgr` directory.

```
446 START_PDV
447 Symbol_Units 'Metrics'
448 Symbol_OutLine 'Thin'
449 Symbol_Width '10'
450 Symbol_Length '10'
451 Symbol_SymSheetSize 'A'
452 Symbol_MaxSymSize '-18000,18000,18000,-18000'
453 Symbol_AutoExpandBus '0'
454 Symbol_GridSize '0.050000'
455 Symbol_Text_Height '25'
456 Symbol_Text_Color '0'
```

Setting Up General Symbol Defaults

The *Library* explorer lists all the parts organized by categories. The *Category* view is enabled by default for all projects. You can change this view from the *Preferences* dialog box.

To disable the *Category* view, do the following:

1. Select *Edit – Preferences*.

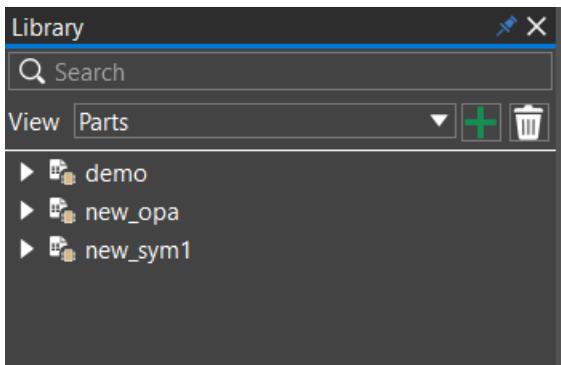
The *Preferences* dialog box is displayed.

2. Click *Symbol Editor — General* to display the *Symbol – General* page.
3. Toggle on *Classification Explorer*.

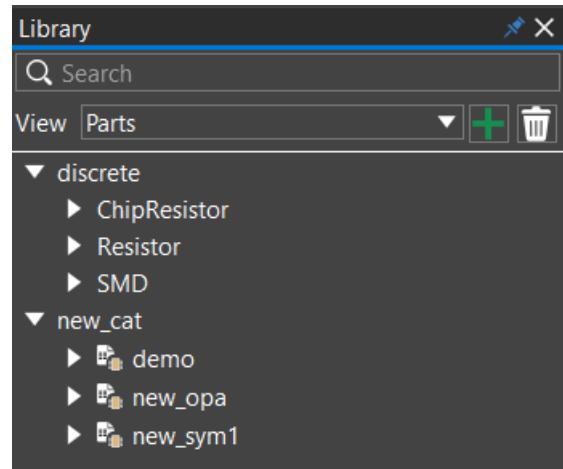
Library Authoring and Editing in Allegro X System Capture

Working with Symbols

4. Click OK.



Classification Explorer Off



Classification Explorer On

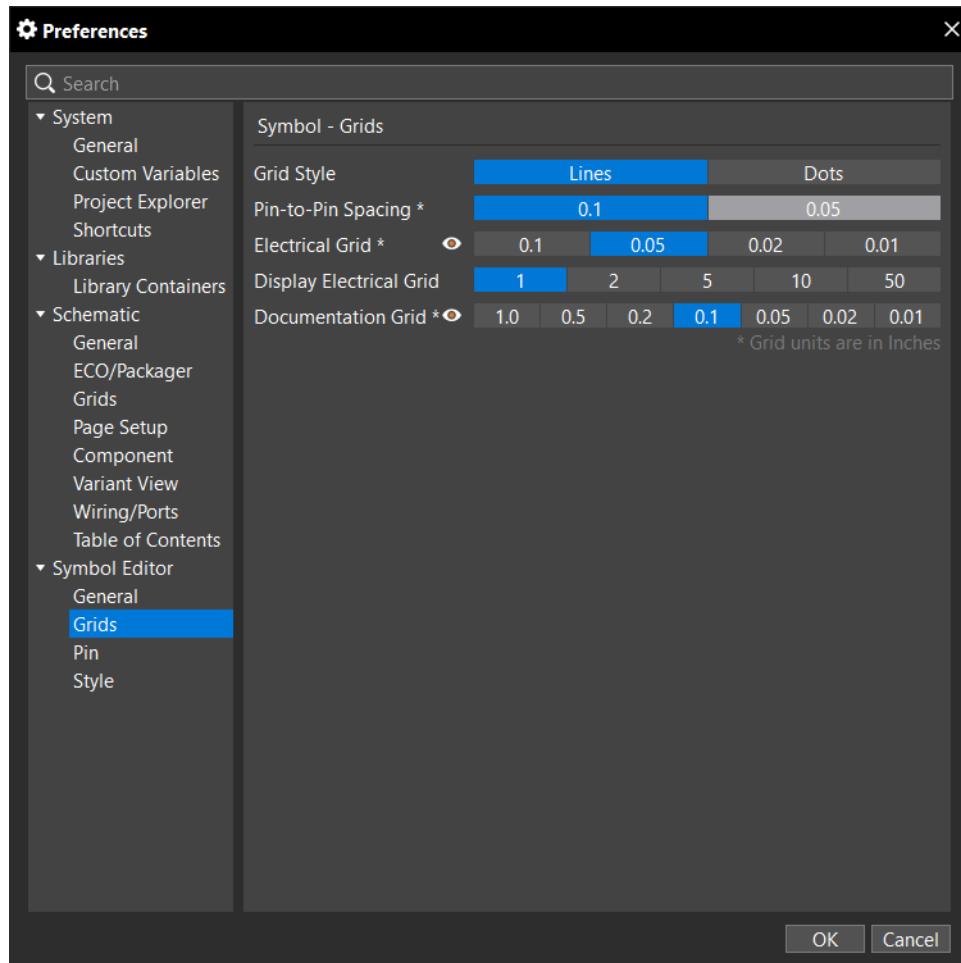
Setting Up Symbol Grid Defaults

System Capture ensures correct connectivity of symbols and pins by ensuring that all the symbol objects are always placed on the grid to avoid any off-grid components in a design.

Library Authoring and Editing in Allegro X System Capture

Working with Symbols

You can configure grid settings, such as grid units, grid style, and grid spacing, from the *Grids* page of the *Preferences* dialog box.



The System Capture canvas supports two types of grids:

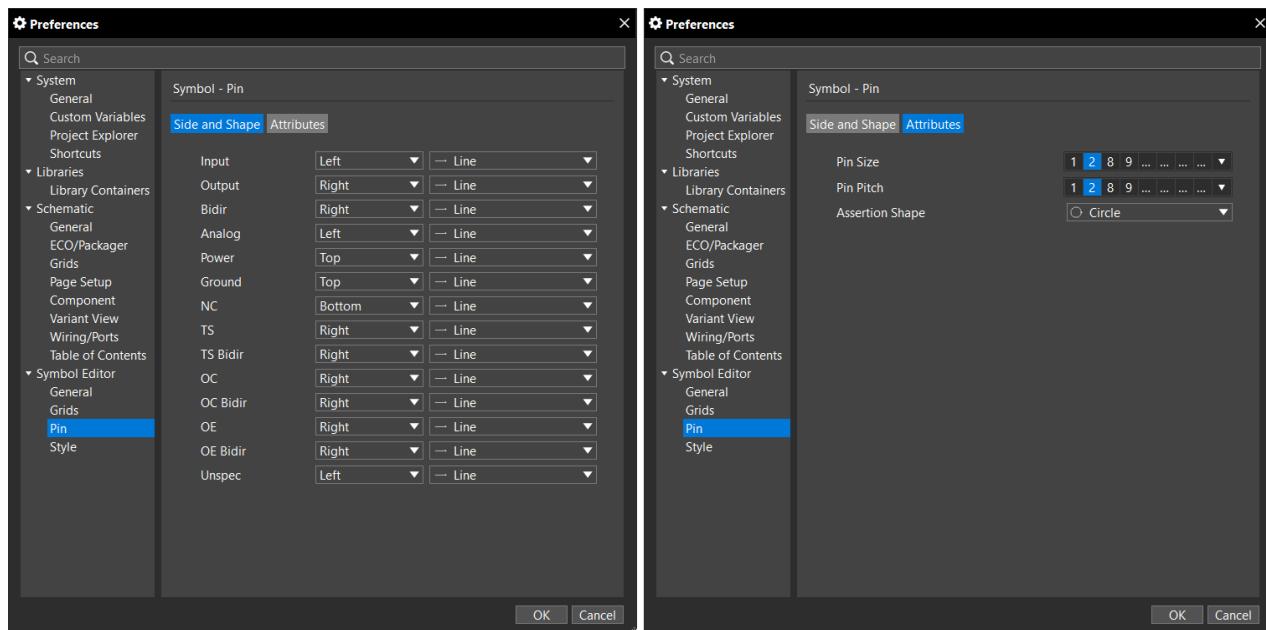
- Electrical Grid
- Documentation Grid

Library Authoring and Editing in Allegro X System Capture

Working with Symbols

Setting Up Symbol Pin Shape and Attribute Defaults

You can set the default pin settings, such as the placement location of different pin types on the symbol outline, pin shape, size, pitch and assertion shape from the *Pins* page in the *Preferences* dialog box.



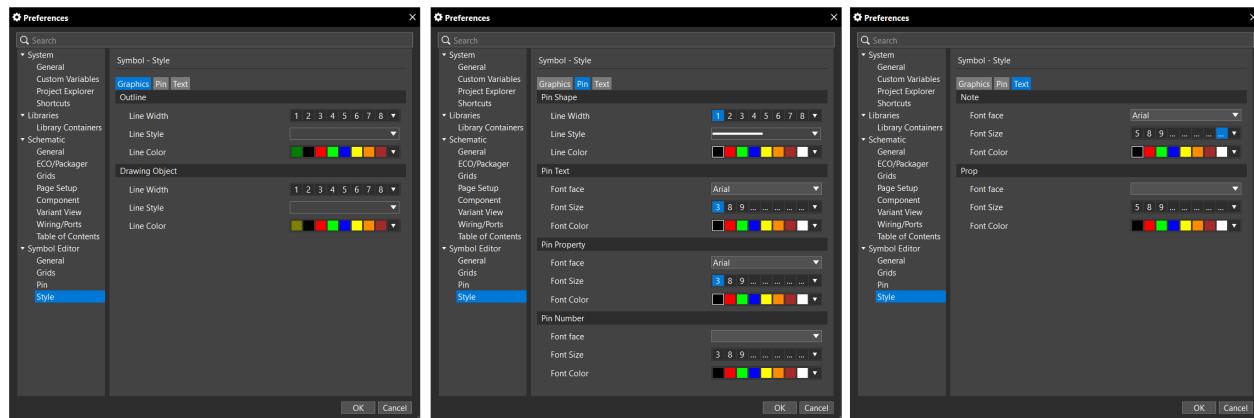
When you save the settings in the *Preferences* dialog box, the changes are updated in the *.cpm* file. You will get the same settings as default values when you add a pin from the *Add Pin* dialog box. You can make the changes when adding a pin to a symbol, if required.

Library Authoring and Editing in Allegro X System Capture

Working with Symbols

Setting Up Pin Style Defaults

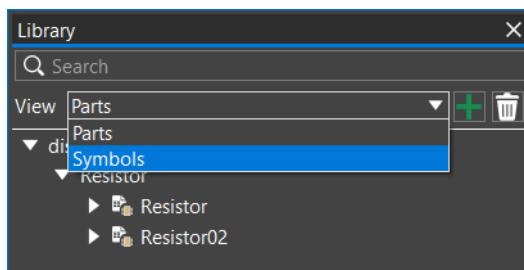
You can change the size, color, and style of pin number, pin property, pin text, and pin shape from the *Pin* tab of the *Style* page. The color and size of drawing objects, notes, and symbol properties can also be changed from the *Graphics* and *Text* tabs.



Creating a New Blank Symbol

Creating new symbols involves adding pins, properties, notes, graphics. To create a new symbol, do the following:

1. Right-click the library in the *Libraries* tree and select *Open Library*.
The *Library* explorer displays a list of available parts in the library.
2. Select *Symbols* from the *View* list.

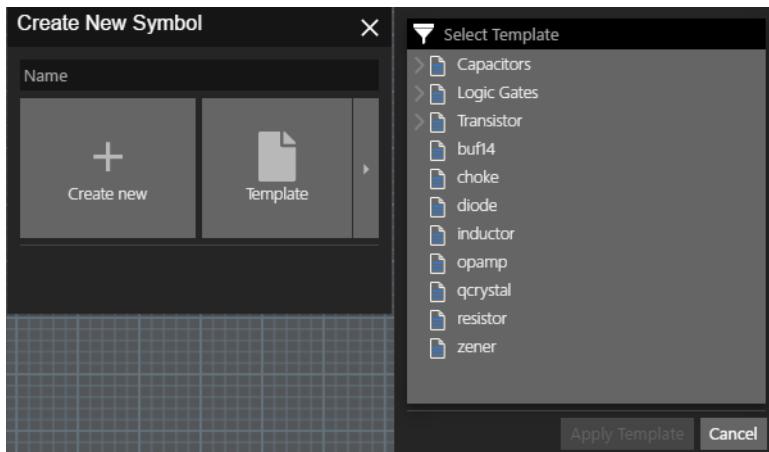


3. Click the *New Symbol* () button.

Library Authoring and Editing in Allegro X System Capture

Working with Symbols

A new symbol window opens and the *Create New Symbol* dialog box is displayed.



4. Type the name of the symbol in the *Name* field and do one of the following:
 - Click *Create new* to create a blank symbol.
A symbol boundary is added to the symbol canvas where you can add pins, properties, and objects to the symbol.
 - Click the *Template* button, select a template from the list, and click the *Apply Template* button.

When you create a new symbol from a template, symbol- and package-level properties are automatically associated with the symbol.

Related Topics

- [Adding Pins using Table View](#)
- [Adding Symbol Properties](#)
- [Adding Notes](#)
- [Drawing Arcs](#)
- [Drawing Lines](#)

Adding Pins using Table View

In Allegro X System Capture, you can add single scalar pins, multiple scalar pins, and vector pins to a symbol. You can add pins to a symbol using *Table View* in the symbol editor of System Capture that displays all the pins added to the symbol.

Library Authoring and Editing in Allegro X System Capture

Working with Symbols

To add pins using *Table View*, do the following:

1. Click  on the toolbar.

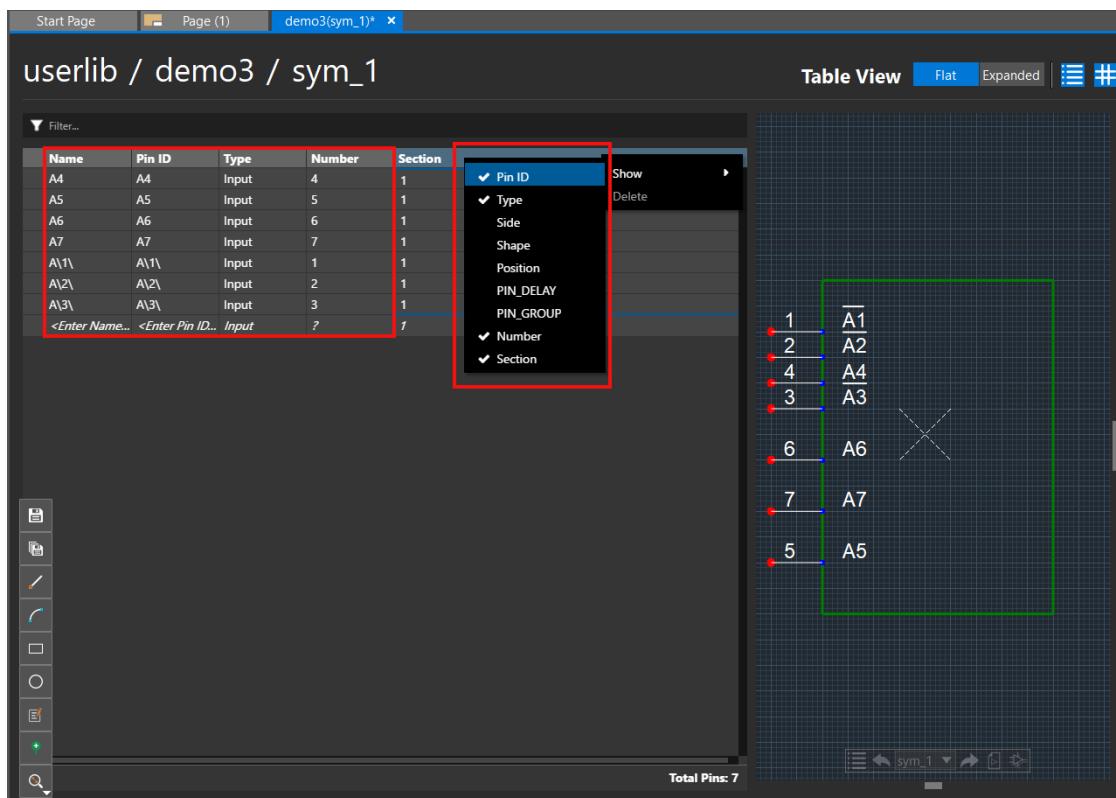
Table View is displayed.

Note: If a greater than sign is displayed in the toolbar after the part name, reduce the size of the *Project Explorer*, *Library Explorer*, or *Properties* panels to display *Table View* and the canvas visibility buttons.

2. Do one of the following to add pins:

- Click a blank cell in the *Name* column, type the pin name, and press *Enter*.
- Add pins by copying a row or by copying multiple rows from a datasheet table. To add a pin by copying and pasting, select one or more rows that you want to copy and press *Ctrl + V* to paste the copied rows.

Pins are added to the symbol boundary on the canvas, and are displayed in the table. By default, the pin type is selected as *Input* and *Pin ID* is the same as *Name*.

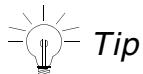
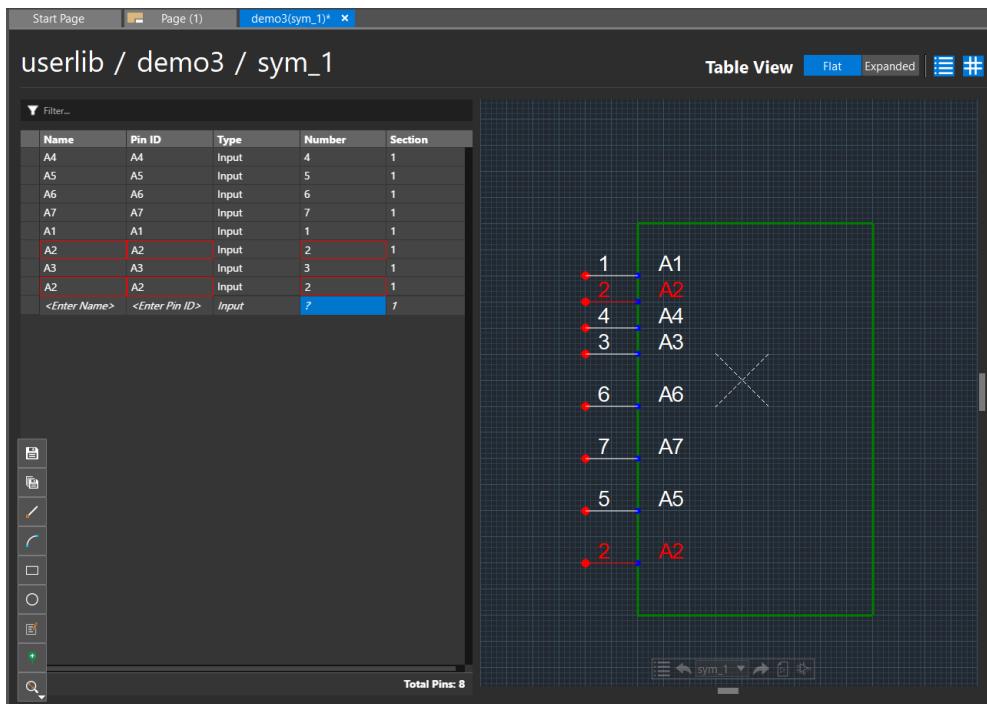


When pasting a row copied from *Table View*, all the details are the same as the copied pin. An error message appears in the *Violation* window prompting you that duplicate pin names and pin numbers are specified for multiple pins. System Capture also highlights

Library Authoring and Editing in Allegro X System Capture

Working with Symbols

the cells in red in the table. To resolve this error, you must change the name and pin number of the newly added pin.



Tip
To quickly add a scalar pin, double-click the symbol outline or an existing pin. To add multiple scalar pins, you can click a pin and keeping the *Shift* key pressed, click and drag the selected pin.

Editing Pins in Table View

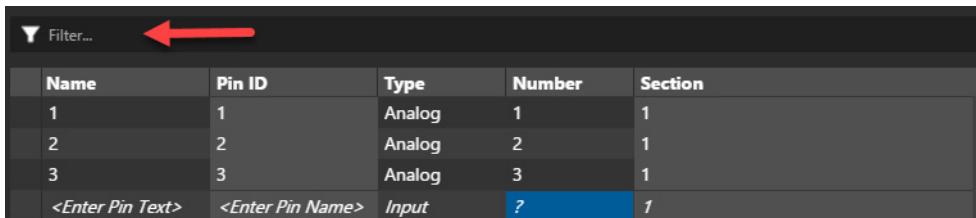
In *Table View*, you can filter pins and delete a single or multiple pins.

- To filter pins, type a string in the *Filter* field.

Library Authoring and Editing in Allegro X System Capture

Working with Symbols

The matching rows are displayed. For example, if you type *out* in the Filter field, only the rows with the string *out* in any of the columns are displayed.



Filter...				
Name	Pin ID	Type	Number	Section
1	1	Analog	1	1
2	2	Analog	2	1
3	3	Analog	3	1
<Enter Pin Text>	<Enter Pin Name>	Input	?	1

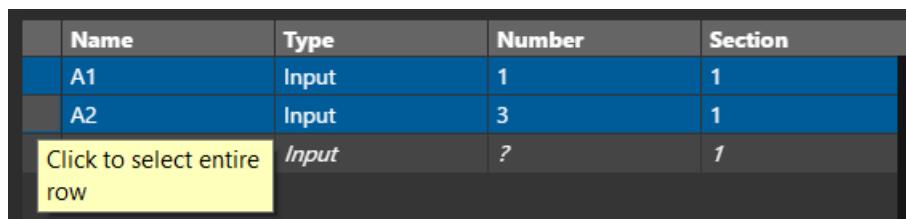
- To delete pins from a symbol, right-click a pin or select multiple pins and select *Delete*. The selected pins are deleted from the symbol.

Adding Repeated Pins in Table View

In *Table View*, you can add multiple pins to a symbol in one go by repeating the pattern of the existing pin data, such as pin name, number, or both. The pin names and numbers of the resultant pins are based on the pattern of the selection — name, number, or both.

To repeat a scalar pin array data pattern, do the following:

- In the spreadsheet table, select the data to be repeated. You can select the:
 - Entire row
 - Name* cell
 - Number* cell

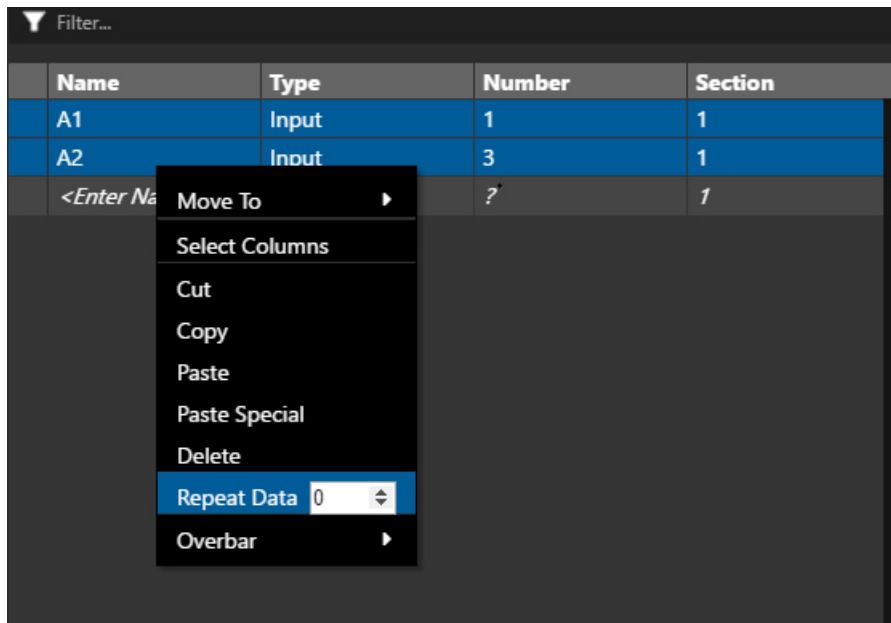


Name	Type	Number	Section
A1	Input	1	1
A2	Input	3	1
Click to select entire row	Input	?	1

Library Authoring and Editing in Allegro X System Capture

Working with Symbols

2. Right-click the selected cells and select *Repeat Data*.



3. Specify a value in the *Repeat Data* field.

This value determines the number of times the selected rows are extended.

4. Press Enter.

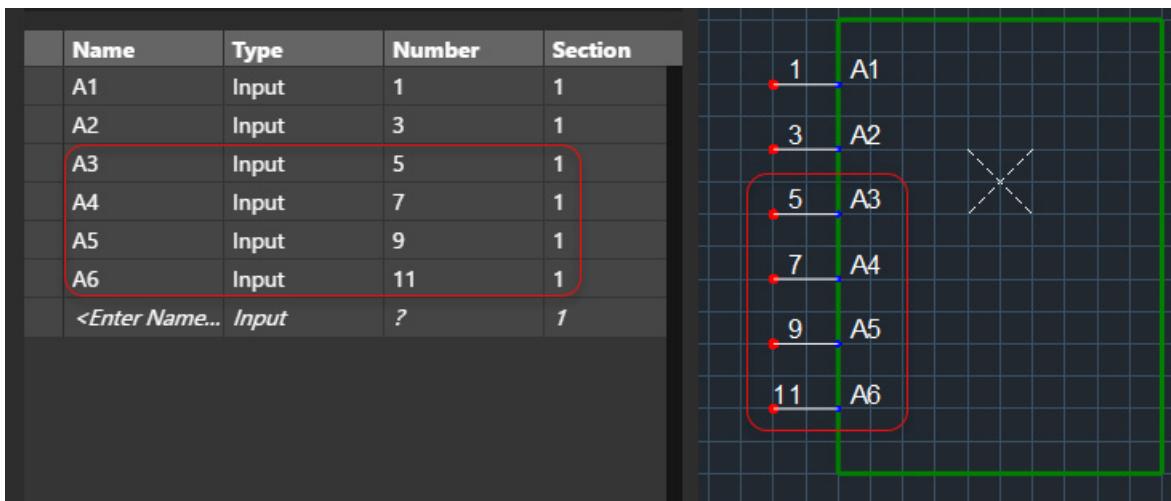
In the example illustrated here, because the value specified in the *Repeat Data* field is 2, the selected pins, A1 and A2, are repeated twice in the resultant array.

The pin name pattern is in the following sequence: A3, A4, A5, A6. The pin numbers are skipped by 5, 7, 9, 11 which is the same as the pattern of the selected pins.

Library Authoring and Editing in Allegro X System Capture

Working with Symbols

The repeated pins are displayed in the spreadsheet view and are also added to the symbol on the canvas.



Adding Pins Using Add Pin Dialog Box

Another convenient way of adding pins to a symbol is using the *Add Pin* dialog box.

Use the *Add Pin* dialog box to add scalar and multi-scalar pins and vector buses and bits to a symbol.

Adding Scalar Pins to Symbols

You can add a single or multi-scalar pins to a symbol using the *Add Pin* dialog box. When adding multi-scalar pins, you can increment the pin numbers by a desired number.

To add a scalar or multi-scalar pin, do the following:

1. Click the *Add Pin* () button on the floating toolbar.

The *Add Pin* dialog box is displayed.

2. In the *Add Pin* dialog box, switch *Style* to *Scalar*.
3. Enter the name of the pin in the *Name* field.
4. For a multi-scalar symbol, specify the range of pins in the *From* and *To* fields.

Ignore this field for a single pin.

Library Authoring and Editing in Allegro X System Capture

Working with Symbols

- This range of the pin array is added with the pin numbers in the specified range.
Example: 1 to 5 to add a total of 5 pins
 - In case of single scalar pin addition, this range remains unspecified.
 - In case of multi-scalar pins, specify the range in the *From* and *To* fields.
5. Specify the *Suffix* to be added at the end of the pin name, if required.
6. Specify the pin number in the *Number* field.
- This pin number is assigned to the first pin name in the defined range of the pins.
- In case of multi-scalar pins, the pin number is incremented till the defined range is exhausted.
- On subsequent launches of this dialog box, the value in this field is one more than the last pin added.
7. Specify an incremental value of the pin in the *Increment* field, if required.
- In case of multi-scalar pins, specify an incremental value of the pin. This value specifies the difference between the two consecutive pin numbers added in a sequence.
 - By default, subsequent pin numbers are incremented by 1. This value cannot be changed for a single pin.
8. Modify the default value of the *Spacing* field, if required. The value added to the *Spacing* field specifies the spacing between the pins on the canvas. The default value is 1.
9. Select the pin *Type* and *Shape*, as required.

10. Click *Add*.

The pin is attached to the cursor.

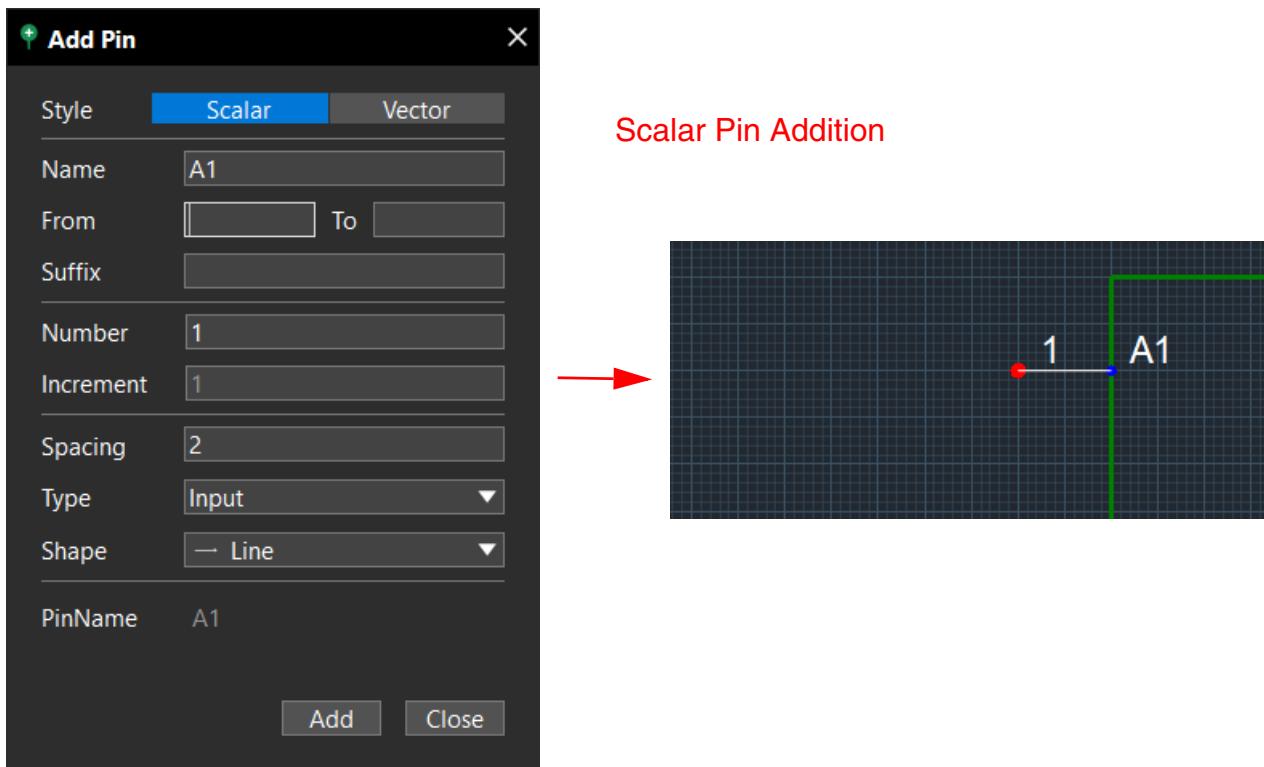
11. Align the pin to the symbol outline and click the pin to attach it to the symbol.

The scalar pins are added to the symbol.

Library Authoring and Editing in Allegro X System Capture

Working with Symbols

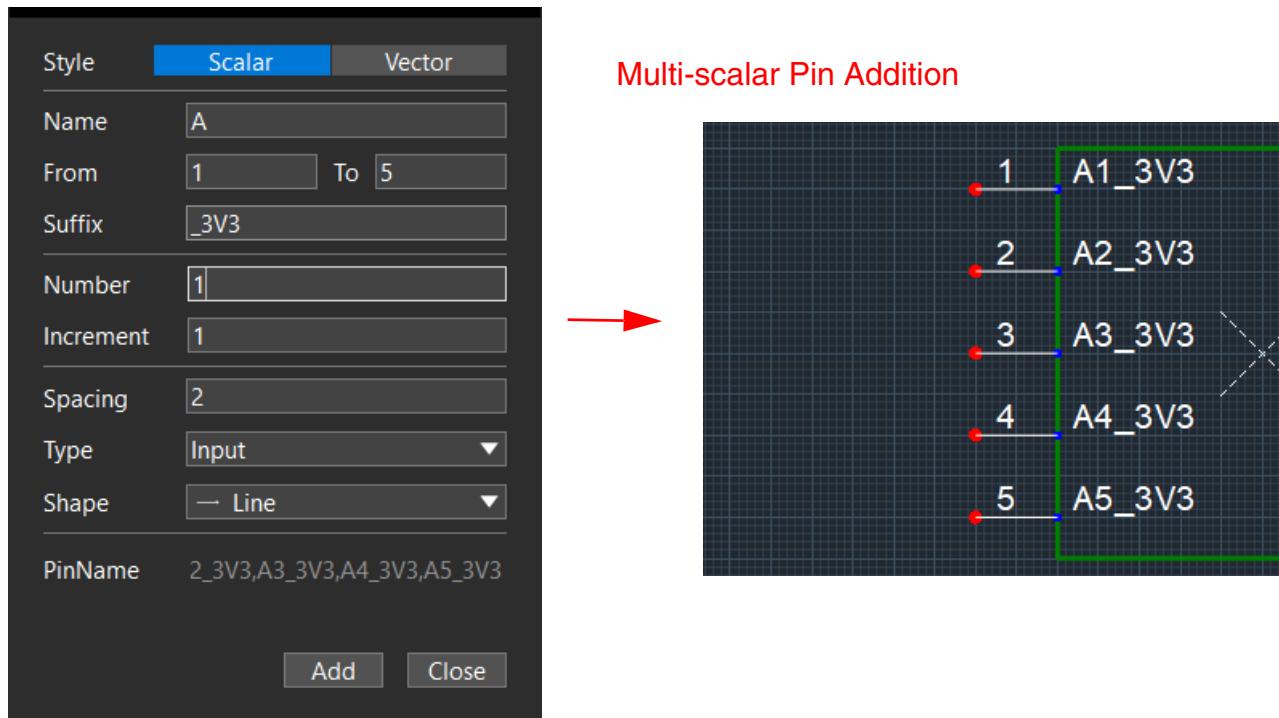
The following figure shows the addition of a single scalar pin to a symbol:



Library Authoring and Editing in Allegro X System Capture

Working with Symbols

The following figure shows the addition of multi-scalar pins to a symbol:



Adding Vector Buses or Bits to Symbols

You can also add a vector bus or vector bits with a thousand pins in a package that might include a group of pins.

To add a vector bus or vector bits to a symbol, do the following:

1. Click the *Add Pin* () button on the floating toolbar.

The *Add Pin* dialog box is displayed.

2. In the *Add Pin* dialog box, switch *Style* to *Vector*.
3. Specify the name of the vector pins in the *Name* field.
 - a. To add a vector bus, specify the name along with the range of the bus in the *Name* field. Example - `A<1..5>`.
 - b. To add vector bits, specify an alphabetic or alphanumeric name in the *Name* field. Example: A, B, A1, B1
4. Specify the range in the *From* and *To* fields to add vector bits.

Library Authoring and Editing in Allegro X System Capture

Working with Symbols

This range of the vector bits is added with the pin name representing the number of bits to be added. Example: 1 to 5 to add a total of 5 vector bits

The *From* and *To* fields are disabled automatically for a vector bus.

The *Suffix* field also remains inactive for vector pins and vector buses.

5. Specify the range of the pin numbers in the *Number* field.

For example, 1 . . 5 translates to the vector pin numbers 1 , 2 , 3 , 4 , 5.

Both, the range and comma-separated values, are supported in the *Number* field.

The *Increment* field is disabled for vector pins.

6. Modify the value in the *Spacing* field, if required.

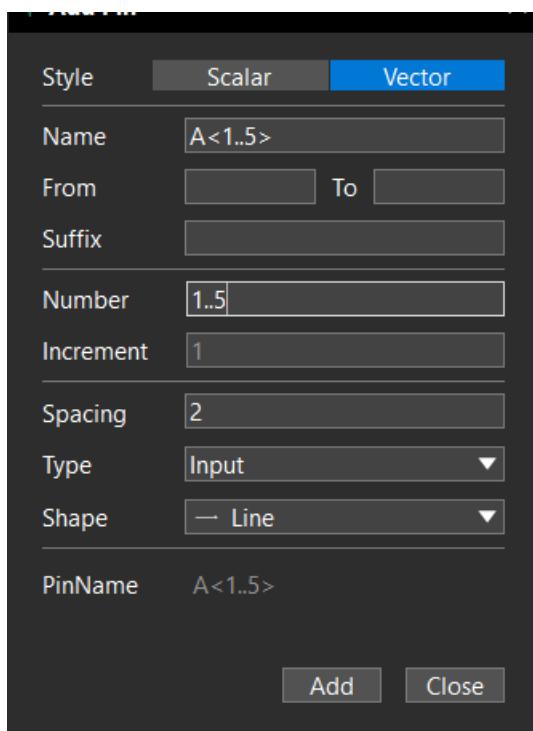
7. Select the pin *Type* and *Shape*, as required.

8. Click *Add*.

The pin is attached to the cursor.

9. Align the pin to the symbol outline and click the pin to attach it to the symbol.

The following figure shows a vector pin added to a symbol:



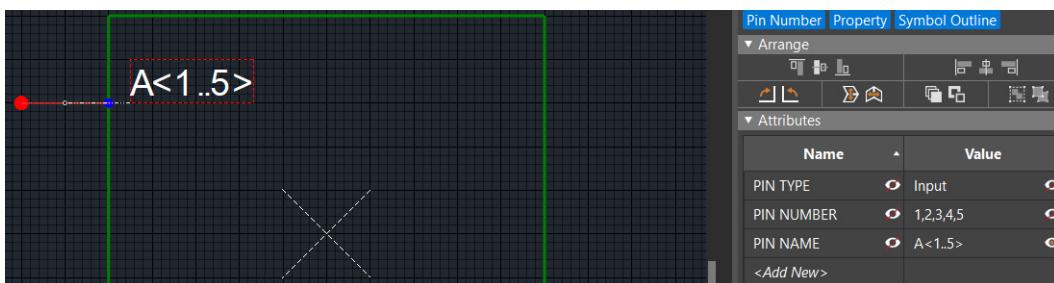
Vector Pin Addition



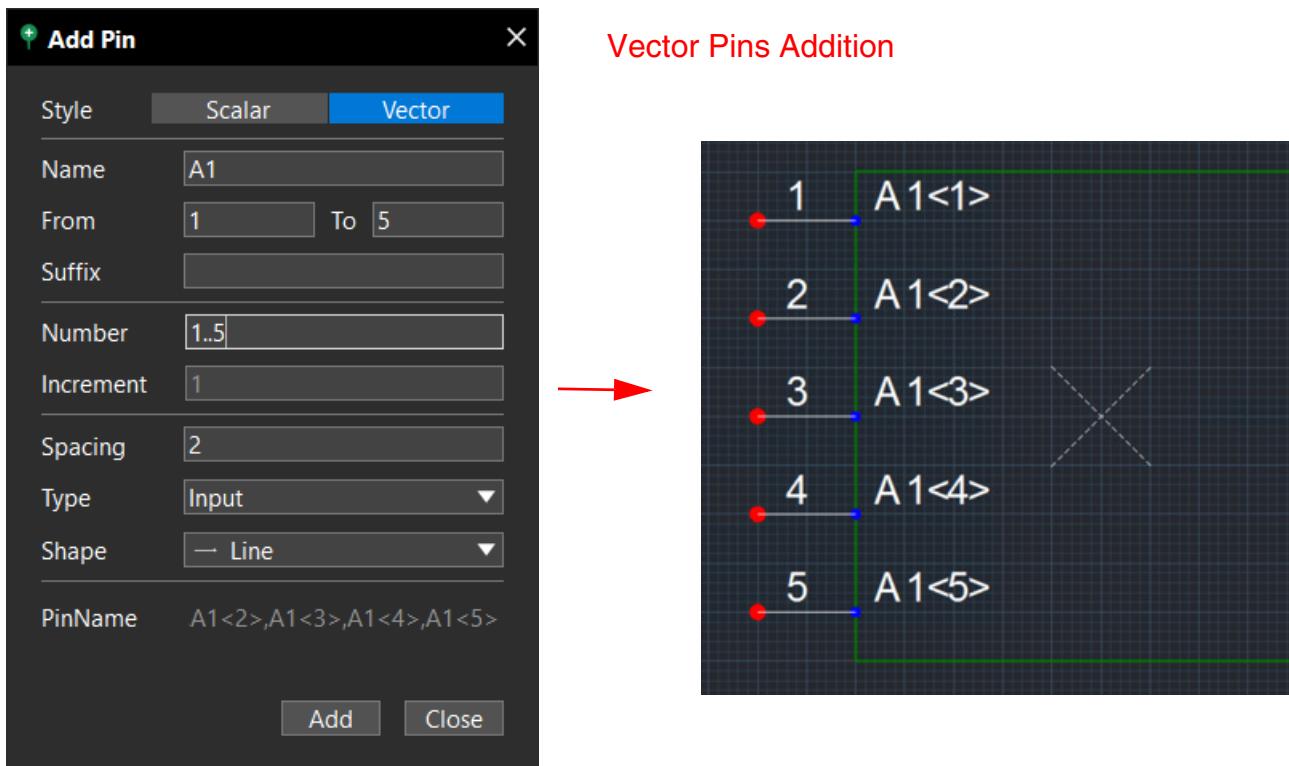
Library Authoring and Editing in Allegro X System Capture

Working with Symbols

You can click the pin name to view the pin numbers displayed in the *Attributes* section of the *Properties* panel.



The following figure shows vector pins added to a symbol:



Adding Low Asserted Pins

You can transform an existing pin into a low-asserted pin or a partially low-asserted pin. To add low asserted pins, do the following:

1. Click  to display *Table View*.

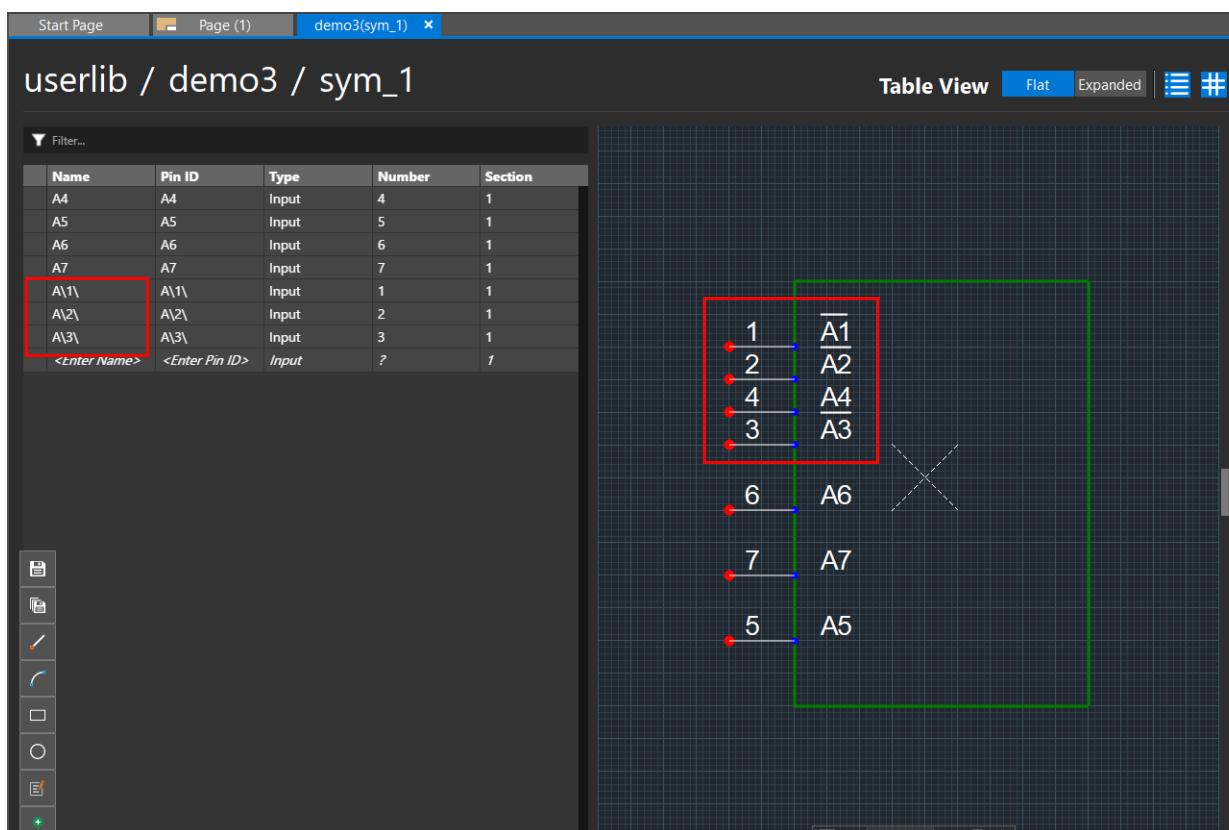
Library Authoring and Editing in Allegro X System Capture

Working with Symbols

2. Do one of the following:

- ❑ To make a pin low asserted, right-click the pin name in the *Name* column, and select *Overbar – On*.
- ❑ To make a pin partially low asserted, double-click the pin name in the *Name* column, select a character of the pin, and select *Overbar – On*.

In *Table View*, a \ (back-slash) is added after the selected or all characters of the pin. A bar also appears on the name or character of the pin in the *Symbol* view indicating that it is a low asserted pin or partially low asserted pin.



Related Topic

- [Adding Pins using Table View](#)

Adding Symbol Properties

In System Capture, you can add or edit a property from the *Additional Properties* section of the *Properties* panel. This panel displays the property name, property value, and visibility of the property.

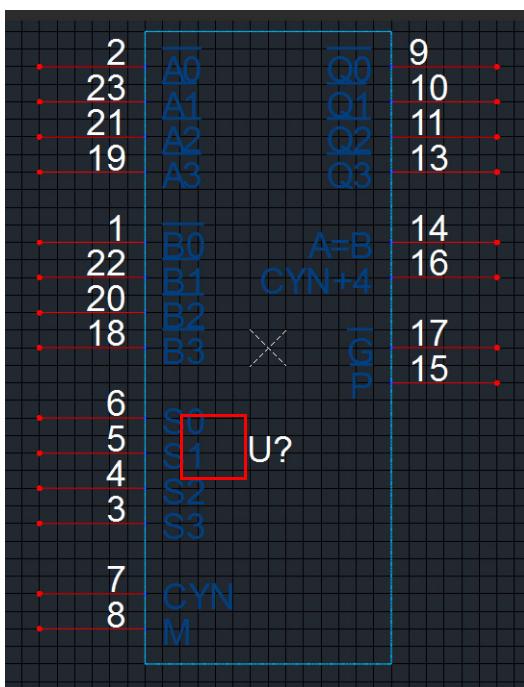
When you select multiple objects, the properties that are common to all the selected objects are displayed in this panel. You can also add the same property to multiple objects in one go.

To add properties to an object, do the following:

1. Select the object.
 2. Double-click <Add New> under the *Additional Properties* section of the *Properties* panel.
- The field becomes editable.
3. Type the property name.
 4. Type the property value in the *Value* column and press *Enter*.

The property and its value are added to the symbol and the property value is displayed on the canvas.

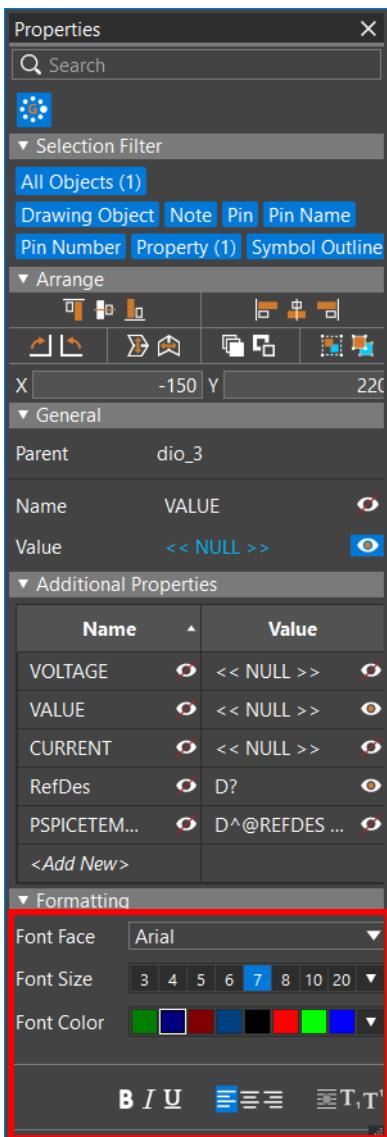
5. Toggle the visibility icon () to hide or display the property on the canvas.



Library Authoring and Editing in Allegro X System Capture

Working with Symbols

The formatting of property text can be changed any time from the *Formatting* section of the *Properties* panel.



Related Topic

- [Adding Part Properties](#)

Adding Notes

You can add notes and images to a symbol to provide additional information about the symbol. To add a note, do the following:

1. Select *Place – Note*.
 2. Click anywhere on the canvas to place the note.
- A blank note is placed on the canvas.
3. Type the detail you want to add and click outside the note.

You can change the formatting of the text from the options in the shortcut menu or the *Arrange* and *Formatting* panes of the *Properties* panel.

Related Topic

- [Setting Up Pin Style Defaults](#)

Adding Images

You can add images to a symbol that can be used to represent the symbol logic.

To add an image:

1. Select *Place – Picture*.
The *Image* dialog box is displayed.
2. Locate and select the image you want to add and click *Open*.
The image is attached to the cursor.
3. Click anywhere on the canvas to place the image.
The image is added to the canvas. You can change the orientation of the image from the *Arrange* pane of the *Properties* panel.

Related Topic

- [Setting Up Pin Style Defaults](#)

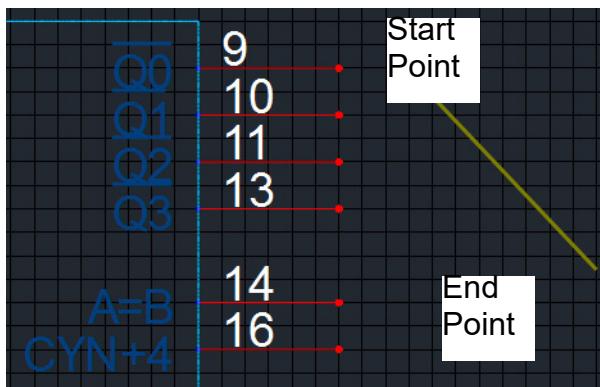
Drawing Arcs

In System Capture, you can easily draw an arc of any angle using the *Place Arc* button.

To draw an arc, do the following:

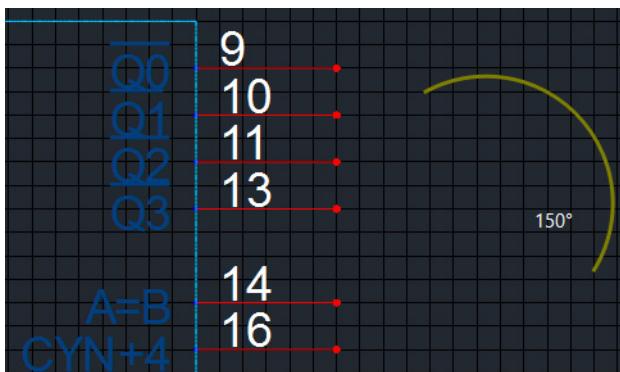
1. Click the *Place Arc* () button on the floating toolbar.
The cursor changes to a cross-hair.
2. Click anywhere on the canvas to specify the start point of the arc.
3. Drag the mouse and click to specify the end point of the arc.

The two points define the two ends of the arc.



After you place the end point of the arc, you can control its curvature dynamically by dragging the mouse.

4. Move the cursor to adjust the angle of the arc and then click to finish drawing the arc.



5. Right-click and select *Done* to dismiss the arc tool.

You can also press *Esc* on the keyboard to dismiss the arc tool.

Arc color and alignment can be changed from the shortcut menu or the *Formatting* pane of the *Properties* panel.

Related Topic

- [Setting Up Pin Style Defaults](#)

Drawing Lines

You can place a line to refer to an image or a note. To draw a line:

1. Click the *Place Line* () button on the toolbar.
The cursor changes to a cross-hair.
2. Click the canvas to specify the start point of the line.
3. Move the cursor and click to specify the end point of the line.
4. Right-click and select *Done* or press the *Esc* key to dismiss the line tool.

The color and alignment of lines can be changed from the shortcut menu and the *Formatting* section of the *Properties* panel.

Related Topic

- [Setting Up Pin Style Defaults](#)

Reusing Existing Symbols

To save time and effort in creating and adding more symbols to a part, you can reuse existing symbols of a part. For each copy of the symbol, the integer count in the symbol name is incremented by 1. For example, if you create a copy of a symbol named `sym_1`, the copies are named `sym_2`, `sym_3`, and so on.

To create a duplicate copy of a symbol, do the following:

1. Right-click the symbol to be copied and select *Duplicate*.

A duplicate copy of the selected symbol is created and the symbol is opened in a new window.

The duplicated symbol is also displayed under the *Symbols* node of the selected part.

2. Select *File – Save All*.

The symbol details are saved.

Renaming a Symbol

You can rename an existing symbol. There are two ways to rename a symbol.

1. Do one of the following to rename a symbol:

- Right-click the symbol and click *Rename*.
- Click a symbol or press F2 when the symbol is selected.

2. Type the new name and press the *Enter* key.

The symbol is renamed as specified.

Deleting a Symbol

You can delete a symbol if it is not being used or if it is no longer required. However, you cannot delete the only symbol available in a part because at least one symbol must exist for a part.

- To delete a symbol, right-click a symbol, and select *Delete* from the menu.

The selected symbol is deleted.

Editing Symbol Objects

Symbol objects, such as arcs, rectangles, properties, text, and so on can be rotated, aligned, or mirrored from the *Arrange* pane of the *Properties* panel.

To align, rotate, or mirror objects, do the following:

1. Click to select the object you want to rotate or mirror.

You can also rotate multiple objects by drawing a selection box around the object.

2. In the *Arrange* pane of the *Properties* panel, do one of the following:

- a. Click the *Rotate Right* button () or press Ctrl + R to rotate the object clockwise.

The selection is rotated by 90 degrees clockwise.

Library Authoring and Editing in Allegro X System Capture

Working with Symbols

- b.** Click the *Rotate Left* button () or press **Ctrl + L** to rotate the object counter-clockwise.
The selection is rotated by 90 degrees counter-clockwise.
- c.** Click the *Mirror Horizontal* button () or press **Ctrl + H** to mirror the object.
The selection is mirrored from side to side across Y-axis.
- d.** Click the *Mirror Vertical* button () or press **Ctrl + T** to mirror the object.
- e.** Click the *Align Left*, *Align Right*, or *Align Center* buttons to align the objects to left, right, or the center.
- f.** Click the *Align Top*, *Align Bottom*, or *Align Middle* buttons to align the objects to the top, bottom, or middle.

The selected objects are modified based on the selected option.

Related Topics

- [Setting Up Pin Style Defaults](#)
- [Renaming Parts](#)

Creation of Shapes

In a project created in the *Allegro Unified* library mode, you can create a new custom shape or a pin shape under a template library and attach it to a pin or the complete symbol. This enables you to have a symbol with graphics that represents the functionality of a symbol.

Note: You can configure existing pin shapes and custom shapes and use them in System Capture projects based on *DE-HDL* library.

Using an *Allegro Unified* library for your project, you can create the following types of shapes:

- [Creating a Custom Shape](#)
- [Creating a Pin Shape](#)

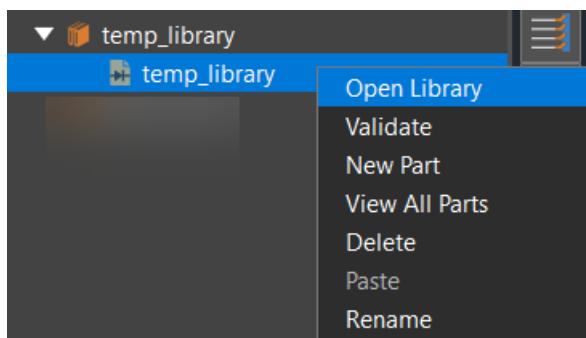
You can also modify an existing custom shape or a pin shape from the library.

Creating a Custom Shape

You can create a custom shape and place it directly on a symbol. A custom shape can be associated with pins or the complete symbol.

To create a new custom shape, do the following:

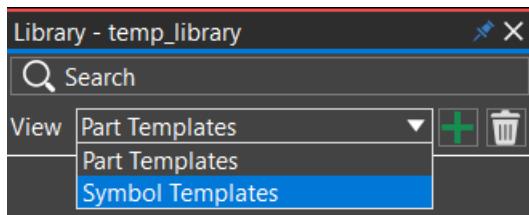
1. Right-click the new template library and select *Open Library*.



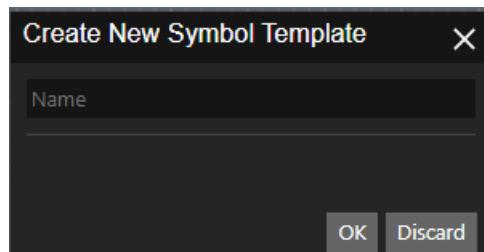
Library Authoring and Editing in Allegro X System Capture

Working with Symbols

2. Select the *Symbol Template* view from the drop-down list and click the *New Symbol Template* button.



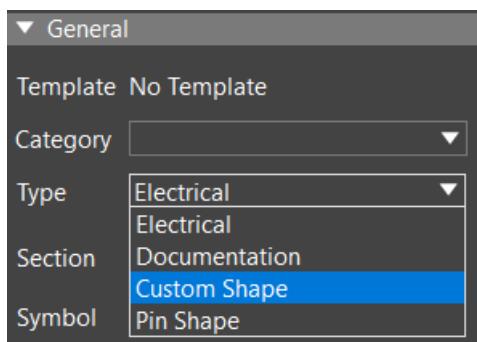
The *Create New Symbol Template* dialog box is displayed.



3. Specify a name for the new symbol template and click *OK*.

A blank canvas is displayed to create an electrical type symbol template.

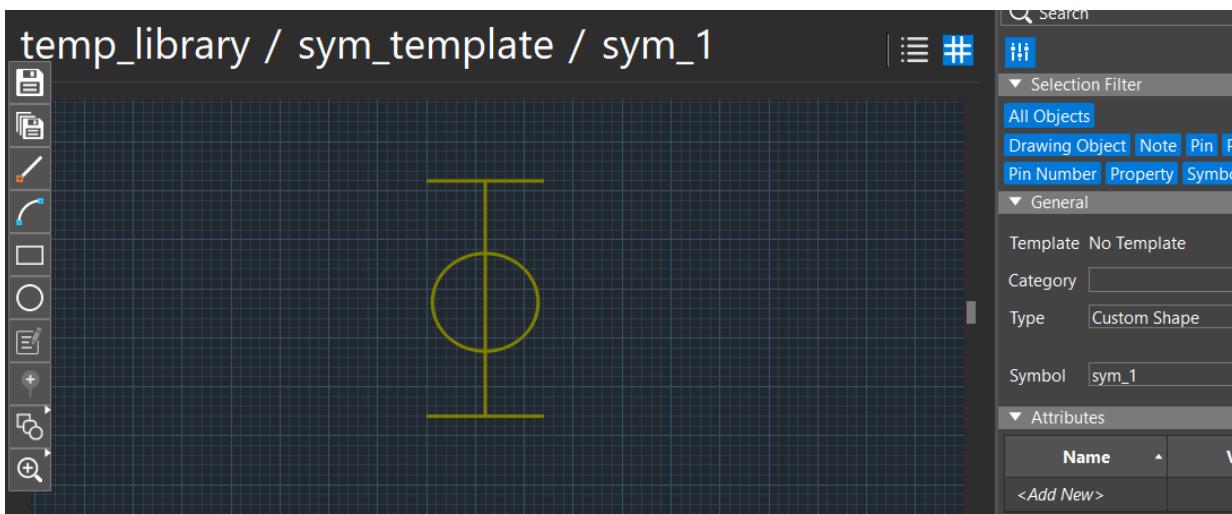
4. In the *General* section of the *Properties* panel, change the *Type* to *Custom Shape*.



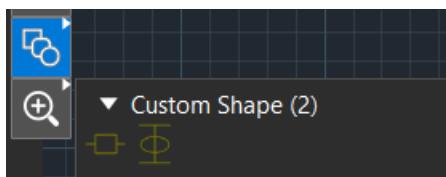
Library Authoring and Editing in Allegro X System Capture

Working with Symbols

5. Draw a custom shape on the canvas using the graphics available in the floating toolbar menu and save it.



The newly created custom shape is visible in the floating toolbar menu under the *Auto Shapes* menu.



Next, when you create a part, you can attach a symbol with a pin that has custom shapes. You can also use custom shapes while creating an electrical symbol.

Creating a Pin Shape

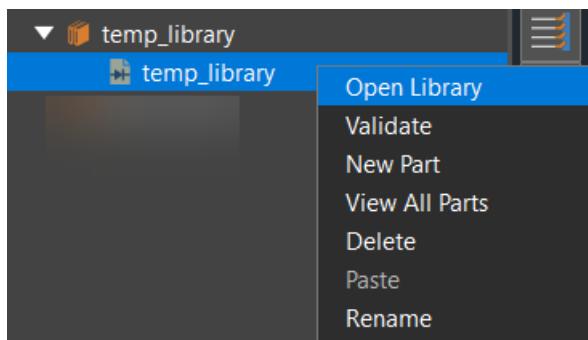
You can create a pin shape and attach it to a symbol pin. A pin shape consists of a connection point, a point where a connection is made with the pin, and an anchor point where a shape is connected to a symbol.

To create a pin shape, do the following:

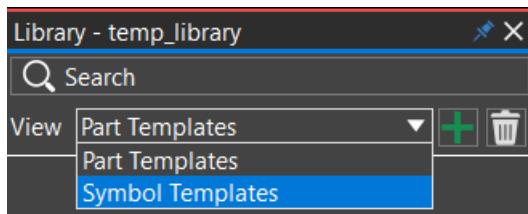
Library Authoring and Editing in Allegro X System Capture

Working with Symbols

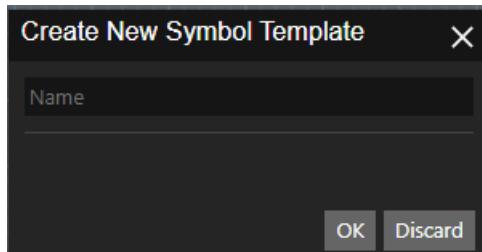
1. Right-click the new template library and select *Open Library*.



2. Select the *Symbol Template* view from the drop-down list and click the *New Symbol Template* button.



The *Create New Symbol Template* dialog box is displayed.



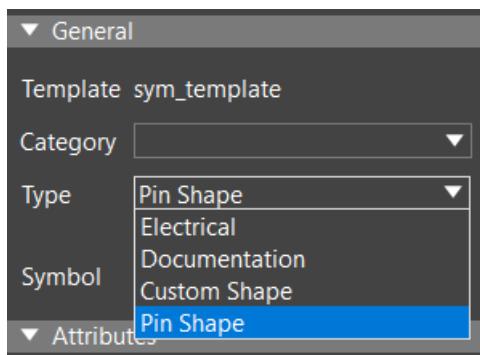
3. Specify a name for the new symbol template and click *OK*.

An electrical type symbol template is created.

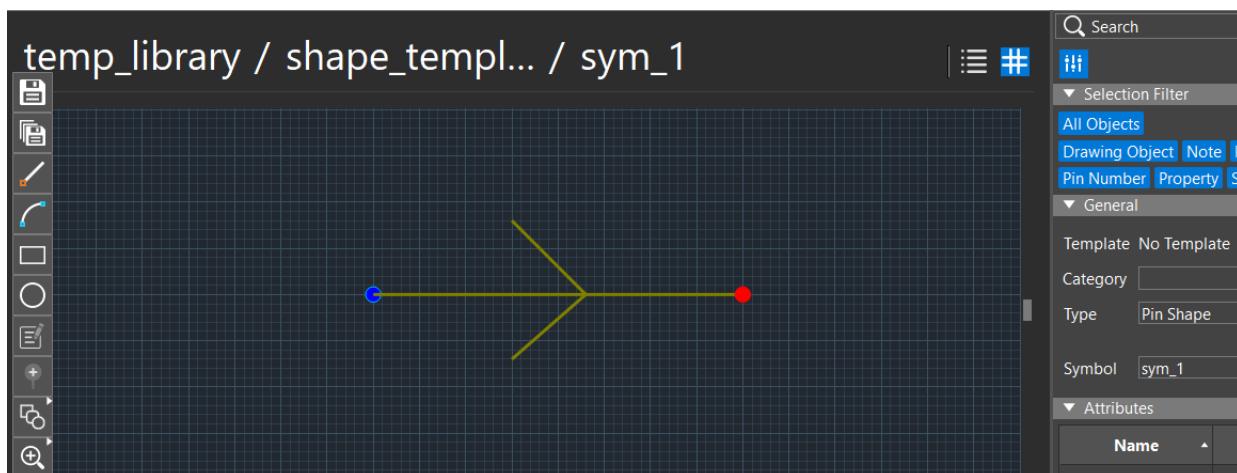
Library Authoring and Editing in Allegro X System Capture

Working with Symbols

4. In the *General* section of the *Properties* panel, change the *Type* to *Pin Shape*.



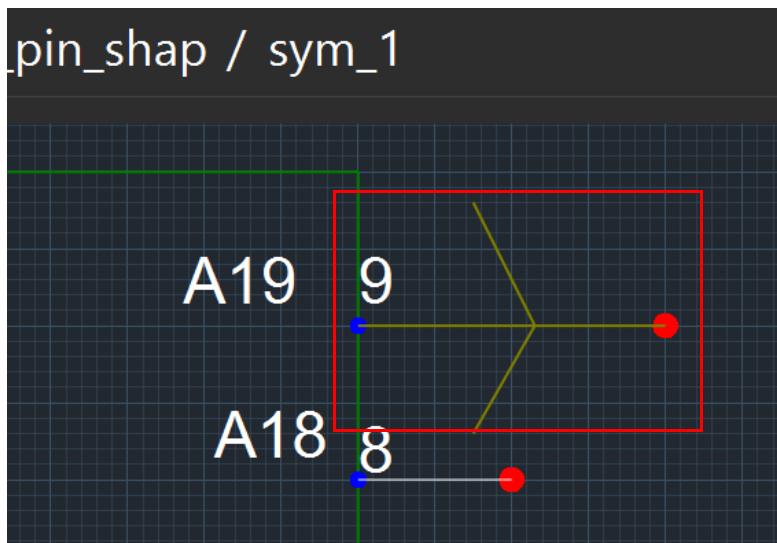
5. Draw a pin shape on the canvas using the graphics available in the floating toolbar menu and save it.



Library Authoring and Editing in Allegro X System Capture

Working with Symbols

The newly created pin shape can now be associated with symbol pins. In the highlighted pin of the following image, the blue dot denotes the anchor point, and the red dot is the origin where a connection is made with the pin.



Related Topics

- [Creating a New Blank Symbol](#)
- [Reusing Existing Symbols](#)

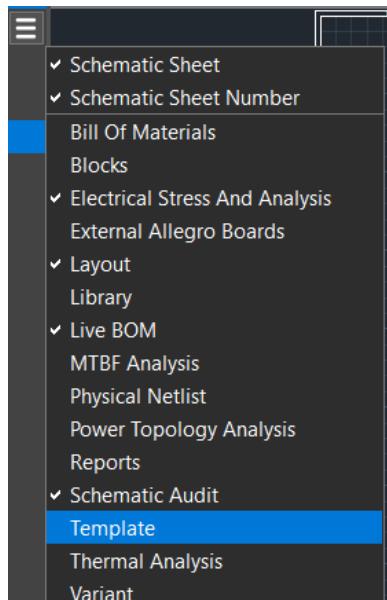
Template Libraries

When working for System Capture projects based on *Allegro Unified* libraries, you can:

- Create a part or symbol using default templates
- Create a new template

Note: You can create only part and symbol templates and add a footprint as a template to a part.

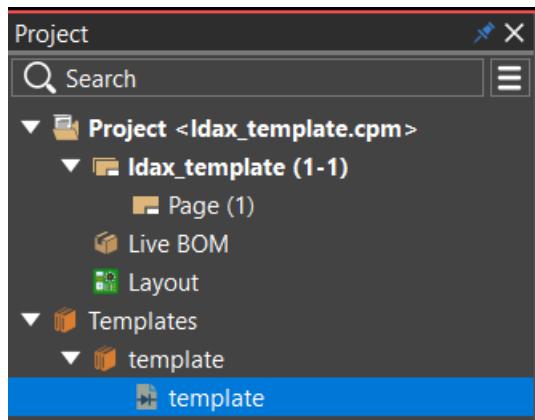
To view the template node, select *Template* in *Project Explorer Settings*.



Library Authoring and Editing in Allegro X System Capture

Template Libraries

A default library container, *Templates*, is displayed. This container has an existing template library with Cadence-supplied templates.

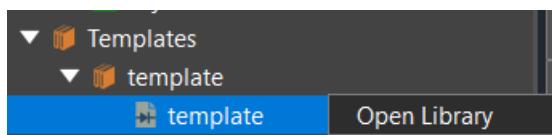


You can create a new template library container under the *Template* node, and create a new part or symbol template in this container. You can also rename or delete a template you created in a template library.

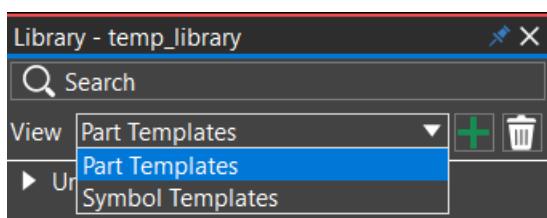
Creating a New Template Library Container

When you create a new library container, a default library is created with the same name as the container. The library contains all the templates available in the default Cadence installation directory.

- To view the Cadence-supplied templates, right-click the *template* library and select *Open Library*.



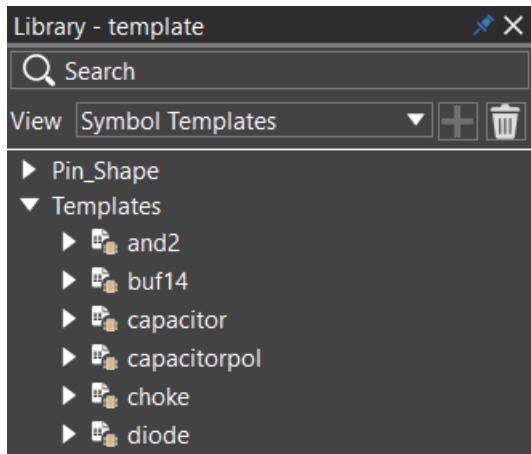
The template library is opened with *Part Templates View*. You can switch it to *Symbol Template View*, if required.



Library Authoring and Editing in Allegro X System Capture

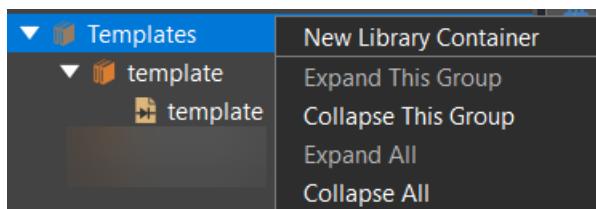
Template Libraries

The existing templates are displayed under the *Part Templates* and *Symbol Templates* views. You cannot edit or delete an existing template.



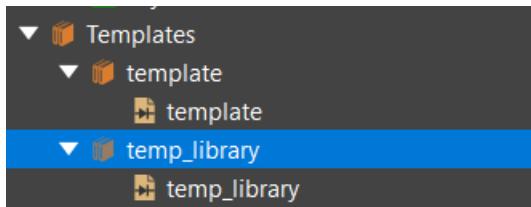
To create a new template library container, do the following:

1. Right-click the *Templates* node and select *New Library Container*.



2. Specify a name for the new library container and click *OK*.

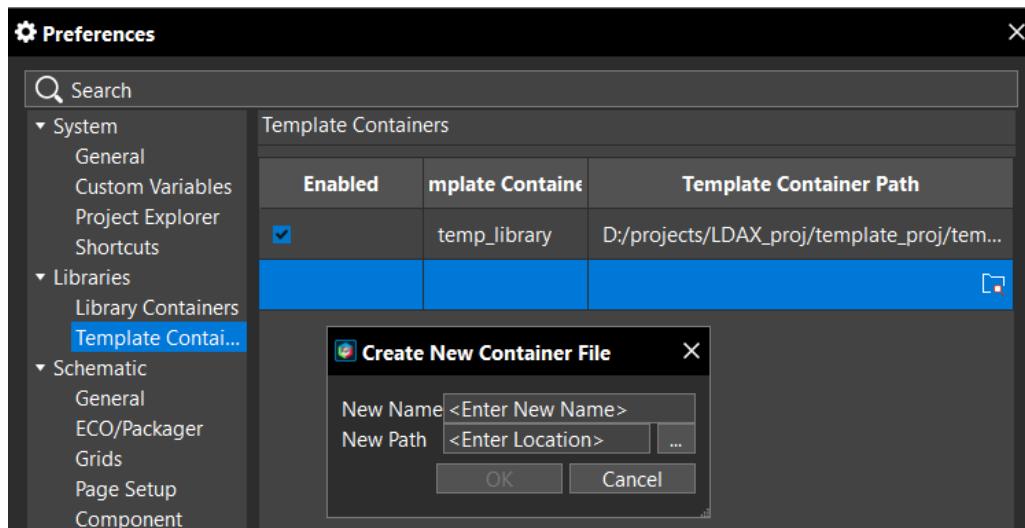
A new template library container is created with a default library, which has the same name as the container.



Library Authoring and Editing in Allegro X System Capture

Template Libraries

You can also create, enable, or disable a template library container from *Preferences* under the *Edit* menu.

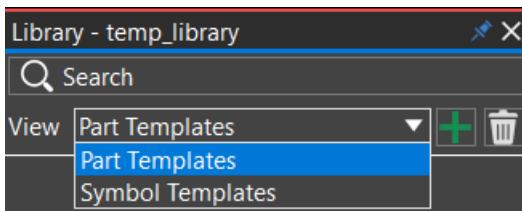


Creating a New Part Template

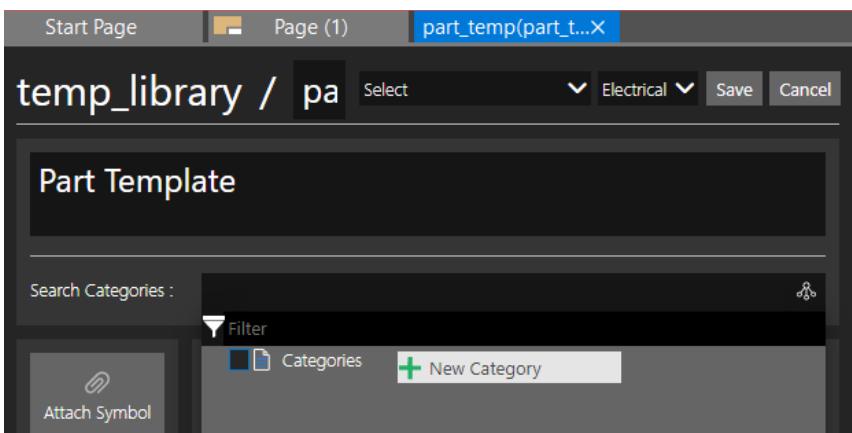
Before assigning a symbol template to a part, you need to create a part template that is associated with the symbol template. With this, all the part template properties are assigned to the symbol template.

To create a new part template, do the following:

1. Open a template library and select the *Part Template* view from the drop-down list.



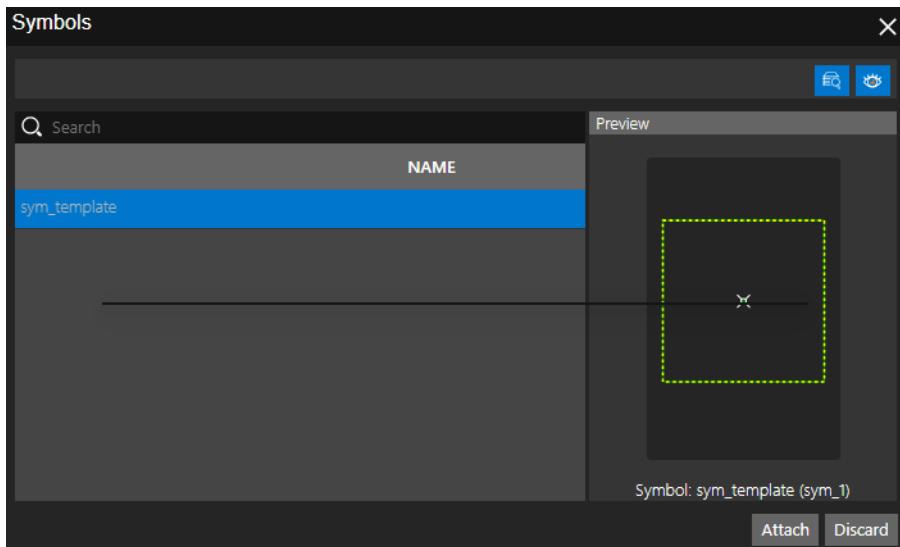
2. Click the *New Part Template* button to create a new part template.
A new part window is displayed.
3. Specify a name and description for the part template.
4. Select the category in which you want to store this template or create a new category for the part template.



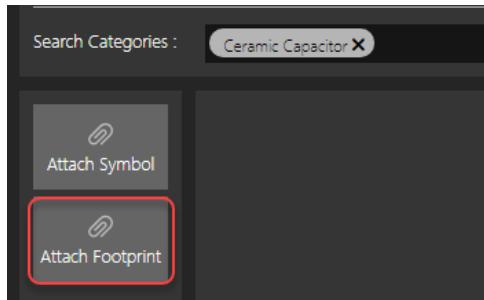
Library Authoring and Editing in Allegro X System Capture

Template Libraries

5. Select an existing symbol template to be attached to the part and click *Attach*.



6. You can also attach a footprint template to the part.



7. Click *Add Property* to specify part-level properties for the part.

Add as many rows as required to specify additional properties.

When you select a property row, the type of property and its respective formats is listed.

Add Property							Type	Format
PROPERTY NAME	VALUE	▲	DESCRIPTION	V	A	E		
VALUE	1K		Resistance Value at 25 degree celsius	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	Validation Rule	...
							Readonly	No
							Versionable	Yes
							Mandatory	No
							Caption	

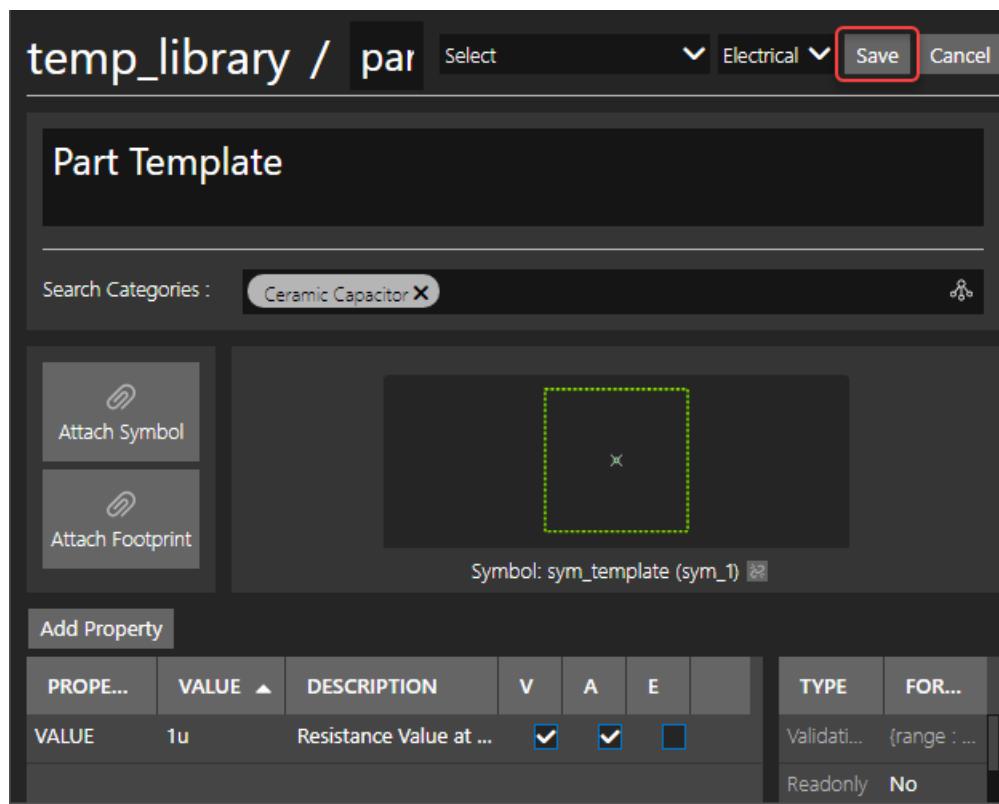
Library Authoring and Editing in Allegro X System Capture

Template Libraries

The following properties are listed:

- ❑ *Validation Rule*: You can create a validation rule for the part properties.
- ❑ *Readonly*: This can be *Yes* or *No* to define whether you can edit the value of this property.
- ❑ *Versionable*: If a property is synchronized with an external system and you do not want to change the value of the property with respect to the other system, this should be *No*. The default value of this type is *Yes*.
- ❑ *Mandatory*: Specify whether property value is mandatory.
- ❑ *Caption*: Use this option if you want to add a caption to the part properties.

8. Click *Save* to save the part template.

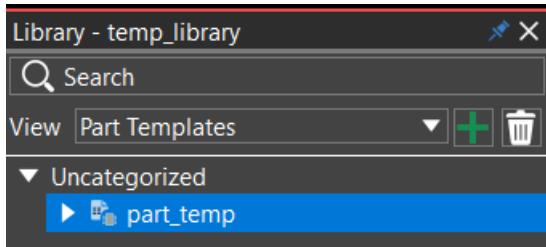


The part template is created in a template library. You can now create a part using this template.

Library Authoring and Editing in Allegro X System Capture

Template Libraries

If you did not select a category while creating the template, the template is by default listed under the *Uncategorized* part template node.



Creating Template Validation Rules

You can create rules based on which templates are validated for a part.

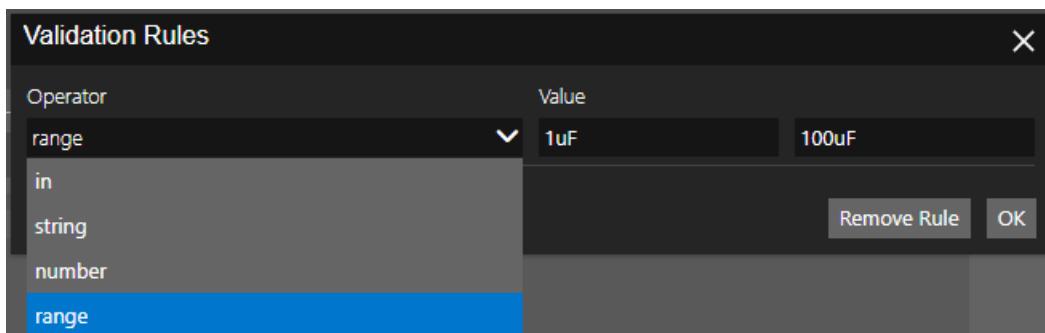
To create a template validation rule for a part property, do the following:

1. Click the browse button under the *FORMAT* column in the *Validation Rule* row.

TYPE	FORMAT
Validation Rule	...
Readonly	No
Versionable	Yes
Mandatory	No
Caption	

A *Validation Rules* dialog box is displayed.

You can select an operator from the drop-down list and specify values for the operator.



2. Click *OK*.

Library Authoring and Editing in Allegro X System Capture

Template Libraries

The validation rule is listed under the *TYPE* column, which defines the type of property.

TYPE	FORMAT
Validation Rule	{range : 1uF..100uF}
Readonly	No
Versionable	Yes
Mandatory	No
Caption	

You can remove a rule at any time by clicking the *Remove Rule* button.

Related Topic

- [Creating a New Symbol Template](#)

Editing an Existing Template

You can also edit existing templates that are in template libraries. To edit an existing part template, do the following:

1. In the *Part Template View* of the template library, click *Edit* to open the part template in the edit mode.
2. Click the *Add Property* button to add or edit a property, edit the part template name, category, and so on.
3. Click *Save* to save the changes.

Related Topic

- [Creating a New Part Template](#)

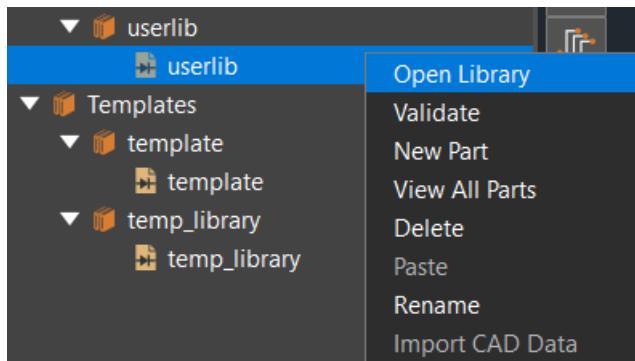
Creating a Part Using a Part Template

To create a part using a part template, do the following:

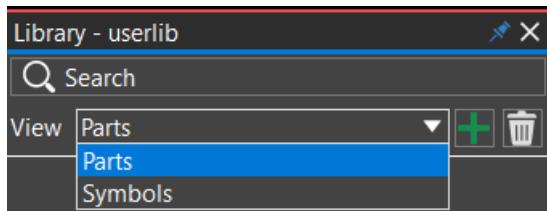
Library Authoring and Editing in Allegro X System Capture

Template Libraries

1. Right-click the library in the *Libraries* tree and select *Open Library*.



2. Open the *Part* view and click the plus (+) button to create a new part.



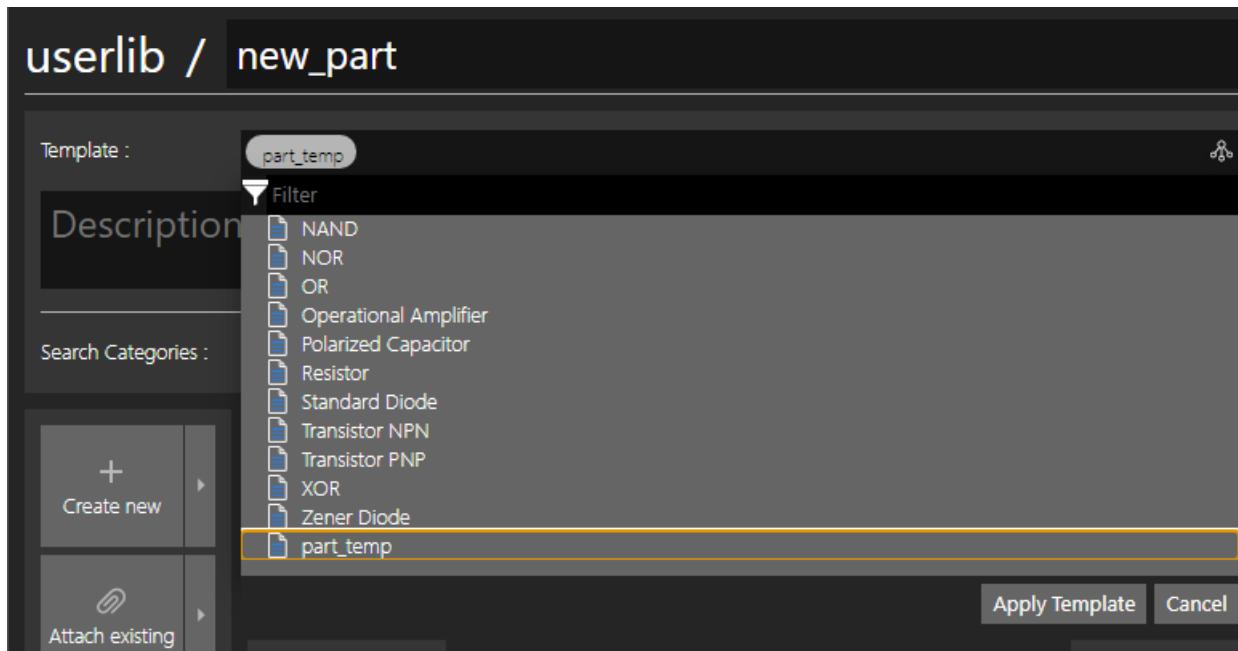
A new part window is displayed.

3. Specify a name for the part and select a part template from the list.

Library Authoring and Editing in Allegro X System Capture

Template Libraries

Note that the new part template is visible under the template list.



4. Click *Apply Template* and the template is assigned to the part.

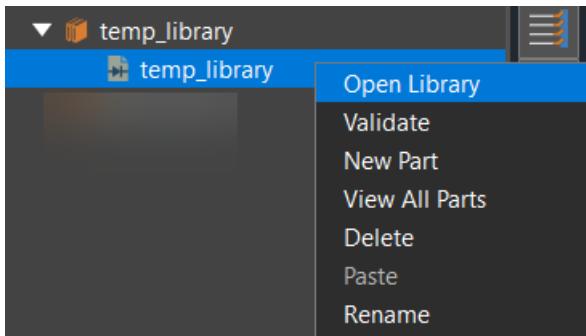
Creating a New Symbol Template

After creating a new library container, you can create a symbol template. Once the template is created, you can:

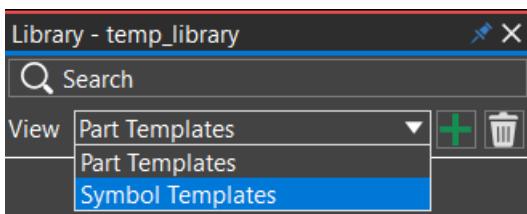
- Attach it to a symbol. See [Creating a Symbol Using a Symbol Template](#)
- Attach it directly to a part. See [Creating a Part Using a Symbol Template](#)

To create a new symbol template, do the following:

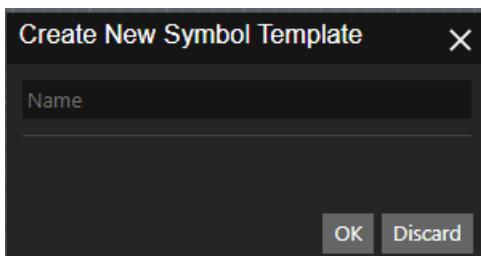
1. Right-click the new template library and select *Open Library*.



2. Select the *Symbol Template* view from the drop-down list and click the *New Symbol Template* button.



The *Create New Symbol Template* dialog box is displayed.

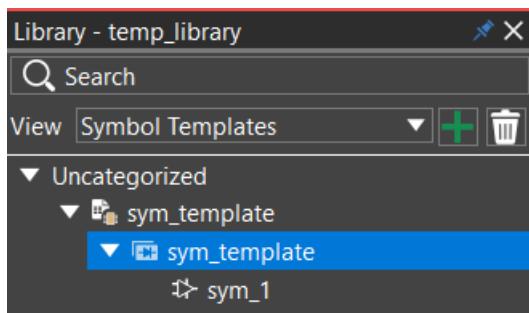


3. Specify a name for the new symbol template and click *OK*.

Library Authoring and Editing in Allegro X System Capture

Template Libraries

A symbol template is created.

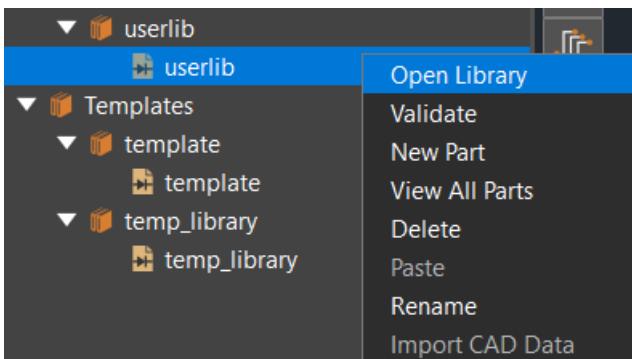


Creating a Symbol Using a Symbol Template

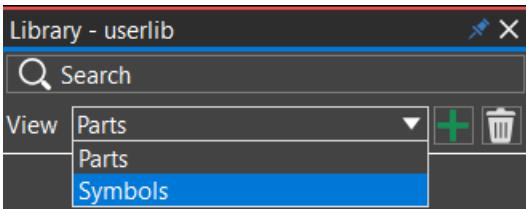
You can assign a symbol template to a symbol and attach the symbol to a part.

To create a symbol using a symbol template, do the following:

1. Right-click the library in the *Libraries* tree and select *Open Library*.



2. Open the Symbol view and click the (+) button to create a new symbol.



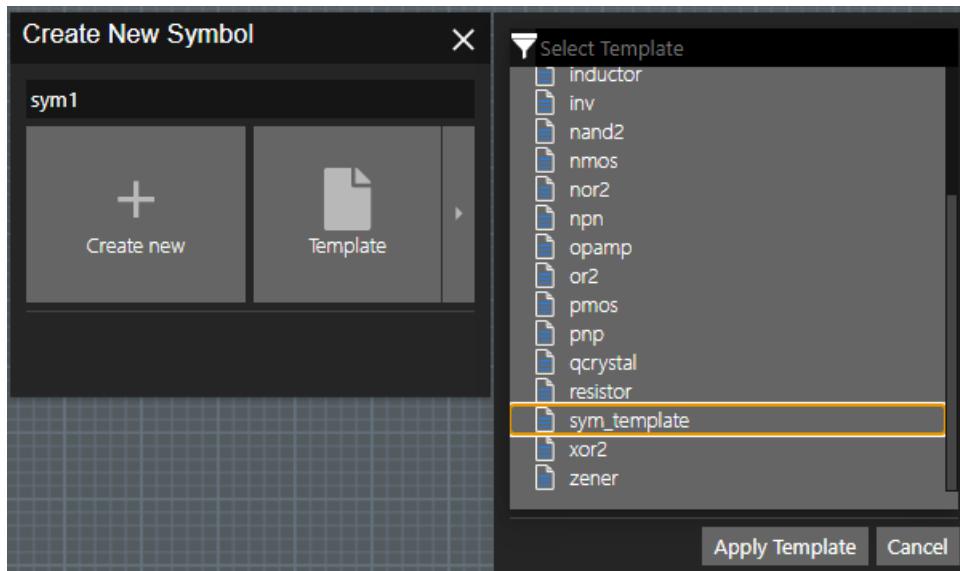
The *Create New Symbol* dialog box is displayed.

3. Specify the name of the new symbol and select a template from the list.

Library Authoring and Editing in Allegro X System Capture

Template Libraries

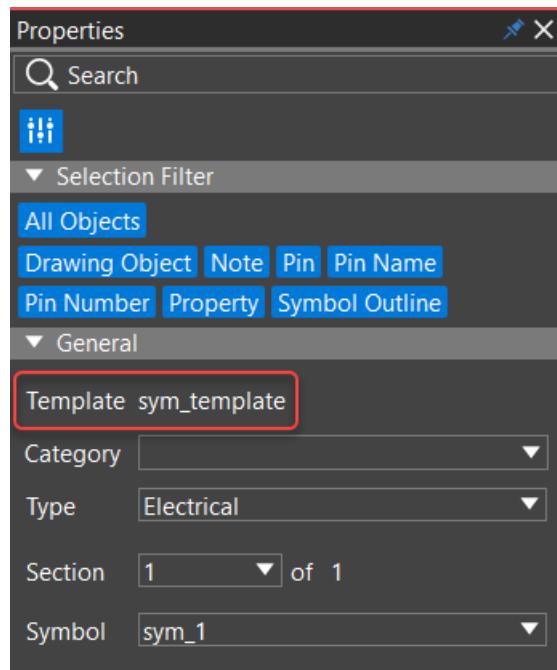
Notice that the new symbol template you created is listed under the *Template* list.



4. Click the *Apply Template* button.

A new template is assigned to the symbol.

You can view the template information under the *General* section of the *Properties* panel.



Library Authoring and Editing in Allegro X System Capture

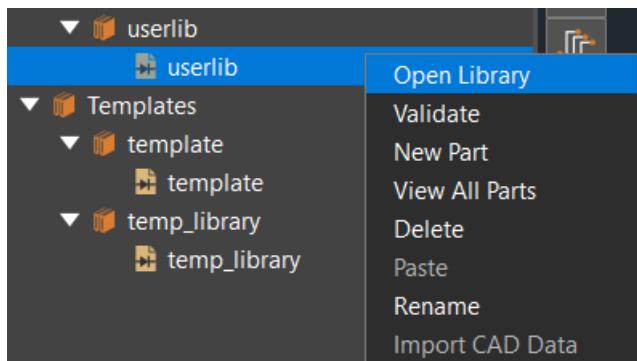
Template Libraries

You can now attach this symbol, which has an associated symbol template, to a part.

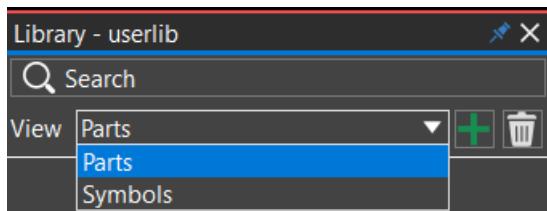
Creating a Part Using a Symbol Template

After creating a symbol template, you attach it to a symbol or a part. To attach a symbol template to a part, do the following:

1. Right-click the library in the *Libraries* tree and select *Open Library*.

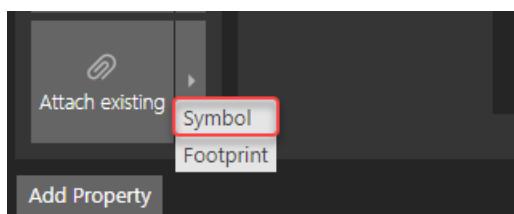


2. Open the *Part* view and click the plus (+) button to create a new part.



A new part window is displayed.

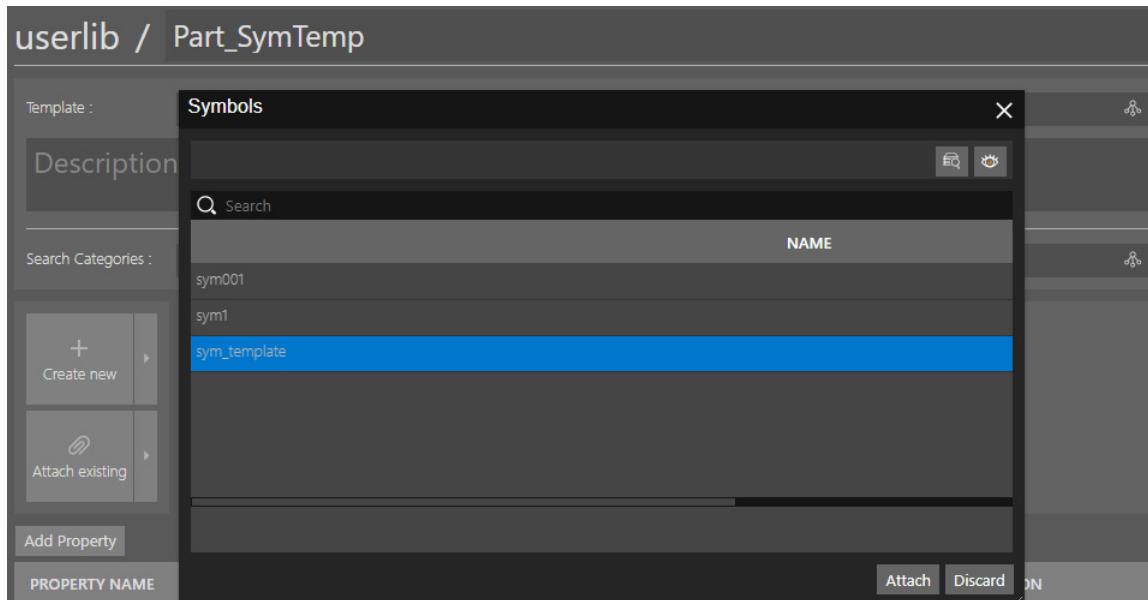
3. Specify a name for the part and select *Attach existing – Symbol*.



Library Authoring and Editing in Allegro X System Capture

Template Libraries

The symbol template you created is listed under the list of existing symbol templates.



4. Click *Attach* and save the part.

A new part is created in a new part template library with an existing symbol template attached to it.

Related Topic

- [Creating a New Part Template](#)

Management of Library Parts

You create parts in Allegro Unified libraries that refer to symbols and footprints. A part can contain a symbol, footprint, datasheet, and other details such as properties. You create a part either using an existing template or by creating a new symbol. Allegro X System Capture supports the creation of electrical parts, such as split, symmetrical, and asymmetrical parts from scratch.

You can also create non-electrical parts, such as page borders, title blocks, and a table of contents. Parts without pins are considered non-electrical parts, and are also referred to as documentation parts.

Related Topics

- [Creating Electrical Flat Parts](#)
- [Creating Electrical Split Parts](#)
- [Creating Symmetrical Parts](#)
- [Creating Asymmetrical Parts](#)
- [Creating Documentation Parts](#)

Creating Electrical Flat Parts

Allegro X System Capture supports the creation of electrical parts with a single or multiple symbols. Electrical parts with a single symbol are called flat parts and include diodes, resistors, capacitors, transistors, and inductors.

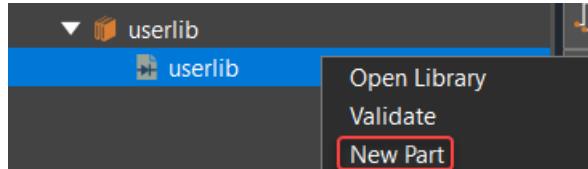
A flat part contains all the pins in one symbol. To create a flat part, follow these steps:

1. Do one of the following:

Library Authoring and Editing in Allegro X System Capture

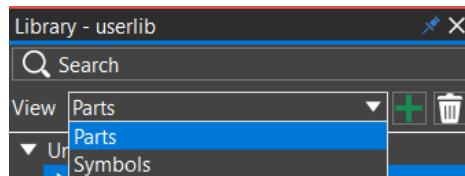
Management of Library Parts

- Right-click a library in the *Libraries* tree and select *New Part*.

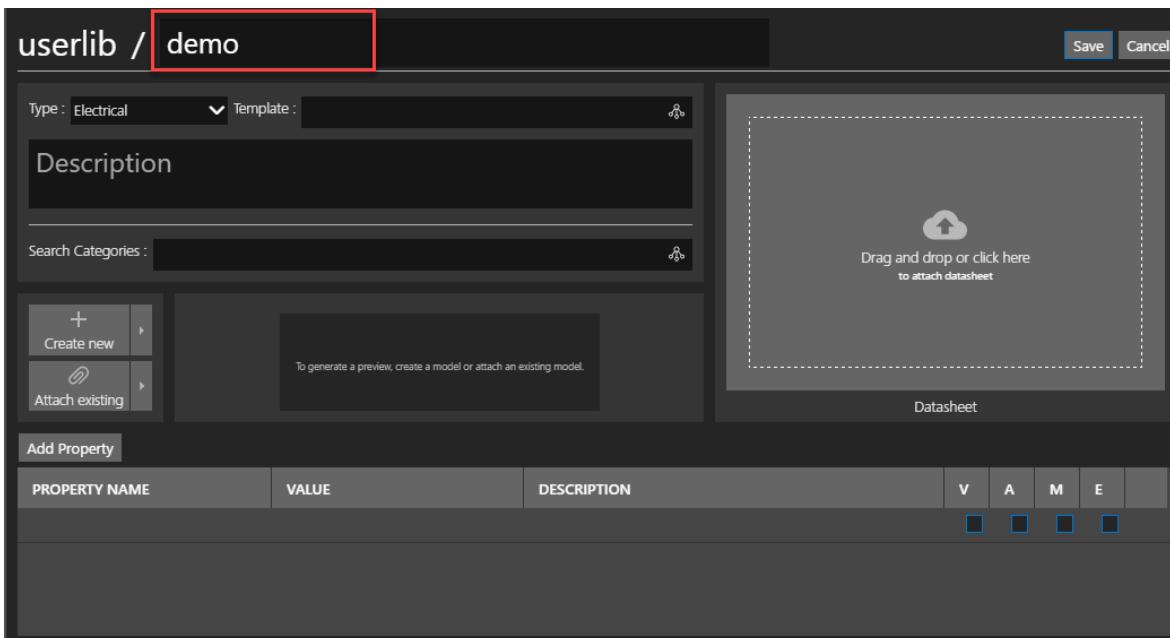


- Right-click a library and select *Open Library*.
- Double-click a library to open it and click the *New Part* button () in the *Library* explorer.

A new part window is displayed with a *Part View*. You can switch it to *Symbols View*.



- Specify a name for the part.



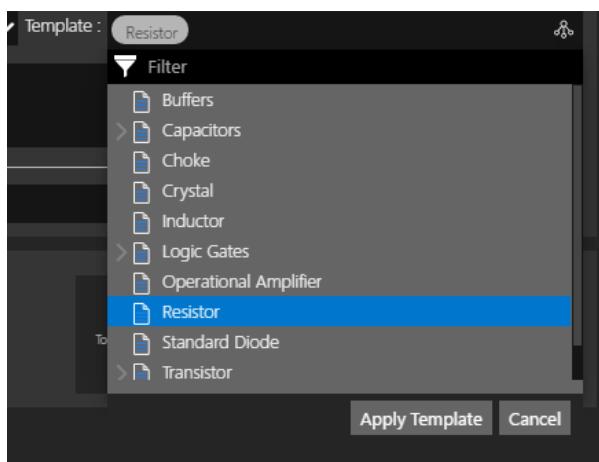
- Select between using an existing template and creating a new symbol for the part.

If you use an existing template, do the following:

Library Authoring and Editing in Allegro X System Capture

Management of Library Parts

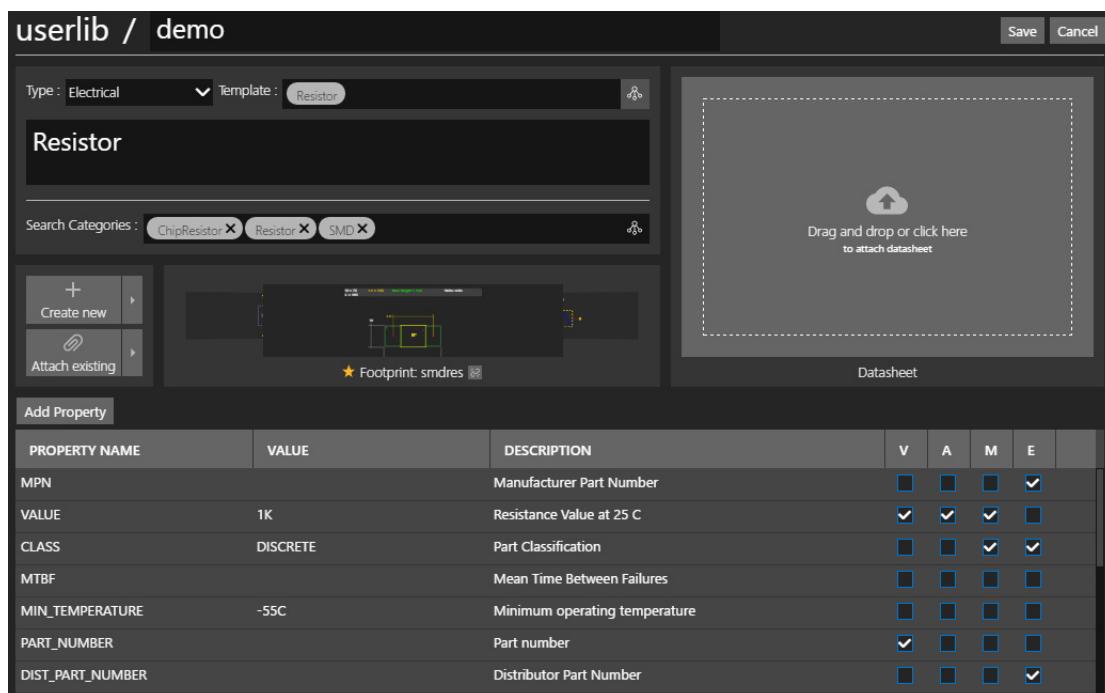
- a. Click  in the *Template* field.



- b. Select a template from the list and click *Apply Template*.

When you select a template, a symbol, footprint, and part-level properties are automatically assigned to the part. You can edit the symbol, if required.

Note that related part categories are also displayed automatically in the *Search Categories* field. You can create a custom category for this template.

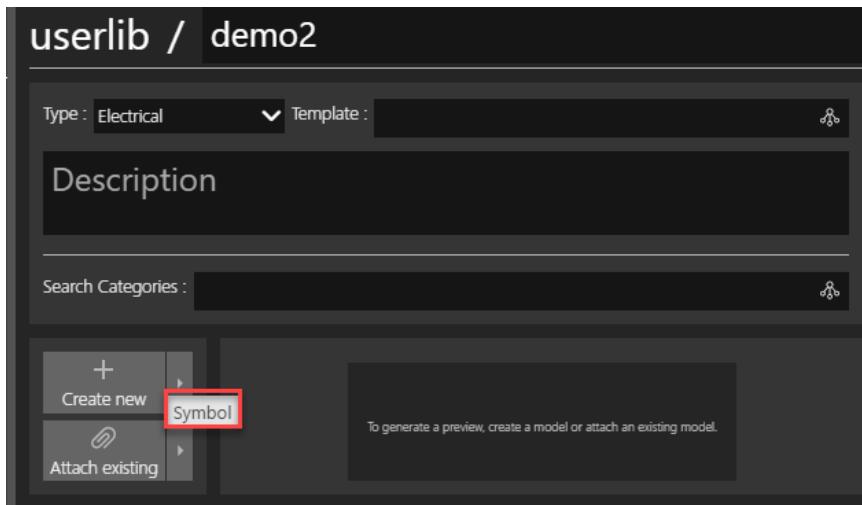


PROPERTY NAME	VALUE	DESCRIPTION	V	A	M	E
MPN		Manufacturer Part Number	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>
VALUE	1K	Resistance Value at 25 C	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>
CLASS	DISCRETE	Part Classification	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
MTBF		Mean Time Between Failures	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
MIN_TEMPERATURE	-55C	Minimum operating temperature	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
PART_NUMBER		Part number	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
DIST_PART_NUMBER		Distributor Part Number	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>

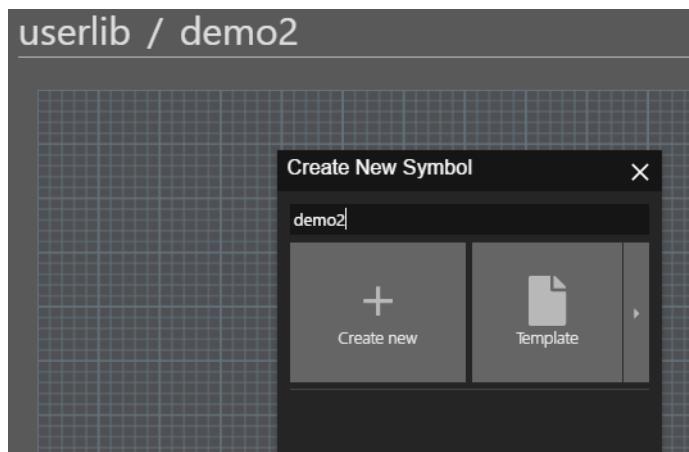
Library Authoring and Editing in Allegro X System Capture

Management of Library Parts

- Alternatively, click *Create new* and select *Symbol* to create a new symbol for the part.



You can create a new symbol from scratch or use a template.



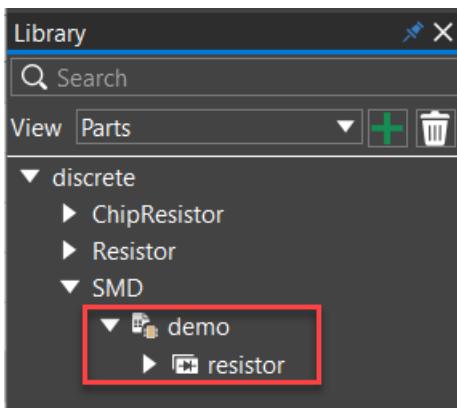
A new symbol window opens. A symbol boundary is added to the symbol canvas where you can add pins, properties, and objects to the symbol.

4. Click *Save*.

Library Authoring and Editing in Allegro X System Capture

Management of Library Parts

The part is created and displayed in the *Library* explorer. The newly created part is immediately available for use in *Unified Search*.



Related Topics

- [Creating a New Blank Symbol](#)
- [Creating a Category for a Specific Part](#)
- [Supported Characters in Part Names](#)
- [Adding Pins using Table View](#)

Creating Electrical Split Parts

Allegro X System Capture supports the creation of electrical parts with multiple symbols, such as split, symmetrical, and asymmetrical parts. These parts can be created from *Table View* by moving pins to different sections or by creating duplicate sections.

A split part contains pins that are split across multiple symbols where each symbol represents a different functionality. Split parts have at least one section in each symbol and are useful for symbols of large pin count devices.

In Allegro X System Capture, you can create split parts with large pin counts in the following ways:

- Add all the pins to a part and move the pins to a new section from *Table View*.
- Create multiple blank sections in a part and then add pins to each section.

Library Authoring and Editing in Allegro X System Capture

Management of Library Parts

Creating Split Parts by Adding Pins

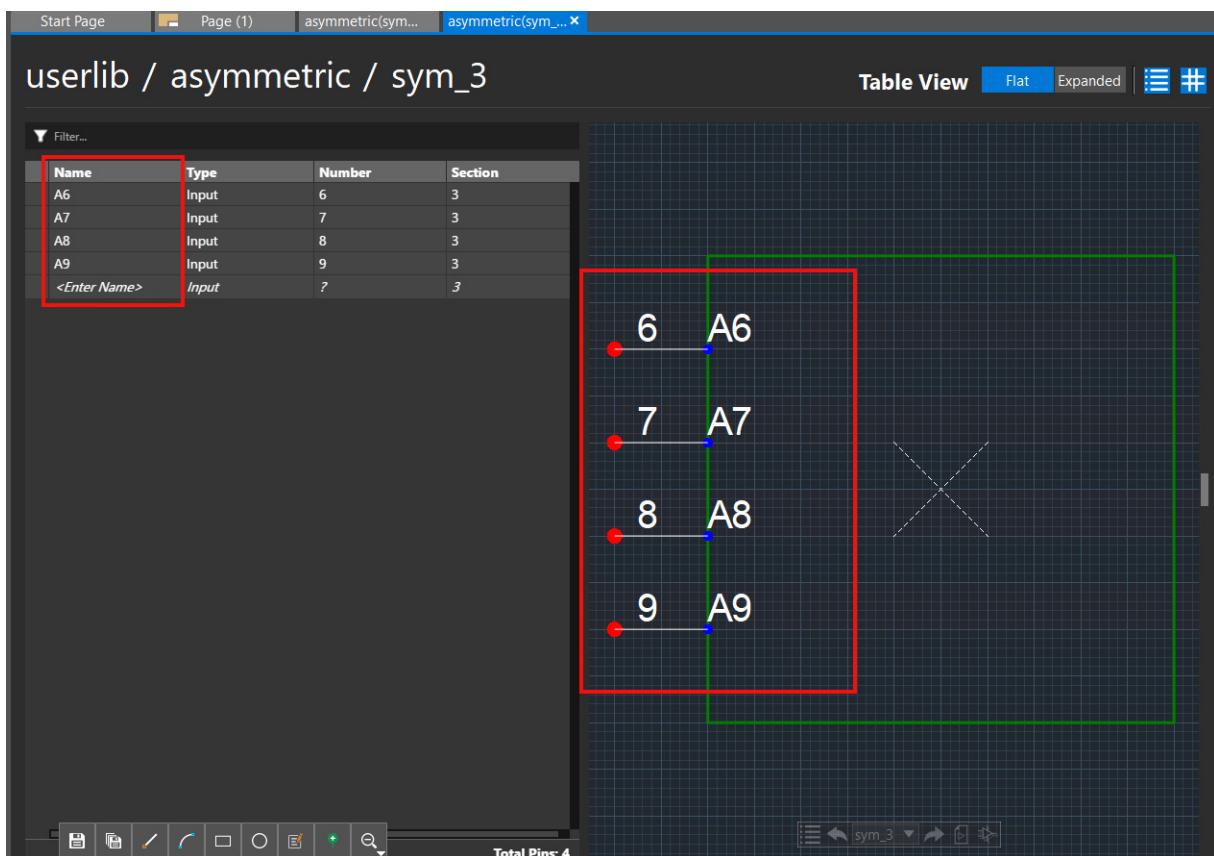
To create a split part by adding pins, do the following:

1. Create a new part.
2. Create a symbol and add the required pins to the symbol.

A new symbol window opens with a boundary of the symbol.

3. Click  to open *Table View*.
4. Select the pin numbers from the *Section* column.
5. Right-click and select *Move To – New Section*.

A new section and a corresponding symbol is created, and the selected pins are moved to the new section.



To view the next or previous sections or symbols from the canvas, click the *Next Section* () or *Previous Section* () buttons on the toolbar at the bottom of the canvas.

6. Select *File – Save All* to save the symbol.

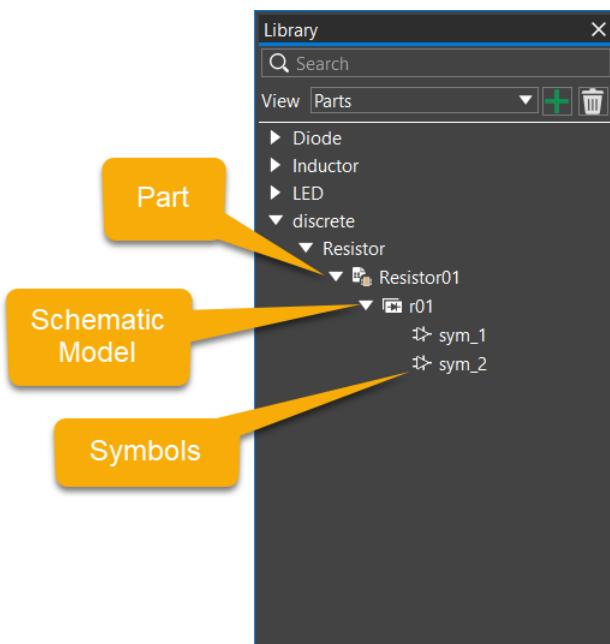
Library Authoring and Editing in Allegro X System Capture

Management of Library Parts

Alternatively, you can click the *Save All* button on the toolbar.



The symbol is saved and displayed in the *Library* explorer under the part.



Related Topics

- [Creating a New Blank Symbol](#)

Library Authoring and Editing in Allegro X System Capture

Management of Library Parts

- [Adding Pins using Table View](#)
- [Adding Low Asserted Pins](#)

Creating Split Parts by Adding Multiple Blank Sections

To create a split part by creating multiple blank sections in a part and then adding pins to the sections, do the following:

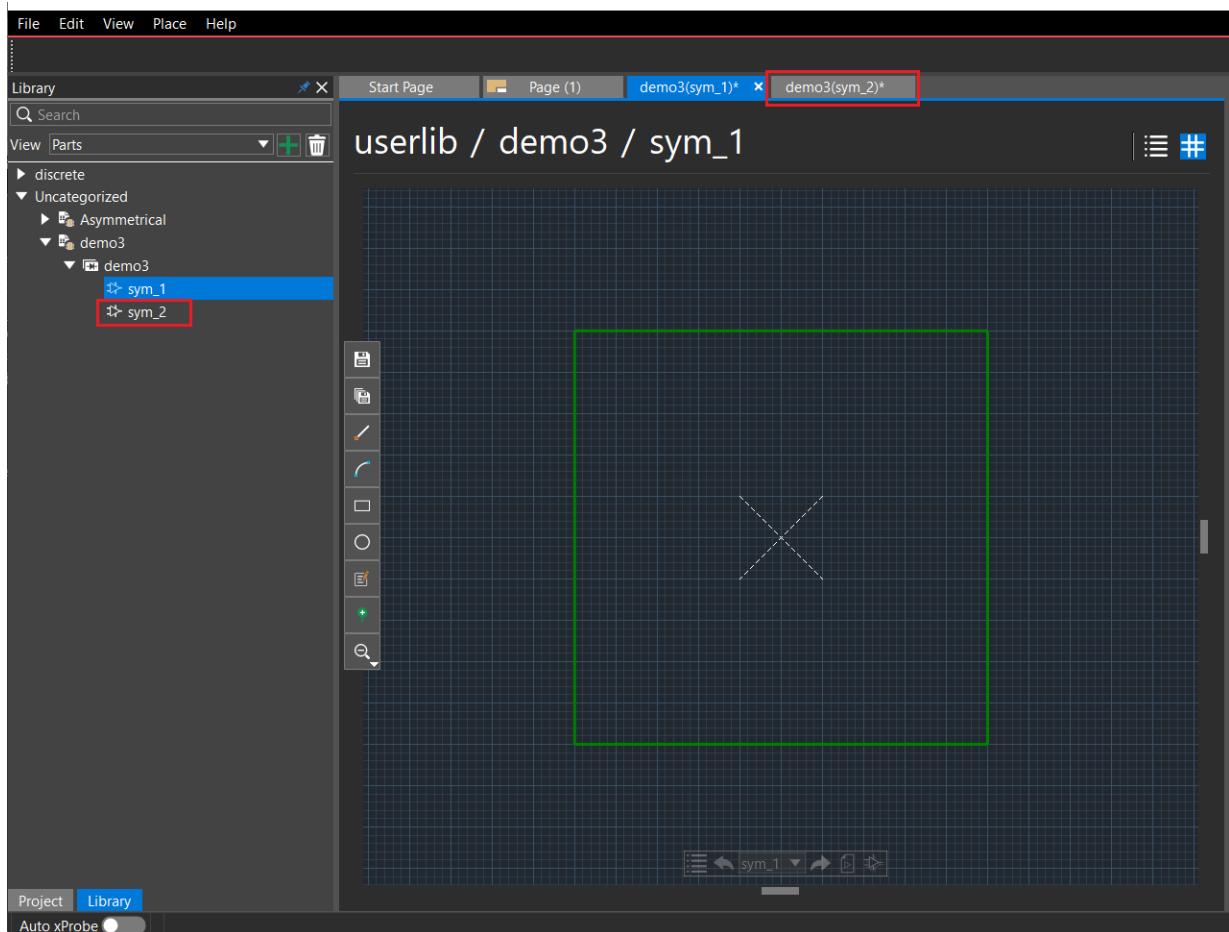
1. Create a new part.
2. Create a symbol.
A new symbol window opens with a boundary of the symbol.
3. Click the *New Section* button at the bottom of the canvas.



Library Authoring and Editing in Allegro X System Capture

Management of Library Parts

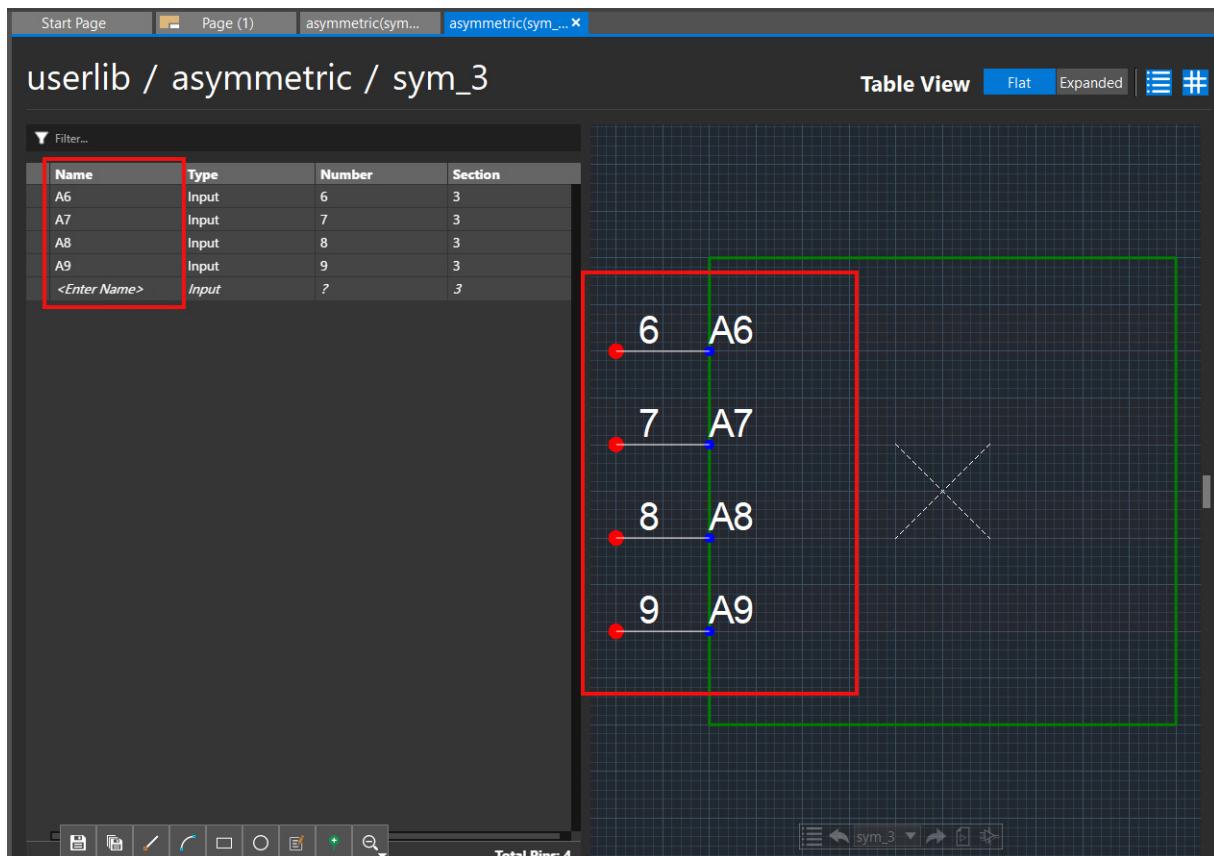
A new symbol is created and opened in a new tab. To view the next or previous sections, click the *Next Section* (➡) or *Previous Section* (⏪) buttons.



Library Authoring and Editing in Allegro X System Capture

Management of Library Parts

4. Add pins to all the symbols.



5. Select *File – Save All* to save the symbol.

The symbol is saved and displayed under the part in the *Library* explorer.

Related Topics

- [Creating a Category for a Specific Part](#)
- [Supported Characters in Part Names](#)
- [Adding Pins using Table View](#)

Creating Symmetrical Parts

An electrical part that has multiple sections represented by one symbol in a package is called a *symmetrical* part. It is also known as *DeMorgan equivalent* part. Each section of a symbol contains the same set of pins but different pin numbers. Any changes to the pins,

other than pin numbers, reflects in all the sections. In System Capture, you can create symmetrical parts by creating a duplicate copy of a section:

To create symmetrical parts, do the following:

1. Create a new part.
2. Create a symbol and add pins to the symbol.
3. Click the *Duplicate Section* icon at the bottom of the canvas.



A new section is created and is visible in *Table View*. This section contains all the pins from the parent section. To view the next or previous sections, click the *Next Section* (➡) or *Previous Section* (⬅) buttons.

You can also move to sections using the *Section* list in the *General* section of the Properties panel.

4. Add pin numbers for pins in the new section.
5. Select *File – Save All* to save the symbol.

The symbol is saved.

Related Topics

- [Creating a Category for a Specific Part](#)
- [Supported Characters in Part Names](#)
- [Adding Pins using Table View](#)
- [Adding Global Pins](#)

Creating Asymmetrical Parts

An electrical part that has multiple sections where each section represents different functions is called an *asymmetrical* part or multi-function sectional part.

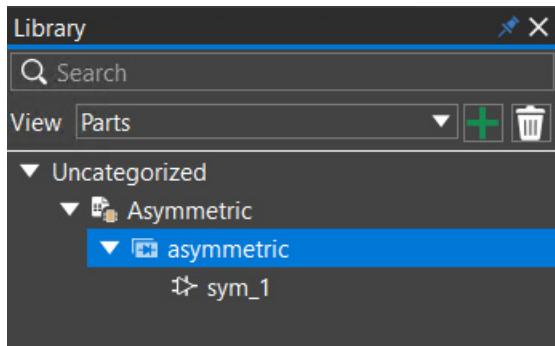
Such parts have one or more schematic symbols and must have a unique set of logical pins so that they can be packaged correctly. Each function in an asymmetrical part might share common and non-common pins.

Library Authoring and Editing in Allegro X System Capture

Management of Library Parts

To create an asymmetrical part, do the following:

1. Create a new part.
2. Create a symbol and add pins to the symbol.
3. Expand a part in the *Library* explorer and click the schematic node under the part name.



The consolidated symbols view is displayed in a new tab.

4. Click *Expanded* to change *Table view* to the expanded view.

The screenshot shows a table view titled 'userlib / asymmetric / sym_1'. At the top right are buttons for 'Table View', 'Flat', 'Expanded' (which is highlighted in blue), and other icons. The table has three columns: 'Name', 'Type', and 'Section 1'. The data rows are as follows:

Name	Type	Section 1
A1	Input	1
A10	Input	10
A2	Input	2
A3	Input	3
A4	Input	4
A5	Input	5
A6	Input	6
A7	Input	7
A8	Input	8
A9	Input	9
<Enter Name>	Input	?

5. Right-click the pins to be moved and select *Move To – New Section*.

A new section is created and the selected pins are moved to the section.

6. Right-click any of the section column headers and select *Duplicate*.

Library Authoring and Editing in Allegro X System Capture

Management of Library Parts

A duplicate copy of the selected section is created.

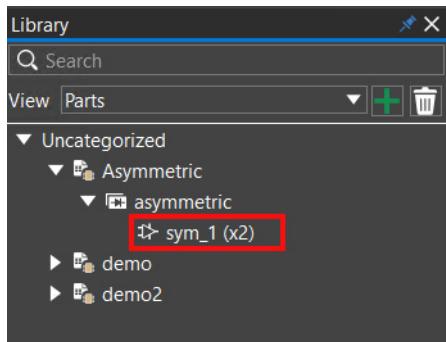
The screenshot shows a software interface for managing library parts. At the top, there are tabs for 'Start Page', 'Page (1)', and 'asymmetric(ASY... x)'. The main area is titled 'userlib / asymmetric / ASYMMETRIC'. On the right, there are buttons for 'Table View' (selected), 'Flat', 'Expanded', and a grid icon. A search bar labeled 'Filter...' is also present. The central part of the screen is a table with the following data:

Pin ID	Type	Section 1	Section 2	Section 3
A6	Input	6		
A7	Input	7		
A8	Input	8		
A9	Input	9		
A10	Input	10		
A1	Input		1	
A2	Input		2	
A3	Input		3	
A4	Input		4	
A5	Input		5	
<Enter Pin ID>		Input		

7. Add pin numbers to the duplicated section.

8. Save the part.

The asymmetrical part is created and displayed in the *Library* explorer. Symbols with duplicate sections are displayed as multiples of one symbol in the *Library* explorer.



Related Topics

- [Creating Electrical Flat Parts](#)
- [Creating a New Blank Symbol](#)
- [Adding Pins using Table View](#)
- [Adding Low Asserted Pins](#)
- [Adding Global Pins](#)

Library Authoring and Editing in Allegro X System Capture

Management of Library Parts

Supported Characters in Part Names

Only the following characters are supported in a part name:

Supported Characters	Description
A-Z and a-z	Letters
0-9	Numbers
-	Dash
@	At
#	Hash
\$	Dollar
^	Carat
&	Ampersand
_	Underscore
%	Percent
/	Slash

Related Topics

- [Creating Electrical Flat Parts](#)
- [Creating Electrical Split Parts](#)
- [Creating Symmetrical Parts](#)
- [Creating Asymmetrical Parts](#)

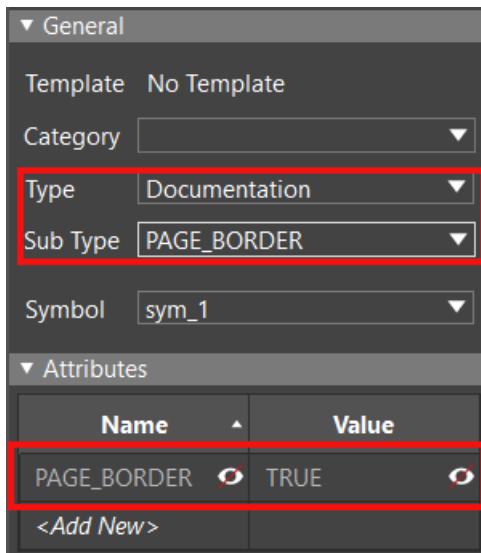
Creating Documentation Parts

Any part without pins is considered a non-electrical part. You can create non-electrical parts in Allegro X System Capture. Non-electrical parts are helpful in documenting information on the schematic, such as title blocks, page borders, and table of contents. Other non-electrical parts that help manipulate signals, such as power or port, can also be created within Allegro X System Capture.

To create a documentation part, do the following:

1. Create a new part.
2. Create a new symbol.
3. In the new symbol window, select *Documentation* from the *Type* list in the *General* section of the *Properties* panel.
4. Select the type of the documentation symbol from the *Sub Type* list.

When you select the *Sub Type*, the symbol-level properties are automatically attached to the symbol.



5. Make the required changes on the canvas.

For example, to create a documentation symbol, such as a title block or table of contents, you can place rectangles and add notes on the canvas. To create symbols that help manipulate signals such as power or ground symbols, change the symbol outline and add the required properties.

6. Save the symbol.

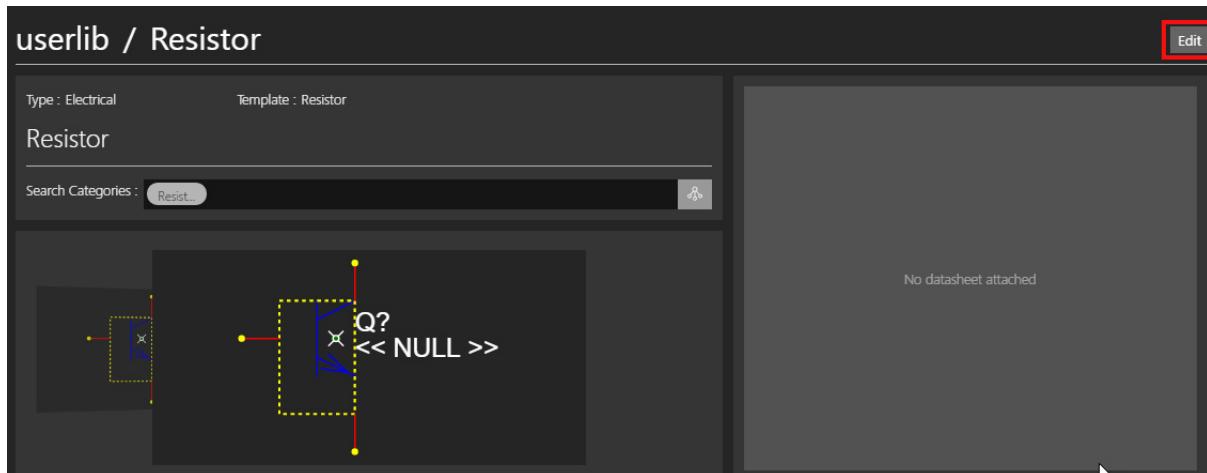
Related Topics

- [Creating a New Blank Symbol](#)
- [Adding Symbol Properties](#)

Associating Existing Symbols

After creating a new part, you can associate existing symbols with a new or existing part. To associate an existing symbol with a part, do the following:

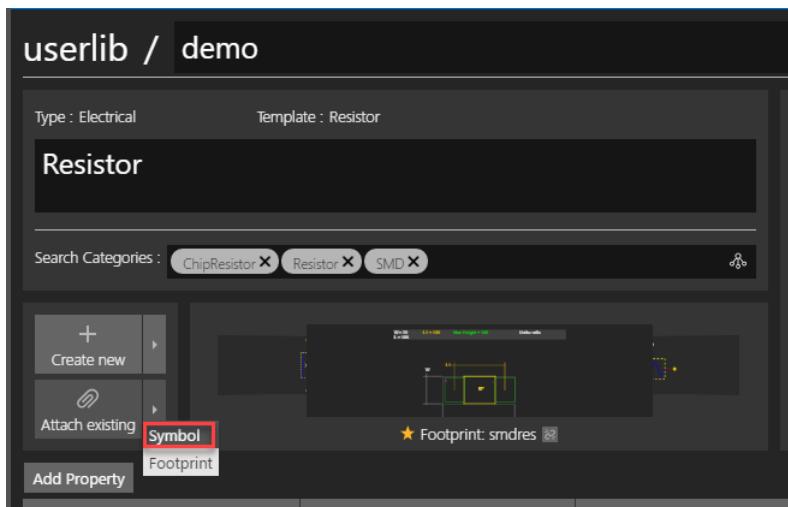
1. Open an existing part from the *Library* explorer.
2. Click the *Edit* button or create a new part.



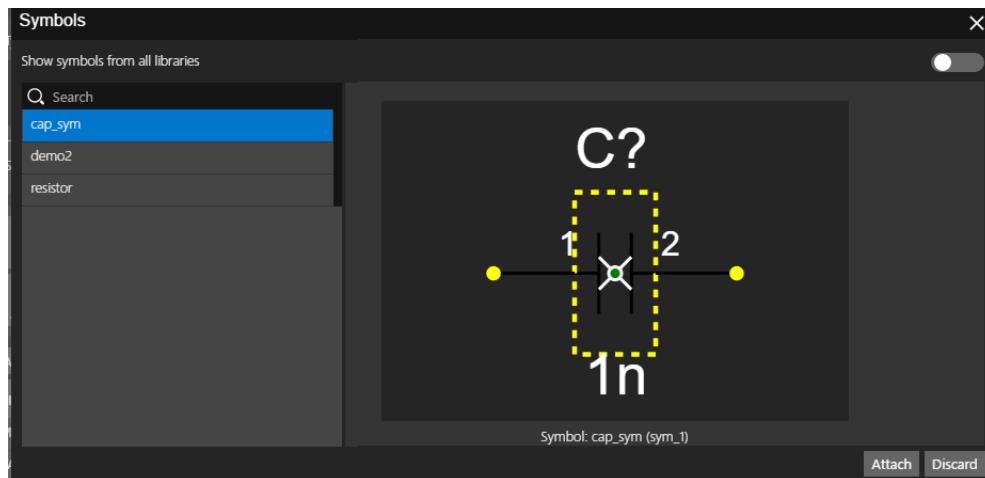
Library Authoring and Editing in Allegro X System Capture

Management of Library Parts

- Click the arrow on the *Attach existing* button and select *Symbol*.

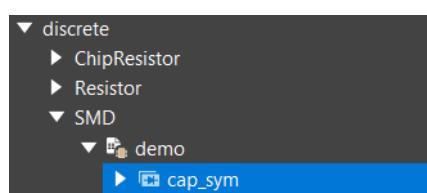


The *Symbols* dialog is displayed.



- Select the required symbol from the list and click *Attach*.
- Click *Save*.

The selected symbol is associated with the part and is displayed under the part in the *Library explorer*.



Related Topics

- [Creating a New Blank Symbol](#)
- [Creating Electrical Flat Parts](#)

Assigning a Footprint

You can also assign footprint information to a new or existing part so that you can refer to it later.

To assign a footprint to a part, do the following:

1. Right-click a part in the *Library* explorer and select *Open Part*.
The selected part window opens.
2. Click the *Edit* button to edit the part.
3. Click the arrow on the *Attach existing* button and select *Footprint*.

Note: In Allegro Unified library, *Symbol* and *Footprint* are the only options available in the drop-down list of the *Attach existing* button.

4. Select the footprint (.dra) file and click *Open*.
5. Click *Save*.

The selected footprint file is assigned to the part and a preview of the footprint is also displayed next to the *Attach existing* button.

Note: Footprint information is not stored in a .ldax file.

Related Topics

- [Creating a New Blank Symbol](#)
- [Creating Electrical Flat Parts](#)

Assigning a Datasheet

You can assign a datasheet to a new or existing part to refer to symbol details.

To assign a datasheet to a part, do the following:

1. Right-click a part in the *Library* explorer and select *Open Part*.

Library Authoring and Editing in Allegro X System Capture

Management of Library Parts

The selected part window opens.

2. Click the *Edit* button to edit the part.
3. Click  in the *Datasheet* section.



4. Browse to the required PDF file.
5. Click *Open*.

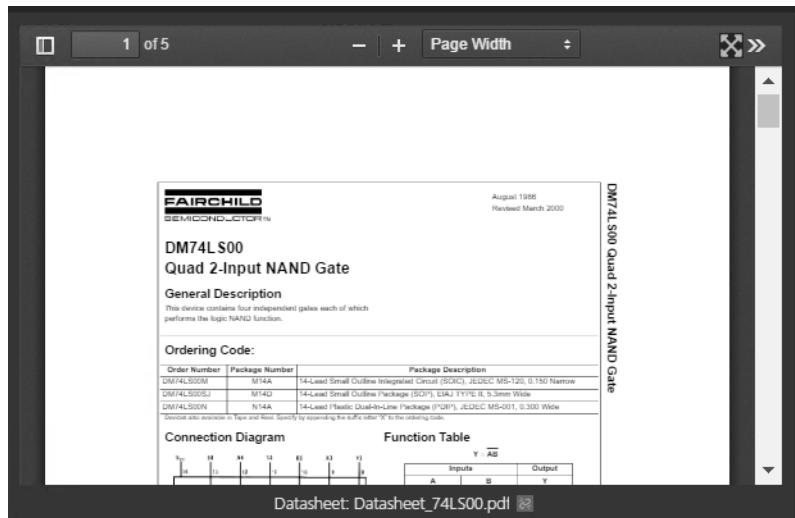
The selected datasheet file is assigned to the part. The assigned file can be viewed and read from the *Datasheet* section.

Note: You can also drag and drop a datasheet, which is in the PDF format, to the

Library Authoring and Editing in Allegro X System Capture

Management of Library Parts

Datasheet section.



Related Topics

- [Creating a New Blank Symbol](#)
- [Creating Electrical Flat Parts](#)

Viewing Details of a Specific Part

To view a part and its details, such as an associated symbol, datasheet, properties, and the footprint information, you can open the part from the *Library* explorer.

To open a specific part and view its details, do the following:

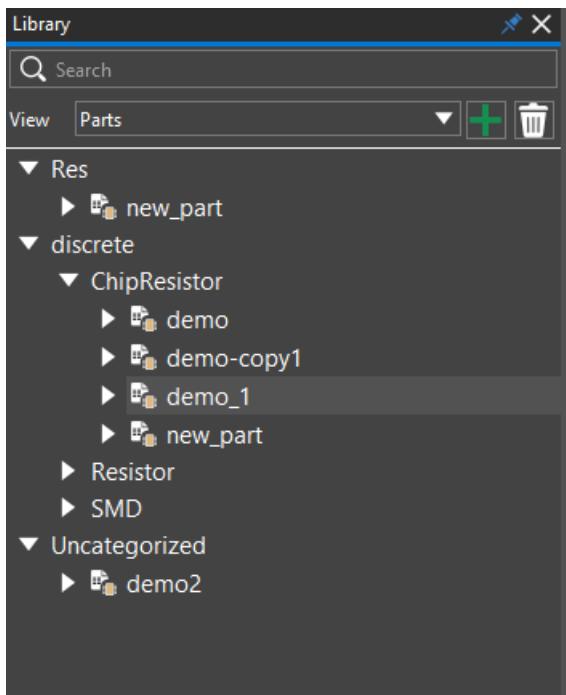
1. In the *Libraries* tree, right-click the library where the part is stored and select *Open Library*.

Alternatively, you can double-click a library to open it.

Library Authoring and Editing in Allegro X System Capture

Management of Library Parts

The *Library* explorer opens and displays the list of available parts in the library.



2. Right-click the part and select *Open Part* to open the part.

Alternatively, you can double-click a part in the *Library* explorer to open it.

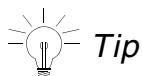
Library Authoring and Editing in Allegro X System Capture

Management of Library Parts

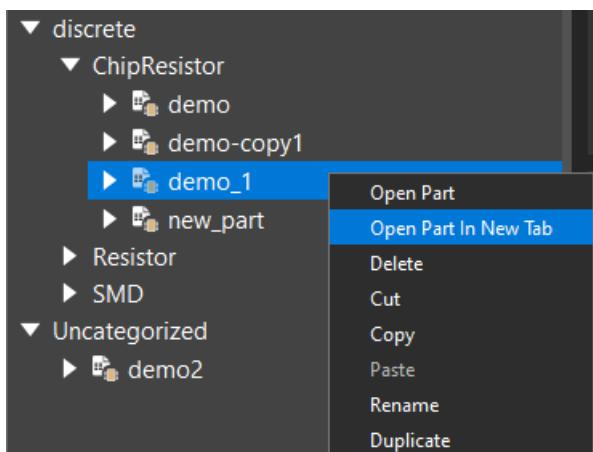
The part is opened in a new tab where you can see the part name, the symbols associated with the part, and properties. A footprint or datasheet assigned to the part is displayed in the *Footprint* and *Datasheet* sections.

The screenshot shows the Allegro X System Capture library editor interface. The top bar has tabs for 'Start Page' and 'demo_1(demo_1)'. The main area is titled 'userlib / demo_1'. It displays a 'Resistor' component with a symbol labeled 'R?' and '1K'. Below the symbol is the text 'Symbol: resistor (sym_1)'. To the right, there is a 'Datasheet' section with the message 'No datasheet attached'. At the bottom, there is a table of properties:

PROPERTY NAME	VALUE	DESCRIPTION	V	A	M	E
MPN		Manufacturer Part Number	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>
VALUE	3K	Resistance Value at 25 C	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>
CLASS	DISCRETE	Part Classification	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
MTBF		Mean Time Between Failures	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
MIN_TEMPERATURE	-40C	Minimum operating temperature	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
PART_NUMBER		Part number	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>



If you open another part from the same library, it opens in the same tab. To open the part in a separate tab, right-click the part and select *Open Part in New Tab*.



Library Authoring and Editing in Allegro X System Capture

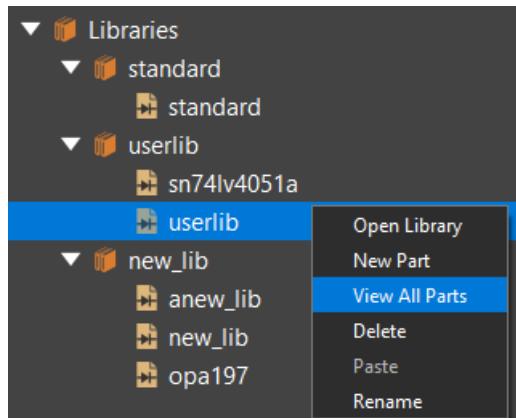
Management of Library Parts

Viewing All Parts in a Library

You can also view all parts and their details in a library.

To view a list of all the parts available in a library, do the following:

1. In the *Libraries* tree, right-click a library and select *View All Parts*.



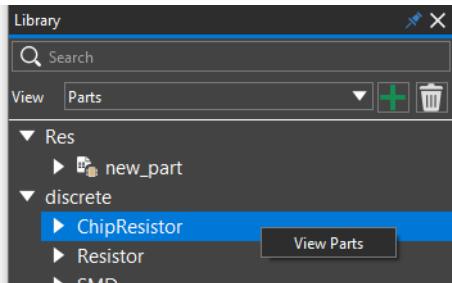
All the parts in the library are displayed in a list in the *Parts View* tab. You can double-click a part in this list to open the part.

NAME	MPN	CLASS	VALUE	MTBF	MIN_TEMPERATURE	PART_NUMBER	PART_NAME
demo		DISCRETE	2K		-55C		CAP_SYM
demo2							RESISTOR
demo_1		DISCRETE	3K		-40C		RESISTOR
demo-co...		DISCRETE	1K		-55C		CAP_SYM
new_part							RESISTOR

Library Authoring and Editing in Allegro X System Capture

Management of Library Parts

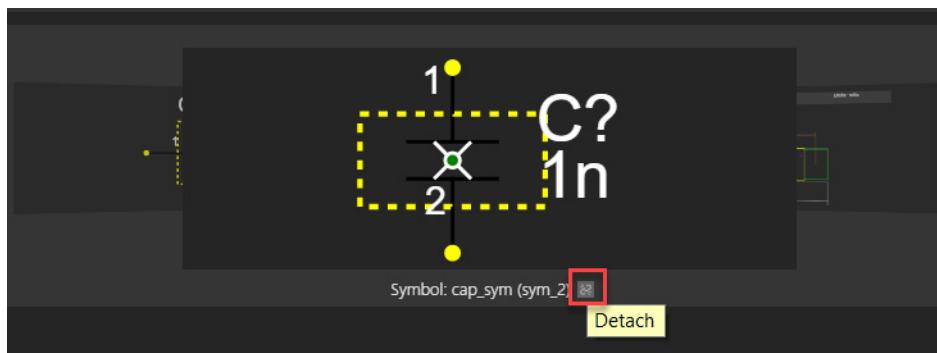
You can also display all the parts in a category by right-clicking the part category in the *Library* explorer and selecting *View Parts*.



Removing Symbols from a Part

To remove a symbol from a part it is associated with, do the following:

1. Right-click the part in the *Library* explorer and select *Open Part*.
The selected part window opens.
2. Click the *Edit* button to edit the part.
3. Click the *Detach* button next to the symbol name under the symbol preview.

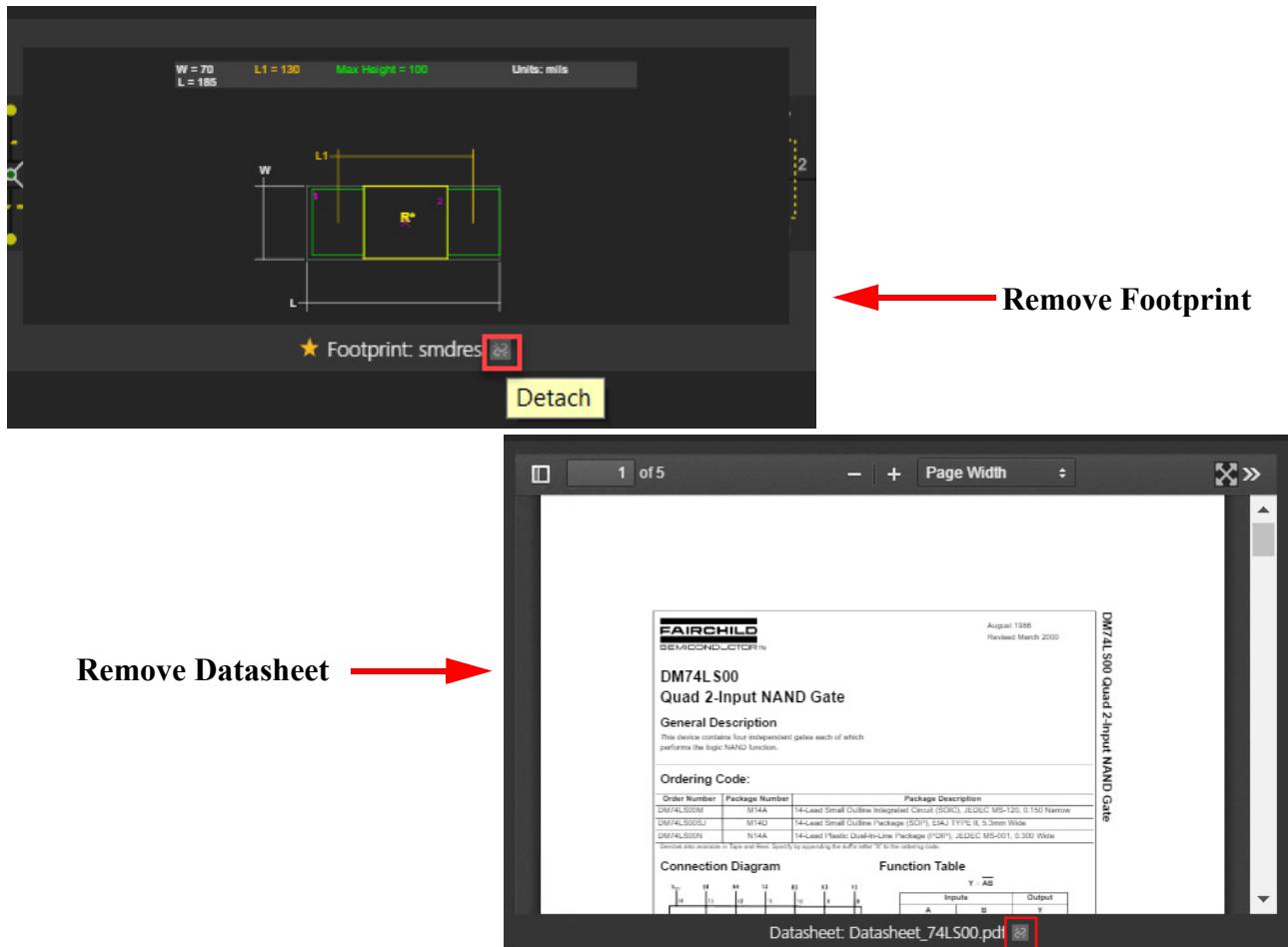


The associated symbol is removed from the part.

Library Authoring and Editing in Allegro X System Capture

Management of Library Parts

Similarly, you can remove footprint and datasheet assigned to a part.



Related Topics

- [Associating Existing Symbols](#)
- [Assigning a Footprint](#)
- [Assigning a Datasheet](#)

Renaming Parts

To rename a part, do the following:

1. Select a part in the *Library* explorer and do one of the following:

- a. Right-click the part and select *Rename* from the context menu.
 - b. Press F2.
 - c. Click the part.
2. Specify a new name for the part and press Tab or Enter.

The part is renamed as specified.

Related Topic

- [Supported Characters in Part Names](#)

Deleting Parts

To delete a part from a library:

- Right-click the part and select *Delete* from the context menu.
The part is immediately removed from the library.

Related Topic

- [Creating Electrical Flat Parts](#)

Deleting a Section

You can also delete specific sections of a multi-section part.

- To delete a section, right-click the section column header and select *Delete*.

This option is disabled in the *Flat* table view or if there is only one section in the symbol.

Related Topic

- [Creating Split Parts by Adding Multiple Blank Sections](#)

Adding Part Properties

In addition to the properties defined in a template, you can add more properties to a part.

Library Authoring and Editing in Allegro X System Capture

Management of Library Parts

To add properties to a part, do the following:

1. Right-click a part in the *Library* explorer and select *Open Part*.

The selected part window opens.

2. Click the *Edit* button to edit the part.

3. Click *Add Property* in the properties section.

Alternatively, click  in the properties section.

A new row is added to the properties section.

Add Property			
PROPERTY NAME	VALUE	DESCRIPTION	A V E
SYMBOL	SYM TEMPLATE-COPY	PART_NAME	<input checked="" type="checkbox"/> <input type="checkbox"/> <input type="checkbox"/>

4. Specify the name of the property in the *PROPERTY NAME* column.
5. Specify the value and description of the property in the *VALUE* and *DESCRIPTION* columns.
6. Select the required check boxes:
 - V*: to make the property visible on the schematic.
 - A*: to annotate the property when the schematic is packaged.
 - E*: to make the property editable on the schematic.
7. Click *Save*.

The property is added to the part and displayed in the *Properties* panel.

Note: When you save a new part, a property, *PART_NAME*, is automatically added to the properties section of the part.

Related Topic

- [Adding Symbol Properties](#)

Modifying Part Properties

You can edit or delete properties defined for a part in the properties section of the part window.

- To edit a property, double-click a cell of the property row and specify a new value.
- To delete a property, hover the cursor over the last column of the property row to be deleted and click the  button.

Related Topic

- [Adding Symbol Properties](#)

Copying a Part Across Libraries

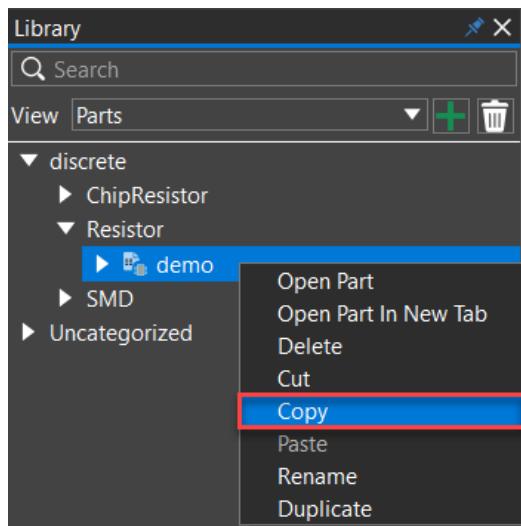
To reuse a library part, you can create copies of an existing Unified Library part across libraries in Allegro X System Capture. You can also duplicate a part within the same library.

To create a copy of a part, do the following:

1. In the *Libraries* tree, double-click the source library.

The *Library* explorer is displayed with a list of available parts in the library.

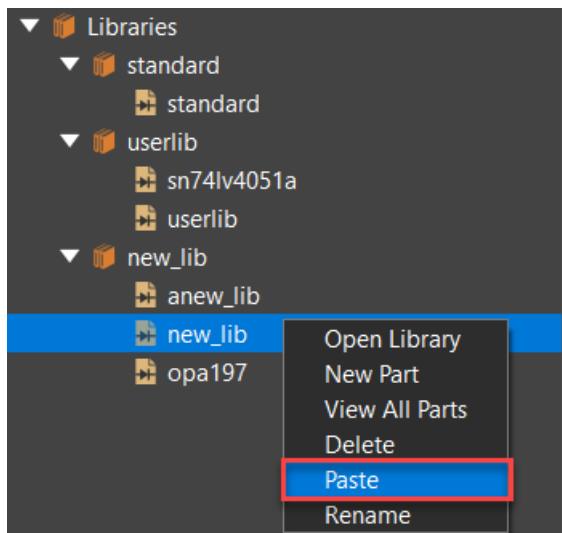
2. Expand a category to view parts in the category.
3. Right-click a part and select *Copy*.



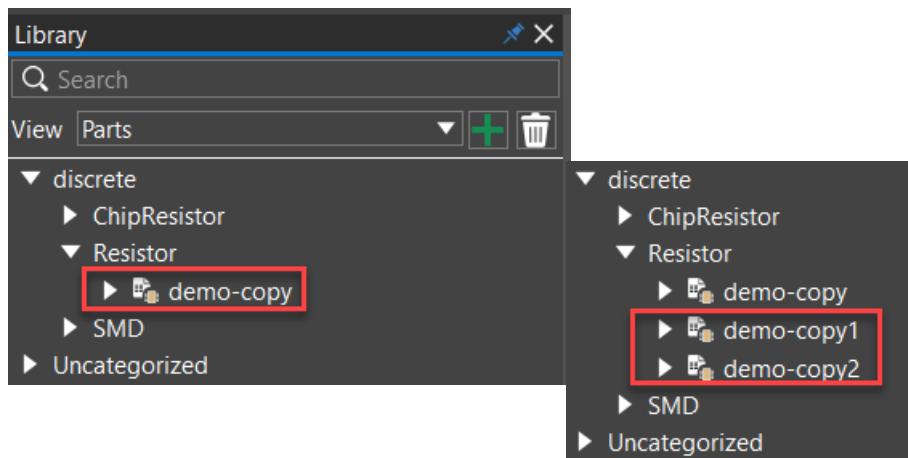
Library Authoring and Editing in Allegro X System Capture

Management of Library Parts

- Back in the *Libraries* tree, right-click the target library and select *Paste*.



The part is added in the selected library with the same name, followed by the *-copy* suffix. If you paste again, the suffix is incremented by 1.



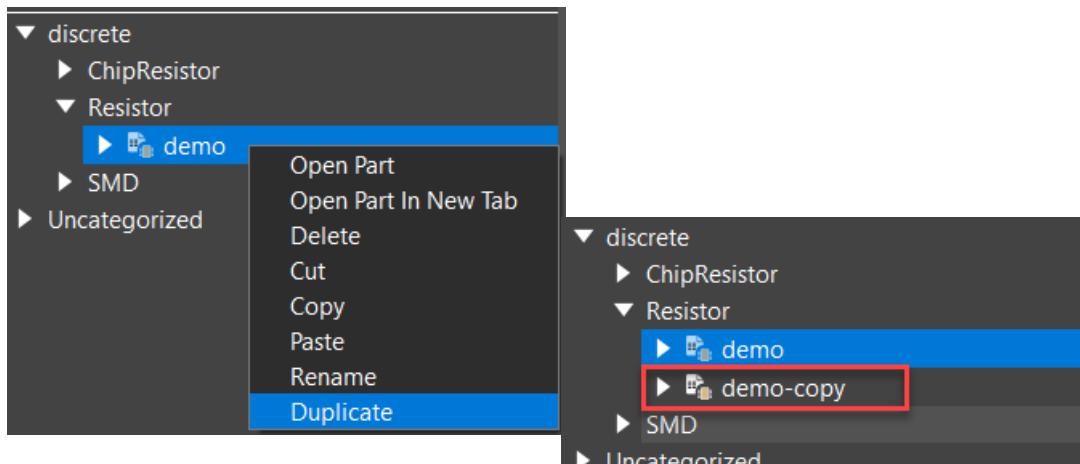
Related Topic

- [Creating Electrical Flat Parts](#)

Duplicating Parts in the Same Library

To create a copy of an existing part in the same library, you can duplicate a part.

- To duplicate a part, right-click a part in the *Library* explorer and select *Duplicate*.
A duplicate copy of the part is added to the same library.



Related Topic

- [Creating Electrical Flat Parts](#)

Library Authoring and Editing in Allegro X System Capture

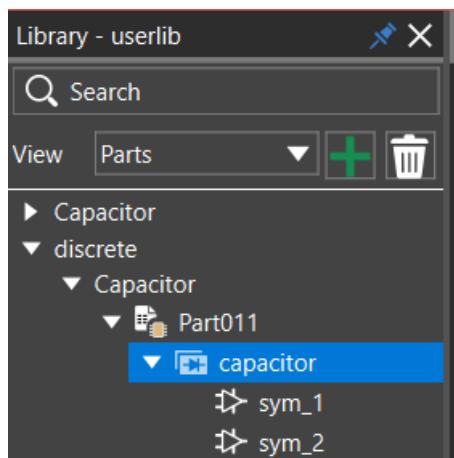
Management of Library Parts

Adding Global Pins

When creating multi-symbol parts, you can add global pins, such as power-implicit or ground-implicit pins, to a symbol.

To add global pins to a symbol, do the following:

1. Expand a part in the *Library* explorer.
2. Click the schematic node under the part name.



The consolidated symbols view is displayed in a new tab.

Pin ID	Type	Number	Section
1	Input	1	1
2	Input	2	1
<Enter Pin ID>	Input		1

3. Double-click *Enter Pin ID*.
4. Type the pin name and press *Enter*.

Library Authoring and Editing in Allegro X System Capture

Management of Library Parts

5. Select an implicit pin type from the *Type* column for the newly added pin.

6. Specify the pin number in the *Number* column.

The global pin is added to all the symbols of the part and can be viewed from the Table View of the symbols.

Symbol 1

Global Pin

Symbol 2

Name	Pin ID	Type	Number	Section
A1	A1	Input	1	1
A2	A2	Input	2	1
A3	A3	Input	3	1
A4	A4	Input	4	1
A5	A5	Input	5	1
VCC	VCC	Power-Implicit	11	
<Enter Pin Text>		<Enter Pin Name>	Input	?

Name	Pin ID	Type	Number	Section
A10	A10	Input	10	2
A6	A6	Input	6	2
A7	A7	Input	7	2
A8	A8	Input	8	2
A9	A9	Input	9	2
VCC	VCC	Power-Implicit	11	
<Enter Pin Text>		<Enter Pin Name>	Input	?

Related Topic

- [Adding Pins using Table View](#)

Editing Multiple Parts in a Grid

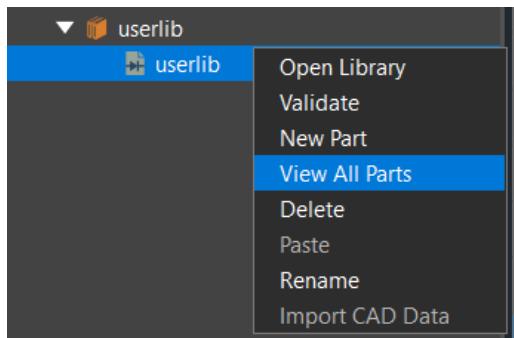
You can also edit library parts from a list in a spreadsheet view. You can update part properties, schematic model, footprint, simulation model, datasheets, and other columns.

To open the spreadsheet view, do the following:

Library Authoring and Editing in Allegro X System Capture

Management of Library Parts

1. Right-click a library and select *View All Parts*.



The spreadsheet view of the parts is displayed.

Library : userlib Parts loaded : 4										Edit	Load more	Filters	▼
STATUS	NAME	CLASSIFICATI...	SUBTYPE	SYMBOL	MARKED DELETED	PART TEMPL...	DESCRIPTION	COMMENT	SYMBOL				
	part_01		Electrical	sym1			new_part		SYM1				
	new_part01	Capacitor	Electrical										
	new_part		Electrical	sym_template		part_temp	Part Template		SYM_TEMPLATE				
	Part_SymT...		Electrical	sym_template					SYM_TEMPLATE				

2. Click the *Edit* button and update the fields, as required.

Library : userlib Parts loaded : 4										Save	Cancel	Load more	Filters	▼
STATUS	NAME	CLASSIFICATI...	SUBTYPE	SYMBOL	MARKED DELETED	PART TEMPL...	DESCRIPTION	COMMENT	SYMBOL					
✗	part_01		Electrical	sym1			new_part		SYM1					
✗	new_part01	Capacitor	Electrical											
✗	new_part		Electrical	sym_template		part_temp	Part Template		SYM_TEMPLATE					
✗	Part_SymT...		Electrical	sym_template				new_part	SYM_TEMPLATE					

3. Click *Save*.

Related Topic

- [Validating Multiple Parts](#)

Category Management

When working on a System Capture project created in the *Allegro Unified* library mode, a classification of models is required to categorize parts, symbols, footprints, pad stacks, and other data models. It improves searchability of components based on their categories.

For example, you can categorize a schematic model, resistor, as *Discrete*, which can be sub-classified as *Resistor*. You can search for a keyword, *Discrete* or *Resistor*, and assign it to a part or a symbol.

You can create a new category or sub-category for a specific part while creating the part or create a category tree within a library that can further be assigned to any part or symbol.

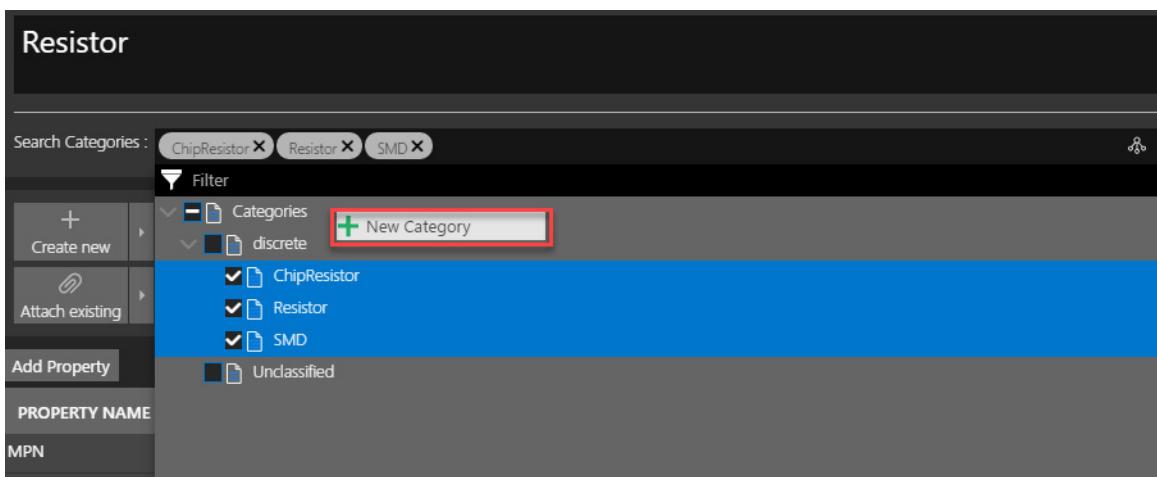
Related Topics

- [Creating a Category for a Specific Part](#)
- [Creating a Category within a Library](#)

Creating a Category for a Specific Part

To create a new category or sub-category, do the following:

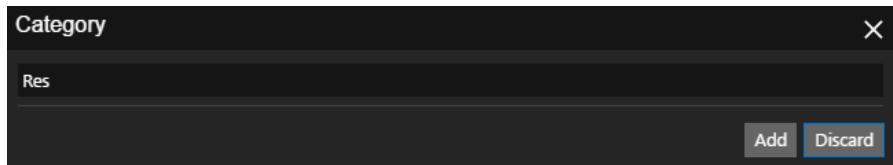
1. Click  in the *Search Categories* field.
2. Right-click *Categories* or an existing category, and select *New Category*.



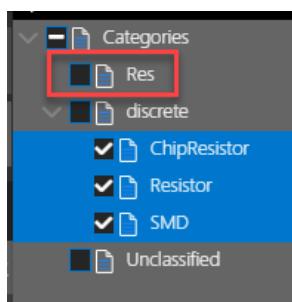
Library Authoring and Editing in Allegro X System Capture

Management of Library Parts

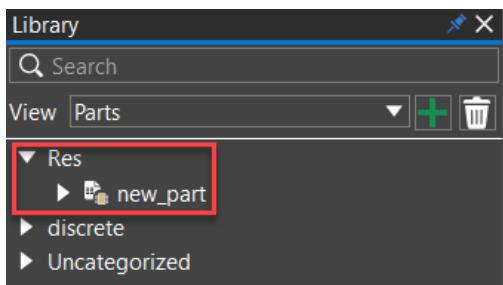
3. Specify the name in the *Category* dialog box and click *Add*.



A new category is added to the list.



4. Select the check box for the newly created category to associate the part with the category.



Creating a Category within a Library

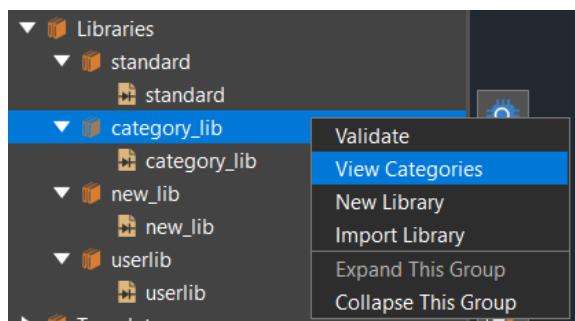
To create a new category or a sub-category, do the following:

1. In the *Library* explorer panel, select a library container.

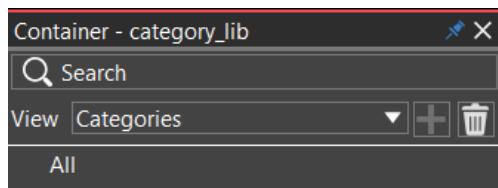
Library Authoring and Editing in Allegro X System Capture

Management of Library Parts

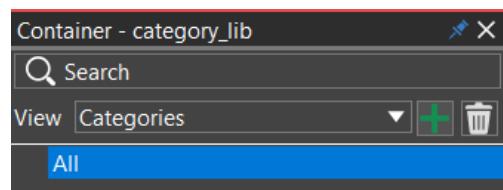
- Right-click the library container and select *View Categories*.



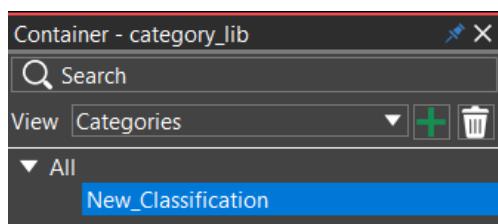
The *Categories* view is displayed.



- Select *All* and click the *New Category* button().



- Specify a name for the new category, which is *New Classification* by default, and save it.

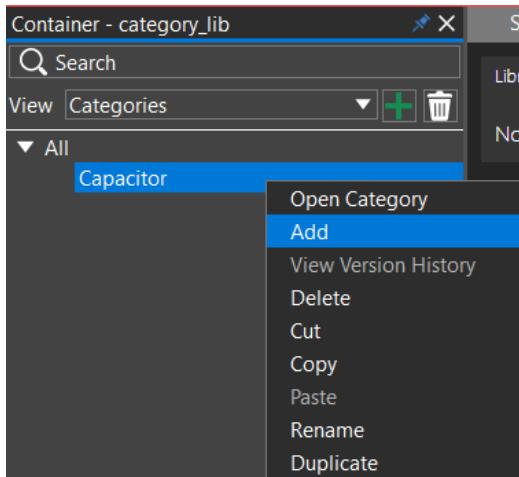


A new category is created and displayed under the category tree.

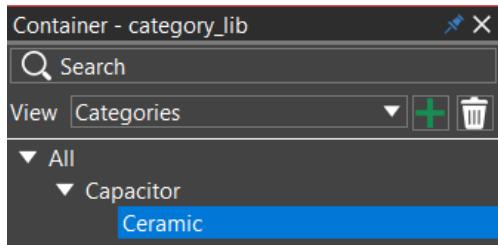
Library Authoring and Editing in Allegro X System Capture

Management of Library Parts

5. Right-click the new category and click *Add* to create a sub-category.



A sub-category is created and displayed under the *Category* tree.



Note: You can rename, delete, duplicate, cut or copy any category or sub-category, if required.

Assigning a Category to a Symbol

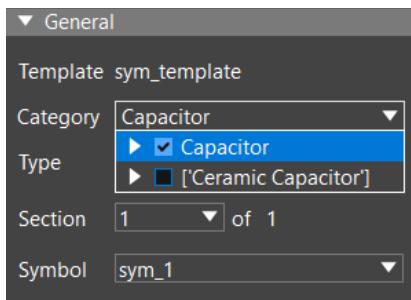
You can assign a category to an existing part or a symbol from the *Category* tree. To assign a category to a part, do the following:

1. Open a library and switch to the *Part* view.
2. Expand a part node and open a symbol.

Library Authoring and Editing in Allegro X System Capture

Management of Library Parts

3. In the *General* section of the *Properties* panel, you can select a category for a part or a symbol.



4. Click *Save*.

A new category is assigned to a symbol.

Adding Parts from External Content Providers

You can add parts from external content providers such as SamacSys or Ultra Librarian to a library and use them in your design. A part can be placed in a design only if it has a symbol and a footprint.

To create a new part, do the following:

1. You are required to log in to Unified Search content providers *SamacSys* and *Ultra Librarian*.

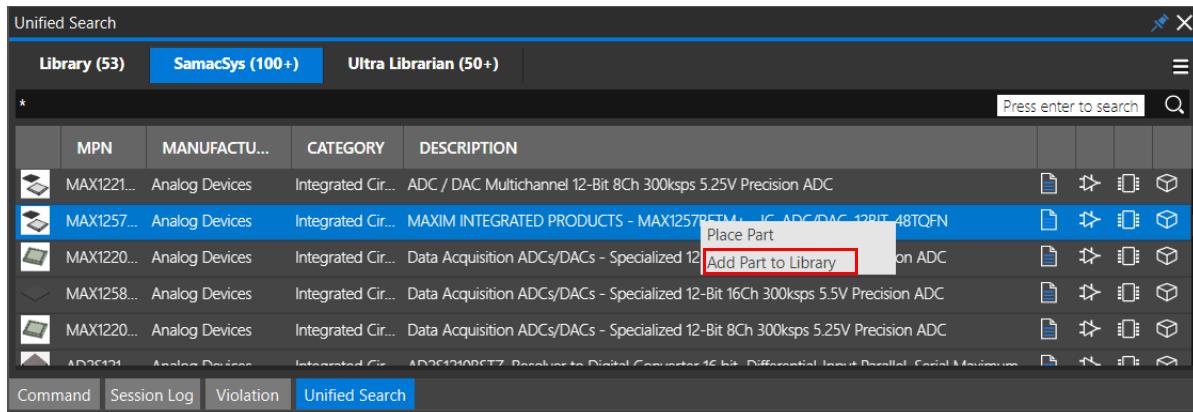
To learn how to login to *SamacSys* and *Ultra Librarian*, refer to the *Logging in to Unified Search Content Providers* section in *Unified Search User Guide*.

After signing in, parts from the selected external content provider are displayed.

Library Authoring and Editing in Allegro X System Capture

Management of Library Parts

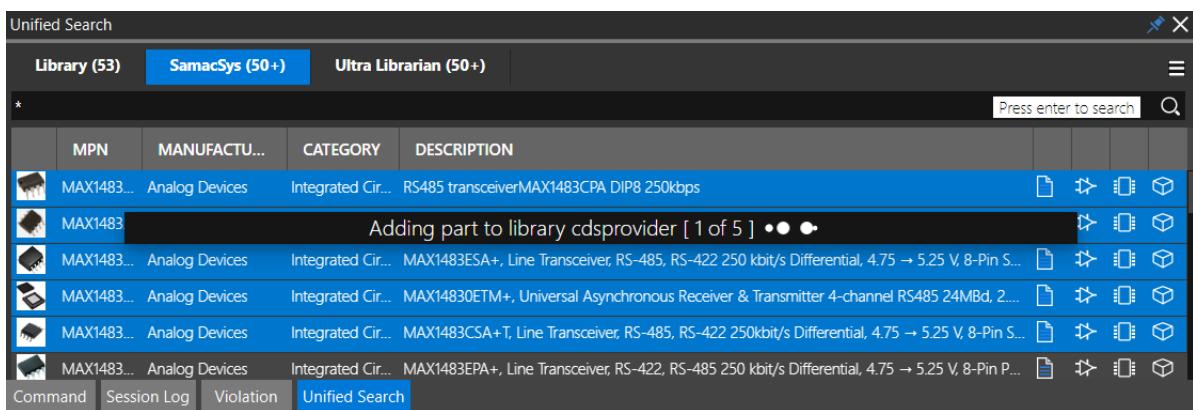
- Right-click a part you want to add and select *Add Part to Library*.



The downloading process starts.

	MPN	MANUFACTU...	CATEGORY	DESCRIPTION
	MAX1221...	Analog Devices	Integrated Cir...	ADC / DAC Multichannel 12-Bit 8Ch 300ksps 5.25V Precision ADC
	MAX1257...	Analog Devices	Integrated Cir...	MAXIM INTEGRATED PRODUCTS - MAX1257BETM+-12-Bit 16-Ch 300ksps 5.25V Precision ADC
	MAX1220...	Analog Devices	Integrated Cir...	Data Acquisition ADCs/DACs - Specialized 12-Bit 8Ch 300ksps 5.25V Precision ADC

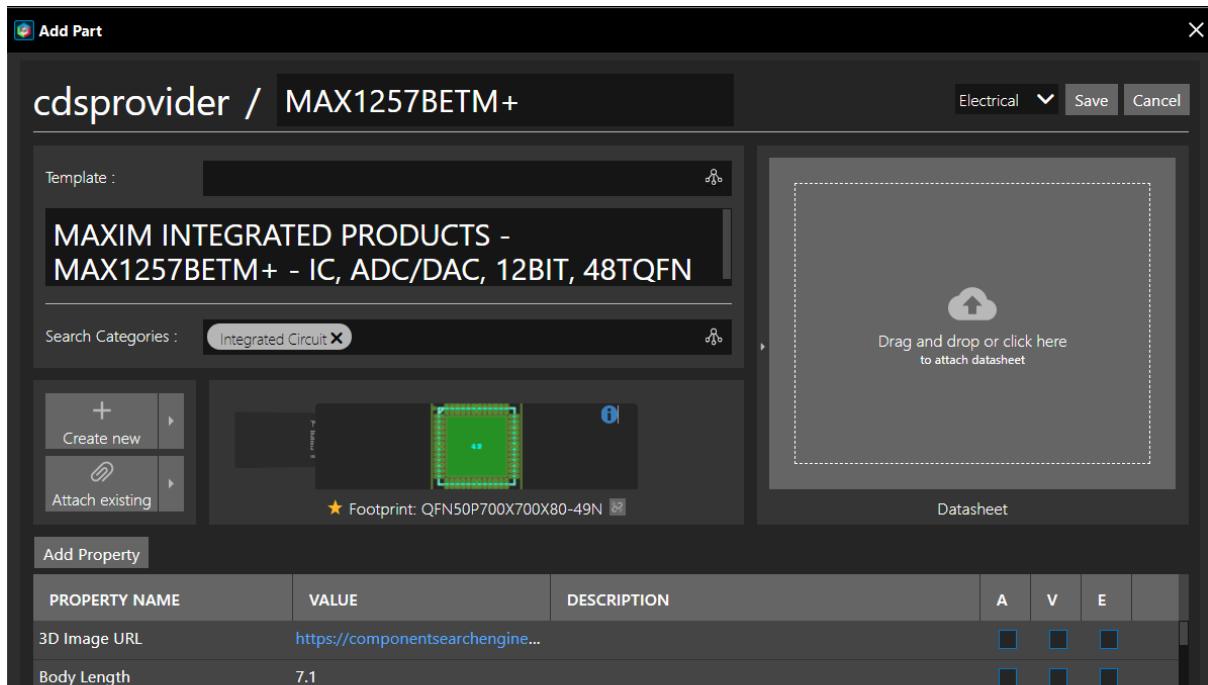
Note: From an external content provider, you can select a maximum of 5 parts at a time and add them to a library in one go.



Library Authoring and Editing in Allegro X System Capture

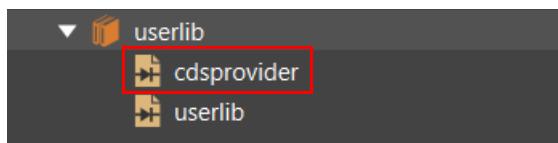
Management of Library Parts

After a part is downloaded, the *New Part* dialog box is displayed with the part details, properties, and footprint associated with it.



3. Click Save in the *Add Part* dialog box.

The part is saved in a *cdsprovider* library created under the library container, *Userlib*.



4. Open the *cdsprovider* library to see the newly added part. You can edit, delete or rename the part, if required.

Library Authoring and Editing in Allegro X System Capture

Management of Library Parts

Note: You cannot duplicate a part created by an external content provider.

The screenshot shows the Allegro X System Capture interface for managing library parts. On the left, a tree view lists various integrated circuits, with 'MAX1257BETM+' and its sub-part 'max1257betm_' selected and highlighted with a red box. The main area displays a table with columns: Pin ID, Type, Number, and Section. The table includes rows for CNVST/AIN15, GPIOA0, GPIOA1, EOC*, GPIOA2, GPIOA3, DVDD, and DGND. A schematic diagram titled 'Section1 sym_1' is shown on the right, featuring a complex circuit with multiple pins labeled AIN1 through AIN8, AIN9 through AIN16, and various control and power pins. A note at the bottom right of the schematic says 'Total Pins: 49'.

- After saving the part under *cdsprovider* library, you can search the part in the *Library* pane of the *Unified Search* window and place it on the schematic.

The screenshot shows the Allegro X Unified Search window. The search term 'max1257betm_' is entered in the search bar. The results table has columns: PART NUMBER, CATEGORIES, FOOTPRINT, and SUMMARY. The first result is 'MAX1257BETM+', categorized as an Integrated Circuit, with a footprint of QFN50P700X70... and a summary indicating it is a MAXIM INTEGRATED PRODUCTS - MAX1257BETM+ - IC, ADC/DAC, 12BIT, 48TQFN. The version is 0.8 and it is released. Below the table, the search term 'max1257betm_' is repeated.

Library Authoring and Editing in Allegro X System Capture

Management of Library Parts

Validation of Models

Libraries and data models including parts, schematic models, footprints, data sheets must be validated to avoid common design errors.

Working with projects created using the *Allegro Unified* library mode, you can validate libraries, parts, and schematic models based on their respective rules defined in the *Library Audit Settings* menu of Allegro X System Capture. You can configure the report type of the specified rules as *Error*, *Warning*, or *Information*.

The Library Validation window reports validation checks performed for the libraries, parts, or schematic models. The Session Log window provides a summary of the validation report analysis.

Working in the *Allegro Unified* library mode, you can do the following validation tasks:

- [Validating a Part](#)
- [Validating a Symbol](#)
- [Validating a Library](#)

Validating a Part

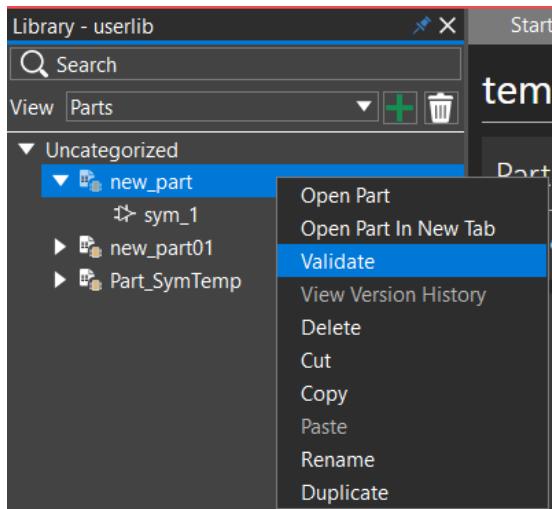
When you run validation for a part, the rules for the parts and the schematic symbol associated with the part are checked. To run validation for a part, do the following:

1. Open a library and select the *Part* view.

Library Authoring and Editing in Allegro X System Capture

Validation of Models

- Right-click the part and select *Validate*.



After the part is validated, a report is generated and displayed in the *Library Validation* window.

Status	View	Source	Message	Type
Passed	part	new_part01	Part name 'new_part01' has valid syntax.	Error
Passed	part	new_part01	Part 'new_part01' has a category assigned.	Warning
Passed	part	new_part01	Part 'new_part01' has mandatory properties.	Error
Passed	part	new_part01	Part 'new_part01' has mandatory properties with valid values.	Error

You can also check the status of the validation run in the *Session Log* window.

Session Log
CDS_SITE=C:/MYLIB CPCB-148: 'test_add_part_lib' project successfully created. Validation Started (userlib::userlib::new_part01) Validation Ended D:/Cadence/SPB_23.1/tools/bin/test_add_part_lib/libraryValidationReports//Run_1_4464fb3a-fa5d-4a55-b232-cea0f106655a.txt
Session Log
Library Validation

Library Authoring and Editing in Allegro X System Capture

Validation of Models

Configuring Library Audit Settings

Library rules are displayed in the list of audit rules accessed from the *Library Audit Settings* dialog box. There are default, predefined rules that are used for auditing a library, parts, or schematic models. You can do the following to customize the level of a library audit:

1. In a Allegro Unified project, open a library from *Library* explorer.
2. Select *Part Integrity – Library Audit Settings*.

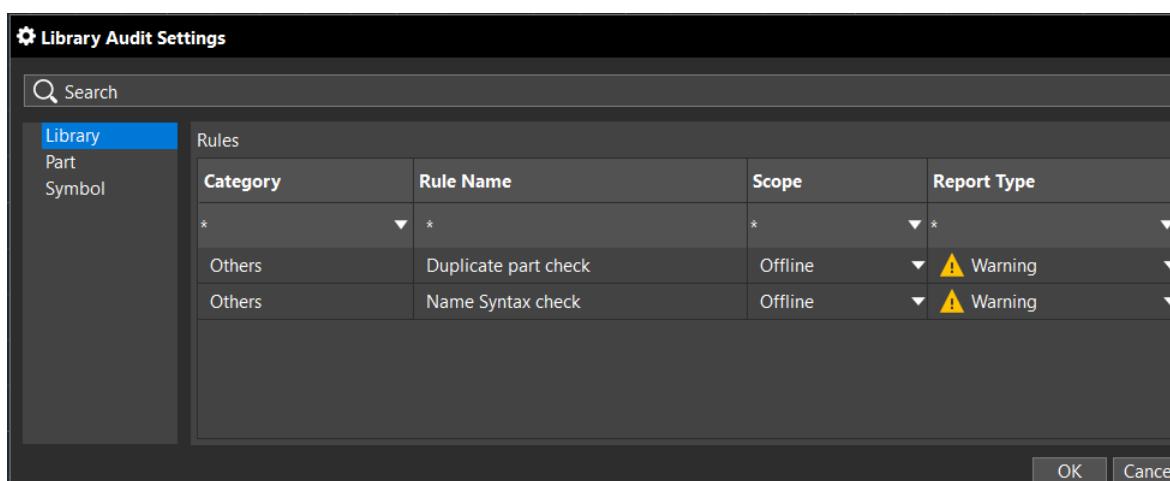


The *Library Audit Settings* dialog box is displayed.

3. You can configure the scope of the rule as *Offline* or *Do not run*.

Rules				
Category	Rule Name	Scope	Report Type	
*	*	*	*	
Others	Duplicate part check	Offline	⚠ Warning	
Others	Library name syntax check	Offline	⚠ Warning	

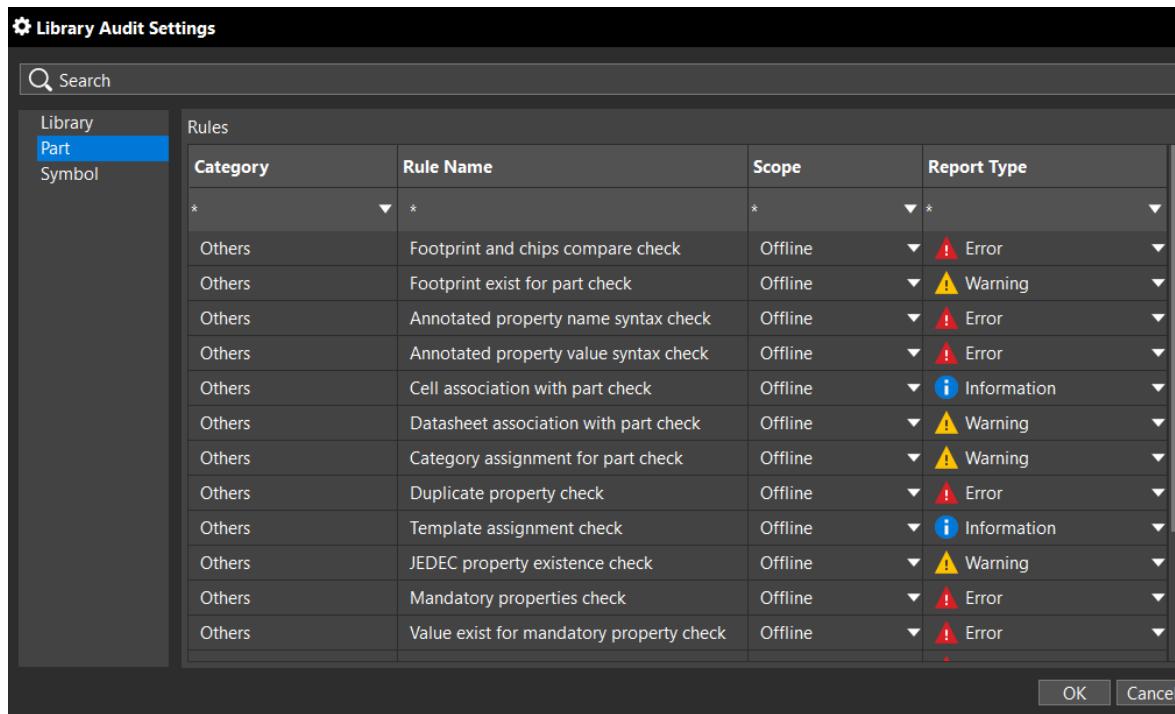
4. Define the severity of a rule as an error, warning, or information message as follows:
 - a. In the *Library* tab, define the severity of a rule for a library.



Library Authoring and Editing in Allegro X System Capture

Validation of Models

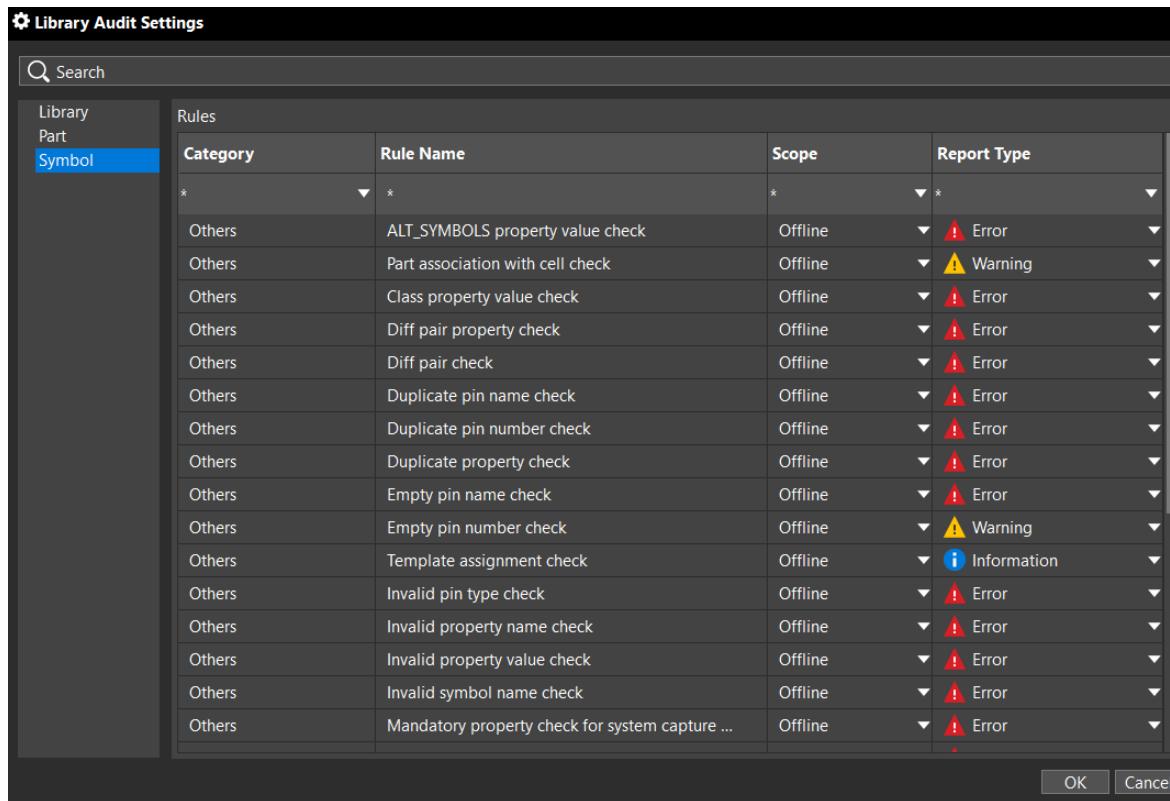
- b. In the *Part* tab, define the severity. Parts are validated based on this severity.



Library Authoring and Editing in Allegro X System Capture

Validation of Models

- c. In the *Symbol* tab, define the severity of a rule to validate symbols.



5. Click *OK* to save the changes.

Configuring Library Validation Window

Status, *View*, *Source*, *Message*, and *Type* are default column headers in the *Library Validation* window. However, you can add more columns and show or hide the columns, if required.

To display a column, do the following:

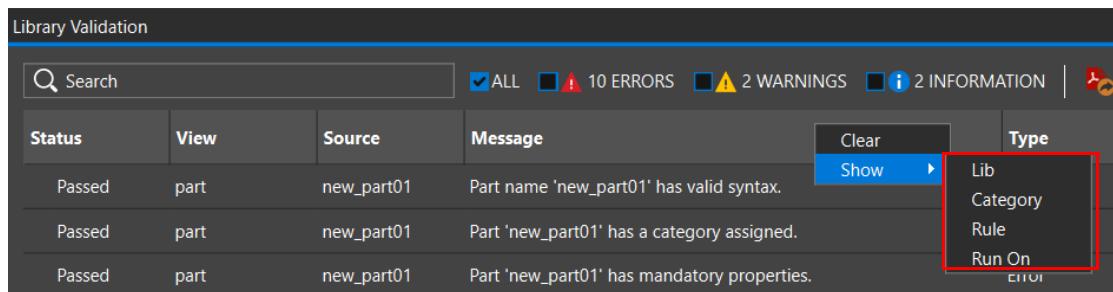
1. Right-click the column header and click *Show*.

Columns are displayed in the drop-down list.

Library Authoring and Editing in Allegro X System Capture

Validation of Models

2. Select the columns to make them visible in the *Library Validation* window.



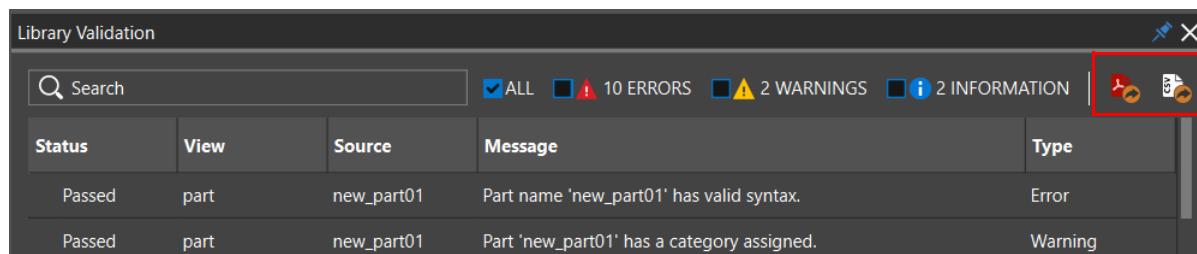
Status	View	Source	Message	Clear	Type
Passed	part	new_part01	Part name 'new_part01' has valid syntax.	Show ▾	Lib Category Rule Run On Error
Passed	part	new_part01	Part 'new_part01' has a category assigned.		
Passed	part	new_part01	Part 'new_part01' has mandatory properties.		

Exporting a Validation Report

You can also export the validation report as PDF and CSV formats from the Library Validation window. This way, you can quickly analyze the reported violations and make the changes on the schematic.

To export the report in PDF or CSV format, do the following:

1. Click the *Export to PDF* or *Export to CSV* button in the *Library Validation* window.



Status	View	Source	Message	Type
Passed	part	new_part01	Part name 'new_part01' has valid syntax.	Error
Passed	part	new_part01	Part 'new_part01' has a category assigned.	Warning

2. Click the *Browse* button and navigate to the directory where you want the file exported.
3. Specify a name for the file.
4. Click *Save*.

The validation report is exported in a PDF or CSV format.

Validating Multiple Parts

To save time, you can also validate more than one part at a time. To run validation for multiple parts, do the following:

1. Open a library in the *Parts* view.
2. Click SHIFT+DOWN ARROW to select multiple parts.
3. Right-click and select *Validate*.

All the selected parts and associated symbols are validated and reported in the *Library Validation* window. You can also search the reports based on keyword matches from the search tab.

Related Topic

- [Validating a Part](#)

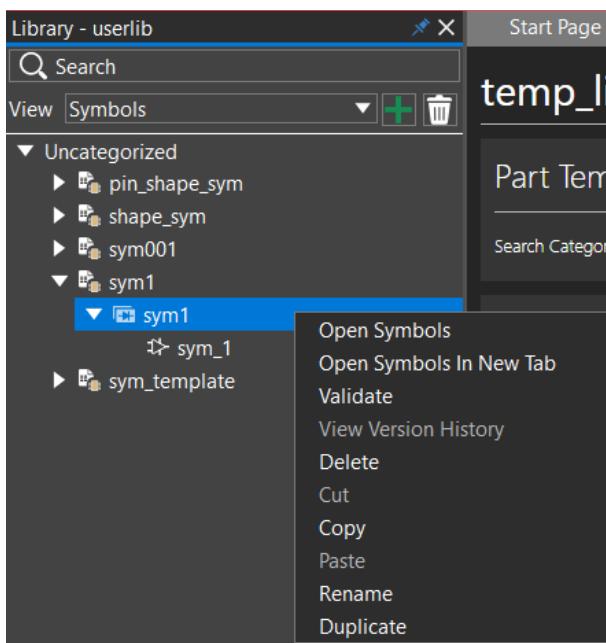
Validating a Symbol

When you run validation for a schematic model, it checks the symbols associated with the schematic model, and the pins associated with the symbol. To run validation for a symbol, do the following:

1. Open a library and switch to the *Symbol* view.

Note: Symbols can also be validated from the *Part* view mode.

2. Select a symbol and click *Validate*.



Library Authoring and Editing in Allegro X System Capture

Validation of Models

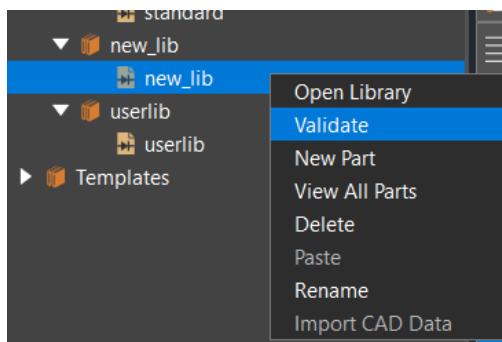
The schematic models are validated and reported in the *Library Validation* window.

Validating a Library

Running validation for a library validates all the parts available in the that library and generates a report.

To run validation for a library, do the following:

- Right-click a library and select *Validate*.



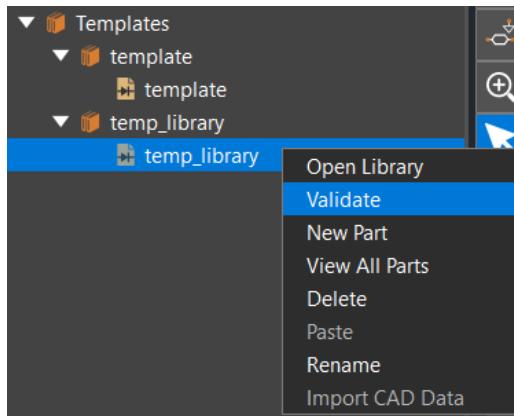
After the library is validated, all the validation checks are reported in the *Library Validation* window.

Library Validation			
Status	View	Source	Message
Passed	system_capture_...	shape_template	System capture model name shape_template has valid syntax
Passed	system_capture_...	shape_template	System capture model name shape_template has valid property names
Passed	system_capture_...	shape_template	System capture model name shape_template has valid property values
Passed	system_capture_...	shape_template	System capture model name shape_template has all symbols associated with package
Failed	part	part_temp	Part part_temp does not have a template assigned
Failed	part	part_temp	Part part_temp does not have a category
Failed	part	part_temp	Part part_temp is not associated with any datasheet
Failed	part	part_temp	Part part_temp does not have JEDEC_TYPE property

Library Authoring and Editing in Allegro X System Capture

Validation of Models

You can perform the same steps to validate a template library created in under *Allegro Unified* library mode.



Working with Legacy Libraries

You can reuse existing OrCAD Capture libraries in Allegro X System Capture. You can also view changes made to imported parts to ensure that they can be used in Allegro X System Capture.

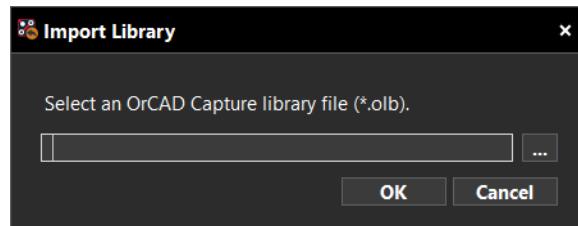
Importing OrCAD Capture Libraries to Unified Library

To use parts from an OrCAD Capture library in a System Capture design, you can import OrCAD Capture library parts into Allegro Unified Library. If there are no Unified Libraries added to the design, the parts are imported to the default `userlib.1dax` library container. You can also import parts to a new library container.

Note: *Import Library* option is available on library container node only.

To import an OrCAD Capture `.olb` library, do the following:

1. Right-click a library container node in the *Libraries* tree and select *Import Library*.
The *Import Library* dialog box is displayed.

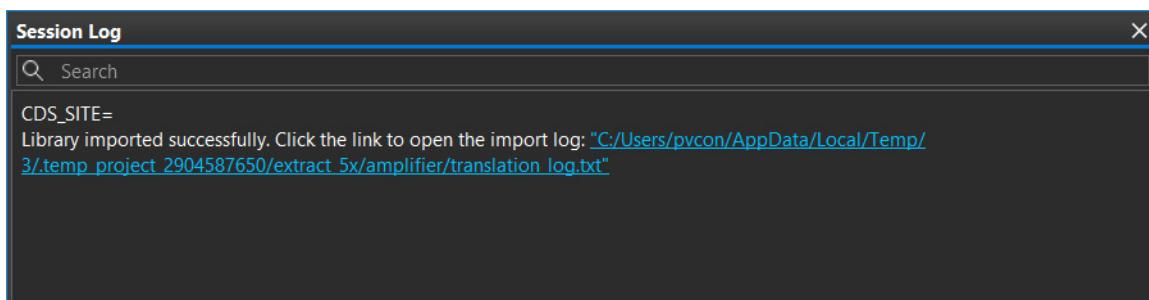


2. Click and navigate to the folder containing the `.olb` file to be imported.
3. Click *OK*.

Library Authoring and Editing in Allegro X System Capture

Working with Legacy Libraries

A confirmation message is displayed in the *Session Log* window.



The library data is imported into the default library container and the library is displayed in the *Libraries* tree.

4. To view the imported parts, right-click a library, and select *View All Parts*.
5. To open any part from the list, double-click the part.

All the imported library parts are immediately available in *Unified Search* for placement on a schematic.

Import Log

After the import is complete, a link to a import log file is displayed in the *Session Log* panel. This file contains a list of the imported parts and details about the import. The following screenshot displays a sample log file:

A screenshot of a Notepad window titled 'translation_log - Notepad'. The window contains a table with four columns: 'Library Name', 'Part Name', 'Message Type', and 'Description'. The data in the table is as follows:

Library Name	Part Name	Message Type	Description
amplifier	NJU7082B/SO	Information	sym_1 is being created.
amplifier	NJU7082B/SO	Information	Property name 'PART_REFERENCE' is converted to 'PART_REFERENCE'
amplifier	NJM2204A	Information	sym_1 is being created.
amplifier	NJM2204A	Information	Property name 'PART_REFERENCE' is converted to 'PART_REFERENCE'
amplifier	NJM2204A	Information	Pin name 'V+' is converted to 'V+#16'
amplifier	NJM2073	Information	sym_1 is being created.
amplifier	NJM2073	Information	Property name 'PART_REFERENCE' is converted to 'PART_REFERENCE'
amplifier	LT1621/SO	Information	sym_1 is being created.
amplifier	LT1621/SO	Information	Property name 'PART_REFERENCE' is converted to 'PART_REFERENCE'
amplifier	LM383/T0220	Information	sym_1 is being created.
amplifier	LM383/T0220	Information	Property name 'PART_REFERENCE' is converted to 'PART_REFERENCE'
amplifier	LM380/DIP8	Information	sym_1 is being created.
amplifier	LM380/DIP8	Information	Property name 'PART_REFERENCE' is converted to 'PART_REFERENCE'
amplifier	LM389	Information	Pin name 'GND' is converted to 'GND#5'
amplifier	LM389	Information	sym_1 is being created.
			Property name 'PART_REFERENCE' is converted to 'PART_REFERENCE'

Enabling OrCAD Library Editing in Allegro X System Capture

You can launch OrCAD Capture from System Capture to create a new OLB schematic model or edit an existing one.

OrCAD library editing is enabled with the following licenses:

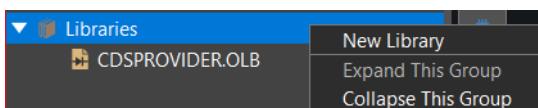
- Allegro System Capture Designer (ASC200)
- Allegro System Capture Venture (ASC300)
- Allegro Managed Library Authoring (PLA300)
- Allegro X EE (ALGX100)
- Allegro(R) PCB Librarian / Allegro Library Authoring (PX3500)

Important

OrCAD library editing requires OrCAD products to be installed and needs OrCAD licenses.

You can add a new OrCAD library that is displayed as a .olb file in *Project Explorer*, under the *Libraries* tree.

- To add a new OrCAD library, right-click the Libraries, and select *New Library*.



OrCAD Capture is launched. You can create a new library in this application.

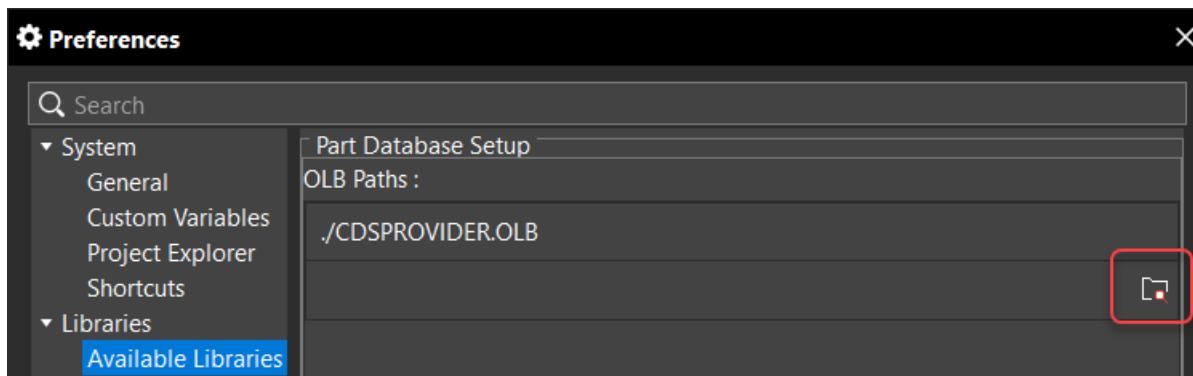
To add the new OrCAD library to System Capture, do the following:

1. Select *Edit – Preferences*.

Library Authoring and Editing in Allegro X System Capture

Working with Legacy Libraries

The *Preferences* dialog box is displayed.



2. Click the *Add Library* button and browse to the library location.
3. Click *OK*.

You can also edit an existing OrCAD library.

- To edit an existing OrCAD library, right-click the library in the *Libraries* tree, and select *Edit Library*.



OrCAD Capture is launched and you can edit the library.

After you make changes in a part from a OrCAD library, a new version of the selected part is available in the library in System Capture.

To reflect the changes in System Capture, select *Tools – Part Manager* to update the part.

Related Topics

- [Creating Library in a New Library Container](#)
- [Viewing Details of a Specific Part](#)

Changes in Imported Part Details

Because of the change in the use model of OrCAD Capture and Allegro X System Capture, all information in an .olb is not translated as is in the corresponding .ldax files. Some modifications are made to the imported data to ensure that it is usable in System Capture. The changes listed in the log file, which is displayed in the *Session Log* panel, are as follows:

- [Changes in Pin Types](#)
- [Changes in Pin Names](#)
- [Duplicate Pin Names](#)
- [Changes in Property Names](#)
- [Properties without Values](#)
- [Reserved Property Names Not Imported](#)
- [Hidden Pins](#)
- [Vector Pins](#)

Changes in Pin Types

System Capture classifies pins as *IN*, *OUT*, and *IN/OUT (IO)* while OrCAD Capture supports nine types of pins.

As the System Capture schematic only supports *IN*, *OUT*, and *IN/OUT (IO)*, when you bring an OrCAD Capture library to the Unified Library, each component pin type gets mapped to these three types as listed in the following table:

Pin Type in OrCAD Capture	Converted to
PASSIVE	<i>In/Out</i>
INPUT	<i>In</i>
OUTPUT	<i>Out</i>
BI-DIRECTIONAL	<i>In/Out</i>
OPEN COLLECTOR	<i>Out</i>
TRI-STATE	<i>In/Out</i>

Library Authoring and Editing in Allegro X System Capture

Working with Legacy Libraries

Pin Type in OrCAD Capture	Converted to
OPEN EMITTER	<i>In/Out</i>
POWER - VISIBLE	<i>In</i>
POWER - INVISIBLE	Not shown on the canvas



Important
Conversion of vector pins from OrCAD libraries to the Unified Library format is not supported.

Changes in Pin Names

When parts are imported from an OrCAD Capture library, pin names with unsupported characters in System Capture are converted as listed in the following table:

Character	Changes to
~	Tilde
'	Acute
!	Exclamation
(Open parenthesis
)	Close parenthesis
:	Colon
;	Semi-colon
"	Double Quote
'	Single Quote
<	Less than
>	Greater Than
,	Comma
	_T
	_Q
	_E
	_LA
	_RA
	_C
	_S
	_DQ
	_A
	_LA# [PIN_NUMBER]
	_RA# [PIN_NUMBER]
	_CO

Duplicate Pin Names

Power pins have the same name in OrCAD Capture, for example, GND. System Capture does not support duplicate pin names so the pin names are changed to GND##1, GND##2, and so on.



Pin name changes do not impact connectivity because it is handled by pin numbers.

Changes in Property Names

System Capture does not support spaces and lowercase characters in property names. OrCAD Capture parts containing spaces or lowercase characters in their property names are converted as follows:

In OrCAD Capture	Changes to
Space in name	Underscore
Lower-case name	Uppercase property name

Properties without Values

Part properties without any values are displayed as *<Property Name>* in OrCAD Capture if the attribute is set to *Value Only*. In System Capture, these properties are displayed as <<NULL>> because System Capture shows <<NULL>> for properties that do not have any values defined.

Reserved Property Names Not Imported

The following table lists the properties that are imported into Unified Library by the import process if they have property names that are reserved in System Capture:

Object	Ignored Properties
Parts	LOCATION REFERENCE SPLIT_INST_NAME SEC CDS_SEC CDS_LOCATION CDS_ERR_LOCATION SIZE HAS_FIXED_SIZE PART_NAME CDS_PART_NAME IREF BIASVALUE_CURRENT BIASVALUE_VOLTAGE BIASVALUE_POWER BIASVALUE HDL_POWER

Hidden Pins

In OrCAD Capture, pins can be hidden without being connected. When an OrCAD Capture library is imported into a Unified Library:

- Power pins that were hidden in the OrCAD Capture design remain hidden in System Capture and the connectivity is maintained.
- Non-power pins that are hidden in OrCAD Capture become visible in System Capture.

Library Authoring and Editing in Allegro X System Capture

Working with Legacy Libraries

Note: There is no loss of connectivity but some overlaps might be seen.

Vector Pins

Vector pins are not imported from OrCAD Libraries into Unified Libraries.

Allegro DE-HDL Libraries in System Capture

System Capture in-tool library authoring and editing functionality is accessible through a System Capture project created in the Allegro DE-HDL library format. A System Capture project created with the Allegro DE-HDL library format provides built-in support to edit DE-HDL designs and libraries.

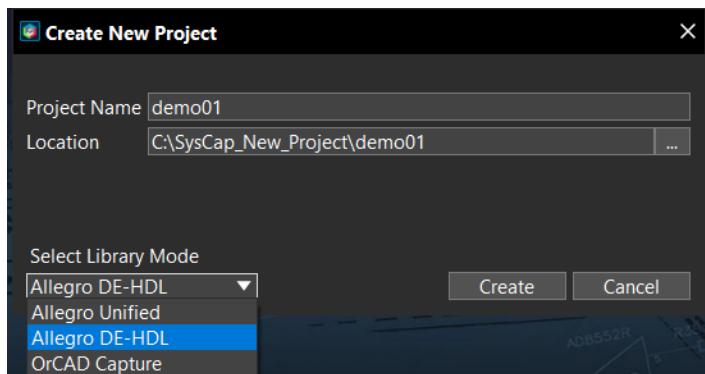
DE-HDL library editing is enabled with the following licenses:

- Allegro System Capture Designer (ASC200)
- Allegro System Capture Venture (ASC300)
- Allegro Managed Library Authoring (PLA300)
- Allegro X EE (ALGX100)
- Allegro(R) PCB Librarian / Allegro Library Authoring (PX3500)

Creating a New Library in DE-HDL

You can create a new library in the DE-HDL library mode in System Capture.

- While creating a new System Capture project, select *Allegro DE-HDL* from the *Select Library Mode* list, and click *Create*.



A new project is created.

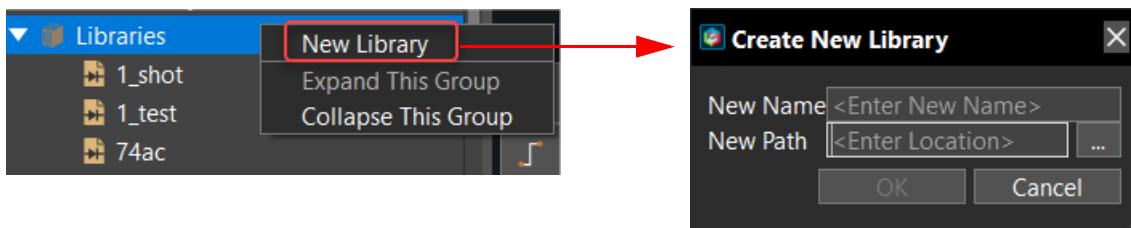
To create a new library in DE-HDL, do the following:

1. Right-click *Libraries*.
2. Select *New Library* to start creating a new library.

Library Authoring and Editing in Allegro X System Capture

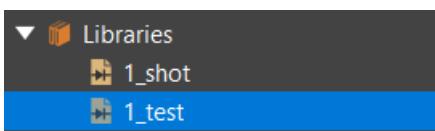
Working with Legacy Libraries

The *Create New Library* dialog box is displayed.



3. Specify the library name and navigate to where you want the library to be stored.
4. Click *OK*.

A new library is created and listed under the *Libraries* tree.



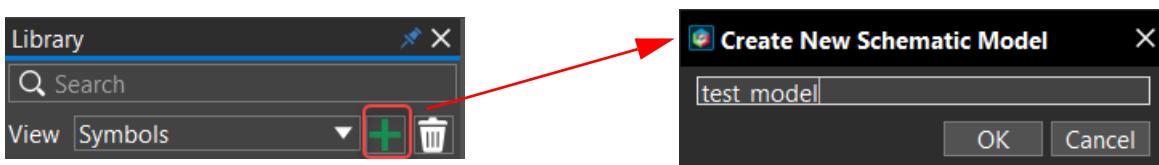
Editing DE-HDL Schematic Model within System Capture

System Capture provides built-in support to edit DE-HDL designs. You can add or edit schematic models in Allegro X System Capture when working with DE-HDL libraries.

To add a new schematic model, do the following:

1. Click the *New Symbol* button in the Library Explorer.

The *Create New Schematic Model* dialog box is displayed.

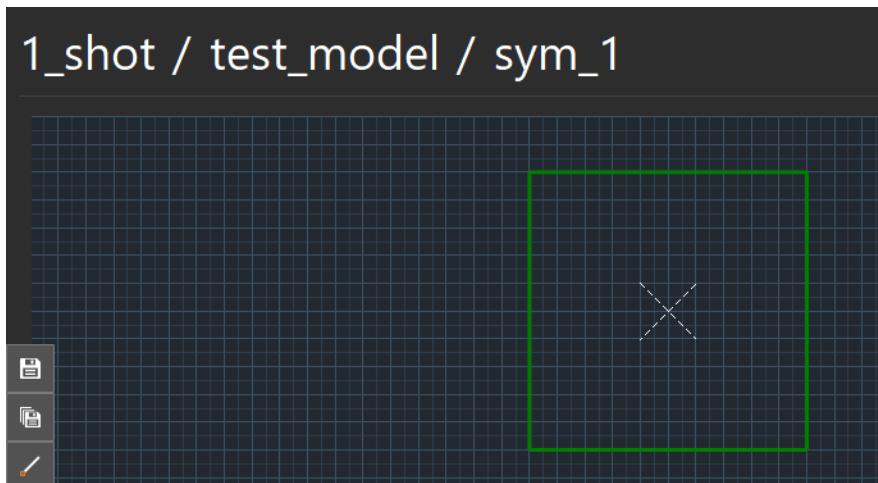


2. Specify a schematic model name.
3. Click *OK*.

Library Authoring and Editing in Allegro X System Capture

Working with Legacy Libraries

A symbol boundary is added to the canvas.



You can switch to the *Table View* where you can add pins, specify a pin name, number, type, and section.

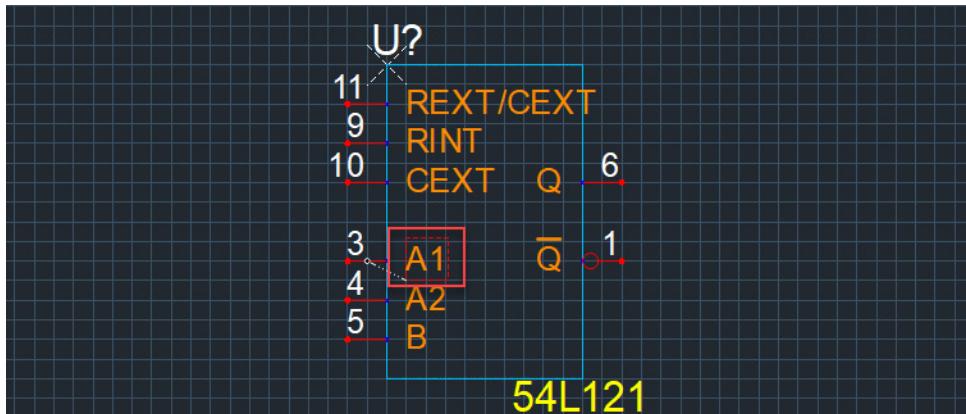
A screenshot of the Allegro X System Capture interface in "Table View" mode. The title bar and toolbar are identical to the previous screenshot. The main area now features a table on the left labeled "Filter..." with columns for Name, Type, Number, and Section. A single row is present with the values "<Enter Name...>" for Name, "Input" for Type, "?" for Number, and "1" for Section. To the right of the table is the same green symbol boundary and dashed crosshair from the first screenshot. At the bottom, there is a toolbar with various icons.

Library Authoring and Editing in Allegro X System Capture

Working with Legacy Libraries

You can edit a library cell in the table view or directly on the canvas.

Editing Cells on Canvas



Editing Cells in Table View

Name	Type	Number	Section
A1	Input	3	1
A2	Input	4	1
B	Input	5	1
CEXT	Input	10	1
Q	Output	6	1
Q\	Output	1	1
REXT/CEXT	Input	11	1
RINT	Input	9	1
GND	Power-Implicit	7	
VCC	Power-Implicit	14	
<Enter Name>	Input	?	1

To view a list of cells in a library, double-click the library.

Based on the file system permissions, you can create, read, update, or delete the cells.



Editing is not supported for sizable parts, multi-primitive parts, tech-independent parts, and library-level blocks. Zero length pins are also not supported. To edit these, select *Tools – Part Developer*.

Library Authoring and Editing in Allegro X System Capture

Working with Legacy Libraries



Operations, such as Rename, Copy-Paste, Save As are not supported.