cadence®

OrCAD X Capture Part Authoring Tutorial

Product Version 23.1
September 2023

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

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

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

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

All other trademarks are the property of their respective holders.

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

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

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

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

Contents

1 5
Module1: Introduction to Part Authoring in OrCAD X Capture 5
Tutorial Overview 6
Tutorial Setup 6
Audience 6
Tutorial Files 6
Using the Tutorial 7
Related Documentation 7
Exploring Component Explorer 8
Launching Capture 8
Creating a New Project 11
Launching Component Explorer 12
2 16
Module 2: Authoring Component Categories as a Librarian 16
Creating Electrical Part Category 16
Adding an Electrical Category 16
Adding Properties 18
Associating Models 20
Associating Symbol 20
Associating Footprint Information 23
Associating PSpice Models 24
Creating a Subcategory 28
Creating Mechanical Part Category 29
3 31
Module 3: Creating New Components in Component Explorer 31
Creating Electrical Parts Based on a Category 31
Creating Mechanical Part Based on a Category 35
Creating Mechanical Assembly 39
4 43
Module 4: Creating New Components from Content Provider Databases 43
Adding Components to Workspace 43

Associating Multiple PSpice Models to a Component 45
Associating Mechanical Models with a Component 48
Associating Manufacturer Part Number from Content Providers 50
5 54
Module 5: Sharing Components with Team Members 54
Publishing Components to Shared Workspace 54
Publishing a Component 57
Editing and Copying Components from Shared Workspace 60
Modifying a Component and Publishing it back to Shared Workspace 63
Restoring Older Versions of Components in Shared Workspaces 67

1

Module1: Introduction to Part Authoring in OrCAD X Capture

This tutorial aims to provide a functional overview of the new OrCAD® X Capture part authoring capability. The part authoring capability can be almost entirely accessed from the Component Explorer interface, which provides a unified environment to:

- Create new a component from scratch
- Add and edit all the relevant details about the component
- Search and add components directly available from different content providers
- Link component details with up-to-date information with live connection with content providers
- Access all the components from this single interface – online and offline
- Access data from anywhere with Cloud storage
- Create separate workspace for sharing components and add team members:
    - To access a central repository of components specific to the project
    - To publish components on the go after review
    - To directly place components specific to the design

Tutorial Overview

This tutorial demonstrates the ease with which you can create electrical and mechanical parts in the OrCAD X Capture solution. You can run through the detailed steps in the tutorial to perform the basic tasks for part authoring. This document does not cover all the features of the tool. It only highlights the tasks that you need to perform in Capture to author parts from existing symbols and footprint information and from external search providers, such as Samacsys, SnapEDA, and Ultra Librarian.

Tutorial Setup

The tutorial is designed for OrCAD X Capture release 23.1. You need the following setup to work with this tutorial:

- OrCAD X Capture installed in the system
- `OrCAD X Professional (POX200 Pro)` license
- An email ID registered with Cadence
- An active subscription to OrCAD X Cloud - Authorization of the email ID to use OrCAD X

> ⚠ Subscription is bundled with the `POX200 Pro` license.

Audience

This tutorial is useful for:

- Designers who want to use OrCAD X Capture to create parts to be consumed in the schematic creation process in a collaborative environment.
- First-time users of OrCAD X Capture.

Tutorial Files

To work with the tutorial:

- Copy *`breakout.lib`* from *`<installation_folder_for_23.1>`*`\tools\pspice\Library` to the following location:

```
%HOME%\cdssetup\workspace\libraries\pspicemodels
```

Using the Tutorial

To run through the complete tutorial, you need OrCAD X Capture available with the `OrCAD X Professional (POX200 Pro)` license.

For a better understanding of the tasks covered in the tutorial, it is strongly recommended to go through the links under Related Documentation.

Related Documentation

- Component Explorer
- Creating Components
- Sharing Components
- Managing Libraries on OrCAD X Cloud
- Managing a Local Component Library
- Configuring Workspaces
- Sharing Workspaces

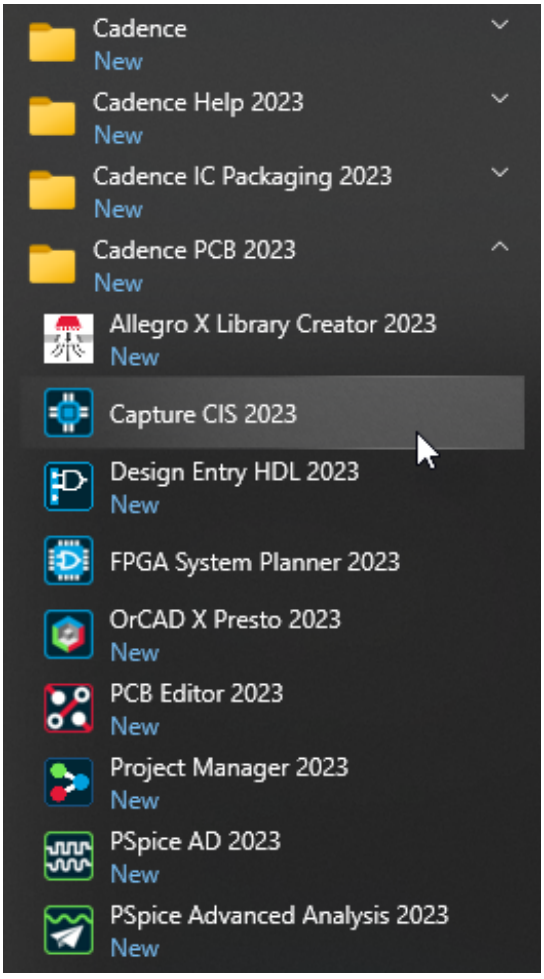
Exploring Component Explorer

With the `OrCAD X Professional (POX200 Pro)` and `OrCAD X Standard (POX100 Standard)` licenses, you can create local libraries of parts sourced from various content providers. Component Explorer is a component management system that provides an intuitive user interface to access components from various sources including Cadence-supplied libraries and content providers.

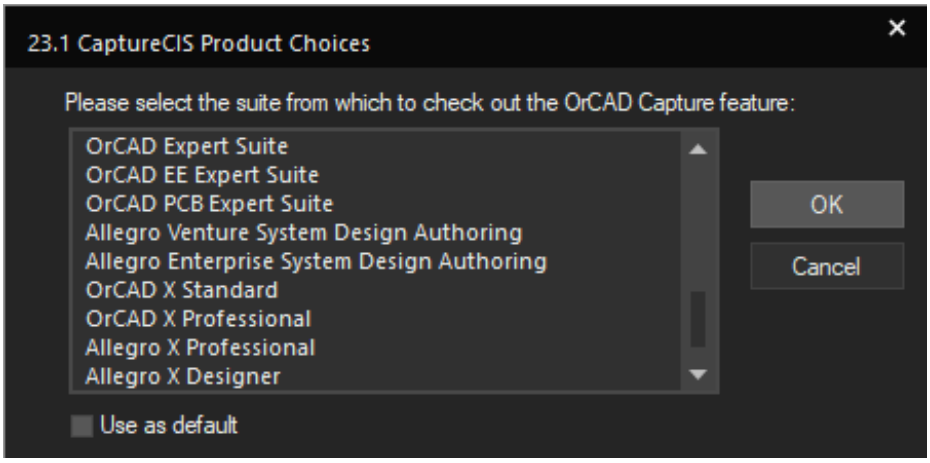
Launching Capture

To start Capture from the Windows Start menu, do the following:

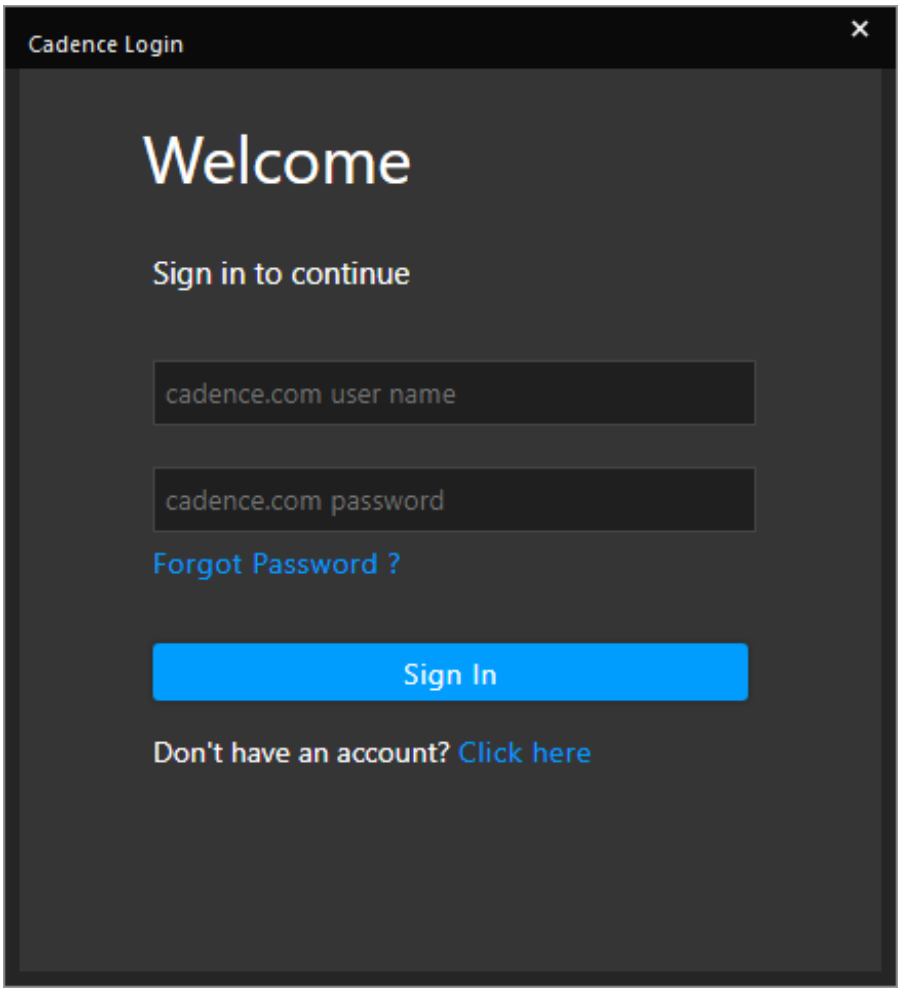
1. Choose *Start – All Programs – Cadence PCB 2023 – Capture CIS 2023.*



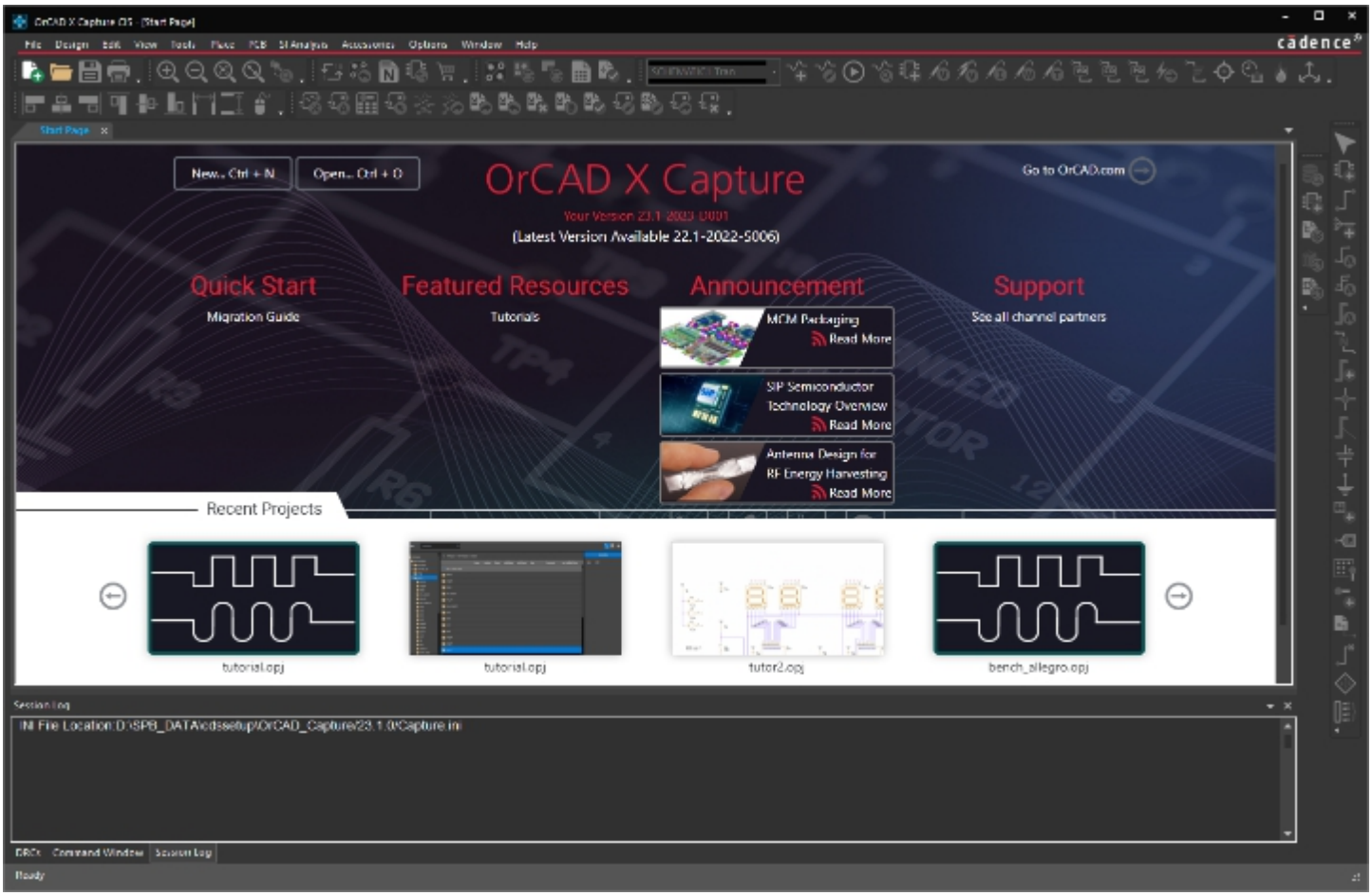
The *23.1 CaptureCIS Product Choices* dialog box is displayed.



2. Choose *OrCAD X Professional* and click *OK*.
   When you launch OrCAD X Capture CIS for the first with this license, you are prompted to specify your Cadence login credentials.



3. Specify the user name and password provided by Cadence and click *Sign In*.

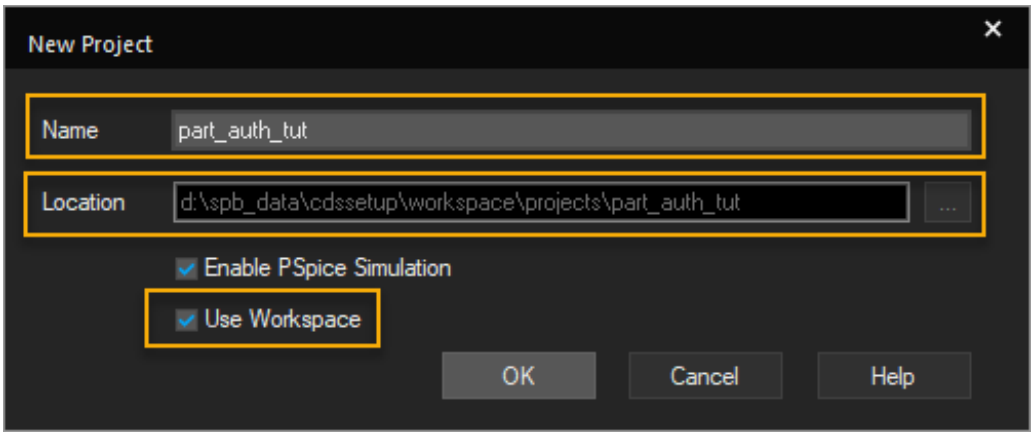


Creating a New Project

To create a project, do the following:

1. Choose *File – New*.
2. In the *Name* text box, type `part_auth_tut` as the name of the new project.

   The *Location* field is pre-seeded with the workspace path (the location where this project will be stored) as the *Use Workspace* check box is selected by default.
3. Select the *Enable PSpice Simulation* check box if you intend to include simulation capabilities in your PCB design.



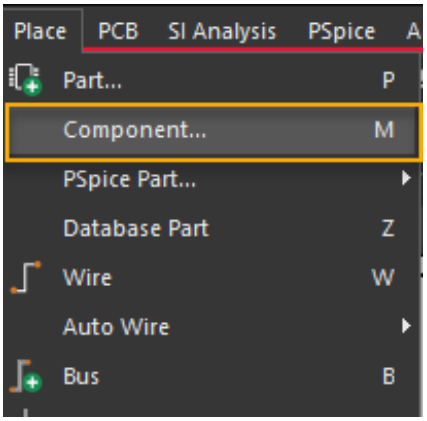
4. Click *OK*.
   A new project is created at the following location and a blank schematic page is displayed:

   ```
   %HOME%\cdssetup\workspace\projects\part_auth_tut
   ```

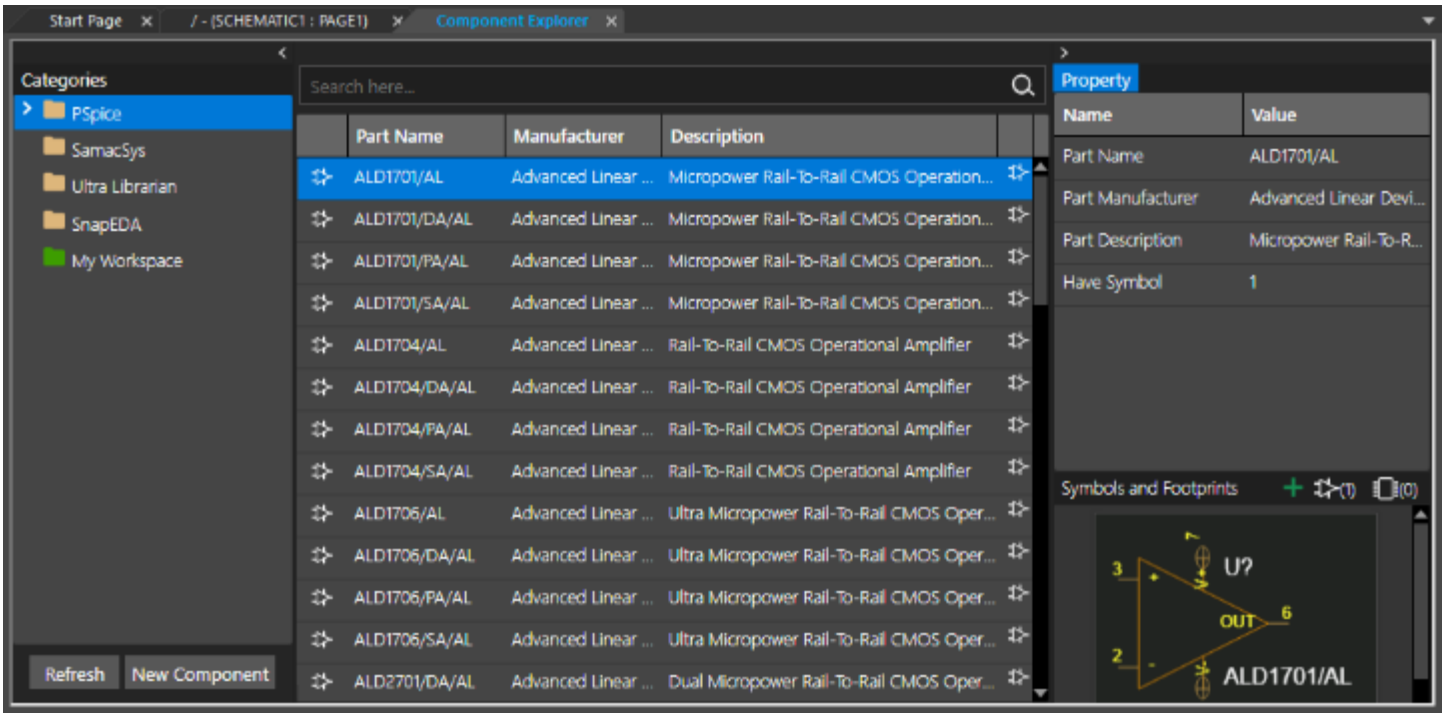
Launching Component Explorer

You can search for components from various sources and place them on the schematic canvas from the Component Explorer interface. You can search and place components from the Cadence-supplied PSpice library, Cadence-supported content providers— SamacSys, Ultra Librarian, SnapEDA, or a workspace with previously-added parts from these sources.

- To launch Component Explorer from the main menu of Capture, choose *Place – Component*. Alternatively, press `M` to display Component Explorer.



The Component Explorer interface opens in a new tab.



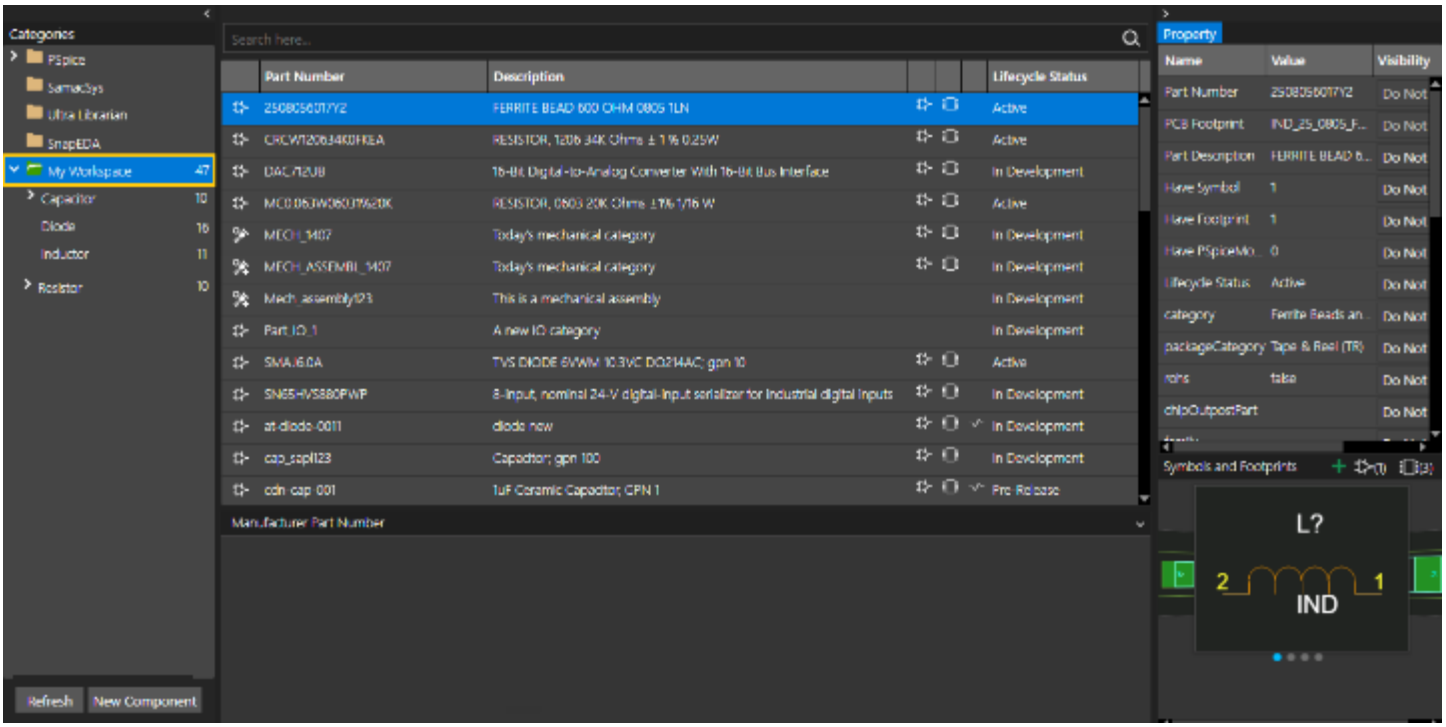
The Component Explorer interface is divided into three distinguishable panes:

Pane	Description
Left Pane	The left pane or the Categories browser displays a list of sources from where you can place components in the design. The Categories tree includes the following nodes: • *PSpice:* Includes thousands of components shipped with the OrCAD X installation. • *SamacSys, Ultra Librarian, and SnapEDA*: Cadence-supported content search providers with a database of components available from various component manufacturers. • *My Workspace:* Includes the new components you create from scratch or add from the search providers. For any component, you can create and assign categories or sub-categories, attach default symbols and footprints, and add Manufacturer Part Number (MPN) details. You can share your local workspace components with others using shared workspaces.

Middle pane	The middle pane or the part browser displays the components from the node selected in the left pane. This pane also includes a *Search* box at the top where you can search for a component within a selected category or subcategory by specifying keywords associated with the component. You can also use search queries to perform a more specific search on the selected source in the left pane. Additionally, for the components in My Workspace, a *Manufacturer Part Number* table shows MPN details in the bottom of the middle pane, if they are linked to the component.
Right pane	The right pane includes the *Property* browser that displays the properties of the component selected in the part browser. The *Symbols and Footprints* section in this pane displays the symbol and footprint information of the selected component in a carousel view. If a part has multiple symbols, the symbol that appears in the Symbols and Footprints view at the time of selection, is placed.

- Select the *My Workspace* node to view all the parts located in your local workspace

  All the components from the Cloud workspace are displayed in the part browser (middle pane).



2

Module 2: Authoring Component Categories as a Librarian

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

In this module, you will do the following tasks:

- Creating Electrical Part Category
- Creating Mechanical Part Category

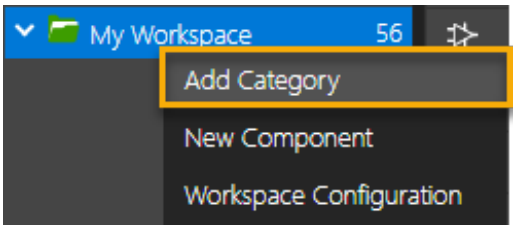
Creating Electrical Part Category

Creating an electrical part category involves adding a new category, associating properties and associating symbols, PSpice models, footprints, and properties with the category.

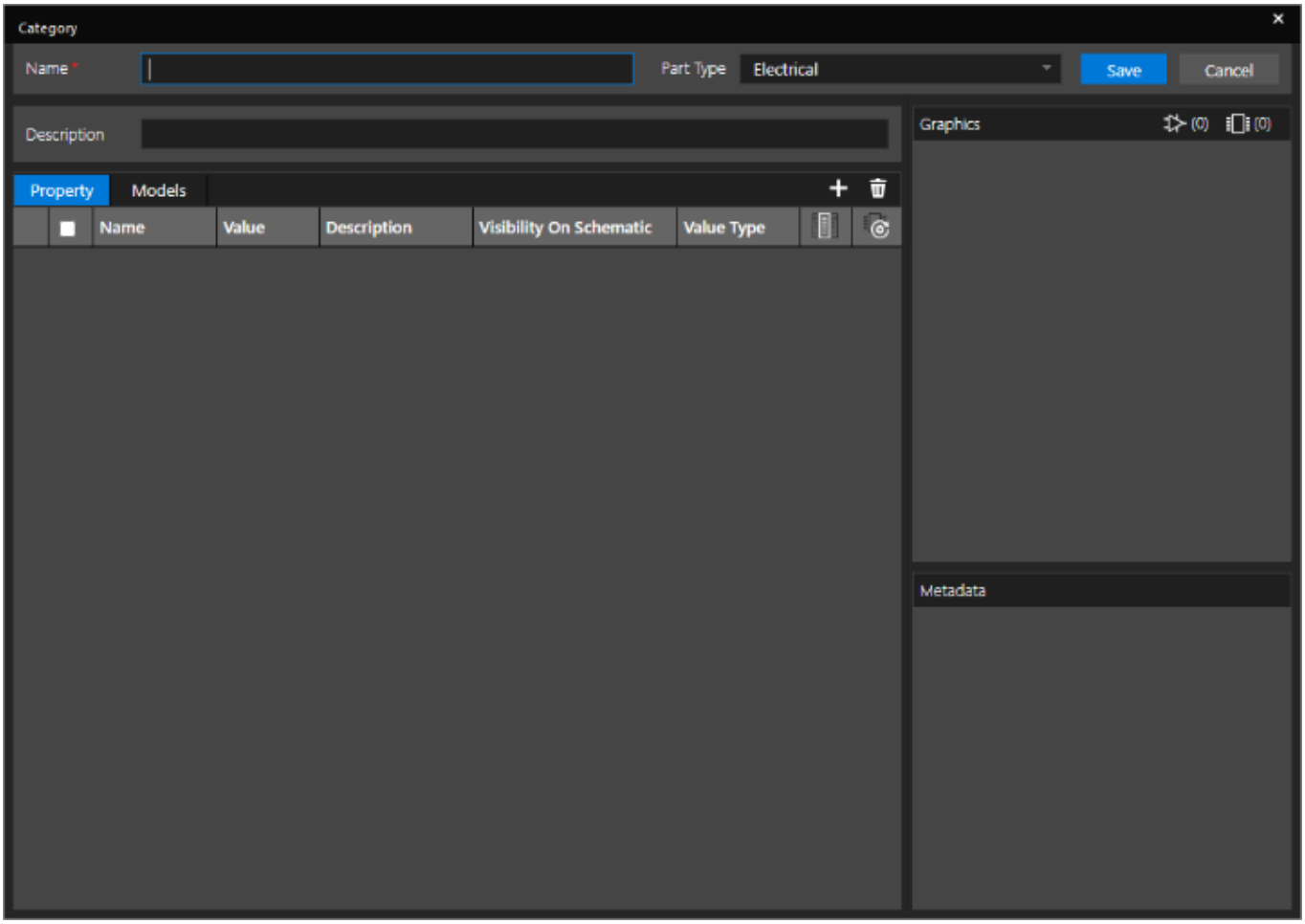
Adding an Electrical Category

To create an electrical part category, do the following:

1. In Component Explorer, right-click *My Workspace* and choose *Add Category*.



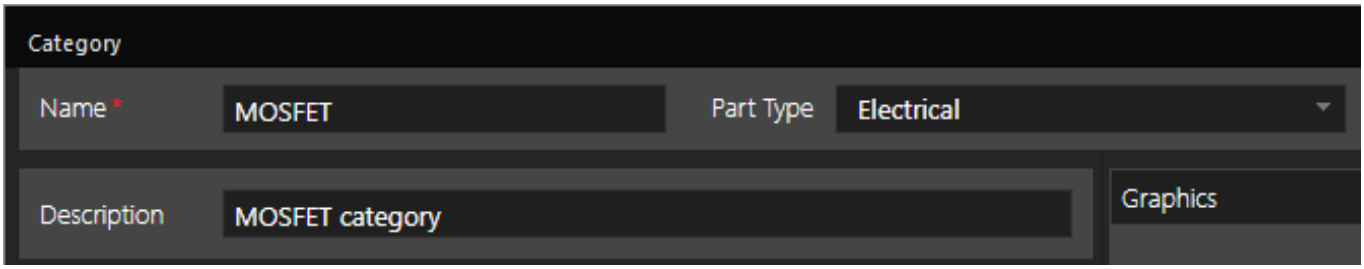
The *Category* dialog box opens. This is where you create a new category.



2. Add fields and values as shown in the following: table.

Field	Value	Description
Name	MOSFET	Name of the category
Part Type	Electrical	A drop-down list to select the type of category

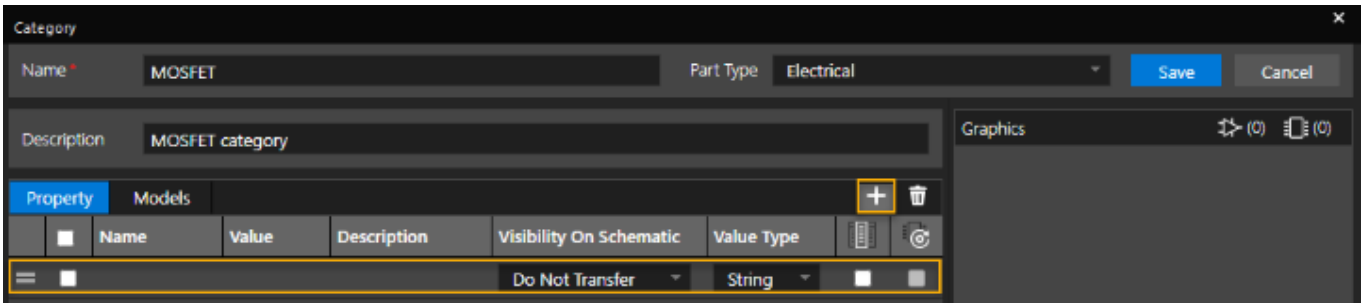
Description	`MOSFET category`	Category description



Adding Properties

To add properties to the part category, do the following:

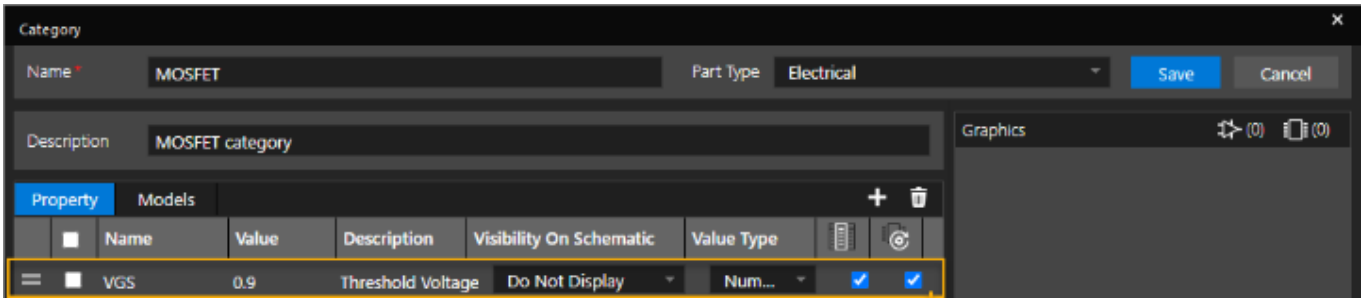
1. Select the + button in the *Property* tab to add a property row:



2. Specify the following values in the respective fields of the property row you added:

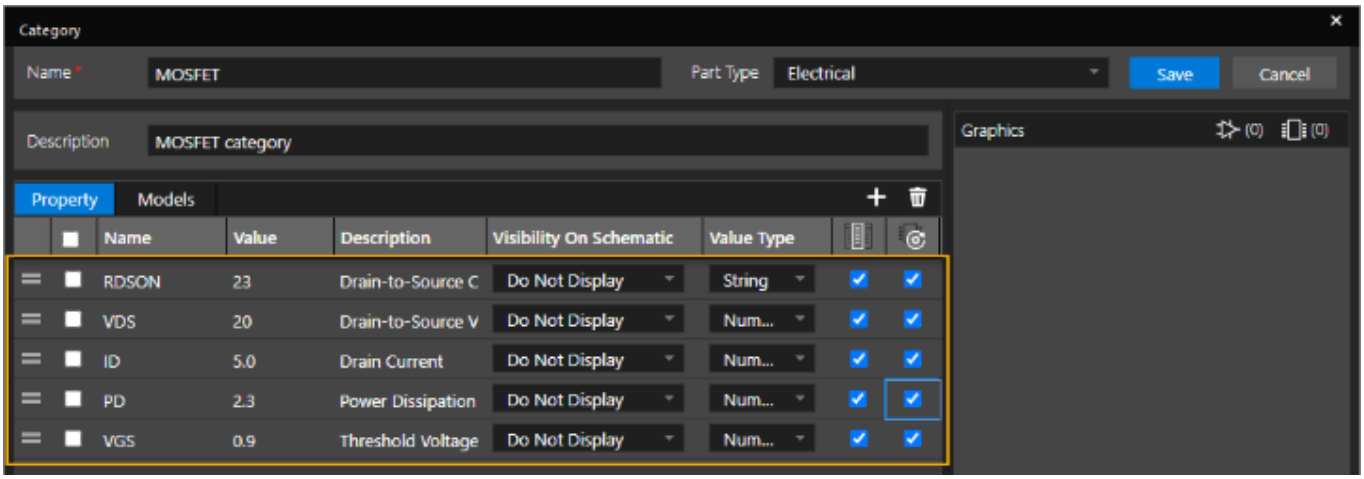
Field	Value
Name	`VGS`
Value	`0.9`
Description	`Threshold Voltage`
Visibility on Schematic	Do Not Display
Value Type	Numeric

Show in Place Component (Checkbox)	Selected
Update Part Property (Checkbox)	Selected



3. Select the + button again to add four more property rows and populate the fields for each property row with the following values:

Name	Value	Description	Visibility On Schematic	Value Type	Show in Place Component (Checkbox)	Update Part Property (Checkbox)
RDSON	23	Drain-to-Source On Resistance	Do Not Display	String	Selected	Selected
VDS	20	Drain-to-Source Voltage	Do Not Display	Numeric	Selected	Selected
ID	5.0	Drain Current	Do Not Display	Numeric	Selected	Selected
PD	2.3	Power Dissipation	Do Not Display	Numeric	Selected	Selected



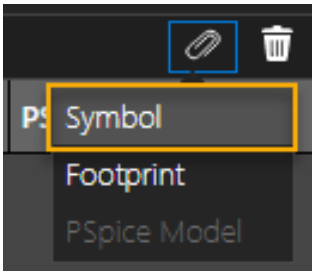
Associating Models

You associate symbols, PSpice models, and primary and alternative footprints with a category in the *Models* tab.

Associating Symbol

To associate symbols with the part category, do the following:

1. Select the *Models* tab.
2. Select the *Attach* (clip) icon.
3. Choose *Symbol* from the drop-down menu.

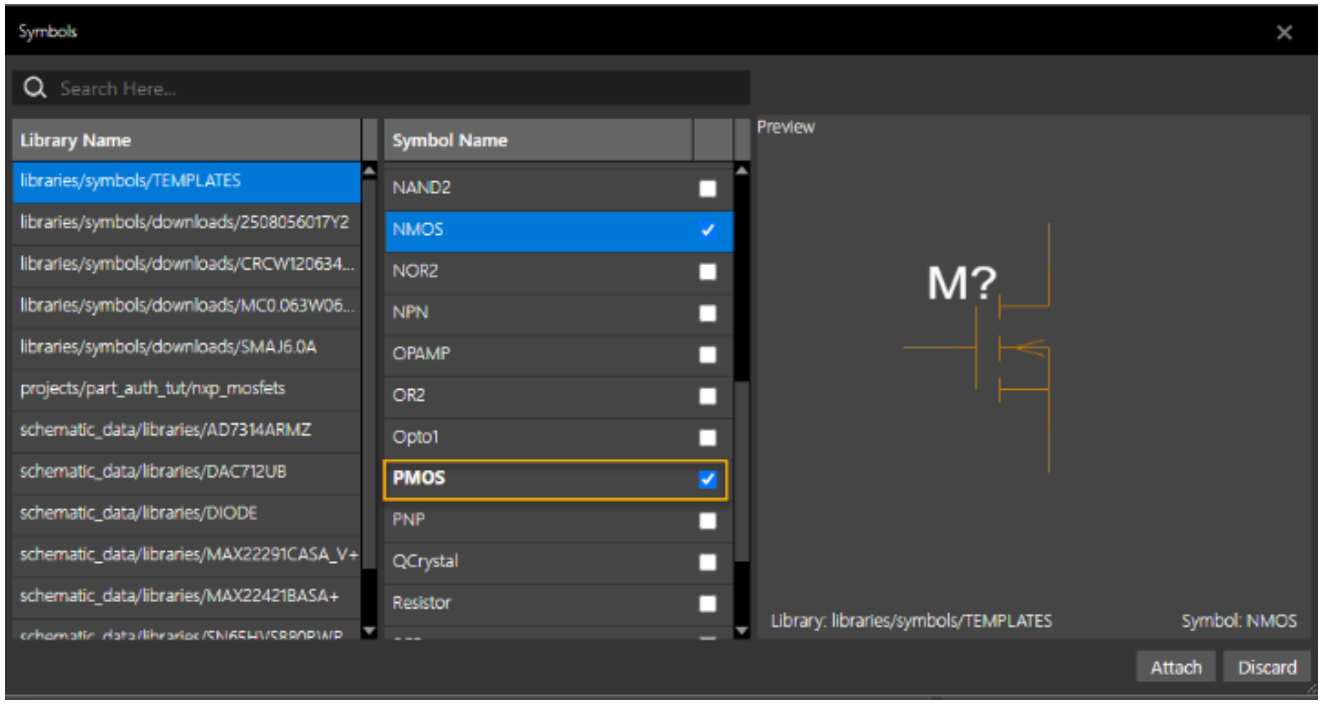


   The *Symbol* dialog box is displayed.

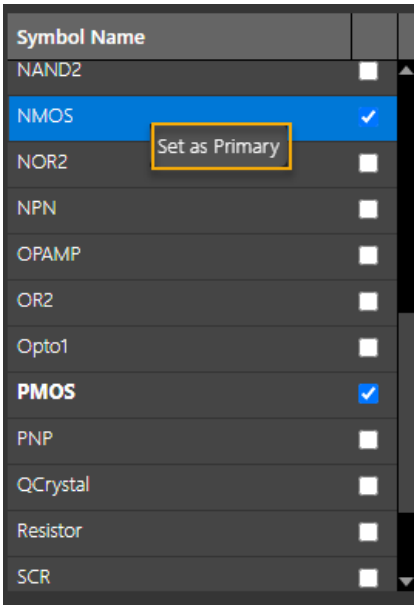
4. In the Symbol dialog box, select the TEMPLATES library from the Library Name list and select the checkboxes to the right of the *PMOS* and *NMOS* symbols.

   The first symbol you select is the primary symbol and is indicated by bold as shown in the

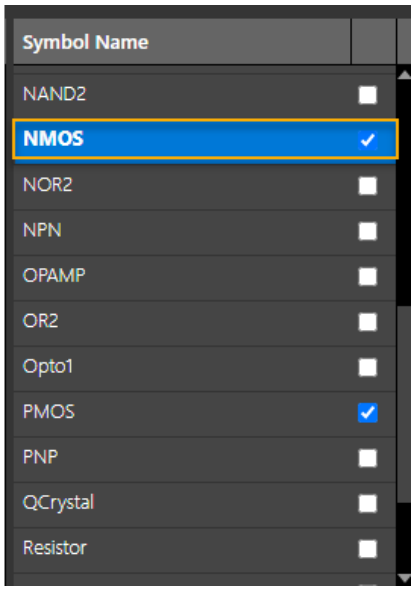
following image:



5. Right-click *NMOS* and choose *Set as Primary* to make NMOS the primary symbol.

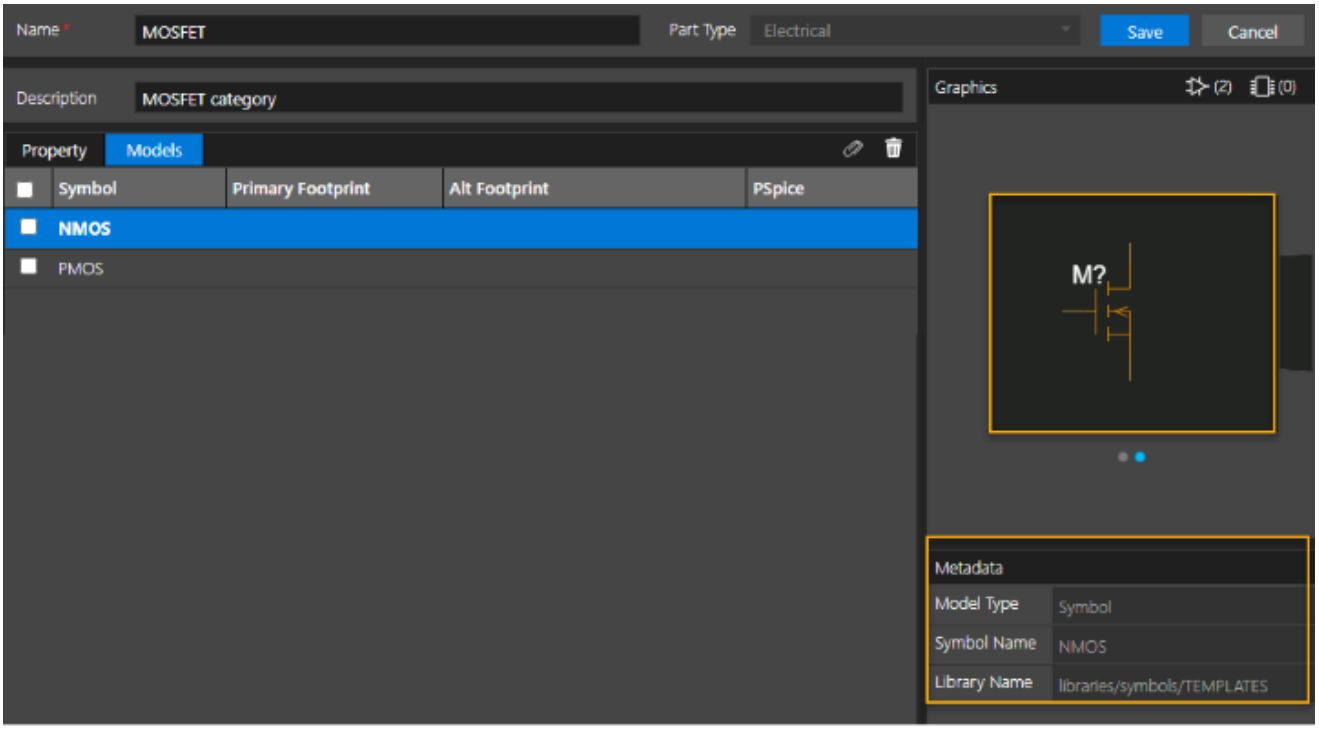


NMOS is the primary symbol



6. Click *Attach*.

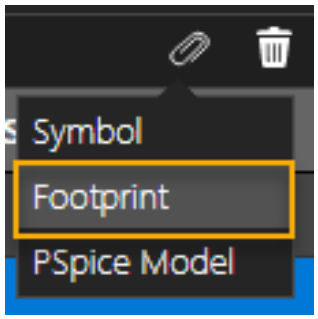
   The selected symbols are associated with the category. The Graphics section provides a preview of the selected symbol and its metadata is also displayed.



Associating Footprint Information

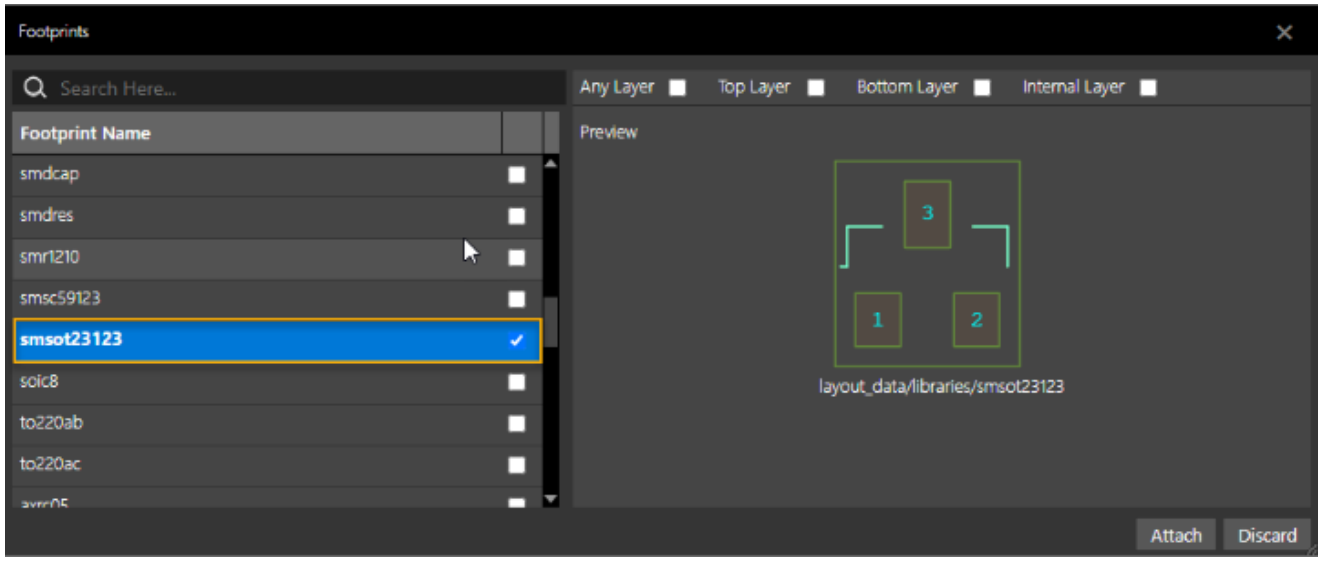
To associate a footprint and alternative footprints with the part category, do the following:

1. Select the *Attach* (clip) icon.
2. Choose *Footprint* from the drop-down menu. Alternatively, double-click the *Primary Footprint* or the *Alt Footprint* cell.

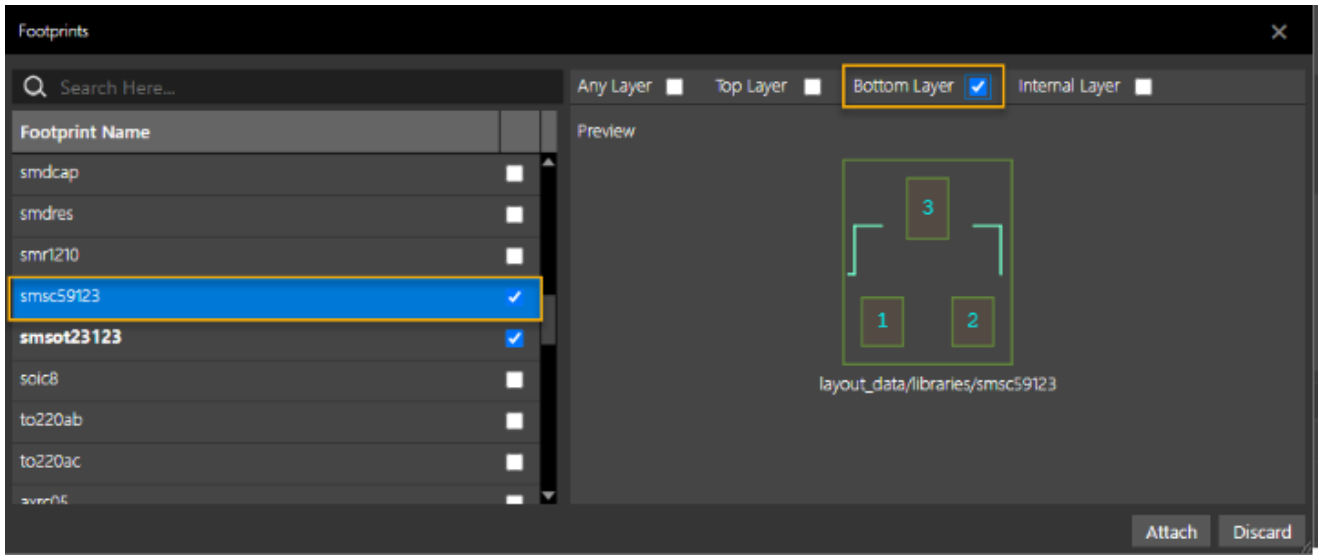


The *Footprints* dialog box is displayed.

3. Search for *smsot23123* and select the checkbox next to it.
The first footprint selected appears in bold identifying it as the primary footprint.

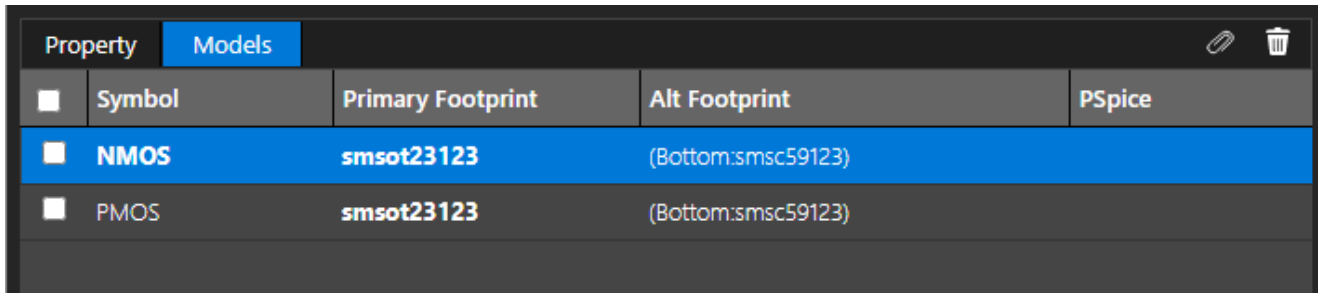


4. Now select the following checkboxes:
   – *smsc59213*
   – *Bottom Layer*



5. Click the *Attach* button.

   All the footprints selected after the primary footprint are assigned to the `ALT_SYMBOL` property of the schematic instance. In the *Component* dialog box, these appear under the *Alt Footprint* field as illustrated in the following image:



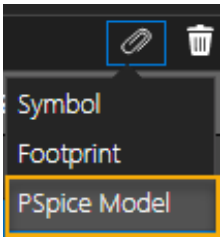
   The footprints are assigned to all the symbols in the *Models* tab.

Associating PSpice Models

You will now associate PSpice models to the symbols.

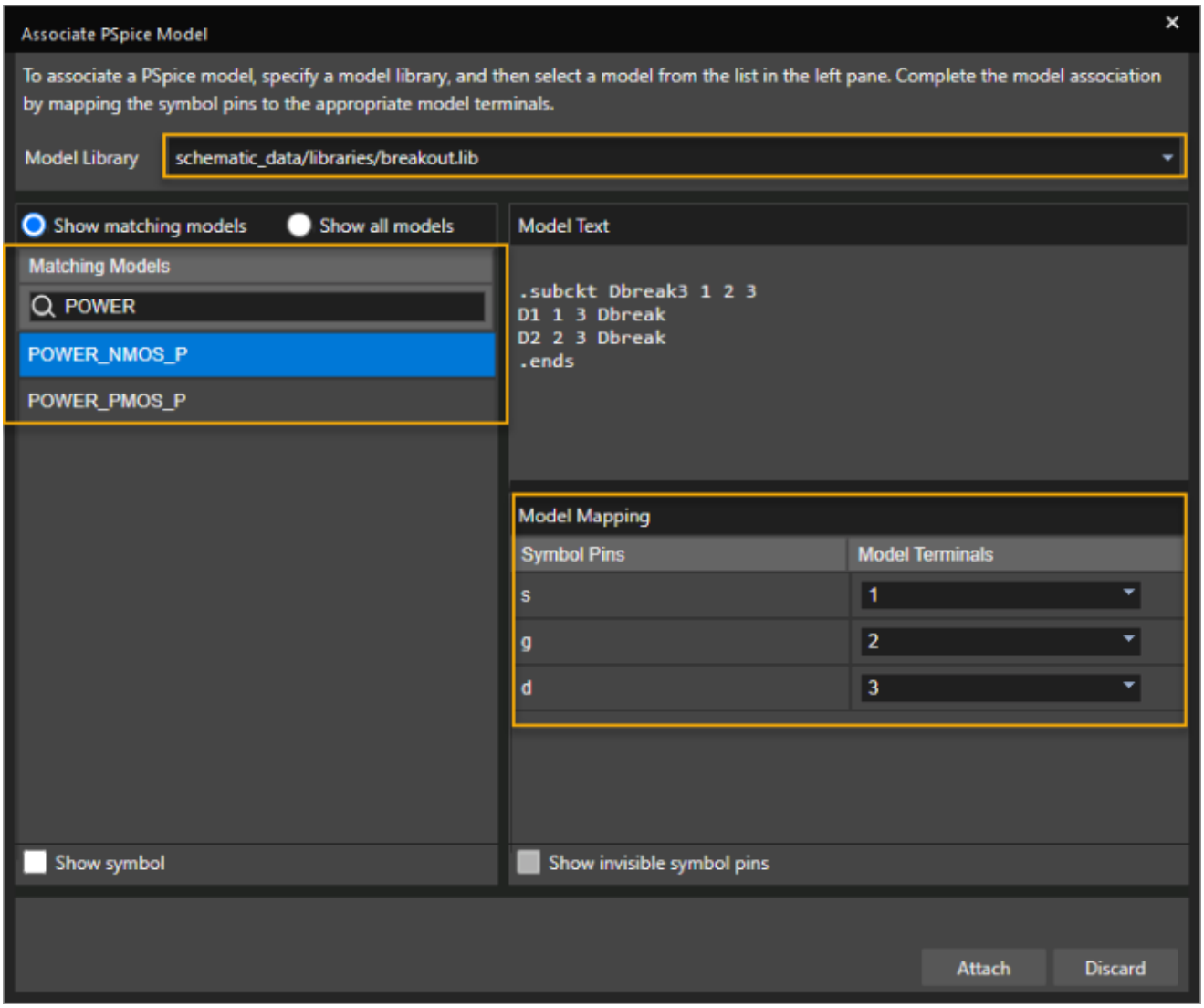
1. Select the checkbox to the left of the *NMOS* symbol.
2. Select the *Attach* (clip) icon.
3. Choose *PSpice Model* from the drop-down menu to launch the Associate PSpice. Alternatively, click the individual cell entry under the *PSpice* column to launch the

A*ssociate PSpice* dialog box.

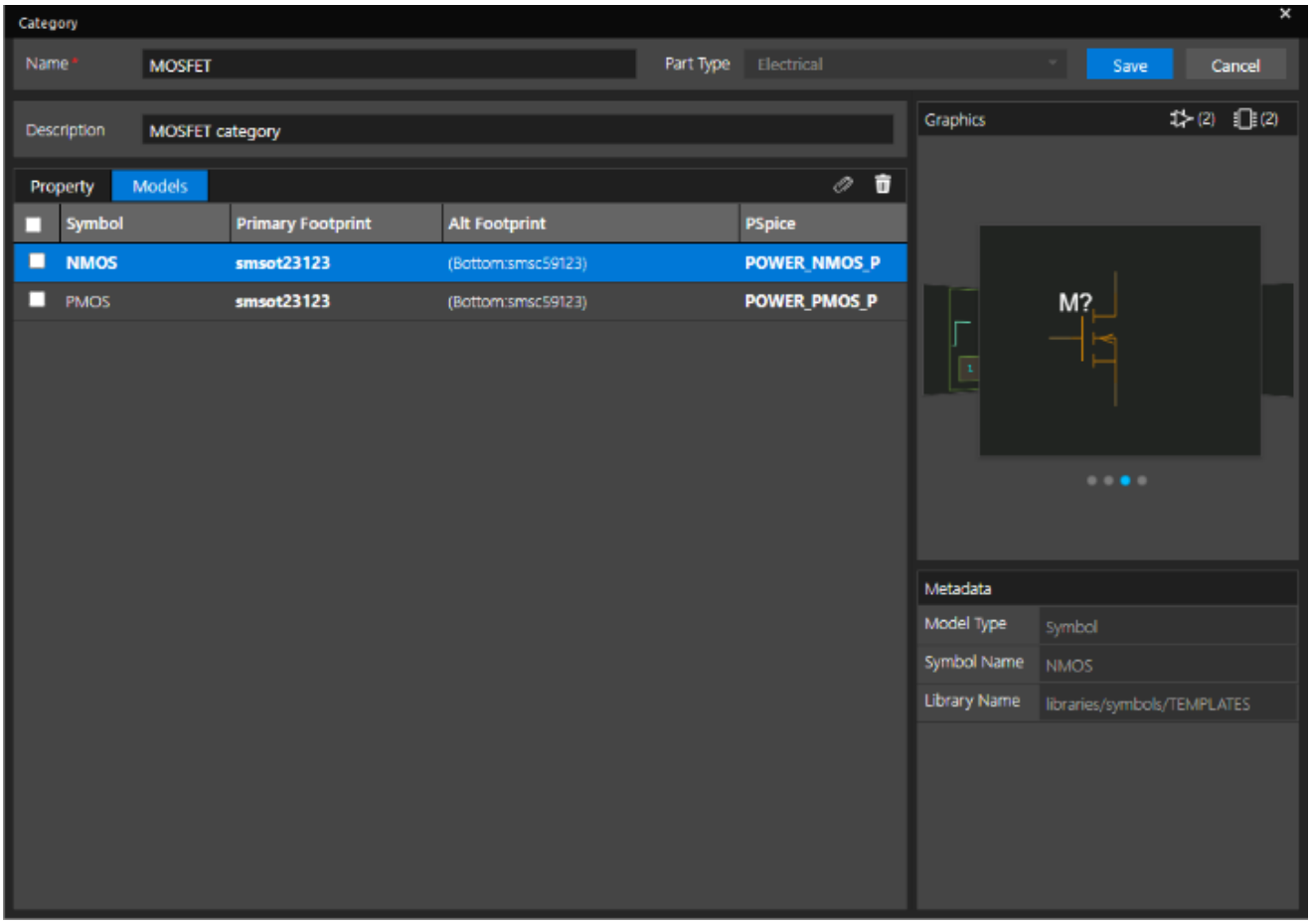


The *Associate PSpice* dialog box is displayed.

4. In the *Associate PSpice* dialog box, do the following:
   – In the Model Library list, select `breakout.lib`
   – In the Search bar, type *POWER.*
5. From the filtered list select the model, *POWER_NMOS_P.*
6. If the model terminals match the symbol pins, they are automatically mapped. Else, map the pins manually as shown in the following image:

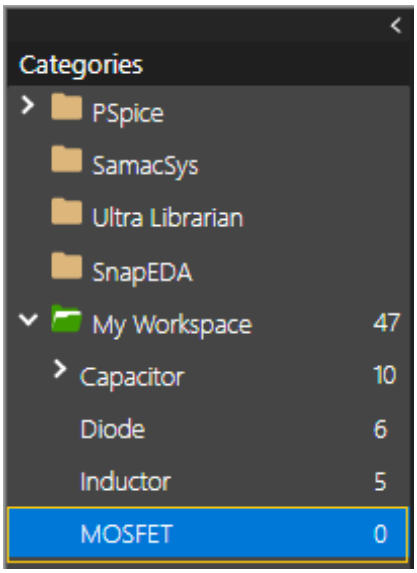


7. Repeat steps 1 to 6 and attach the *POWER_PMOS_P* model to the *PMOS* symbol.



8. Click *Save* to save the category.

   The category is created and appears under the *My Workspace* node in the Categories pane:

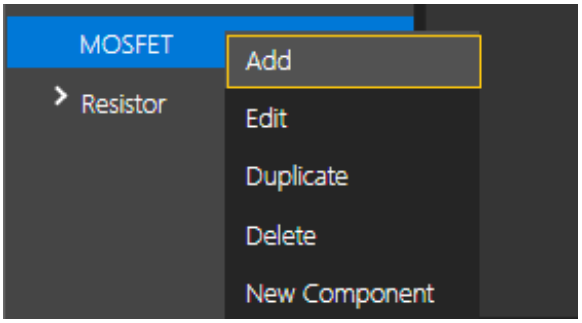


Creating a Subcategory

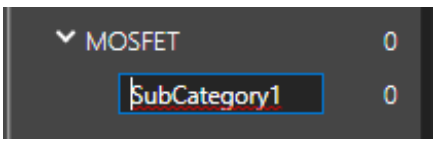
After creating the category, you will now create a subcategory under it.

To create a subcategory, do the following steps:

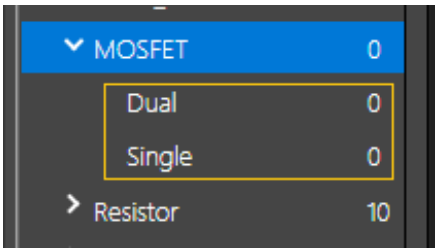
1. Right-click the *MOSFET* category under the *My Workspace* node and choose *Add.*



2. A subcategory is added under *MOSFET*.



3. *S*pecify *Single* as a name for the subcategory, overwriting the default name, *SubCategory1*.
4. Add another subcategory and name it *Dual*.

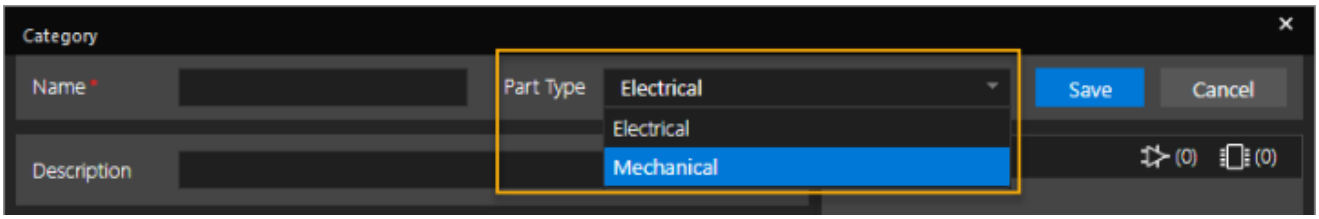


Creating Mechanical Part Category

In addition to electrical components, some non-electrical components on a PCB are used for mechanical support, such as screws and connectors. You can create categories or templates for mechanical assemblies and parts and then associate them with components.

To create a mechanical category, do the following:

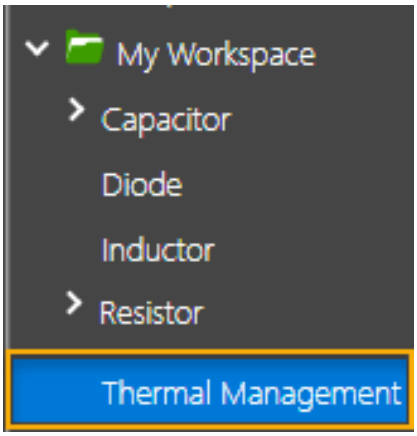
1. In Component Explorer, right-click the *My Workspace* node and choose *Add*.
2. In the *Part Type* field, select *Mechanical*.



3. Specify the name as `Thermal Management`.
4. Specify the description as `Thermal Management Category`.

5. Click *Save*.

   The category is added under *My Workspace*.


My Workspace
Capacitor
Diode
Inductor
Resistor
Thermal Management

3

Module 3: Creating New Components in Component Explorer

Use the Component Explorer interface to create new electrical and mechanical parts, and mechanical assemblies.

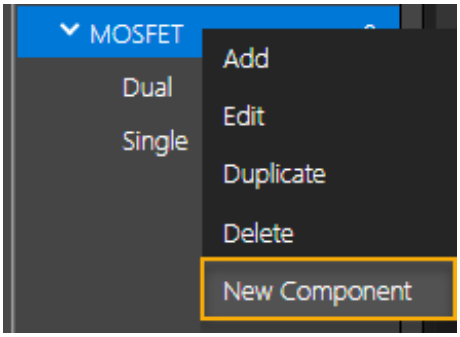
In this module, you will complete the following tasks:

- Creating Electrical Parts Based on a Category
- Creating Mechanical Part Based on a Category
- Creating Mechanical Assembly

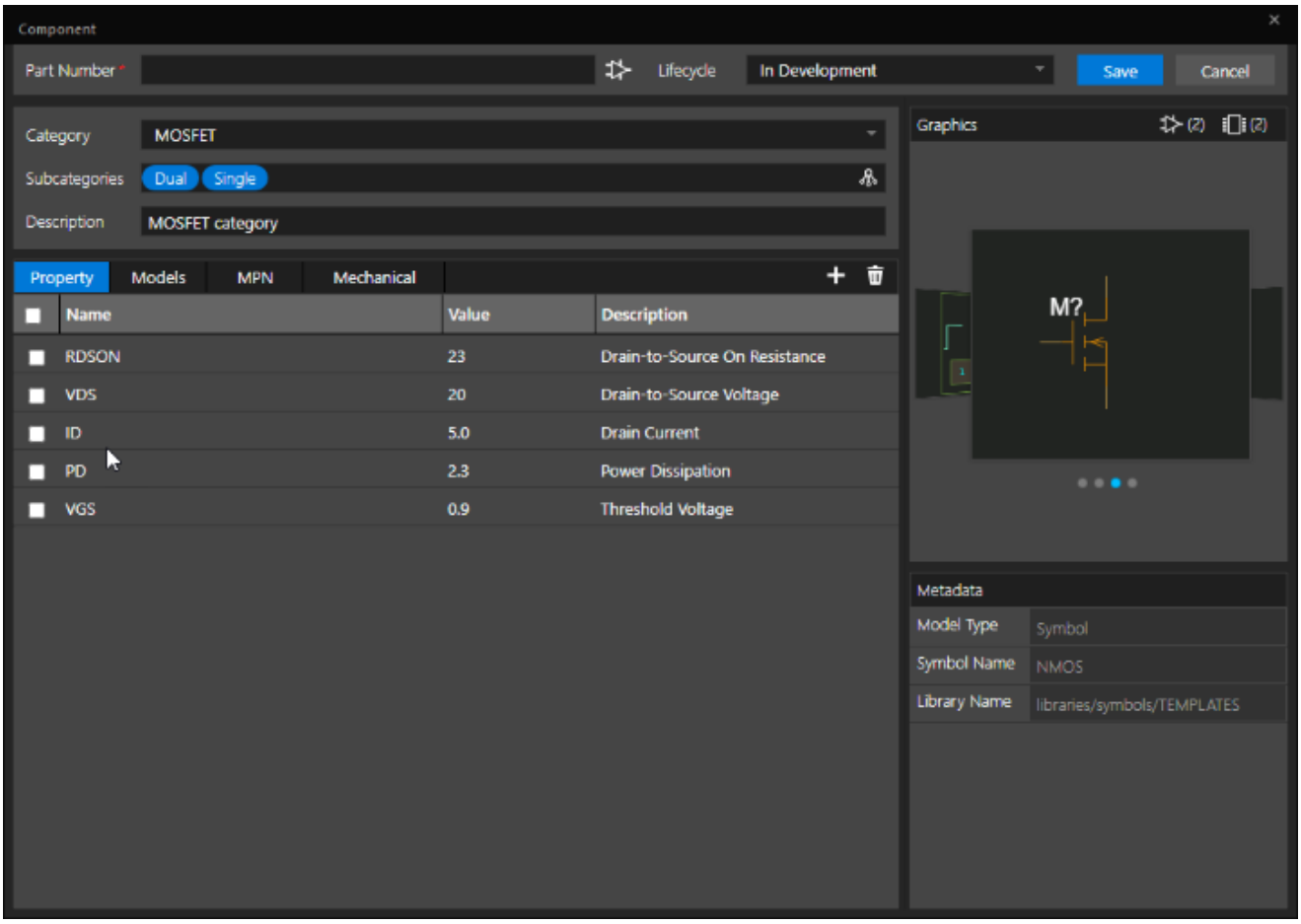
Creating Electrical Parts Based on a Category

To create an electrical part, do the following:

1. In Component Explorer, right-click the *MOSFET* category and choose *Add*.



The *Component* dialog box opens with pre-filled values from the MOSFET category.



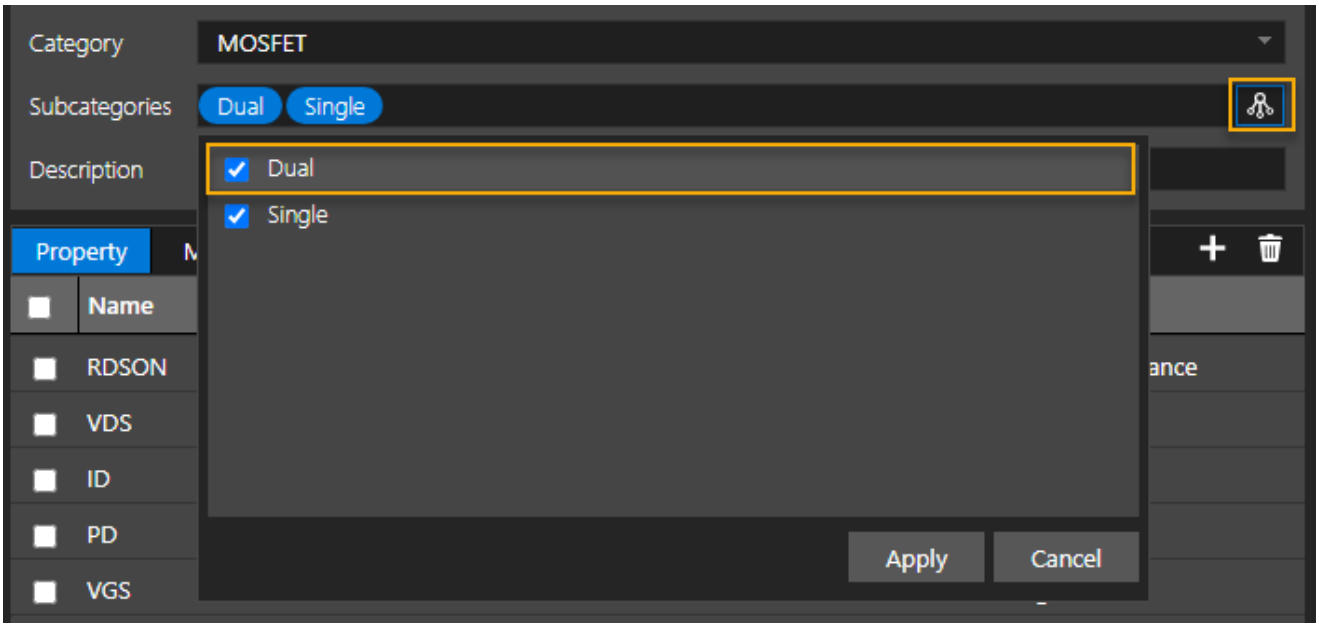
2. Specify the following details:

   Part Number: cdn-nmos-001

   Lifecycle: Active

   Description: 20-V, N channel single MOSFET, 19.2 mOhm

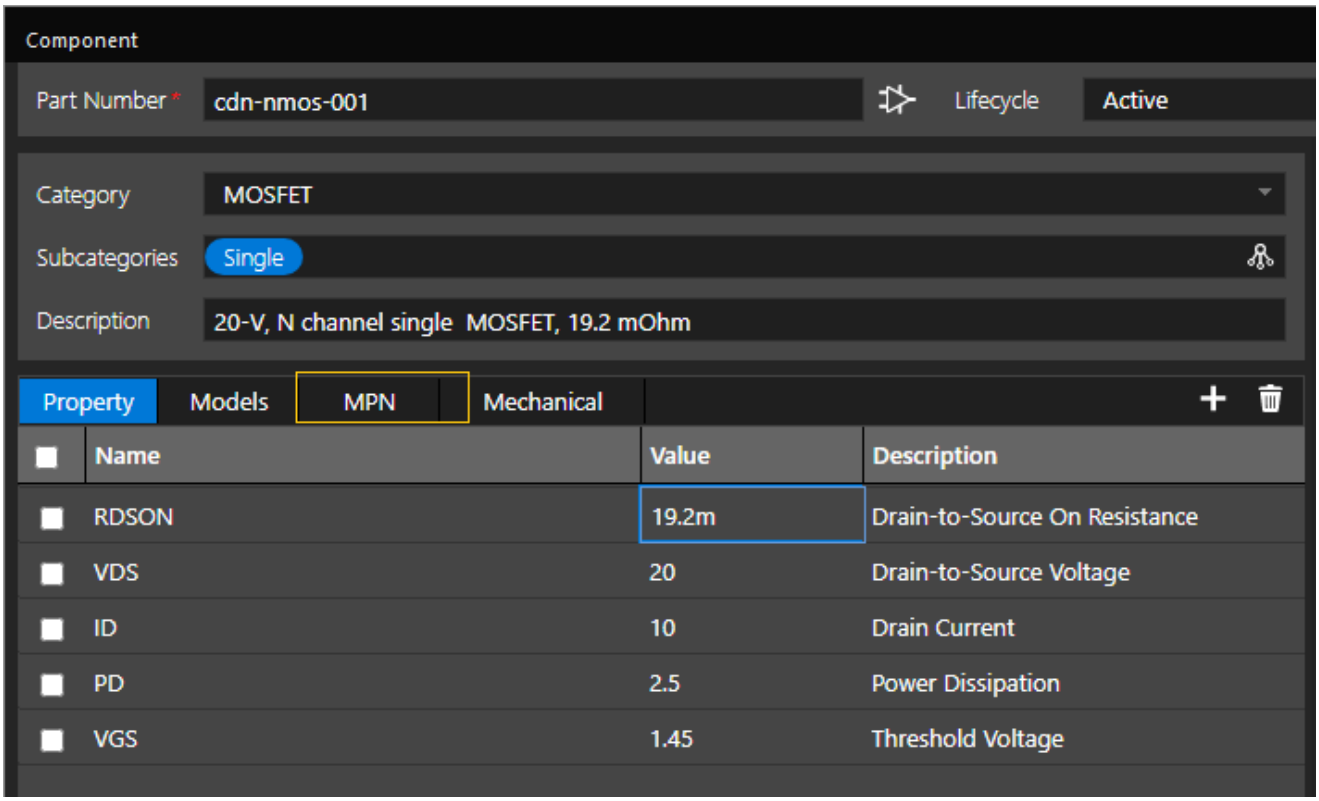
3. In the *Subcategories* field, click the *Category* icon and deselect the checkbox for the *Dual* subcategory and click *Apply*.



4. Update the property values for the component as given in the following table:

Property Name	Value
RDSON	19.2m
VDS	20
ID	10
PD	2.5
VGS	1.45

5. Select the *MPN* tab.



6. Click the *Add* icon and specify the following values in the MPN row.

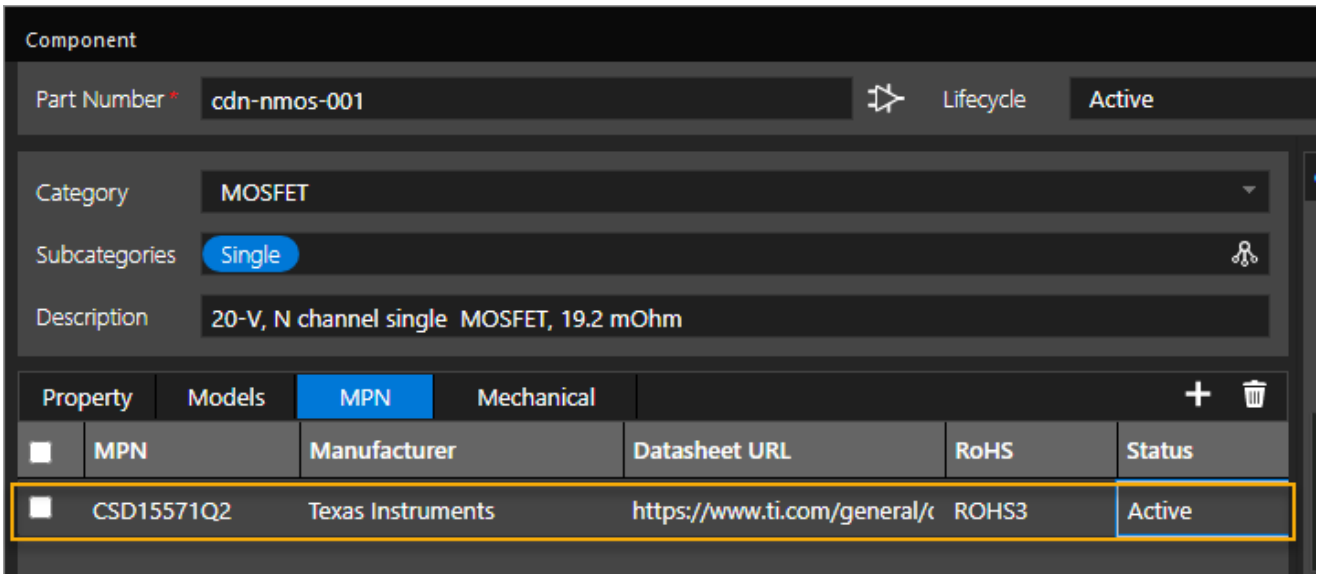
   MPN: `CSD15571Q2`

   Manufacturer: `Texas Instruments`

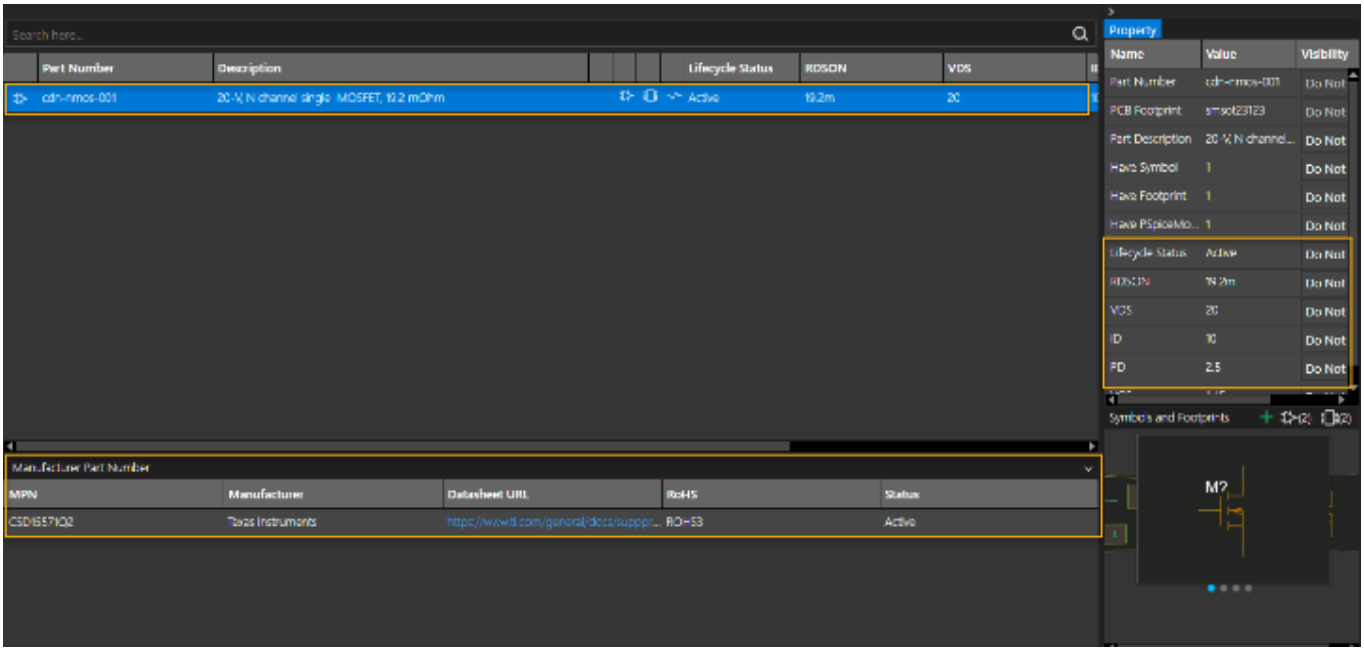
   Datasheet URL: `https://www.ti.com/general/docs/suppproductinfo.tsp?distId=10&gotoUrl=https%3A%2F%2Fwww.ti.com%2Flit%2Fgpn%2Fcsd15571q2`

   RoHS: `ROHS3`

   Status: `Active`



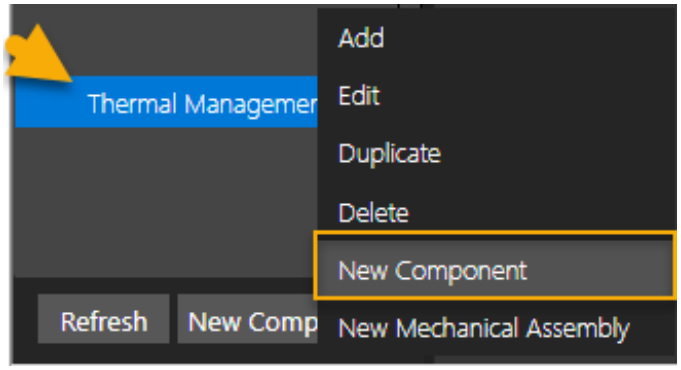
7. Click *Save*.
   The component is added and all the details you specified are available in the *Property* and *Manufacturer Part Number* sections.



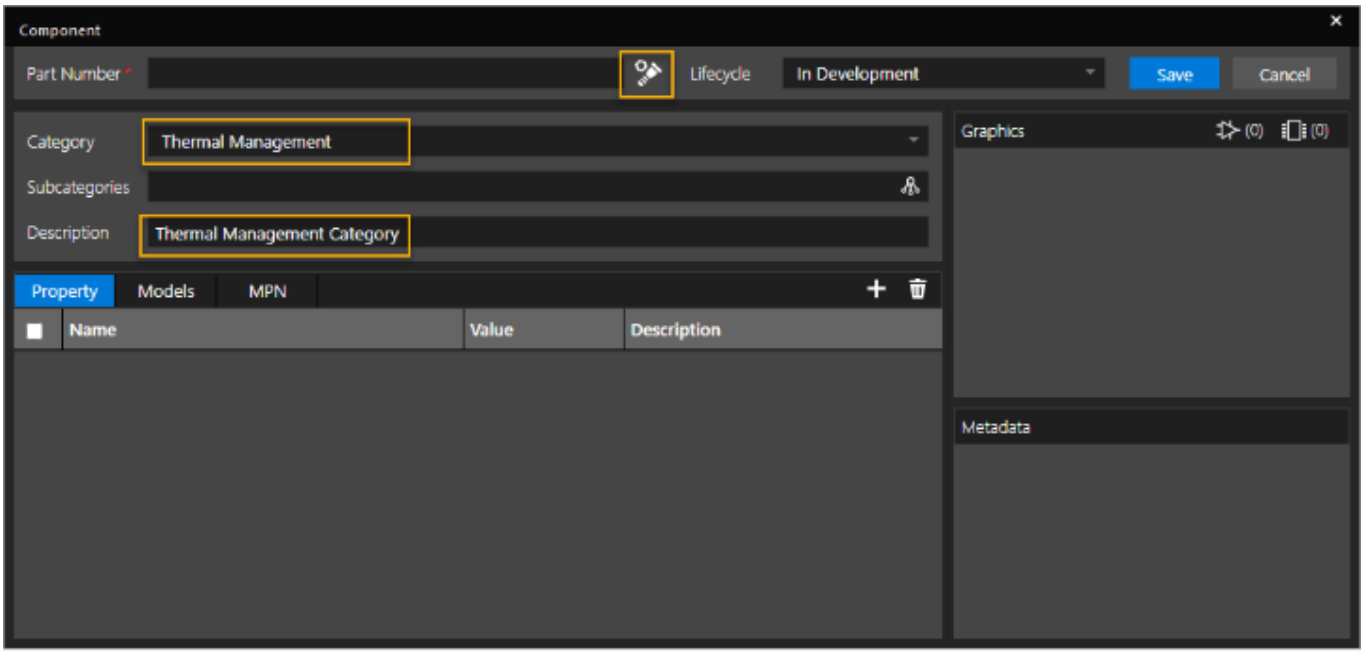
Creating Mechanical Part Based on a Category

To create a mechanical part, do the following:

1. Right-click *Thermal Management* and choose *New Component.*

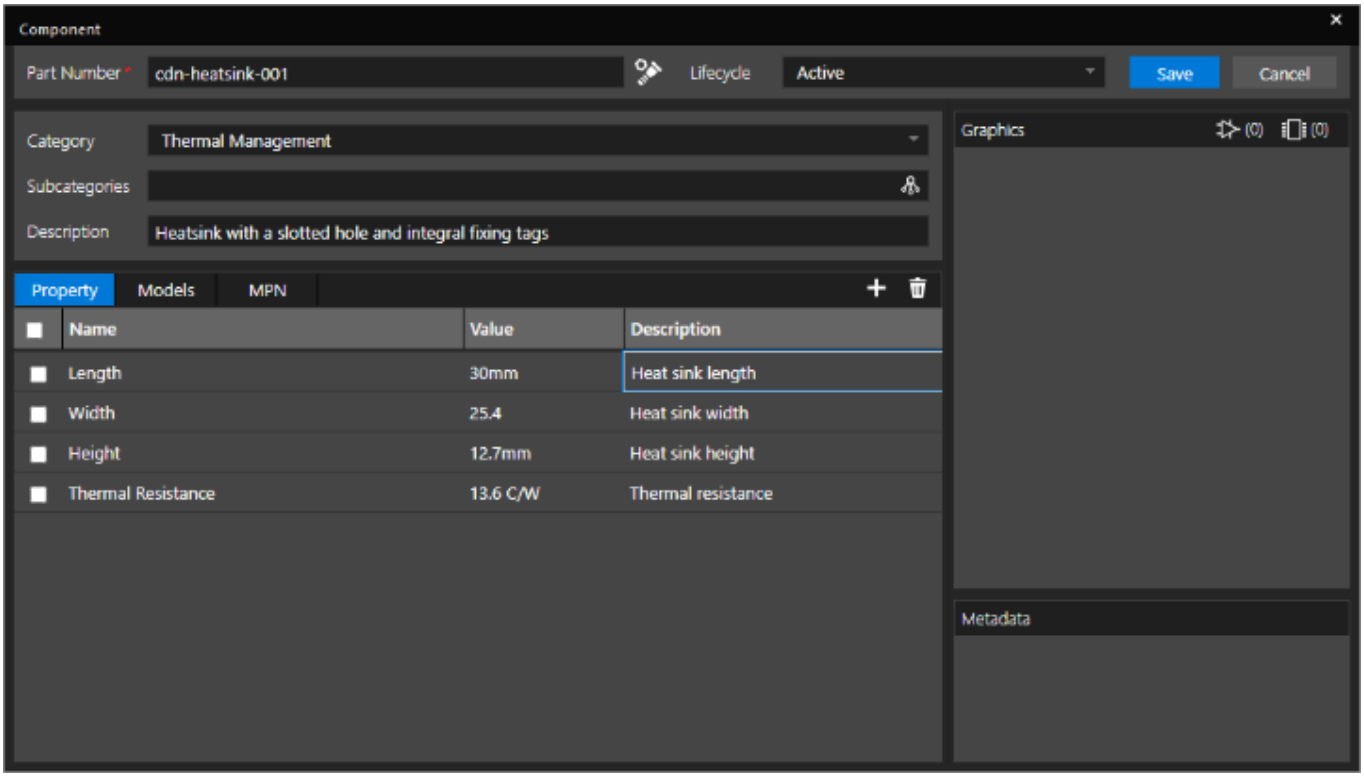


The *Component* dialog box opens pre-filled with the values inherited from the category.



2. Specify the following part details:
   Part Number: cdn-heatsink-001
   Lifecycle: Active
   Description: Heatsink with a slotted hole and integral fixing tags
3. Click the *Add* (⊕) icon in the *Property* tab to add new property rows with the following properties:

Property Name	Value	Description
Thermal Resistance	13.6 C/W	Thermal resistance
Height	12.7mm	Heat sink height
Width	25.4mm	Heat sink width
Length	30mm	Heat sink length



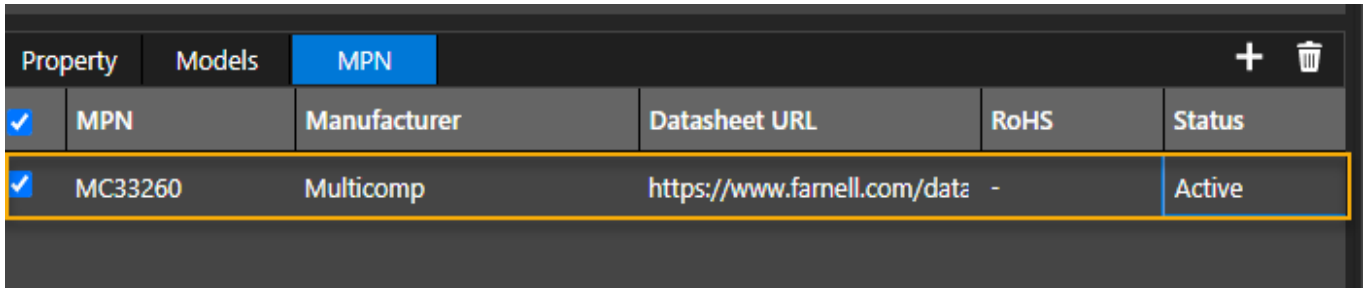
4. Select the *MPN* tab.
5. Click the *Add* (⊕) icon to add a new row and add the following MPN details:

   MPN: MC33260

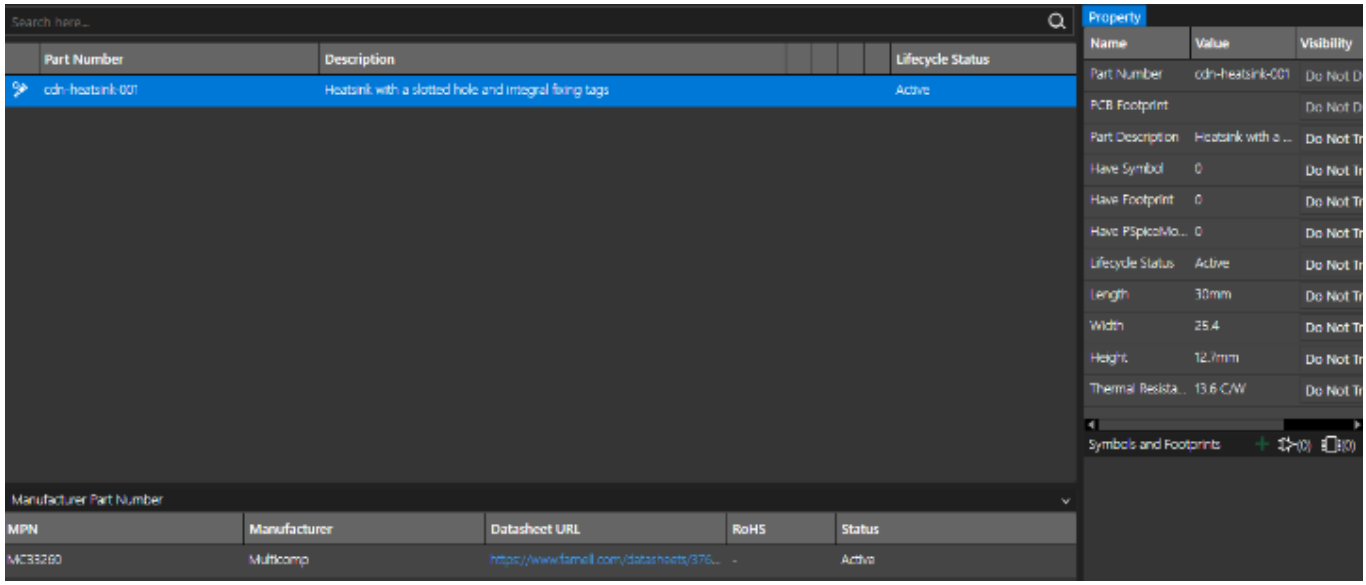
   Manufacturer: Multicomp

Datasheet URL: https://www.farnell.com/datasheets/3760481.pdf

RoHS: -
Status: Active



6. Click the *Save* button.



7. Similarly, create two more mechanical parts with the following details:

- Screw washer set
Part Number: cdn-screw-001
Lifecycle: Active
Description: M3 - 8mm (Heat Sink Screw)

For this part, add new property rows in the *Property* tab and add the following properties:

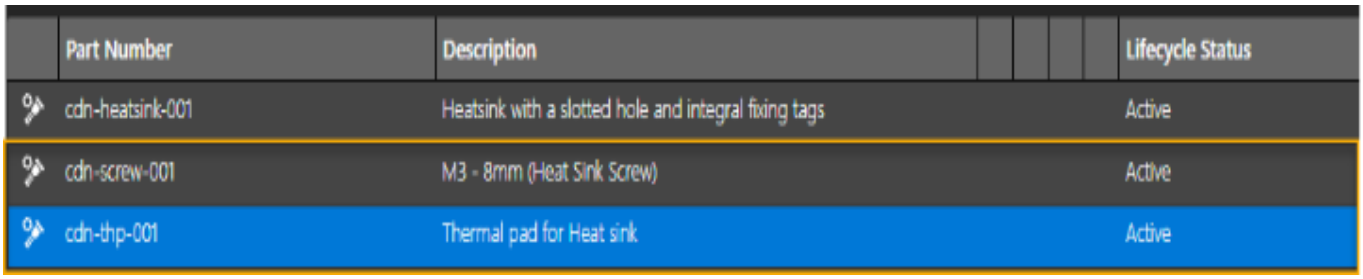
Property Name	Value	Description

Length	8mm	Screw length below head
Thread Size	M3	Thread size
Head Type	Pan Head	Screw head shape type

- Thermal pad
  Part Number: cdn-thp-001
  Lifecycle: Active
  Description: Thermal pad for Heat sink

  For this part, add new property rows in the *Property* tab and add the following properties:

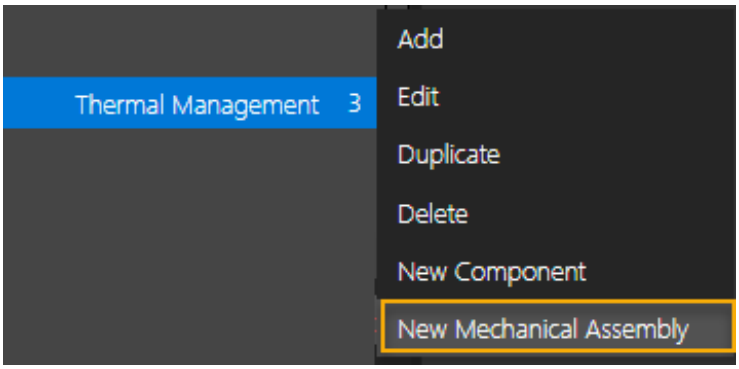
Property Name	Value	Description
Width	19.05mm	Thermal pad width
Length	12.70mm	Thermal pad length



Creating Mechanical Assembly

To create a new mechanical assembly, do the following:

1. Right-click the mechanical category, choose *New Mechanical Assembly*.

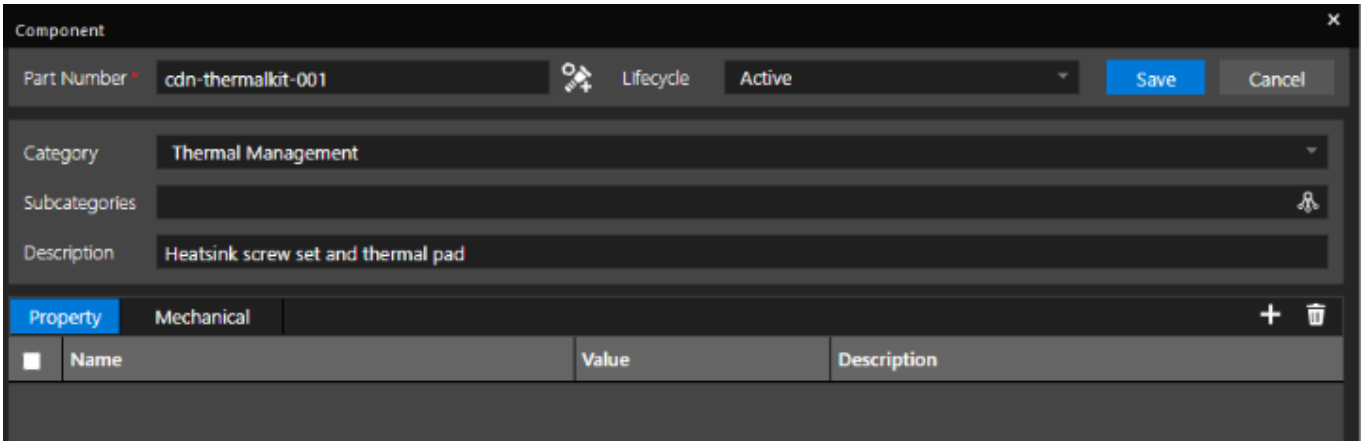


2. In the *Component* dialog box, specify the following details:

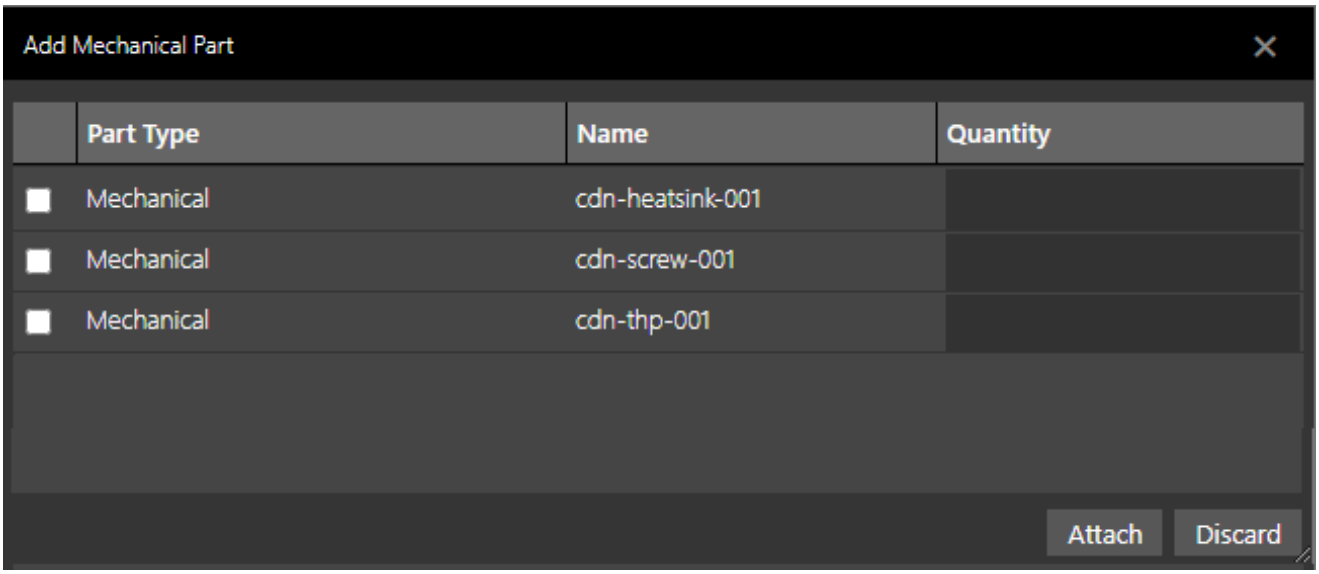
   Part Number: cdn-thermalkit-001

   Lifecycle: Active

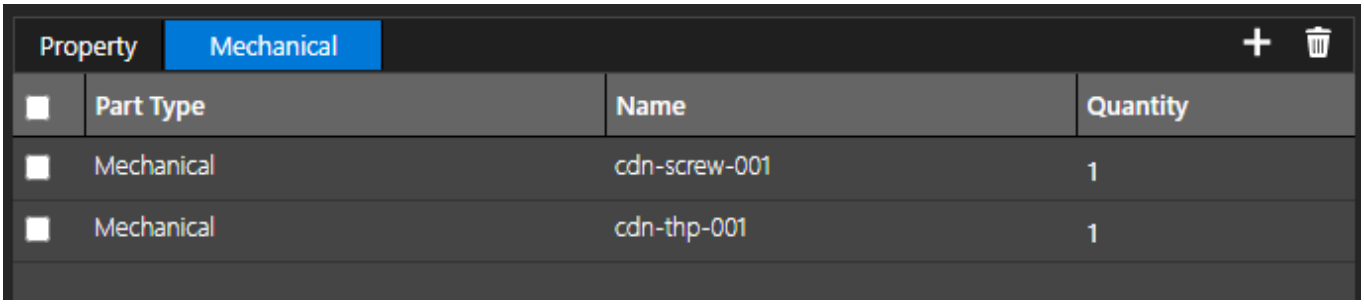
   Description: Heatsink screw set and thermal pad



3. Select the *Mechanical* tab.
4. Click the *Add* (⊕) icon.
   The *Add Mechanical Part* dialog box opens. A list of available mechanical parts is displayed.

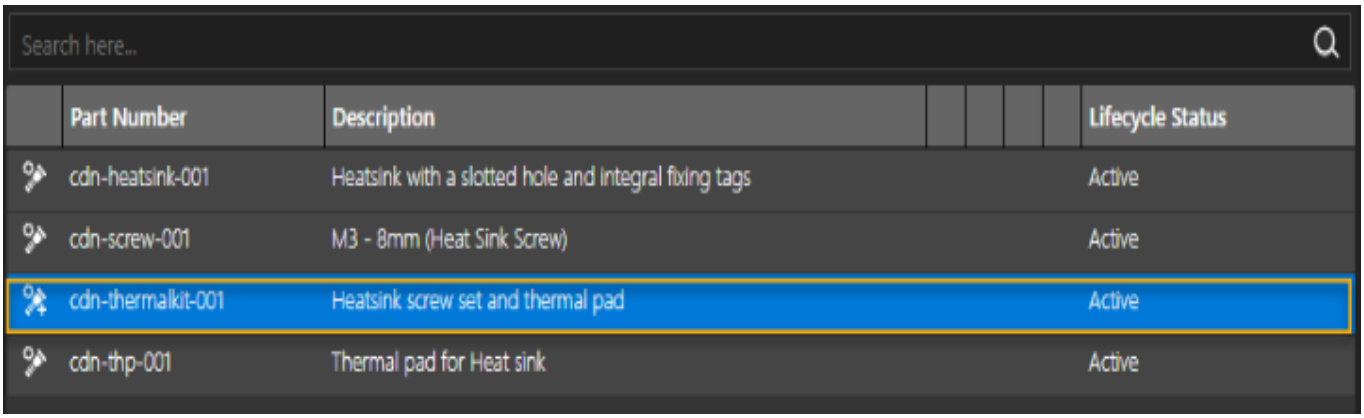


5. Select the check box to the left of *cdn-screw-001* and *cdn-tp-001.*
   The quantity is automatically set as 1 for each of the mechanical parts to be included in the assembly.
6. Click *Attach*.
   The selected parts are added in the *Mechanical* tab.



7. Click *Save.*

   The mechanical assembly is displayed in the part browser (middle pane) of Component Explorer.


Search here...
Part Number
Description
Lifecycle Status
cdn-heatsink-001
Heatsink with a slotted hole and integral fixing tags
Active
cdn-screw-001
M3 - 8mm (Heat Sink Screw)
Active
cdn-thermalkit-001
Heatsink screw set and thermal pad
Active
cdn-thp-001
Thermal pad for Heat sink
Active

4

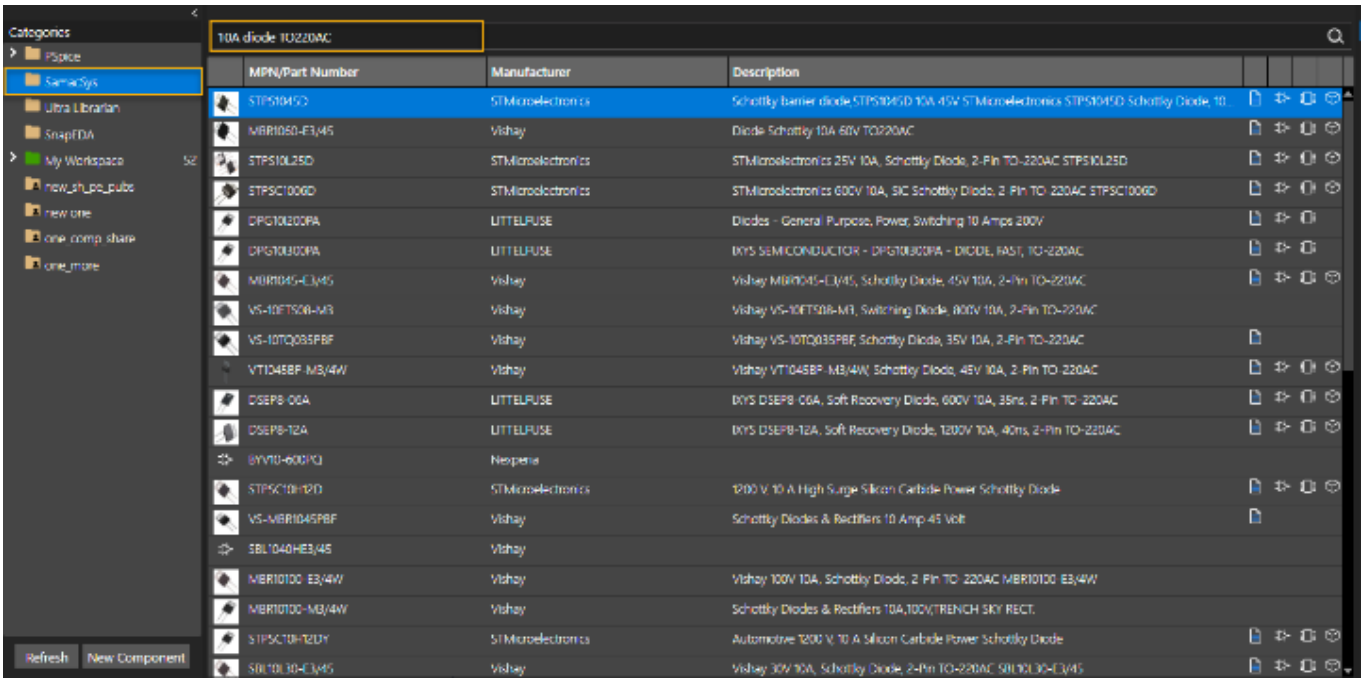
Module 4: Creating New Components from Content Provider Databases

You can use parts provided by external content providers, such as SamacSys, Ultra Librarian, and SnapEDA to create new parts and save them in the workspace.

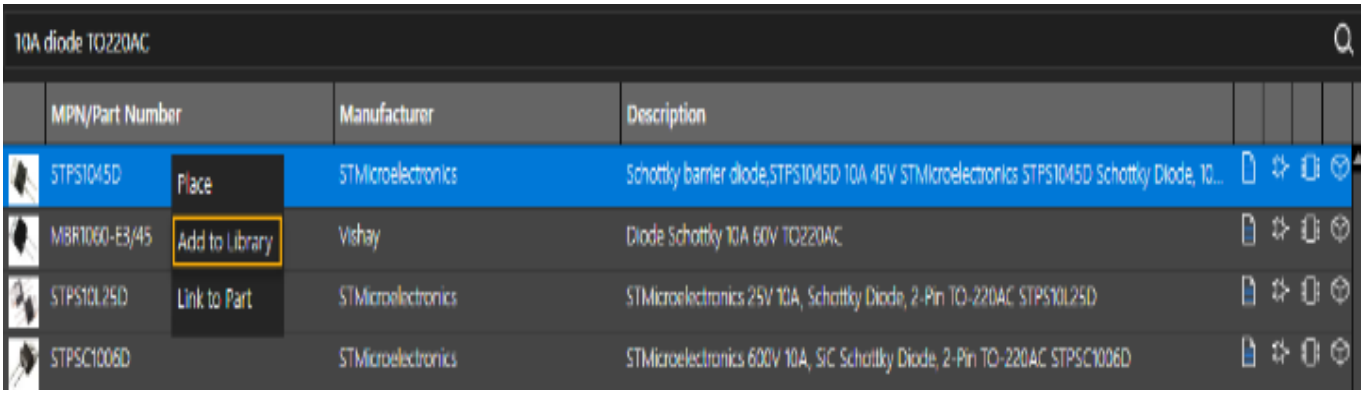
Adding Components to Workspace

To create an electrical part, do the following:

1. In Component Explorer, select the *SamacSys* node.
2. Type the string, `10A diode TO220AC`, in the Search bar and press `Enter`.
   Search results matching the specified search string are displayed:

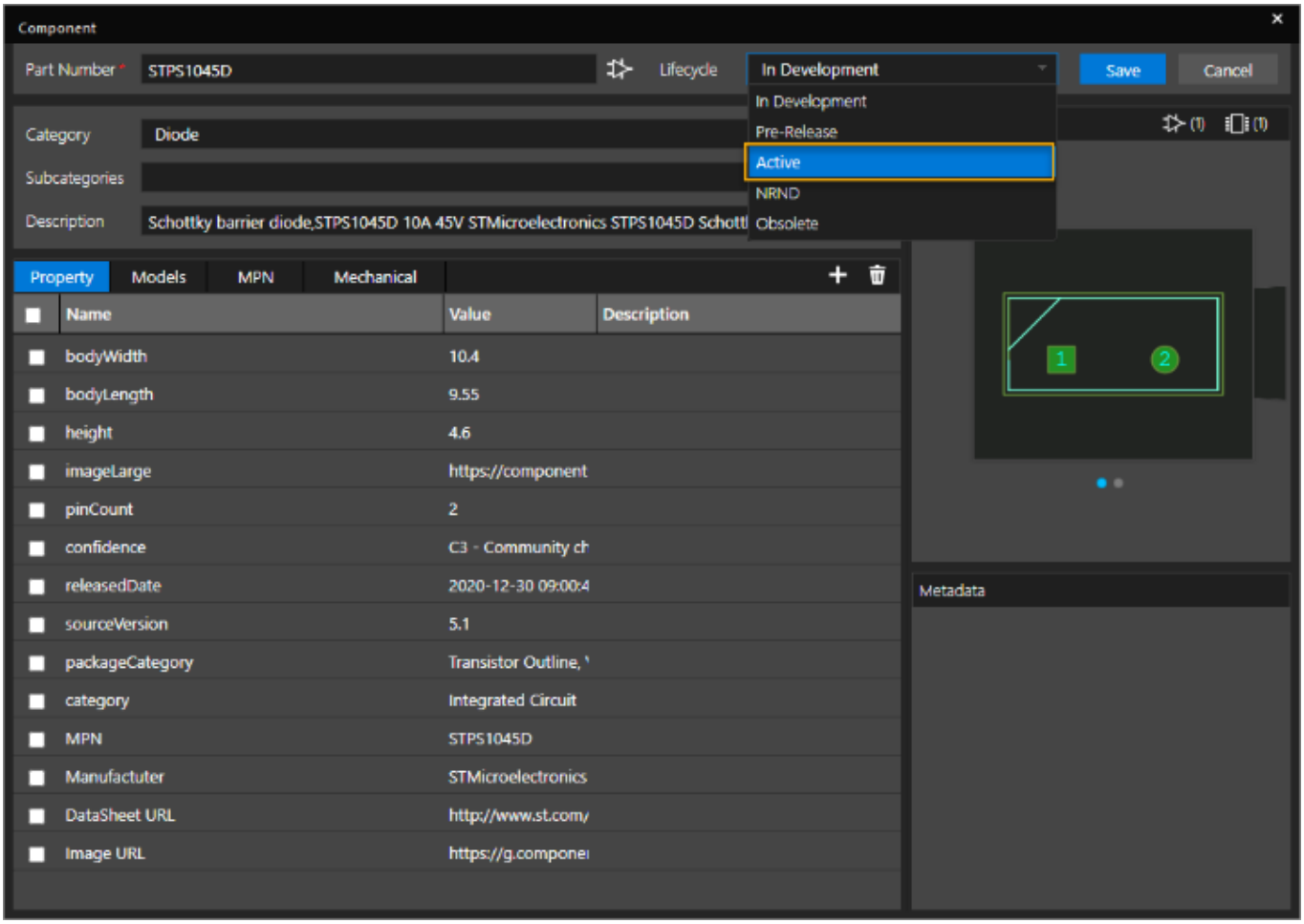


3. Right-click the component row, *STPS1045D*, and choose *Add to Library*.



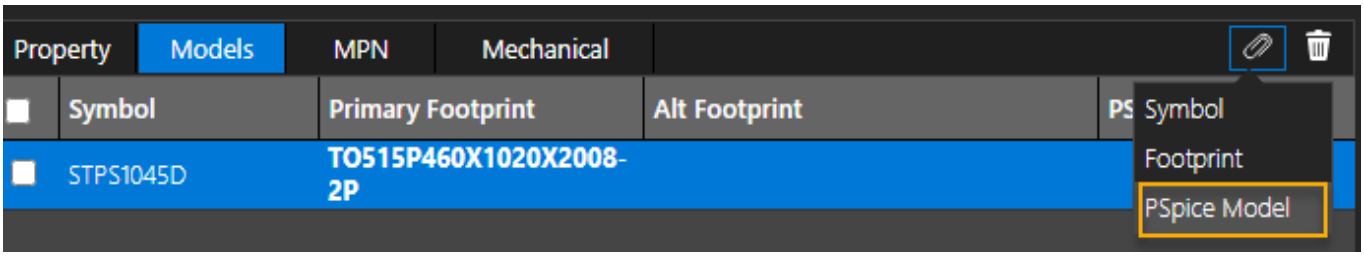
The *Component* dialog box opens with the details of the component provided in the SamacSys database.

4. Change the Lifecycle status of the component from *In Development* to to *Active*.

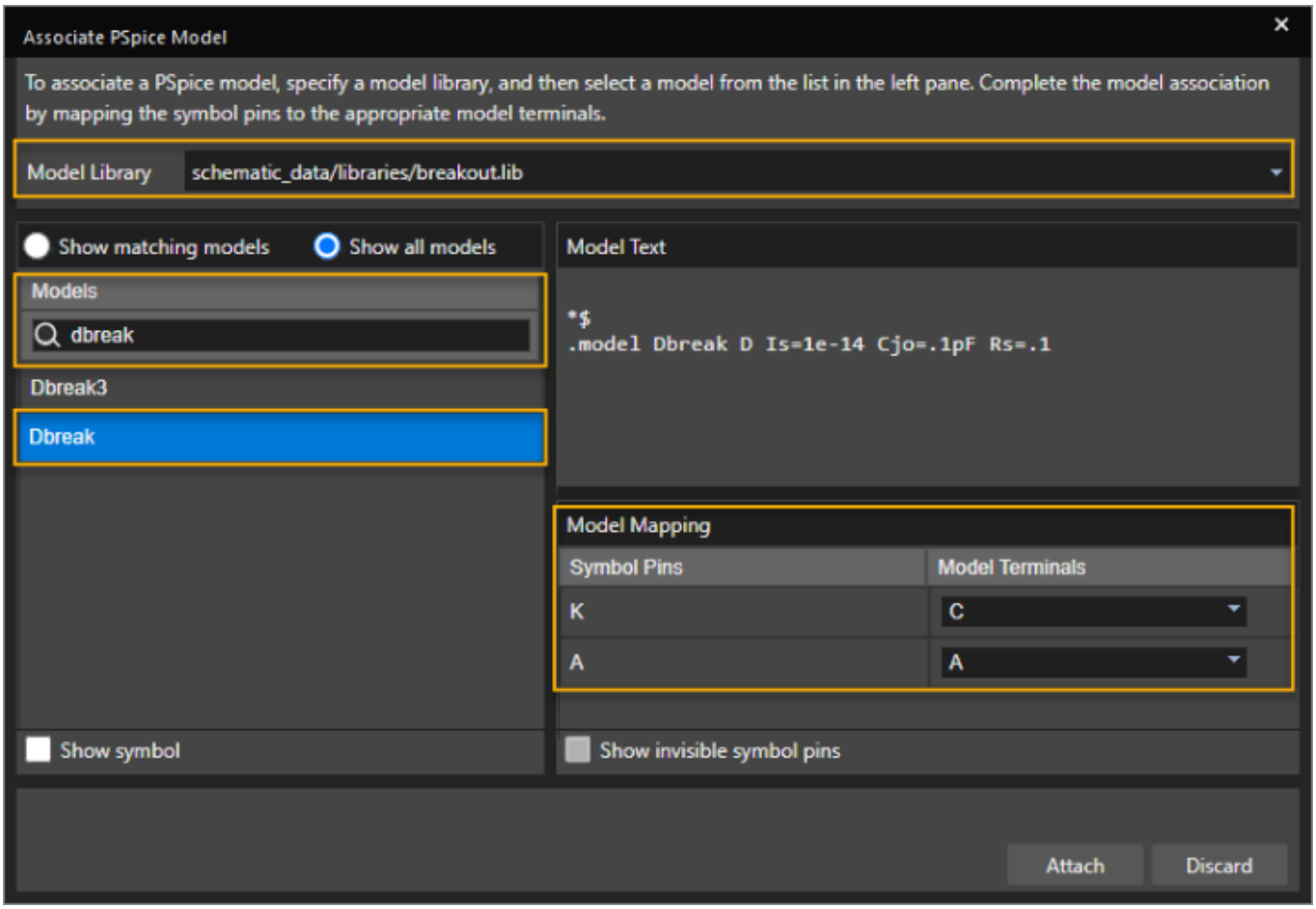


Associating Multiple PSpice Models to a Component

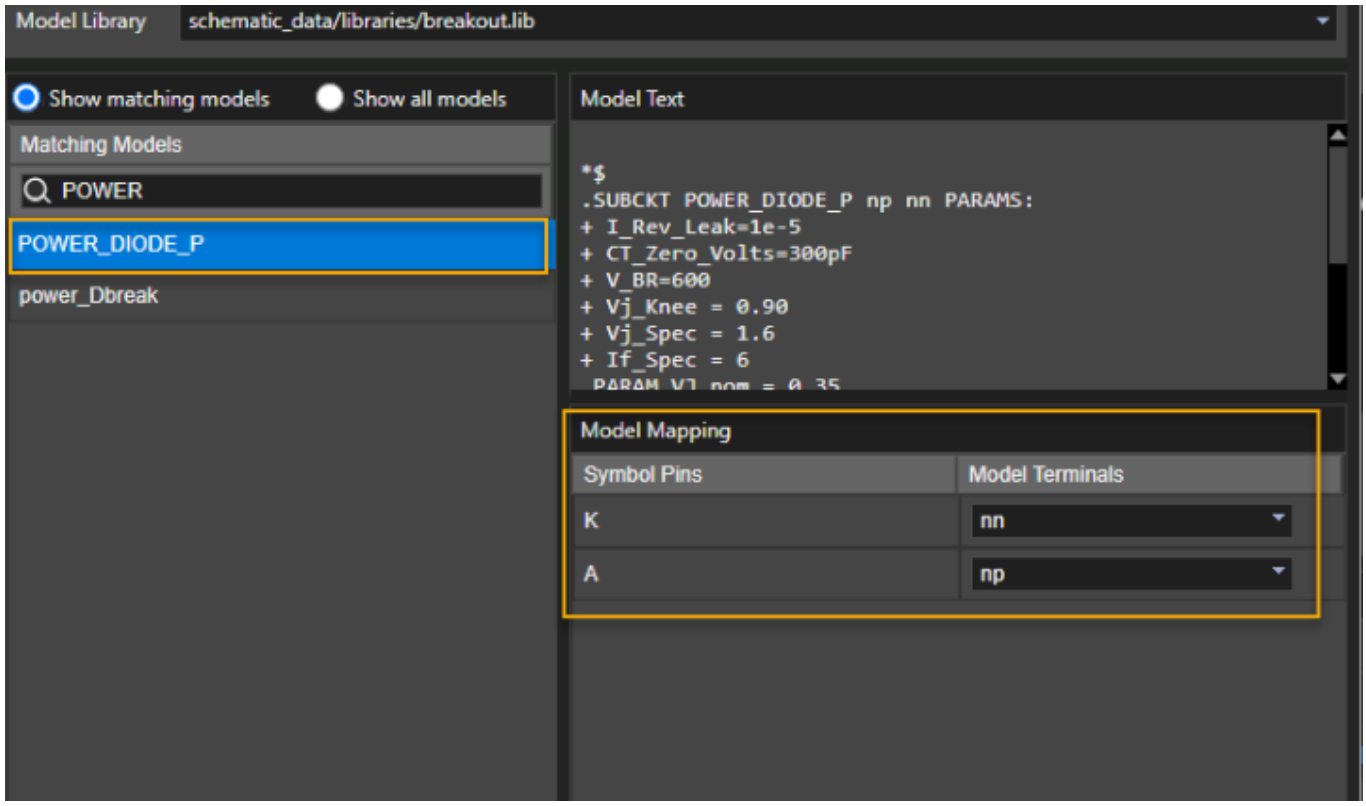
1. In the *Models* tab, select the model, click the *Attach* (clip) icon and choose *PSpice Model*.



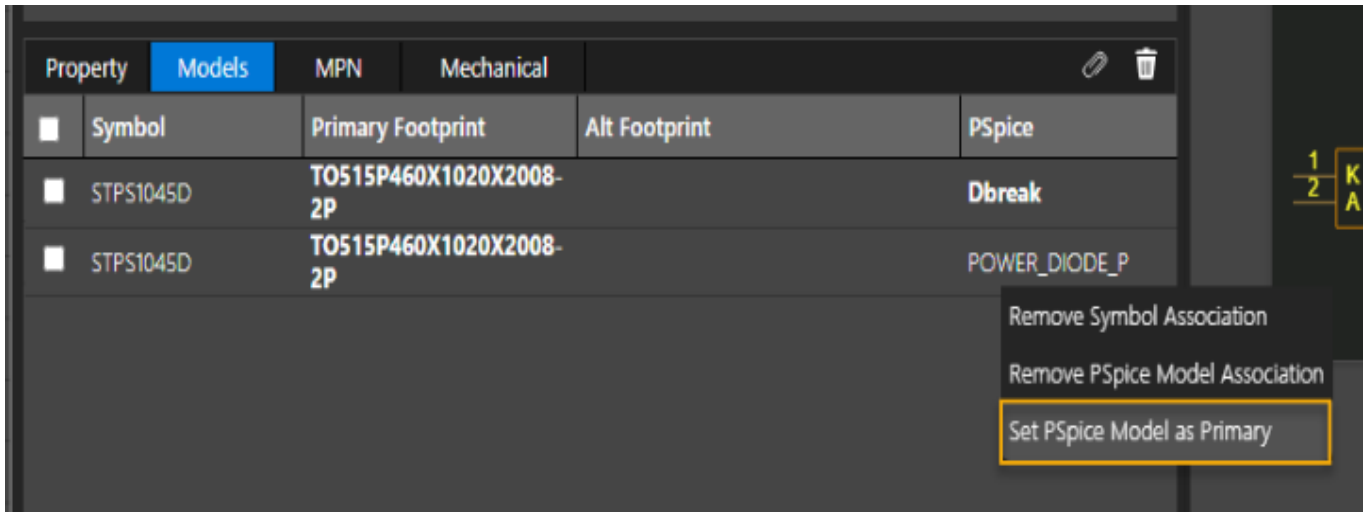
2. In the *Associate PSpice Model* dialog box, search for the PSpice model, *Dbreak* and map the model pins with the component pins as shown in the following image:



3. Click *Attach*.
4. Repeat steps 1-3 to associate another PSpice model, `POWER_DIODE_P`, with the symbol.



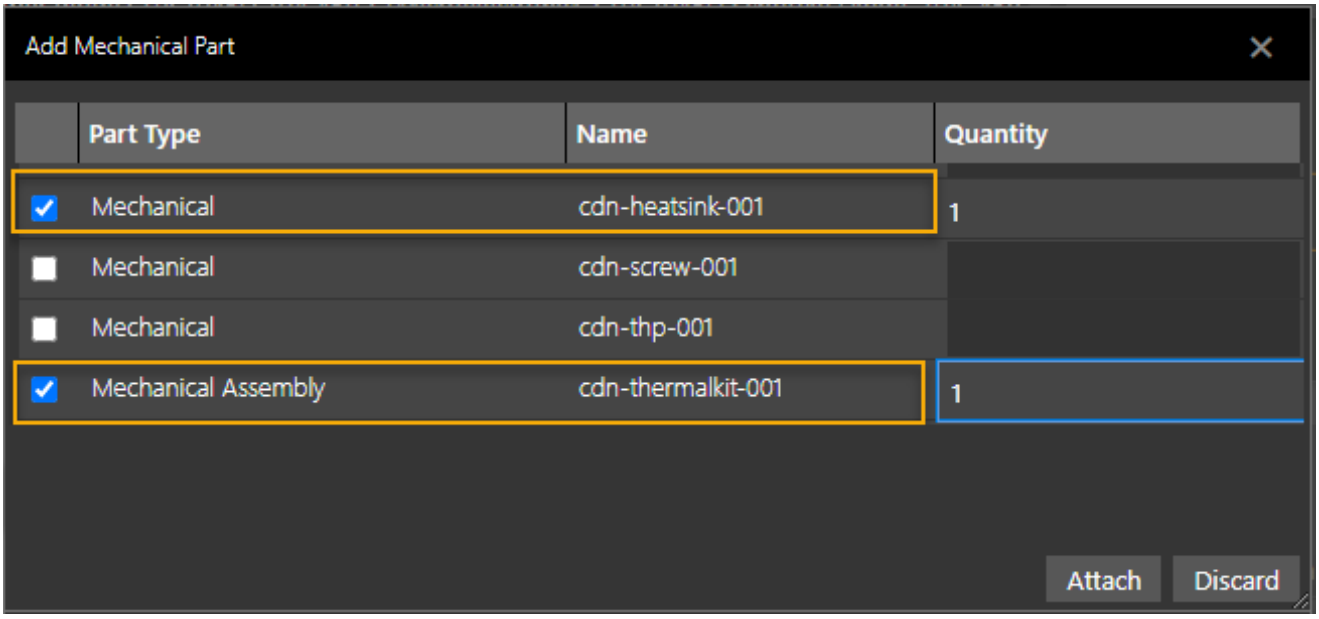
5. Right-click the second row and choose *Set PSpice Model as Primary*.



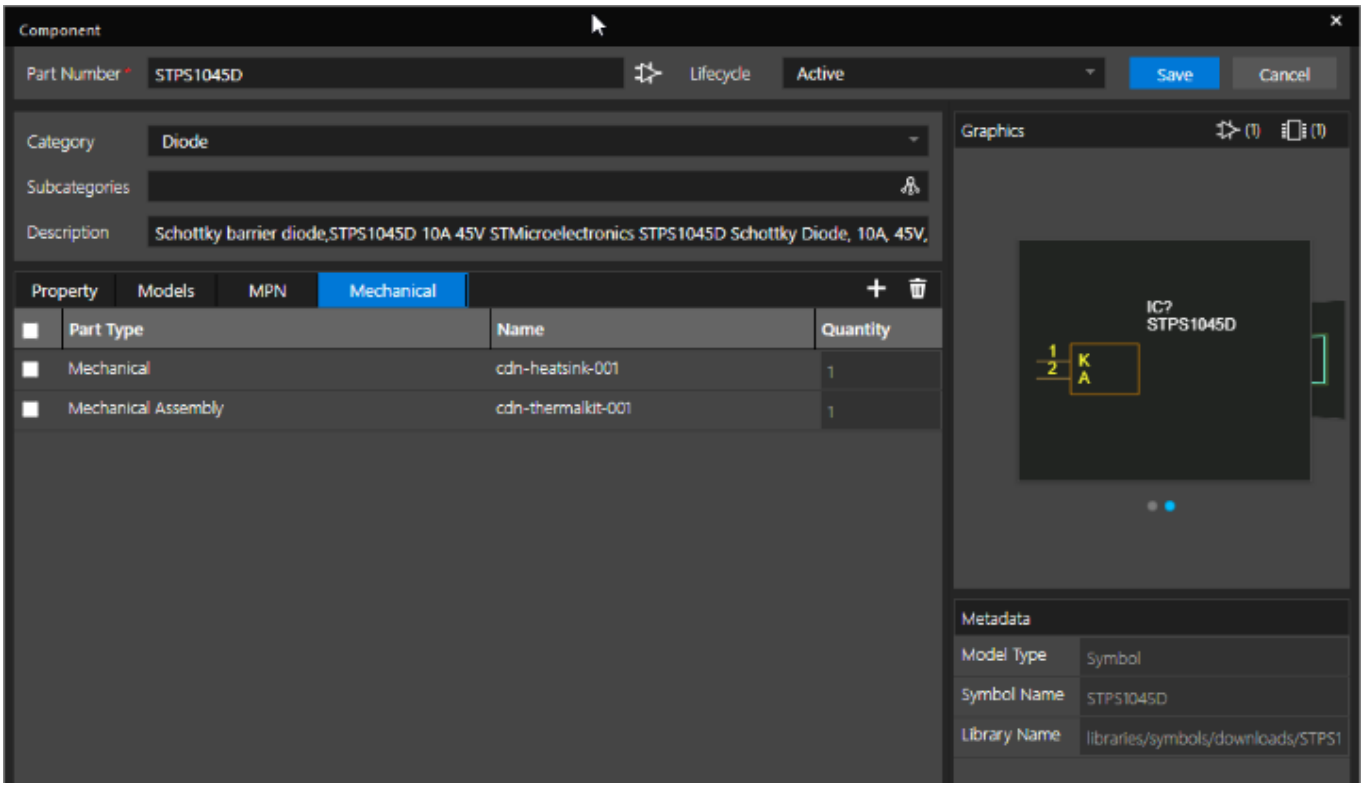
Associating Mechanical Models with a Component

To associate a mechanical model with a component, do the following:

1. Select the *Mechanical* tab.
2. Click the Add (⊕) icon to open the *Add Mechanical Part* dialog box.
3. Select the checkboxes for *cdn-heatsink-001* and *cdn-thermalkit-001.*
   As you select a mechanical part, a quantity of `1` is automatically assigned to the part.

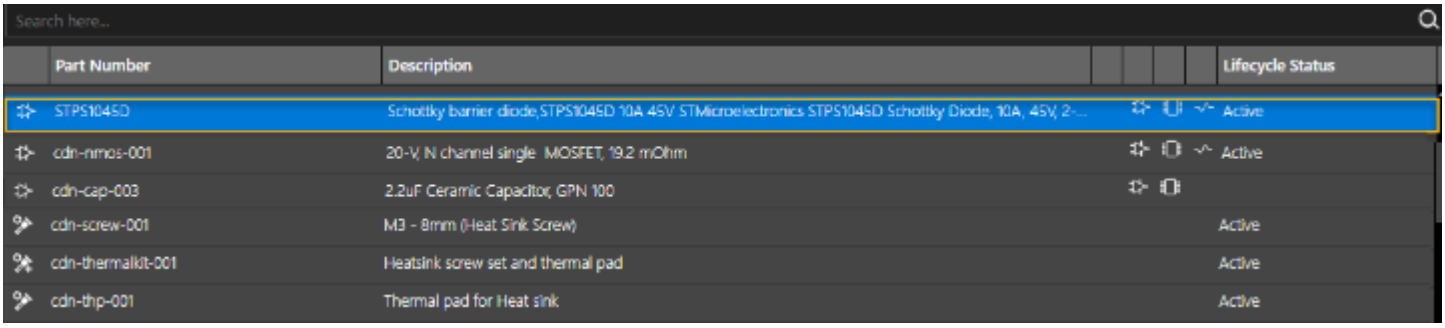


4. Click *Attach*.



5. Click *Save*.

The component is added to *My Workspace*.

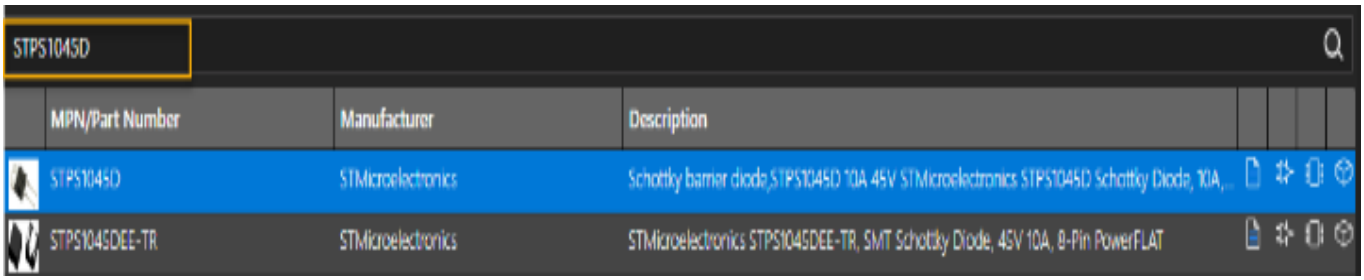


Associating Manufacturer Part Number from Content Providers

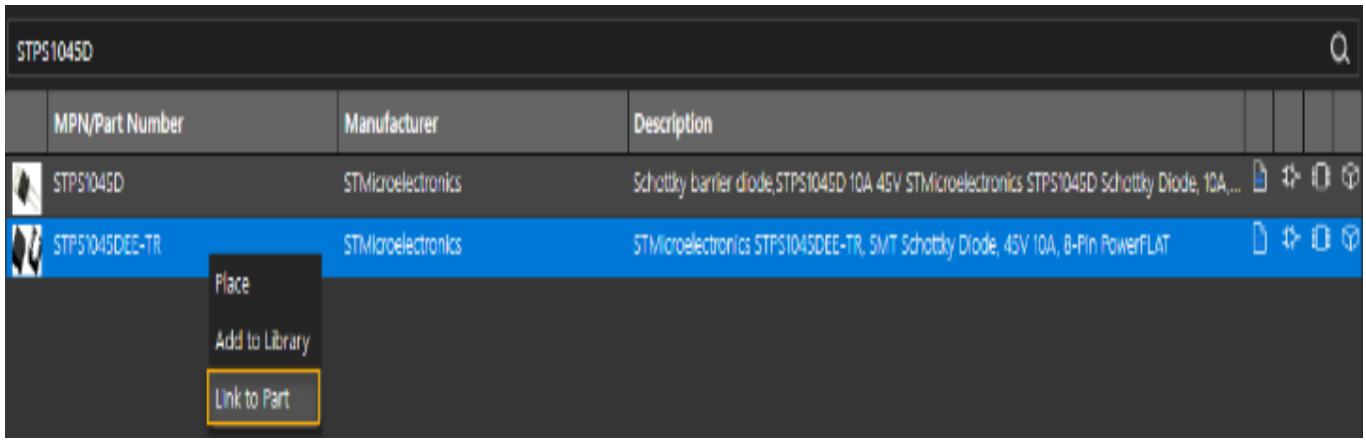
Next, you will associate a Manufacturer Part number (MPN) from the content provider to a component in your local workspace.

To associate an MPN with an existing component, do the following:

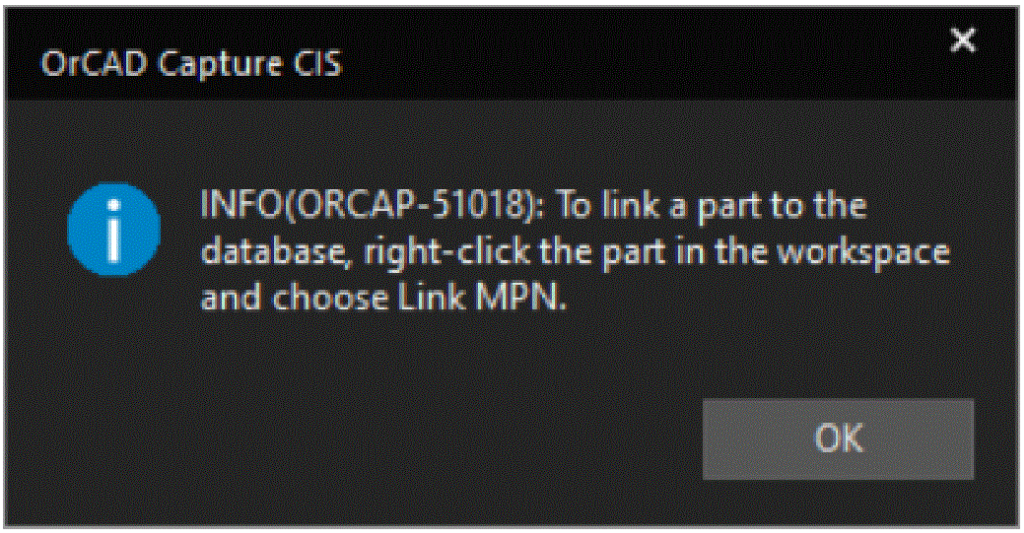
1. In Component Explorer, select the *SamacSys* node and type *STPS1045D* in the search bar and press Enter.



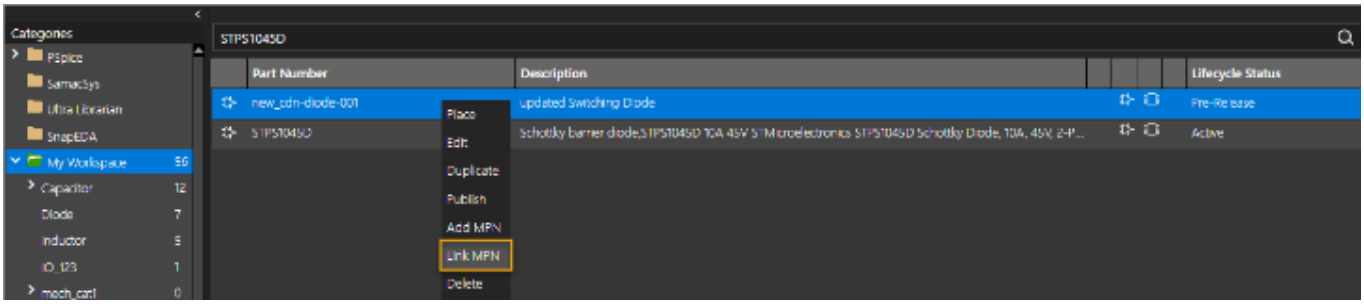
2. Right-click the component row for MPN *STPS1045DEE-TR* and choose *Link to Part*.



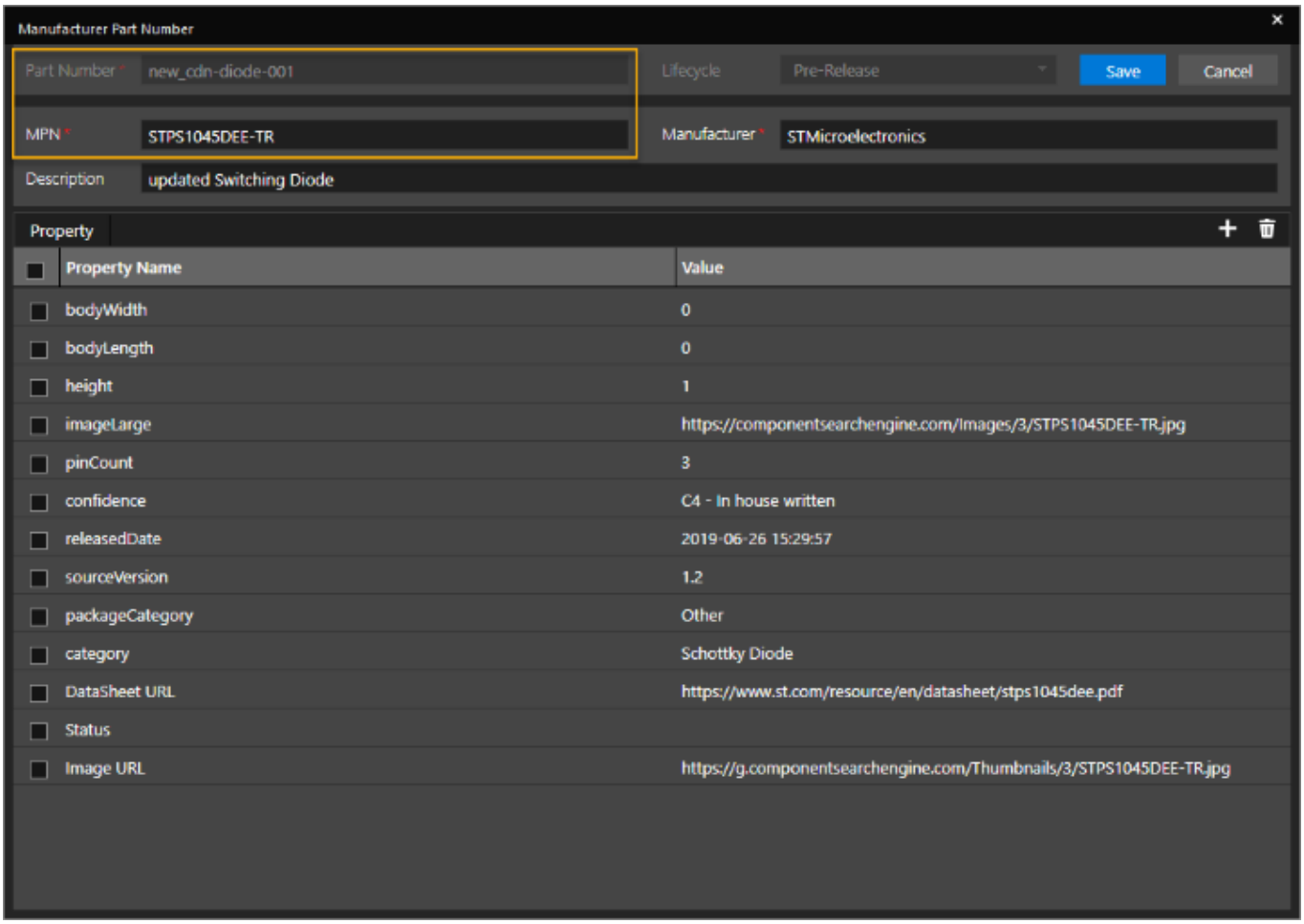
A message pops up informing about the next step.



3. Click *OK*.
4. With the same search string in search bar, select the *My Workspace* node.
   The part browser displays filtered results from *My Workspace*. The matching component in the workspace is *new_cdn-diode-001*.
5. Right-click the component to be linked and choose *Link MPN*.

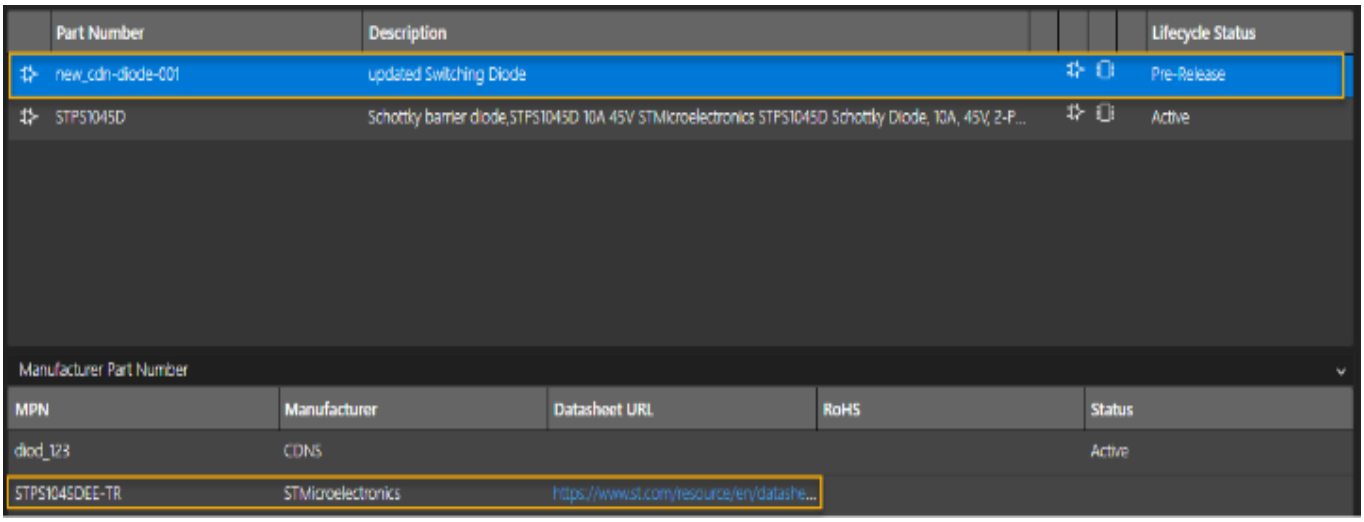


The *Manufacturer Part Number* dialog box is displayed.



6. Click *Save* to associate this MPN with the component *new_cdn-diode-001.*

   The associated MPN is now seen in the *Manufacturer Part Number* section in the part browser.


Part Number
Description
Lifecycle Status
new_cdn-diode-001
updated Switching Diode
Pre-Release
STPS1045D
Schottky barrier diode,STPS1045D 10A 45V STMicroelectronics STPS1045D Schottky Diode, 10A, 45V, 2-P...
Active
Manufacturer Part Number
MPN
Manufacturer
Datasheet URL
RoHS
Status
diod_123
CDNS
Active
STPS1045DEE-TR
STMicroelectronics
https://www.st.com/resource/en/datashe...

5

Module 5: Sharing Components with Team Members

In this module, we will share the components created in this tutorial with team members using a shared workspace.

In this module, you will do the following tasks:

- Publishing Components to Shared Workspace
- Editing and Copying Components from Shared Workspace
- Modifying a Component and Publishing it back to Shared Workspace
- Restoring Older Versions of Components in Shared Workspaces

Publishing Components to Shared Workspace

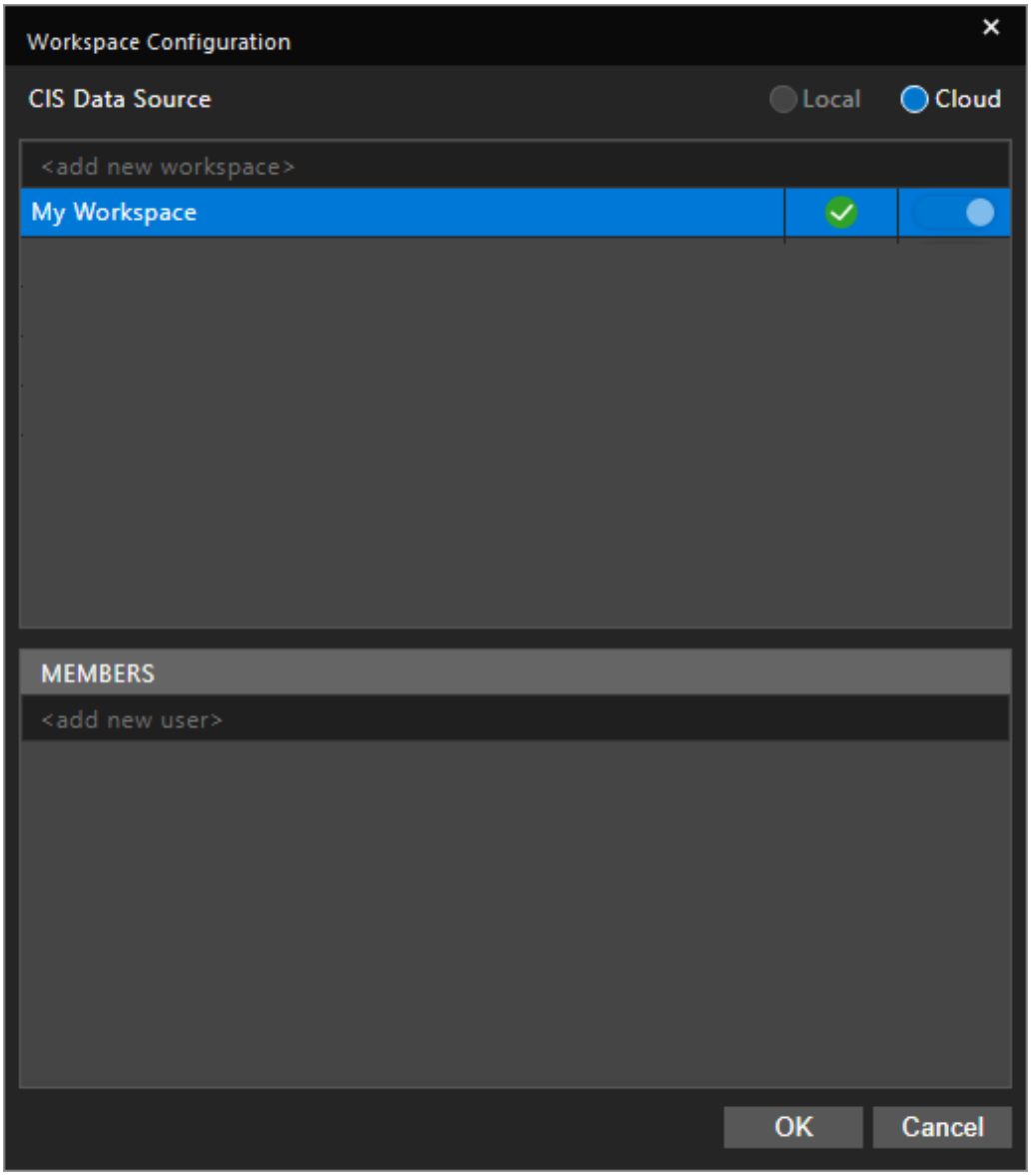
Before you can publish components, you need to create a shared workspace and add members to the shared workspace from the *Workspace Configuration* dialog box. To share the workspace, you need to provide team members access to the workspaces by assigning them roles with predefined access permissions to the workspaces. See Sharing Workspaces to learn about various roles and their associated privileges.

Create a Shared Workspace

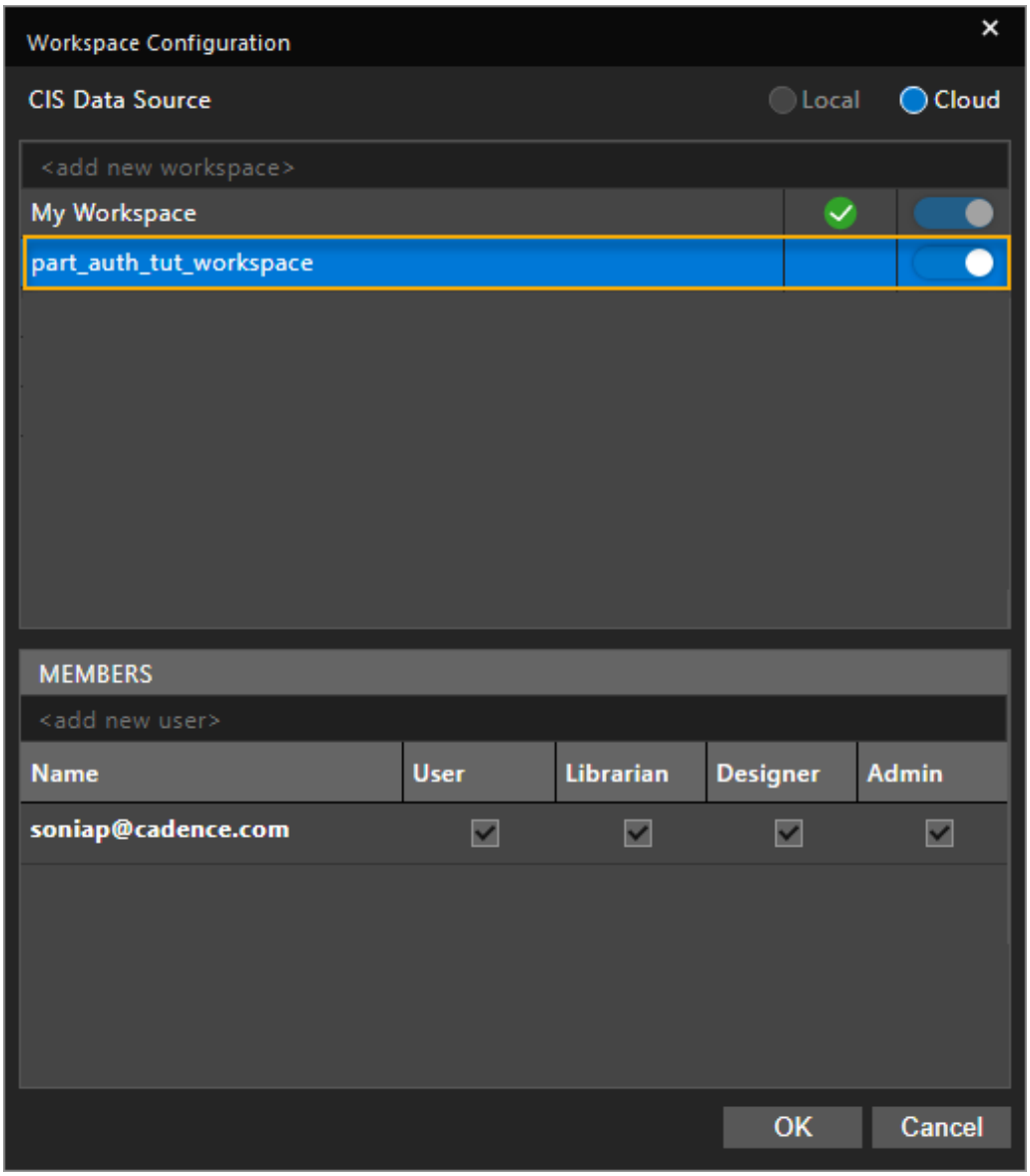
To create a shared workspace, do the following:

1. Choose *View – Workspace – Configuration* from the Capture main menu.

The *Workspace Configuration* dialog box is displayed.



2. Now click the input field with the *<add new workspace>* string, specify the workspace name as *part_auth_tut_workspace*, and press `Enter`.



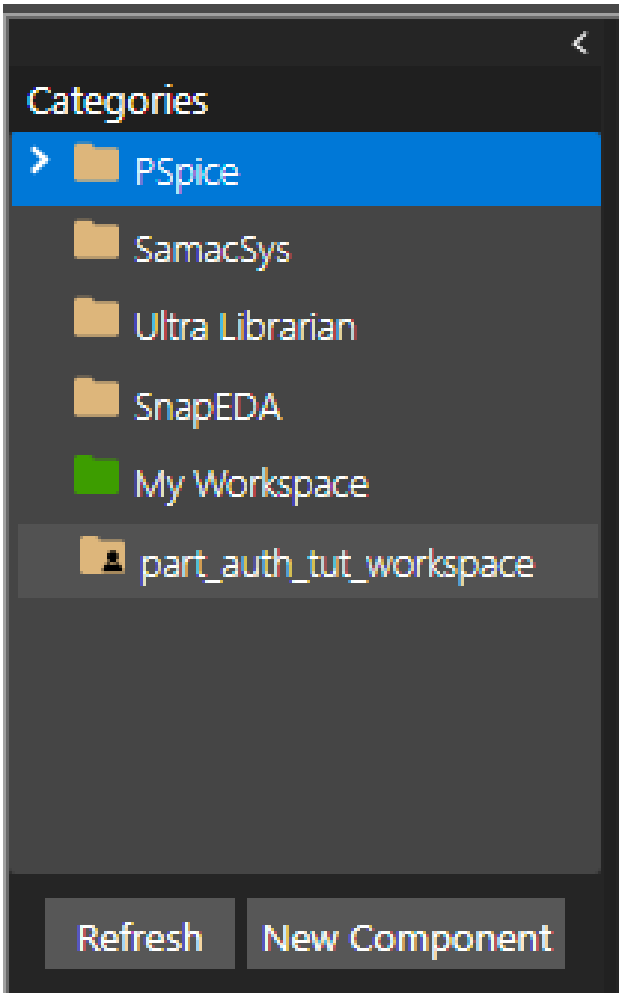
The user who created the shared workspace is the default owner of the workspace with *Admin* rights. The username appears in bold in the MEMBERS section. You can grant access rights to your team members on this workspace from the MEMBERS section.

See Sharing Workspaces to learn more.

3. Click *OK* to create the new shared workspace.

   The shared workspace is created.

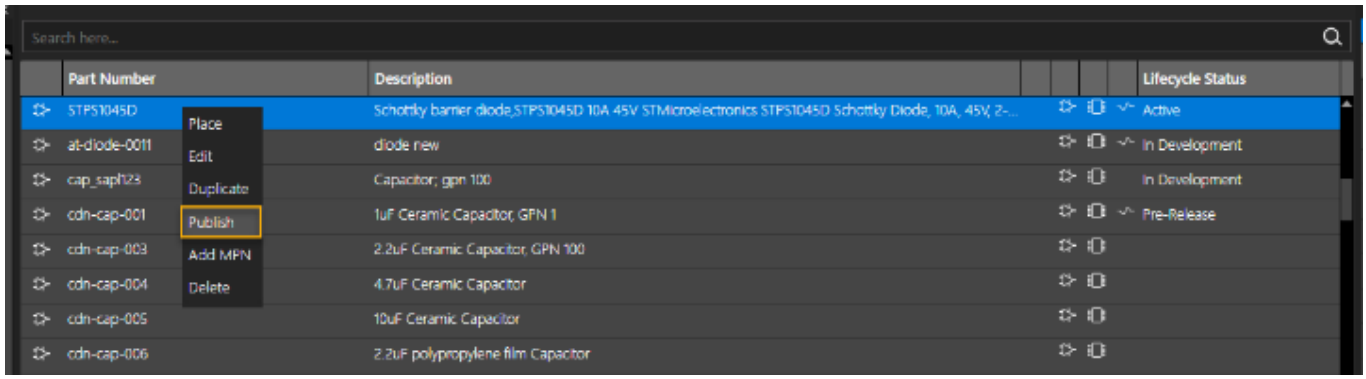
4. Choose *Place – Component* to launch the *Component Explorer* UI and view the newly added shared workspace in the *Categories* tree.



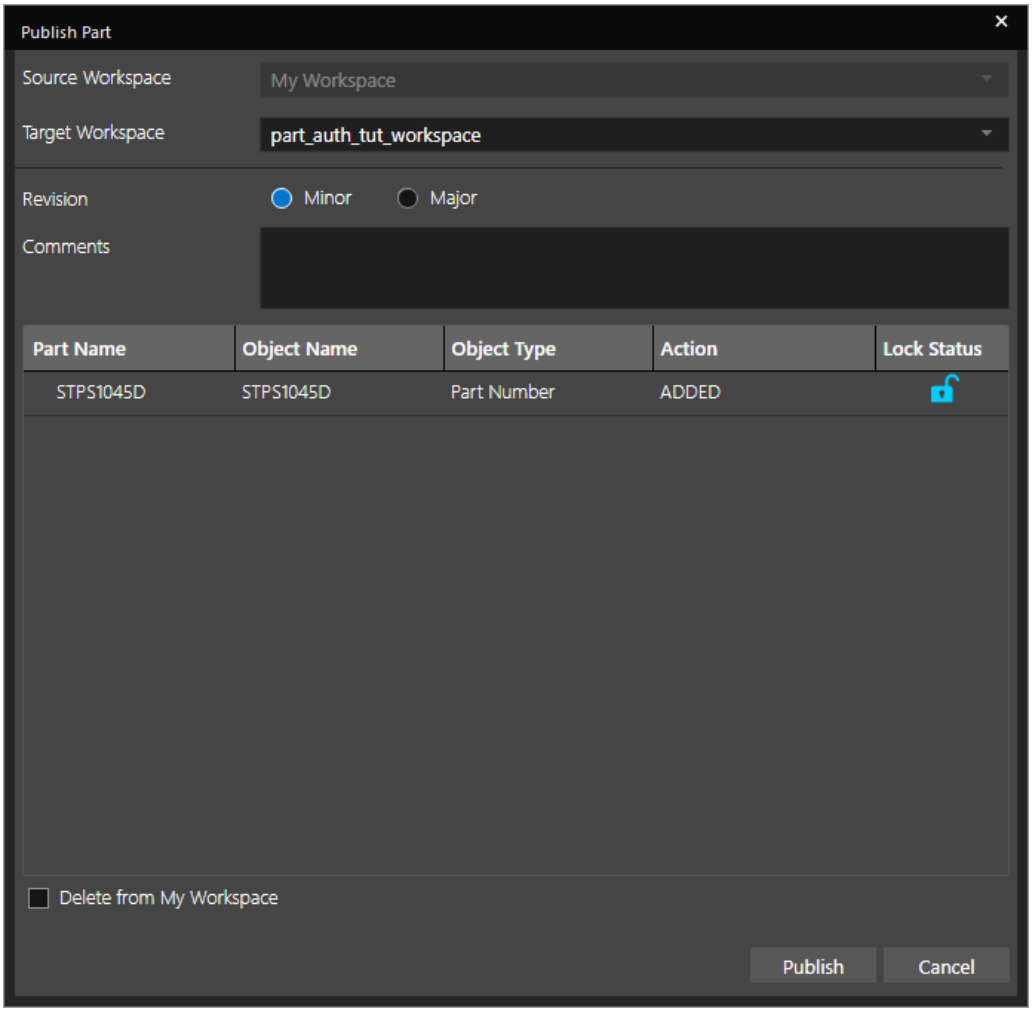
Publishing a Component

To publish a component , do the following:

1. Select *My Workspace* in Component Explorer.
   The *Workspace Configuration* dialog box is displayed.
2. Select part row *STPS1045D*.
3. Right-click and choose *Publish*.



The *Publish* dialog box is displayed.

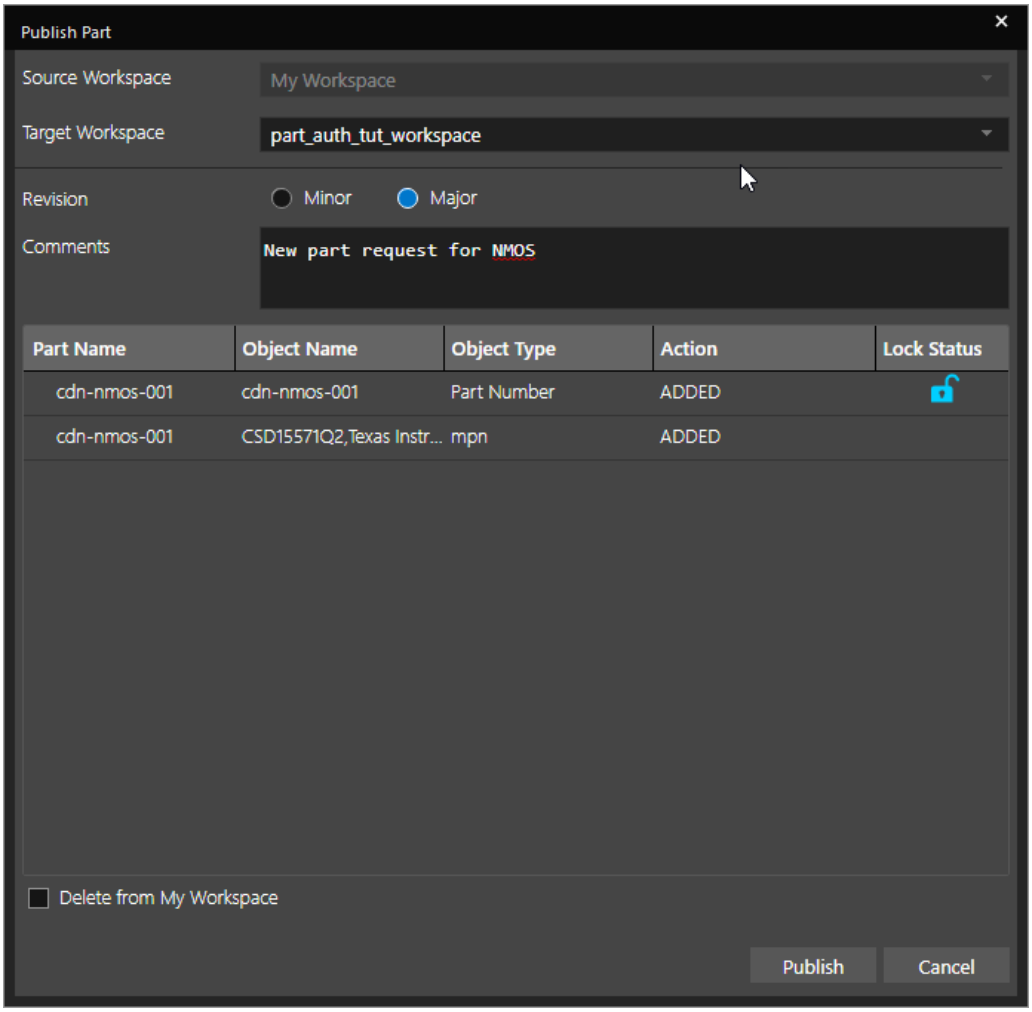


4. Specify the following entries for revision and comments:

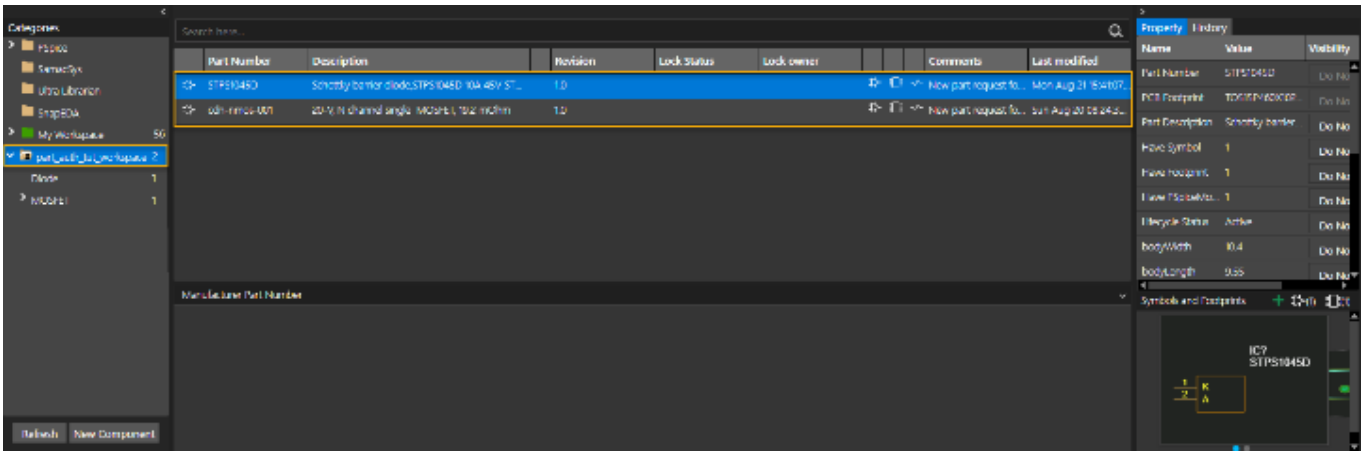
Revision: *Major*
Comments: *New part request for tutorial*

As this component is being published for the first time, the *Action* column shows the value as *Added*.

5. Click *Publish*.
6. Similarly, publish the component, *cdn-nmos-001* that you created in the previous module.



The published components can be accessed in the shared workspace by all the users with whom the workspace is shared.

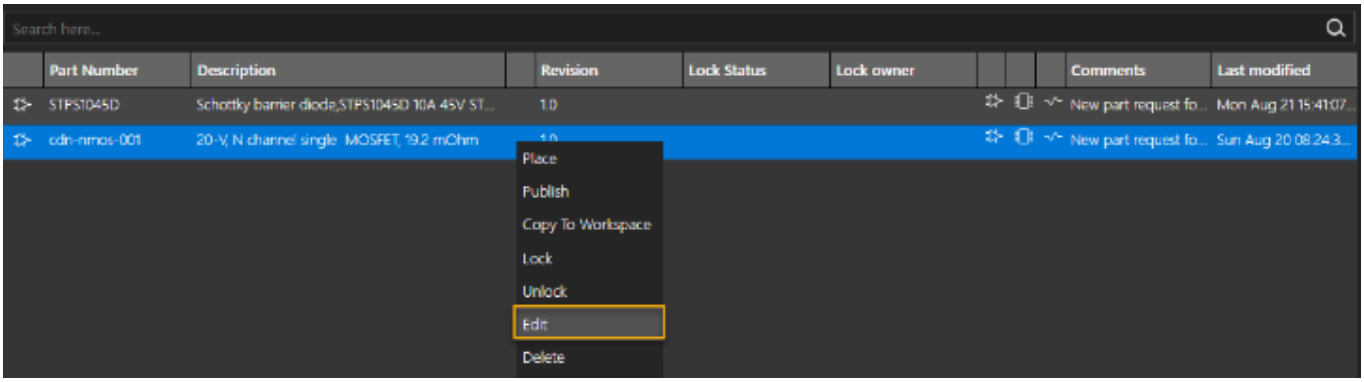


Editing and Copying Components from Shared Workspace

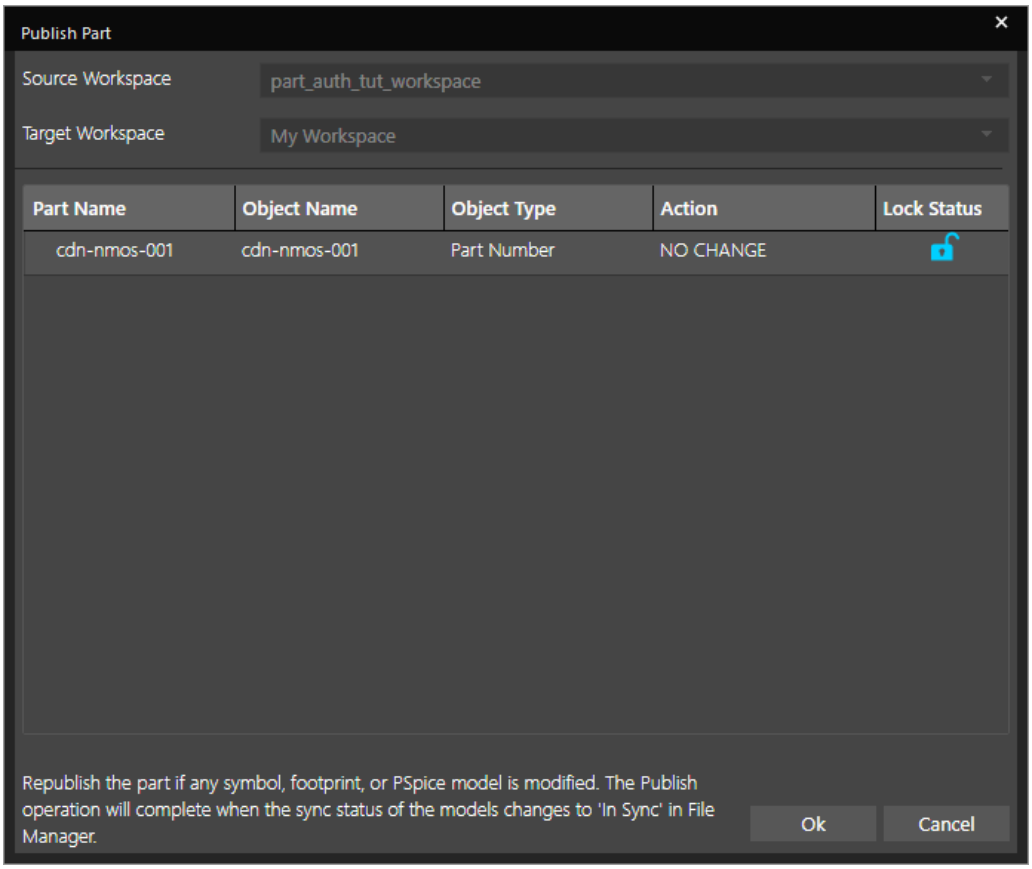
For this exercise, you can move to another system using another user id with access to the shared workspace and a *Librarian* or *Admin* role to work on the library components.

To edit a component on a shared workspace, do the following:

1. In Component Explorer, select the shared workspace, *part_auth_tut_workspace*.
2. In the part browser, right-click the component row for *cdn-nmos-001* and choose *Edit*.

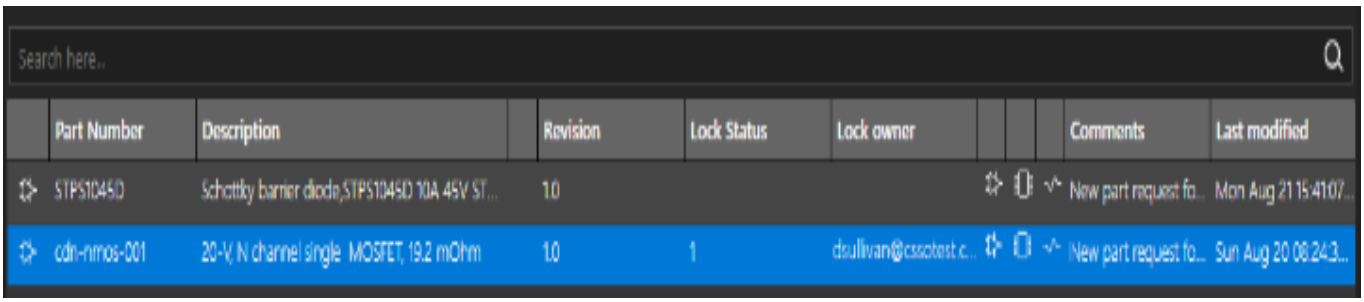


The *Publish Part* dialog box is displayed.



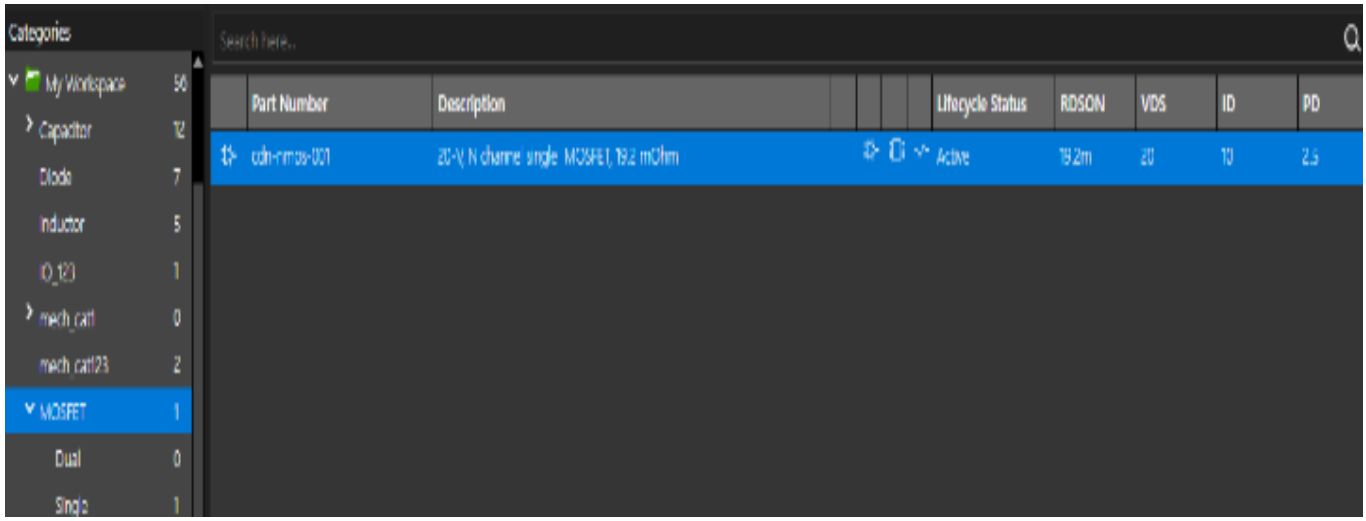
3. Click *Ok*.

   This component is locked in the shared workspace. The lock status is visible to all the team members of this workspace.



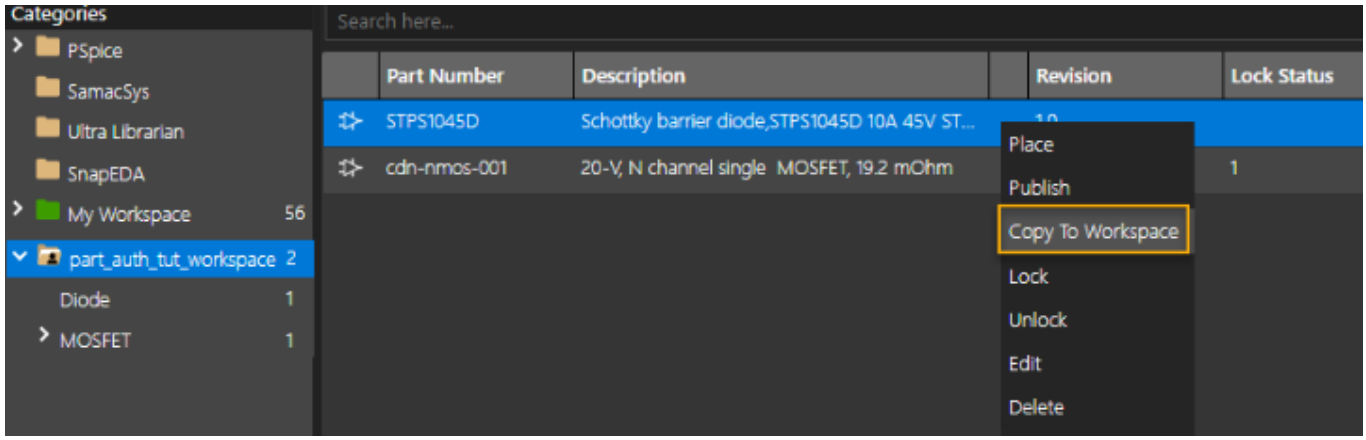
The component is now seen in *My workspace* of the current user. Also, the associated

category (MOSFET) and subcategories (Dual, Single) are also copied.



Similarly, you need to copy the component, *STPS1045D* from the shared workspace to *My Workspace*.

4. In the shared workspace, *part_auth_tut_workspace*, right-click *STPS1045D* and choose *Copy To Workspace.*



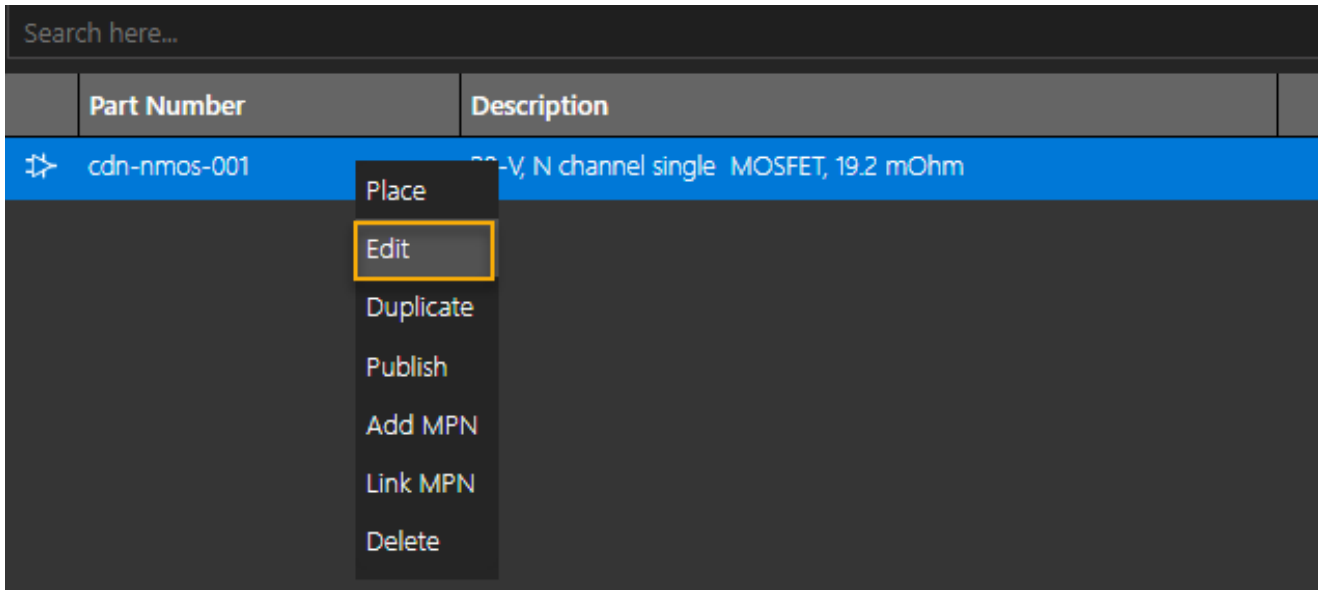
The component is copied to *My Workspace*, but it is not locked in the shared workspace.

Modifying a Component and Publishing it back to Shared Workspace

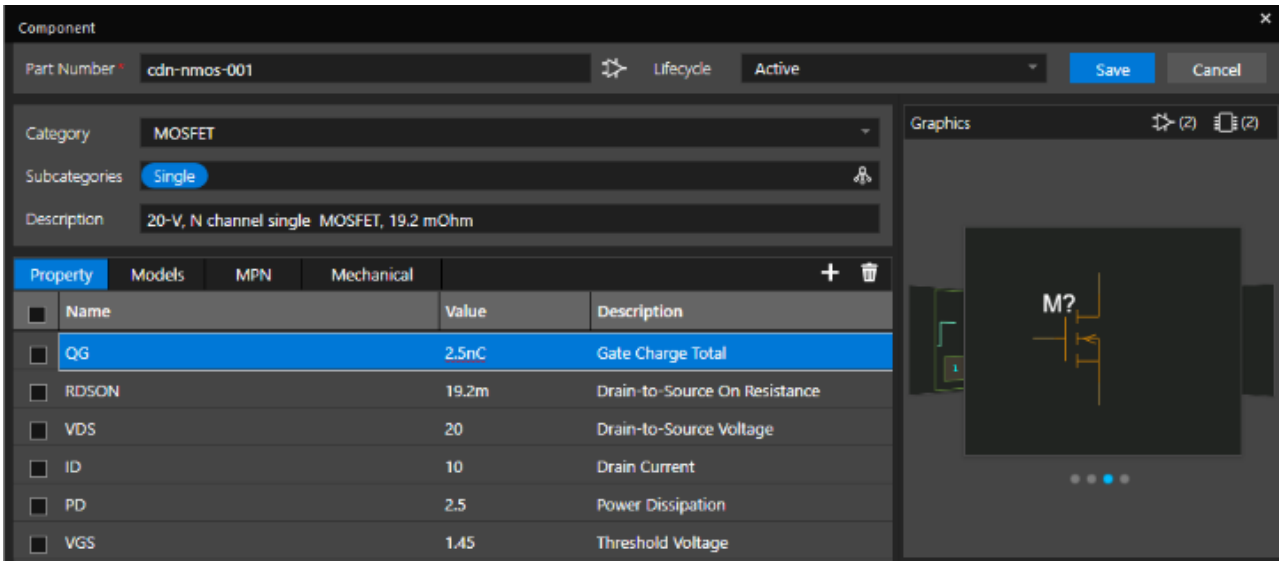
Let's now modify the component, *cdn-nmos-001* in the local workspace of the current user who has locked the component.

To modify a component, do the following:

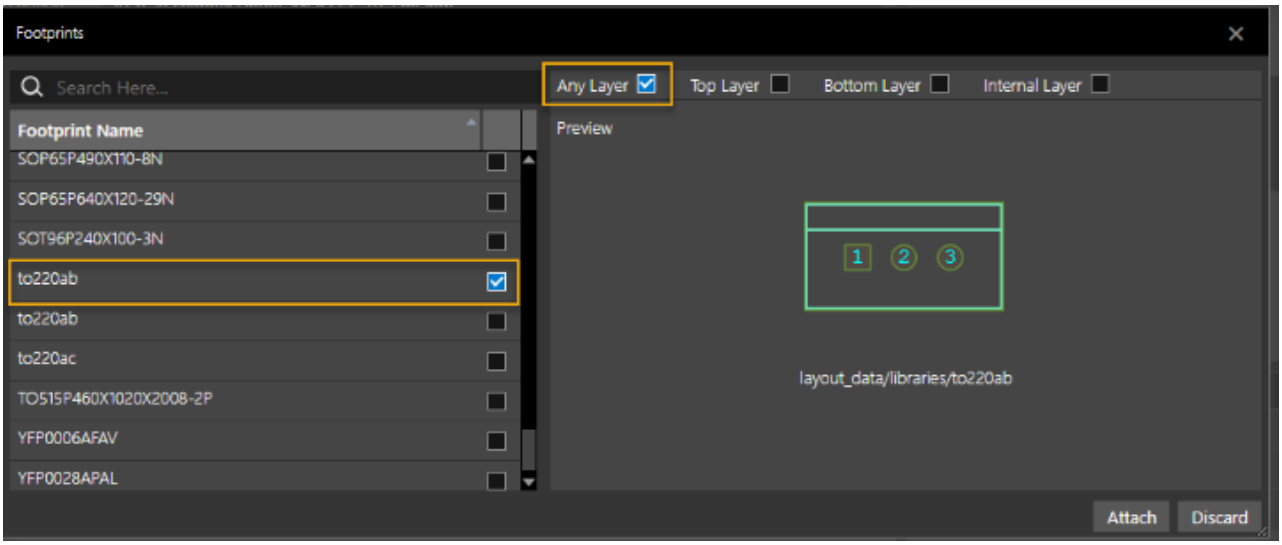
1. In Component Explorer, select the shared workspace, *part_auth_tut_workspace*.
2. In the part browser, right-click the component row for *cdn-nmos-001* and choose *Edit*.

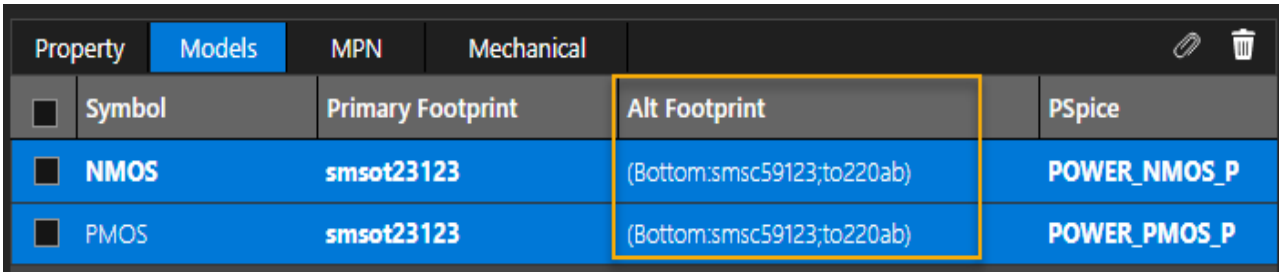


3. In the *Component* dialog box, make the following changes:
   a. Select the Add (⊕) icon to add a new property row with the following details:
      - Name: `QG`
      - Value: `2.5nC`
      - Description: `Gate Charge Total`



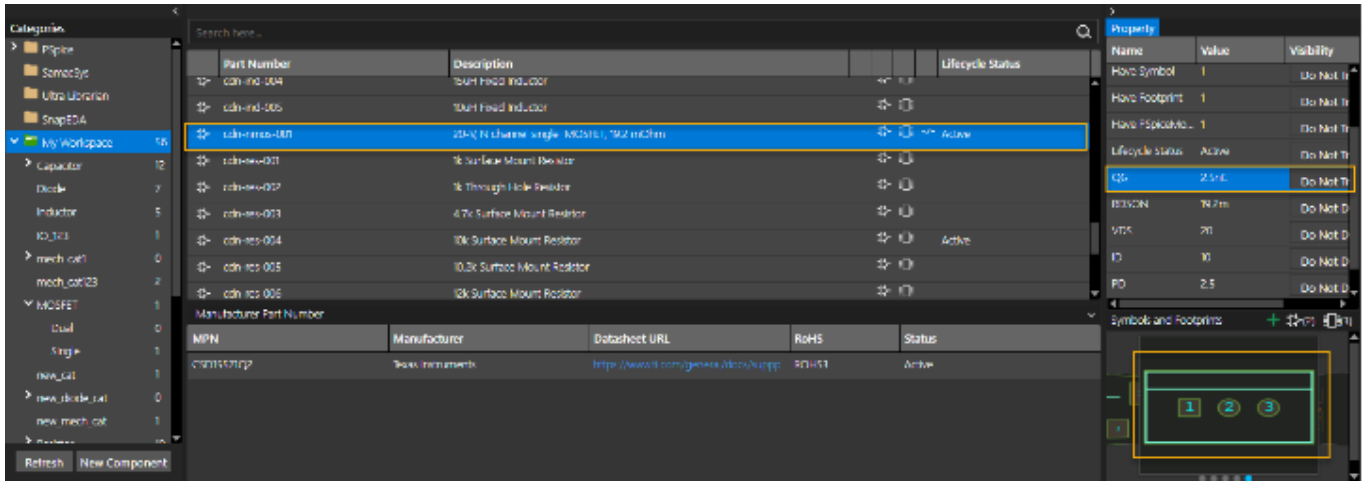
b. Switch to the *Models* tab and add an alternative footprint, *to220ab*, to *Any Layer*.





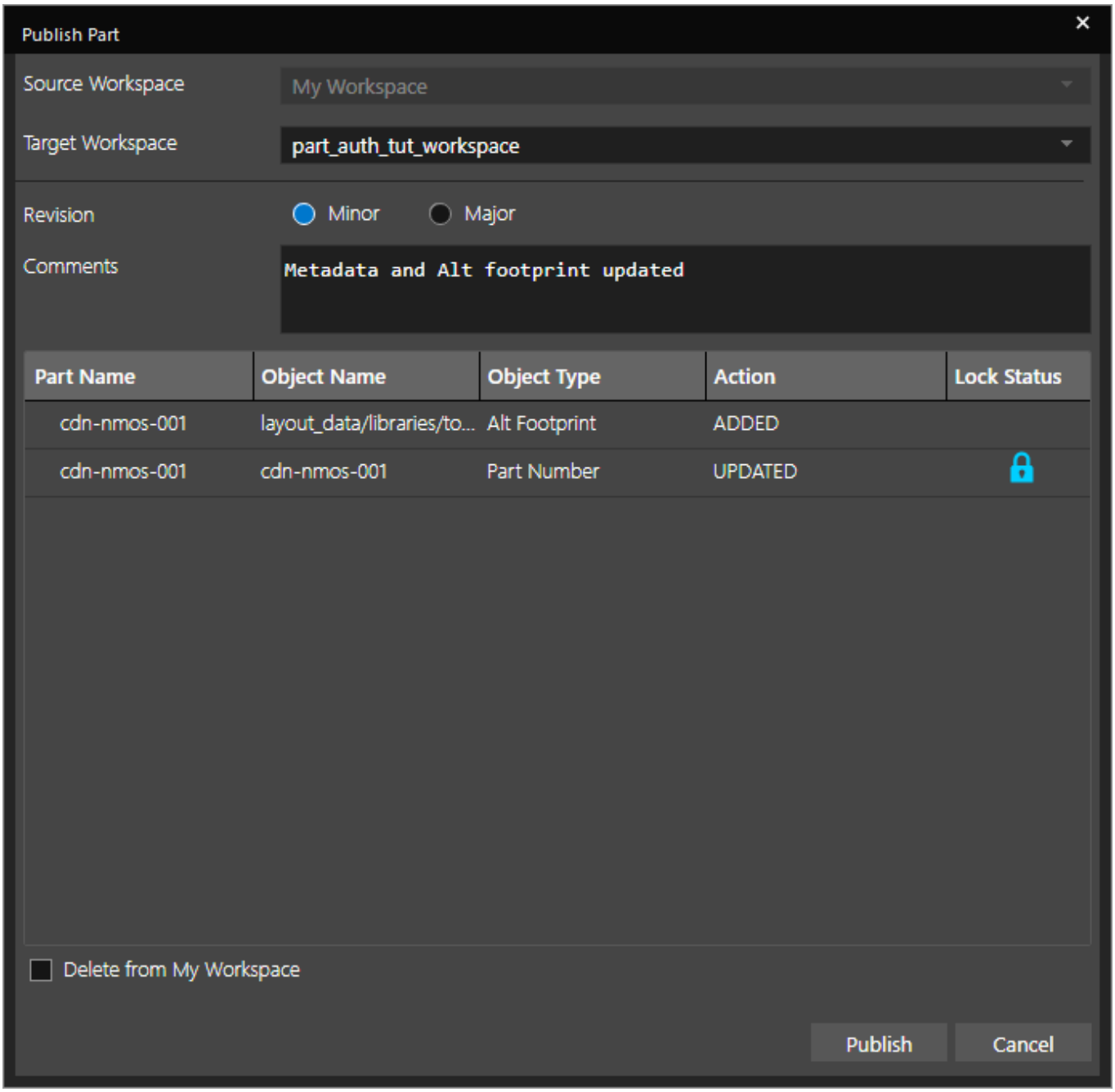
4. Click *Save*.

The component appears with the updated metadata and alternative footprint.



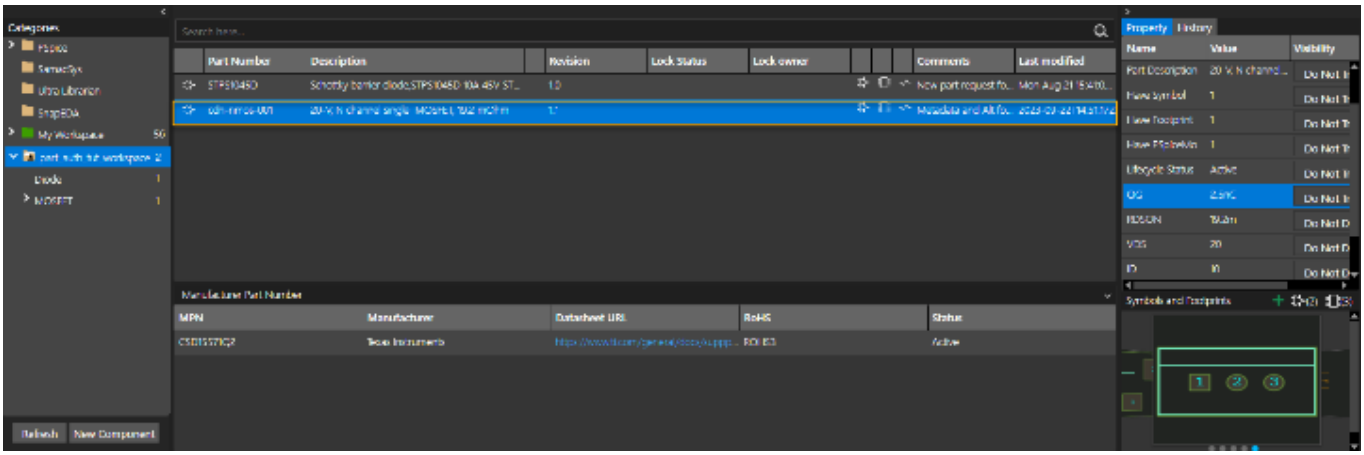
Now let us publish this updated component to the shared workspace,

5. Right-click *cdn-nmos-001* and choose *Publish*.



6. Click *Publish* to publish the component to the shared workspace.

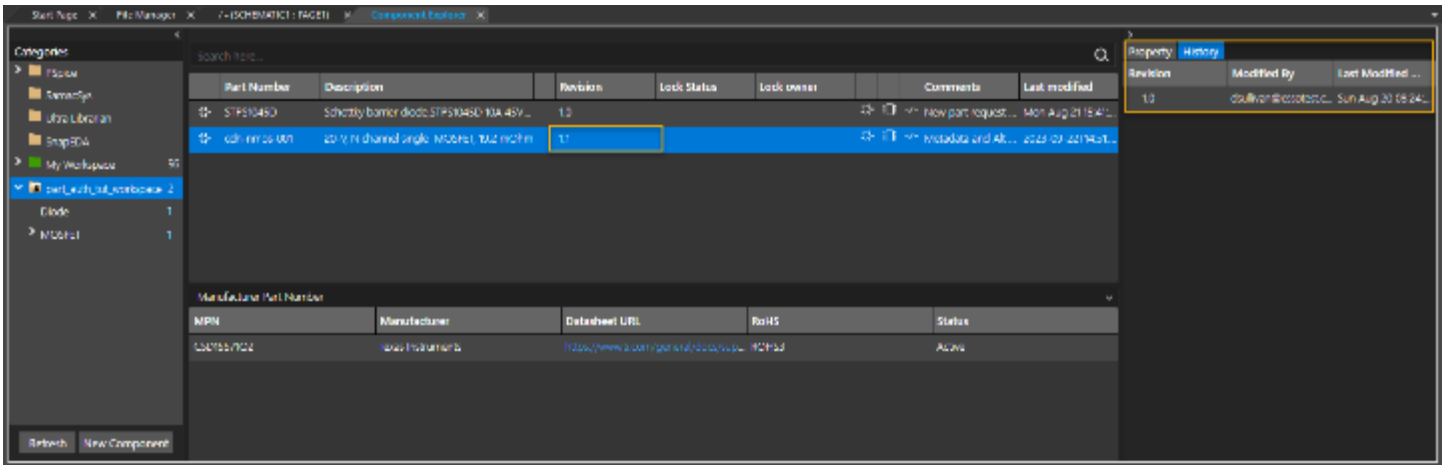
   The updated component with updated revision (`1.1`) is seen in the shared workspace. Also, notice that the lock is released.



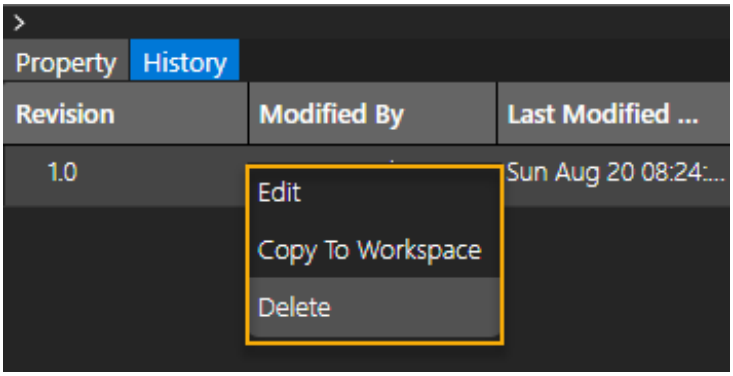
Restoring Older Versions of Components in Shared Workspaces

From the shared workspace, you can view and restore the older revisions of shared components with multiple revisions in the *History* section.

To restore an older version of a component, select the *History* tab from the right-pane of Component Explorer.



The History section shows the following details.

- Revision
- Modified By (user/team members ID)
- Last Modified Date.

You can restore or roll back to a previous version of the component using the *Edit* command from the shortcut menu.