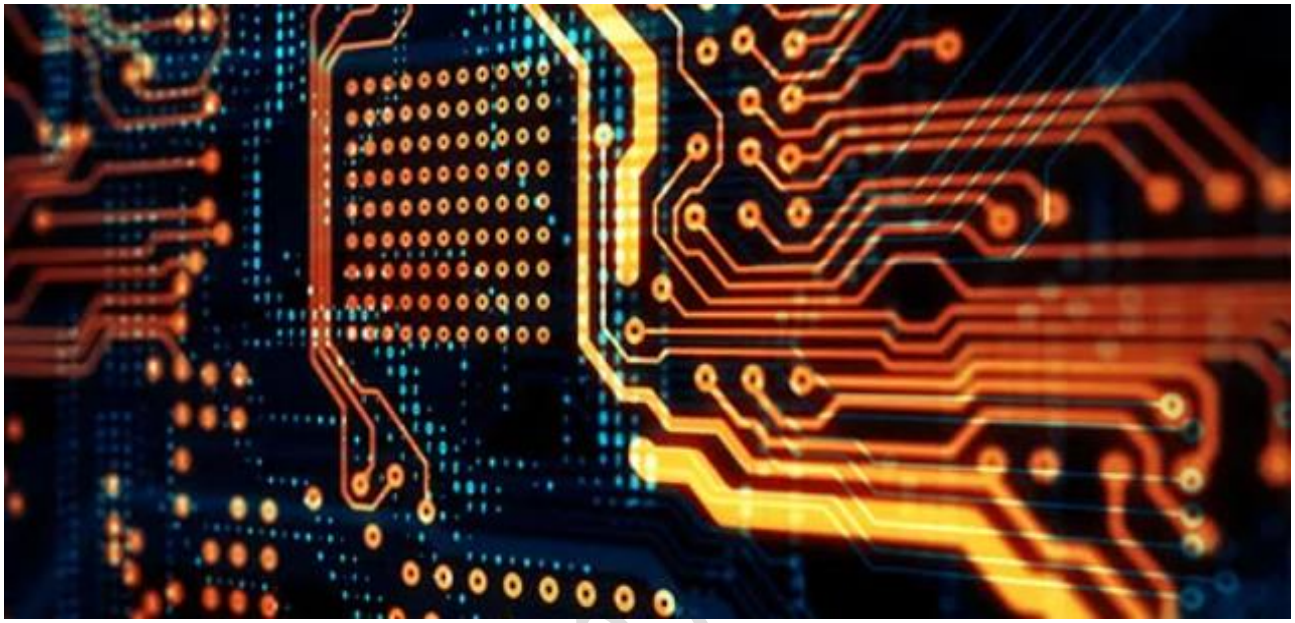


OrCAD Capture CIS/CIP Usage Training

Version 23.1

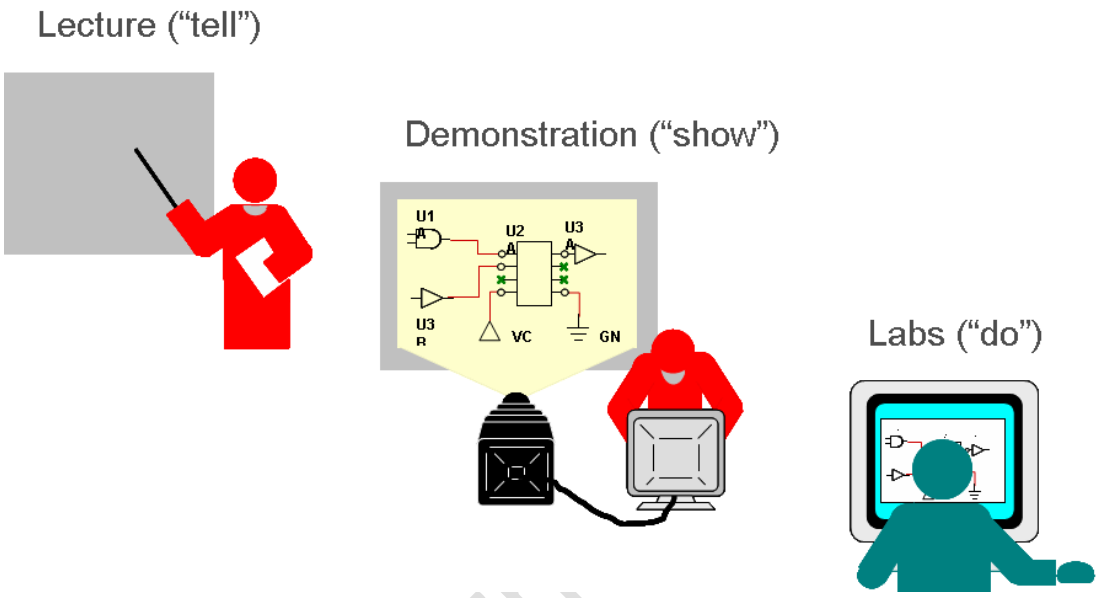
EMA Education Services

Table of Contents

***This training manual was written using OrCAD Capture CIS version 23.1-2023-P001 in February of 2024. All references will refer to that version and may change in later versions or hotfixes of the software.

Introduction

This document is designed to accompany formal training for OrCAD Capture's CIS and EMA Design Automation's Component Information Portal (CIP). Usage methodology and best practices are covered. Concepts and skills taught during this course are based on an "explain," "show," and "hands-on" method.



Each lesson begins with an explanation of application features and associated dialog boxes. Lab exercises follow that provide guided hands-on experience.

Course Agenda

Lesson 1: Component Information Portal (CIP) Overview
Lesson 2: OrCAD Capture CIS Fundamentals
Lesson 3: OrCAD Capture CIS Part Manager
Lesson 4: Finalizing and Documenting the Design

Related EMA Courses

All classes offered by EMA can be viewed at:

https://www.ema-eda.com/learning/classes

You will see a list of courses that are offered in a variety of delivery modes – Classroom-based, Live-Online, and E-Learning. Custom classes, tailored to your specific company needs, can also be requested.

Formatting Conventions

The following formatting conventions are used throughout this training manual:

- When lab procedures instruct you to click a dialog box button, tab, option, or toolbar icon, the item is formatted in ***bold, italic*** text.
- When lab procedures instruct you to select a file name, folder, or schematic page in either the OrCAD Capture Project Manager window or a directory tree, the name of the file, folder, or page is formatted in `courier` text.
- When lab procedures instruct you to access a directory path, the path is formatted in `courier` text. When you are instructed to select a menu option, the option is formatted in ***bold, italic*** text.
- When you are instructed to select a series of menu options, the primary and secondary menu options are separated by the ***(>)*** symbol.
- When lab procedures instruct you to press a key on your PC keyboard, the name of the key is enclosed in brackets. For example: ***<key>***.
- When lab procedures instruct you to use ***<LMB>*** you are expected to press and hold the ***<Ctrl>*** key on your keyboard while you click design objects using your left mouse button. This technique if often used to select multiple design objects.
- Throughout this manual ***(LMB)*** refers to your "**L**eft-**M**ouse- **B**utton". The instructor may also refer to this as the "***Control, Select"*** method of choosing objects.
- Throughout this manual ***(RMB)*** refers to your "**R**ight-**M**ouse- **B**utton".
- When lab procedures instruct you to select a design name or object, the name or object is formatted in `courier` text.
-

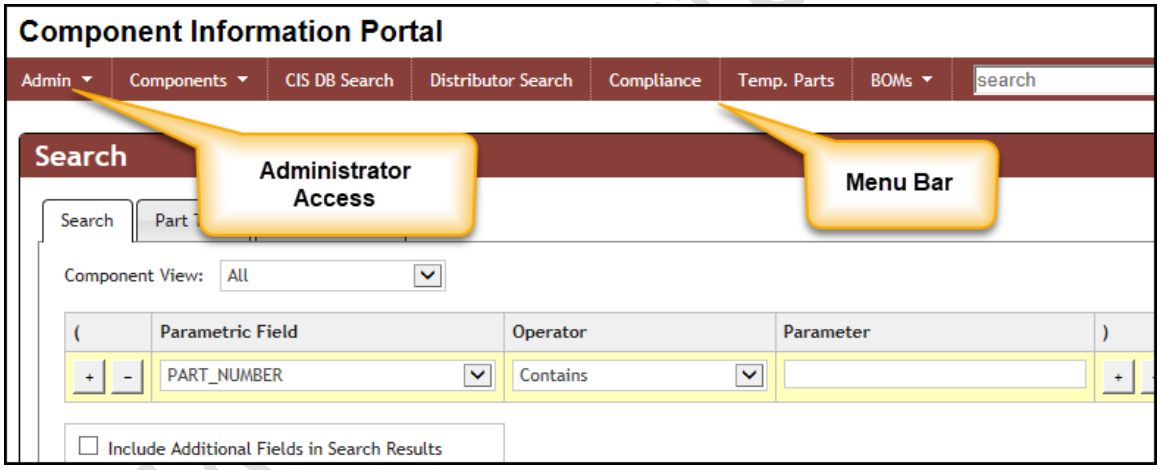
Lesson 1: CIP Overview

Objectives

By the time you are finished with this lesson you will be able to:

- Log into CIP
- Perform CIS database searches within CIP
- Use the Distributor Search tab to search for distributor parts
- Use CIP to add new parts to the database
- Track part history
- Add mechanical parts and associate them with existing database parts

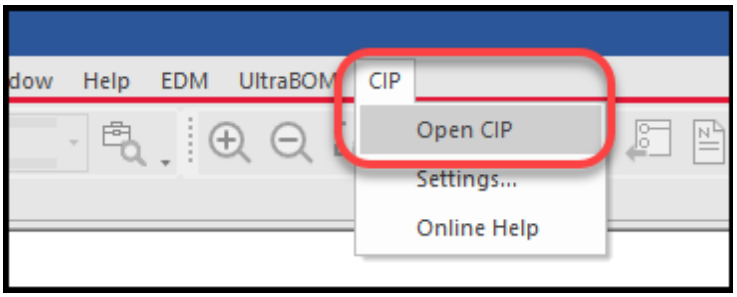
CIP provides a powerful interface for entering, deleting, or modifying data. Users can search for parts within CIP as well as generate temporary parts. CIP facilitates the process of converting, or promoting, temporary parts to formal parts.



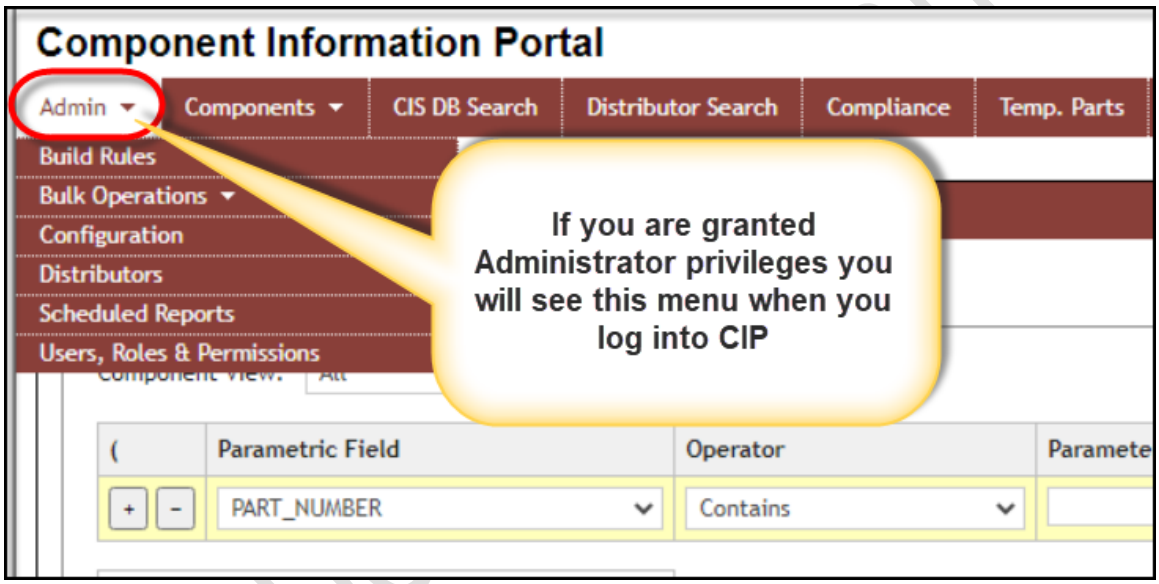
Logging in to CIP

Component Information Portal (CIP) can be accessed in two ways – either from an external web browser or from within the OrCAD Capture environment. In this class, CIP will be accessed from within the OrCAD environment. The CIP install program enables this capability during installation and is accessible after downloading the CIP client. This information is covered during the installation process.

To access CIP from within the OrCAD environment, select ***CIP > Open CIP***.

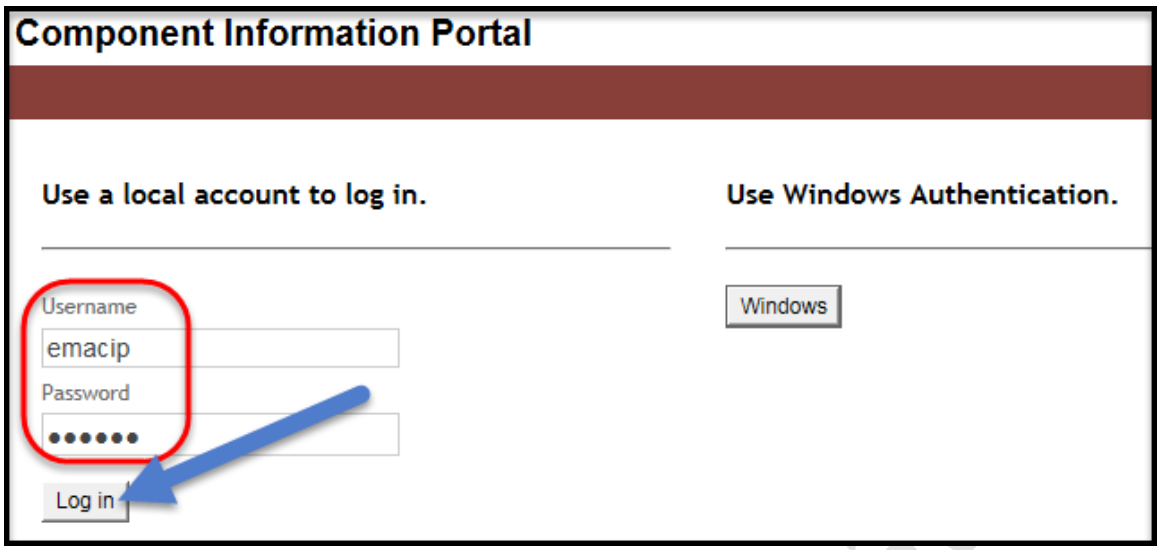


The login you use will determine the privileges you have while working in CIP. This is defined by an Administrator. If you are granted administrator privileges, you will see the **Admin** menu across the top of the CIP menu bar, which contains features that grant logins and roles, and defines CIP preferences.



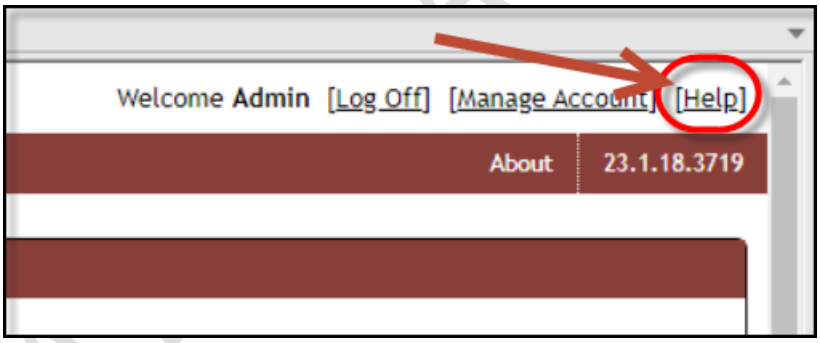
For this training course you will be logging in as a user, and you will be able to add parts, edit parts, and delete TMP parts. You will not see the Admin menu across the top menu bar. The login is:

Username: **emacip**
Password: **Emacip_01**

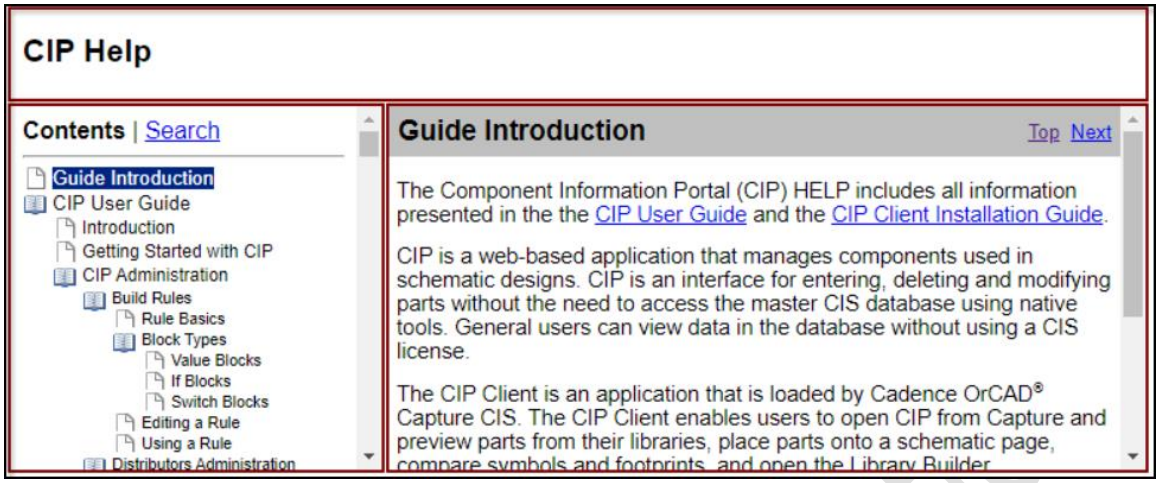


Using Help

Once you login, the main **Help** link will be located at the top right area of the CIP window. Here you also see the login name, as well as links to log off and change password.

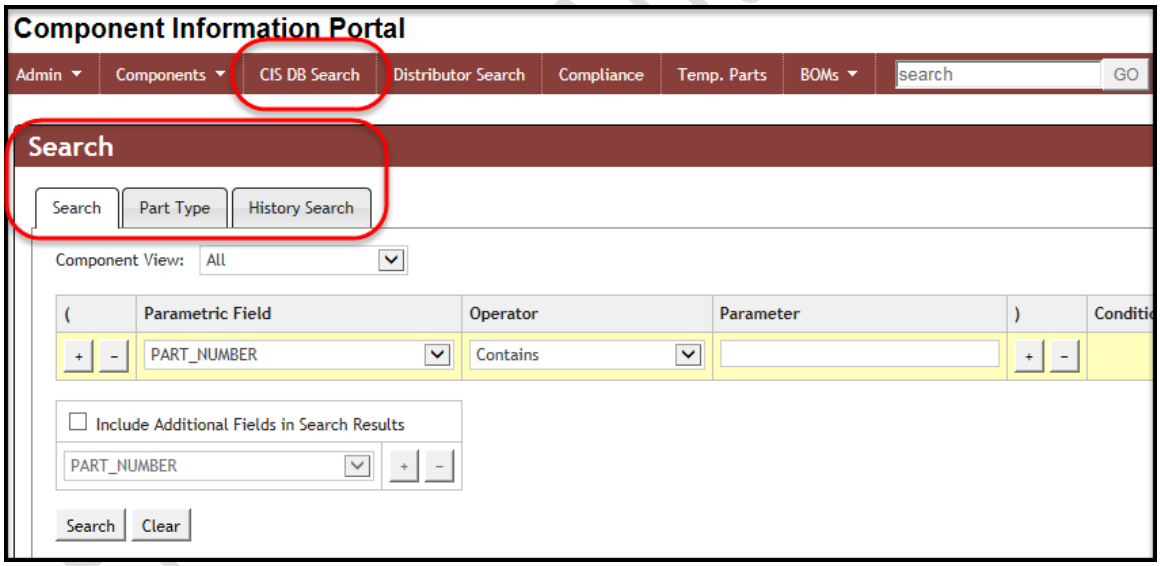


When the Help Doc opens you will be able to browse or search through the help documents.



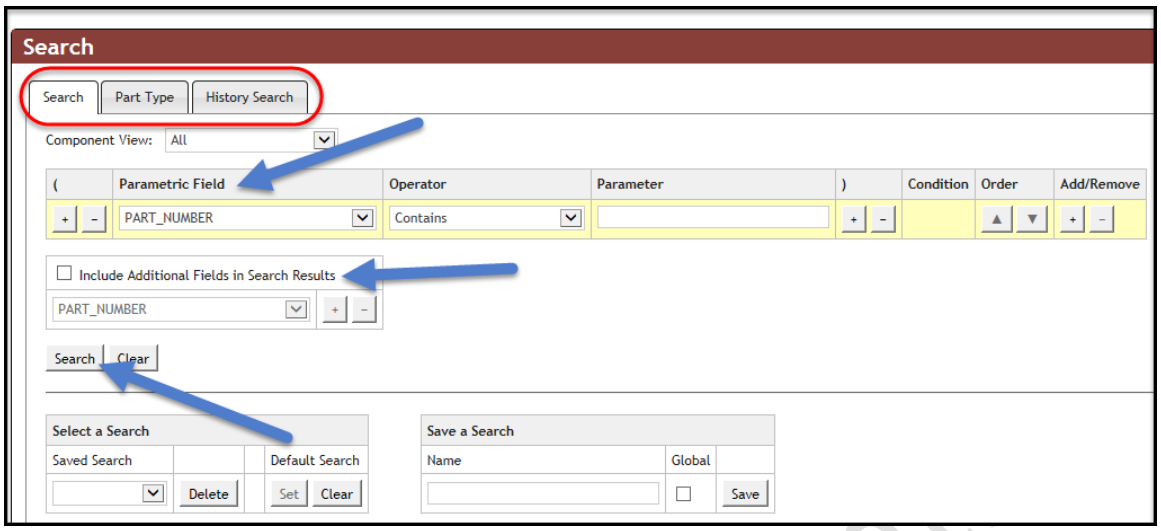
Using CIS DB Search

Part searches in the local database can be performed in the CIS DB Search. Simple searches can be performed, or searches can be customized to look for parts with specific parametrics (PART_NUMBER, VALUE, TOLERANCE, etc.).



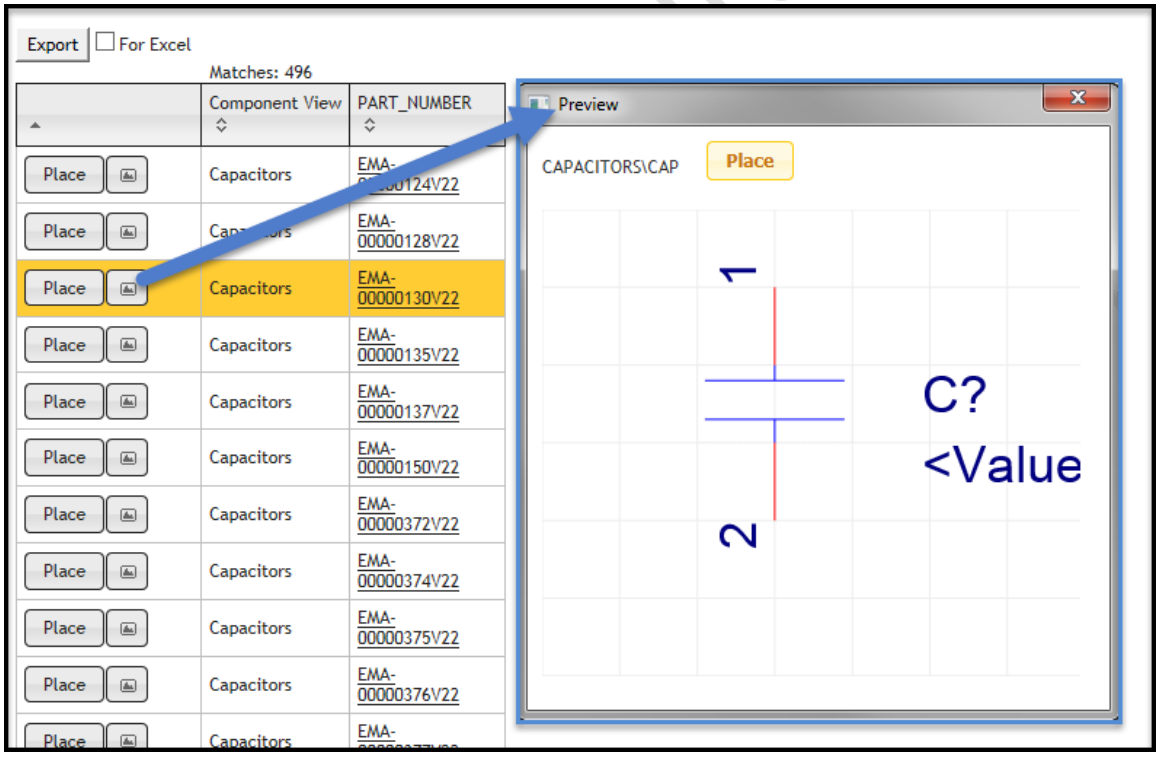
Performing a CIS DB Search

If more refinement is needed, parts can be searched using parametric data. In a more refined search, you can choose the table, parametric fields, operator, parameter, and condition. Additional fields can be added to the search results. For example, a search can be created that will return specific results but will always show the requested additional fields in the results.

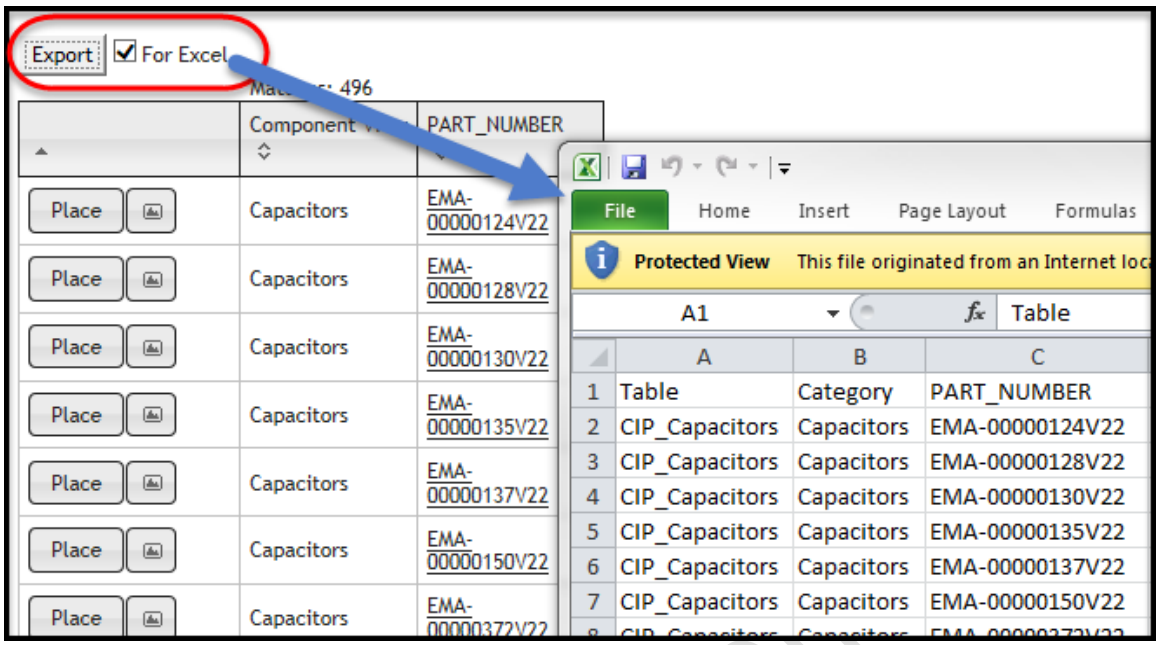


Reviewing Database Search Results

Symbols can be previewed and/or placed directly from the search.

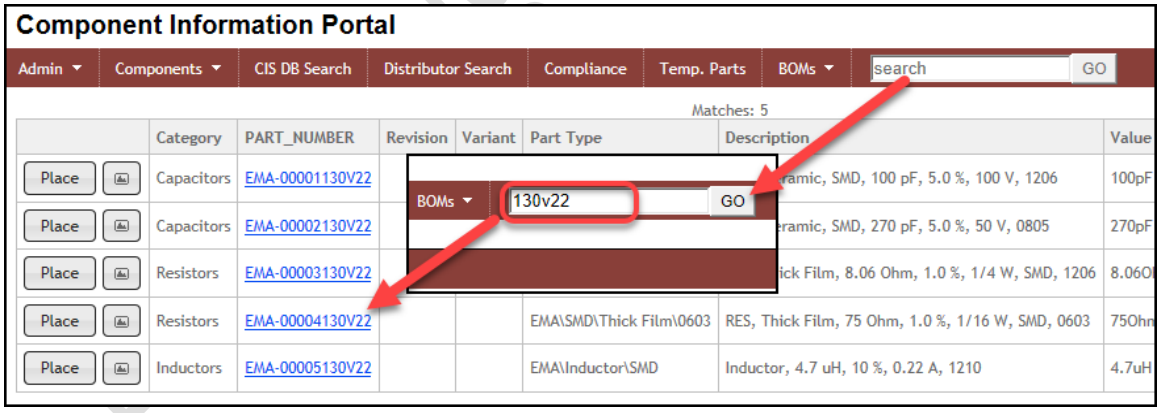


Clicking on a column header will order the search results. These results can be exported into Excel using the ***Export*** button.



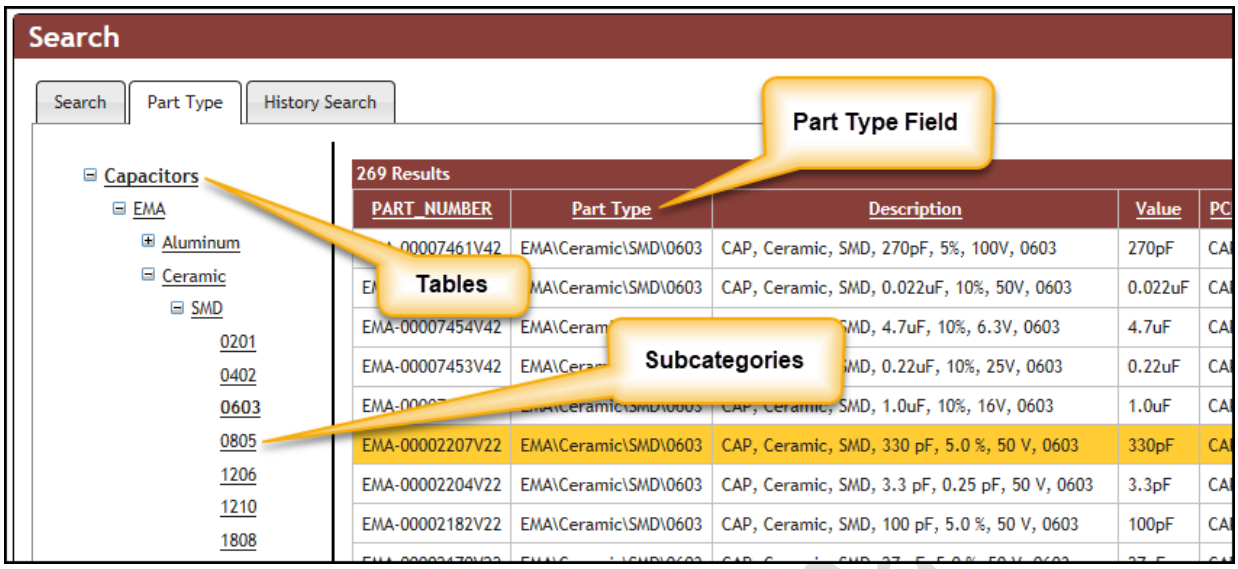
Using Quick Search

A quick search can be run by entering search data in the **Search** box on the Menu bar, then clicking the ***GO*** button to initiate the search. Quick search queries the PART_NUMBER, Part Type, Description, Value, PCB Footprint, and Schematic Part fields to match the search string entered.



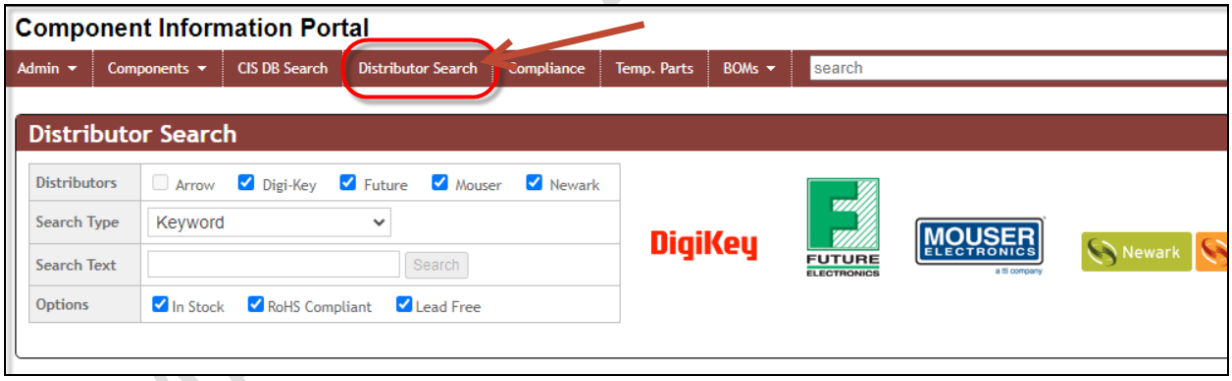
Using the Part Type Search

The Part Type specifies where a part resides in the database. Users with administrator privileges can create Part Types. The **Part Type** tab allows you to search by expanding tables and subcategories.



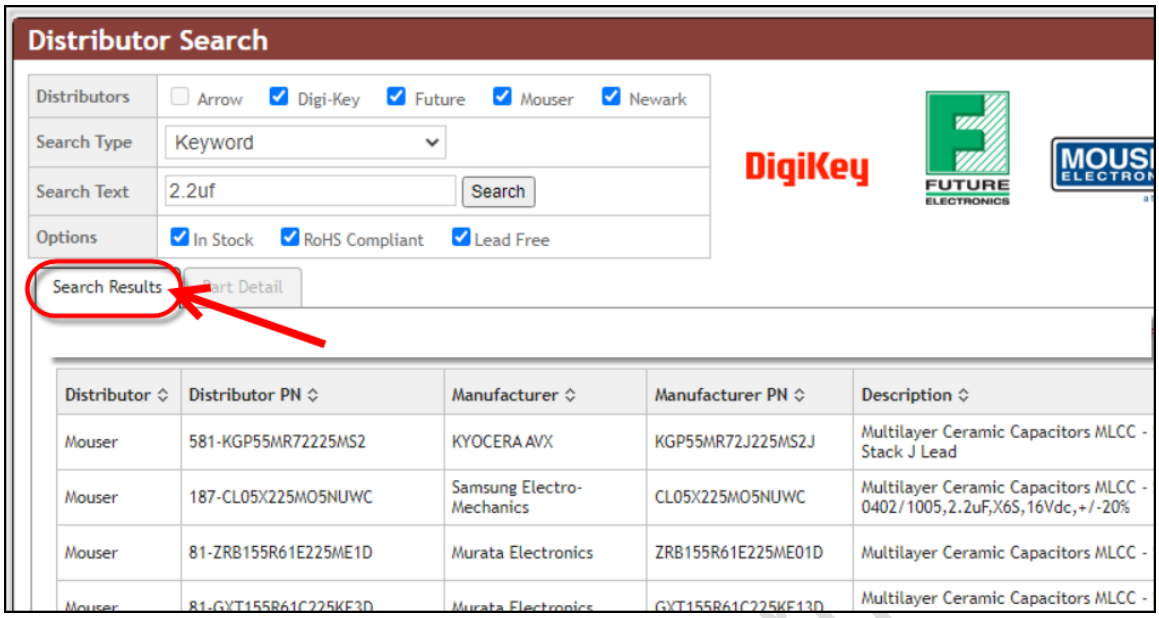
Using the Distributor Search Tab

The **Distributor Search** tab provides a way to directly search for parts that are available from Newark, Arrow, Future, Mouser, and Digi-Key. Options are available to search for parts that are **In Stock**, **RoHS Compliant**, and **Lead Free**.

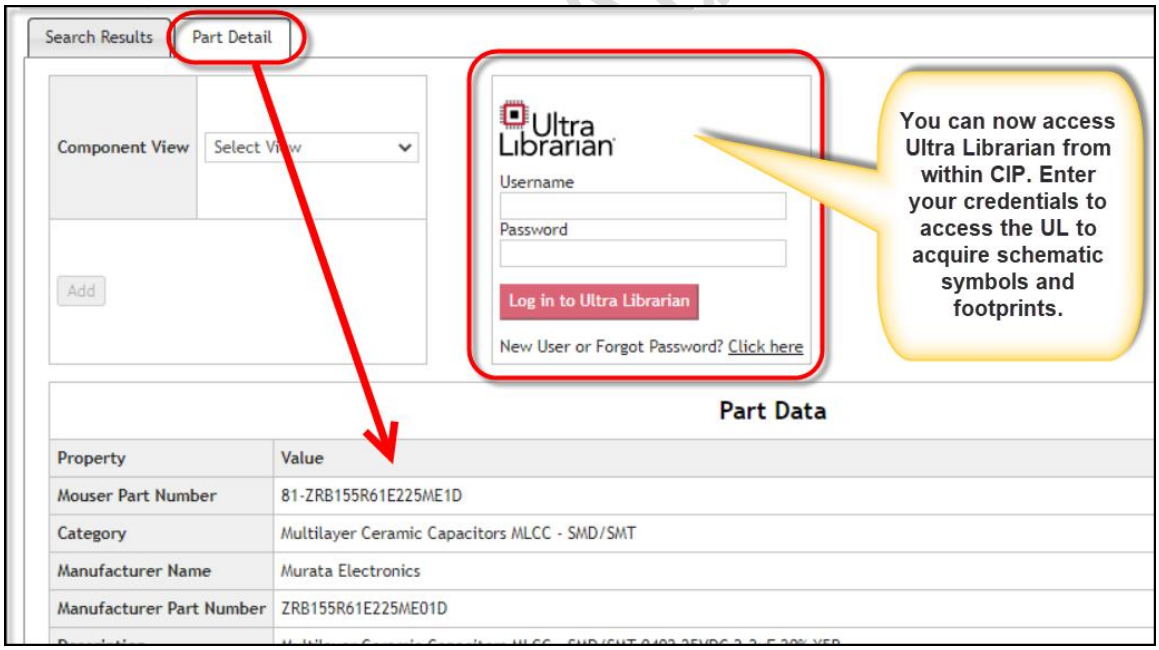


Distributor parts can be searched by **Keyword** or **Manufacturer Part Number**. Results that match the entered text will be returned.

Search results contain a list of relevant parts with distributor part numbers, manufacturer part numbers, part descriptions, category, and quantity on hand.

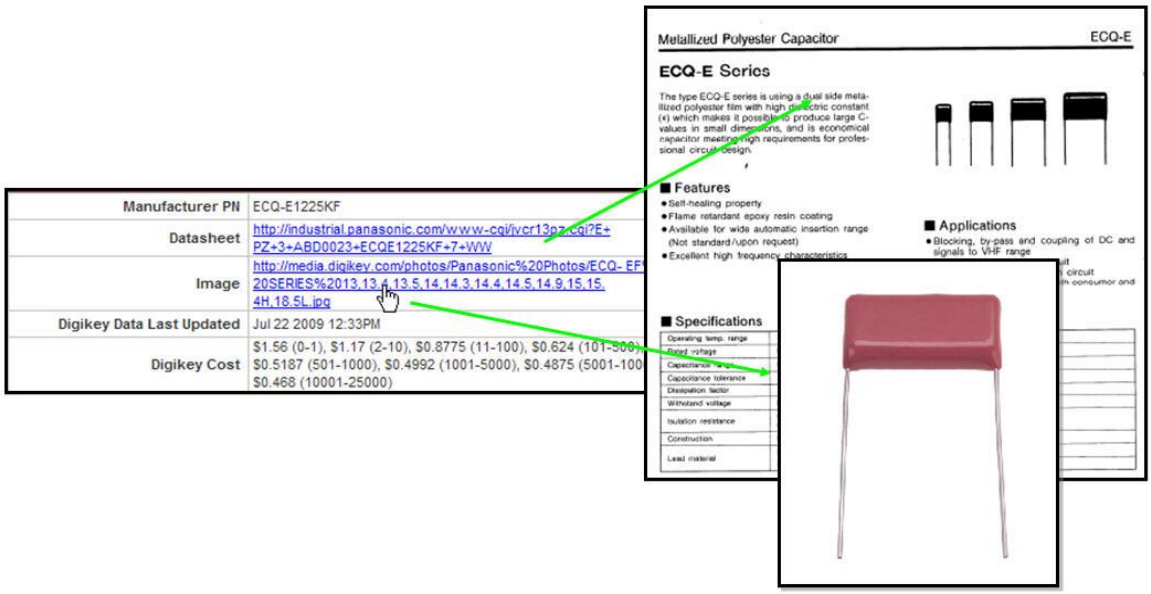


When a part is selected from the list, part details will be displayed in the ***Part Detail*** tab. If you have a subscription to **Ultra Librarian** you can also access the UL parts to find and associate a schematic symbol and footprint to your chosen part.



Distributor Links

When a TMP part is created, links are generated for the datasheet and part image.



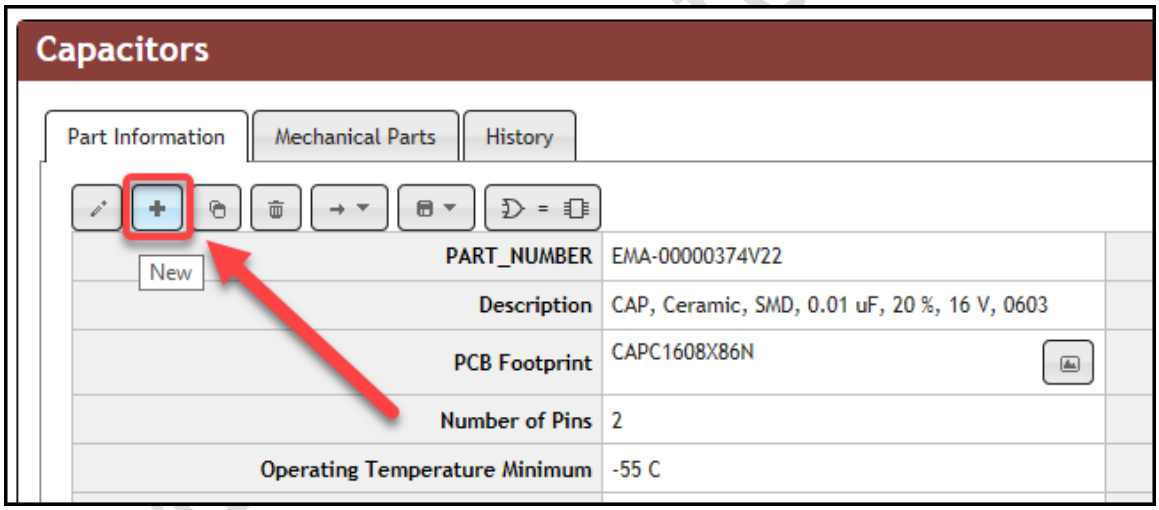
Creating New Parts in CIP

There are several ways to generate new parts in CIP:

- Generate a blank part and fill in the appropriate fields.
- Copy an existing part, then update the fields that differ in the new part. This method automatically generates the part with a TMP part number.
- Use the Distributor Search to locate the part from a distributor, then add it to the database. When it is added, the parametric data will be auto populated into the corresponding fields and the part will receive a TMP part number.

New Blank Part

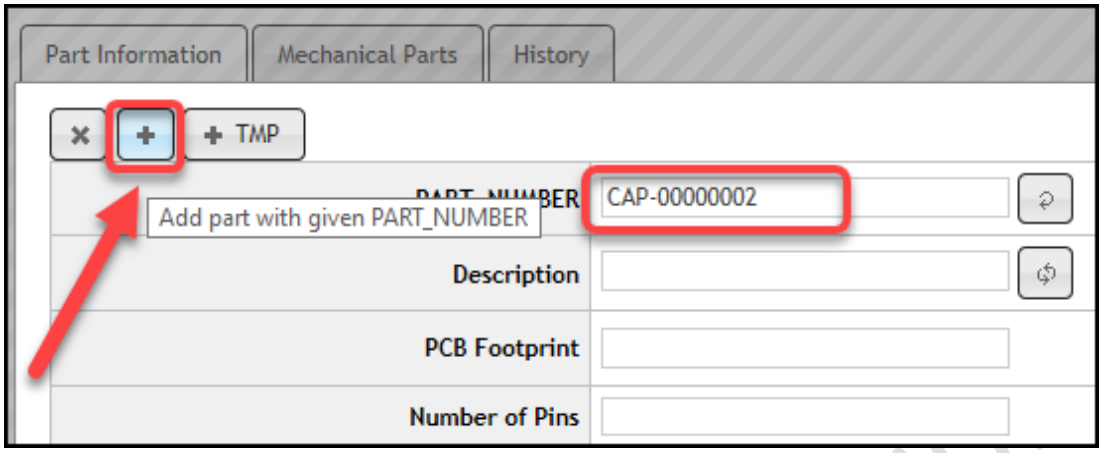
To create a new part from an existing part, select the ***New*** button.



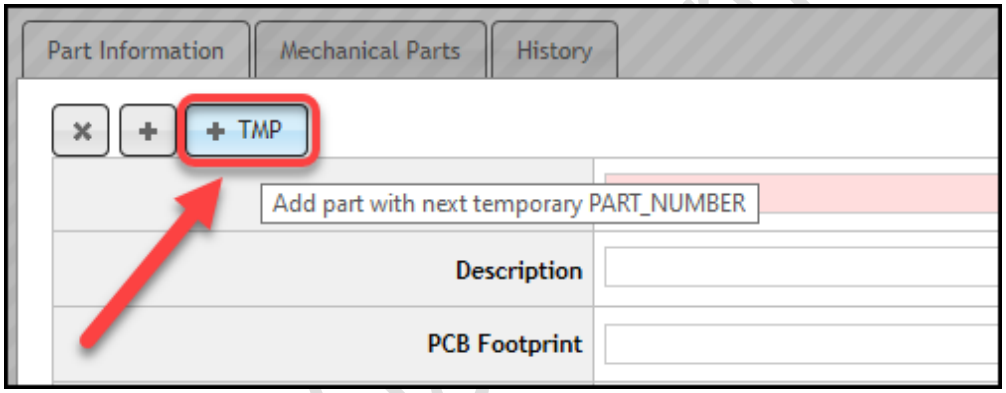
This opens an editing page where all fields are blank. Data can be entered for the new part, including a new part number.

Part numbers can be added using one of the methods below:

- Enter a preferred part number in the PART_NUMBER field:

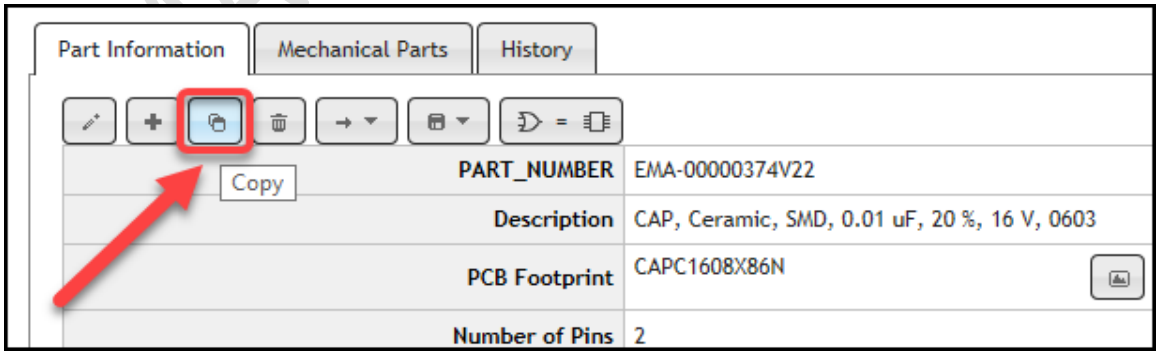


- Add the part with a TMP part number. Clicking the ***Add part with next temporary PART_NUMBER*** button will assign the next available TMP part number in the database.



Copy an Existing Part

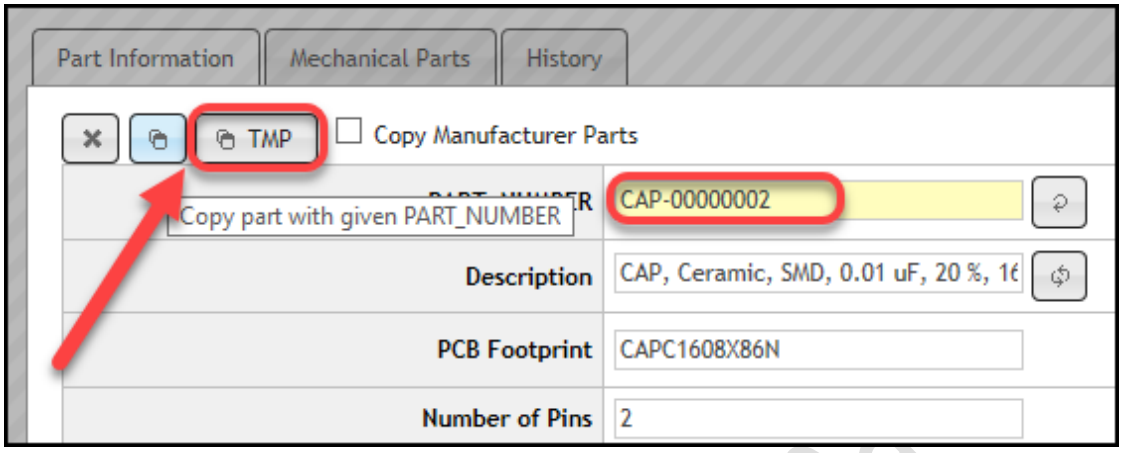
To create a new part by copying an existing part, select the ***Copy Part*** button.



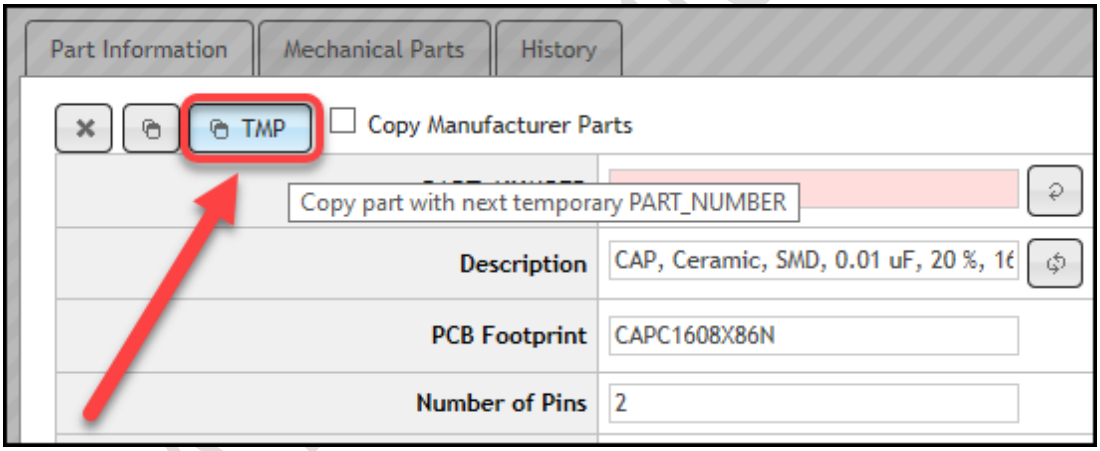
This opens an editing page where all fields are blank. Data can be entered for the new part, including a new part number.

Part numbers can be added using one of the methods below:

- Enter a preferred part number in the PART_NUMBER field:

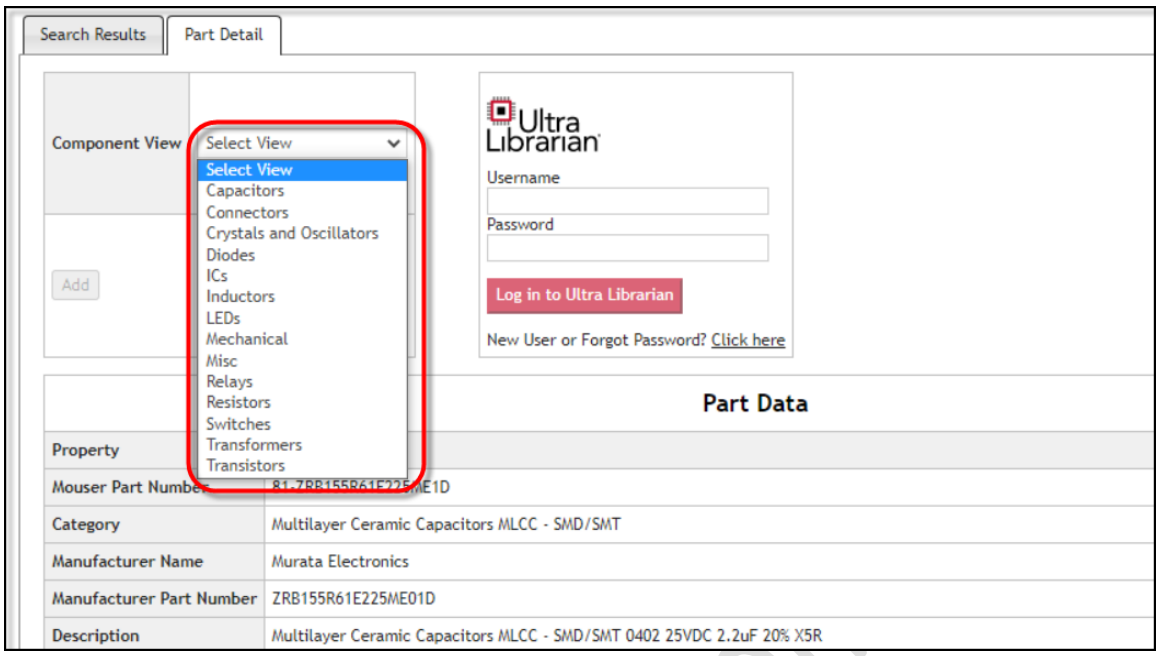


- Add the part with a TMP part number. Clicking the ***Add part with next temporary PART_NUMBER*** button will assign the next available TMP part number in the database.

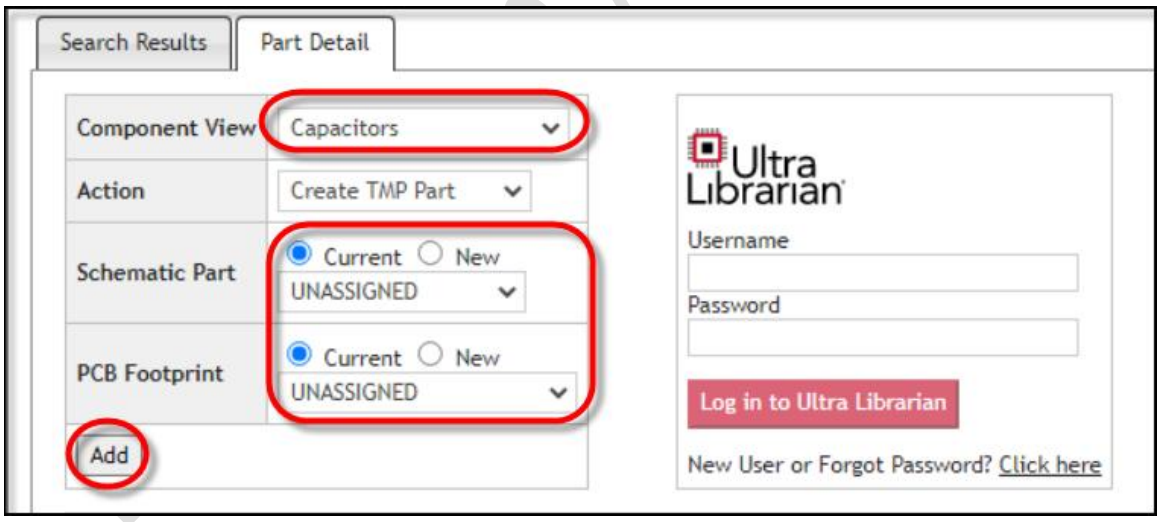


Create a TMP Part from a Distributor Search Part

There are a few steps involved in generating a temporary part from a distributor. Once a part is selected, it will need to be assigned to one of the categories in the database. From the **Component View** dropdown, select the CIS database table where the part will be stored.

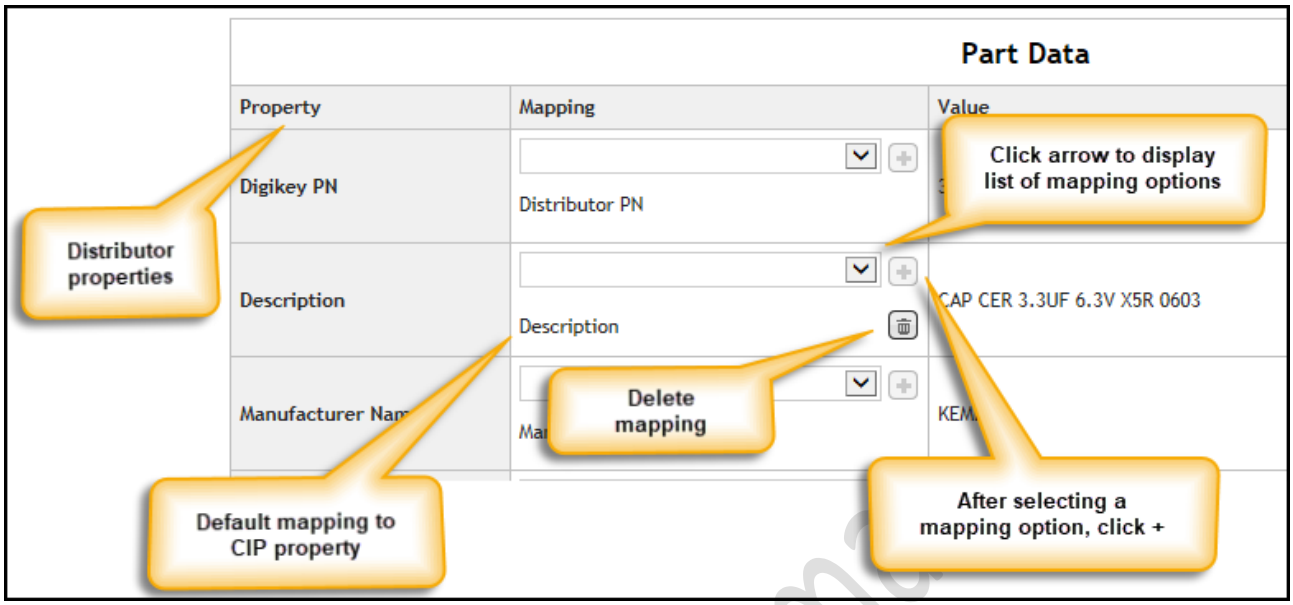


Once a table is selected, the part can be associated with a schematic symbol and a PCB footprint. The dropdown lists allow the selection of schematic symbols and PCB footprints that are already associated with other parts in the same table. Select the preferred schematic symbol and PCB footprint from the dropdown lists.

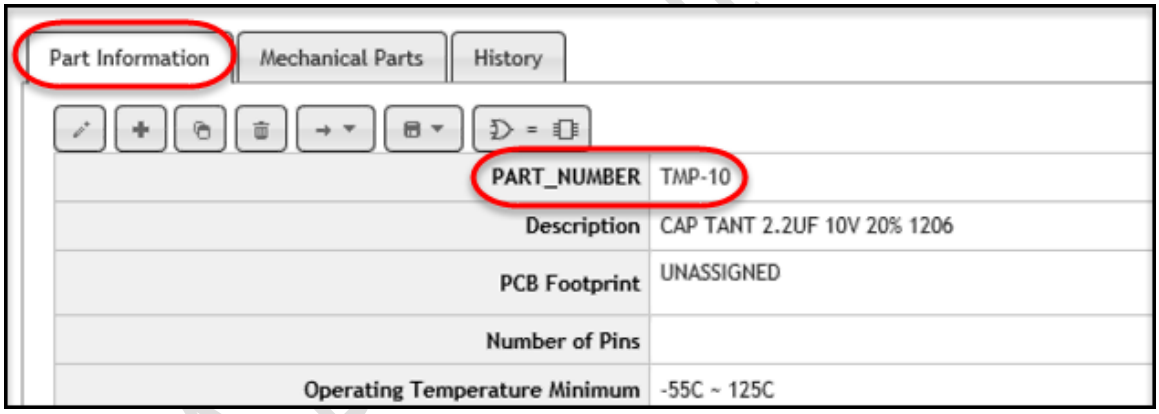


Alternately, these can be left unassigned and can be assigned later.

To change the property mapping from the suggested default mapping, or if a distributor property needs to be mapped to more than one CIP property, change the mapping configuration before creating the TMP part. After selecting a mapping option, click the **Add Mapping (+)** button to add the property mapping.

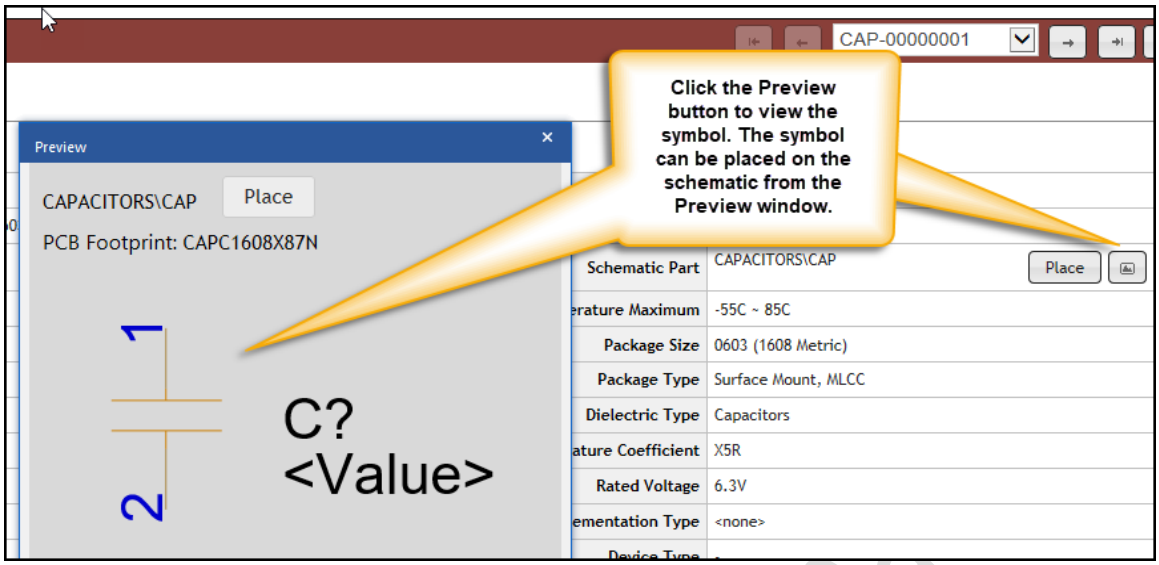


Click ***Add*** to create the new TMP part with the specified part data. This TMP part is added to the CIS database and is available for placement in the schematic.



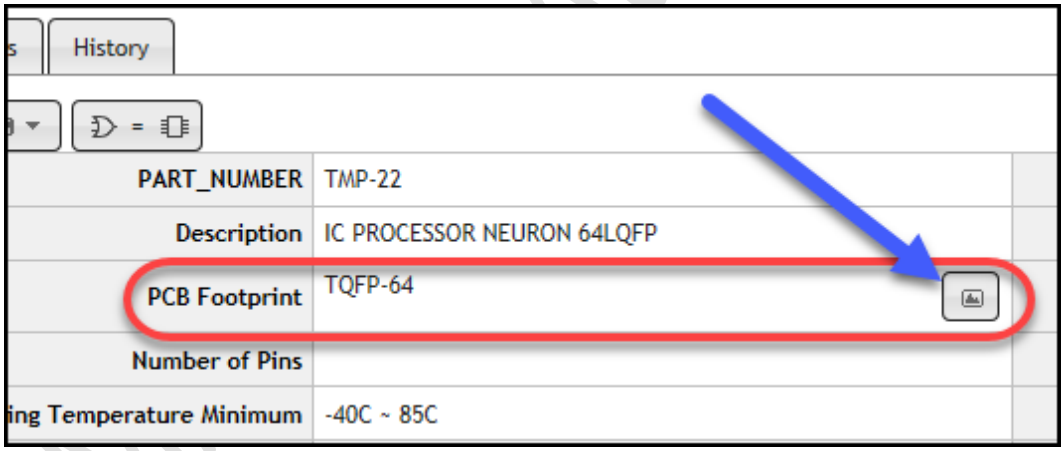
Preview the Schematic Symbol

The schematic symbol can be previewed and placed on a schematic page directly from CIP in the **Schematic Part** section of the **Part Information** tab. There are two options, one for placing the part in the schematic, and the other for previewing the part prior to placement.



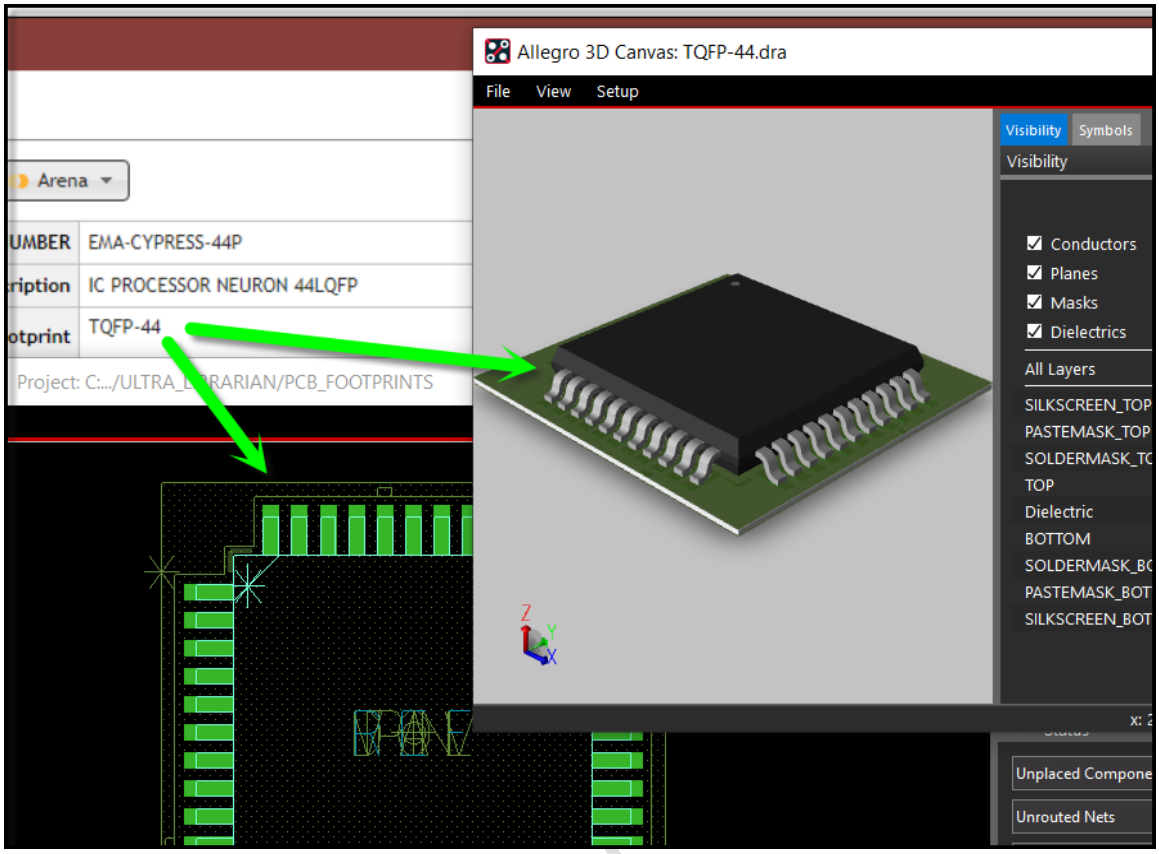
Preview the PCB Footprint

If the footprint symbol can be found in the defined library path, the PCB footprint can be viewed from the Part Information tab.



This path is defined in
`%HOME%\cdssetup\OrCAD_Capture\<version>\Capture.ini`:

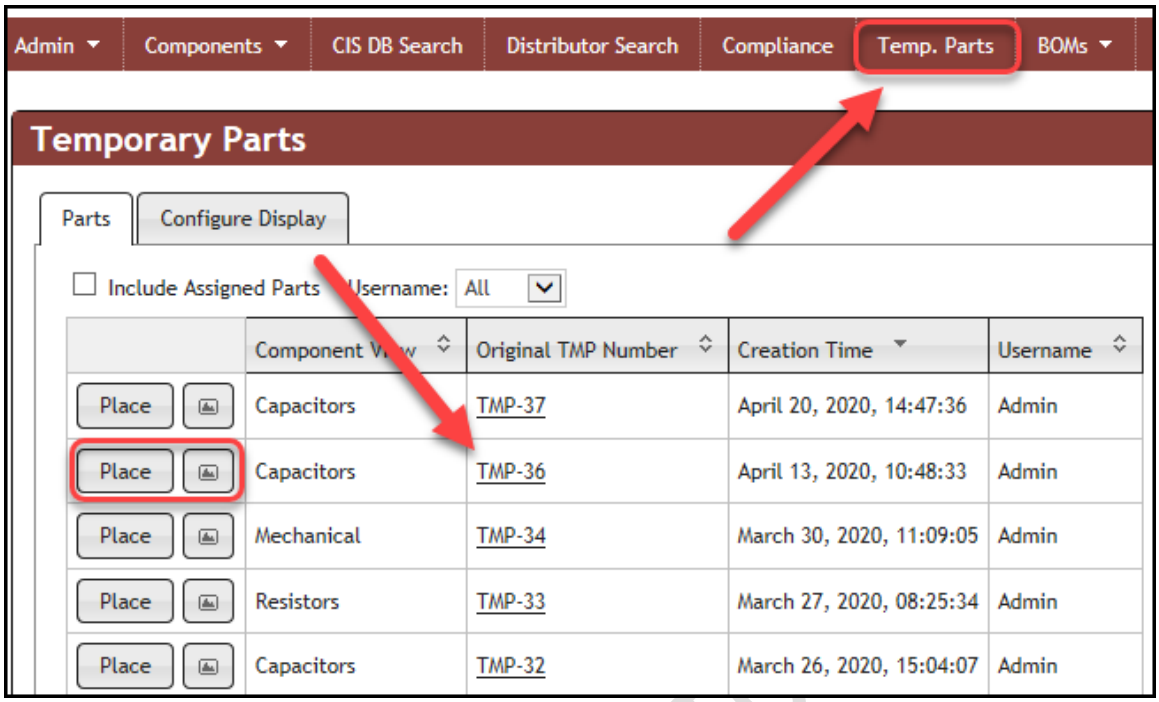
```
[Allegro Footprints]
dir0=<path to directory with Allegro footprints>
```

If a STEP Model has been generated and associated with the PCB Footprint, it can be viewed in the 3D Canvas.

Using the Temp Parts Tab

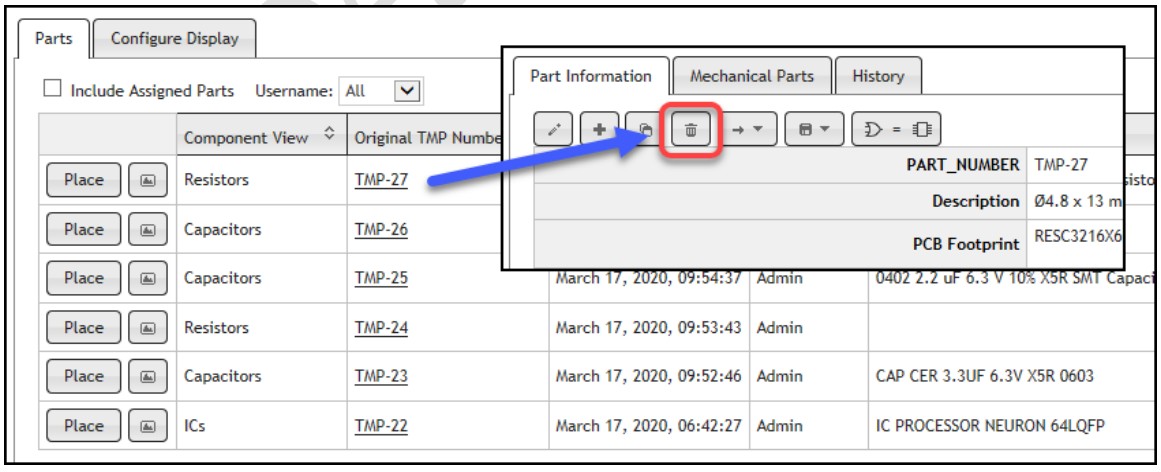
The **Temp Parts** tab contains a list of temporary parts that have been added to the database. These parts do not have formal part numbers assigned to them. The list can be sorted in ascending or descending order by clicking on the column title. Additionally, the schematic symbol can be previewed and placed from the Search results.



Configuring the TMP Parts Display

The properties that are displayed for the TMP parts can be configured in the **Configure Display** tab. This is useful when only specific field data needs to be shown.

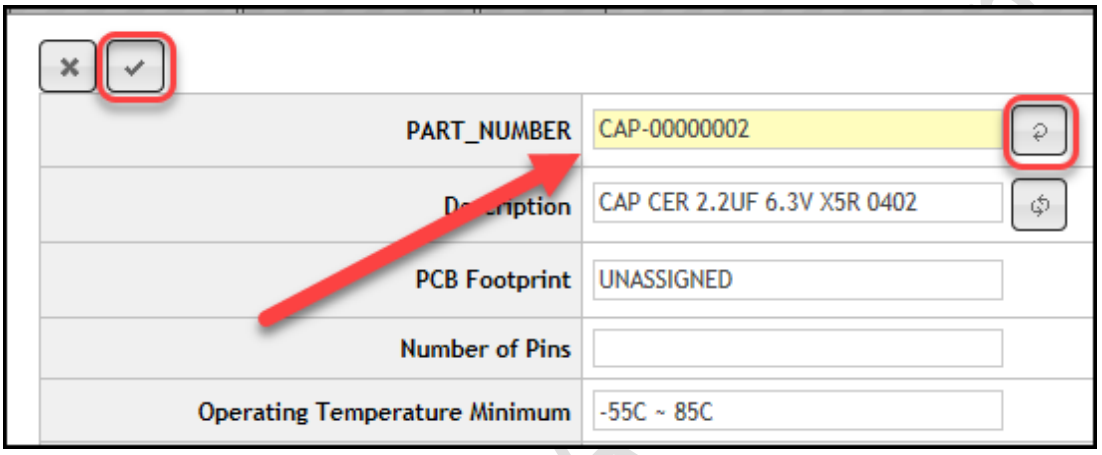
Temporary parts can be deleted, although it is best to verify that it is not currently being used.



Once a Temp Part has been created, the temporary part number should be replaced by a formal part number.

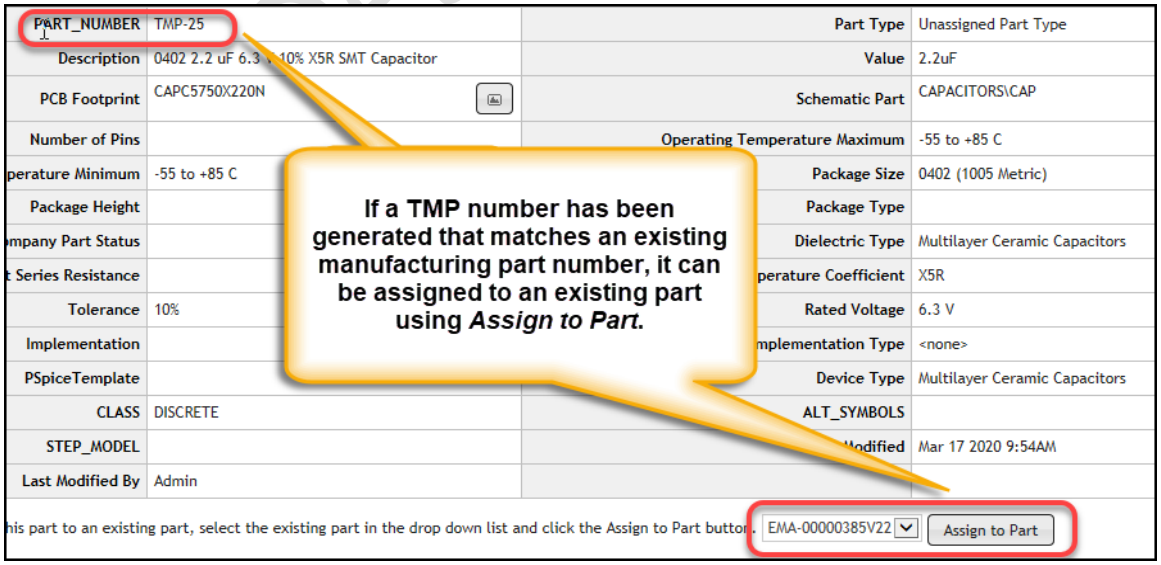
Assigning the Next Part Number

As an option, CIP can generate part numbers using the **Assign Next PN** button. Incremental part numbers can be enabled and assigned in the ***Admin > Configuration > Incremental Part Number*** menu. Once a part number has been assigned to the part, the librarian would then notify the engineer that the part has been updated so the engineer can use the ***Link Database Part*** command in CIS to update the part. Part process will be discussed in greater detail later in this training.



Assign to Part

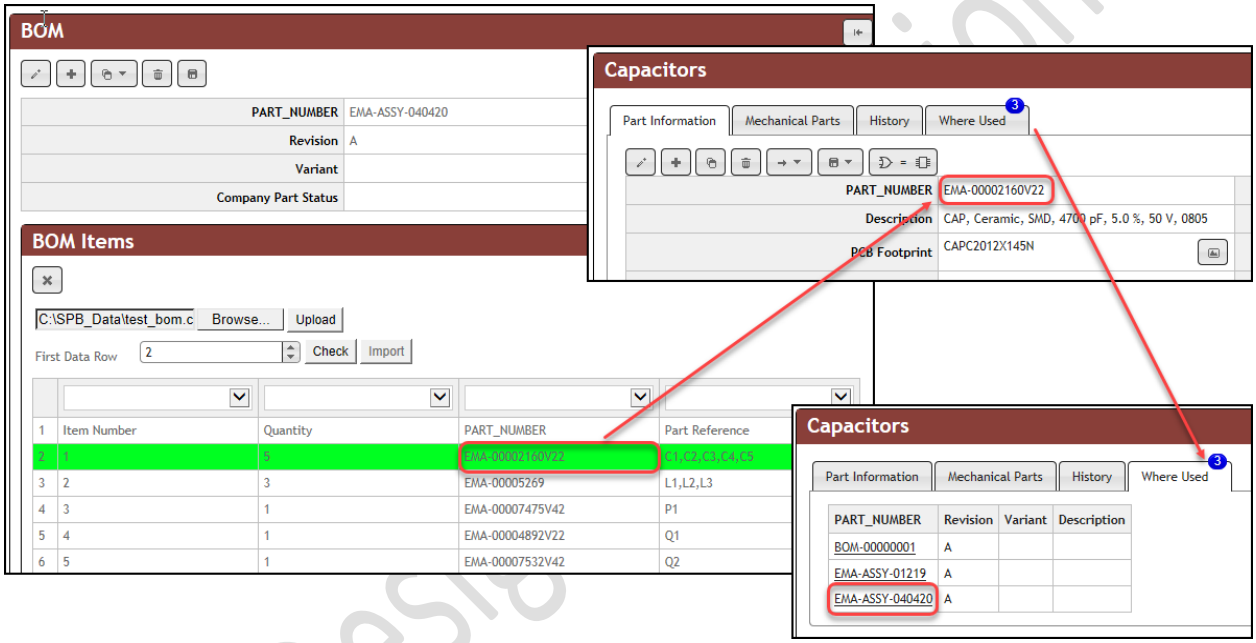
If a temporary part is found to be a duplicate of an existing part, it can be assigned to the existing part number by using the **Assign to Part** feature.



Uploading BOMs into CIP

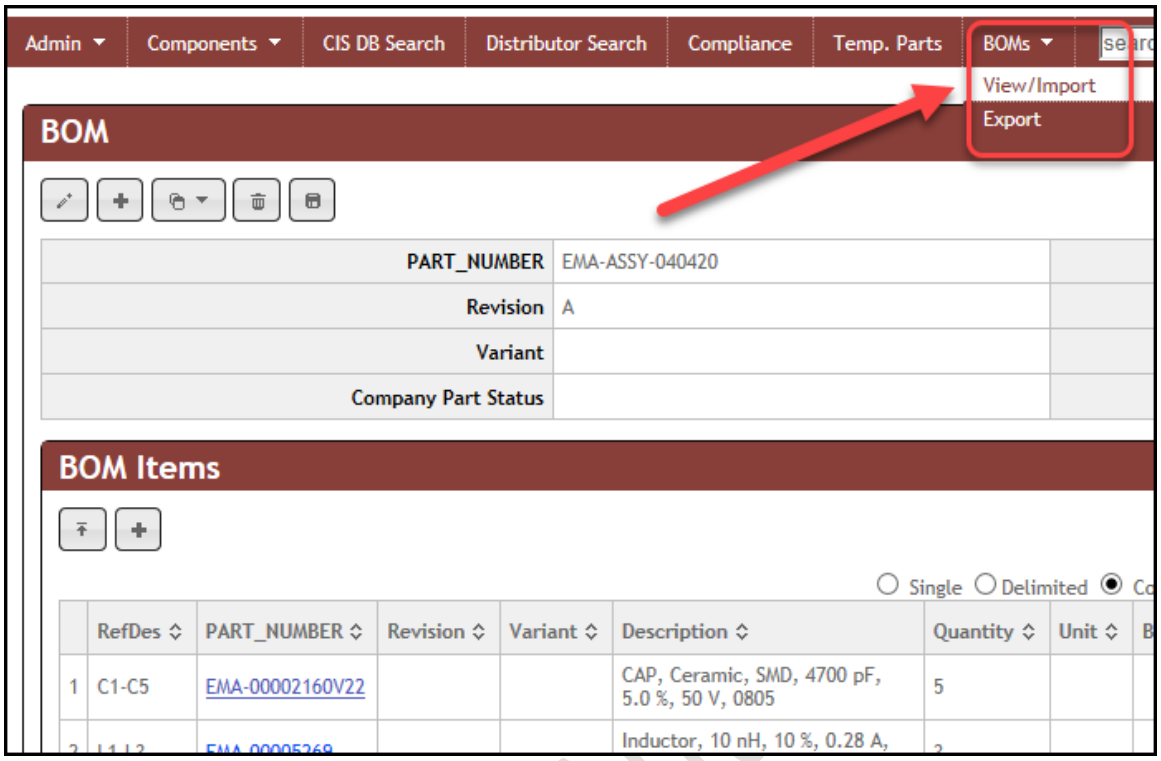
When searching for a part, it may be necessary to determine whether parts have been used in other designs. Parts lists/Bill of Materials (BOMs) can be uploaded into CIP to allow for searching. Any number of BOMs can be imported.

Each imported BOM is assigned a unique part number, and a parts list file is selected to import the list of parts into CIP. After import, when the detail information on the part is viewed, the **Where Used** tab will identify the list(s) where the part appears.

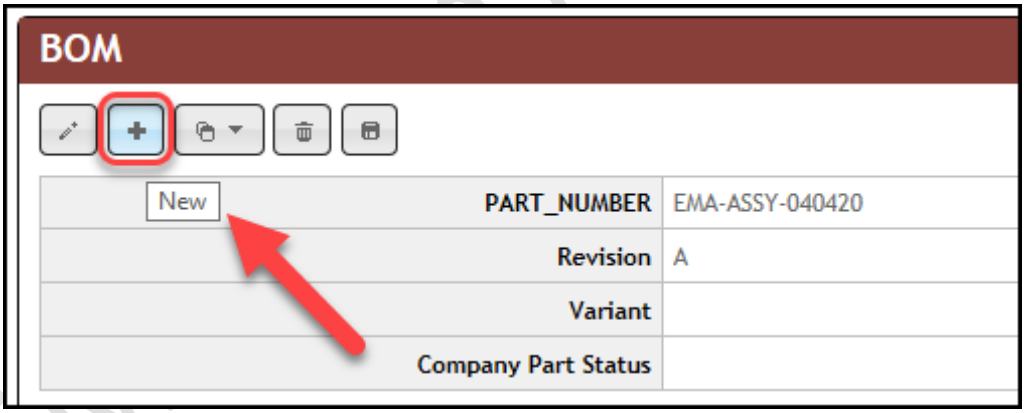


Steps for uploading BOMs into CIP:

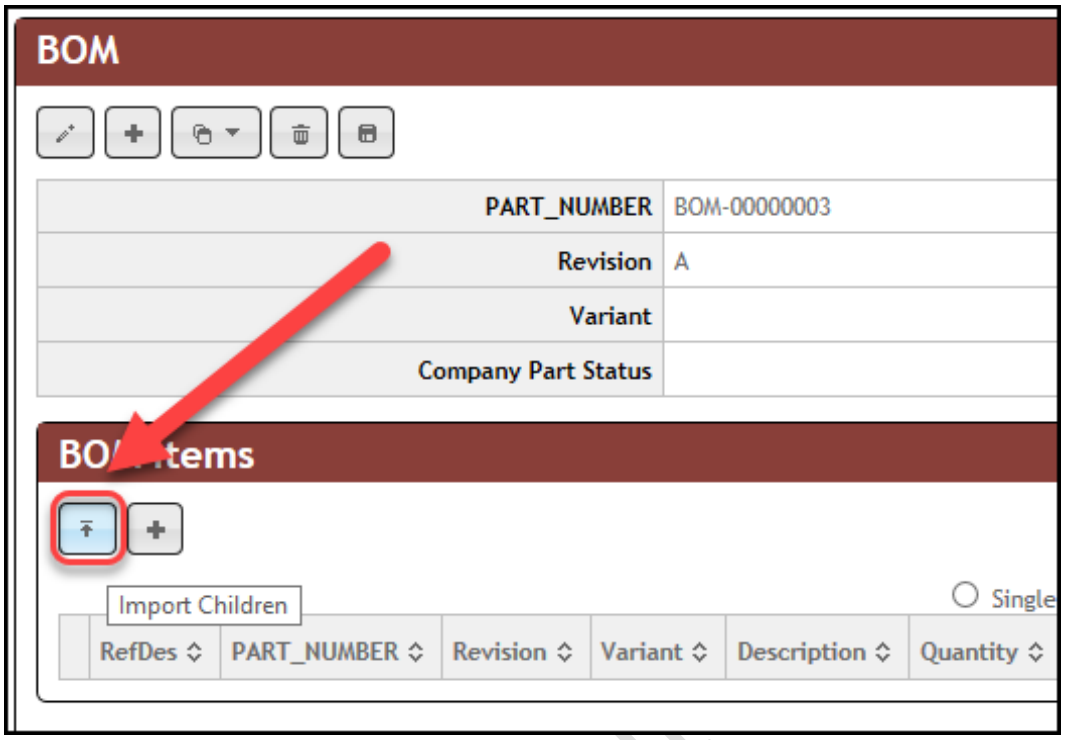
- Select ***BOMs > View/Import***



- Select the plus sign to assign a new BOM number and BOM revision.



- Under ***BOM*** Items, select the ***Import Children*** button.

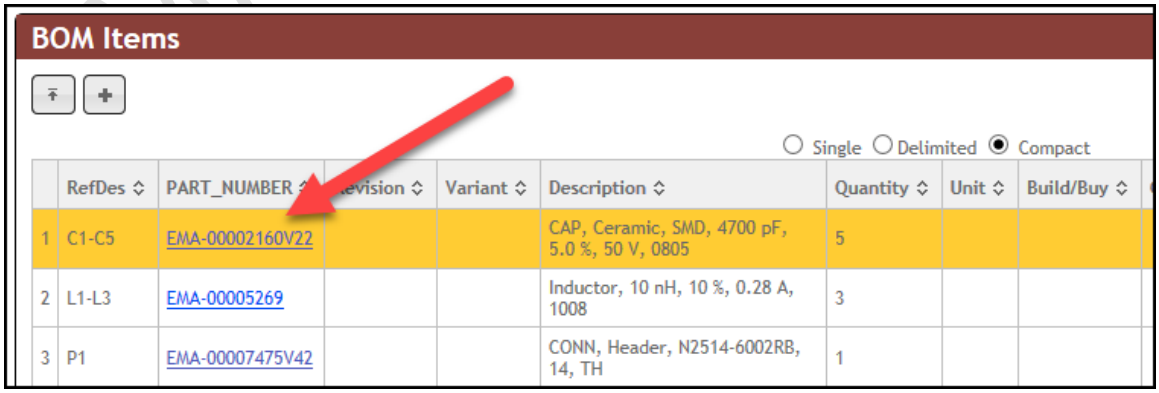


- **Browse** to locate and **Upload** a CSV file. **NOTE:** If a design has variants, a variant BOM can be chosen for upload.
- **Add Headers** and validate the file

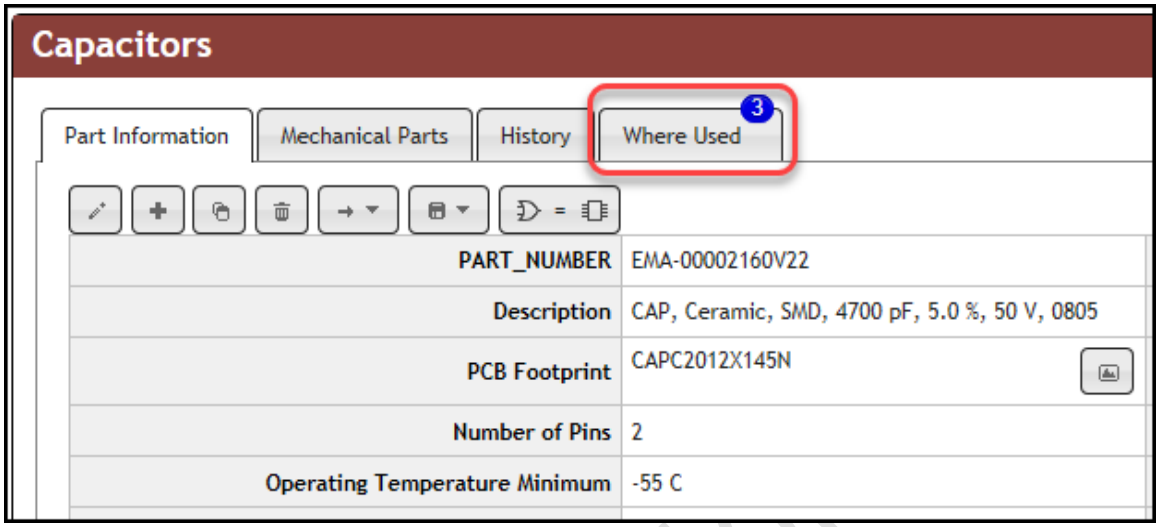
NOTE: In an upcoming lab you will step through uploading a BOM into CIP.

Determining Where Parts are Used

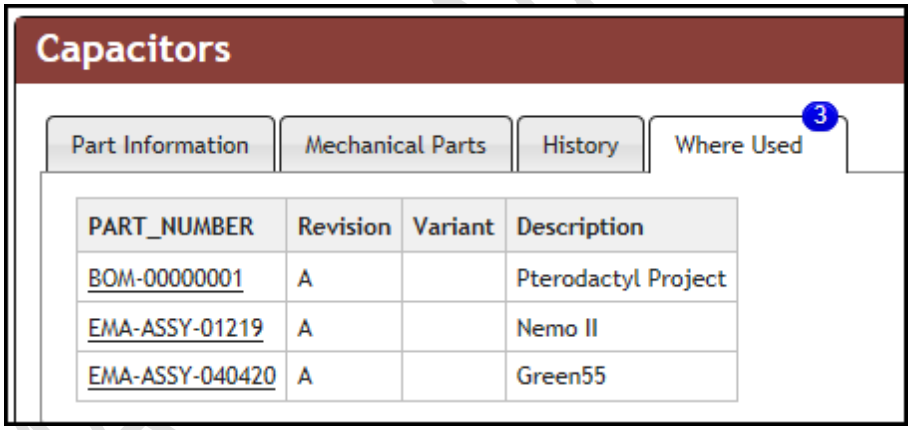
Once you have several BOMs uploaded into CIP you can begin tracking part usage with the **Where Used** function. With a BOM part number selected, the list of parts it contains will be displayed.



By selecting a part number from the list of parts, the **Part Information** page will open, showing the information associated with the part. The **Where Used** tab will display other BOMs where this part has been used.

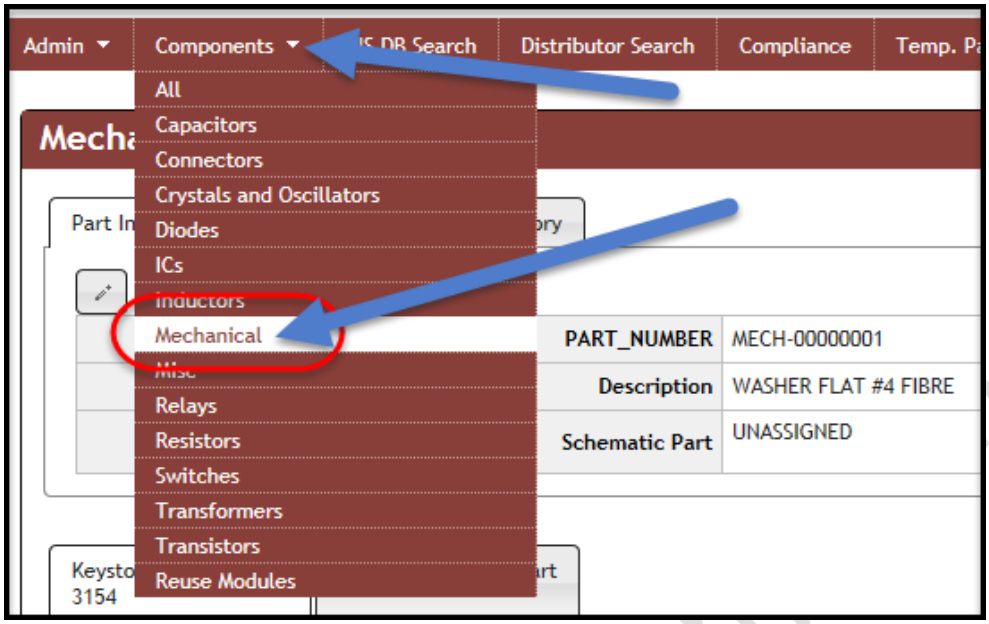


The results will open with a list showing where the part has been used.

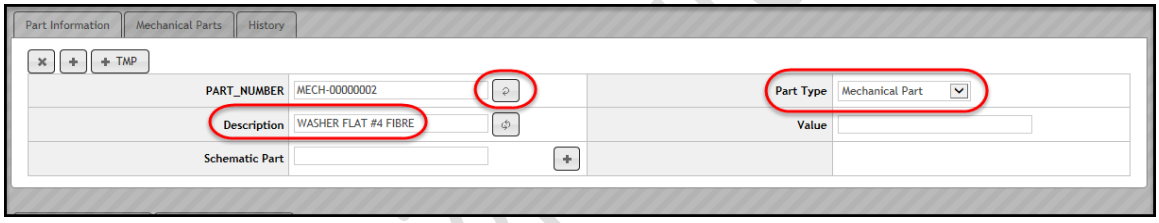


Adding Mechanical Parts

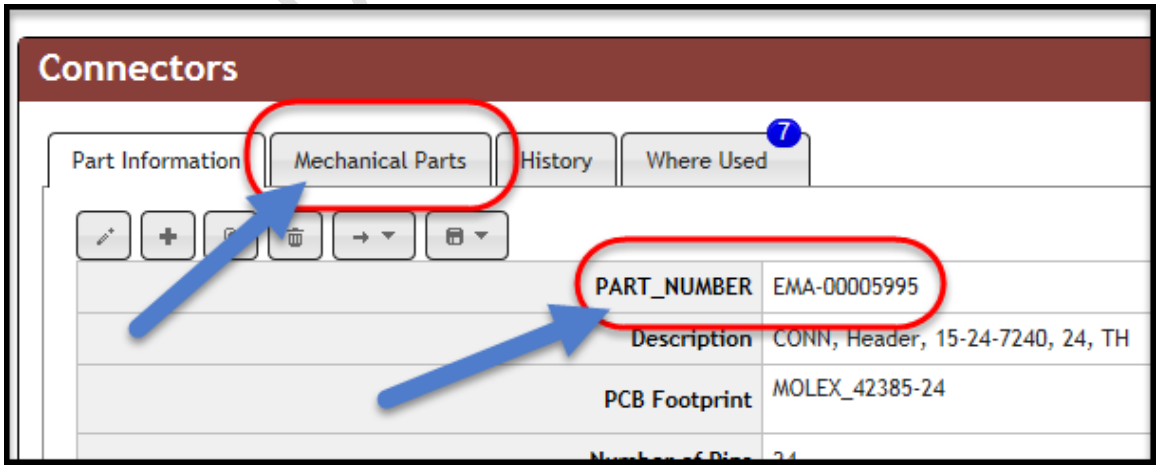
Mechanical parts, such as screws and washers, may now be added to existing parts in the database. These parts can be included when BOMs are generated in Capture.

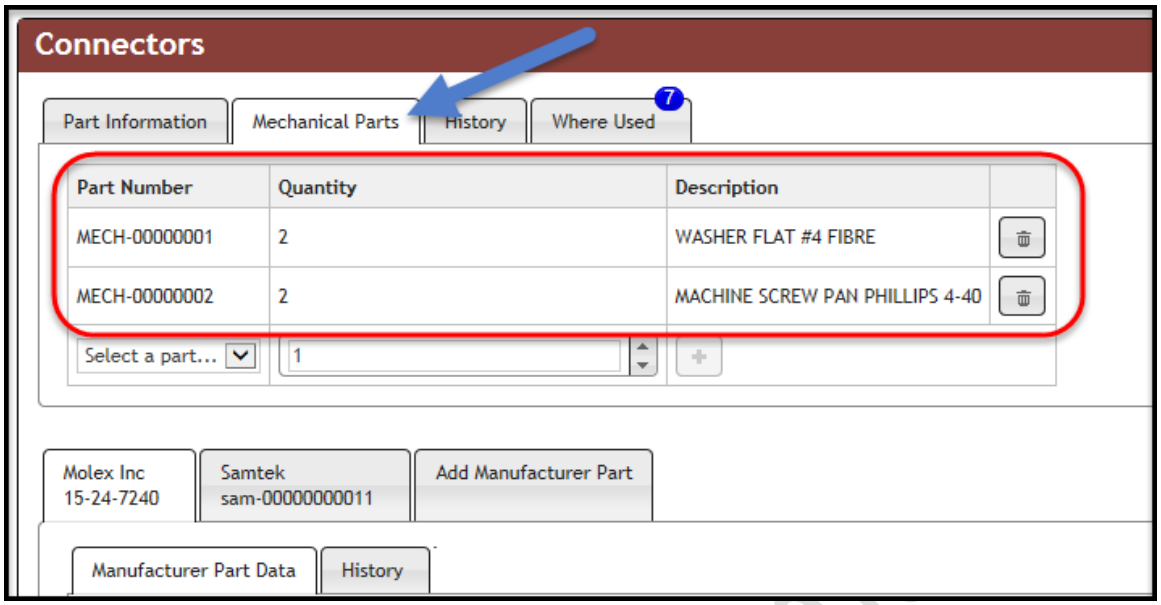


Selecting the **New** button will open the editor to add a new mechanical part.



Once new mechanical parts have been added, they can be associated to other parts. For example, a connector may also need mechanical parts to secure it to the printed circuit board.



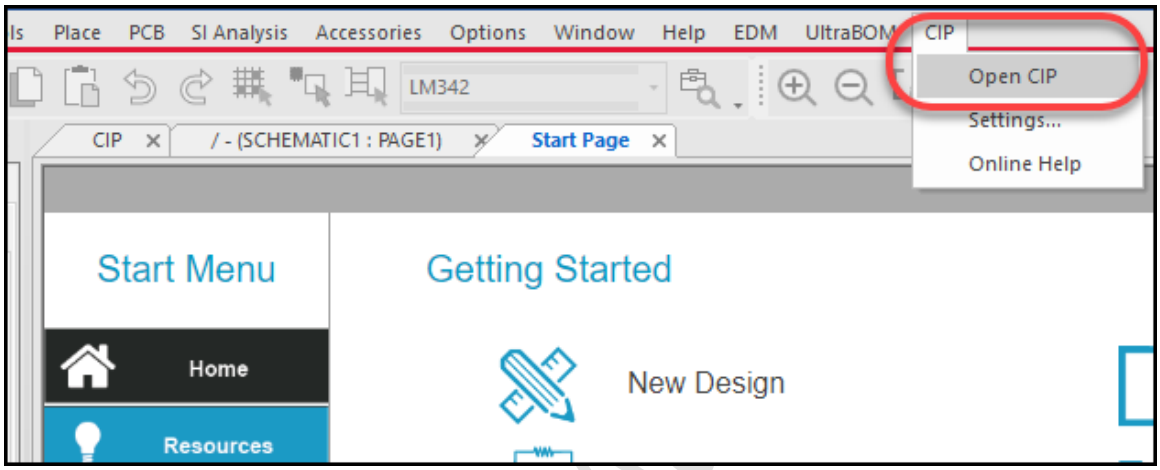


Each time the part is placed on a schematic, the mechanical parts that have been associated to the original part will automatically become part of the design. Once mechanical parts have been associated with other parts, they can be exported to a BOM.

Lab 1-1: Logging in to CIP

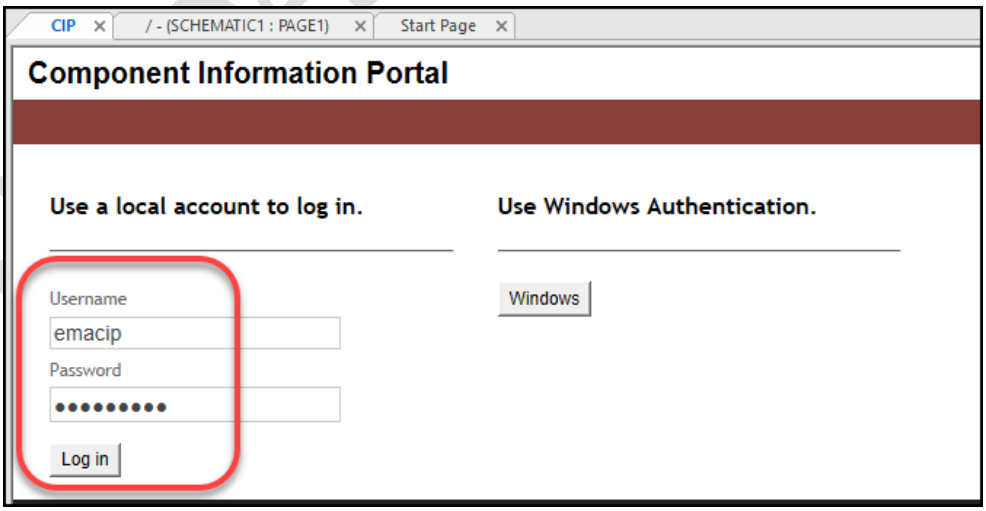
For this training all the tools you will be using are located on the AWS (Amazon Work Space) machines that have been assigned to you.

1. Open OrCAD Capture CIS on your remote desktop. Select ***CIP > Open CIP***.



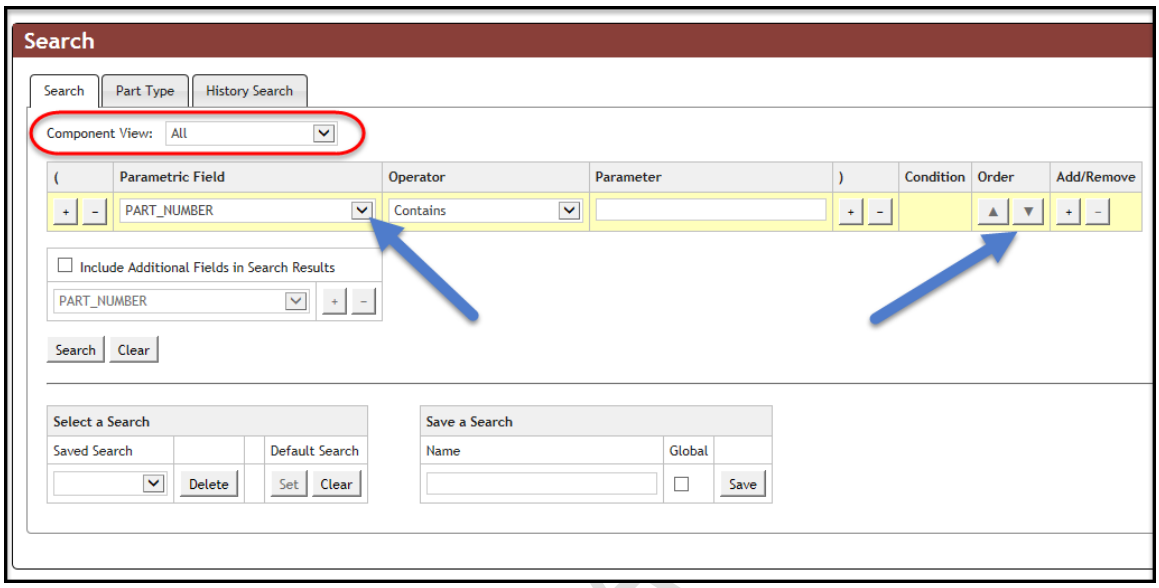
2. Once CIP opens enter the **Username** and **Password**.

   - **Username**: **`emacip`**
   - **Password**: **`Emacip_01`**



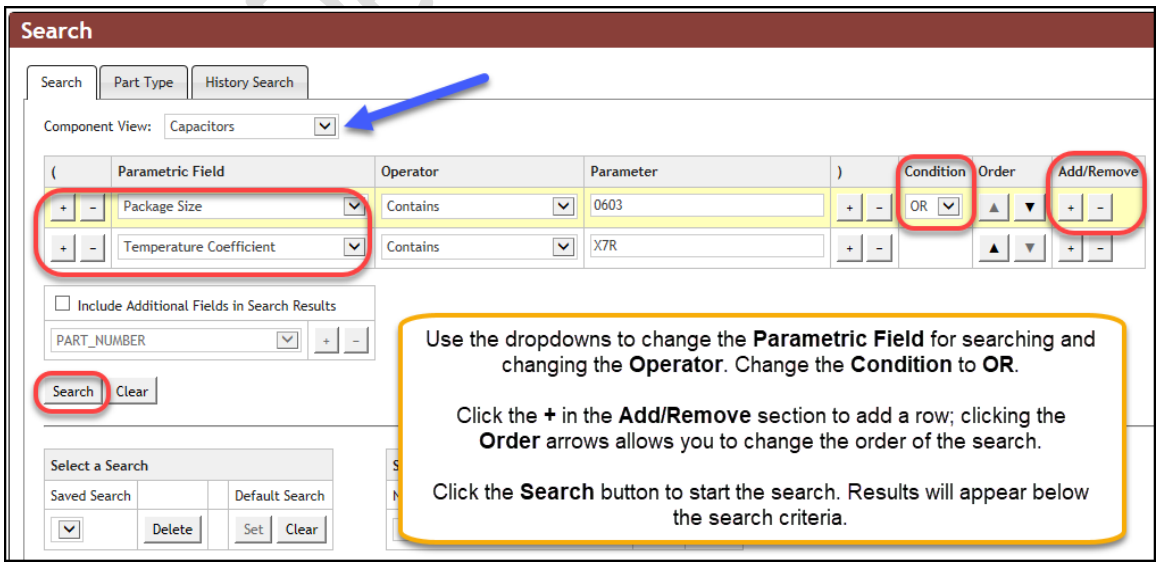
Lab 1-2: Performing a CIS DB Search

1. Once you are logged in, the **CIS DB Search** Tab will be shown as the default. In the ***Component View*** dropdown, select ***Capacitors***.



Change the Default Search Data

Note that the default property in the Parametric Field is PART_NUMBER, but you can change this along with the order of the search parameters.



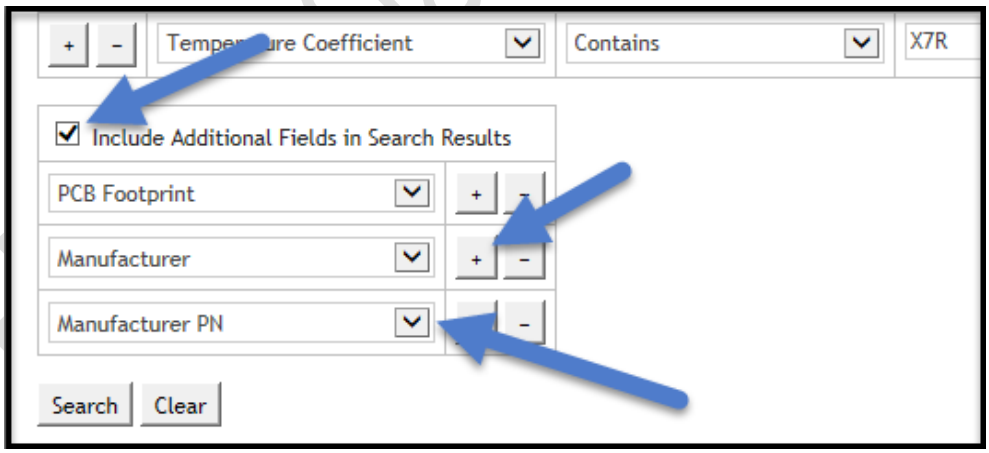
2. In the ***Parametric Field***, use the dropdown to select ***Package Size***. Change the ***Operator*** to ***Contains***. In the ***Parameter*** block enter ***0603***.
3. Click the ***+ button*** in the ***Add/Remove*** area to add another row.
4. In the ***Condition*** area use the dropdown to select ***OR***.
5. Use the dropdown to change the ***Parametric Field*** in the newly added row to ***Temperature Coefficient***.
6. Use the dropdown to change the ***Operator*** in the newly added row to ***Contains***.
7. Enter ***X7R*** n the ***Parameter*** field of the new row**. <u>Do not start the search yet</u>.**

Include Additional Fields

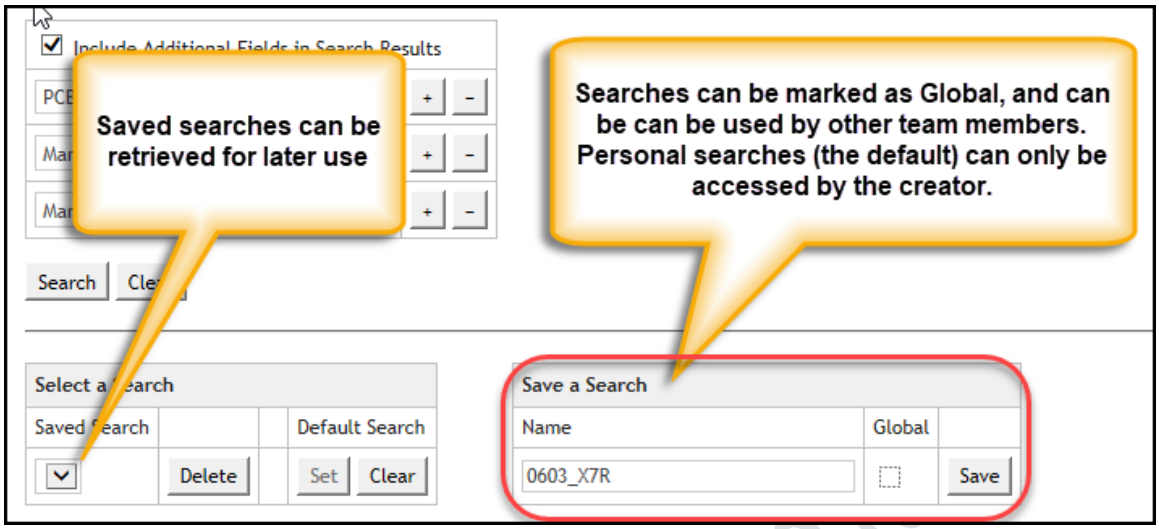
Additional fields can be included to further refine the search.

1. Enable ***Include Additional Fields in Search Results***.
2. Click on the ***Add Field +*** button twice to add 2 more rows.
3. Click in the dropdown boxes to select the following fields: ***PCB Footprint***, ***Manufacturer*** (you will have to scroll down in the dropdown list in the ***Manufacturer*** fields to find this), and ***Manufacturer PN*** (you will have to scroll down in the dropdown list in the ***Manufacturer*** fields to find this).

   <u>**Do not start the search yet**</u>**.** You will save it so you can retrieve it later.



Save the Search

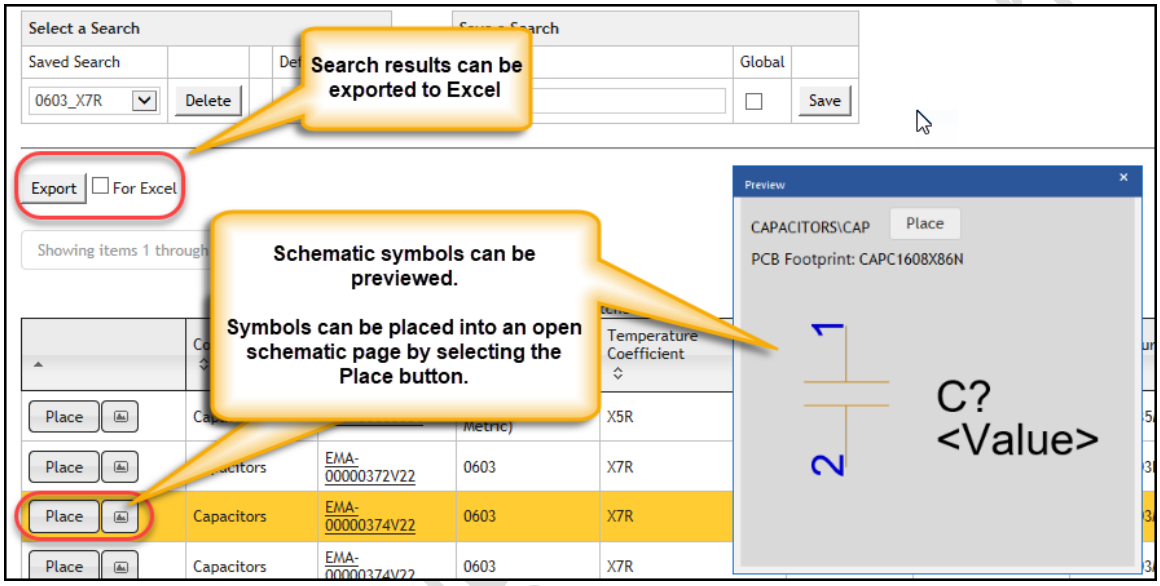


1. In the **Save a Search** area, enter the name ***0603_X7R***. Click the ***Save*** button to save the search.
2. Click on the ***Search*** button to start the search. The included fields will be reflected in the results.

Lab 1-3: Reviewing Search Results

Search results will be displayed below the criteria and can be exported to an Excel file.

1. Scroll down to review the search results. When you hover the mouse over an item in the results list, it becomes highlighted. Click on one of the items to review the part data.

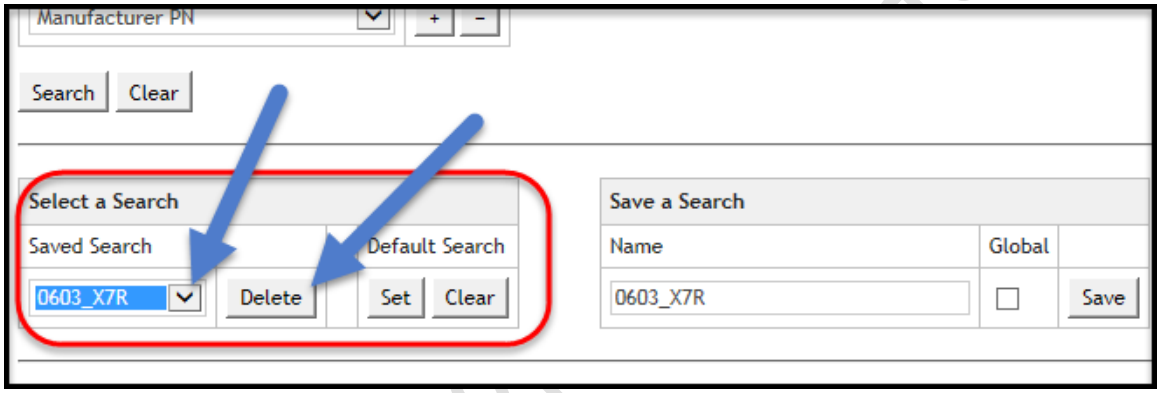


2. Click on the ***Preview*** button to review the schematic symbol. Do not place it.

Lab 1-4: Retrieving Saved Searches

Once a search is generated, it can be saved for later use. Globally defined searches are available to all CIP users. Personal searches are only accessible to the user who has generated them. In this exercise, you will retrieve a saved search.

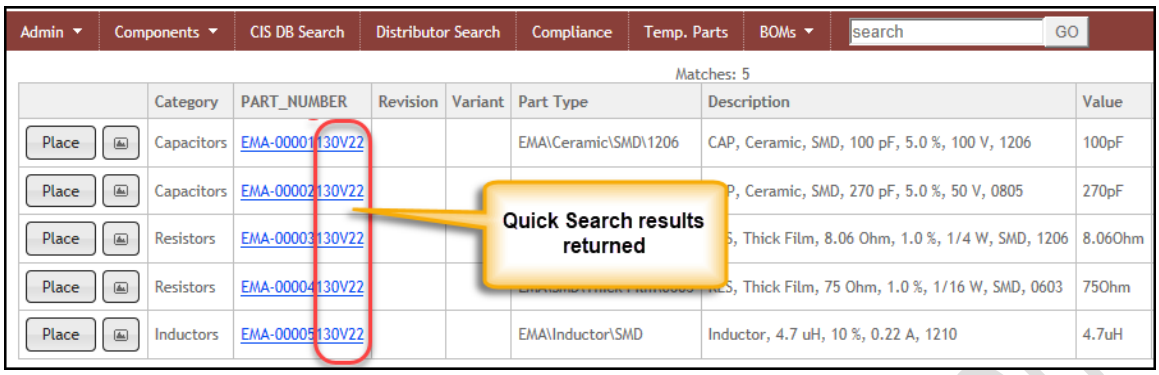
1. Select the ***CIS DB Search*** menu to go back to the Search criteria page.
2. In the ***Select a Search*** area (towards the bottom of the search page), use the ***Saved Search*** dropdown to select the previously saved search ***0603_X7R***. The parametric fields repopulate with the data from the 0603_X7R search.
3. Searches can also be deleted. Select the ***Delete*** button to remove the saved search.



Using Alternate Searching

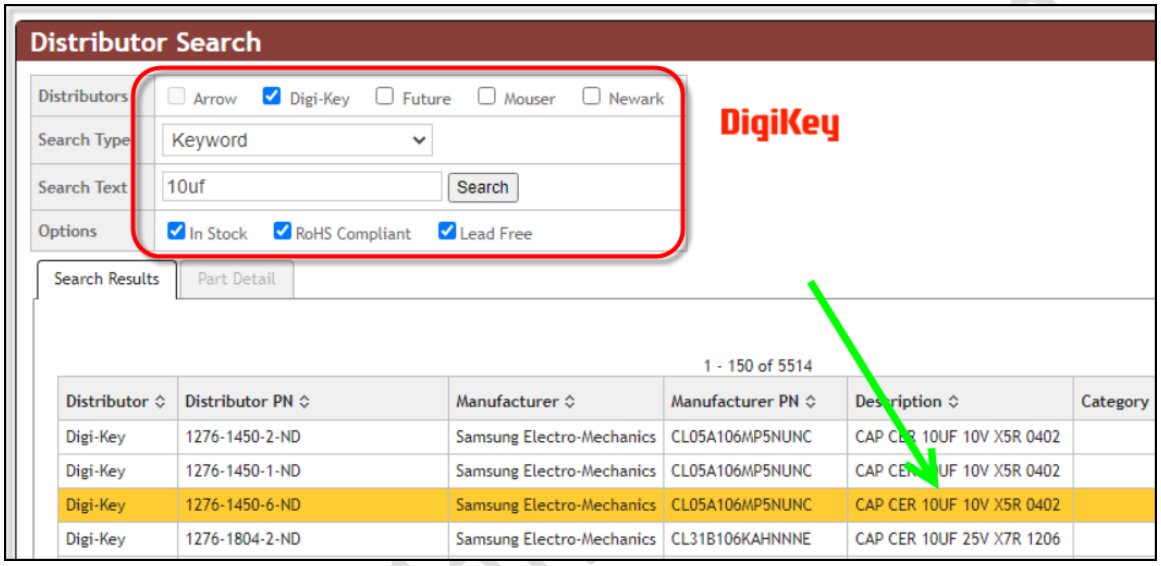
An alternate Quick Search is available in CIP for finding parts by typing a string into the search block.

2. In the search block, enter ***130V22***.
3. Click the ***GO*** button to start the search. Review the results. The string entered is in the part number.



Lab 1-5: Performing a Distributor Search

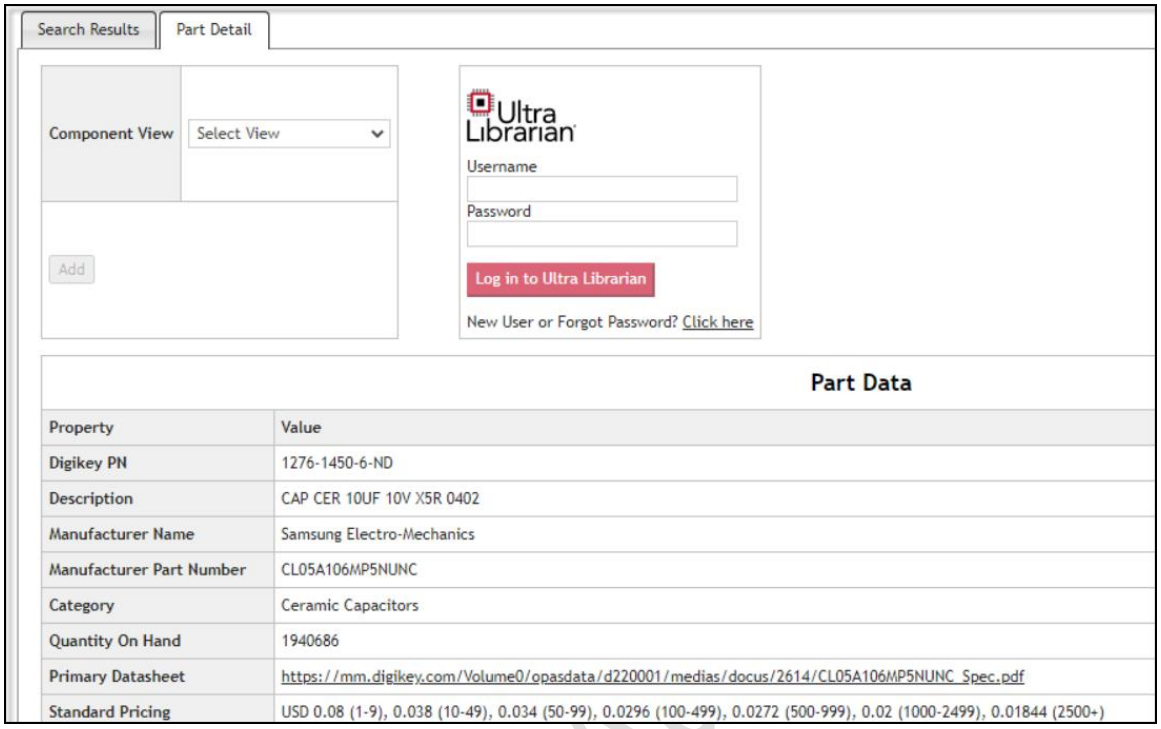
1. Click on the ***Distributor Search*** menu.
2. Select ***Digi-Key***.
3. Select the ***Keyword*** option and enter the value ***10uF***.
4. Check the options for ***In Stock***, ***RoHS Compliant***, and ***Lead Free***.
5. Click ***Search*** and review the results.



Reviewing Part Details

1. Click on a part in the search results.

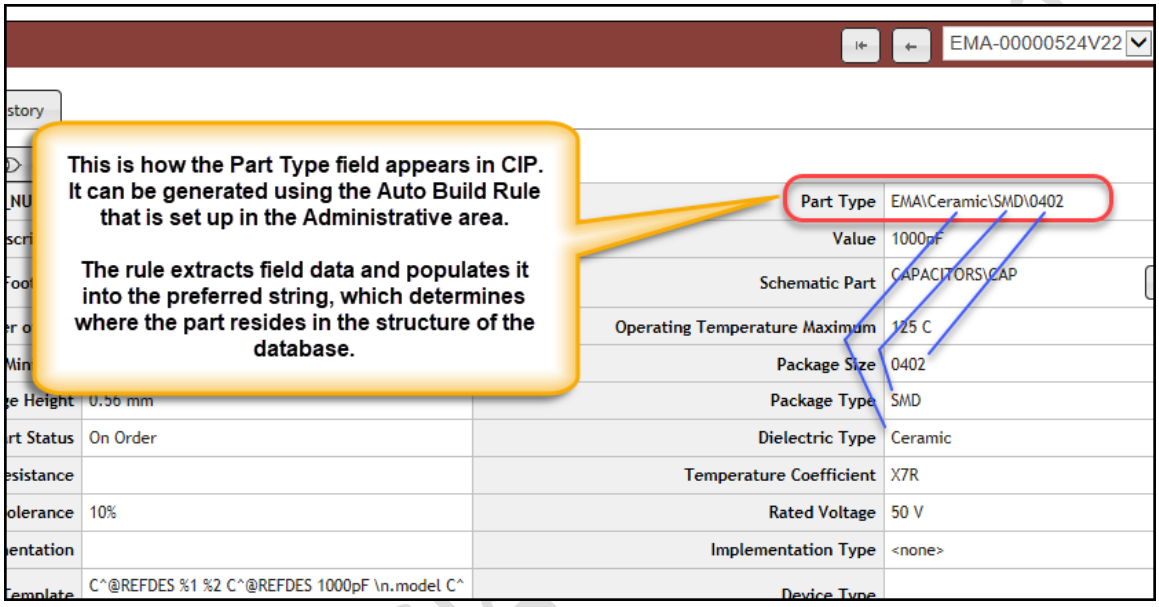
Note: The part you select may differ from the following picture.



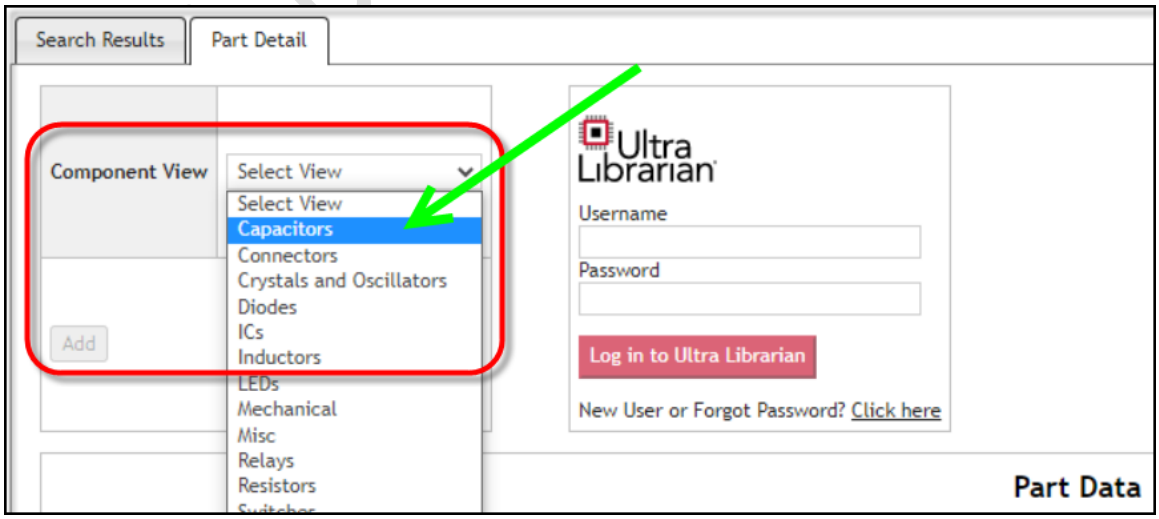
Do not close the part detail page. You will use this part for the next exercise.

Lab 1-6: Creating a TMP Part Using the Distributor Search Portal

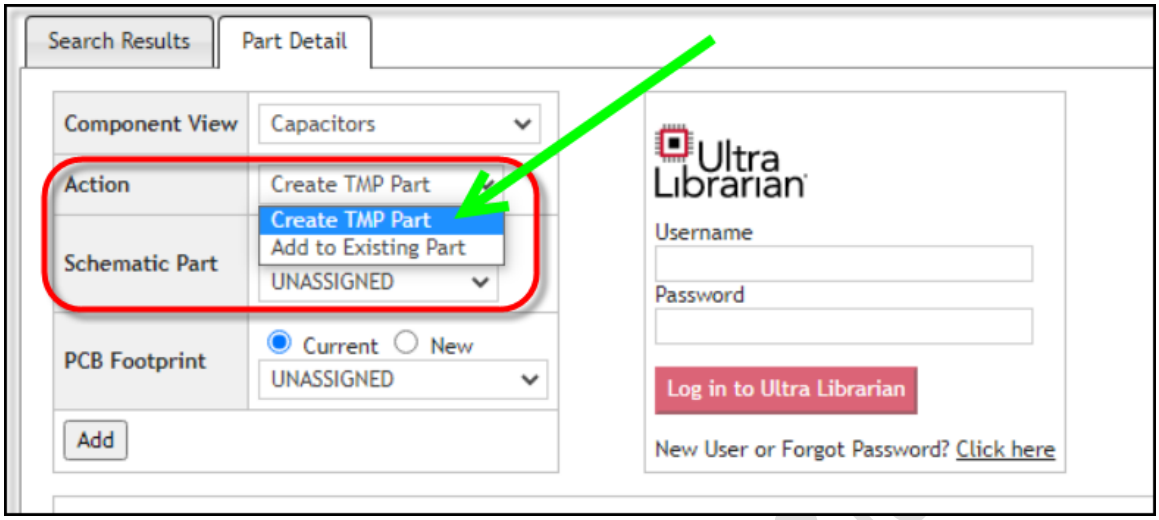
The Category, or Table, is the primary location where a part resides in the database. The **Part Type** field defines the subcategories. You can define this in the CIP editor. The following image shows how the Part Type appears in CIP.



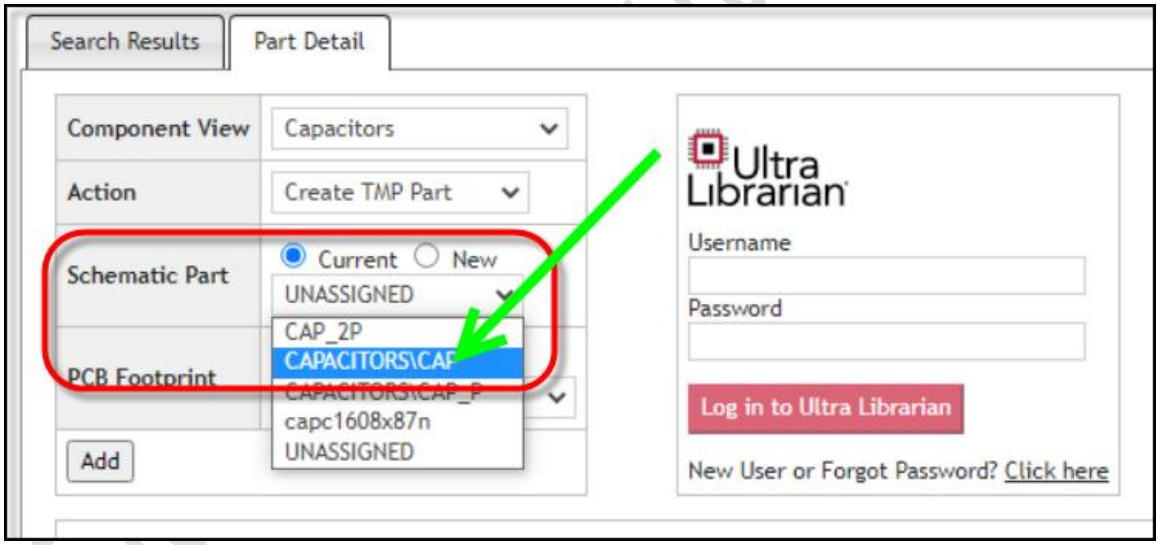
1. The part detail from the previous exercise should be open. Towards the top of the **Part Detail** page, use the dropdown in the ***Component View*** area to select ***Capacitors***. This is the category table where the part will reside in the database.



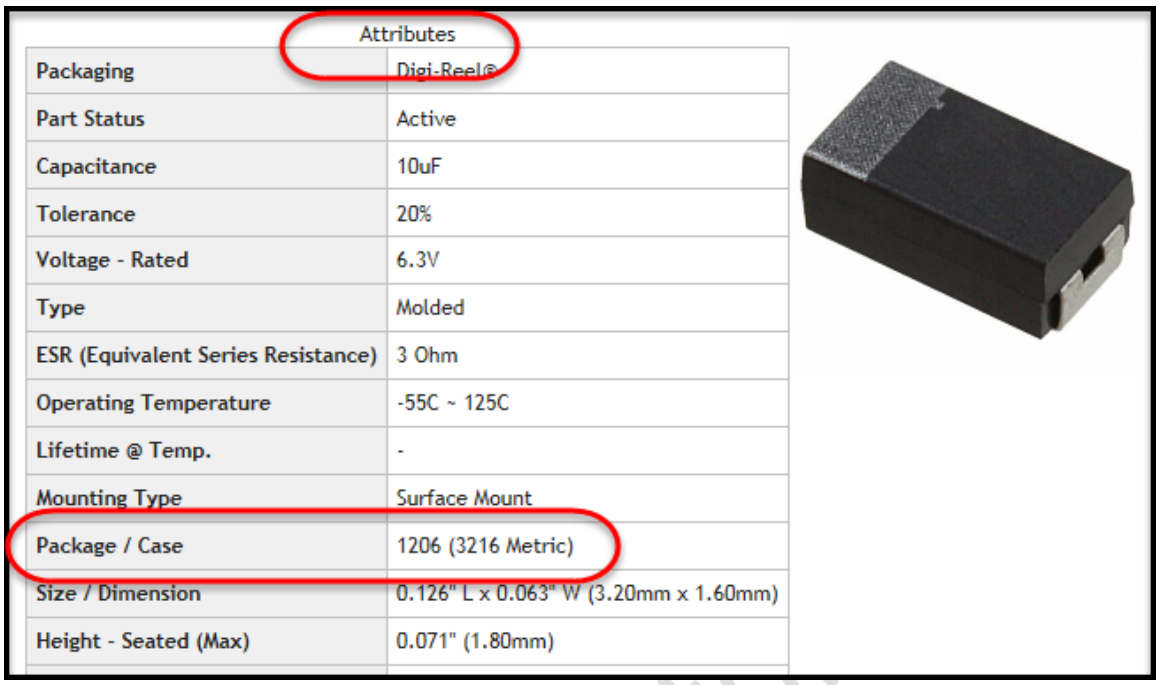
2. Make sure the ***Action*** is set to ***Create TMP Part***.



3. In the ***Schematic Part*** area, select ***Current*** and use the dropdown to select ***CAPACITORS\CAP***.



A PCB Footprint may exist for the part and can be selected in the **PCB Footprint** area. For this exercise, because you chose to search for a part that is an **0603**, the EMA library contains PCB footprints that will work for this part. Typically, you would check the package size provided in the part detail and look for the footprint that matches this part in the PCB Footprint dropdown. In general, you can check the Package/Case in the Attributes to find the package size:

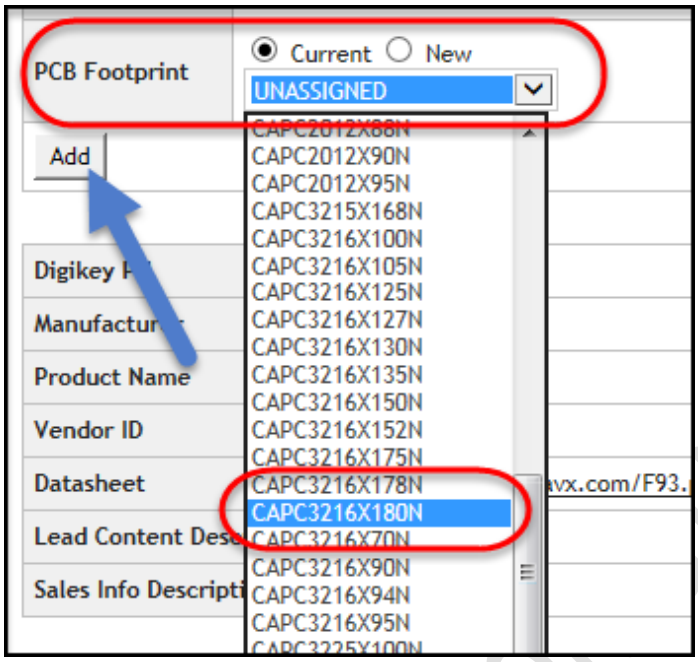


Attributes	
Packaging	Digi-Reel®
Part Status	Active
Capacitance	10uF
Tolerance	20%
Voltage - Rated	6.3V
Type	Molded
ESR (Equivalent Series Resistance)	3 Ohm
Operating Temperature	-55C ~ 125C
Lifetime @ Temp.	-
Mounting Type	Surface Mount
Package / Case	1206 (3216 Metric)
Size / Dimension	0.126" L x 0.063" W (3.20mm x 1.60mm)
Height - Seated (Max)	0.071" (1.80mm)

NOTE: When you use the dropdown to see the footprints that are available for this cap, you will see the EMA parts that have been installed. These parts have been built based on IPC criteria; therefore, the naming convention for these parts is based on IPC naming conventions. For example, the PCB Footprint you will choose from the list is named **CAPC3216X180N**.

IPC naming convention defines this as: **CAPC** (CAP CHIP); **3216** (metric package size); **X180N** (by height; all measurements are NOMINAL).

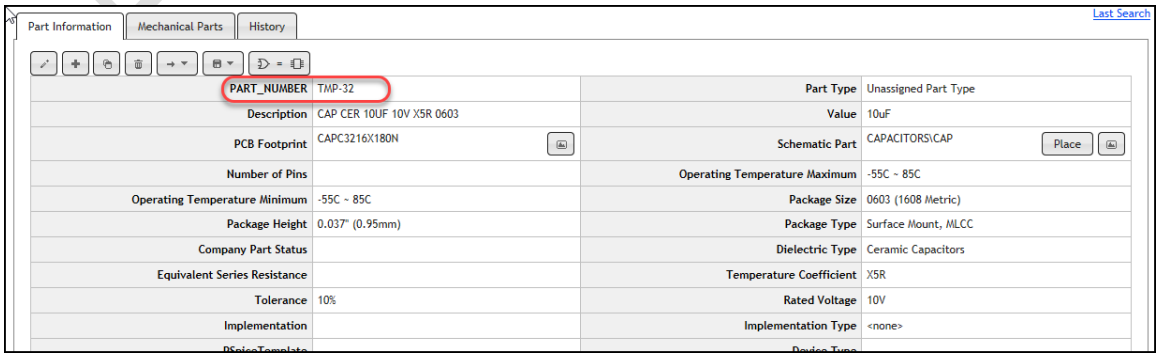
4. In the ***PCB Footprint*** area, select ***Current*** and use the dropdown to select ***CAPC3216X180N***.
5. Click ***Add***.



When you map a schematic symbol and PCB footprint at this stage that they do NOT come from the distributor or vendor. These are symbols and footprints that *already exist in the database*. CIP offers you a choice to map to an existing part.

To supply customer demand for content, EMA has implemented **Ultra Librarian**. Ultra Librarian™ for OrCAD provides a comprehensive, cloud-based library of over sixteen million components, eliminating the need for manual building and maintenance. Ultra Librarian for OrCAD offers engineers the option to search, preview, and place components quickly without leaving the native CAD design environment, saving time and eliminating errors. More information can be found at https://www.ema-eda.com/products/orcad/ultra-librarian-for-orcad.

The part has automatically been created and assigned a TMP part number. This means the part has been added to the database. The part data, along with parametric data, is transferred from the Distributor to the local CIS database.

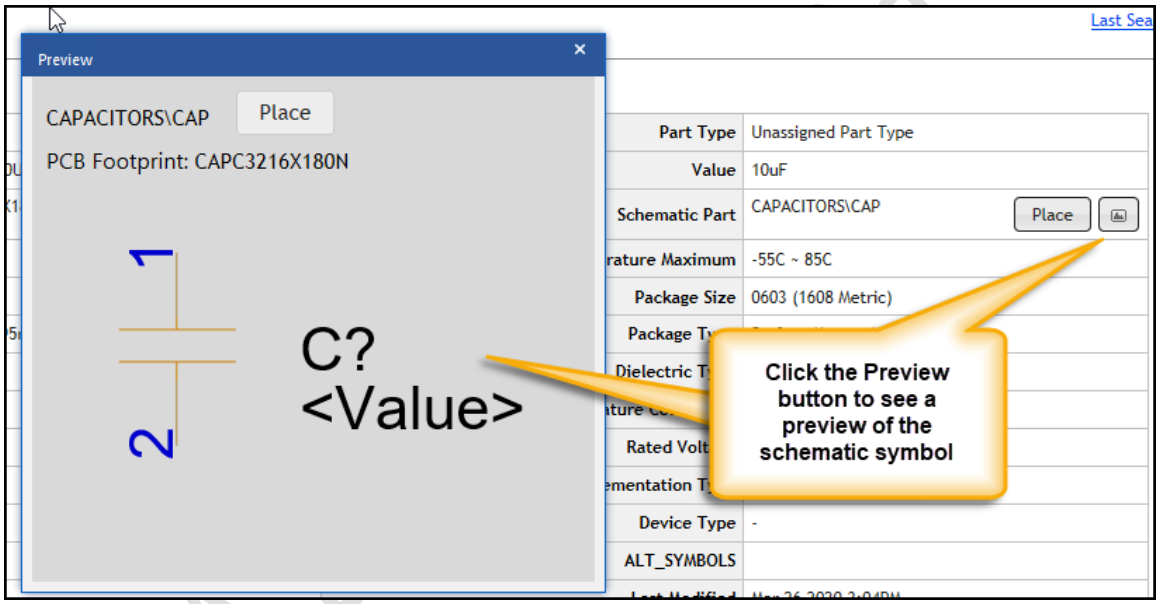


Note: Your TMP part number may be different than in the image above. This is due to the different databases on each remote training machine. Most important to note is that CIP automatically tracks new temporary parts and their part numbers.

Previewing the Schematic Symbol

In CIP, you can preview and place the schematic symbol on the schematic page. You will be placing a part from CIP in a later exercise. In the following exercise you will preview the part.

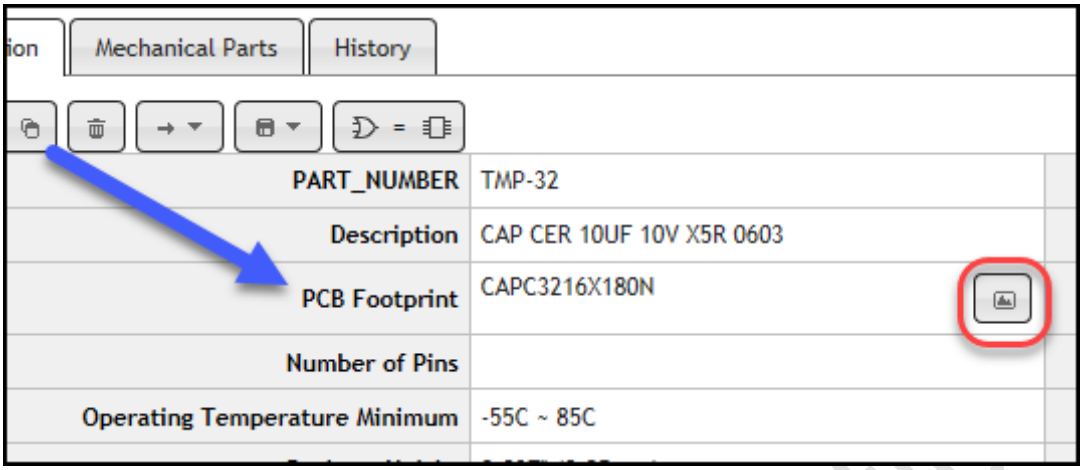
1. In the ***Schematic Part*** field of the ***Part Information*** tab, there are options for placing the part in the schematic and for previewing the part prior to placement. Select the ***Preview*** button to view the symbol. Close the Preview window.



Previewing the PCB Footprint

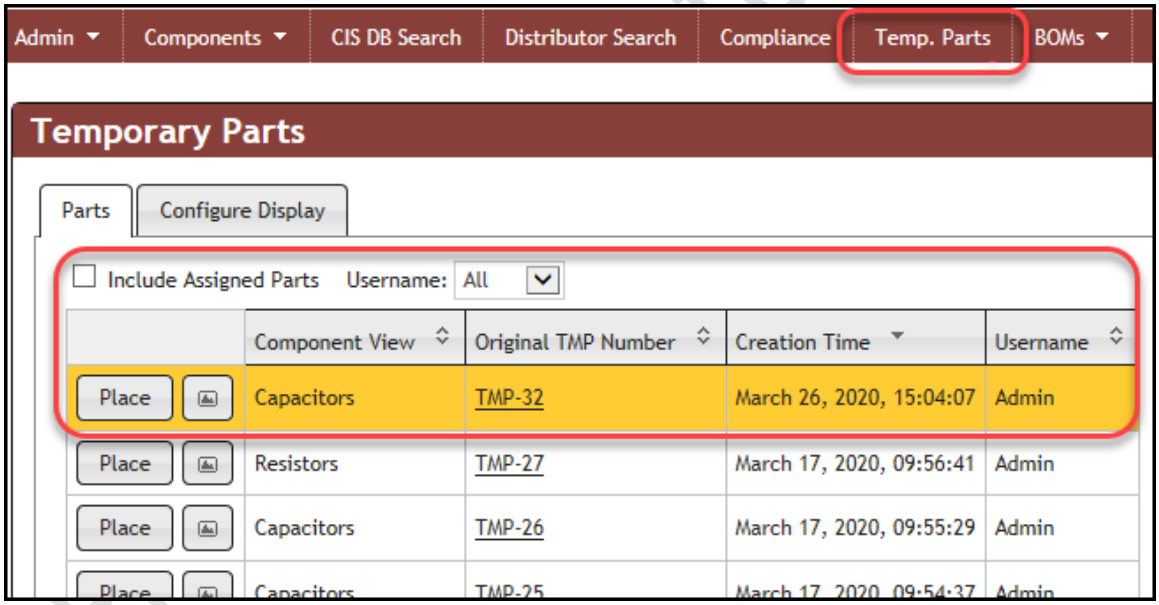
If you mapped a PCB Footprint symbol to the part that was previously created, you can choose to preview it.

1. In the ***PCB Footprint*** field, click the ***Preview*** button.



Checking the Temp Parts Tab

When you check the **Temp Parts** menu, you can see all the temporary parts that have been generated, the creation time, and the user who generated them.

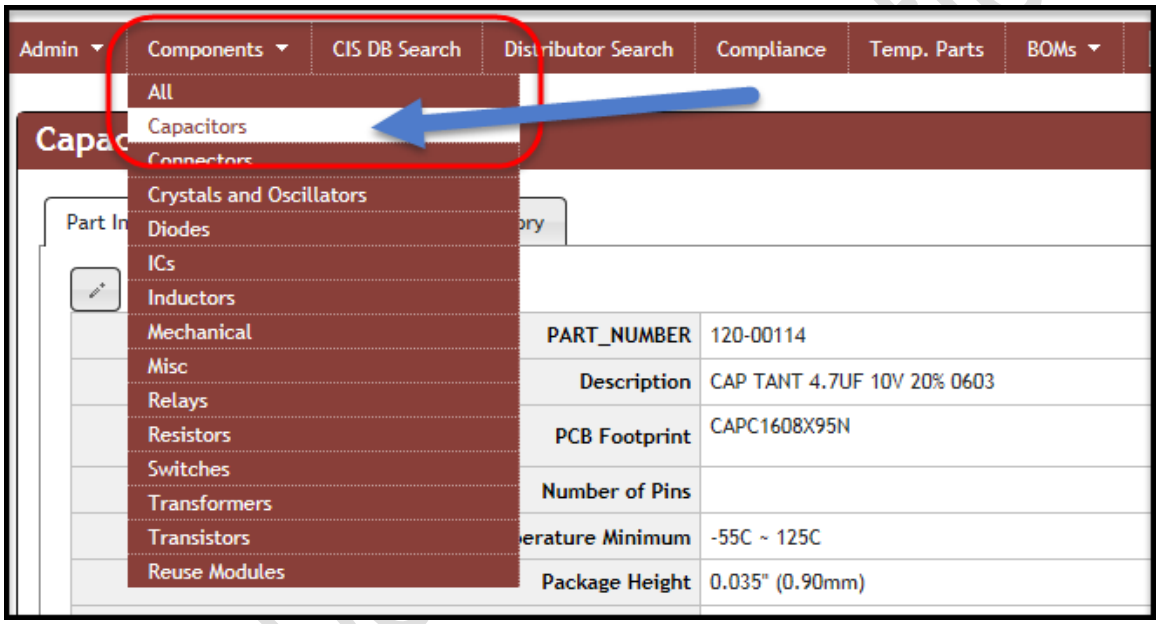


1. Click on the ***Temp. Parts*** menu to view the list of temporary parts that have been generated.
2. Click on a part to select it and view the part details.

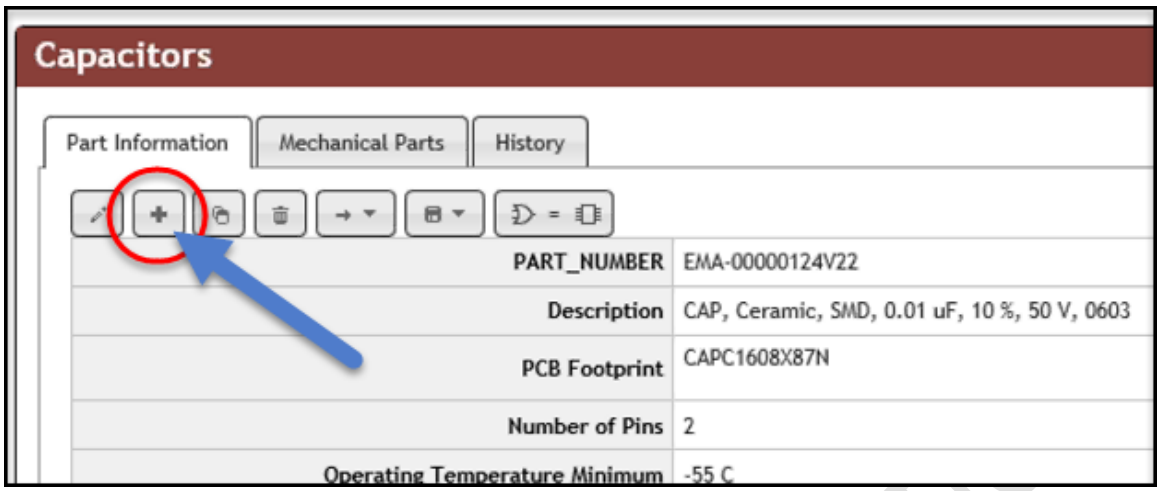
Lab 1-7: Adding a New (Blank) Part

There are several ways to generate new parts within CIP, such as copying from an existing part to create a new part, starting with a blank form, and manually entering the data, or adding a part using the **Distributor Search** menu. In the following exercise, you will generate a new capacitor by using the NEW option.

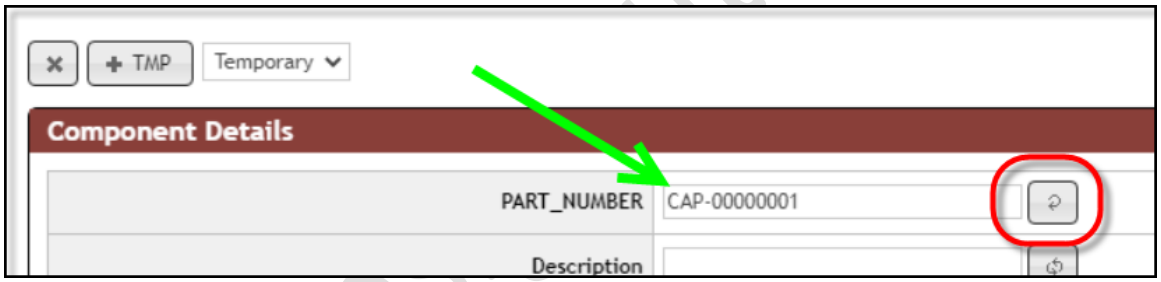
1. In the ***Components*** menu, use the dropdown to select ***Capacitors***. The part detail will appear for the first capacitor part number in the Capacitors table. You can use this part as the template to copy to a new part.



2. Once the part detail appears, click on the ***New*** button  .

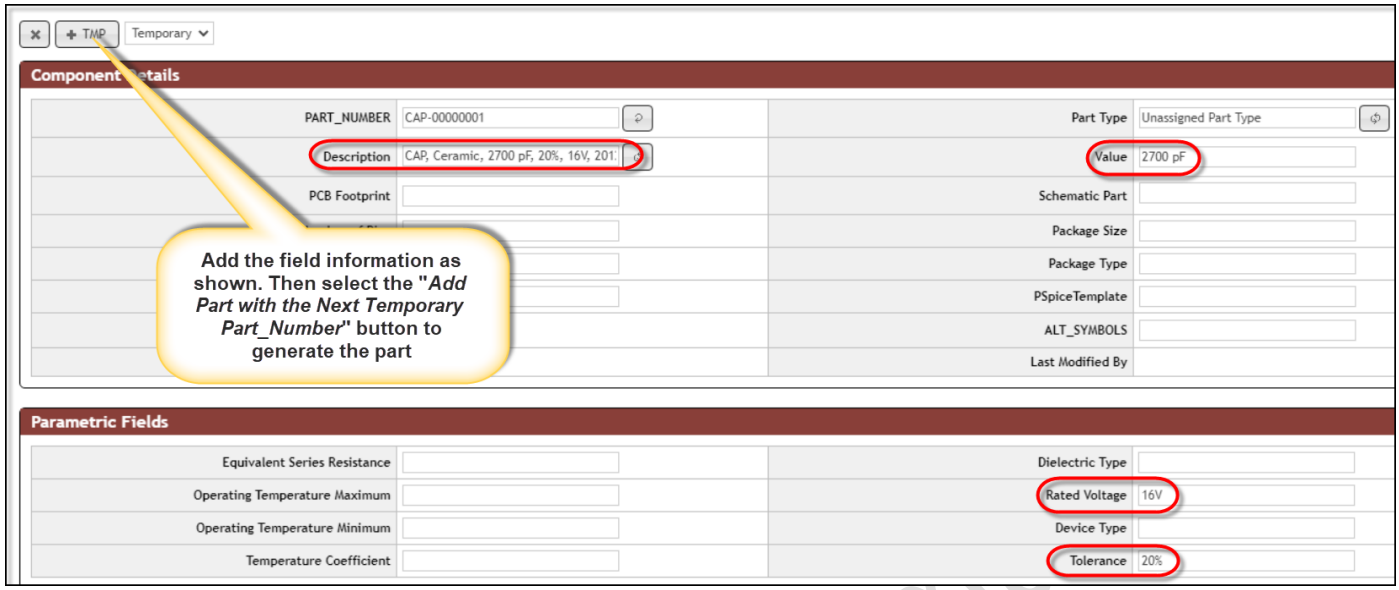


3. In the ***PART_NUMBER*** field, click on the **Next PN** button to populate the next available CIP part number for Capacitors. If your company does not already have a part numbering system, you could use the one provided in CIP. This applies to all categories of parts.



4. Populate the remaining fields as shown:

   DESCRIPTION = CAP, Ceramic, 2700 pF, 20%, 16V, 2012
   VALUE = 2700 pF
   TOLERANCE = 20%
   RATED VOLTAGE = 16V



5. Click the ***Add Part with the next Temporary Part_Number*** button  to generate the new part.

Next you will add manufacturing data for the new part.

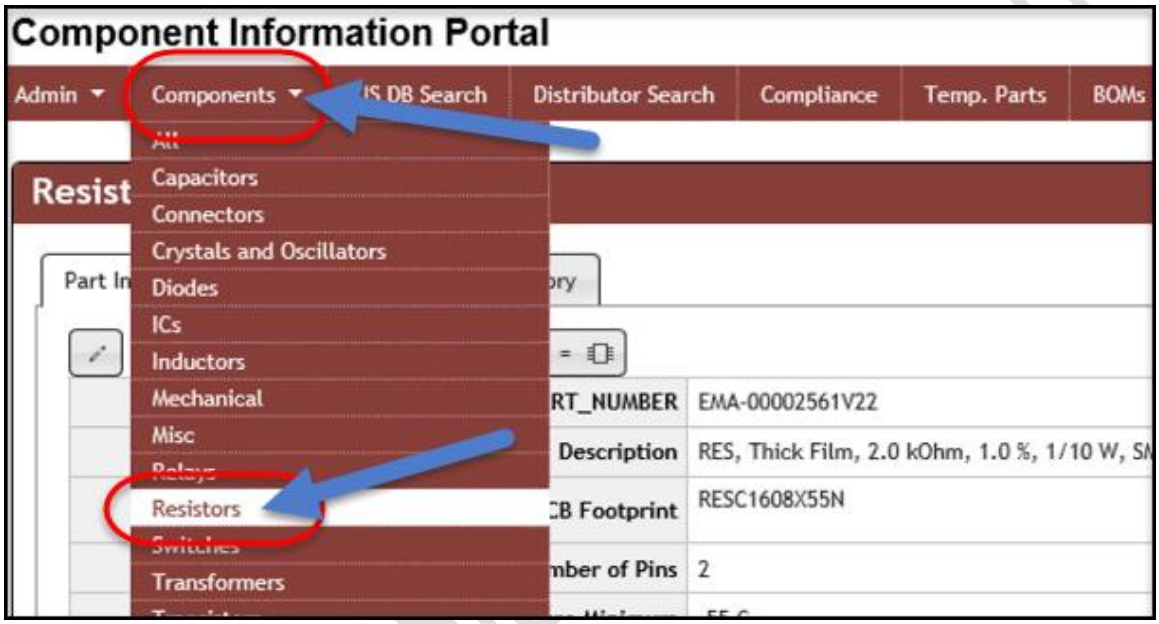
6. Scroll down to the ***Add Manufacturer Part*** area. Enter ***MFR-NAME-01*** in the ***Manufacture*** field.
7. Enter ***MFR-PN-01b*** in the ***Manufacturer PN*** field.
8. Click the ***Add*** button to update the manufacturing data.

When you add a “real” part to your database you will most likely spend more time adding actual manufacturing information and other relevant data. The sample we used in the previous exercise was to demonstrate how to input the data.

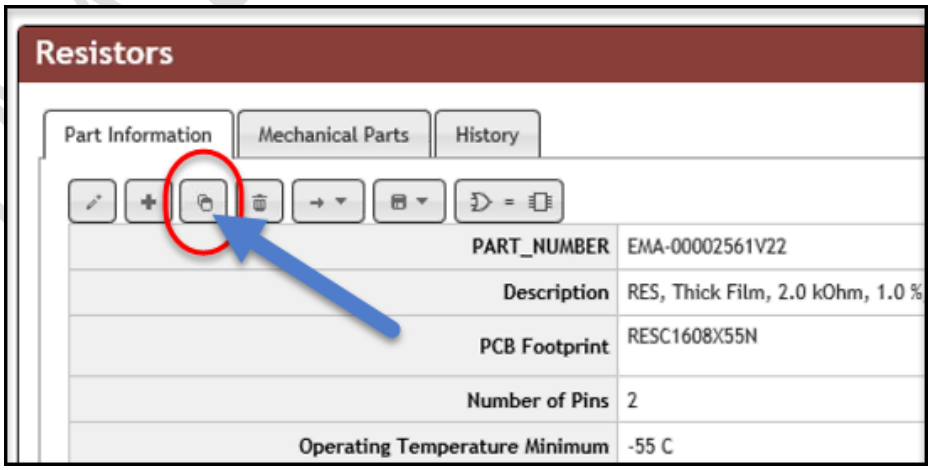
Lab 1-8: Copying an Existing Part

Use the Copy feature to copy an existing part that meets most of the criteria for a new part you want to generate. In the next exercise, you will generate a new resistor from an existing one.

1. Select ***Components > Resistors***. The Part Data for the first resistor in the Resistors table will appear.



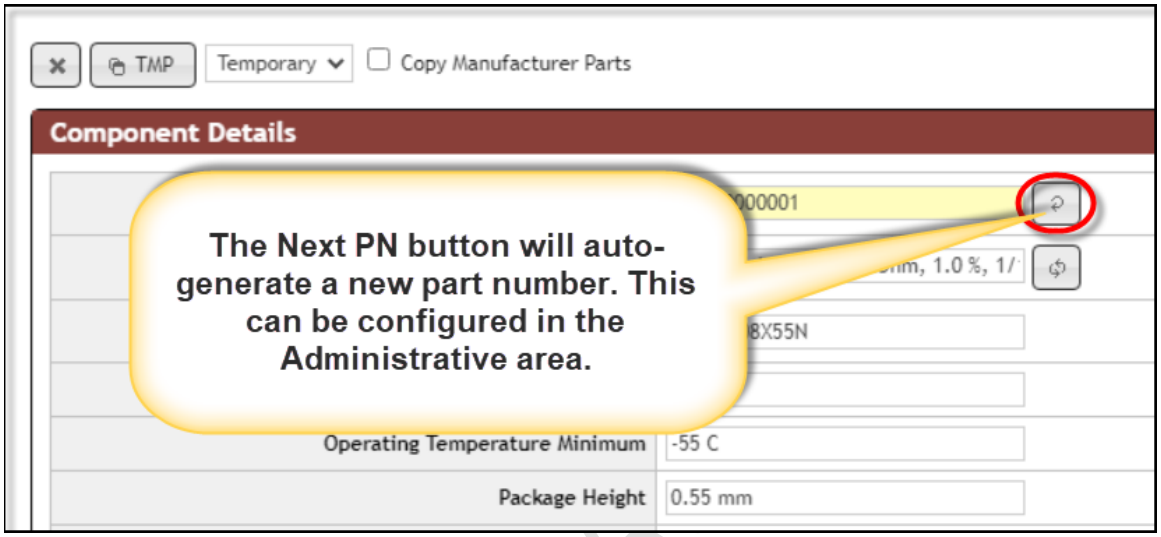
2. Click on the ***Copy*** button. You will alter several field parameters.



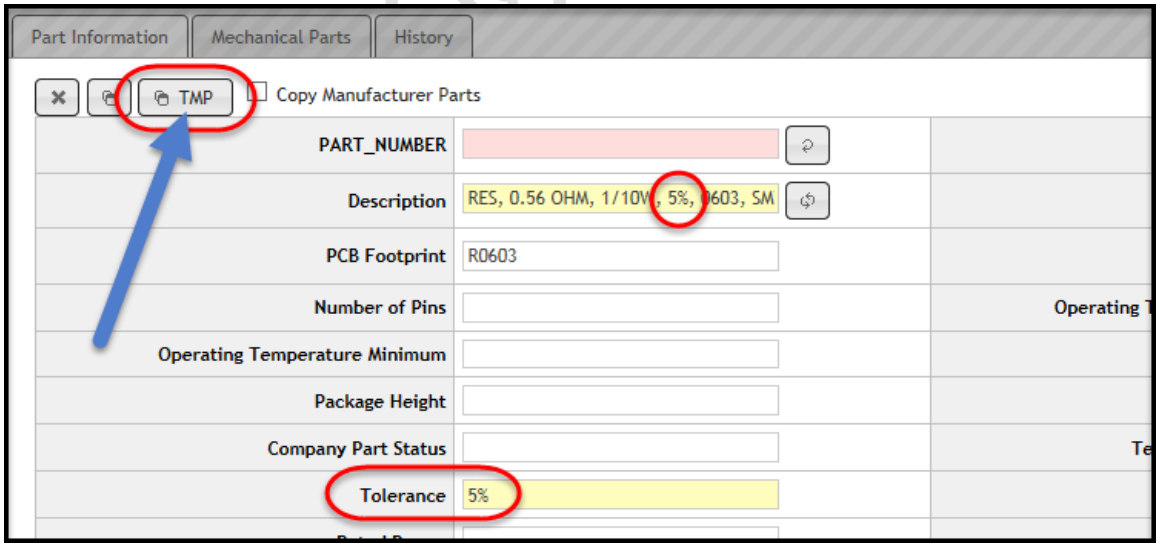
3. The part is now in edit mode. In the ***Description*** field, change the ***Tolerance*** to ***5%***.

4. In the ***Tolerance*** field, add ***5%***.

There are a couple of ways you could add this part in its new, updated state. Selecting the **PART_NUMBER > Next PN** button would auto-generate a new part number. This would be determined by Administrative settings that can be set to use CIP's internal part numbering scheme. (**NOTE:** You will not be using this option for this part).



Another way to add the part is to assign it a TMP number. This will allow the part to be validated prior to assigning it a formal part number. This is recommended when copying a part from an existing part.



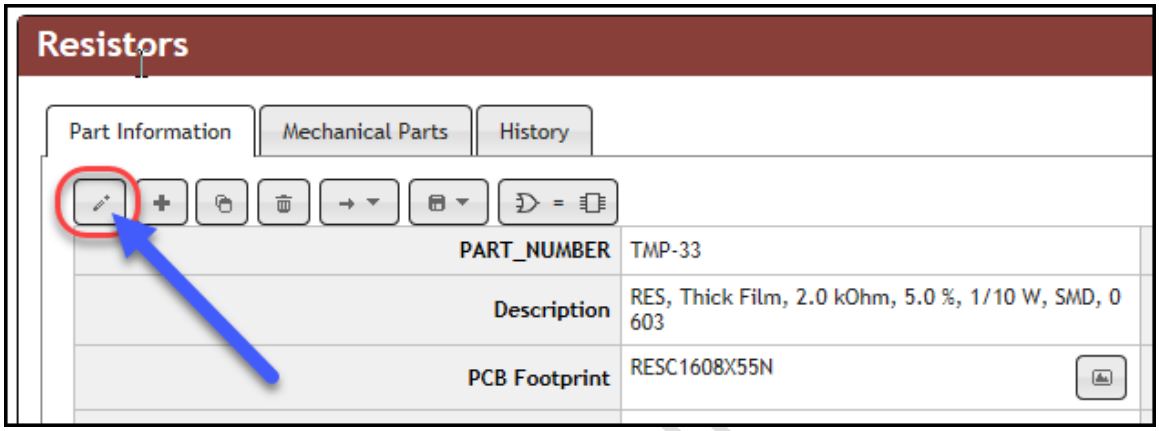
5. Click the ***TMP*** button to generate the new part with a TMP number.

NOTE: Because you copied this part from an existing part it assumes the Part Type of the original part.

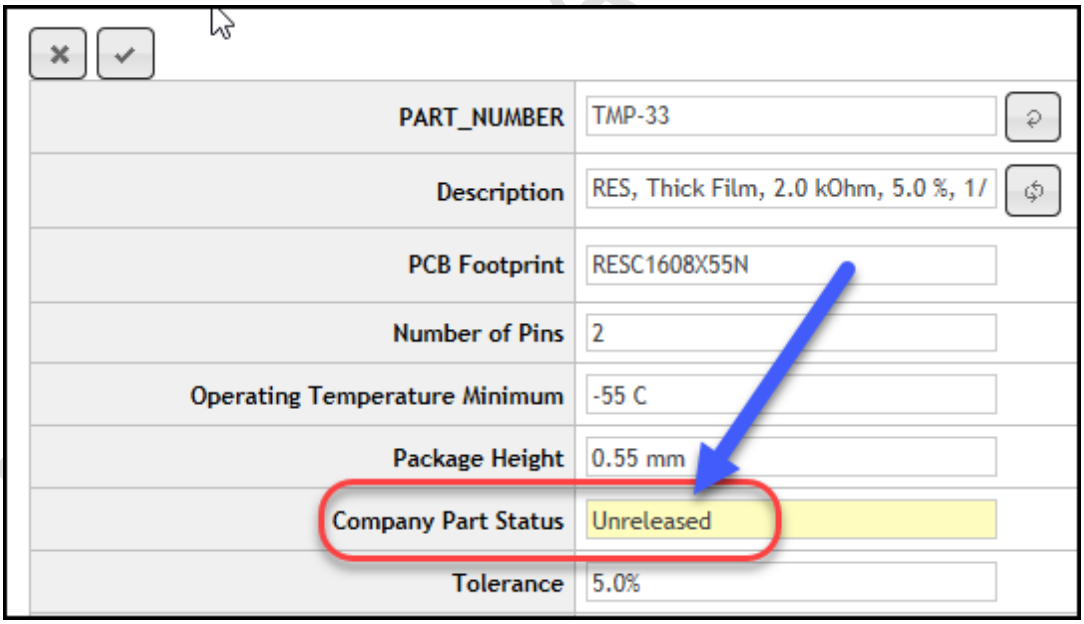
Lab 1-9: Tracking Part History

Changes to parts can be tracked using the **History** feature. History can be checked for both Part Information and the Manufacturing Parts.

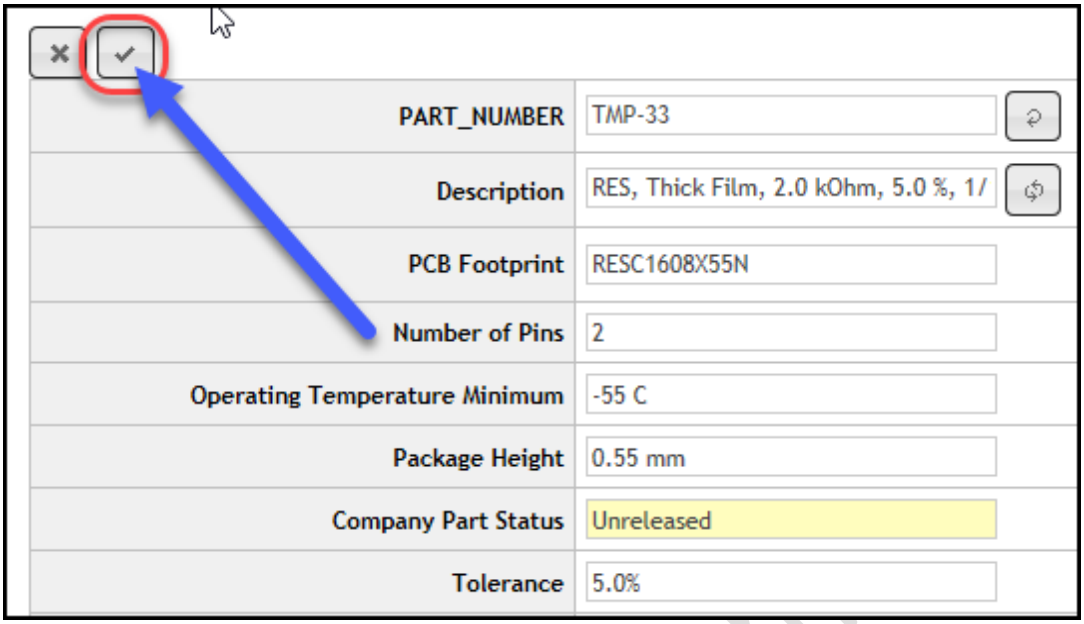
1. Using the part you just generated, select ***Edit*** to make a change to the part.



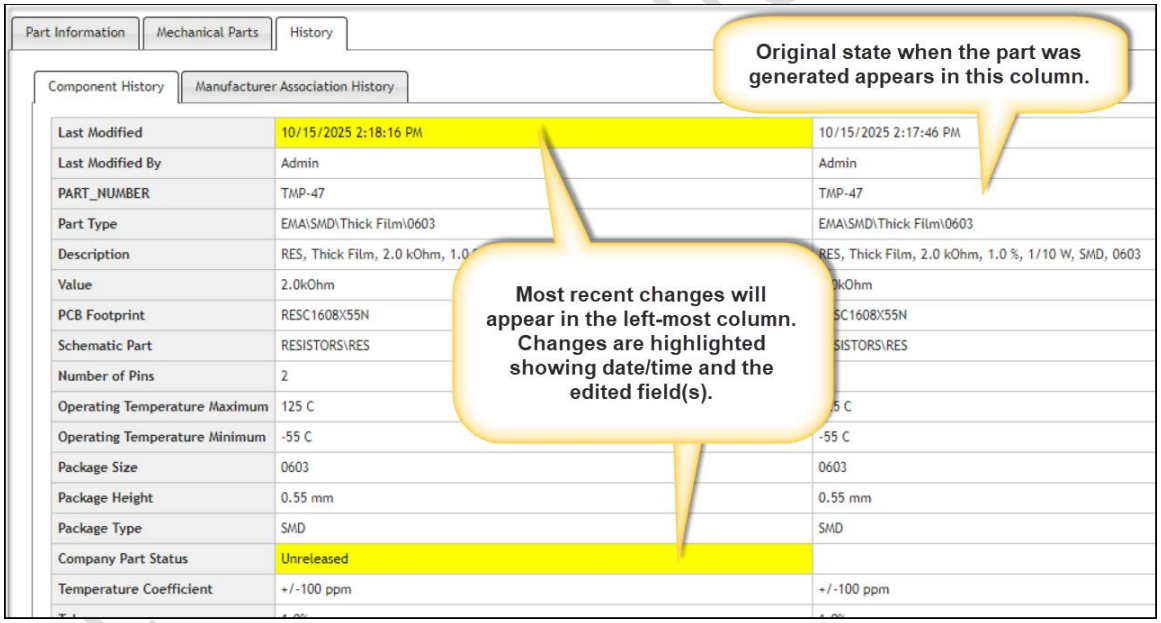
2. In the ***Company Part Status*** field, enter ***Unreleased***.



3. Click the ***Save*** button to save the change.

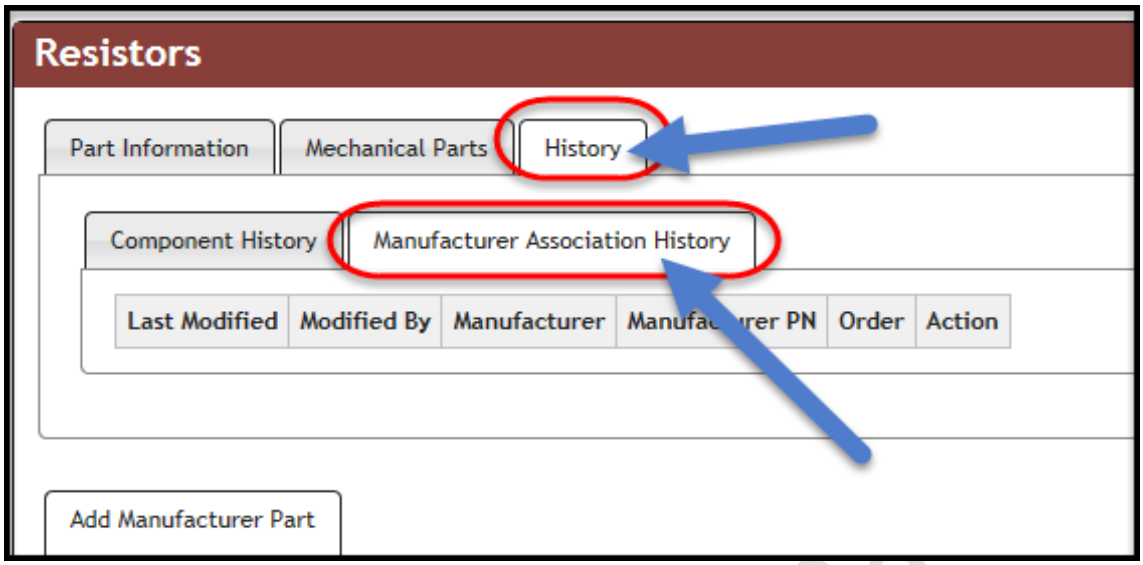


4. Select the ***History*** tab.



Checking Manufacturing History

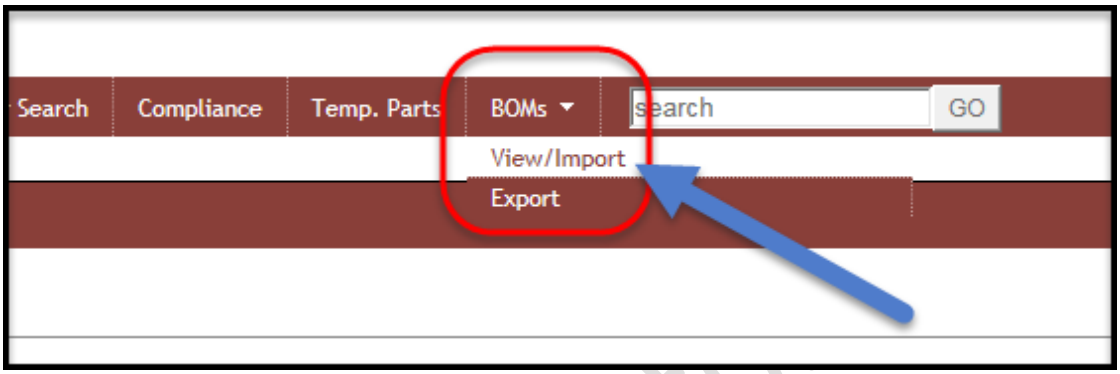
Manufacturing history can be tracked for changes. The **Manufacturer Association History** tab displays this information.



Lab 1-10: BOM Import

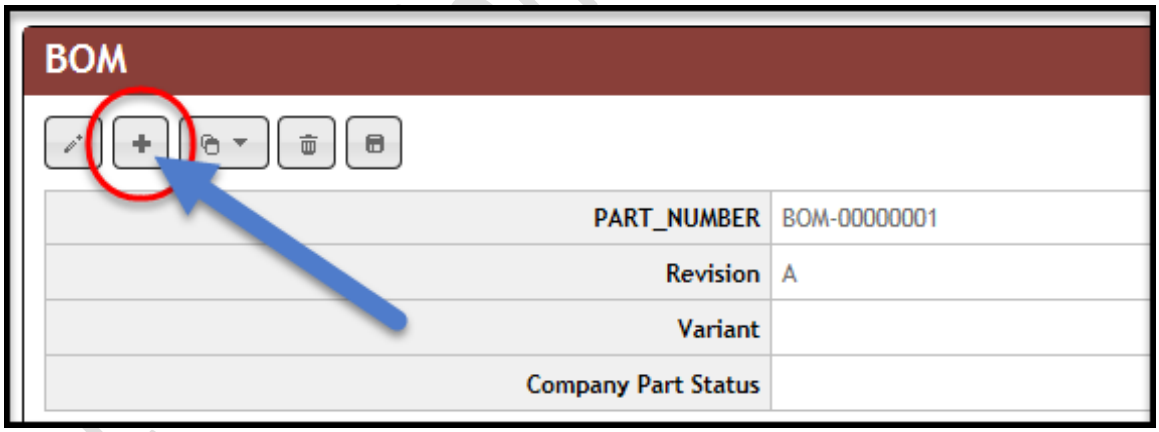
In this lab you will learn how to import Bills of Materials into CIP. BOMs must be in a CSV format in order to be imported.

1. Select ***BOMs > View/Import***.

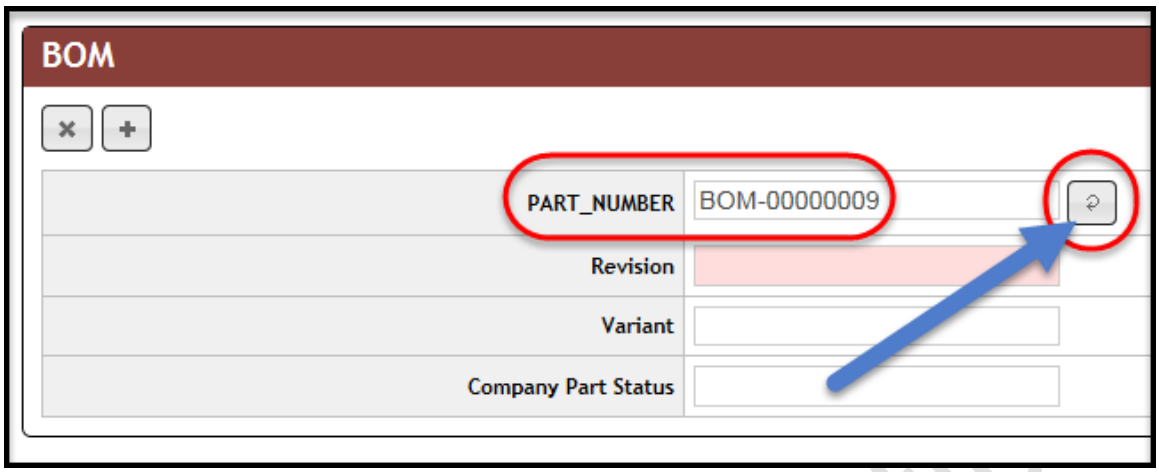


In the next steps you will be importing a sample BOM. The BOM has already been exported from OrCAD Capture CIS using the ***Reports > CIS Bill of Materials > Standard*** command and saved as CSV files.

2. Click on the ***Plus +*** sign to start a new BOM import.



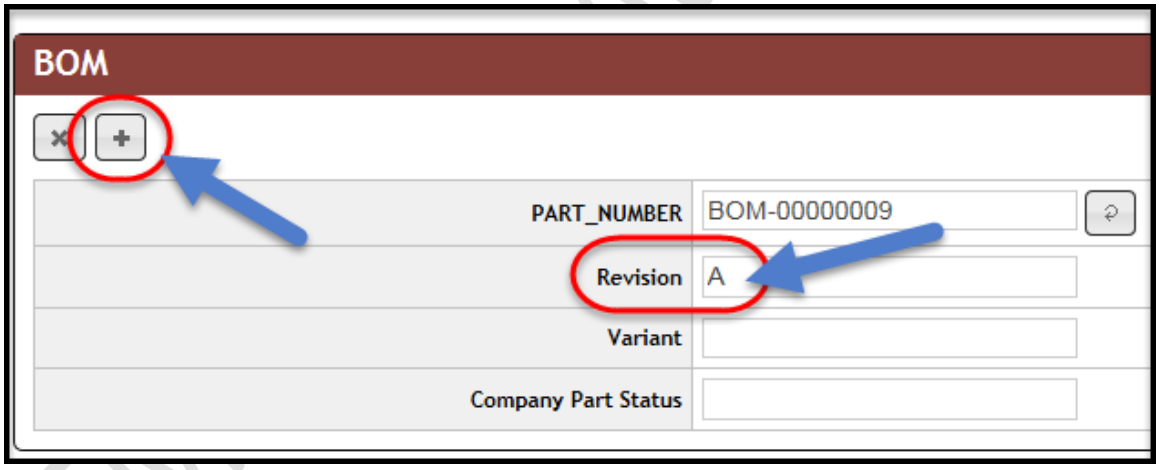
3. In the ***PART_NUMBER*** field, click the ***Next_PN*** button to add a BOM part number.
   Note: The BOM number that is assigned may be different from the one showing in the image below.



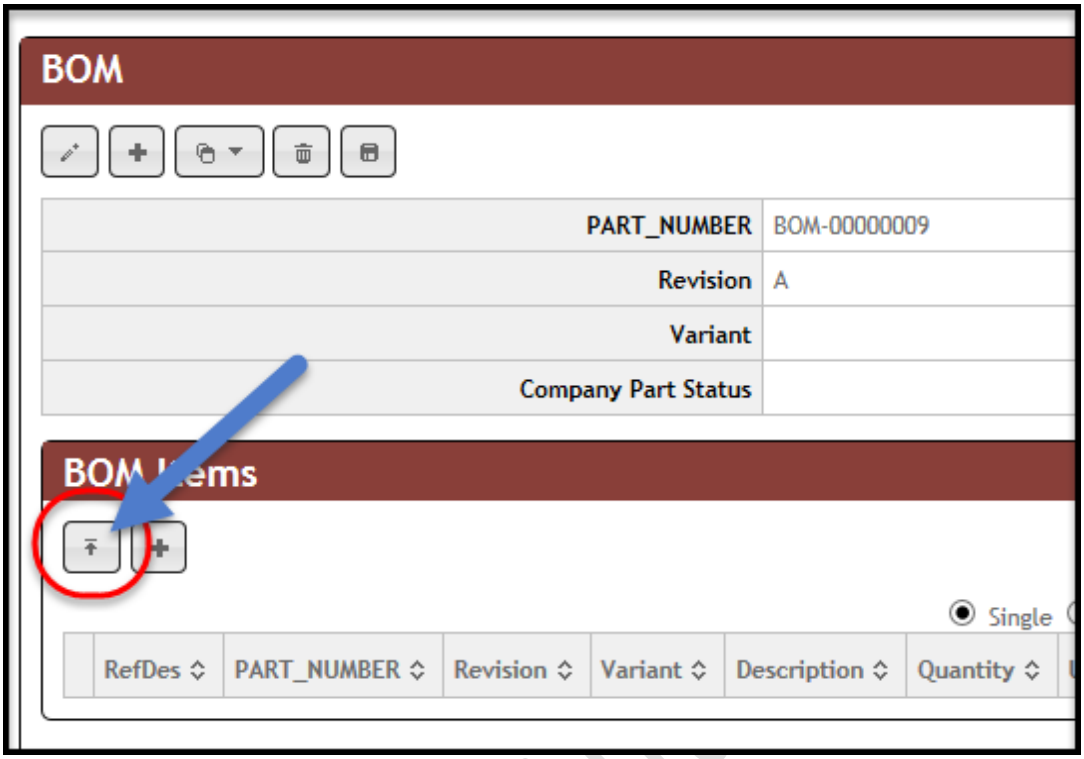
In a typical setting the BOM part number will already be established and can be retrieved for upload. In this exercise, you will use the auto-part numbering system to identify the BOM part number prior to uploading.

Additionally, there are other fields you can edit, such as description, cost, and build, to help identify what the BOM represents.

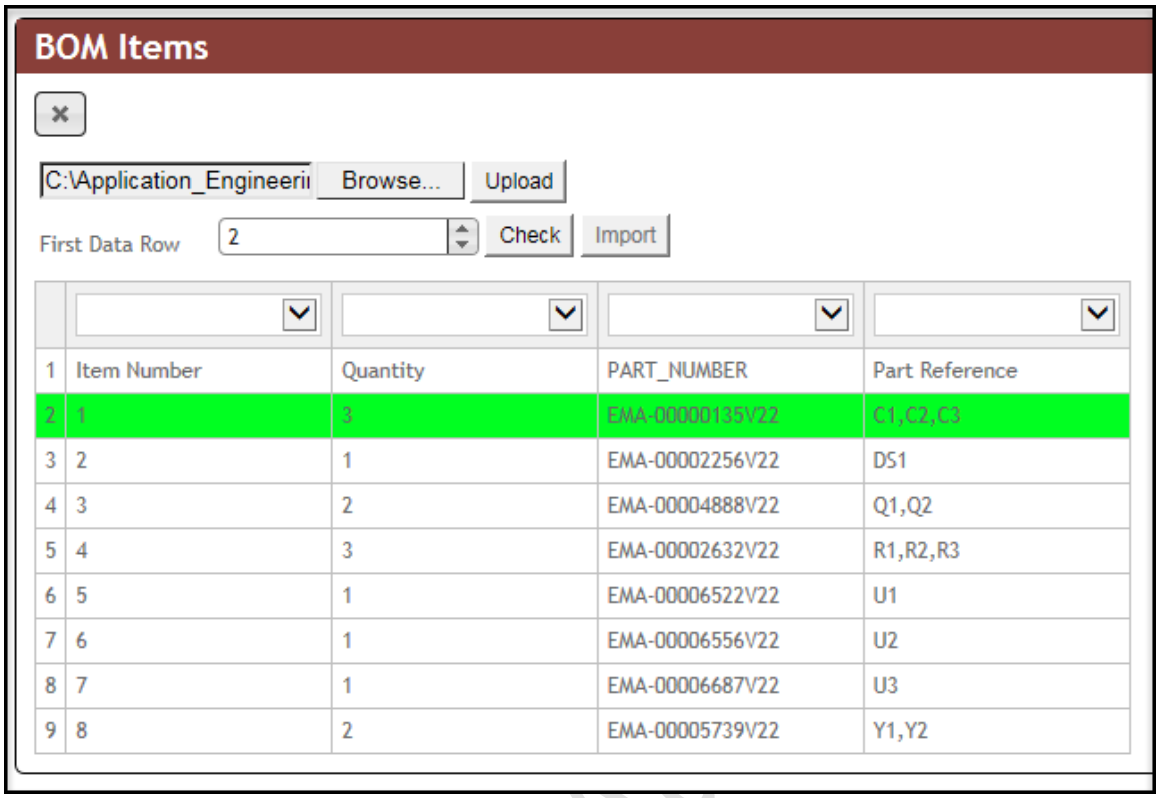
4. In the Revision area type ***A***. When finished, click the ***Plus +*** sign to enter the BOM.



5. Click on the ***Import Children*** button to import that BOM.



6. Browse to and select `C:\EMA_Training\CIS_CIP_Usage_23.1\Board2_BOM.CSV`.
7. Select ***Upload***.

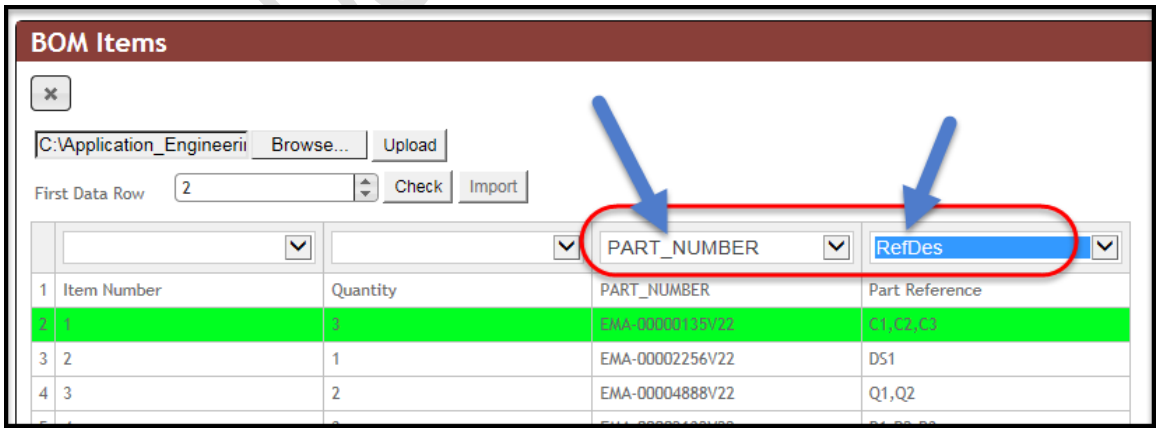


1	Item Number	Quantity	PART_NUMBER	Part Reference
2	1	3	EMA-00000135V22	C1,C2,C3
3	2	1	EMA-00002256V22	DS1
4	3	2	EMA-00004888V22	Q1,Q2
5	4	3	EMA-00002632V22	R1,R2,R3
6	5	1	EMA-00006522V22	U1
7	6	1	EMA-00006556V22	U2
8	7	1	EMA-00006687V22	U3
9	8	2	EMA-00005739V22	Y1,Y2

Adding Headers to the BOM

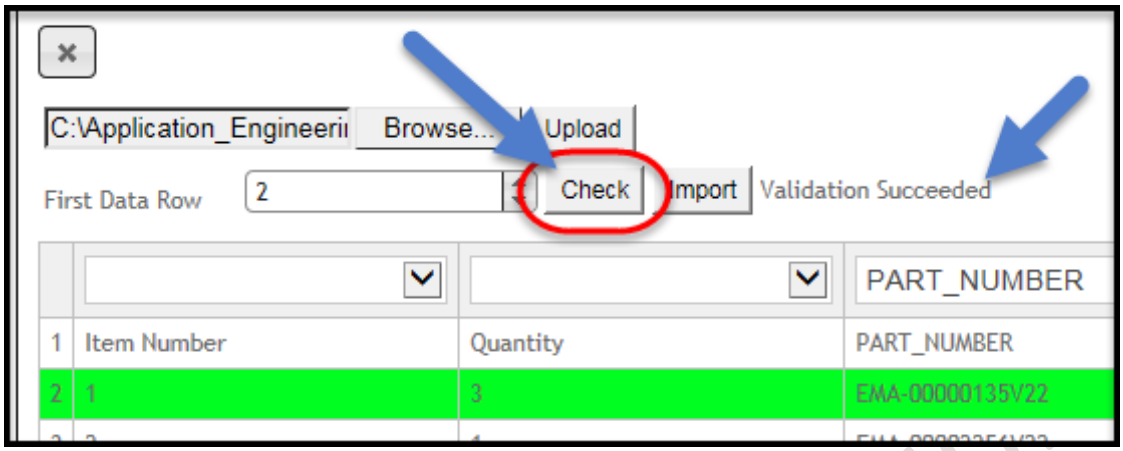
Next you will add the headers and complete the import.

1. In the ***PART_NUMBER*** column, click the dropdown to select ***PART_NUMBER***.
2. In the ***Part Reference*** column, click the dropdown and select ***RefDes***.

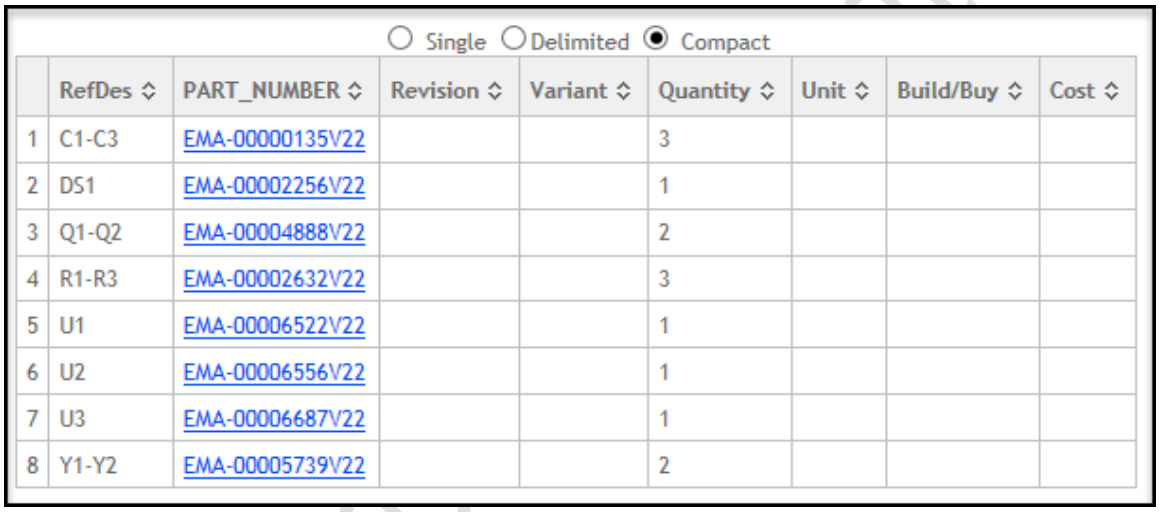


1	Item Number	Quantity	PART_NUMBER	Part Reference
2	1	3	EMA-00000135V22	C1,C2,C3
3	2	1	EMA-00002256V22	DS1
4	3	2	EMA-00004888V22	Q1,Q2

3. Once the headers have been added, click ***Check*** to validate prior to importing.



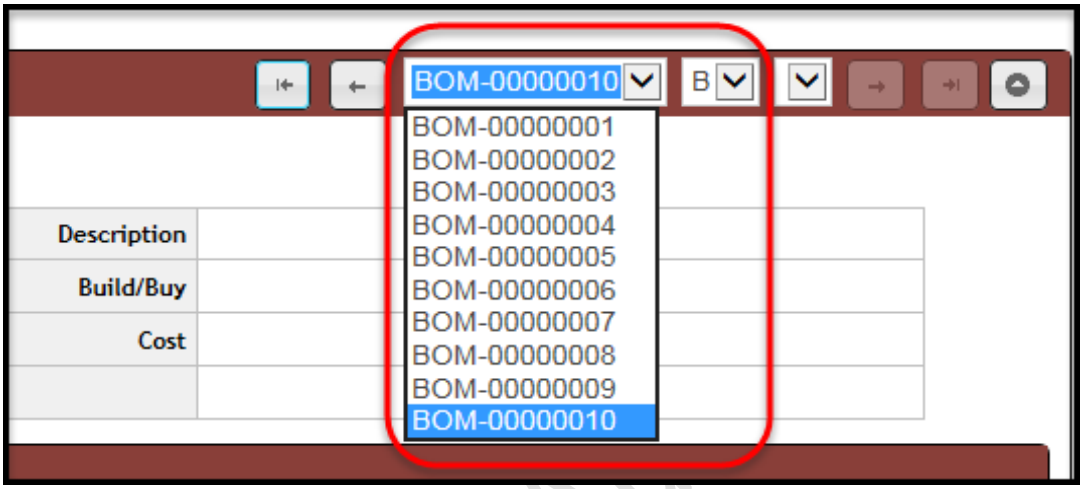
4. Click ***Import*** to import the BOM into CIP. The BOM will upload as shown below.



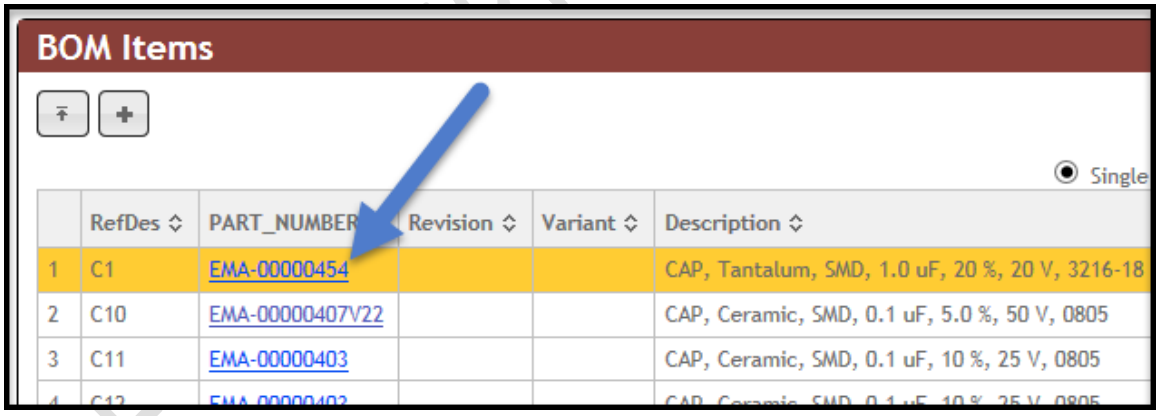
	RefDes	PART_NUMBER	Revision	Variant	Quantity	Unit	Build/Buy	Cost
1	C1-C3	EMA-00000135V22			3			
2	DS1	EMA-00002256V22			1			
3	Q1-Q2	EMA-00004888V22			2			
4	R1-R3	EMA-00002632V22			3			
5	U1	EMA-00006522V22			1			
6	U2	EMA-00006556V22			1			
7	U3	EMA-00006687V22			1			
8	Y1-Y2	EMA-00005739V22			2			

Lab 1-11: Where Used

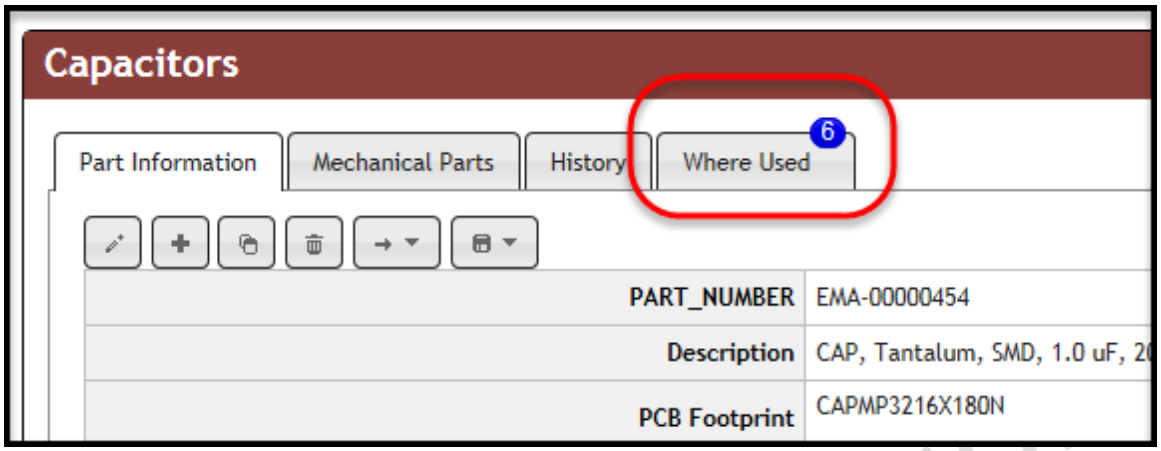
Now that a BOM has been uploaded into CIP you will be able to see it in a dropdown area in the upper right area of CIP. Note that the number of BOMs you see in the list on your training machine may differ than what appears in the following image.



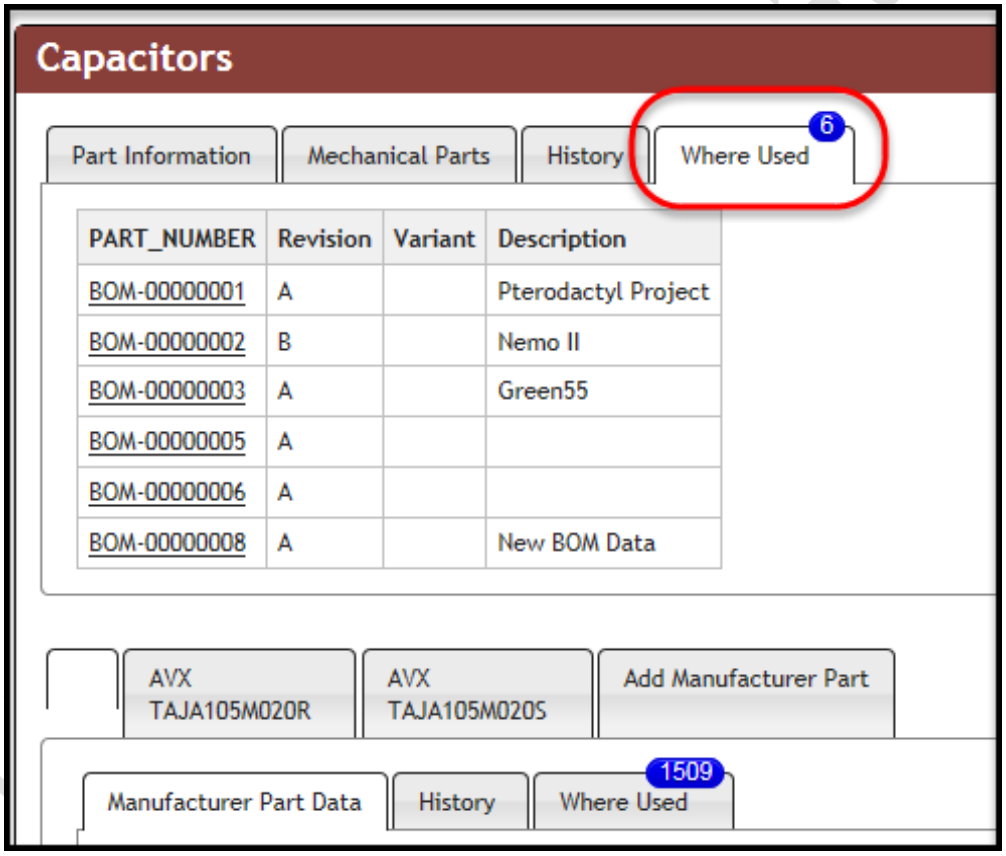
1. From the BOM dropdown, select the BOM you just uploaded.
2. Click on the first part number in the list.



3. When the Part Detail appears, click on the ***Where Used*** tab.



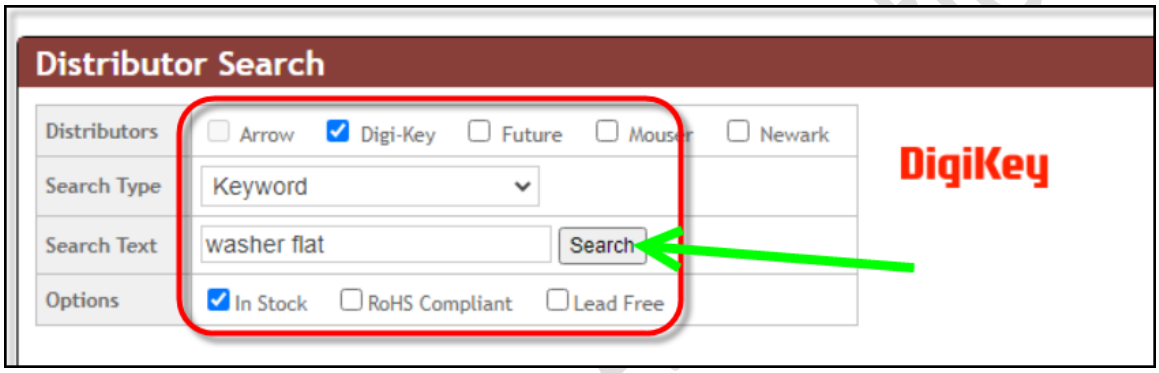
The BOM you previously uploaded should appear in the list of Where Used items.



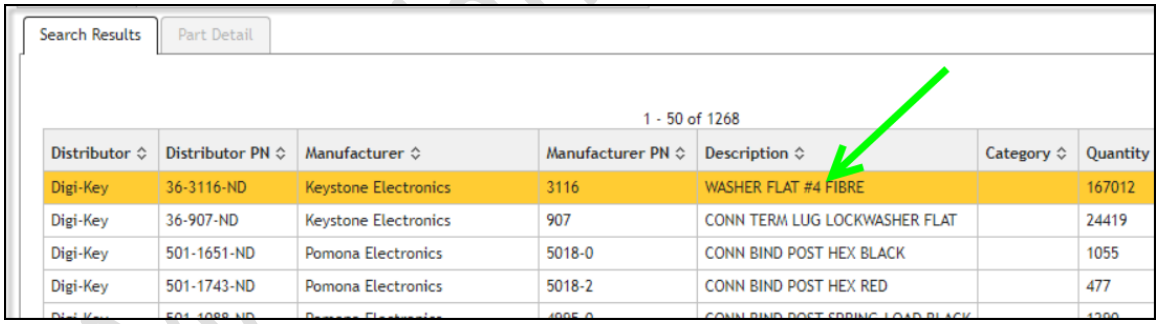
Lab 1-12: Adding Mechanical Parts

Just as you can create new parts using the distributor portal you can also copy existing parts to generate new ones. For this lab you will use the distributor portal to generate new mechanical parts.

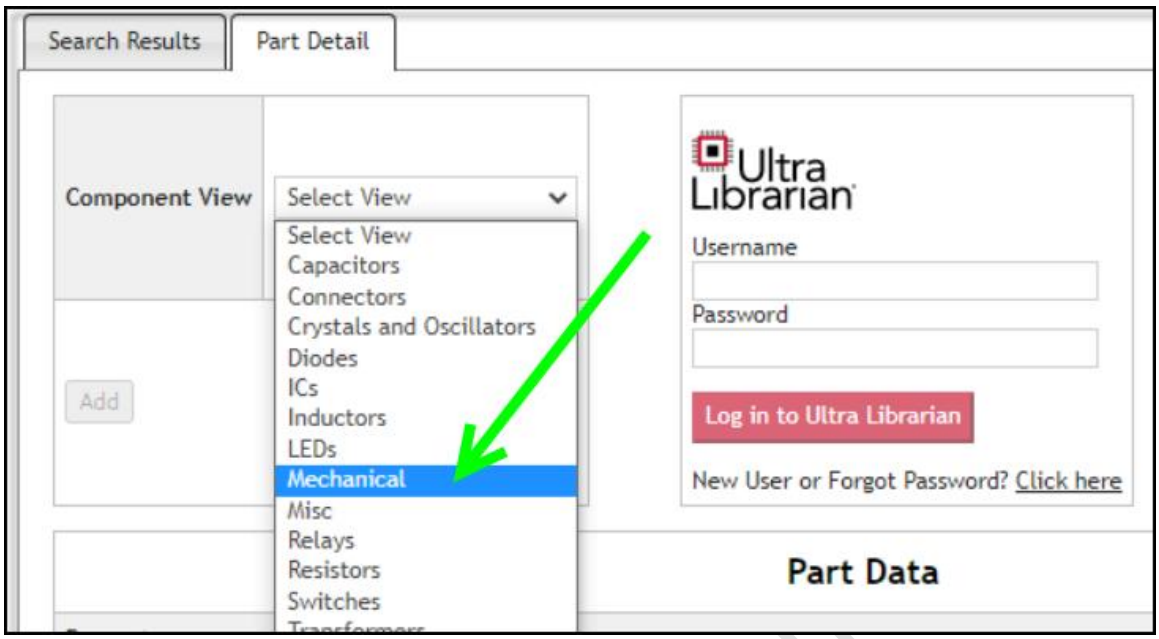
1. Select the ***Distributor Search*** tab and select ***Digi-Key***.
2. Leave the ***Search Type*** set to ***Keyword***.
3. In the ***Search Text*** area type **washer flat**.
4. In the ***Options*** area, check ***In Stock***, then click the ***Search*** button to begin the search.



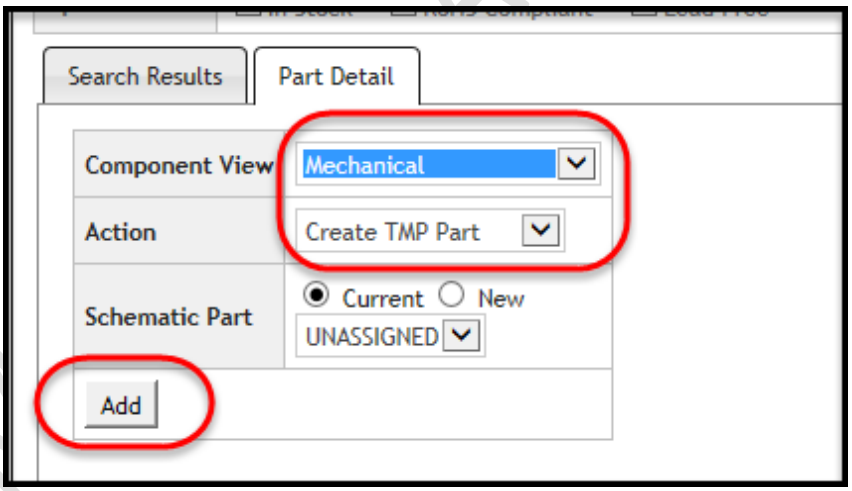
5. Select any #4 washer from the search results.



6. In the ***Part Detail*** tab, use the dropdown to select ***Mechanical***.



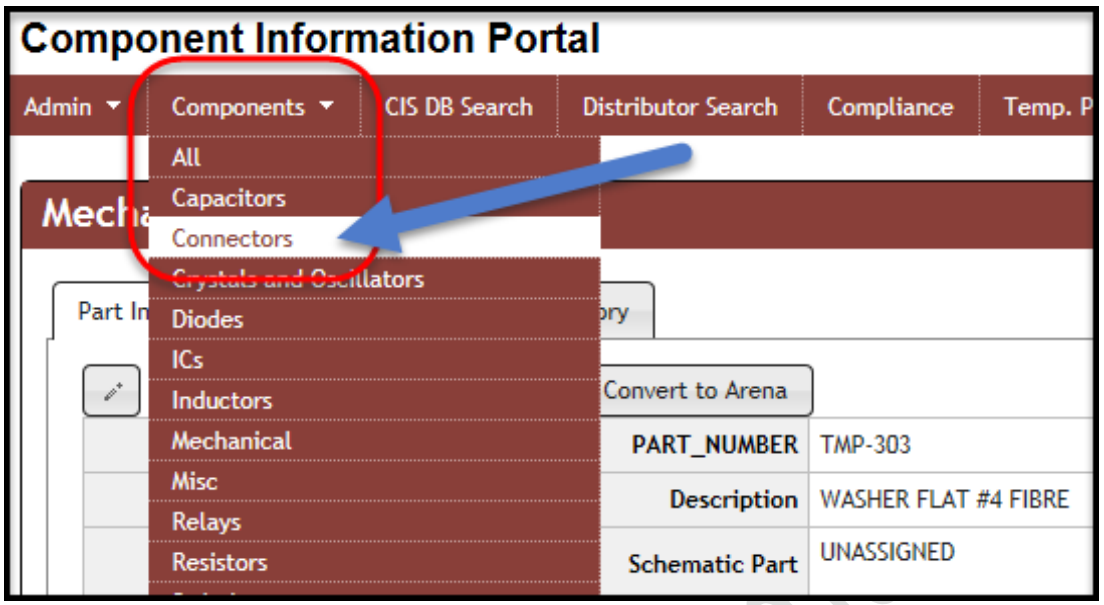
7. Leave the ***Action*** set to ***Create TMP Part*** and ***Add*** the part. **Note**: If you had a graphical representation of mechanical parts available in the libraries you could associate the part with the schematic part graphic. In this instance we will leave it UNASSIGNED.



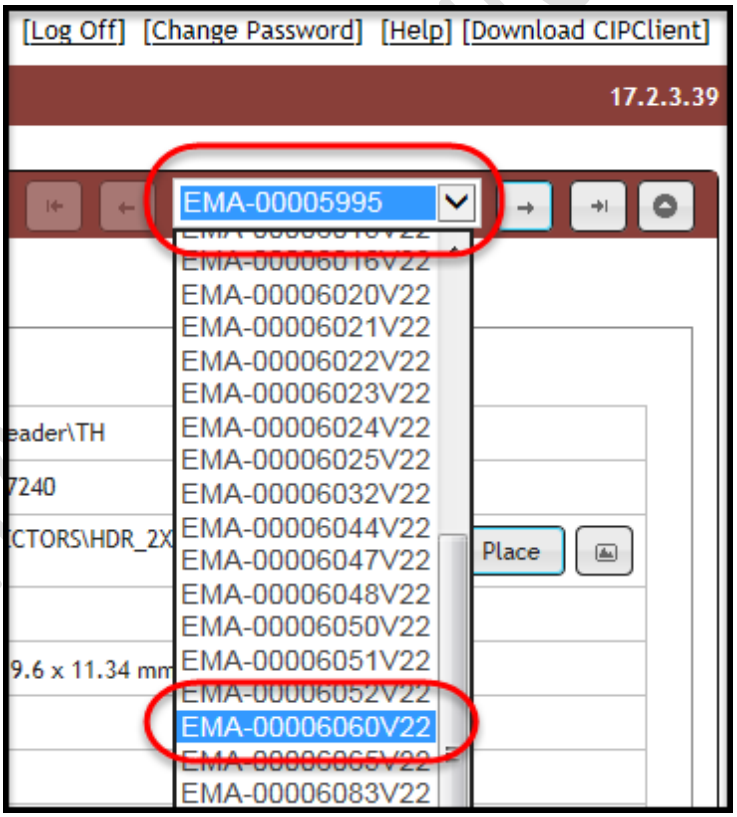
Associating a Mechanical Part with a Component

Once a mechanical part has been added to the database it can be associated to a component so that each time that component is added to a schematic, the associated mechanical part will travel with it.

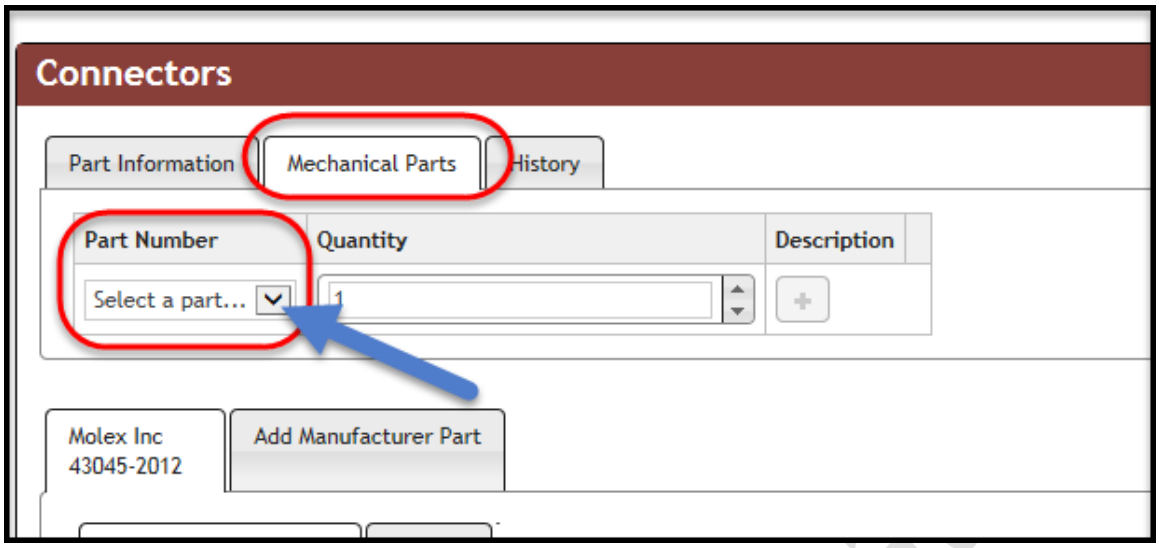
1. From the ***Components*** tab, use the dropdown to select ***Connectors***.



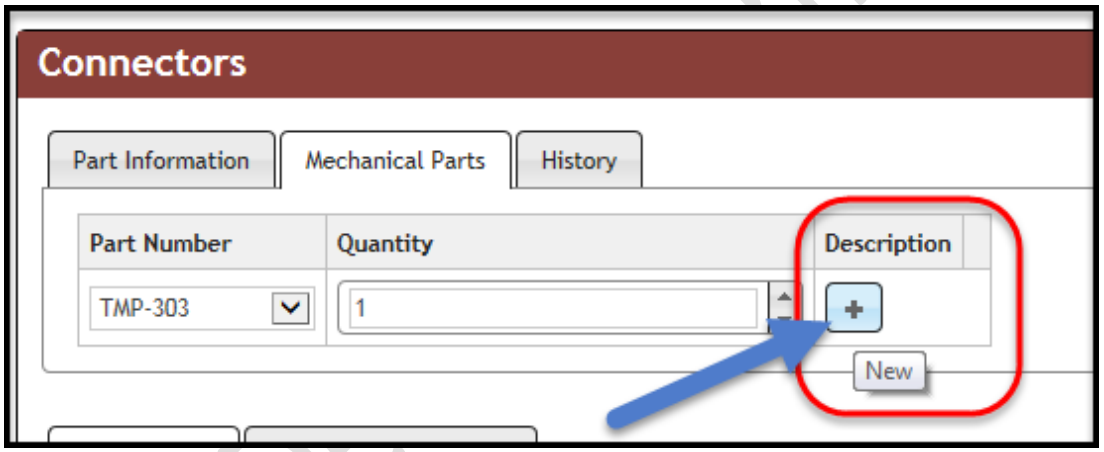
2. Using the pat number dropdown, scroll down to locate and select part number ***EMA-00006060V22***.



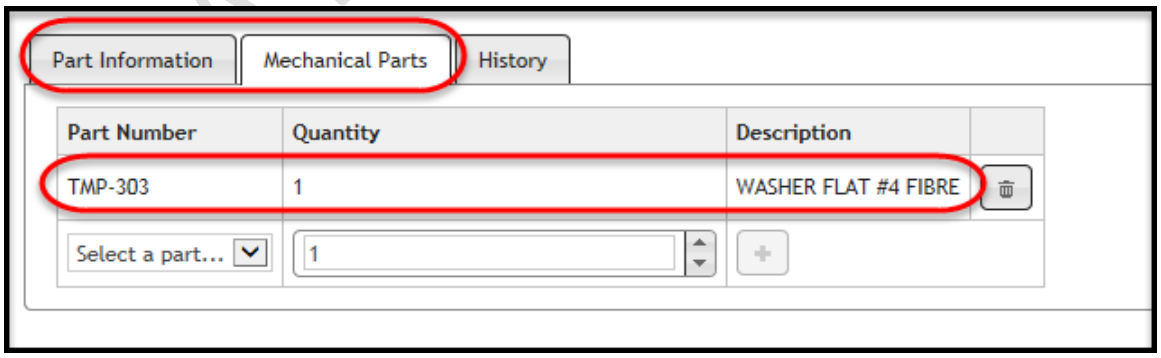
3. Once the part information opens, select the ***Mechanical Parts*** tab. Use the ***Part Number*** dropdown to select the washer you just added. **Note**: It will probably have a TMP part number.



4. Click the ***New*** button in the ***Description*** area.



Notice the part is now added to the original component.



Each time you place the part on a schematic, the mechanical parts that have been associated to the original part will automatically become part of the design. Once mechanical parts have been associated with other parts, they can be exported to a BOM.

Lesson 2: OrCAD Capture CIS Fundamentals

OrCAD Capture's Component Information System, or CIS, is a part management tool that organizes and coordinates the part placement process. CIS provides access to a local centralized database containing all the relevant information for parts used within a schematic design. Primary input for part definition typically comes from the Engineering department. However, other groups, such as Manufacturing and Purchasing, also need to provide and/or edit part information.

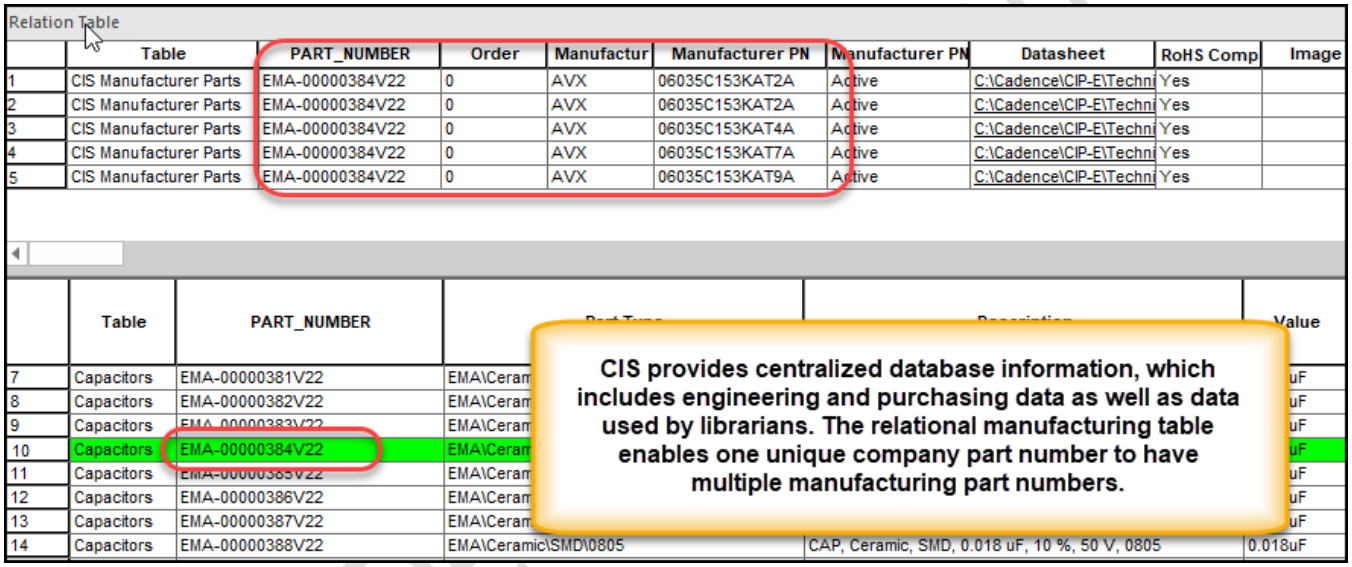


	Table	PART_NUMBER	Order	Manufactur	Manufacturer PN	Manufacturer PN	Datasheet	RoHS Comp	Image
1	CIS Manufacturer Parts	EMA-00000384V22	0	AVX	06035C153KAT2A	Active	C:\Cadence\CIP-E\Techni	Yes	
2	CIS Manufacturer Parts	EMA-00000384V22	0	AVX	06035C153KAT2A	Active	C:\Cadence\CIP-E\Techni	Yes	
3	CIS Manufacturer Parts	EMA-00000384V22	0	AVX	06035C153KAT4A	Active	C:\Cadence\CIP-E\Techni	Yes	
4	CIS Manufacturer Parts	EMA-00000384V22	0	AVX	06035C153KAT7A	Active	C:\Cadence\CIP-E\Techni	Yes	
5	CIS Manufacturer Parts	EMA-00000384V22	0	AVX	06035C153KAT9A	Active	C:\Cadence\CIP-E\Techni	Yes	

	Table	PART_NUMBER			Value
7	Capacitors	EMA-00000381V22	EMA\Ceram		uF
8	Capacitors	EMA-00000382V22	EMA\Ceram		uF
9	Capacitors	EMA-00000383V22	EMA\Ceram		uF
10	Capacitors	EMA-00000384V22	EMA\Ceram		uF
11	Capacitors	EMA-00000385V22	EMA\Ceram		uF
12	Capacitors	EMA-00000386V22	EMA\Ceram		uF
13	Capacitors	EMA-00000387V22	EMA\Ceram		uF
14	Capacitors	EMA-00000388V22	EMA\Ceramic\SMD\0805	CAP, Ceramic, SMD, 0.018 uF, 10 %, 50 V, 0805	0.018uF

CIS provides a core solution in overall library management and facilitates a streamlined flow for product design from schematic, to simulation, to PC board.

Benefits of CIS

The primary benefit of using CIS is its ability to organize engineering data into a centralized, usable database. Additional benefits to consider are:

- Automatically consolidates part definition input from various departments
- Automatically enter new parts into the approval process
- Updates schematics to reflect changes made to approved parts
- Ensures production designs only contain approved parts
- Ensures Bill of Materials contains the latest data
- Maintains a link between the schematic part and its database record
- Promotes data consistency and use of preferred parts

- Transfers selected data from the database to the schematic
- Ensures a unified and compact library
- Ensures part selection from the approved centralized library
- Promotes sharing and reuse of centralized parts information
- Eliminates duplication and separate part libraries

CIS Interface and Functionality

CIS streamlines the front-end design process into two graphical user interfaces, the CIS Explorer window and the Part Manager window.

CIS Explorer Window

The CIS Explorer window provides access to the parts that have been approved and reside in the centralized database. Parts can be searched, parametrics can be reviewed, schematic symbols and PCB footprints can be viewed, and parts can be placed in the schematic.

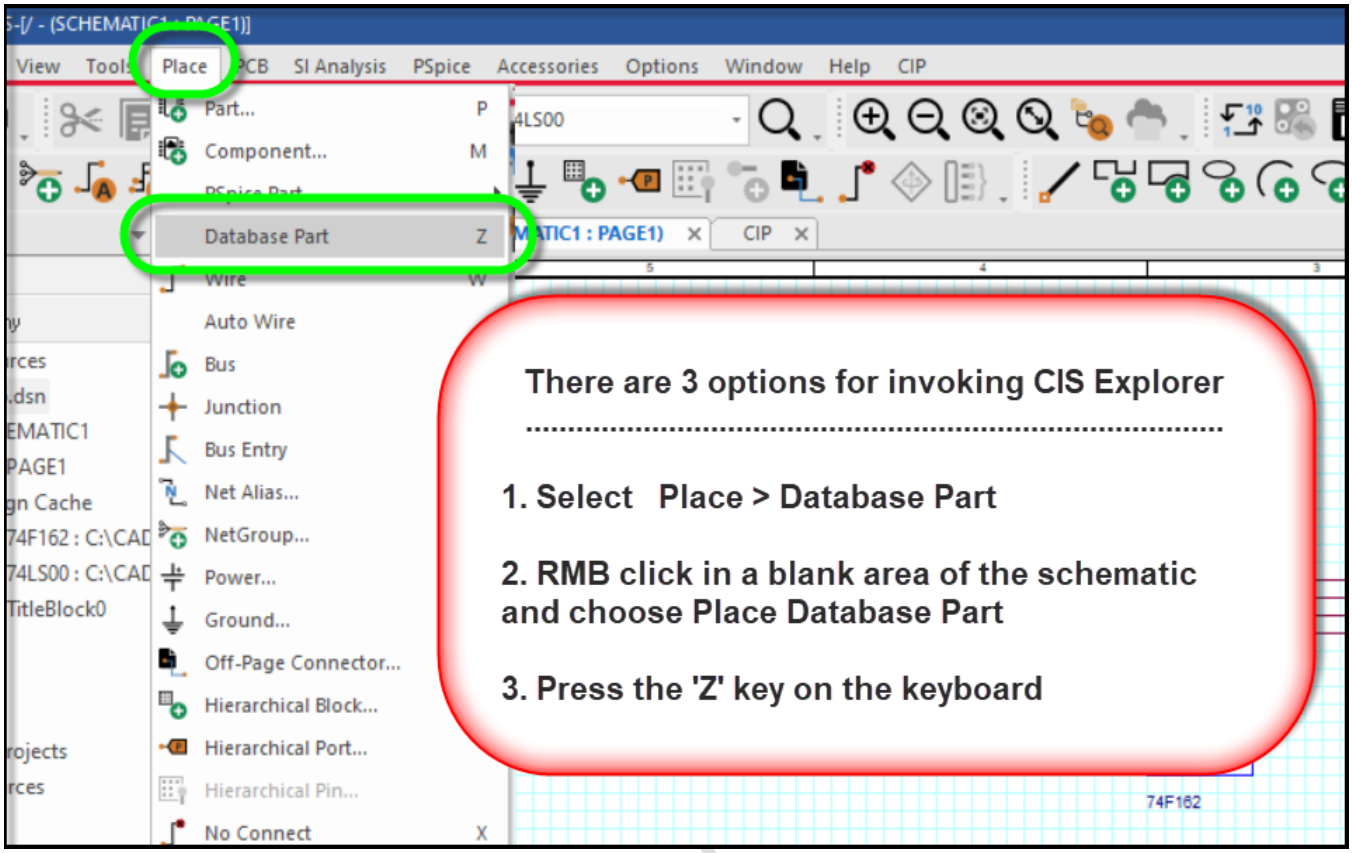
Part Manager Window

The Part Manager window keeps an accurate status of every part in a design. It provides a graphical interface where the status of each part will be indicated by a specific color. Parts can be linked to updated database parts and verified of their current status before going out for Bill of Materials. Variants can also be defined and maintained using the Part Manager window.

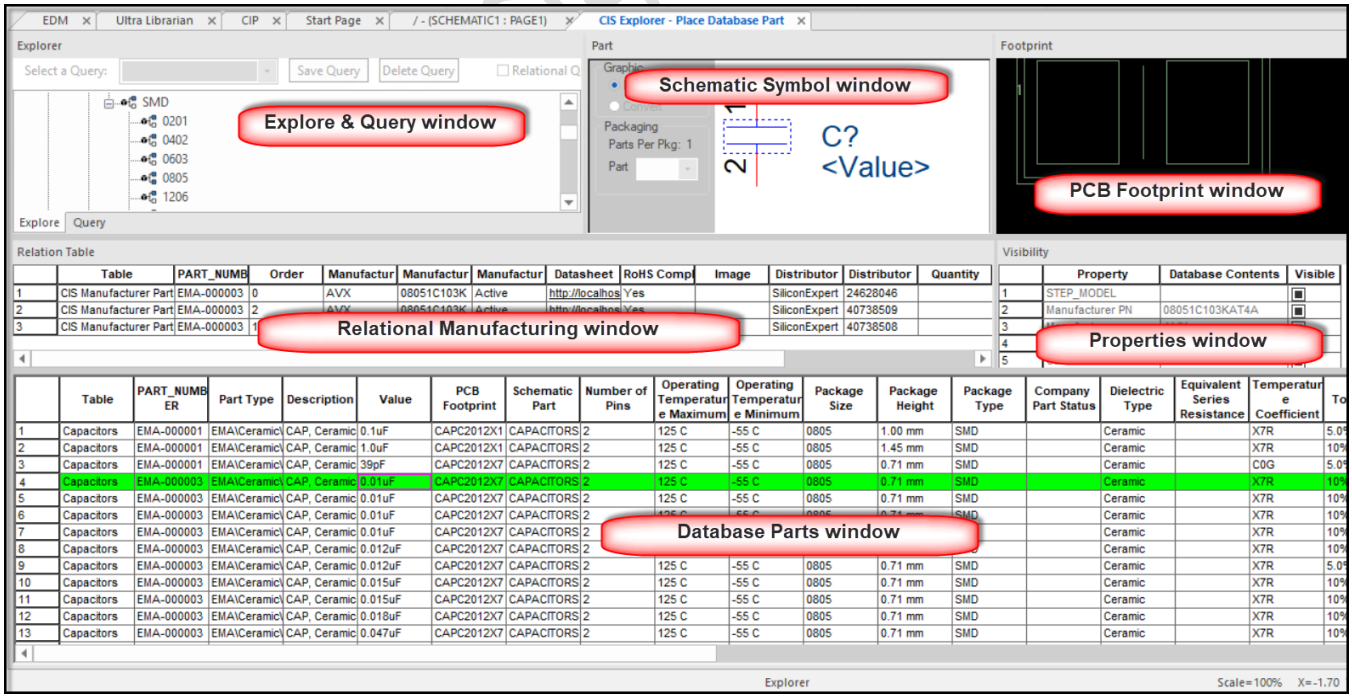
Accessing CIS Explorer

The CIS Explorer window can be accessed in several ways:

- Select ***Place > Database Part***
- Right click on any location in the schematic and select ***Database Part***
- From any location in the schematic, press the ***Z*** shortcut key

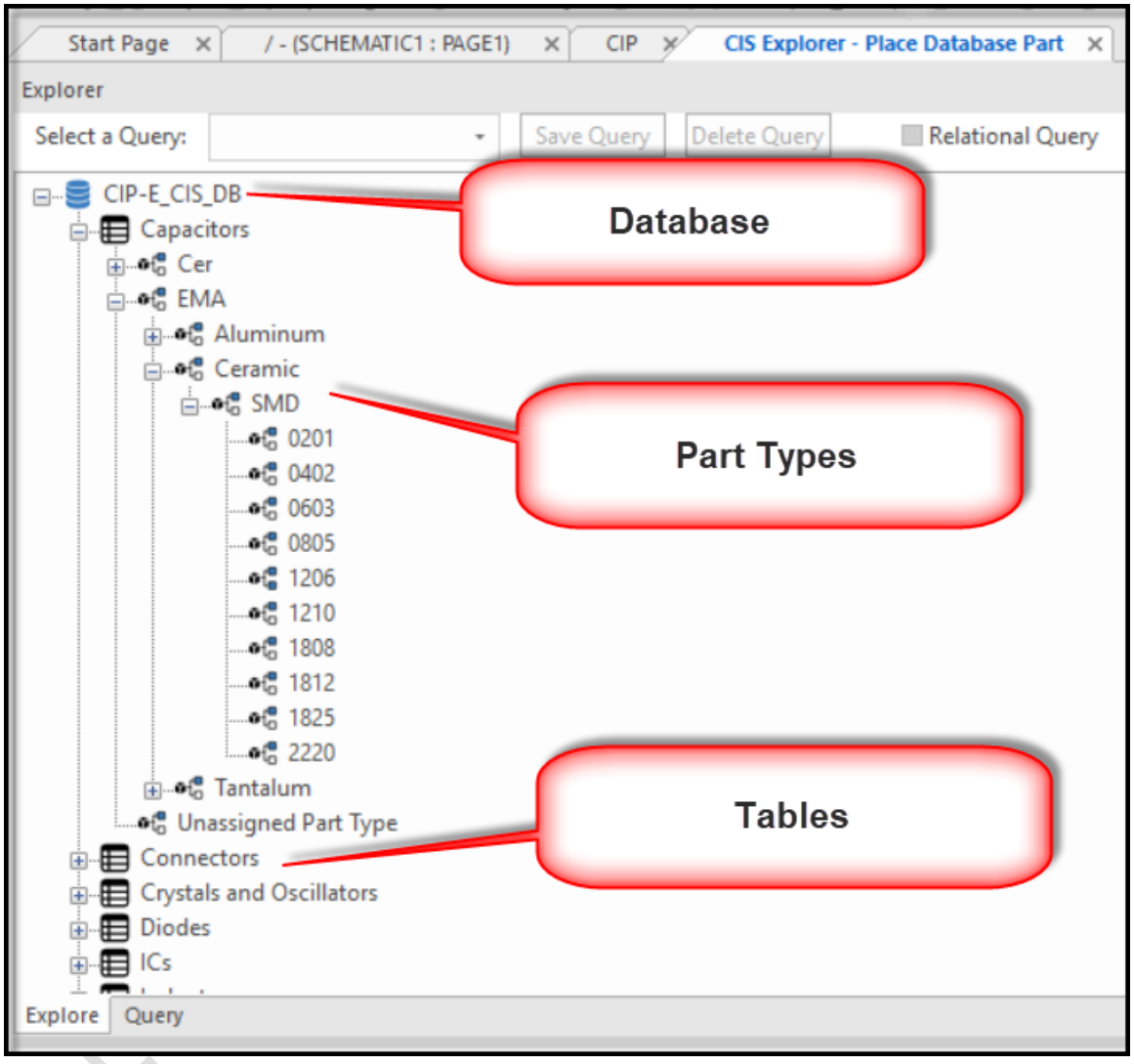


CIS Explorer Window



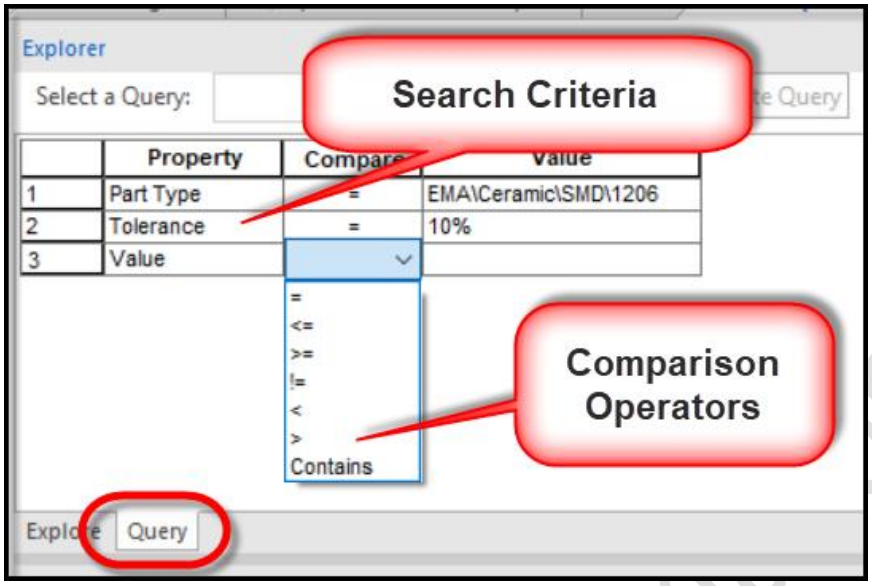
Explore Database and Query Window

There are two tabs in the database window, the **Explore** tab and the **Query** tab. The **Explore** tab allows browsing through the database hierarchy to find parts in their relevant part categories. The **Query** tab can be used to perform a refined search based on parametric data.



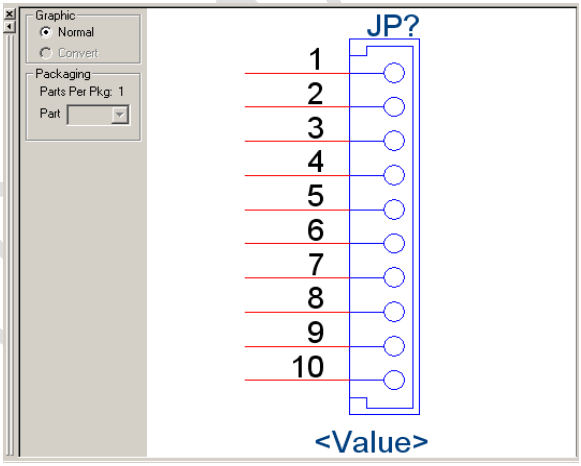
Explore Window Query Tab

The **Query** tab provides an efficient way to find parts and returns a list based on the search criteria. Three fields can be used to refine a search – Property, Compare, and Value. Once each field entry is filled, another row will be added enable refinement of the search. Queries can be saved for later use.



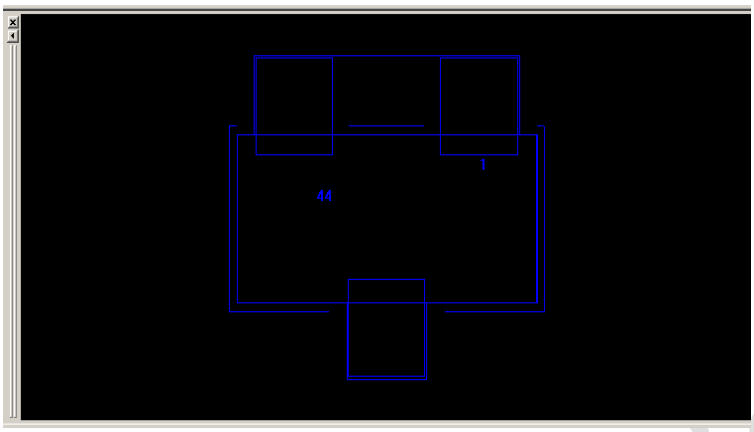
Parts Window

The **Parts** window displays the schematic part associated with a selected part. The **Parts** window can show the Normal graphic or the Convert Part (the DeMorgan Equivalent) as well as the specified part in a multiple part package.



PCB Footprint Window

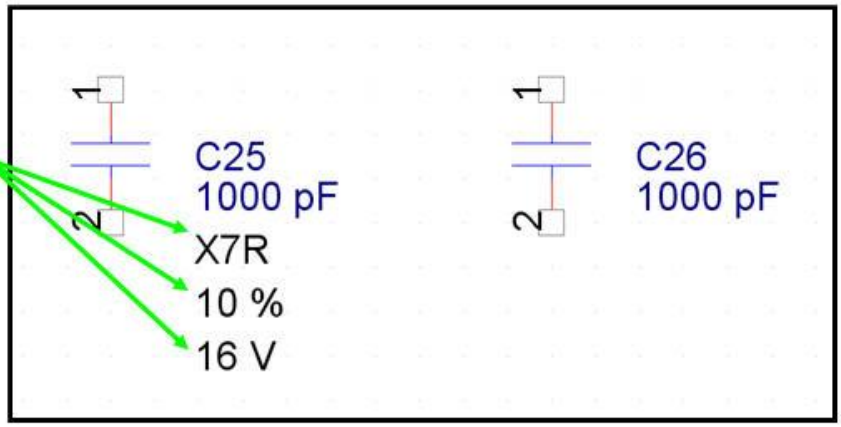
The **PCB Footprint** window displays the footprint for the currently selected part. The window can display only Allegro footprints or OrCAD Layout based footprints.



Visibility Window and Settings

The **Visibility** window displays the default settings for the visibility of part properties on the schematic page. This window can be used to override the default settings. Custom visibility settings can also be set for the current part.

	Property	Database Contents	Visible
1	CLASS	DISCRETE	☑
2	PSpiceTemplate		☑
3	Implementation Type	<none>	☑
4	Implementation		☑
5	Rated Voltage	16 V	☑
6	Tolerance	10 %	☑
7	Temperature Coefficie	X7R	☑
8	Dielectric Type	Ceramic	☑
9	Package Size	0402	☑
10	PCB Footprint	CAPC1005X56N	☑
11	Value	1000 pF	☑
12	Description	CAP, Ceramic, SMD, 10	☑
13	PART_NUMBER	EMA-00000522V22	☑
14	Schematic Part	CAP	☑
15	Part Type	EMA\Ceramic\SMD\0402	☑
16	Number of Pins	2	☑
17	Operating Temperatur	125 C	☑
18	Operating Temperatur	-55 C	☑
19	Package Height	0.56 mm	☑
20	Package Type	SMD	☑
21	Company Part Status		☑
22	Equivalent Series Resi		☑
23	Device Type		☑



There are four possible visibility settings.

- ☑ CIS displays the property with the part on the schematic page
- ☐ CIS does not display the property with part on the schematic
- ☑ CIS does not modify the property visibility. If the property is not set to be transferred from the database to the schematic it will not be visible
- ■ CIS does not allow this property to be visible with the schematic part on the design

Database Parts Window

The **Database Parts** window displays results in spreadsheet format. In the table, the columns can be sorted by clicking on the field header, column widths can be adjusted, the order of the columns can be changed by dragging their position, and columns can be hidden or unhidden.

	Table	PART_NUMBER	Part Type	Description	Value	PCB Footprint	Schematic Part	Number of Pins	Operating Temperature Maximum	Oper Temp e Min
1	Capacitors	EMA-00000465	EMA\Ceramic\	CAP, Ceramic, SMD, 10 uF, 10 %, 6.3 V,	10uF	CAPC3216X94N	CAPACITORS\CAP	2	85 C	-55 C
2	Capacitors	EMA-00000501	EMA\Ceramic\	CAP, Ceramic, SMD, 100 pF, 10 %, 100	100pF	CAPC3216X94N	CAPACITORS\CAP	2	125 C	-55 C
3	Capacitors	EMA-00000502V22	EMA\Ceramic\	CAP, Ceramic, SMD, 100 pF, 5.0 %, 100	100pF	CAPC3216X94N	CAPACITORS\CAP	2	125 C	-55 C
4	Capacitors	EMA-00000503	EMA\Ceramic\	CAP, Ceramic, SMD, 100 pF, 5.0 %, 50 V	100pF	CAPC3216X94N	CAPACITORS\CAP	2	125 C	-55 C
5	Capacitors	EMA-00000545V22	EMA\Ceramic\	CAP, Ceramic, SMD, 1000 pF, 10 %, 100	1000pF	CAPC3216X94N	CAPACITORS\CAP	2	125 C	-55 C
6	Capacitors	EMA-00000546V22	EMA\Ceramic\	CAP, Ceramic, SMD, 1000 pF, 10 %, 50	1000pF	CAPC3216X94N	CAPACITORS\CAP	2	125 C	-55 C
7	Capacitors	EMA-00000548V22	EMA\Ceramic\	CAP, Ceramic, SMD, 1000 pF, 5.0 %, 50	1000pF	CAPC3216X94N	CAPACITORS\CAP	2	125 C	-55 C
8	Capacitors	EMA-00000549V22	EMA\Ceramic\	CAP, Ceramic, SMD, 1000 pF, 10 %, 100	1000pF	CAPC3216X94N	CAPACITORS\CAP	2	125 C	-55 C
9	Capacitors	EMA-00000569V22	EMA\Ceramic\	CAP, Ceramic, SMD, 1200 pF, 5.0 %, 50	1200pF	CAPC3216X94N	CAPACITORS\CAP	2	125 C	-55 C

Placing a Database Part

Within the CIS Explorer window, when a desired part has been found, select the part in the part table. When it turns green it is ready for placement on the schematic. Double clicking on the part will attach the part to the cursor, allowing it to be placed it in the schematic.

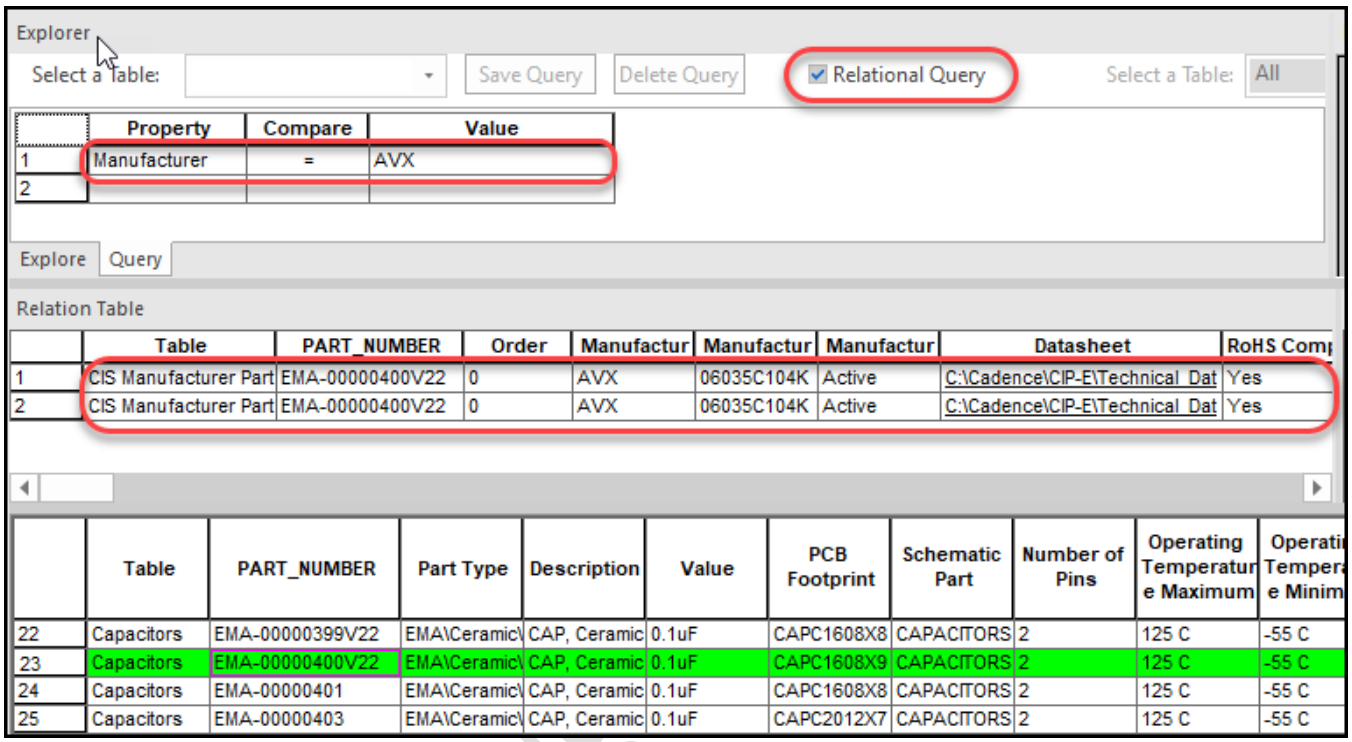
	Table	PART_NUMBER	Part Type	Description	Value	PCB Footprint	Schematic Part	Number of Pins	Operating Temperature Maximum	Operating Temperature Minimum	Pack Siz
1	Capacitors	EMA-00000128V22	EMA\Ceramic\	CAP, Ceramic	0.1 uF	CAPC2012X100N	CAP	2	125 C	-55 C	0805
2	Capacitors	EMA-00000137V22	EMA\Ceramic\	CAP, Ceramic	1.0 uF	CAPC2012X145N	CAP	2	125 C	-55 C	0805
3	Capacitors	EMA-00000150V22	EMA\Ceramic\	CAP, Ceramic	39 pF	CAPC2012X71N	CAP	2	125 C	-55 C	0805
4	Capacitors	EMA-00000375V22	EMA\Ceramic\	CAP, Ceramic	0.01 uF	CAPC2012X71N	CAP	2	125 C	-55 C	0805
5	Capacitors	EMA-00000376V22	EMA\Ceramic\	CAP, Ceramic	0.01 uF	CAPC2012X71N	CAP	2	125 C	-55 C	0805
6	Capacitors	EMA-00000377V22	EMA\Ceramic\	CAP, Ceramic	0.01 uF	CAPC2012X71N	CAP	2	125 C	-55 C	0805

Browsing Part Properties

Browsable properties are set within the CIS configuration file. The property is generally used to open datasheets associated with parts in the database.

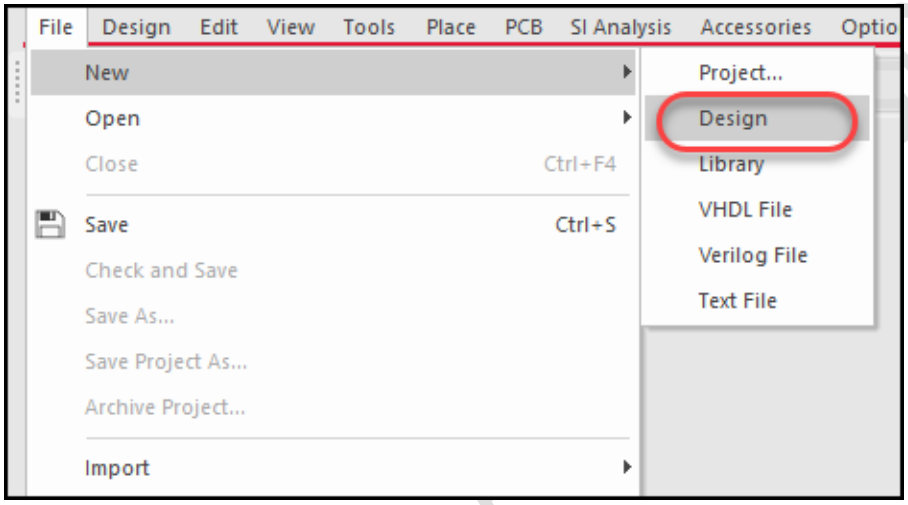
Relational Database Support

The relational support provided allows users to see a one-to-many relationship between the vendor table and the part table. The primary key is the **PART_NUMBER**. This allows for one corporate part number to be associated with multiple vendor part numbers.

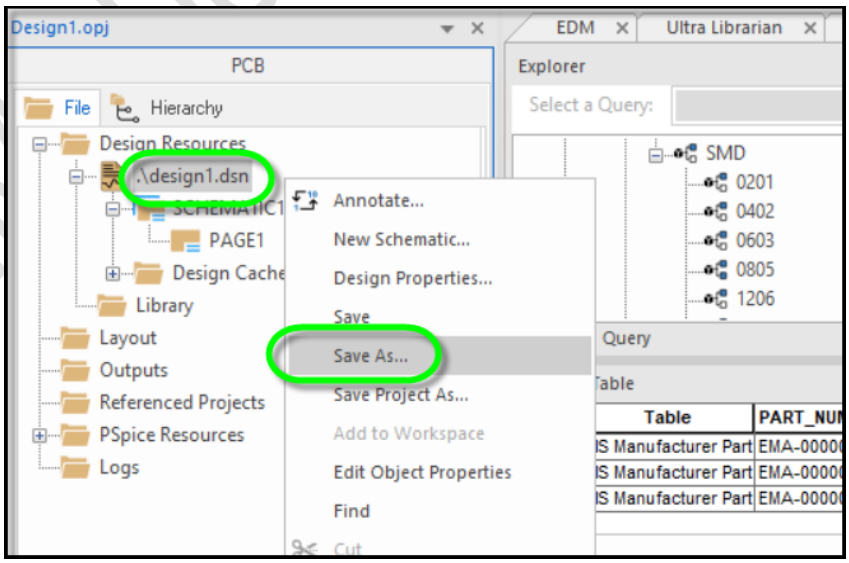


Lab 2-1: Creating a New Design in CIS

1. On your desktop, locate and double click the ***OrCAD Capture CIS*** icon
2. If the ***Product Choice*** window appears, select ***OrCAD Capture CIS*** and click ***OK***.
3. Click on ***File > New > Design***.

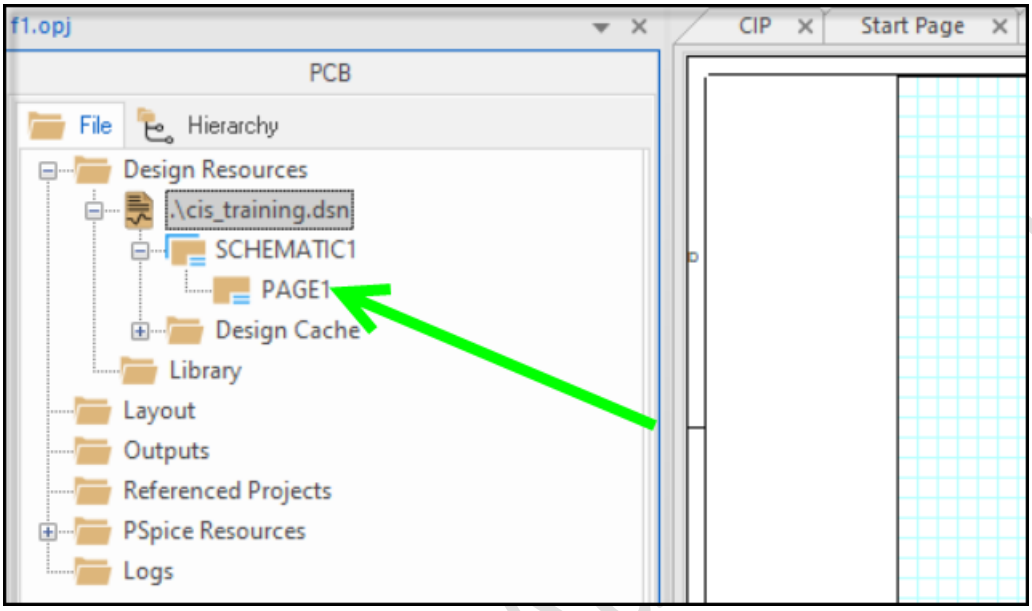


4. A new design will be created. Right click on the design and select ***Save As***.
5. Save the new design as `C:\EMA_Training\CIS_CIP_Usage_23.1\CIS_Training.dsn`. This design will be used in future exercises.

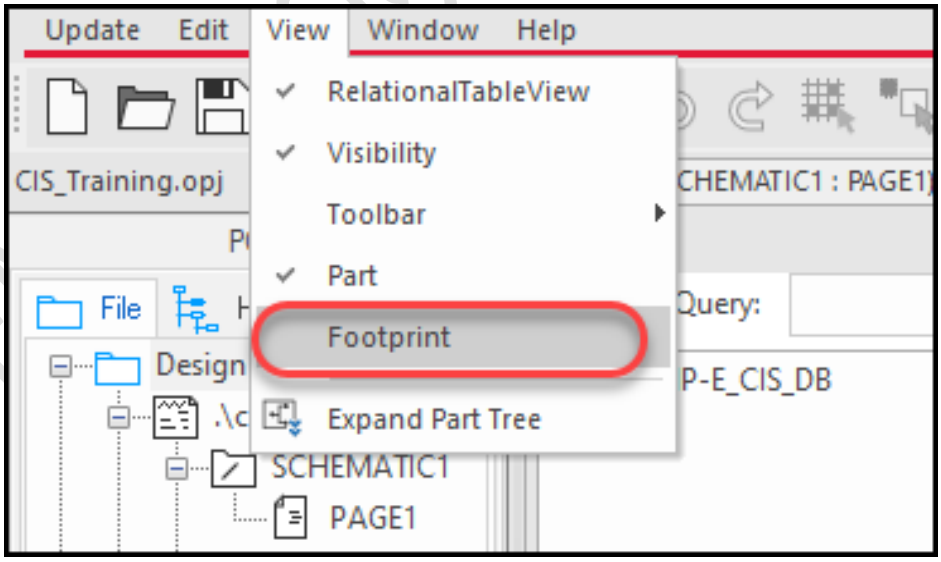


Lab 2-2: Configuring CIS Explorer

1. Click on the ***PAGE1*** tab of the schematic.

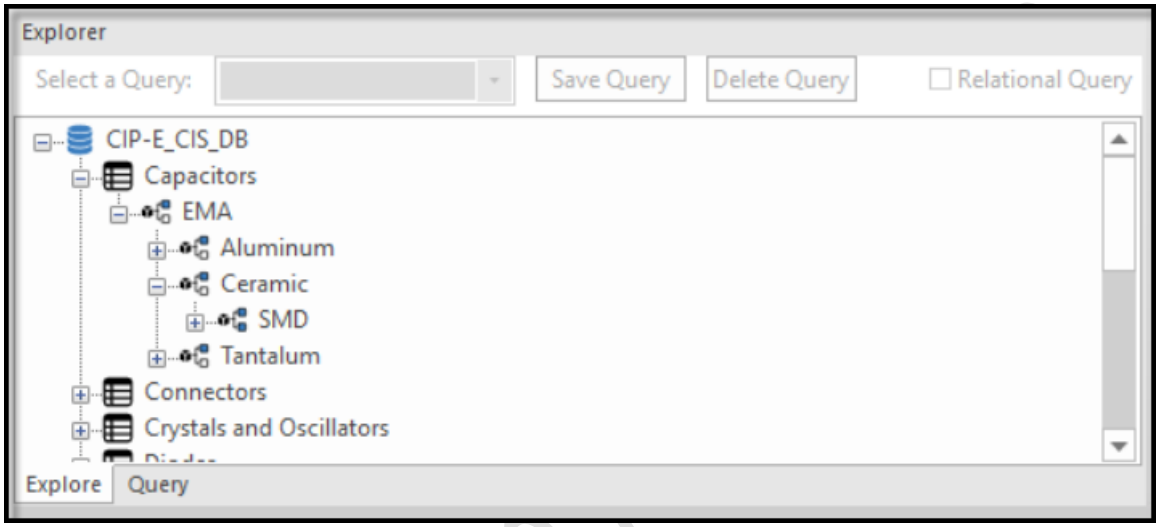


2. Select ***Place > Database Part***. The CIS Explorer window opens.
3. Click ***View > Footprint***. When the **Footprint** option in this menu is unchecked the footprint window will not appear in CIS Explorer.
4. Click ***View > Footprint*** again to have the footprint window reappear.

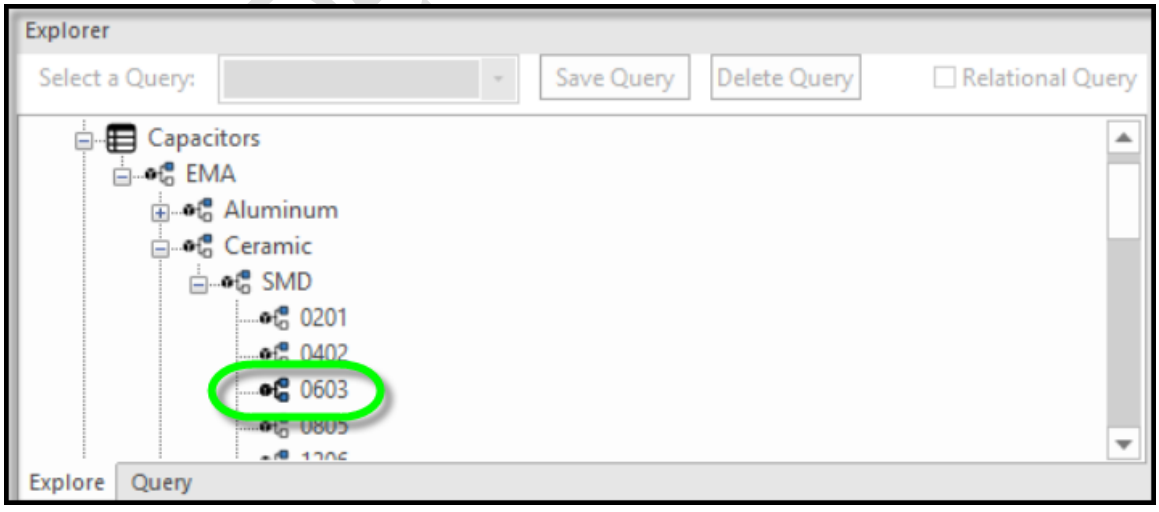


Lab 2-3: Using CIS Explorer to Search and Place Parts

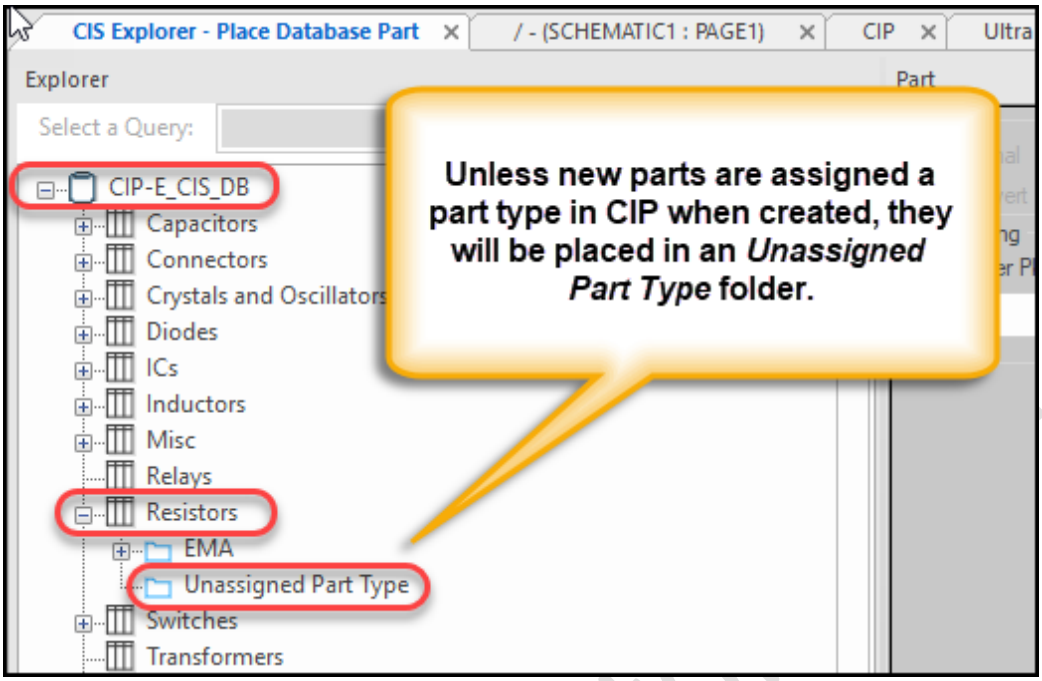
1. The upper left window of CIS Explorer is the database window. Click on the plus sign to expand the top-level database. When expanded, you will see part folders (or part tables) where parts are located.



2. Expand the folders by clicking the Plus (+) sign next to each one.
3. Expand the ***Capacitors*** table first. This will display all associated CIS Part Types for Capacitors.
4. Expand the hierarchy for ***Capacitors > EMA > Ceramic > SMD > 0603***.



If a new part is generated, unless the Part Type is defined in CIP, the part will be added to a new folder named ***Unassigned Part Type***. If a part is copied from another part that already has the Part Type defined, it will assume that Part Type.



5. Go to the ***Database Parts*** window and review the results of the 0603 category you just chose.

	Table	PART_NUMBER	Part Type	Description	Value	PCB Footprint	Schematic Part
1	Capacitors	EMA-00000372V22	EMA\Ceramic\SMD\0603	CAP, Ceramic, SMD, 0.01 uF, 10 %, 16 V, 0603	0.01uF	CAPC1608X86N	CAPACITORS\CAP
2	Capacitors	EMA-00000374V22	EMA\Ceramic\SMD\0603	CAP, Ceramic, SMD, 0.01 uF, 20 %, 16 V, 0603	0.01uF	CAPC1608X86N	CAPACITORS\CAP
3	Capacitors	EMA-00000381V22	EMA\Ceramic\SMD\0603	CAP, Ceramic, SMD, 0.012 uF, 10 %, 50 V, 0603	0.012uF	CAPC1608X90N	CAPACITORS\CAP
4	Capacitors	EMA-00000384V22	EMA\Ceramic\SMD\0603	CAP, Ceramic, SMD, 0.015 uF, 10 %, 50 V, 0603	0.015uF	CAPC1608X90N	CAPACITORS\CAP
5	Capacitors	EMA-00000387V22	EMA\Ceramic\SMD\0603	CAP, Ceramic, SMD, 0.018 uF, 5.0 %, 50 V, 0603	0.018uF	CAPC1608X90N	CAPACITORS\CAP
6	Capacitors	EMA-00000389V22	EMA\Ceramic\SMD\0603	CAP, Ceramic, SMD, 0.022 uF, 10 %, 50 V, 0603	0.022uF	CAPC1608X90N	CAPACITORS\CAP

Lab 2-4: Rearranging CIS Columns

You can rearrange CIS columns to suit your preferences for each table. This information will be stored in the Capture.ini file and will be restored the next time you open the altered CIS table.

1. In the parts table, select the ***Schematic Part*** field header and drag it to the left. You' will notice a red line as you drag that indicates where to drop the column.

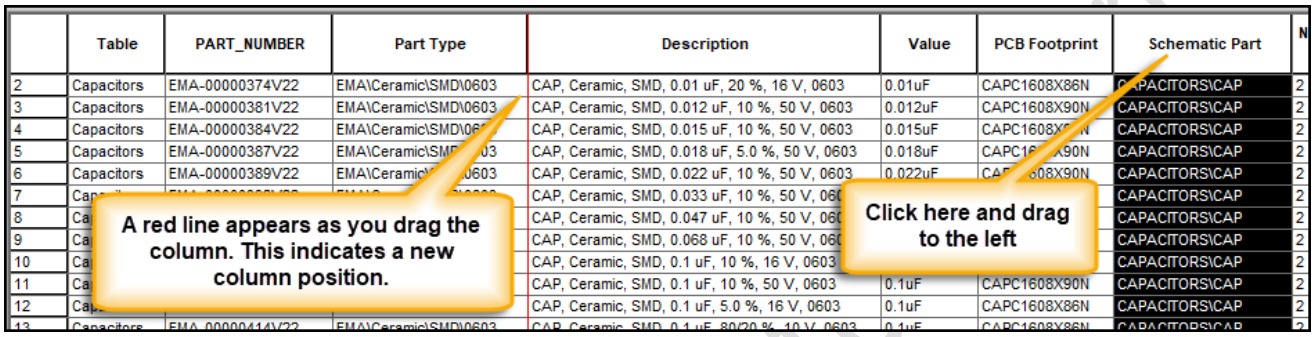


	Table	PART_NUMBER	Part Type	Description	Value	PCB Footprint	Schematic Part	N
2	Capacitors	EMA-00000374V22	EMA\Ceramic\SMD\0603	CAP, Ceramic, SMD, 0.01 uF, 20 %, 16 V, 0603	0.01uF	CAPC1608X86N	CAPACITORS\CAP	2
3	Capacitors	EMA-00000381V22	EMA\Ceramic\SMD\0603	CAP, Ceramic, SMD, 0.012 uF, 10 %, 50 V, 0603	0.012uF	CAPC1608X90N	CAPACITORS\CAP	2
4	Capacitors	EMA-00000384V22	EMA\Ceramic\SMD\06	CAP, Ceramic, SMD, 0.015 uF, 10 %, 50 V, 0603	0.015uF	CAPC1608X	CAPACITORS\CAP	2
5	Capacitors	EMA-00000387V22	EMA\Ceramic\SM 03	CAP, Ceramic, SMD, 0.018 uF, 5.0 %, 50 V, 0603	0.018uF	CAPC16 X90N	CAPACITORS\CAP	2
6	Capacitors	EMA-00000389V22	EMA\Ceramic\ 0603	CAP, Ceramic, SMD, 0.022 uF, 10 %, 50 V, 0603	0.022uF	CA 08X90N	CAPACITORS\CAP	2
7	Cap			CAP, Ceramic, SMD, 0.033 uF, 10 %, 50 V, 06			CAPACITORS\CAP	2
8	Ca			CAP, Ceramic, SMD, 0.047 uF, 10 %, 50 V, 06			CAPACITORS\CAP	2
9	Ca			CAP, Ceramic, SMD, 0.068 uF, 10 %, 50 V, 06			CAPACITORS\CAP	2
10	Ca			CAP, Ceramic, SMD, 0.1 uF, 10 %, 16 V, 0603			CAPACITORS\CAP	2
11	Ca			CAP, Ceramic, SMD, 0.1 uF, 10 %, 50 V, 0603	0.1uF	CAPC1608X90N	CAPACITORS\CAP	2
12	Cap			CAP, Ceramic, SMD, 0.1 uF, 5.0 %, 16 V, 0603	0.1uF	CAPC1608X86N	CAPACITORS\CAP	2
13	Capacitors	EMA-00000414V22	EMA\Ceramic\SMD\0603	CAP, Ceramic, SMD, 0.1 uF, 80/20 %, 10 V, 0603	0.1uF	CAPC1608X86N	CAPACITORS\CAP	2

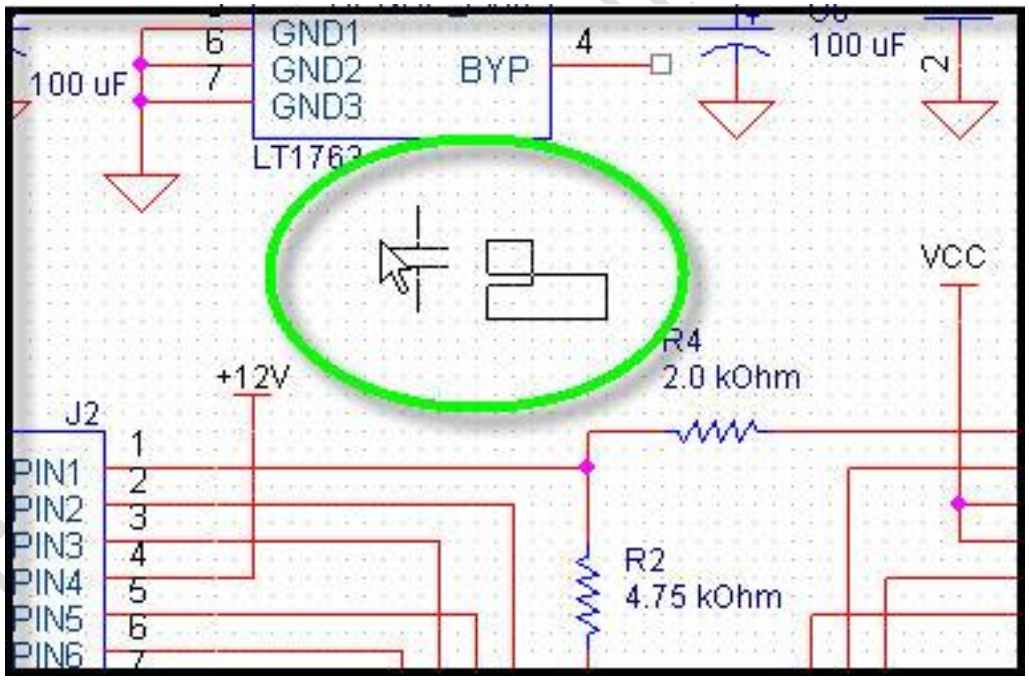
2. Drop the column to the right of the ***Part Type*** column.

Lab 2-5: Placing a Database Part

1. Click on one of the parts in the parts table – the part turns green. This indicates that the part can be placed on the schematic.
2. Double click on the part that you have selected.

	Table	PART_NUMBER	Part Type	Value	Description	Schem
1	Capacitors	EMA-00000372V22	EMA\Ceramic\SMD\0603	0.01uF	CAP, Ceramic, SMD, 0.01 uF, 10 %, 16 V, 0603	CAPACITOR
2	Capacitors	EMA-00000374V22	EMA\Ceramic\SMD\0603	0.01uF	CAP, Ceramic, SMD, 0.01 uF, 20 %, 16 V, 0603	CAPACITOR
3	Capacitors	EMA-00000381V22	EMA\Ceramic\SMD\0603	0.012uF	CAP, Ceramic, SMD, 0.012 uF, 10 %, 50 V, 0603	CAPACITOR
4	Capacitors	EMA-00000384V22	EMA\Ceramic\SMD\0603	0.015uF	CAP, Ceramic, SMD, 0.015 uF, 10 %, 50 V, 0603	CAPACITOR
5	Capacitors	EMA-00000387V22	EMA\Ceramic\SMD\0603	0.018uF	CAP, Ceramic, SMD, 0.018 uF, 5.0 %, 50 V, 0603	CAPACITOR
6	Capacitors	EMA-00000389V22	EMA\Ceramic\SMD\0603	0.022uF	CAP, Ceramic, SMD, 0.022 uF, 10 %, 50 V, 0603	CAPACITOR
7	Capacitors	EMA-00000392V22	EMA\Ceramic\SMD\0603	0.033uF	CAP, Ceramic, SMD, 0.033 uF, 10 %, 50 V, 0603	CAPACITOR
8	Capacitors	EMA-00000394V22	EMA\Ceramic\SMD\0603	0.047uF	CAP, Ceramic, SMD, 0.047 uF, 10 %, 50 V, 0603	CAPACITOR
9	Capacitors	EMA-00000398V22	EMA\Ceramic\SMD\0603	0.068uF	CAP, Ceramic, SMD, 0.068 uF, 10 %, 50 V, 0603	CAPACITOR

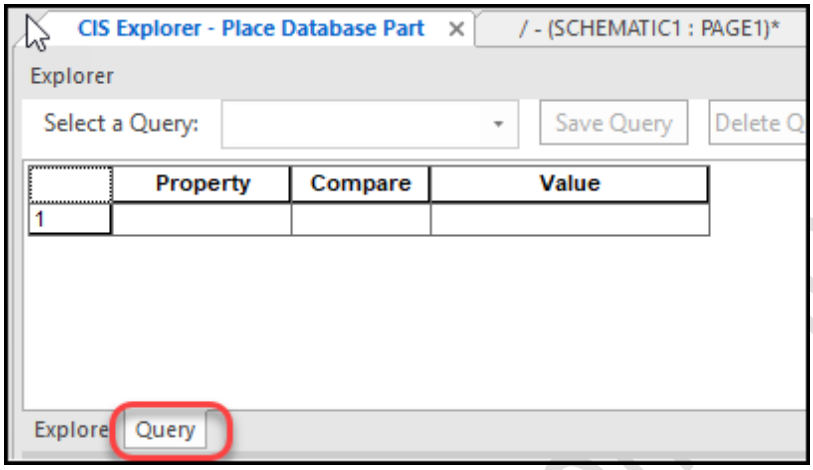
3. The part becomes attached to the cursor and the schematic page opens to enable placement.



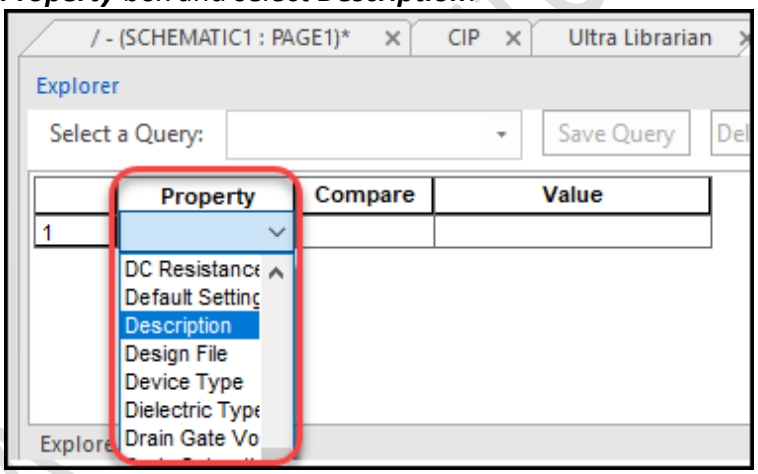
4. Click to place the part on the schematic.
5. Hit the ***<Esc>*** key twice to exit part placement.
6. Right click in the schematic and select ***Place Database Part*** to return to CIS Explorer.

Lab 2-6: Using Query to Search

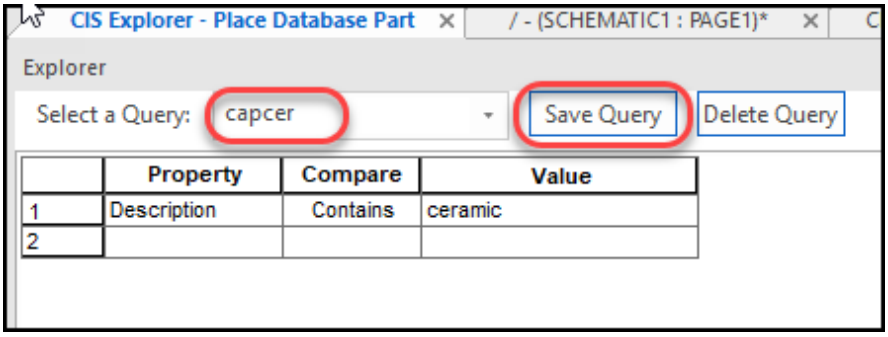
1. In the CIS Explorer window, click on the ***Query*** tab.



2. Click in the ***Property*** box and select ***Description***.



3. Click in the ***Compare*** box and select ***Contains***.
4. Click in the ***Value*** box and enter ***ceramic***. Hit the **<Enter>** key.
5. Save this search by entering the name ***capcer*** in the ***Select a Query*** field, then click ***Save Query*.** This will enable you to retrieve the search for future use.



6. Review the search results.

Sorting Search Results

1. Double clicking on any column header in the parts table will sort the results based on that field. Double click on the ***Value*** column to sort based on value.

	Table	PART_NUMBER	Part Type	Description	Value	PCB Footprint	Sc
4	Capacitors	EMA-00000375V22	EMA\Ceramic\SMD\0805	CAP, Ceramic, SMD, 0.01 uF,	0.01uF	CAPC2012X71N	CA
5	Capacitors	EMA-00000376V22	EMA\Ceramic\SMD\0805	CAP, Ceramic, SMD, 0.01 uF,	0.01uF	CAPC2012X71N	CA
6	Capacitors	EMA-00000377V22	EMA\Ceramic\SMD\0805	CAP, Ceramic, SMD, 0.01 uF,	0.01uF	CAPC2012X71N	CA
7	Capacitors	EMA-00000378V22	EMA\Ceramic\SMD\0805	CAP, Ceramic, SMD, 0.01 uF,	0.01uF	CAPC2012X71N	CA
8	Capacitors	EMA-00000381V22	EMA\Ceramic\SMD\0603	CAP, Ceramic, SMD, 0.012 u	0.012uF	CAPC1608X90N	CA
9	Capacitors	EMA-00000382V22	EMA\Ceramic\SMD\0805	CAP, Ceramic, SMD, 0.012 u	0.012uF	CAPC2012X71N	CA
10	Capacitors	EMA-00000383V22	EMA\Ceramic\SMD\0805	CAP, Ceramic, SMD, 0.012 u	0.012uF	CAPC2012X71N	CA
11	Capacitors	EMA-00000384V22	EMA\Ceramic\SMD\0603	CAP, Ceramic, SMD, 0.015 u	0.015uF	CAPC1608X90N	CA
12	Capacitors	EMA-00000385V22	EMA\Ceramic\SMD\0805	CAP, Ceramic, SMD, 0.015 u	0.015uF	CAPC2012X71N	CA
13	Capacitors	EMA-00000386V22	EMA\Ceramic\SMD\0805	CAP, Ceramic, SMD, 0.015 u	0.015uF	CAPC2012X71N	CA
14	Capacitors	EMA-00000387V22	EMA\Ceramic\SMD\0603	CAP, Ceramic, SMD, 0.018 u	0.018uF	CAPC1608X90N	CA
15	Capacitors	EMA-00000388V22	EMA\Ceramic\SMD\0805	CAP, Ceramic, SMD, 0.018 u	0.018uF	CAPC2012X71N	CA
16	Capacitors	EMA-00000389V22	EMA\Ceramic\SMD\0603	CAP, Ceramic, SMD, 0.022 u	0.022uF	CAPC1608X90N	CA
17	Capacitors	EMA-00000390V22	EMA\Ceramic\SMD\1825	CAP, Ceramic, SMD, 0.022 u	0.022uF	CAPC4564X155N	CA
18	Capacitors	EMA-00000392V22	EMA\Ceramic\SMD\0603	CAP, Ceramic, SMD, 0.033 u	0.033uF	CAPC1608X90N	CA
19	Capacitors	EMA-00000394V22	EMA\Ceramic\SMD\0603	CAP, Ceramic, SMD, 0.047 u	0.047uF	CAPC1608X90N	CA
20	Capacitors	EMA-00000395V22	EMA\Ceramic\SMD\0805	CAP, Ceramic, SMD, 0.047 u	0.047uF	CAPC2012X71N	CA

Placing Parts

1. Click on one of the capacitors in the search results.
2. When it turns green, double click to place it in the schematic.
3. When the schematic page appears, click 3 times on the schematic page to place 3 capacitors.
4. Save the design. Do not close CIS.

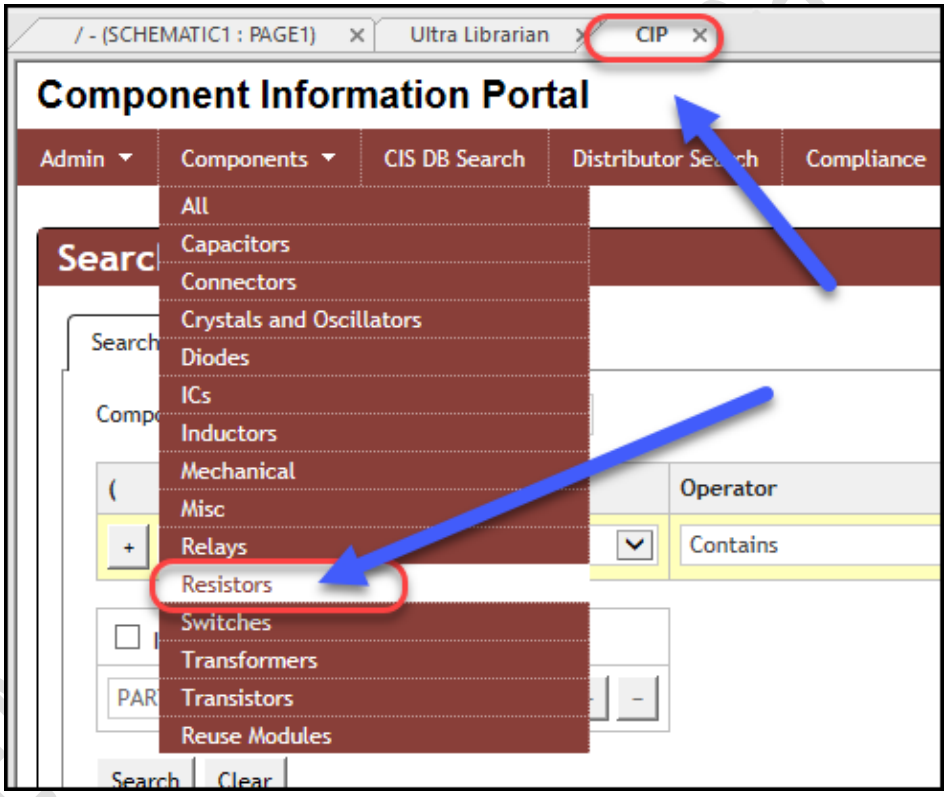
Lab 2-7: Add Parts to the Schematic from CIP

The schematic named `CIS_Training.dsn` should be open. In Lab 2-6, you learned how to place a part on the schematic from the CIS Explorer window.

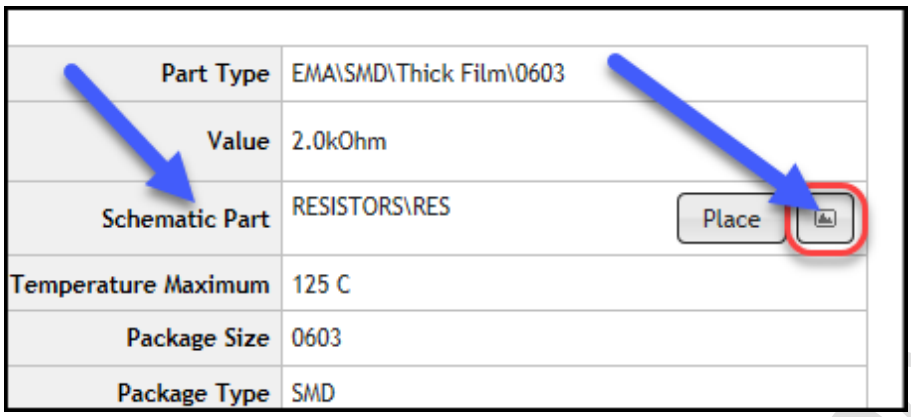
Next, you will place the part from within the CIP window.

Placing a Schematic Part from CIP

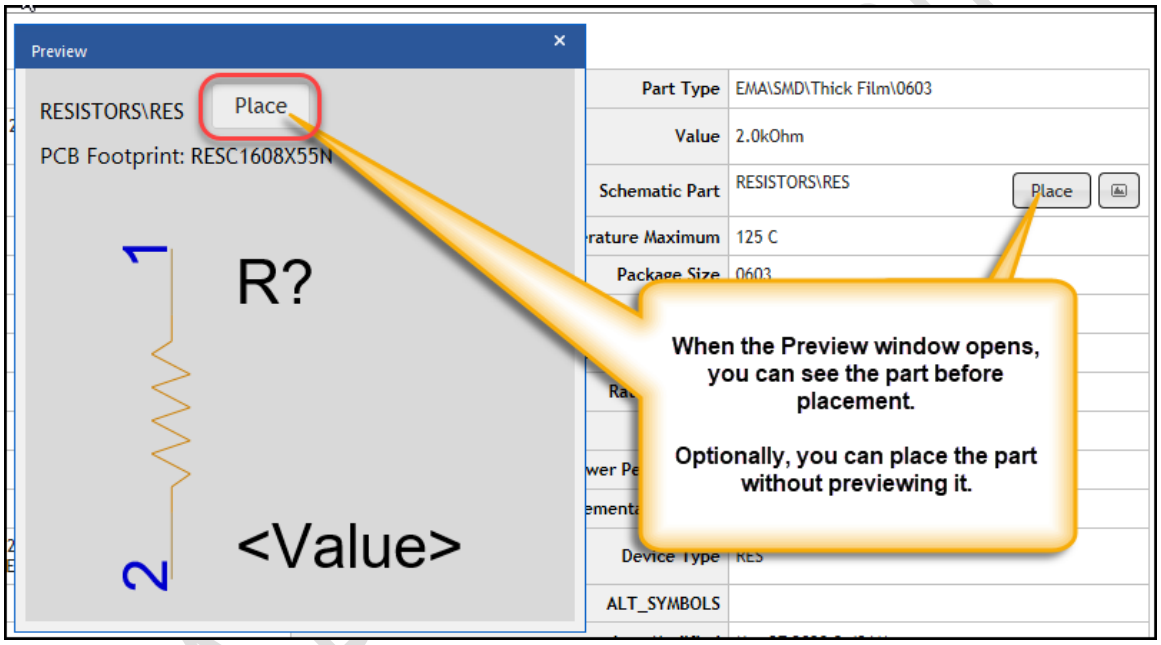
1. Select the ***CIP*** tab and open the ***Resistors*** table.



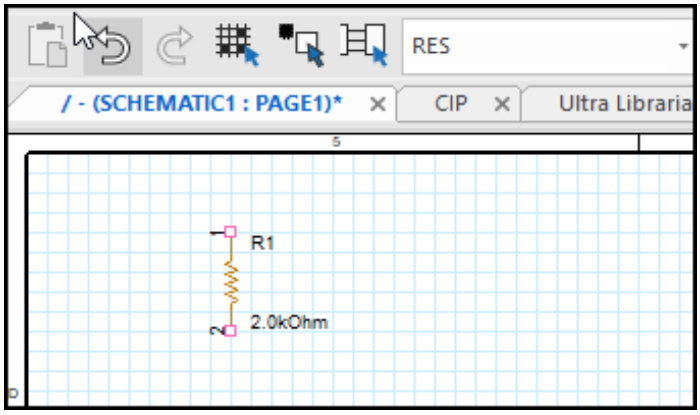
2. The first part number in the table will show in the Part Detail. In the ***Schematic Part*** field, select the ***Preview*** icon to preview the part before placement.



When the Preview window opens you can examine the part before placing it.



3. From the ***Preview*** window, select the ***Place*** button. The schematic page will open, and the part will be attached to the cursor, allowing you to place it on the schematic.



Optionally, if you do not need to preview the symbol before placing, you could have chosen the ***Place*** button from the ***Schematic Part*** field in CIP.

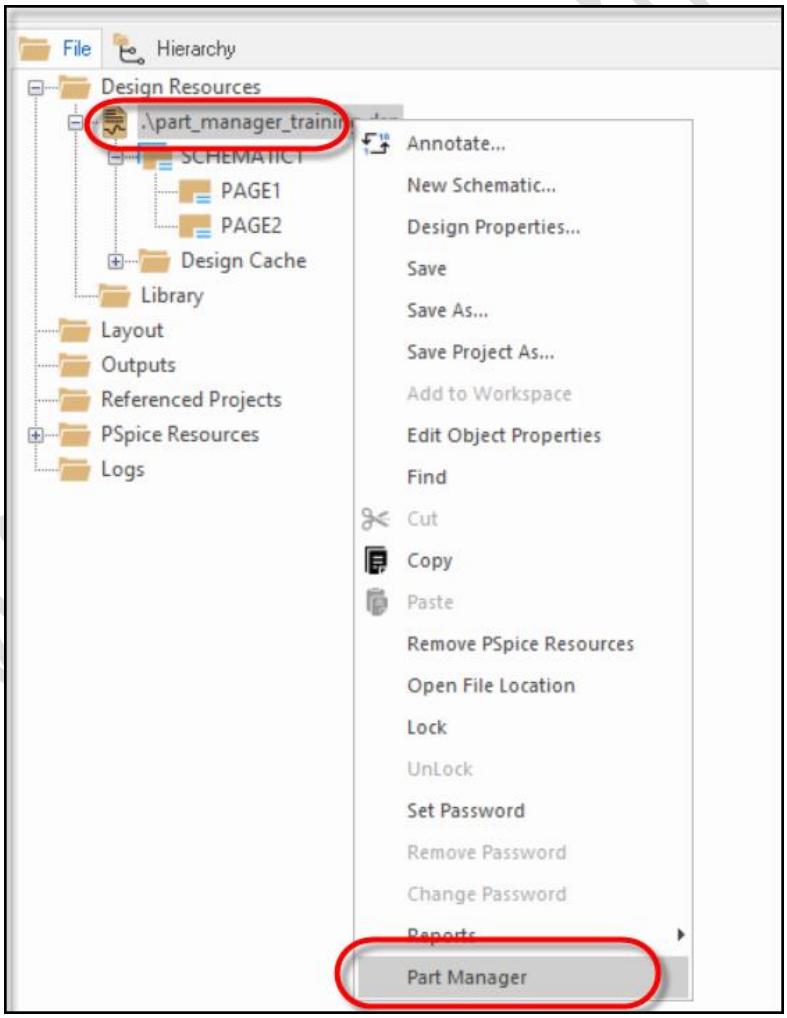
Lesson 3: CIS Part Manager

The Part Manager's role provides a constant checking of the status of each part residing in a design as well as providing functionality to update part status and create and manage variants of the schematic.

Accessing Part Manager

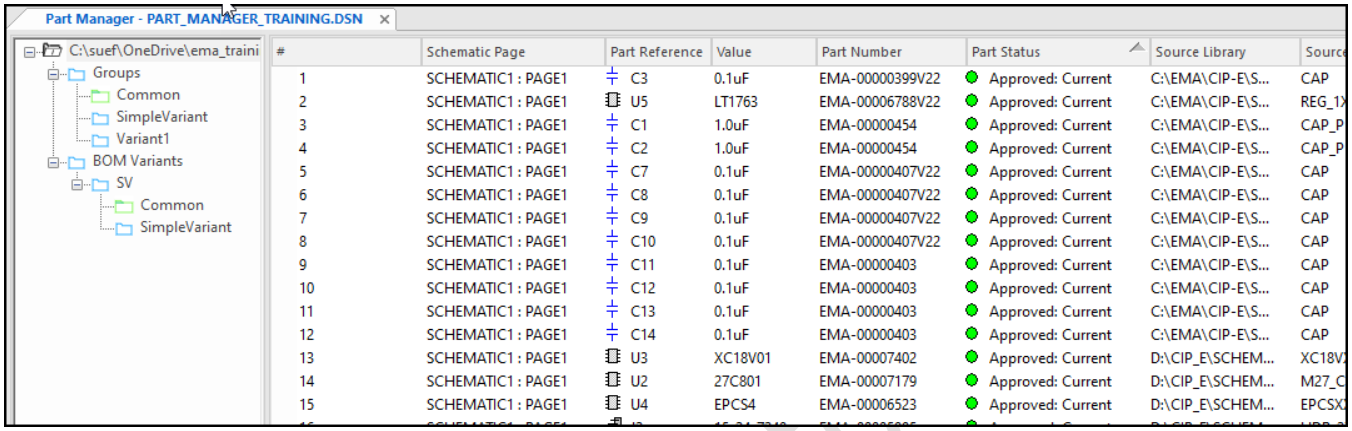
Part Manager can be accessed in a couple of ways. The first way is to select the design (.dsn) in the Project Manager, then select ***Tools > Part Manager > Open***.

The second method is to right click on the design (.dsn) in the Project Manager. And select ***Part Manager***.



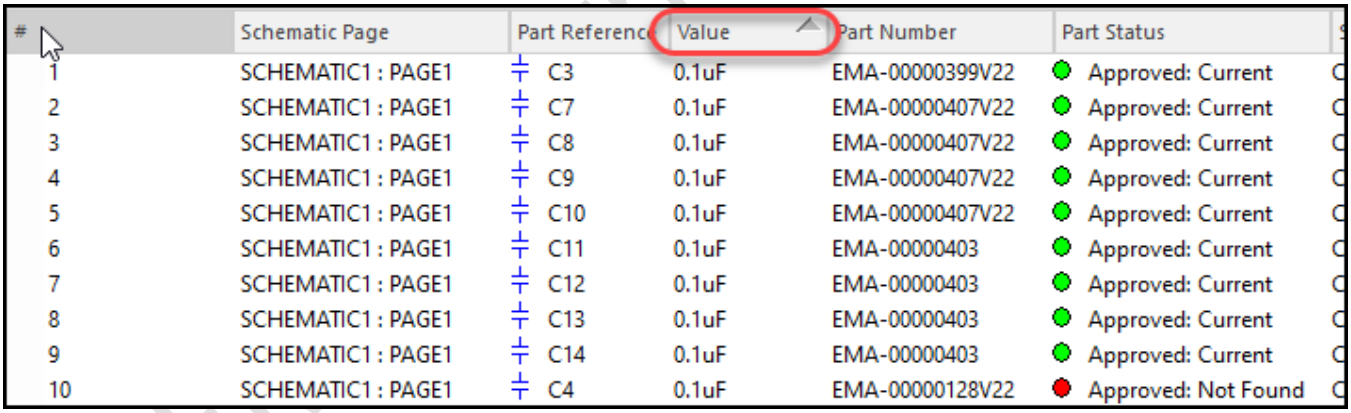
Part Manager Window

The Part Manager window contains two panes. The left pane is the ***Tree View*** and is used for creating groups and subgroups used for BOM variants. The ***List View***, in the right pane, displays all parts used in the schematic design and shows the status of each part.



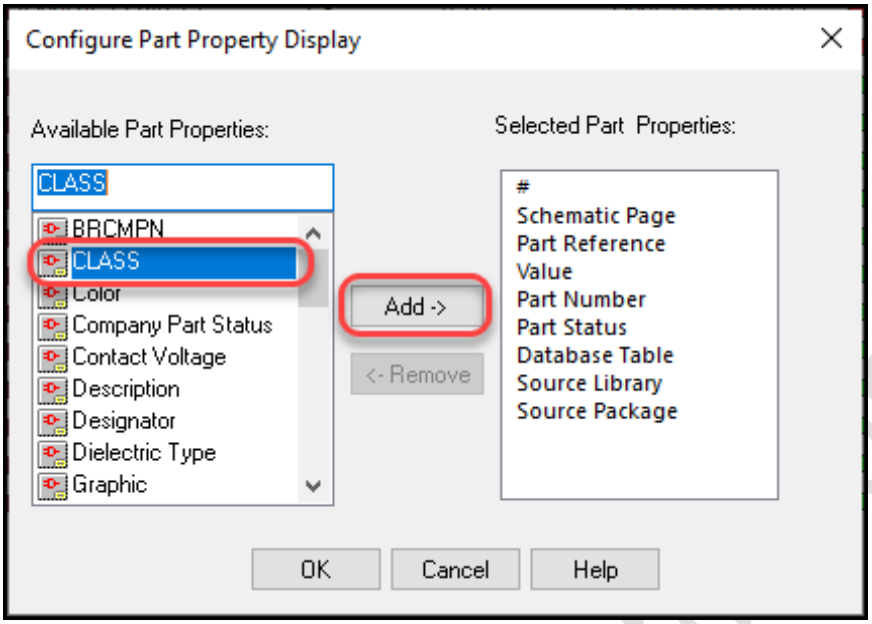
Sorting the Part Manager Window

Data in the Part Manager can be sorted by clicking on the column headers.



Configuring the Part Property Display

In addition to the order, the display of part properties can be configured in the Part Manager window by selecting ***View > Configure Part Properties Display***. Choose an ***Available Part Property*** from the list on the left, click ***Add ->*** to add it to the ***Selected Part Properties*** on the right, then click ***OK***. The part property will be displayed in the Part Manager.



Placed Part Status

There are several states, or part status indicators, associated with each part in a design. The part status for each part can be viewed in the Part Manager.

Status Dot Color	Placed Status Part	Description
●	Approved: Current	The part number property value on the placed part matches the database part, and all of the transferrable properties match
●	Approved: Defined	The placed part has a defined part number property but it has not yet been checked against the database part
●	Approved: Undefined part reference	The placed part has an undefined part reference value (such as "R?")
●	Temporary: Current	The placed part has temporary part number and all the transferrable properties match the database part
●	Temporary: Defined	The placed part has a temporary part number but it has not yet been checked against the database part
●	Approved: Package out of date	The symbol in the schematic does not match the symbol in the database
●	Approved: Not Current	A part number property exists in the database but one or more of the transferrable properties or the schematic symbol do not match the database part
●	Approved: Duplicate	The part number on the placed part occurs more than once in the parts database. This status only occurs if your configuration file does not allow duplicate part numbers
●	Approved: Not Found	The part number property on the placed part does not exist in the parts database
●	Undefined	The placed part does not have a part number property
●	Temporary: Not Current	One or more of the transferrable properties or the schematic symbol on the temporary part do not match the database part
●	Temporary: Duplicate	This status only occurs if you intentionally duplicate a temporary part number
●	Temporary: Not Found	The part number property valuie on the placed temporary part does not exist in the parts database

Part Manager Options

There are several options you can apply to parts within Part Manager. They are:

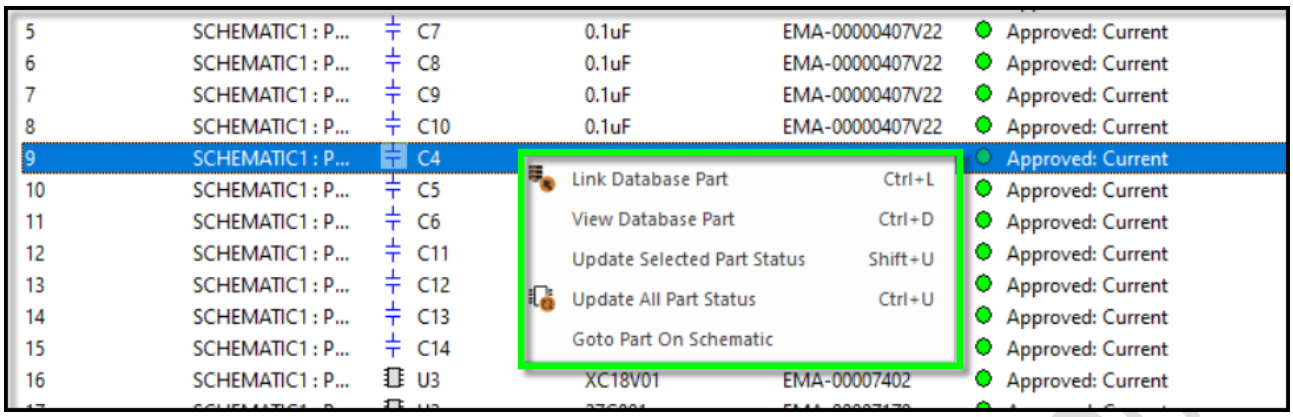
Link Database Part – associates a component on the schematic page with a part you choose from the CIS database

View Database Part – displays CIS Explorer and the selected part and its database properties are displayed

Update Selected Part Status – ensures that the selected part exists in the CIS database. Any differences between the schematic parts and its database properties are displayed.

Update All Part Status – ensures all parts exist in the CIS database. Any differences between the schematic parts and its database properties are displayed.

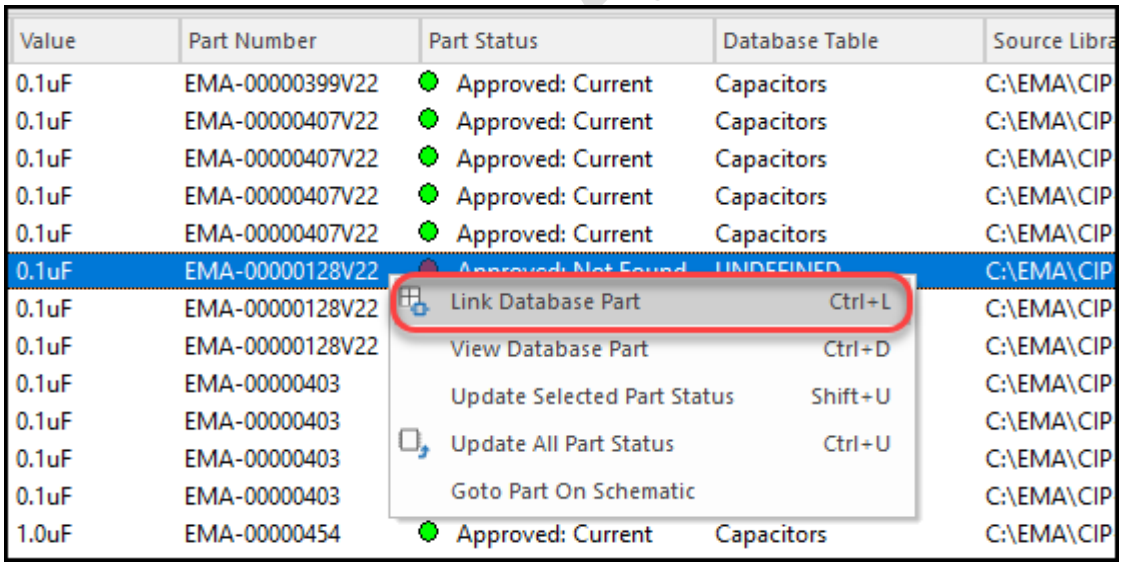
Goto Part On Schematic – displays the schematic page and highlights the part



Linking Database Parts

The ***Link Database Part*** operation provides a way to link one or more part(s) in a design to a part in the database. This feature may be necessary when bringing older, legacy schematics with parts that may not reside in the database into sync with the CIS database.

To link a part, while Part Manager is open, right click on the desired part and select ***Link Database Part***.



This will open CIS Explorer. The table view will populate with similar parts based on value. Scroll through the list to find a suitable part, making sure the PCB footprint is compatible. Double click on the database part to link it to the part in Part Manager. The Part Manager will reappear, showing the newly linked part. At this point, select the newly linked part, right click, and select ***Update Selected Part***. This operation will run a check of the new part against the database. If everything matches, the part status will be ***Approved: Current***.

While linking a part, if the database part is an alias, the schematic part property of the database part will not be transferred to the placed part on the schematic. CIS does not differentiate between a package and its alias.

Updating All Part Status

CIS checks each placed part against the database part to which it is linked. The part database is searched for the Part Number property that matches the placed part, and transferred properties that are configured to be updated are compared.

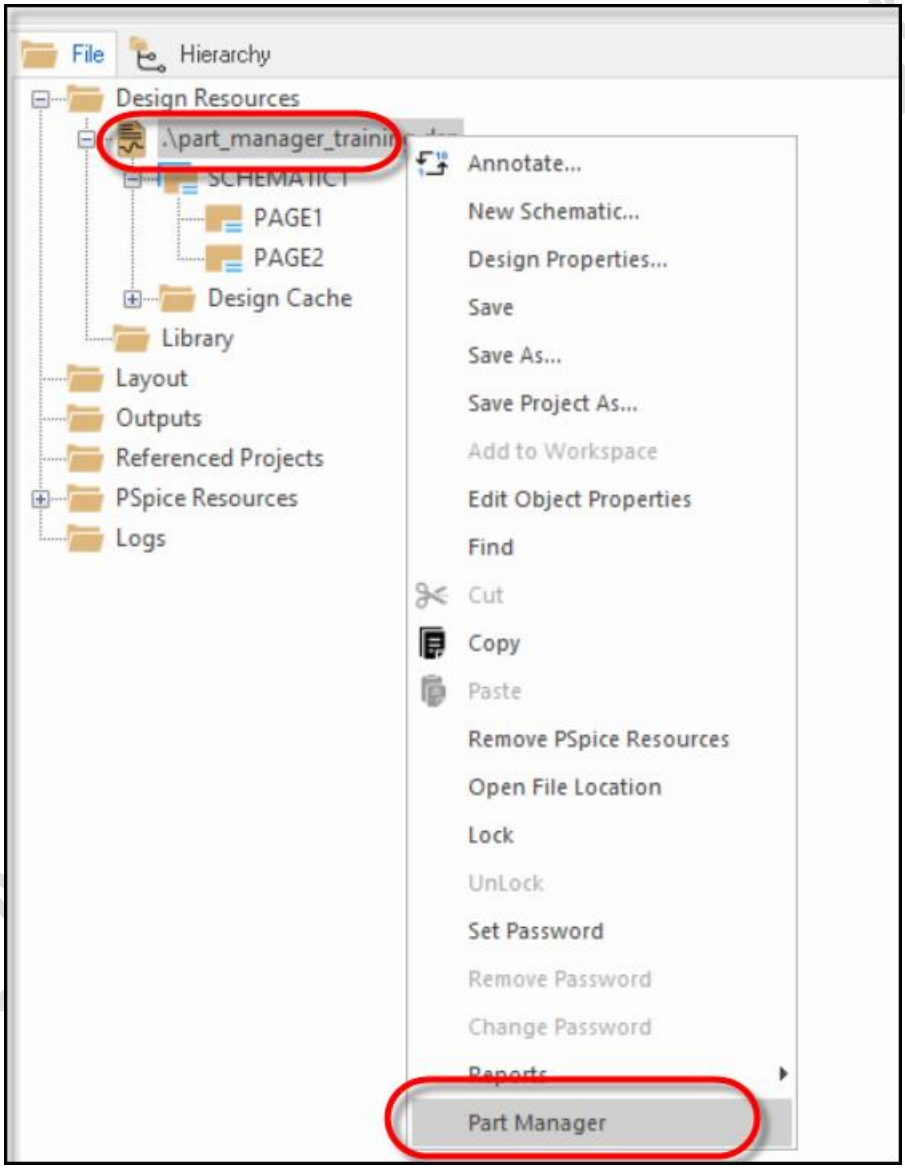
For each placed part that is not current, you are prompted with the **Update Part** dialog box. This dialog box lets you decide whether you want to update the placed part properties with the transferred properties from the database part.

The part status is based only on the part properties have been specified to be transferred from the part database. Other properties that may reside in the part are not checked.

Lab 3-1: Opening Part Manager

Part Manager provides an ongoing check of the status of each part residing in the design. In this exercise, you will explore the various features and options within Part Manager.

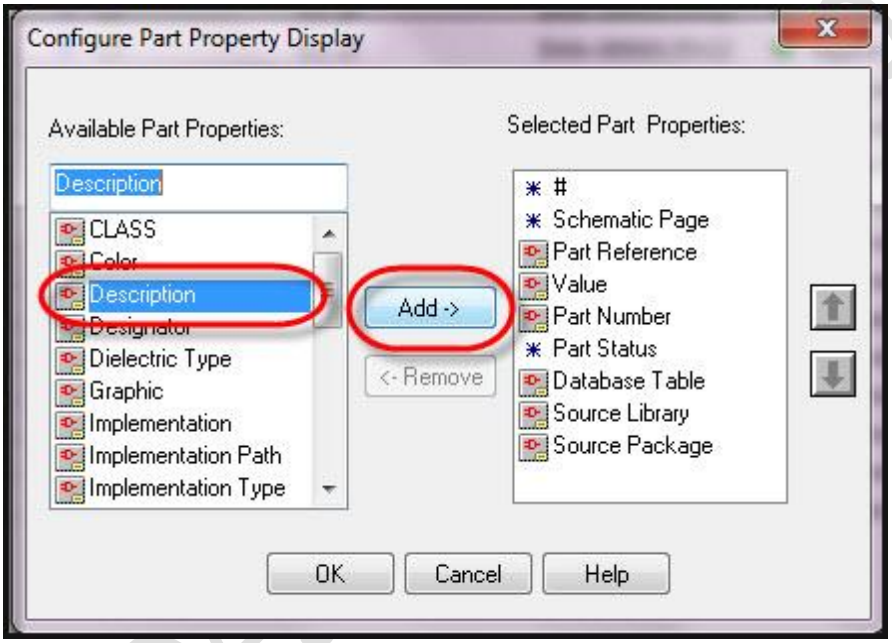
1. In CIS, select ***File > Open*** and open `part_manager_training.dsn`.
2. In the Project Manager, right click on the design and select ***Part Manager***.



Lab 3-2: Configuring Part Manager

You can add properties that are shown in the Part Manager, adjust their column locations, and sort parts based on preferred fields in ascending or descending order.

1. In Part Manager, select ***View > Configure Part Properties Display***.
2. The **Configure Part Property Display** dialog will appear. In the left column select ***Description***, then click the ***Add*** button to push it to the ***Selected Part Properties*** that will be displayed in Part Manager.

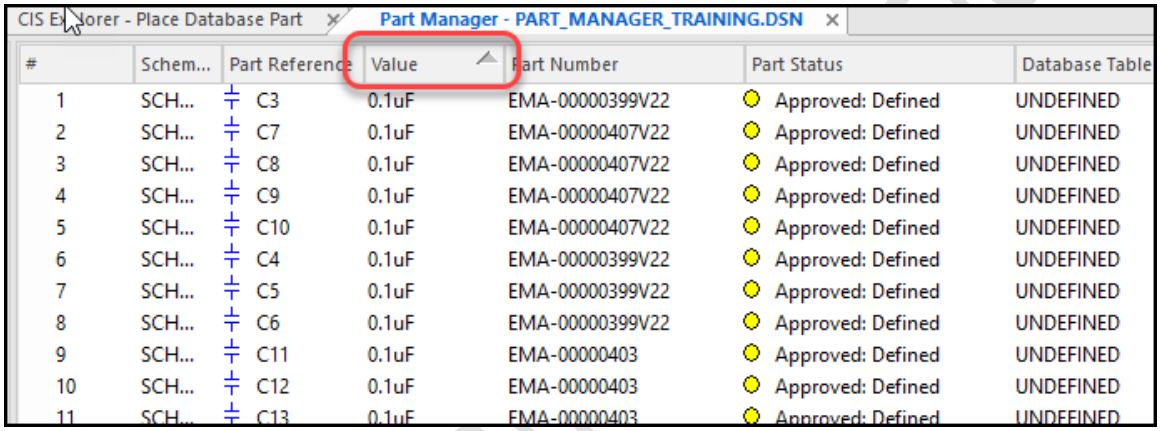


Do not close Part Manager.

Lab 3-3: Sorting Data within Part Manager

Often, data is sorted based on field preference. This can be especially helpful if you plan to link, for example, all capacitors of one value to another value. Sorting by Value can allow you to easily select multiple parts at a time for this type of operation.

1. Click on the ***Value*** header to sort the parts based on value. This will sort by value in ascending order.



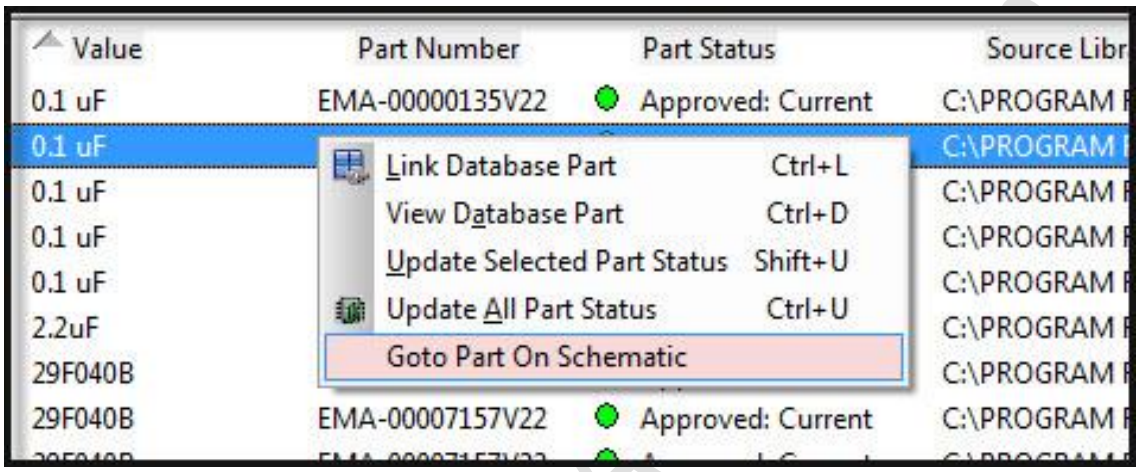
#	Schem...	Part Reference	Value	Part Number	Part Status	Database Table
1	SCH...	C3	0.1uF	EMA-00000399V22	Approved: Defined	UNDEFINED
2	SCH...	C7	0.1uF	EMA-00000407V22	Approved: Defined	UNDEFINED
3	SCH...	C8	0.1uF	EMA-00000407V22	Approved: Defined	UNDEFINED
4	SCH...	C9	0.1uF	EMA-00000407V22	Approved: Defined	UNDEFINED
5	SCH...	C10	0.1uF	EMA-00000407V22	Approved: Defined	UNDEFINED
6	SCH...	C4	0.1uF	EMA-00000399V22	Approved: Defined	UNDEFINED
7	SCH...	C5	0.1uF	EMA-00000399V22	Approved: Defined	UNDEFINED
8	SCH...	C6	0.1uF	EMA-00000399V22	Approved: Defined	UNDEFINED
9	SCH...	C11	0.1uF	EMA-00000403	Approved: Defined	UNDEFINED
10	SCH...	C12	0.1uF	EMA-00000403	Approved: Defined	UNDEFINED
11	SCH...	C13	0.1uF	EMA-00000403	Approved: Defined	UNDEFINED

2. Click again on the ***Value*** field to sort in descending order based on Value.
3. Do not close Part Manager.

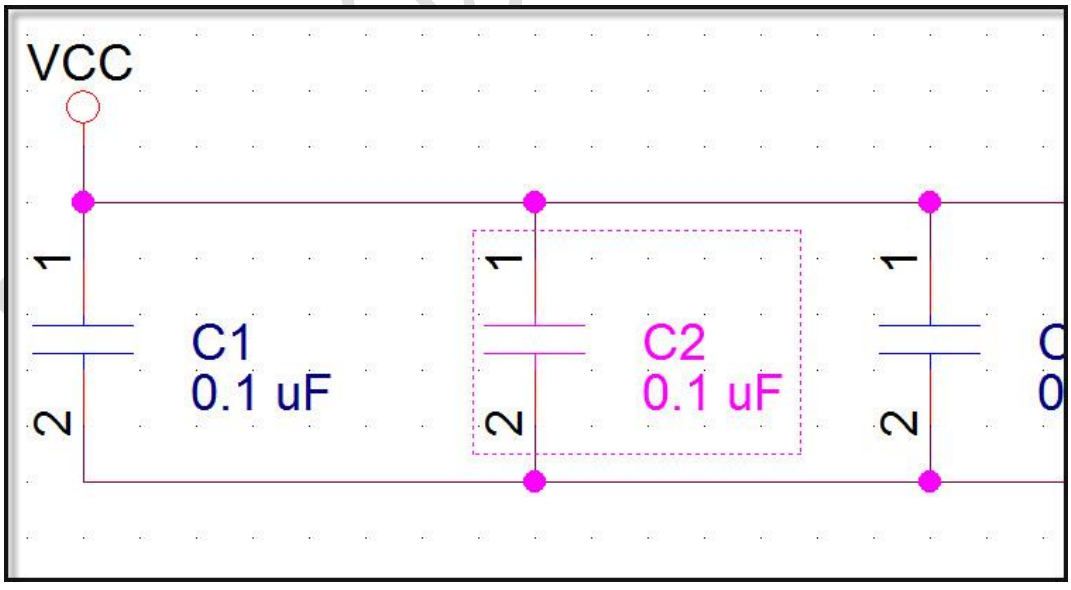
Lab 3-4: Go to Part on Schematic

While in Part Manager, you can go to any one of the parts to see its symbol and placement within the schematic.

1. Right click on any part in the Part Manager list and choose ***Goto Part on Schematic***.



The schematic will appear showing the highlighted part. Below is a sample of a part appearing highlighted on the schematic after choosing **Goto Part on Schematic** in Part Manager.

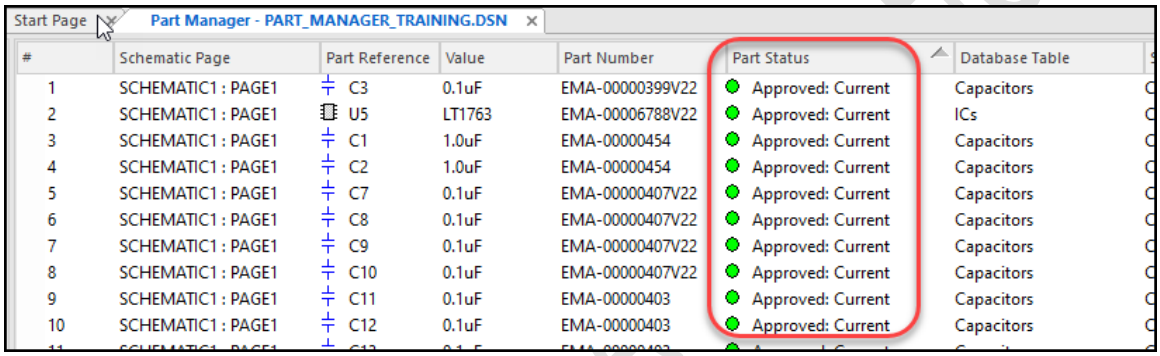


Go back to the Part Manager window.

Lab 3-5: Update All Part Status

When you opened Part Manager on this design, all the parts have an **Approved: Defined** (Yellow) Part Status. This is because the parts have not yet been checked against the database to see if any changes have occurred either with the parts or within the database.

1. Right click on a part in the list and select ***Update All Part Status***.
2. Click ***Yes*** to the Undo/Redo message. The parts in this design will now have an **Approved: Current** status (Green).

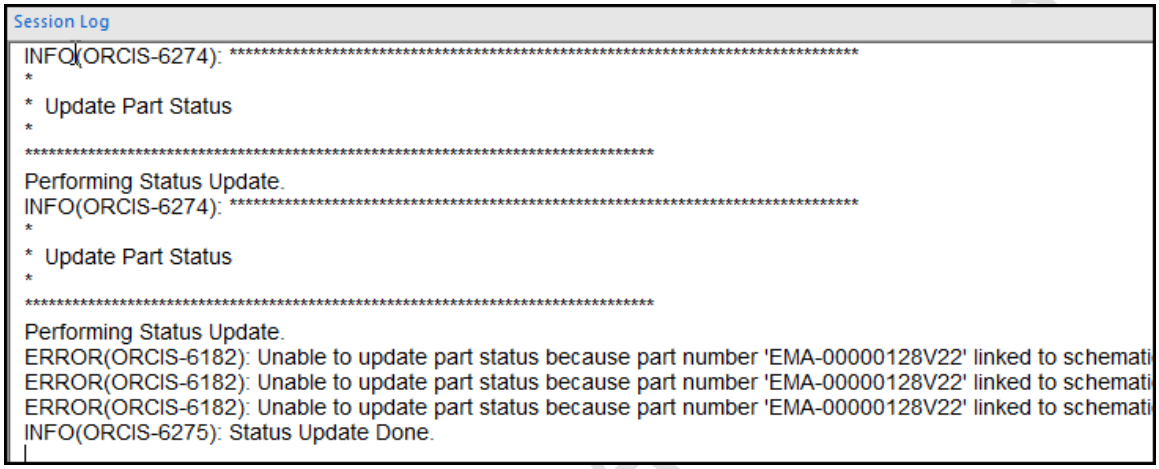


#	Schematic Page	Part Reference	Value	Part Number	Part Status	Database Table
1	SCHEMATIC1 : PAGE1	C3	0.1uF	EMA-00000399V22	Approved: Current	Capacitors
2	SCHEMATIC1 : PAGE1	U5	LT1763	EMA-00006788V22	Approved: Current	ICs
3	SCHEMATIC1 : PAGE1	C1	1.0uF	EMA-00000454	Approved: Current	Capacitors
4	SCHEMATIC1 : PAGE1	C2	1.0uF	EMA-00000454	Approved: Current	Capacitors
5	SCHEMATIC1 : PAGE1	C7	0.1uF	EMA-00000407V22	Approved: Current	Capacitors
6	SCHEMATIC1 : PAGE1	C8	0.1uF	EMA-00000407V22	Approved: Current	Capacitors
7	SCHEMATIC1 : PAGE1	C9	0.1uF	EMA-00000407V22	Approved: Current	Capacitors
8	SCHEMATIC1 : PAGE1	C10	0.1uF	EMA-00000407V22	Approved: Current	Capacitors
9	SCHEMATIC1 : PAGE1	C11	0.1uF	EMA-00000403	Approved: Current	Capacitors
10	SCHEMATIC1 : PAGE1	C12	0.1uF	EMA-00000403	Approved: Current	Capacitors

Lab 3-6: Viewing the Session Log

At any time while you are working within OrCAD Capture CIS you can check the session log for any messages.

1. If the session log is not displayed, select ***Window > Session Log***.
2. The session window will appear at the bottom of the window. Review the results.

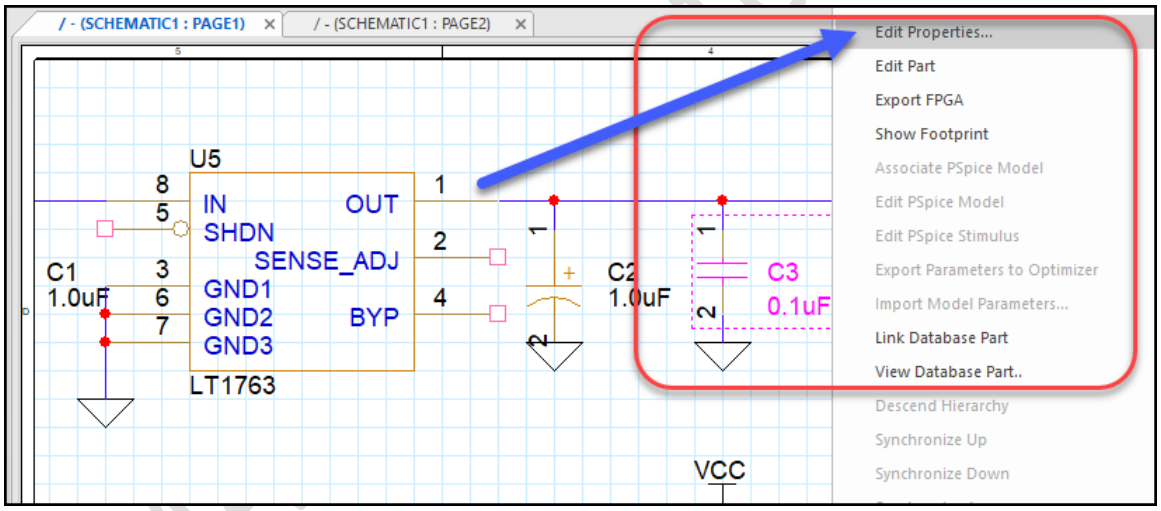


Lab 3-7: Editing Properties

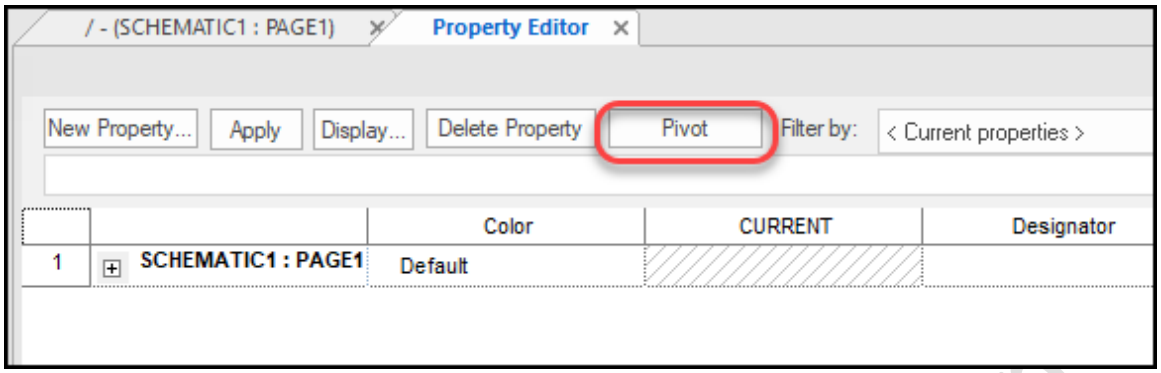
Capture schematic entry can update properties on parts while in the schematic. However, when this occurs, the Part Status indicator changes after running Update Part Status, indicating that there are differences between the schematic part and the database part.

As a best practice, when moving to a database methodology that OrCAD Capture CIS provides, it is recommended that no edits to part properties be made at the schematic level, because that would diverge from the master part record that is stored in the database. Instead, make changes to existing part properties in the database itself. Symbol changes should be made in the master library. If symbol graphics have changed, you should update your Design Cache to reflect the updated symbol in the schematic.

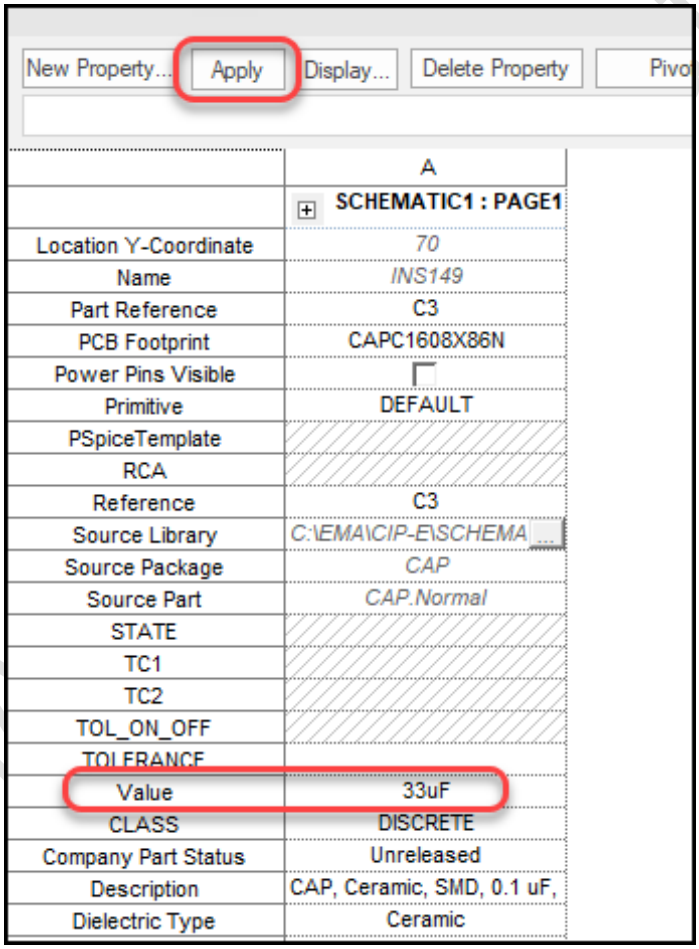
1. Locate C3 on the schematic (upper left area), select the part, right click, and select ***Edit Properties***.



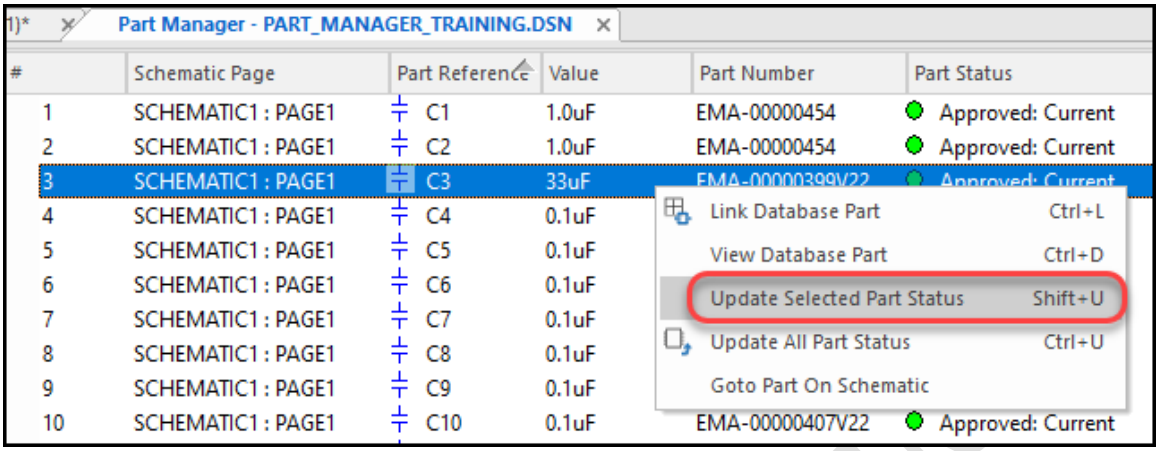
2. The Property Editor window will appear. If the properties appear in one horizontal row you can pivot the view so it will be easier to read. To pivot the view, select the ***Pivot*** button.



3. Change the ***Value*** from ***0.1uF*** to ***33uF***.
4. Click ***Apply***.



5. Go back to Part Manager.
6. Select ***C3***. Notice the value has changed to 33uF.
7. Right click on ***C3*** and select ***Update Selected Part Status***. This will run a check to see if there are differences between the schematic part and the database part.



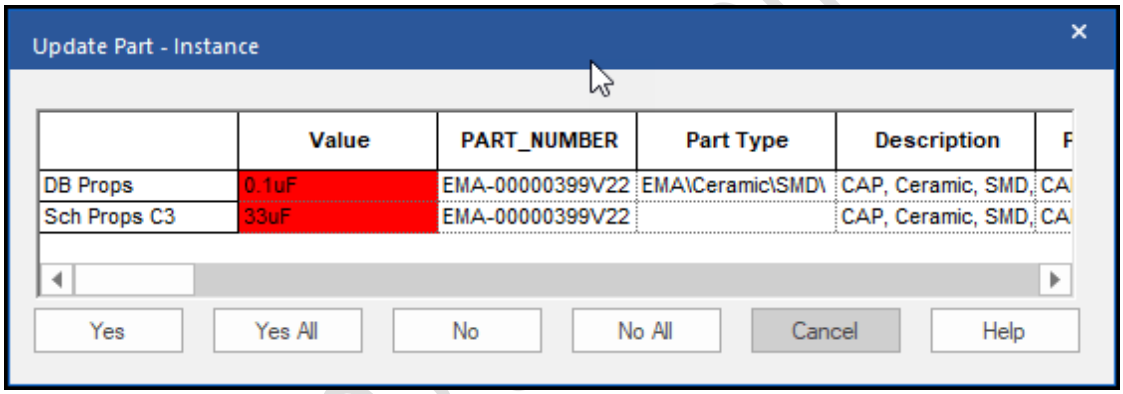
8. Click ***Yes*** to the Undo/Redo message. The **Update Part Instance** window will appear. In the next lab you will work with this window. Do not close it yet.

Lab 3-8: Update Part Instance

The **Update Part Instance** window will appear showing the **Value** highlighted in red, indicating the discrepancy between the schematic value and the database value.

Selecting ***No*** means you ***do not want*** Part Manager to update the part to be current with the database. It is important to note that if this choice is made, this part will be out of sync with the database. The changed part will contain the part number of the database part, which could cause a conflict in the Bill of Materials.

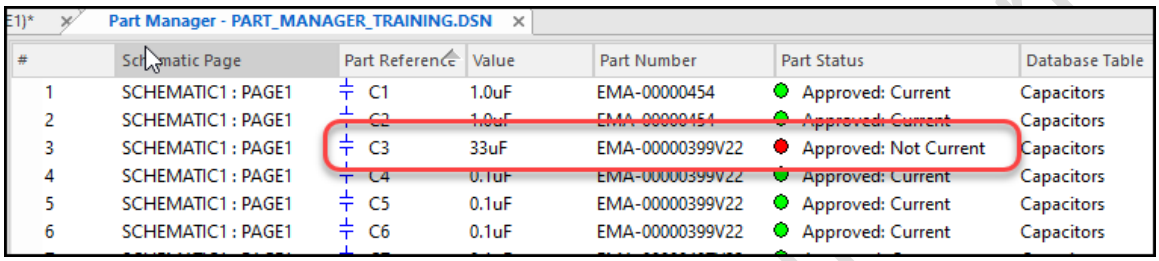
If you need the change the value on a part, you should **first** search the database for the part you need. If it does not exist, consider creating a new TMP part in CIP, or deriving a database part in CIS, so there are no conflicts with corporate part numbers.



1. Click ***No*** to keep the changed part's value at 33uF.
2. Do not close Part Manager.

Lab 3-9: Part Status Notification

Part Manager will now show an ***Approved: Not Current*** status for this part, which means that the part number is found within the database but one or more of the properties do not match. Notice the Part Number is the same as other capacitors, but the values are different.

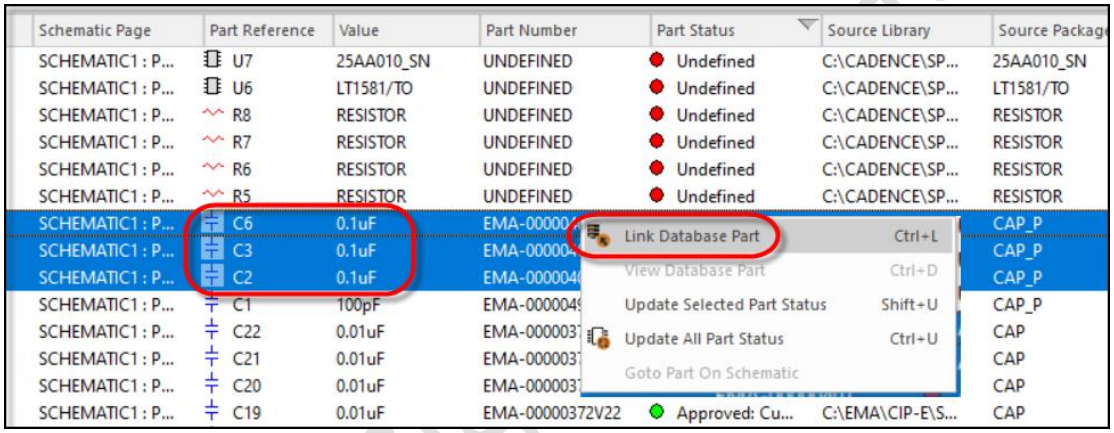


1. Right click on ***C3*** in Part Manager and select ***Update Selected Part Status***. Click ***Yes*** to continue.
2. Select ***Yes*** to update the part value back to the database value of 0.1uF. Part Manager now shows an **Approved: Current** status for this part.

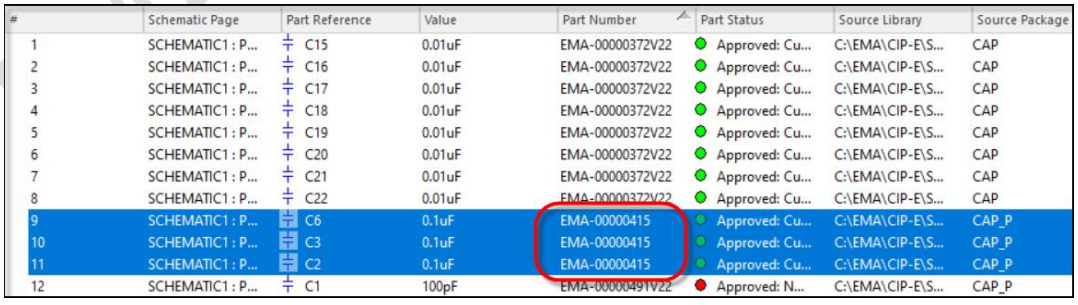
Lab 3-10: Linking a Database Part

There are several situations where you may want to "link" a part to a different database part. For example, you may find that all 10K resistors need to be 100K instead. The linking feature in Part Manager allows you to accomplish this. In the next exercise, you will link a group of capacitors to a different one.

1. In Part Manager, sort the component list based on Value by clicking on the ***Value*** header.
2. Hold down the ***<Shift>*** key and select ***C6, C3*** and ***C2***.
3. Right click and select ***Link Database Part***.



4. Click ***Yes*** to continue. CIS Explorer will appear with a list of capacitors matching the value of C1 and C2 (linking will search based on Value). Select the first capacitor in the list (Part Number ***EMA-00000415***).
5. Double click on the selected part. Click ***OK*** to continue. The Part Manager will reappear. Notice that C6, C3 and C2 are now linked to a different part number.



6. Right click on ***C2*** and select ***Goto Part on Schematic***.
7. Double click on ***C2*** to view its properties.
8. Save the design.

Lab 3-11: Using Link Database Part to Update a Legacy Design

In this exercise you will be updating a schematic design whose parts may be undefined or do not exist in the database. For some parts you will link to existing parts in the database, and for other parts you will have to add new part entries in the database.

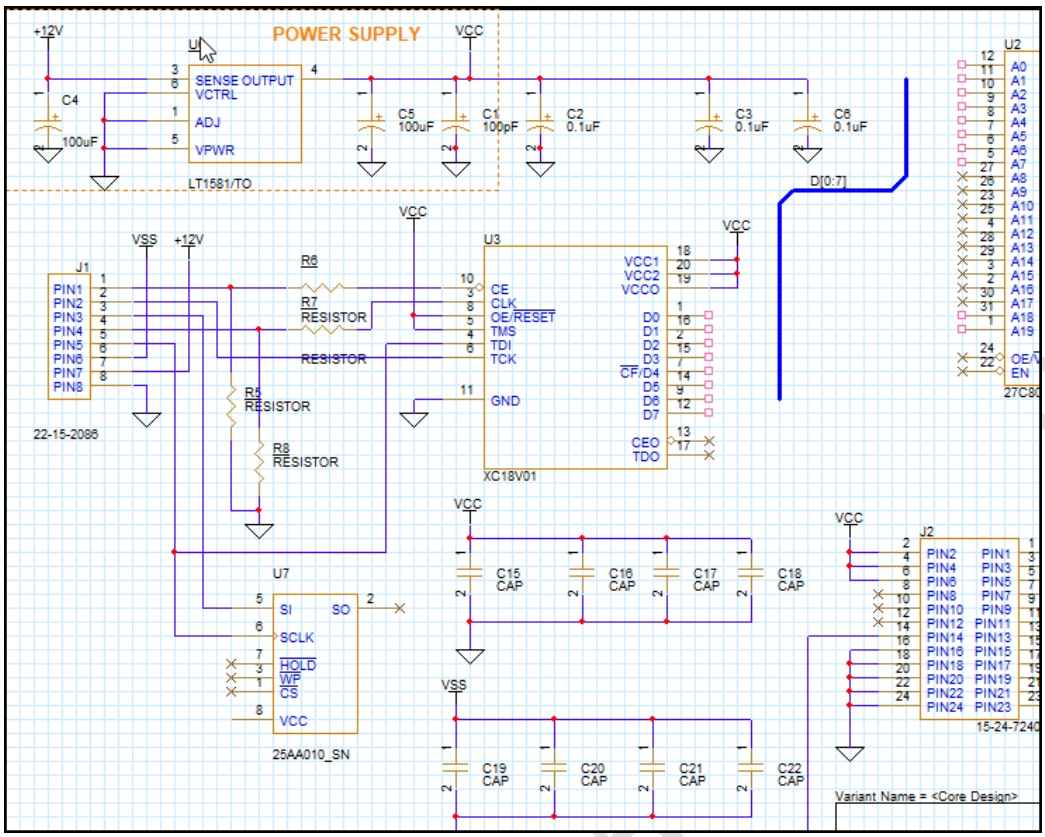
These are all common tasks used when updating older schematics and aligning them with the CIS/CIP database. After this exercise you will be able to:

- Link undefined parts to existing database parts
- Add new parts to the database
- Swap schematic parts for parts that exist in the database
- Ensure that all parts in a schematic are *Approved: Current*

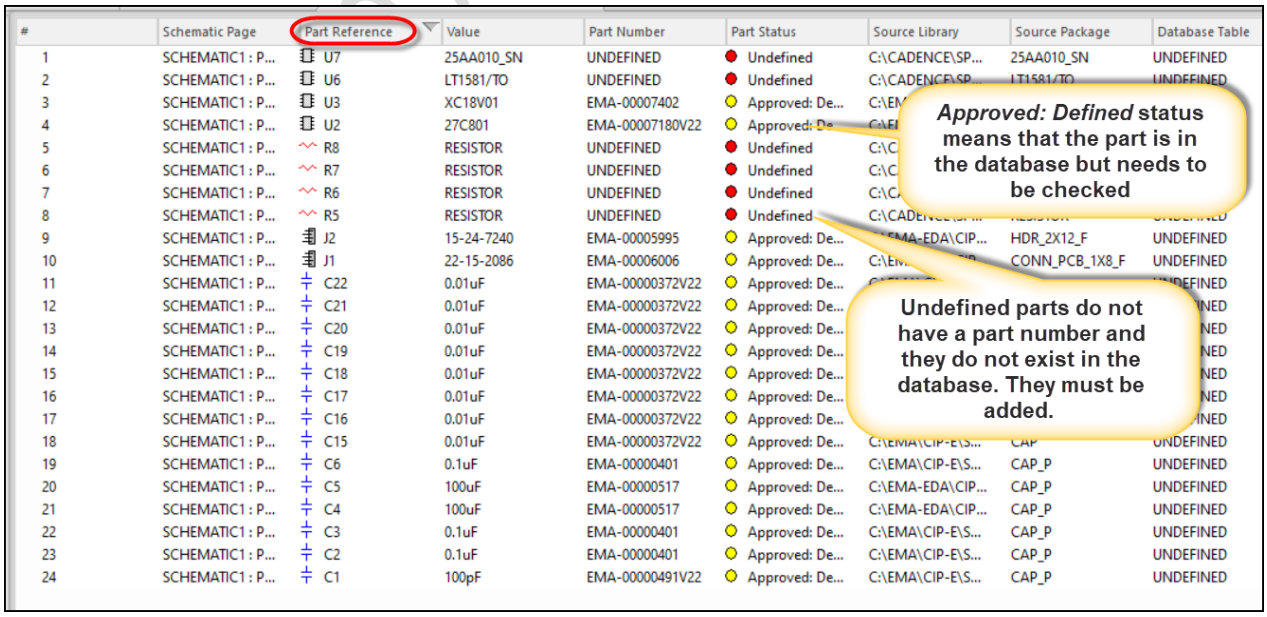
Note *It is important to understand that the lab steps in this lesson are based on the CIP standard database. If you are performing the steps via EMA e-Learning then you may or may not be using the CIP standard database, or you may be using your own companies database in which case the components may vary from the lab.*

Reviewing the Design and Updating All Part Status

1. Open the design `C:\EMA_Training\CIP_CIP_Usage\legacy.dsn`.
2. Open schematic page 1 and review the schematic.



3. Right click on the design in the Project Manager and select ***Part Manager***. Note that many of the Part Status indicators are ***Undefined*** (RED) status. You will be updating these parts.
4. Click the **Part Reference** header to organize the list by reference designator.



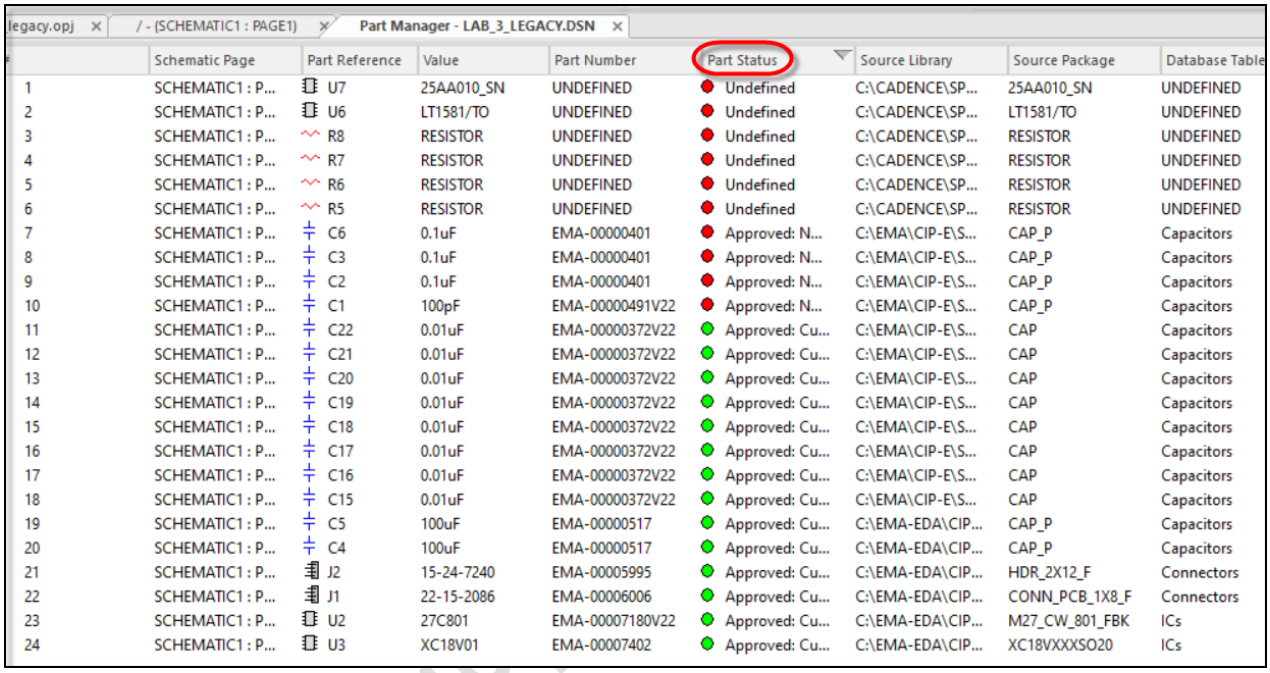
#	Schematic Page	Part Reference	Value	Part Number	Part Status	Source Library	Source Package	Database Table
1	SCHEMATIC1 : P...	U7	25AA010_SN	UNDEFINED	Undefined	C:\CADENCE\SP...	25AA010_SN	UNDEFINED
2	SCHEMATIC1 : P...	U6	LT1581/TO	UNDEFINED	Undefined	C:\CADENCE\SP...	LT1581/TO	UNDEFINED
3	SCHEMATIC1 : P...	U3	XC18V01	EMA-00007402	Approved: De...	C:\EM		
4	SCHEMATIC1 : P...	U2	27C801	EMA-00007180V22	Approved: De...	C:\EM		
5	SCHEMATIC1 : P...	R8	RESISTOR	UNDEFINED	Undefined	C:\C		
6	SCHEMATIC1 : P...	R7	RESISTOR	UNDEFINED	Undefined	C:\C		
7	SCHEMATIC1 : P...	R6	RESISTOR	UNDEFINED	Undefined	C:\C		
8	SCHEMATIC1 : P...	R5	RESISTOR	UNDEFINED	Undefined	C:\CADENCE\SP...		
9	SCHEMATIC1 : P...	J2	15-24-7240	EMA-00005995	Approved: De...	EMA-EDA\CIP...	HDR_2X12_F	UNDEFINED
10	SCHEMATIC1 : P...	J1	22-15-2086	EMA-00006006	Approved: De...	C:\EM	CONN_PCB_1X8_F	UNDEFINED
11	SCHEMATIC1 : P...	C22	0.01uF	EMA-00000372V22	Approved: De...			
12	SCHEMATIC1 : P...	C21	0.01uF	EMA-00000372V22	Approved: De...			
13	SCHEMATIC1 : P...	C20	0.01uF	EMA-00000372V22	Approved: De...			
14	SCHEMATIC1 : P...	C19	0.01uF	EMA-00000372V22	Approved: De...			
15	SCHEMATIC1 : P...	C18	0.01uF	EMA-00000372V22	Approved: De...			
16	SCHEMATIC1 : P...	C17	0.01uF	EMA-00000372V22	Approved: De...			
17	SCHEMATIC1 : P...	C16	0.01uF	EMA-00000372V22	Approved: De...			
18	SCHEMATIC1 : P...	C15	0.01uF	EMA-00000372V22	Approved: De...			
19	SCHEMATIC1 : P...	C6	0.1uF	EMA-00000401	Approved: De...	C:\EMA\CIP-E\S...	CAP_P	UNDEFINED
20	SCHEMATIC1 : P...	C5	100uF	EMA-00000517	Approved: De...	C:\EMA-EDA\CIP...	CAP_P	UNDEFINED
21	SCHEMATIC1 : P...	C4	100uF	EMA-00000517	Approved: De...	C:\EMA-EDA\CIP...	CAP_P	UNDEFINED
22	SCHEMATIC1 : P...	C3	0.1uF	EMA-00000401	Approved: De...	C:\EMA\CIP-E\S...	CAP_P	UNDEFINED
23	SCHEMATIC1 : P...	C2	0.1uF	EMA-00000401	Approved: De...	C:\EMA\CIP-E\S...	CAP_P	UNDEFINED
24	SCHEMATIC1 : P...	C1	100pF	EMA-00000491V22	Approved: De...	C:\EMA\CIP-E\S...	CAP_P	UNDEFINED

Approved: Defined **status means that the part is in the database but needs to be checked**

Undefined parts do not have a part number and they do not exist in the database. They must be added.

5. Begin by checking the part properties for every part in the list. Right click on any component and select ***Update All Part Status***. Click ***Yes*** (or Yes All) to the message to continue.

Some of the parts are now ***Approved: Current*** (GREEN). This is the ideal status and what you will be targeting for the rest of the parts in this schematic. Notice that some of the parts remain as ***Undefined***. These are the parts that need to be added to the database.



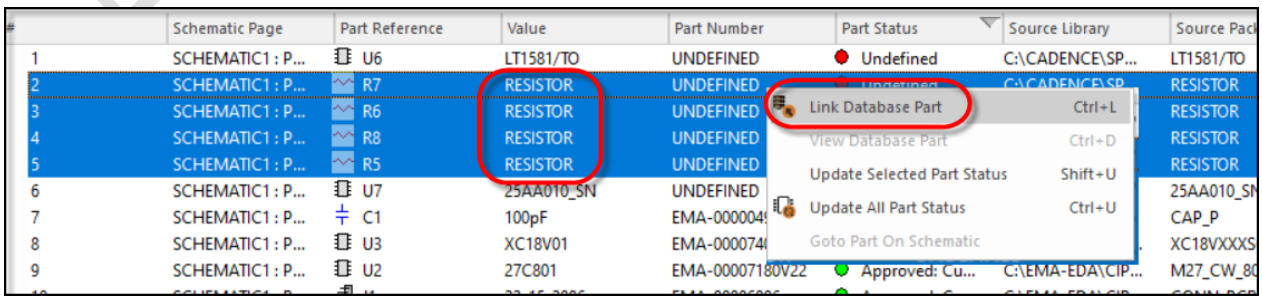
	Schematic Page	Part Reference	Value	Part Number	Part Status	Source Library	Source Package	Database Table
1	SCHEMATIC1 : P...	U7	25AA010_SN	UNDEFINED	Undefined	C:\CADENCE\SP...	25AA010_SN	UNDEFINED
2	SCHEMATIC1 : P...	U6	LT1581/TO	UNDEFINED	Undefined	C:\CADENCE\SP...	LT1581/TO	UNDEFINED
3	SCHEMATIC1 : P...	R8	RESISTOR	UNDEFINED	Undefined	C:\CADENCE\SP...	RESISTOR	UNDEFINED
4	SCHEMATIC1 : P...	R7	RESISTOR	UNDEFINED	Undefined	C:\CADENCE\SP...	RESISTOR	UNDEFINED
5	SCHEMATIC1 : P...	R6	RESISTOR	UNDEFINED	Undefined	C:\CADENCE\SP...	RESISTOR	UNDEFINED
6	SCHEMATIC1 : P...	R5	RESISTOR	UNDEFINED	Undefined	C:\CADENCE\SP...	RESISTOR	UNDEFINED
7	SCHEMATIC1 : P...	C6	0.1uF	EMA-00000401	Approved: N...	C:\EMA\CIP-E\S...	CAP_P	Capacitors
8	SCHEMATIC1 : P...	C3	0.1uF	EMA-00000401	Approved: N...	C:\EMA\CIP-E\S...	CAP_P	Capacitors
9	SCHEMATIC1 : P...	C2	0.1uF	EMA-00000401	Approved: N...	C:\EMA\CIP-E\S...	CAP_P	Capacitors
10	SCHEMATIC1 : P...	C1	100pF	EMA-00000491V22	Approved: N...	C:\EMA\CIP-E\S...	CAP_P	Capacitors
11	SCHEMATIC1 : P...	C22	0.01uF	EMA-00000372V22	Approved: Cu...	C:\EMA\CIP-E\S...	CAP	Capacitors
12	SCHEMATIC1 : P...	C21	0.01uF	EMA-00000372V22	Approved: Cu...	C:\EMA\CIP-E\S...	CAP	Capacitors
13	SCHEMATIC1 : P...	C20	0.01uF	EMA-00000372V22	Approved: Cu...	C:\EMA\CIP-E\S...	CAP	Capacitors
14	SCHEMATIC1 : P...	C19	0.01uF	EMA-00000372V22	Approved: Cu...	C:\EMA\CIP-E\S...	CAP	Capacitors
15	SCHEMATIC1 : P...	C18	0.01uF	EMA-00000372V22	Approved: Cu...	C:\EMA\CIP-E\S...	CAP	Capacitors
16	SCHEMATIC1 : P...	C17	0.01uF	EMA-00000372V22	Approved: Cu...	C:\EMA\CIP-E\S...	CAP	Capacitors
17	SCHEMATIC1 : P...	C16	0.01uF	EMA-00000372V22	Approved: Cu...	C:\EMA\CIP-E\S...	CAP	Capacitors
18	SCHEMATIC1 : P...	C15	0.01uF	EMA-00000372V22	Approved: Cu...	C:\EMA\CIP-E\S...	CAP	Capacitors
19	SCHEMATIC1 : P...	C5	100uF	EMA-00000517	Approved: Cu...	C:\EMA-EDA\CIP...	CAP_P	Capacitors
20	SCHEMATIC1 : P...	C4	100uF	EMA-00000517	Approved: Cu...	C:\EMA-EDA\CIP...	CAP_P	Capacitors
21	SCHEMATIC1 : P...	J2	15-24-7240	EMA-00005995	Approved: Cu...	C:\EMA-EDA\CIP...	HDR_2X12_F	Connectors
22	SCHEMATIC1 : P...	J1	22-15-2086	EMA-00006006	Approved: Cu...	C:\EMA-EDA\CIP...	CONN_PCB_1X8_F	Connectors
23	SCHEMATIC1 : P...	U2	27C801	EMA-00007180V22	Approved: Cu...	C:\EMA-EDA\CIP...	M27_CW_801_FBK	ICs
24	SCHEMATIC1 : P...	U3	XC18V01	EMA-00007402	Approved: Cu...	C:\EMA-EDA\CIP...	XC18VXXXSO20	ICs

6. Sort the parts in the list by clicking on the ***Part Status*** header. An efficient way to replace multiple parts is to select them as a group and use ***Link Database Part***.

Linking the Resistors

Next, you will use Link Database Part to replace the undefined resistors.

1. In Part Manager, ***<Shift>-select*** all components with the value ***RESISTOR***. Right click and select ***Link Database Part***. Click ***Yes*** to continue.

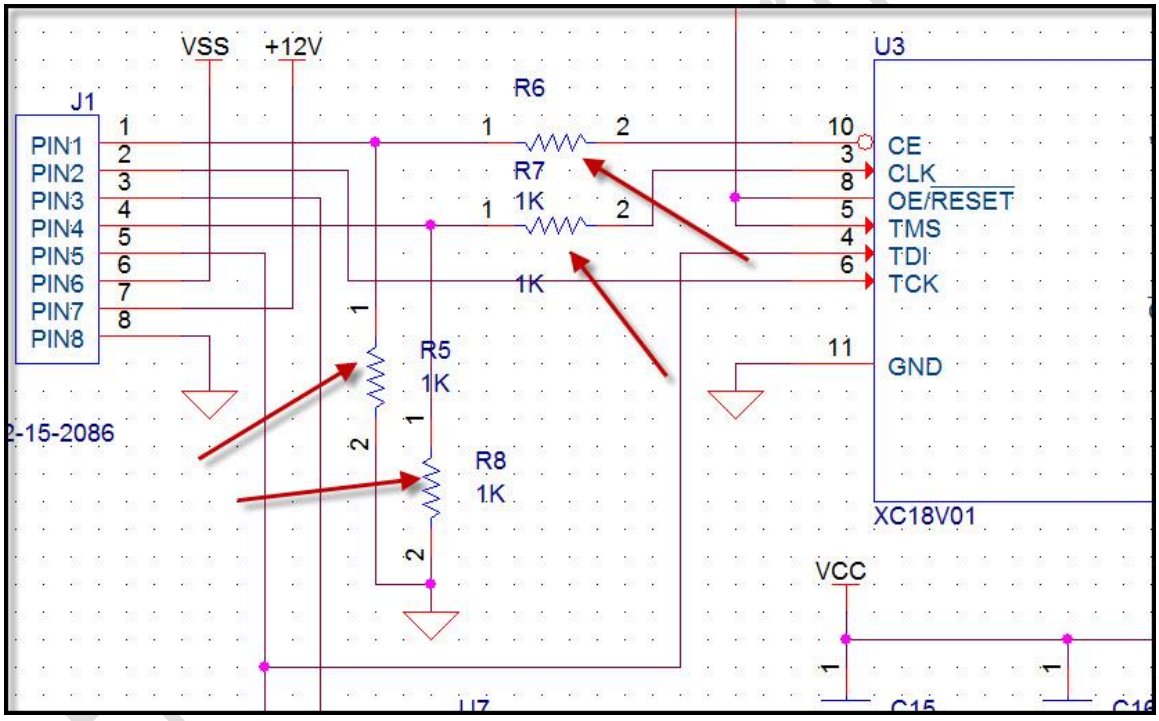


Note there is no component in CIS Explorer that has the value of RESISTOR.

2. In the ***Query*** tab, set the ***Property*** field to ***Value*** and the ***Compare*** field to **=**. Enter ***1k*** in the ***Value*** field, then hit ***<Enter>*** to begin the search.
3. In the Parts List, select the component with the Part Number ***EMA-00007504V42***. Notice the part turns yellow. This is because the graphical representation of the part is different than the current schematic symbol.
4. Double click on the part to replace the schematic part with this part.
5. Click ***OK*** to the message that the parts differ from the database part. The newly updated part is shown in Part Manager.
6. Right click on any part and select ***Update All Part Status*** to recheck all parts against the database. Click ***Yes*** to continue.

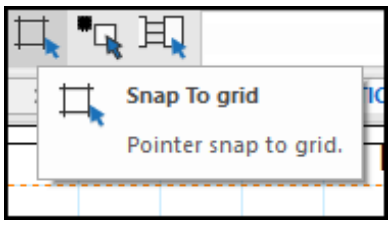
Note that the RESISTORS are now updated to the Part Number EMA-00007504V42 and have an ***Approved: Current*** (GREEN) status.

7. Select one of the updated resistors in Part Manager, right click, and select ***Goto Part on Schematic***.



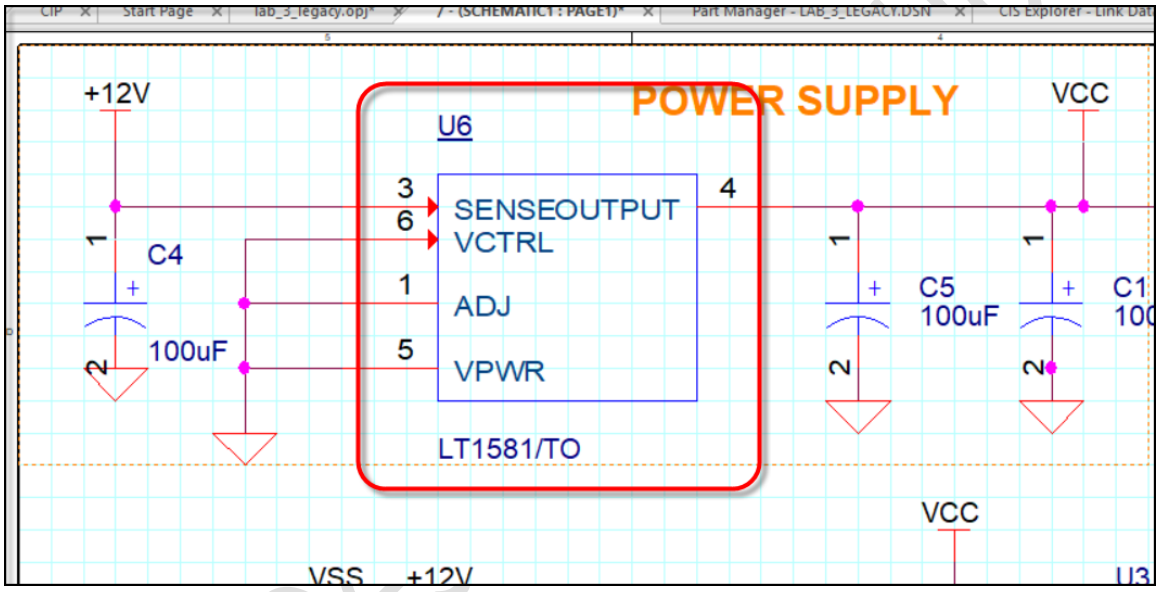
It is always important to check the connectivity when you replace parts using Link Database Part. As you can see, the connectivity may be intact, but you may want to adjust the reference designator and value text placement.

Tip: If you are adjusting the placement of text on the schematic, you can turn off ***Snap to Grid*** so the text can be placed closer to the component. ***Always turn Snap to Grid back on before sliding wires or making connections!***



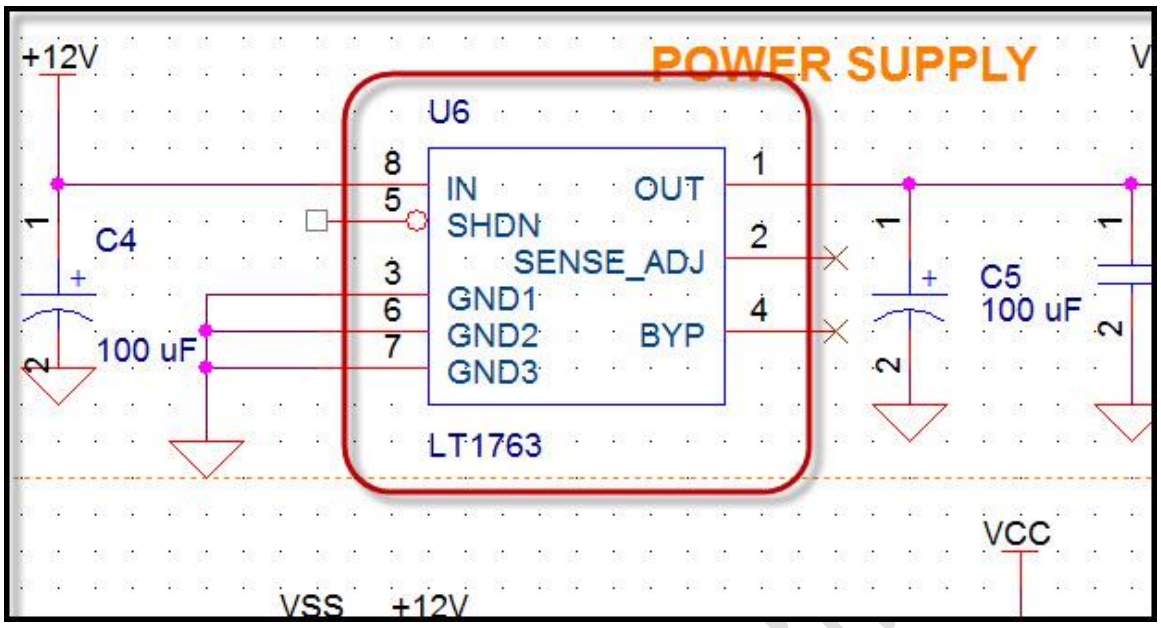
Replacing U6

In the next step, you will update the regulator (U6) located in the upper left area of the schematic.

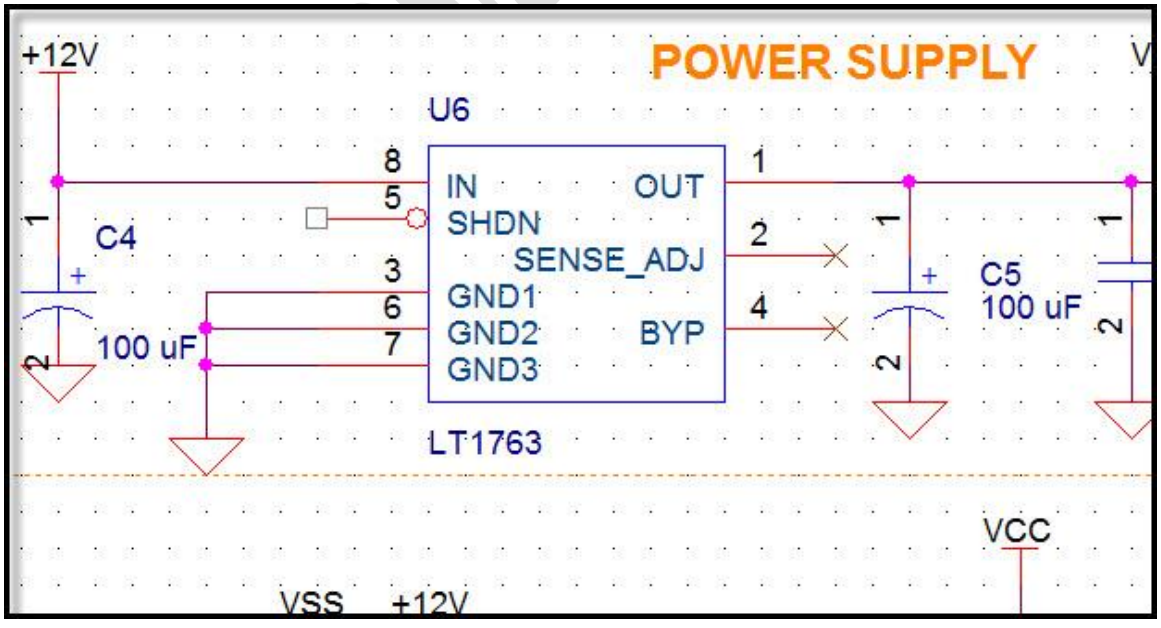


1. In Part Manager, sort by ***Part Status*** so that the remaining undefined parts are at the top of the list.
2. Right click on ***U6*** and select ***Link Database Part***. Click ***Yes*** to continue. There are no parts with this value in CIS Explorer.

The regulator part in the database is shown below (the schematic has been rewired to connect the new part). The pinouts are different, so you will need to make sure the connectivity is updated when you update the part.

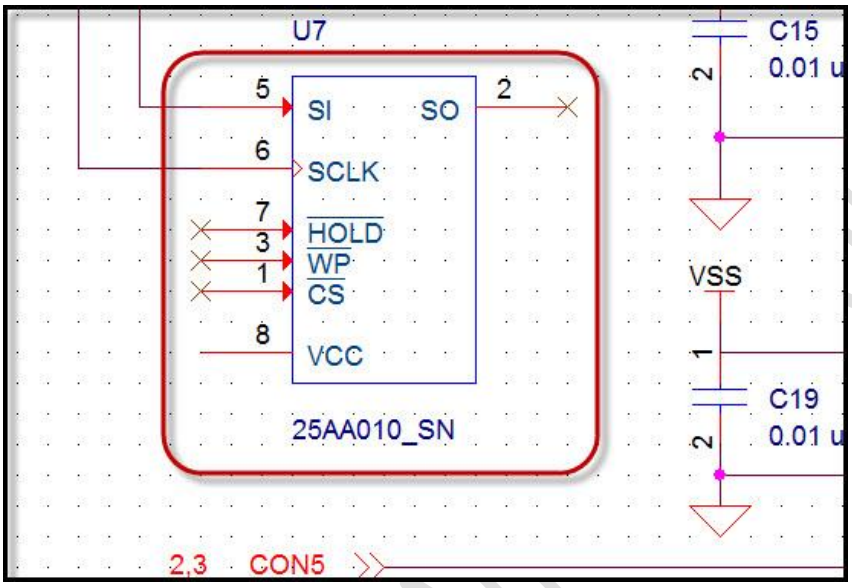


3. In the ***Query*** tab of CIS Explorer, search for a part with the value of ***LT1763***. There are several components that show up in the parts list.
4. Select the component with the Part Number ***EMA-00006787V22***. Double click on the part. Click ***OK*** to continue.
5. Part Manager now shows U6 as an LT1763. Right click on ***U6*** and select ***Update All Part Status***. Click ***Yes*** to Continue. U6 is now updated.
6. Right click on ***U6*** and select ***Goto Part on Schematic***. Edit the connectivity to match the following graphic (be sure to turn on Snap To Grid if it was turned off in the previous exercise).



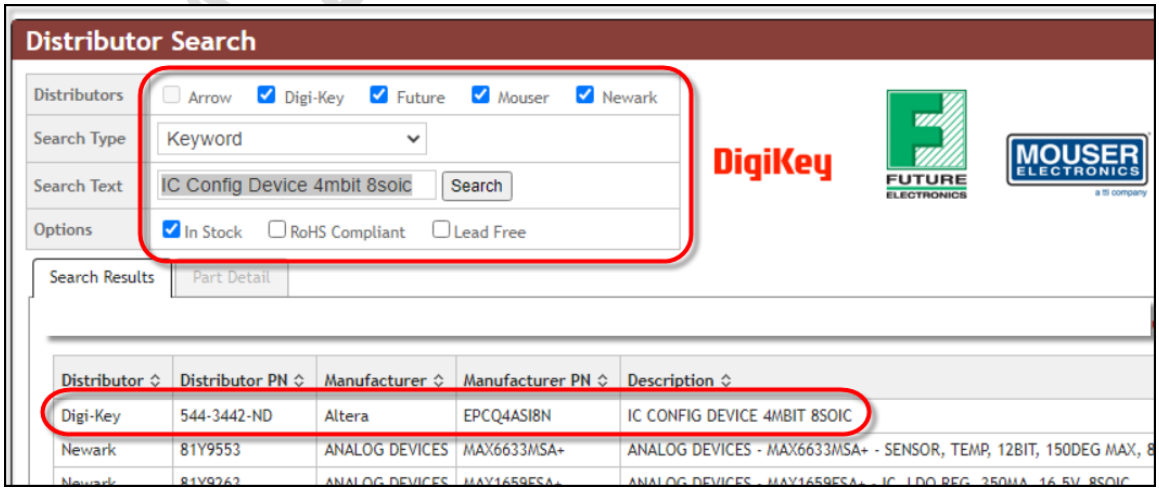
Adding the New PROM

The final IC you will update is the PROM, located in the lower left side of the schematic page.

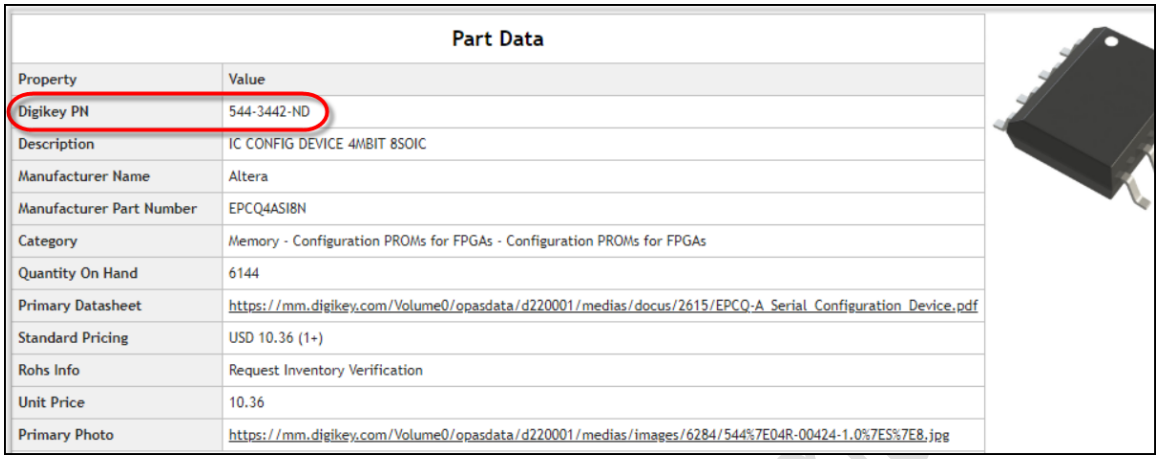


This part does not exist in the database so you will create a new part entry using a different part. You will do this from Component Information Portal.

1. Open CIP and log in.
2. Select the ***Distributor Search*** tab.
3. Select the ***Digi-Key***, ***Future***, ***Mouser***, and ***Newark*** distributors.
4. Set the ***Search Type*** to ***Keyword*** and enter the keyword ***IC Config Device 4mbit 8soic***.
5. Click the ***Search*** button to start the search.



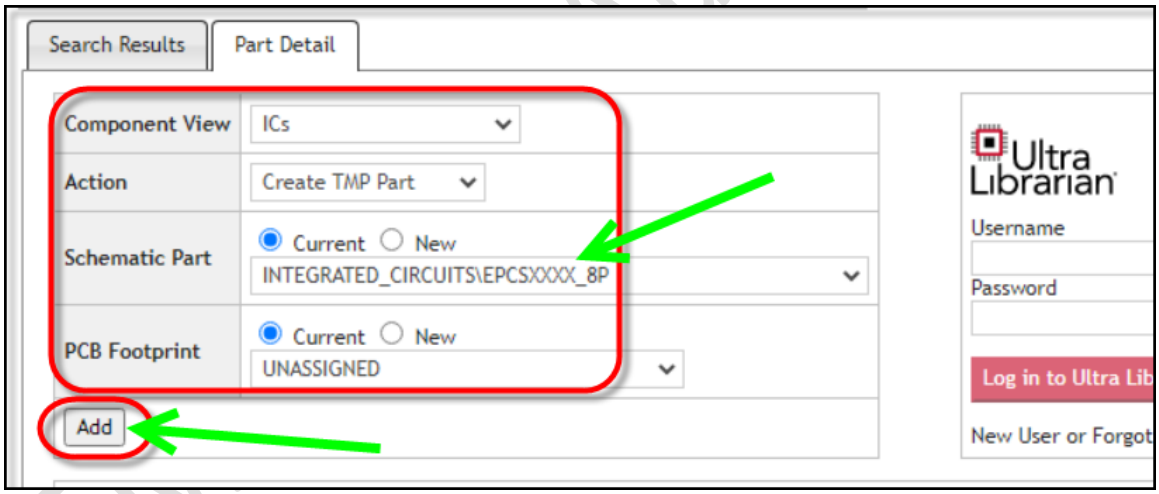
6. When the Search Results return, select the **Altera** component with the manufacturing PN of ***EPCQ4AS18N*** and the Digi-Key Distributor PN ***544-3442-ND***.
7. Scroll down to view the Part Detail.



Part Data

Property	Value
Digikey PN	544-3442-ND
Description	IC CONFIG DEVICE 4MBIT 8SOIC
Manufacturer Name	Altera
Manufacturer Part Number	EPCQ4ASI8N
Category	Memory - Configuration PROMs for FPGAs - Configuration PROMs for FPGAs
Quantity On Hand	6144
Primary Datasheet	https://mm.digikey.com/Volume0/opasdata/d220001/medias/docus/2615/EPCQ-A_Serial_Configuration_Device.pdf
Standard Pricing	USD 10.36 (1+)
Rohs Info	Request Inventory Verification
Unit Price	10.36
Primary Photo	https://mm.digikey.com/Volume0/opasdata/d220001/medias/images/6284/544%7E04R-00424-1.0%7ES%7E8.jpg

8. Select ***ICs*** in the ***Component View*** dropdown.
9. Leave the ***Action*** set to ***Create TMP Part***.
10. Map the Schematic Part to **INTEGRATED_CIRCUITS\EPCSXXXX_8P**.
11. Leave the ***PCB Footprint*** set to ***UNASSIGNED***.



12. Click ***Add*** to add the part to the database.

Assigning Properties and Part Number to the PROM

Next, you will add and edit properties, such as Part Number and Part Type.

1. Click the Edit button  to go into edit mode.
2. In the PART_NUMBER field, change the part number to ***EMA-NEWPART-01***.
3. Change the ***Package Size*** to ***SOIC-8***.

4. Change the ***Package Type*** to ***SMD***.
5. Change the ***Company Part Status*** to ***Preliminary***.
6. Change the ***Device Type*** to ***Programmable Devices***.
7. Change the ***Number of Pins*** to **8**.
8. In the ***Part Type*** field enter ***EMA\SMD\Programmable Devices\Config Device***.
9. Click on the ***Update*** button to enter the changes.

Now that you have entered the part, you will go back to Part Manager and use ***Link Database Part*** to swap the old PROM with the newly created part.

Replacing the PROM in Part Manager

1. Reopen Part Manager if it is not already open.
2. Right click on ***U7*** and select ***Link Database Part***. Click ***Yes*** to continue.

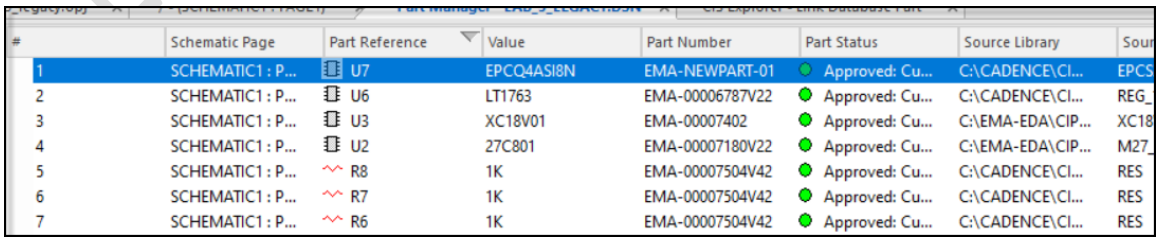
CIS Explorer opens – there is no part that matches this value. You will replace this part with the one you created.

3. Using the ***Explore*** tab, expand the categories for **ICs\EMA\SMD\Programmable Devices\Config Device**.
4. In the Part List, select the part you just generated in CIP, ***EMA-NEWPART-01***.

Alternately, you could use the ***Query*** tab to search for the value ***EPCQ4AS18N***. Notice the part turns yellow since the schematic symbol is a different representation from the part being replaced. Also notice some of the properties in the properties window are highlighted in red, indicating the differences between this part and the part on the schematic.

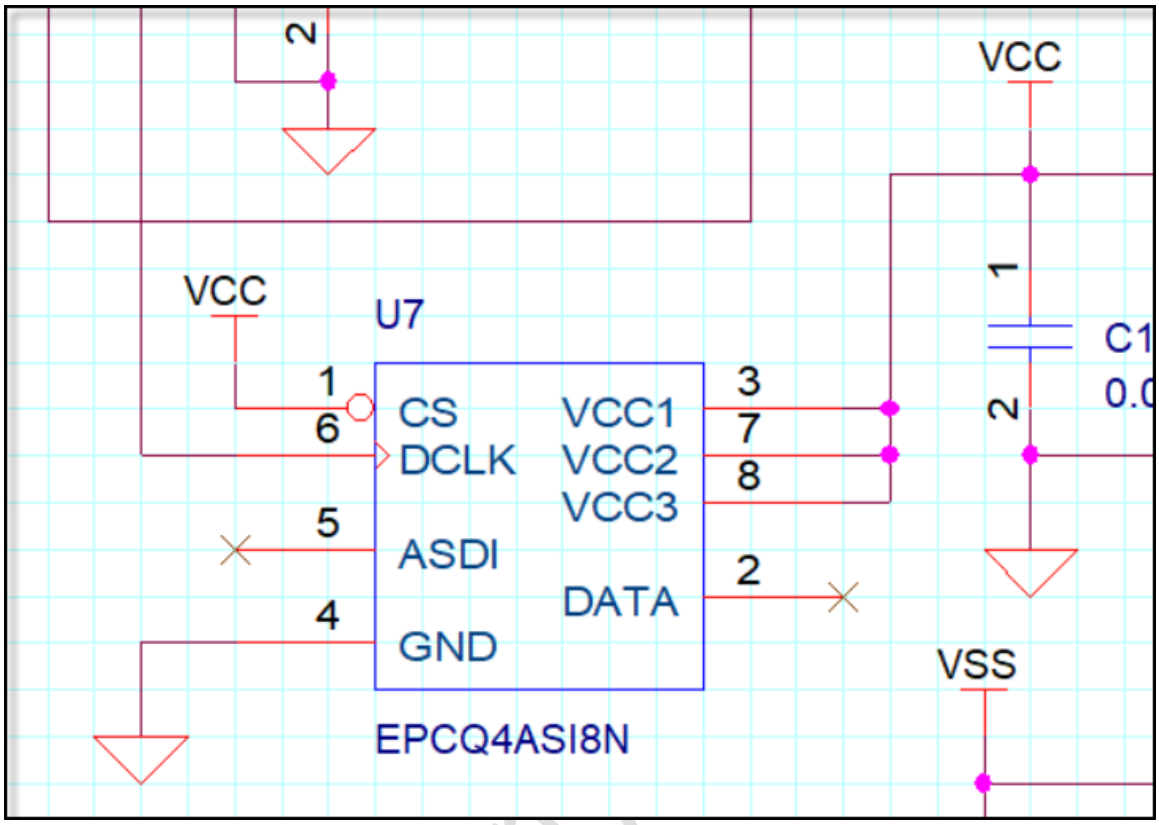
5. Double click on the part to replace it in the schematic. There is a message indicating the part currently on the schematic differs graphically from the part replacing it. You will need to check the connectivity of this part on the schematic. Click ***OK*** to continue.
6. In Part Manager, ***U7*** is replaced with ***EMA-NEWPART-01***. Right click on ***U7*** and select ***Update All Part Status***.

All part status indicators in Part Manager should now reflect ***Approved: Current*** status.



#	Schematic Page	Part Reference	Value	Part Number	Part Status	Source Library	Sour
1	SCHEMATIC1 : P...	U7	EPCQ4ASI8N	EMA-NEWPART-01	Approved: Cu...	C:\CADENCE\CI...	EPCS
2	SCHEMATIC1 : P...	U6	LT1763	EMA-00006787V22	Approved: Cu...	C:\CADENCE\CI...	REG_
3	SCHEMATIC1 : P...	U3	XC18V01	EMA-00007402	Approved: Cu...	C:\EMA-EDA\CIP...	XC18
4	SCHEMATIC1 : P...	U2	27C801	EMA-00007180V22	Approved: Cu...	C:\EMA-EDA\CIP...	M27_
5	SCHEMATIC1 : P...	R8	1K	EMA-00007504V42	Approved: Cu...	C:\CADENCE\CI...	RES
6	SCHEMATIC1 : P...	R7	1K	EMA-00007504V42	Approved: Cu...	C:\CADENCE\CI...	RES
7	SCHEMATIC1 : P...	R6	1K	EMA-00007504V42	Approved: Cu...	C:\CADENCE\CI...	RES

7. In Part Manager, right click on ***U7*** and select ***Goto Part on Schematic***.



Checking the Connectivity

You will need to adjust the connectivity on this part. Using the illustration above, change the connectivity on U7 to match the following:

1. ***U7 pin 6*** connects to ***J1 pin 3***.
2. ***U7 pins 1, 3, 7, and 8*** connects to ***VCC***.
3. ***U7 pin 4*** connects to ***GND***.
4. ***U7 pins 2 and 5*** are No Connects  .

The newly replaced component should look like the above graphic.

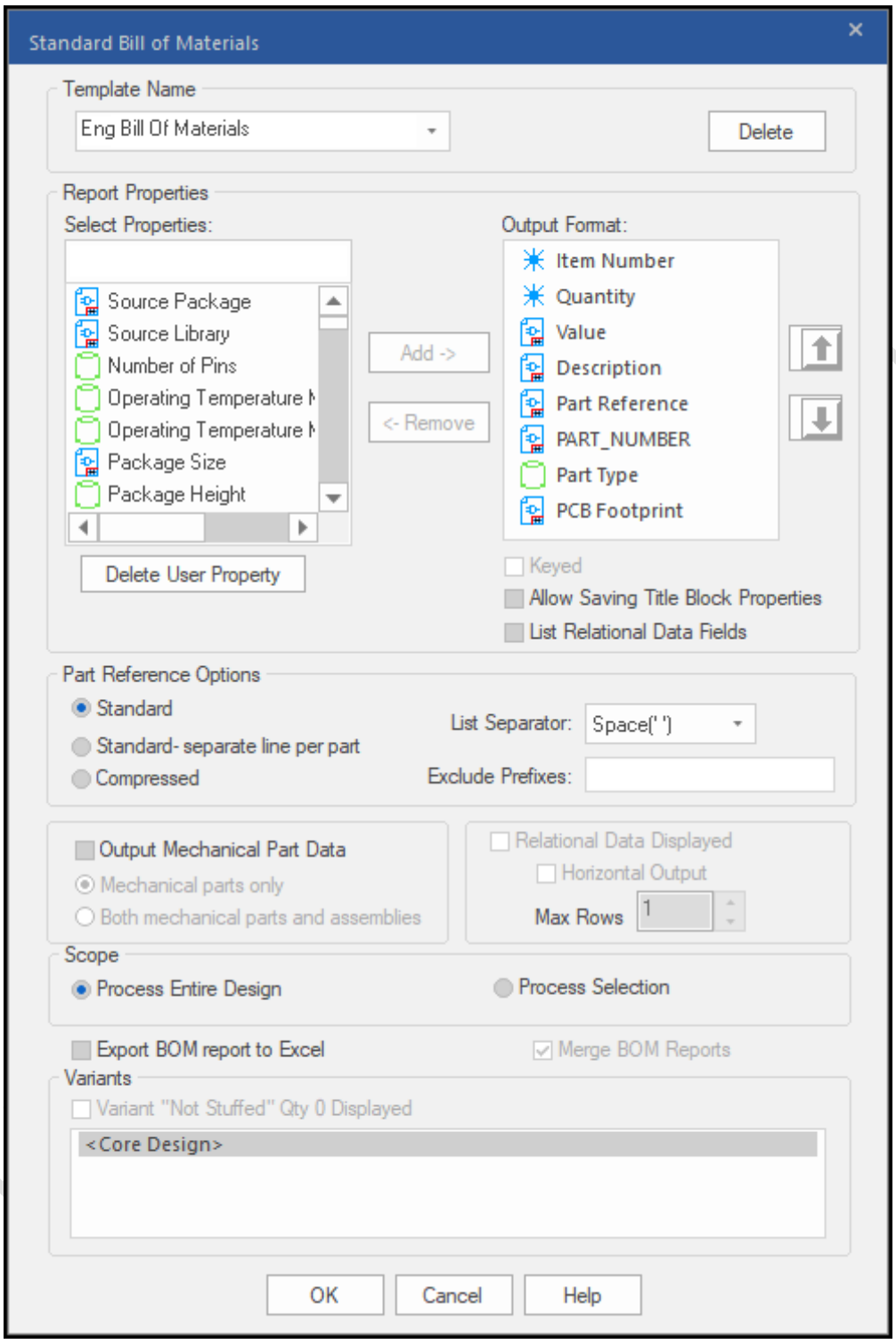
It is not necessary to finish the schematic page. This exercise is now complete. Close and save the design.

Lesson 4: Finalizing and Documenting the Design

This section will deal with creating Bills of Materials and using CIS to create and manage design variants. Variations on a design are common in cases where you may have the same schematic for differing products where the value of components may be different depending on product requirements.

Standard Bill of Materials

You can use the standard CIS Bill of Materials feature to create multiple named report templates so that you can generate separate bills of materials for the different requirements. This is useful when preparing BOMs for purchasing and manufacturing.

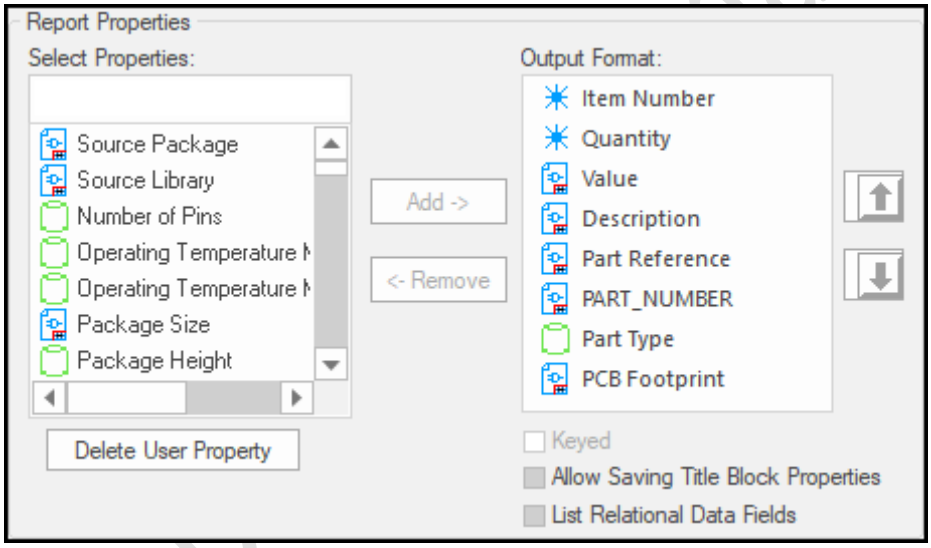


In the ***Report Properties*** section, you can select any property in the ***Select Properties*** list and click ***Add*** to add it to the ***Output Format*** of the final Bill of Materials. In the ***Output Format*** section, the properties can be ordered by selecting the property and clicking the UP or DOWN arrows located on the right.

The following table identifies the property icons and their descriptions.

Icon	Description
	Default CIS Property
	The property is transferred from the parts database to the placed schematic part
	The property comes directly from the parts database
	Title Block property
	The property is transferred from ICA

These property icons appear in the ***Report Properties*** area.



Keyed Option

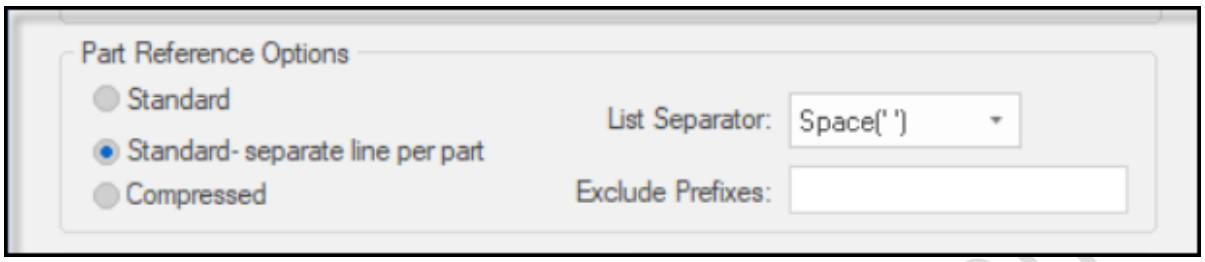
The ***Keyed*** option allows you to check to key the selected property in the ***Output Format*** list. Keyed properties are grouped as a single item in the report. At least one property must be keyed. Typically, the Part Number property would be keyed.

Allow Saving Title Block Properties

Check this if you want to save the title block property information along with the BOM information when you save the BOM report as a BOM; .CSV; or .FWC file.

Part Reference Options

Several options in this area will help you control how the BOM out appears.



Standard

Groups parts with matching keyed properties on a single line.

Standard-separate line per part

Lists each part on its own line.

Compressed

Presents a group of the same parts on a single line and compresses the part references into a range whenever possible.

List Separator

Choose either a space or a comma to separate the part references.

Exclude Prefixes

Allows you to enter in the prefixes you want to exclude from the report. To exclude more than one prefix, enter the prefixes separated by spaces.

Output Mechanical Data

This area deals with mechanical parts and assemblies.

Mechanical Parts Only

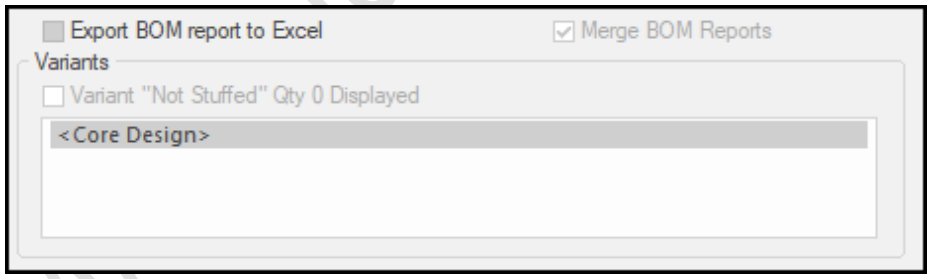
Displays the quantity of all the mechanical parts, including the ones that are in the assemblies.

Both Mechanical Parts and Assemblies

Displays only those mechanical parts and assemblies that are available at the root level of your design. In this case, mechanical parts within the assemblies will not be displayed.

Export BOM Report to Excel

Select this option to export a report in spreadsheet format. It will open automatically in Excel when you generate a Bill of Materials.



Merge BOM Reports

If this option is selected, and you select more than one variant in the Variant List box, the Bill of Materials report contains all BOM variant data separated by variant, in the same report. You cannot sort the report when you merge BOM reports into one report.

Variants

Select either the core design or one or more of the design variants for which you want to generate the Bill of Materials.

Standard BOM Output

Below is a sample of the “Standard” Bill of Materials output.

Capture CIS Standard Bill Of Materials - Standard Report
Report Created on Wednesday Apr 08 16:47:45 2020

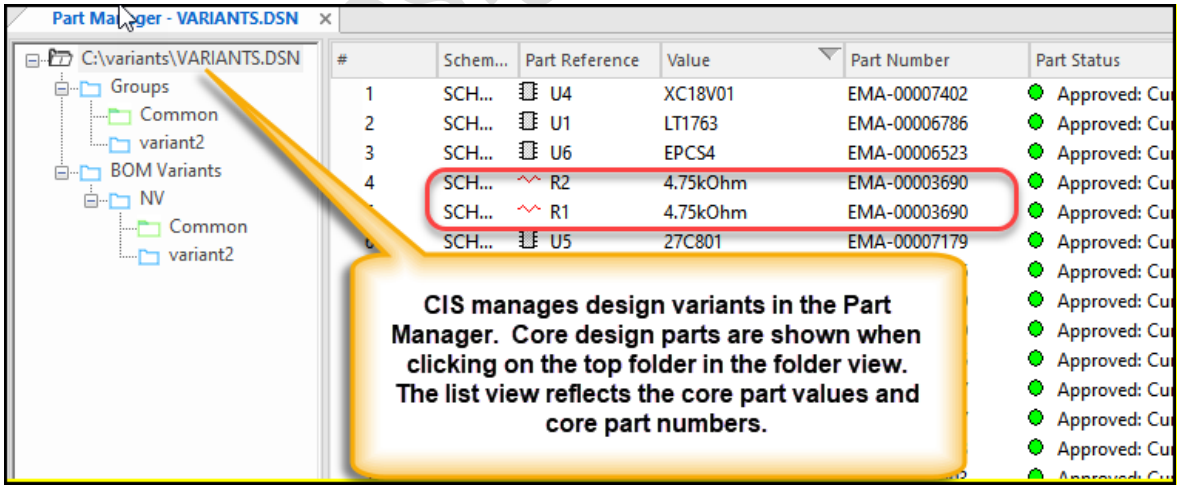
Item Number	Quantity	Value	Description	Part Reference	PART_NUMBER	Part Type	PCB Footprint
1	1	100pF	CAP, Ceramic,...	C1	EMA-00000491V22	EMA\Cera...	CAPC1608X86N
2	3	0.1uF	CAP, Ceramic,...	C2 C3 C6	EMA-00000401	EMA\Cera...	CAPC1608X86N
3	2	100uF	CAP, Tantalu...	C4 C5	EMA-00000517	EMA\Tanta...	CAPMP6032X280N
4	8	0.01uF	CAP, Ceramic,...	C15 C16 C17 C18 C1...	EMA-00000374V22	EMA\Cera...	CAPC1608X86N
5	1	22-1...	CONN, Heade...	J1	EMA-00006006	EMA\Head...	MOLEX_4455A-8
6	1	15-2...	CONN, Heade...	J2	EMA-00005995	EMA\Head...	MOLEX_42385-24
7	4	1K	RES, Thick Fil...	R5 R6 R7 R8	EMA-00007504V42	EMA\SMD\...	RESC1608X55N
8	1	27C801	IC, Memory D...	U2	EMA-00007180V22	EMA\SMD\...	PLCC127P1244X1...

Design Variants

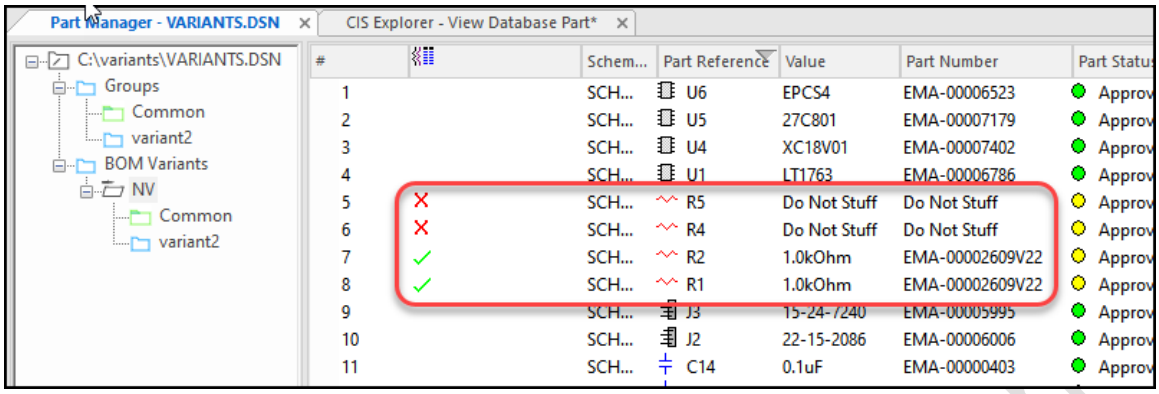
A design variant refers to variations of components in a core design, resulting in different product assemblies or functions. For each design variation, some components may change, and some may not be present.

Design variants are needed to support different models of the product (for example, memory, capacity, speed, or additional functionality), and to support different requirements, depending on where the product is sold. Having a single PCB board reduces cost by allowing a single board design to represent multiple products.

Design variants are also needed based on the manufacturing requirements in different countries.



To view the variant parts, click in the variant folder(s). Parts can either be identified as a different part value (with a different part number), or as **Do Not Stuff**.



Terminology

Groups – Describes multiple components generally used to support a particular function or module (for example, a power or memory module). These components are defined as a group and have varying version numbers.

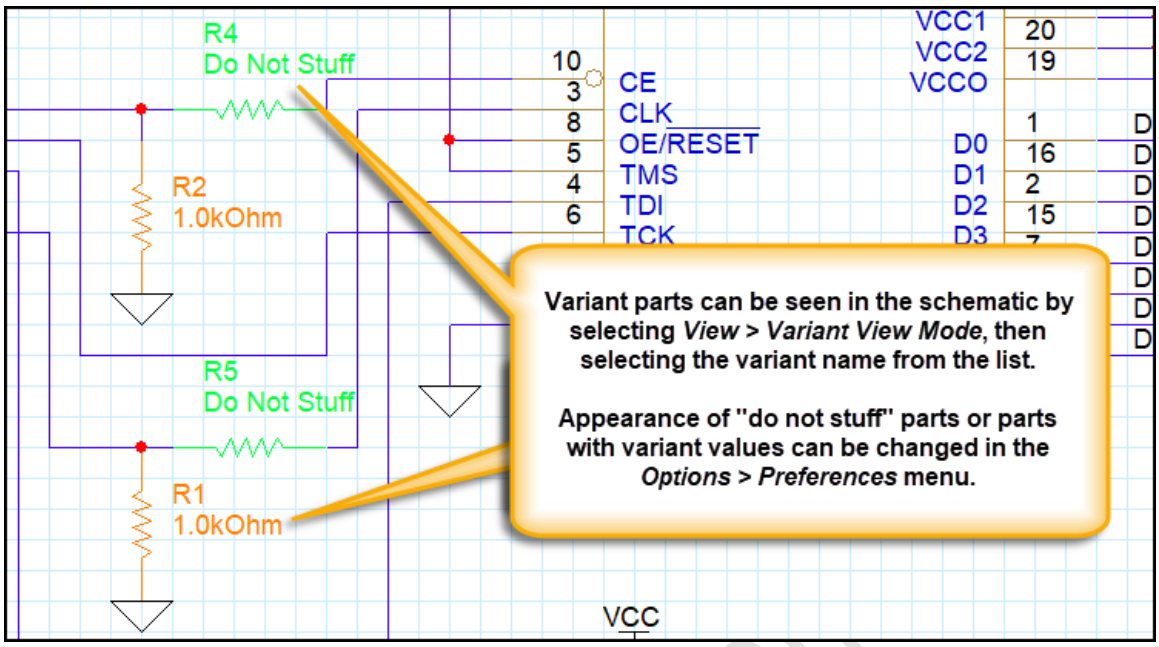
Subgroups – Each subgroup represents a version or assembly of the parent group. For example, if your power module has different assemblies for Europe and Asia, then the Power group would have two subgroups. The set of components in each subgroup is the same as the parent group.

Core Design – The core design is the base schematic and PCB from which design variants can be created.

Common – All components that are not part of a group but are still part of the core design. These modules or functions remain unchanged in all assemblies

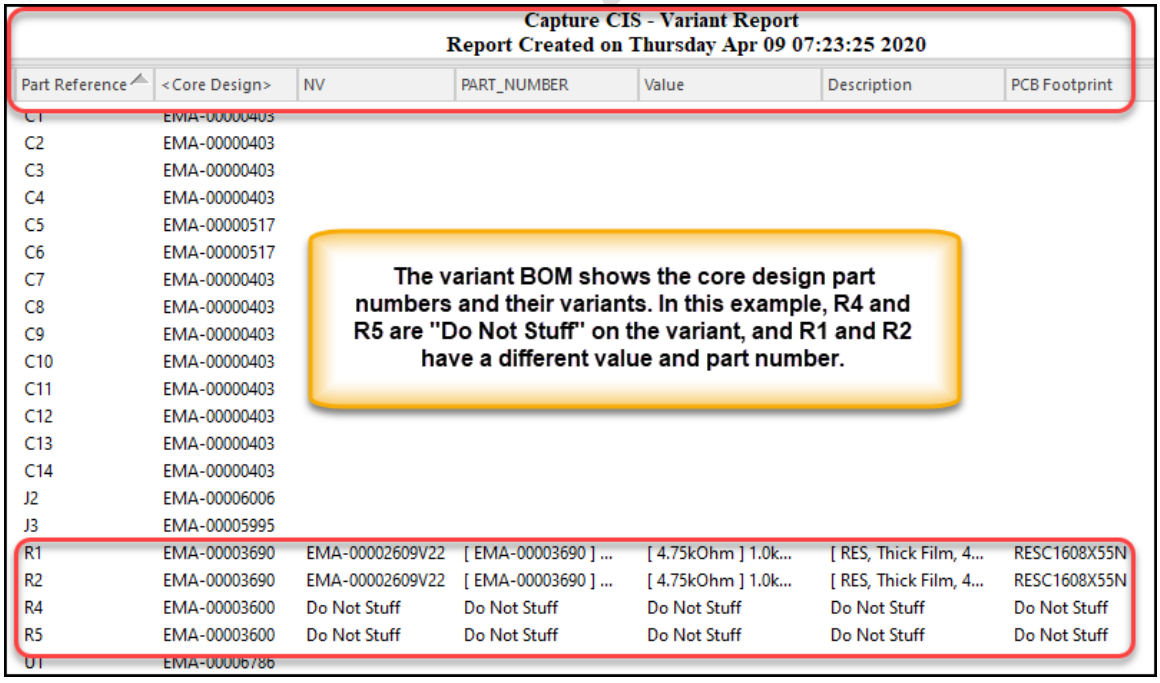
Variant View Mode

OrCAD Capture CIS allows you to view variant information for all design variants defined in your project. The variant information includes different property values for common components or different or not present components for identical footprints on a schematic page. You can use the ***Variant View Mode*** command to display the variant information on a schematic page.



Variant BOM Reports

The following image shows a sample of a variant BOM report.



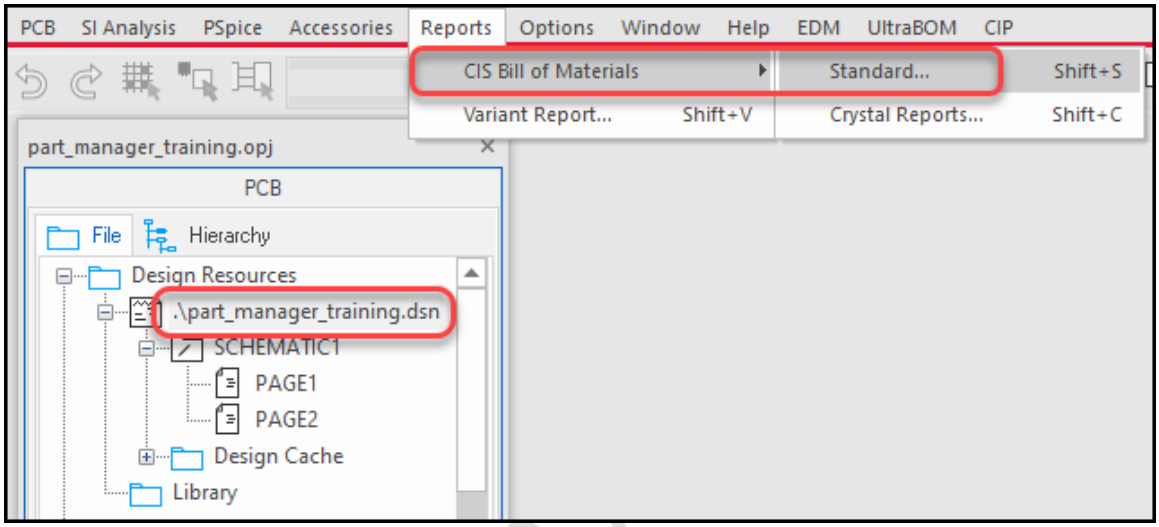
Capture CIS - Variant Report
Report Created on Thursday Apr 09 07:23:25 2020

Part Reference	<Core Design>	NV	PART_NUMBER	Value	Description	PCB Footprint
C2	EMA-00000403					
C3	EMA-00000403					
C4	EMA-00000403					
C5	EMA-00000517					
C6	EMA-00000517					
C7	EMA-00000403					
C8	EMA-00000403					
C9	EMA-00000403					
C10	EMA-00000403					
C11	EMA-00000403					
C12	EMA-00000403					
C13	EMA-00000403					
C14	EMA-00000403					
J2	EMA-00006006					
J3	EMA-00005995					
R1	EMA-00003690	EMA-00002609V22	[EMA-00003690] ...	[4.75kOhm] 1.0k...	[RES, Thick Film, 4...	RESC1608X55N
R2	EMA-00003690	EMA-00002609V22	[EMA-00003690] ...	[4.75kOhm] 1.0k...	[RES, Thick Film, 4...	RESC1608X55N
R4	EMA-00003600	Do Not Stuff	Do Not Stuff	Do Not Stuff	Do Not Stuff	Do Not Stuff
R5	EMA-00003600	Do Not Stuff	Do Not Stuff	Do Not Stuff	Do Not Stuff	Do Not Stuff

The variant BOM shows the core design part numbers and their variants. In this example, R4 and R5 are "Do Not Stuff" on the variant, and R1 and R2 have a different value and part number.

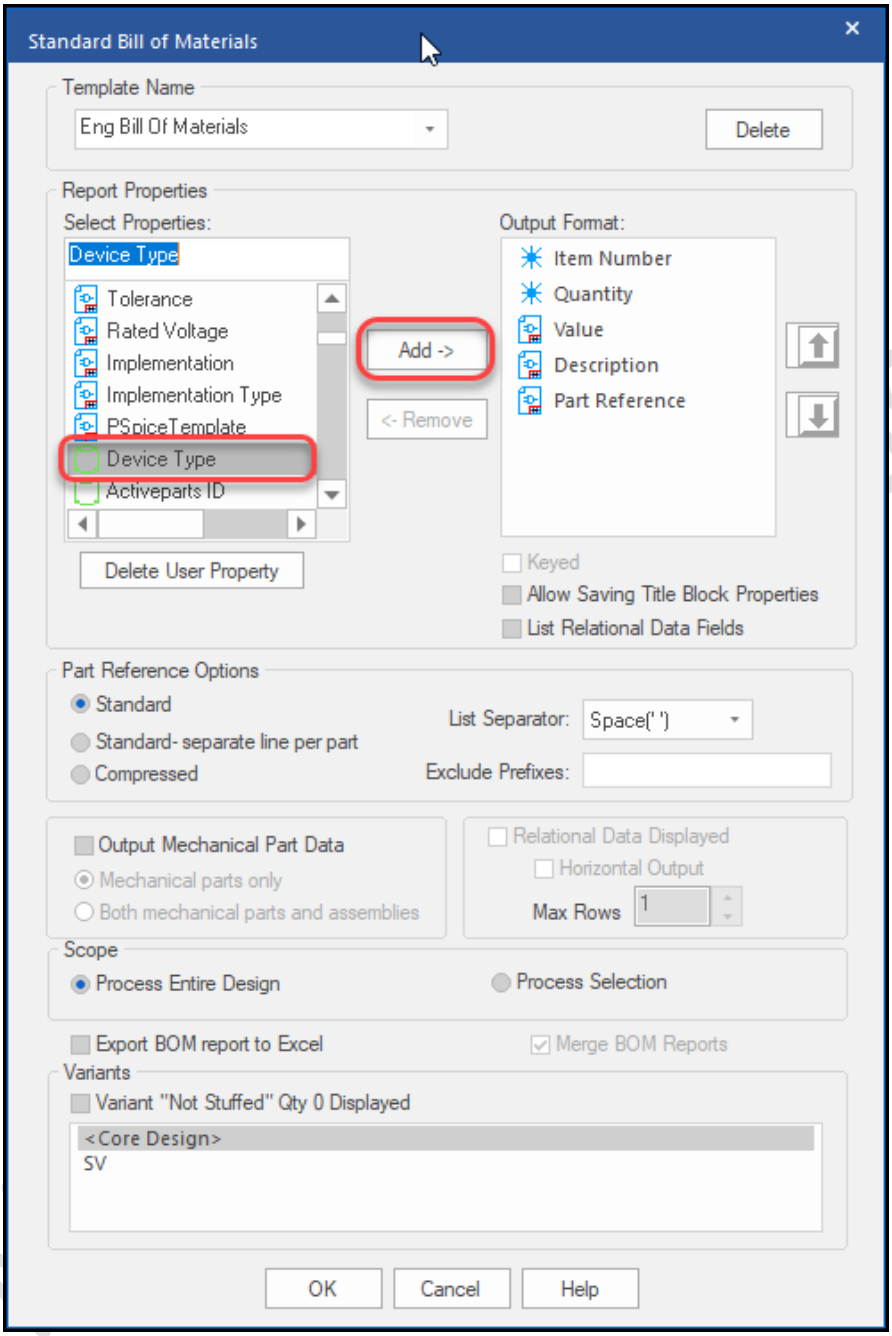
Lab 4-1: Creating a Standard CIS BOM

1. In the Project Manager, select `part_manager_training.dsn`, then select **Reports > CIS Bill of Materials > Standard**.



BOM Templates

BOM templates can be created and used for future reports. The desired properties can be added and arranged in the template, then the template is named.



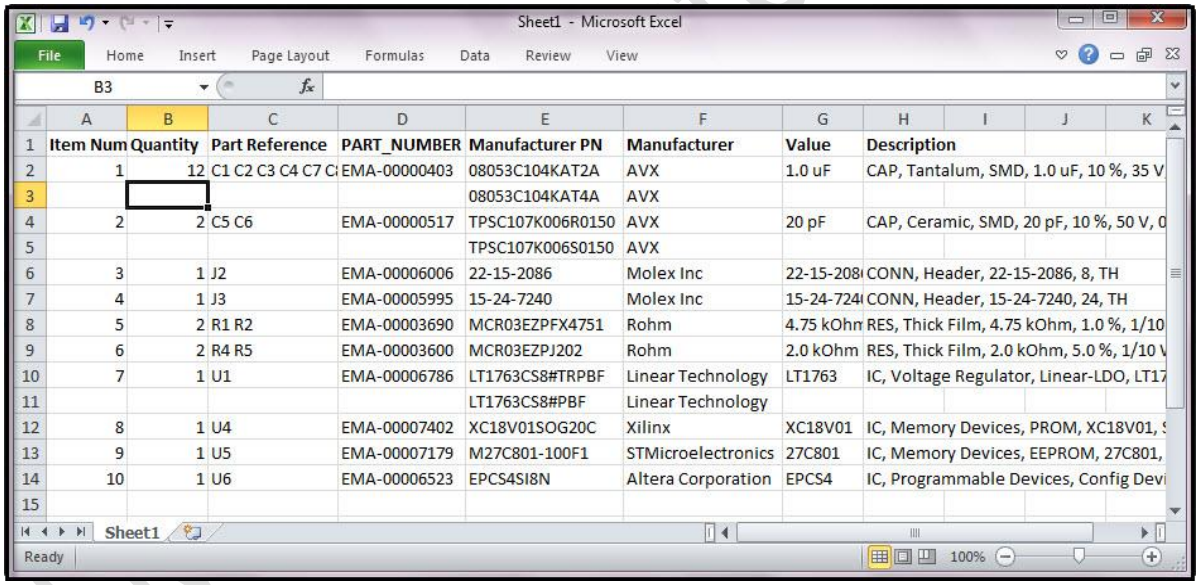
2. Scroll through the ***Select Properties*** list and select ***Device Type***, then click ***Add*** to add it to the ***Output Format***.
3. In the ***Template Name*** area enter the name **New Template**, then hit ***<Tab>***.

Lab 4-2: Creating the Report

1. In the ***Part Reference Options*** section, leave the default as ***Standard*** output.
2. Leave the list separator as ***Space (' ')***.
3. Check the ***Export BOM report to Excel*** option.
4. Click ***OK*** to generate the BOM.

Reviewing the Results

If there are multiple manufacturing part numbers associated with a single corporate part number, you can export this information to the BOM by selecting ***List Relational Data Fields*** in the BOM form. Additionally, select **Relational Data Displayed**, and set the max rows to 5. Typically, the Output Format would include the ***Manufacturer PN*** and ***Manufacturer*** fields.



Item Num	Quantity	Part Reference	PART_NUMBER	Manufacturer PN	Manufacturer	Value	Description
1	12	C1 C2 C3 C4 C7 C	EMA-00000403	08053C104KAT2A	AVX	1.0 uF	CAP, Tantalum, SMD, 1.0 uF, 10 %, 35 V
				08053C104KAT4A	AVX		
2	2	C5 C6	EMA-00000517	TPSC107K006R0150	AVX	20 pF	CAP, Ceramic, SMD, 20 pF, 10 %, 50 V, 0
				TPSC107K006S0150	AVX		
3	1	J2	EMA-00006006	22-15-2086	Molex Inc	22-15-208	CONN, Header, 22-15-2086, 8, TH
4	1	J3	EMA-00005995	15-24-7240	Molex Inc	15-24-724	CONN, Header, 15-24-7240, 24, TH
5	2	R1 R2	EMA-00003690	MCR03EZPFX4751	Rohm	4.75 kOhm	RES, Thick Film, 4.75 kOhm, 1.0 %, 1/10
6	2	R4 R5	EMA-00003600	MCR03EZPJ202	Rohm	2.0 kOhm	RES, Thick Film, 2.0 kOhm, 5.0 %, 1/10
7	1	U1	EMA-00006786	LT1763CS8#TRPBF	Linear Technology	LT1763	IC, Voltage Regulator, Linear-LDO, LT17
				LT1763CS8#PBF	Linear Technology		
8	1	U4	EMA-00007402	XC18V01SOG20C	Xilinx	XC18V01	IC, Memory Devices, PROM, XC18V01,
9	1	U5	EMA-00007179	M27C801-100F1	STMicroelectronics	27C801	IC, Memory Devices, EEPROM, 27C801,
10	1	U6	EMA-00006523	EPCS4SI8N	Altera Corporation	EPCS4	IC, Programmable Devices, Config Devi

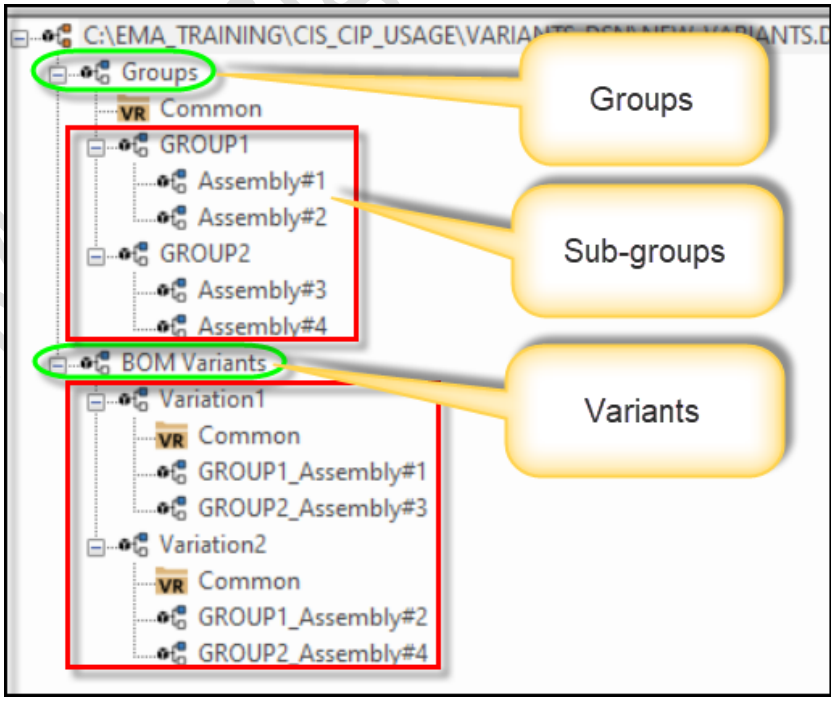
Lab 4-3: (OPTIONAL) Steps to Generate a Variant Design

In the earlier releases of OrCAD Capture CIS, the recommended method for creating and managing variants was slightly different in that you would create each variant in and of itself, then create the next one, and so on. Today – the recommended method shown below is better able to handle any ambiguity that may arise when you have a more complex set of variants. At the end of this section is a video link that will demonstrate the preferred way to generate variants in CIS.

Overview of Variant Generation

There are several steps to follow to generate variants in a design. Variants are created and managed in CIS Part Manager. The basic steps to create a variant are:

- **Create the group(s)**
  - Create the sub-groups (if any)
  - Add selected parts to the top level group folder(s)
  - Modify the parts of the sub-group(s)
- **Create the BOM Variant folder(s)**
  - Add the modified group(s) to the BOM Variant folder(s)



The following exercise will demonstrate how to create two variants using two different groups and sub-groups resulting in two BOM variants.

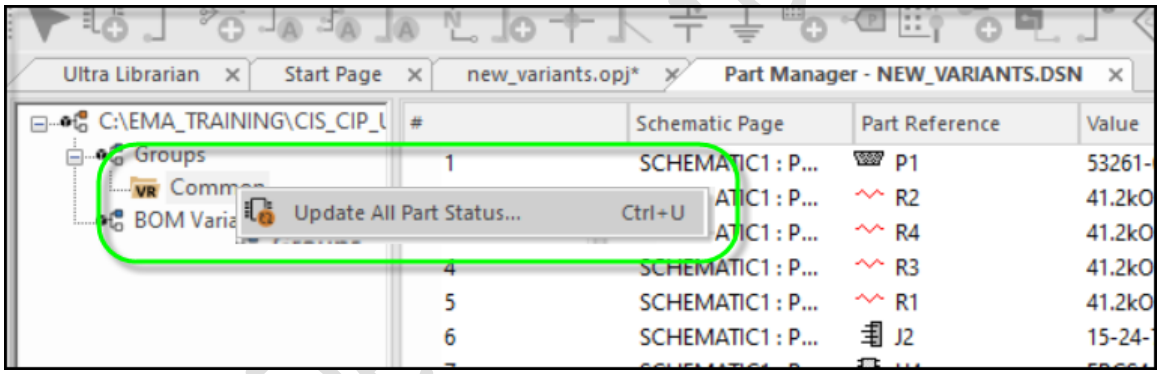
Some designs may require multiple variants, and these can be further refined by including sub-groups. Sub-groups represent a version or assembly of the parent group. Regardless of how many variants need to be created for any design, the basic steps will be the same.

Create the Groups

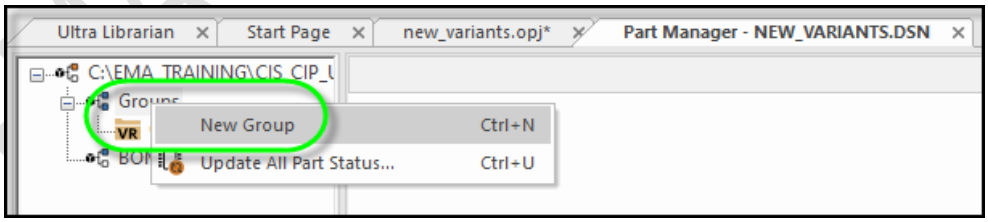
1. Open `C:\EMA_Training\CIS_CIP_Usage_23.1\new_variants.dsn`.
2. In the Project Manager, right click on the design and select ***Part Manager***.

When Part Manager opens all parts in the design appear in the list on the right. On the left are folders identifying any groups that may exist, common parts, and BOM Variants.

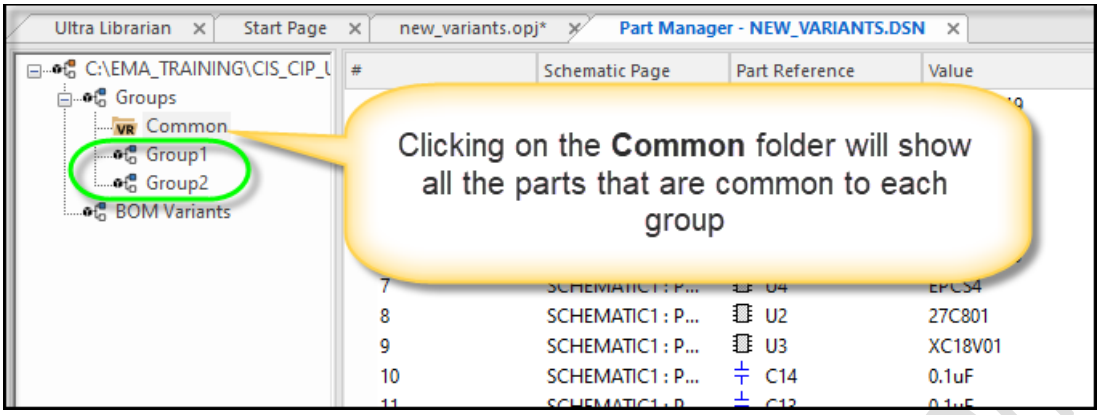
3. In the left window pane, right-click on all the Common folder and choose to ***Update All Part Status***



4. Right click on the folder named ***Groups*** and select ***New Group***.
5. Name the new group ***Group1***.



The new folder is created but does not yet contain any parts. Clicking on the ***Common*** folder will show the parts that are common to both the core design and any variants that may be created.

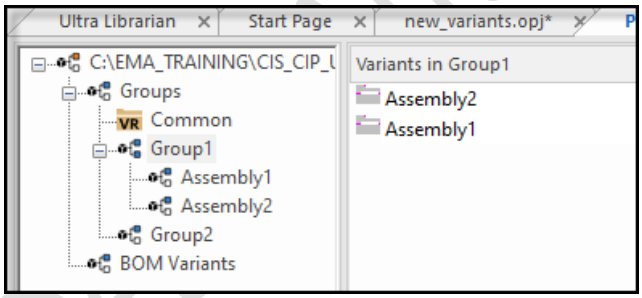


6. Create another group and name it Group2.

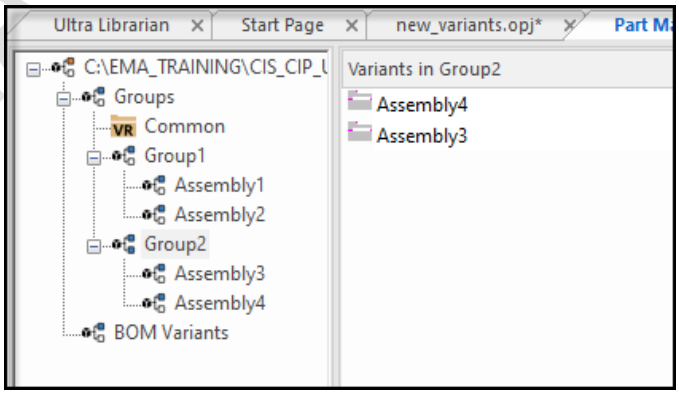
Create the Sub-groups

Next you will create sub-groups for each parent group.

1. Right click on Group1 and create a sub-group named Assembly1.
2. Do the same and create another sub-group named Assembly2.



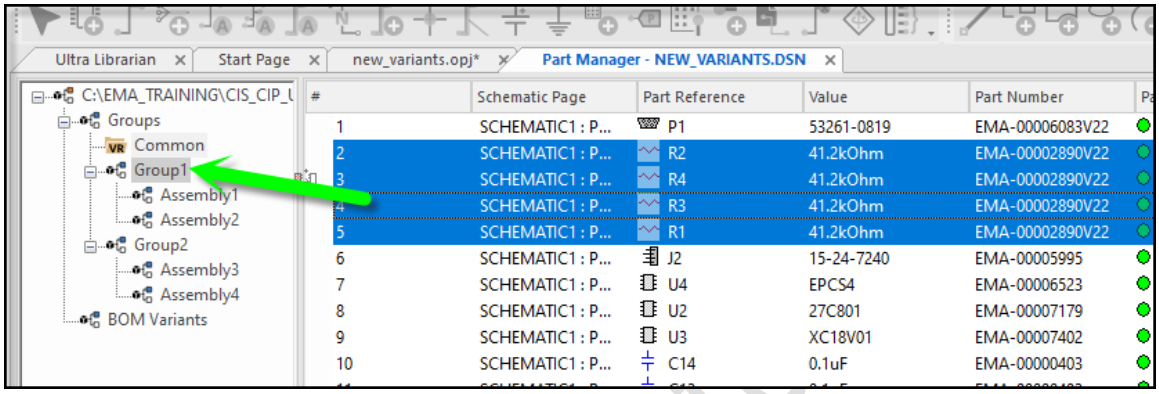
3. Right click on Group2 and create 2 more sub-groups named Assembly3 and Assembly4.



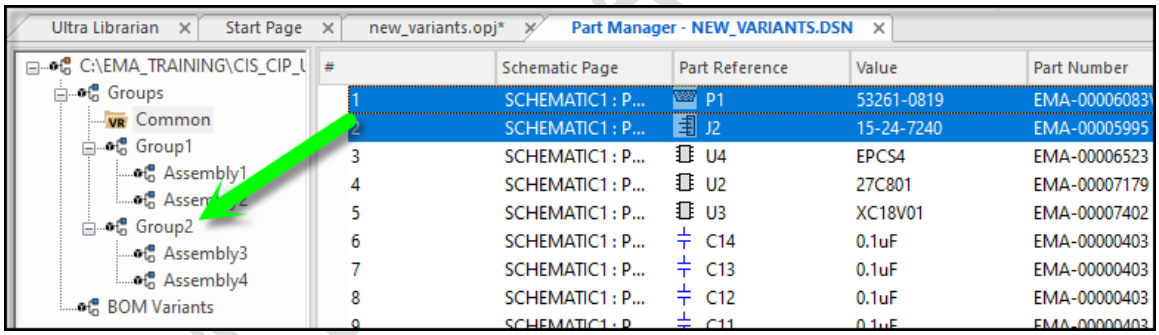
Adding Parts to the Groups

Now that you have the groups and sub-groups created you can begin adding the parts which will make up the potential combination for Assemblies 1 through 4.

1. Click on the Common folder under Groups.
2. In the list of parts, select R1, R2, R3, and R4 from the list and drag them to the Group1 folder.



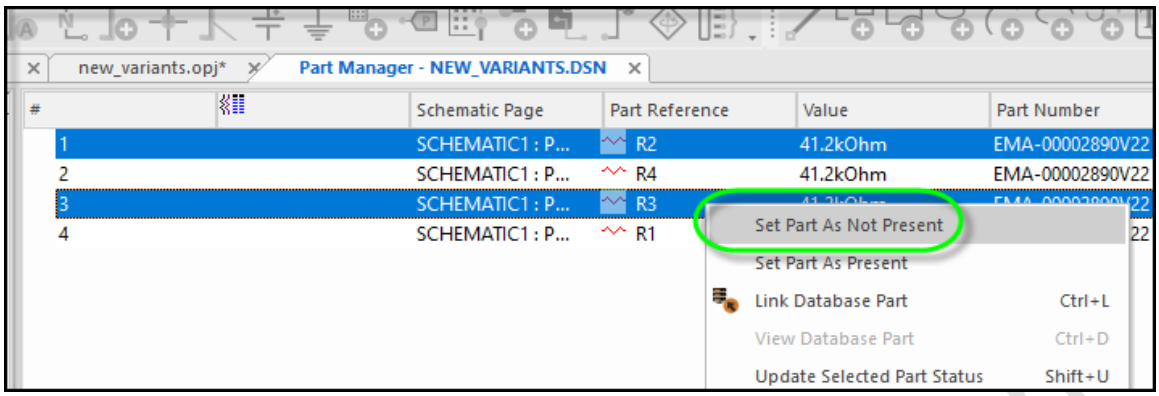
3. Next, from the parts list select components J1 and P1, and drag them to the Group2 folder.



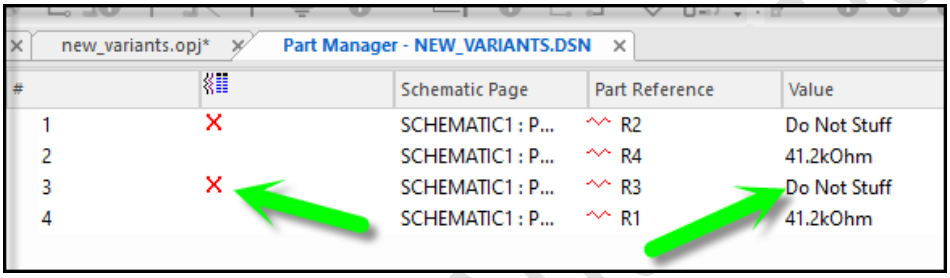
Modifying the Parts in Group1

The parts in the Assembly1 folder under Group1 will remain as is (42.2kOhm). You will only be changing the parts in Assembly2.

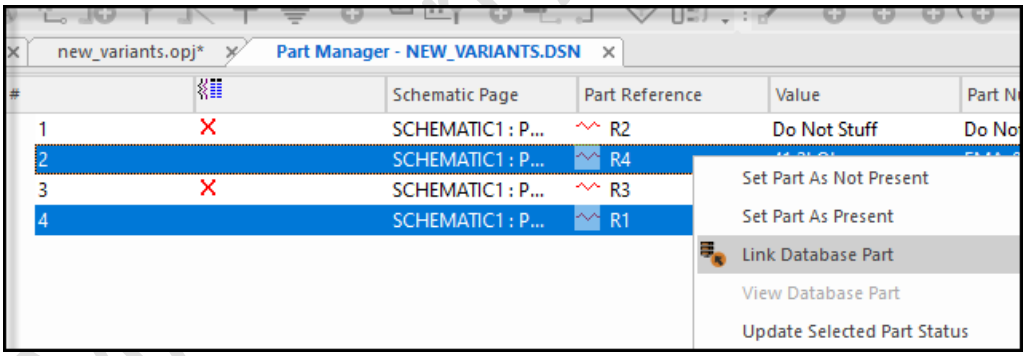
1. Select the Assembly2 folder under Group1.
2. Select R2 and R3 from the list, right click and choose **Set Part As Not Present**



Notice they now show an 'X' and their Value has changed to 'Do Not Stuff'

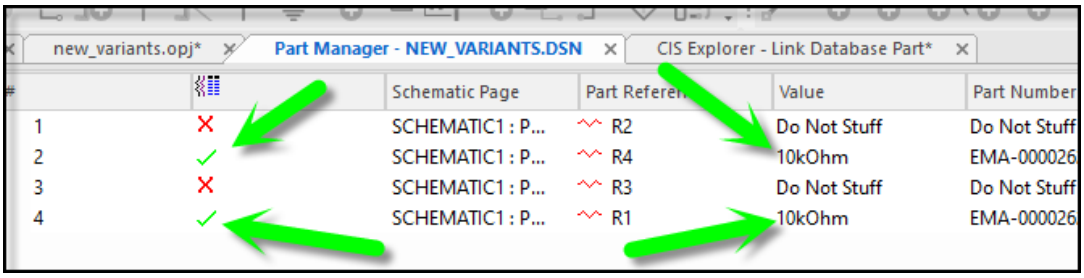


3. Select R4 and R1 from the same list, then right click and choose **Link Database Part**



This will launch CIS Explorer where you will search for a 10K resistor to replace R1 and R4. In CIS Explorer you can either use the Query tab to search for a 10K resistor or you can expand the database tree.

4. Locate a 10K resistor in CIS Explorer and select it from the parts list. It should turn green.
5. Once selected you can review the attributes. Double click on it to complete the replacement of the parts.

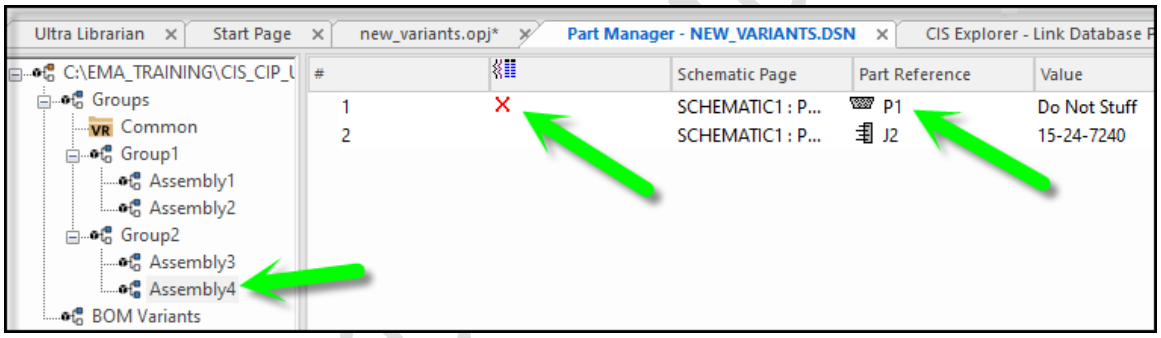


Notice the Values for R1 and R4 show 10kOhm and also show a check mark indicating a differing value from the original core set of parts.

Modifying the Parts in Group2

The parts showing in the Group2 Assembly 3 folder will remain as is. You will only make a change to the parts in the Assembly 4 folder.

1. Select the Assembly4 folder
2. Right click on the P1 part and choose **Set Part As Not Present**



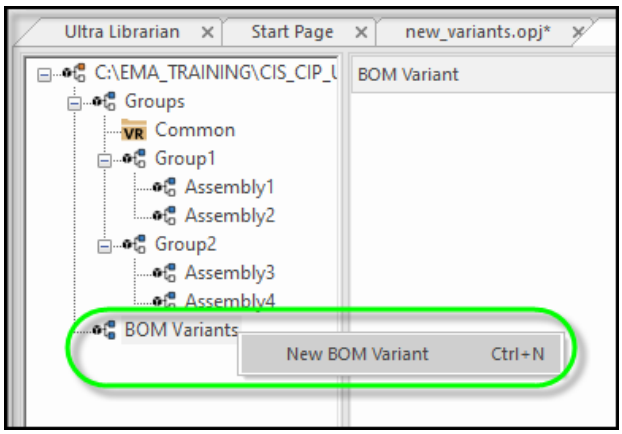
This completes creating the Groups, and sub-groups and modifying the parts within the sub-groups. Next, you will create the BOM Variants.

Create the BOM Variants

Now you can create a few variants from the sub-groups. In this step you will create two BOM Variants.

Create BOM Variants

1. Right click on the BOM Variants folder to create a new BOM variant. Name the BOM variant Variant1.

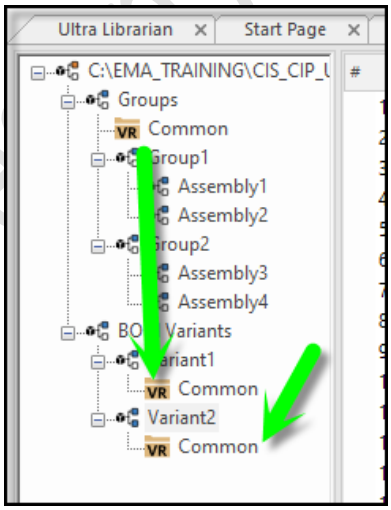


2. Right click on the BOM Variants folder again and create a new BOM variant named Variant2.

Assemble the Variants

Next you will need to add the Common folder to the BOM Variants folder so that each variant has the Common set of components represented.

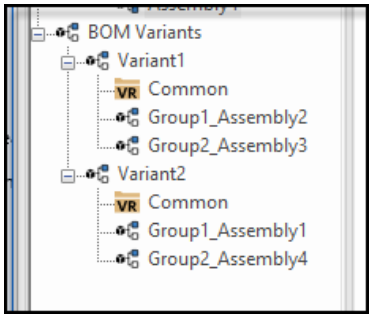
1. Select and drag the Common folder to each of the Variant1 and Variant2 folders.



Create Variant1 and Variant2

Now that the common set of parts are representing both variants you can select from the sub-groups to complete the variant.

1. Click and drag the Assembly2 and Assembly3 folders into the Variant1 folder.
2. Click and drag the Assembly1 and Assembly4 folders into the Variant2 folder.
3. Now examine each of the Variants.

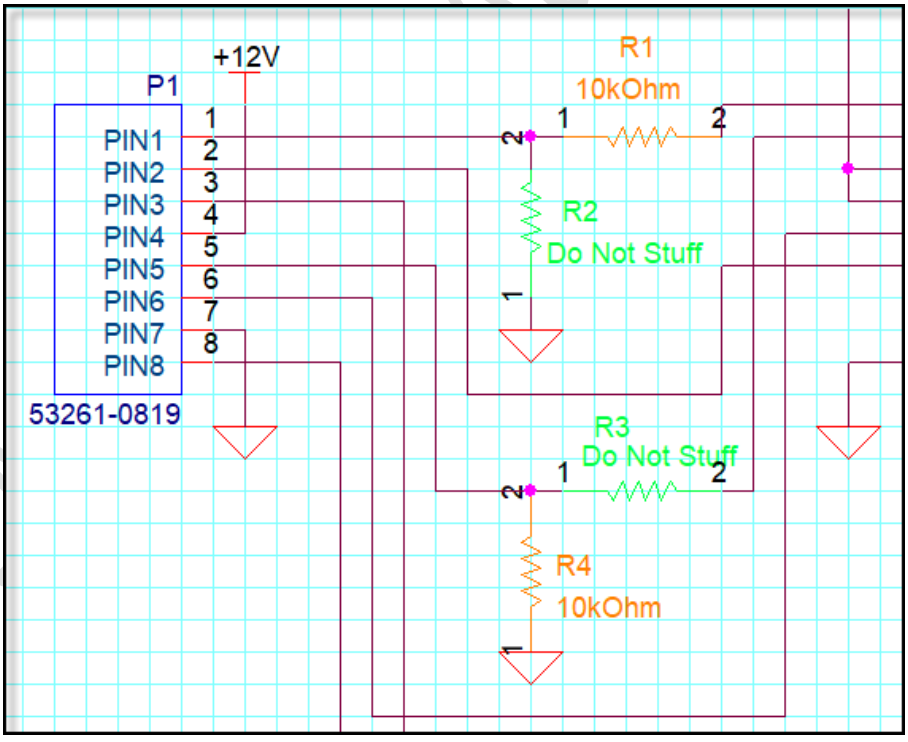


Viewing the Variants in the Schematic

Now that you have created the BOM Variants they can be viewed in the schematic.

1. Open the schematic page.
2. Select ***View > Variant View Mode***.
3. Select ***Variant1*** from the ***Select a Design Variant*** window.
4. Click ***OK*** to save the design.

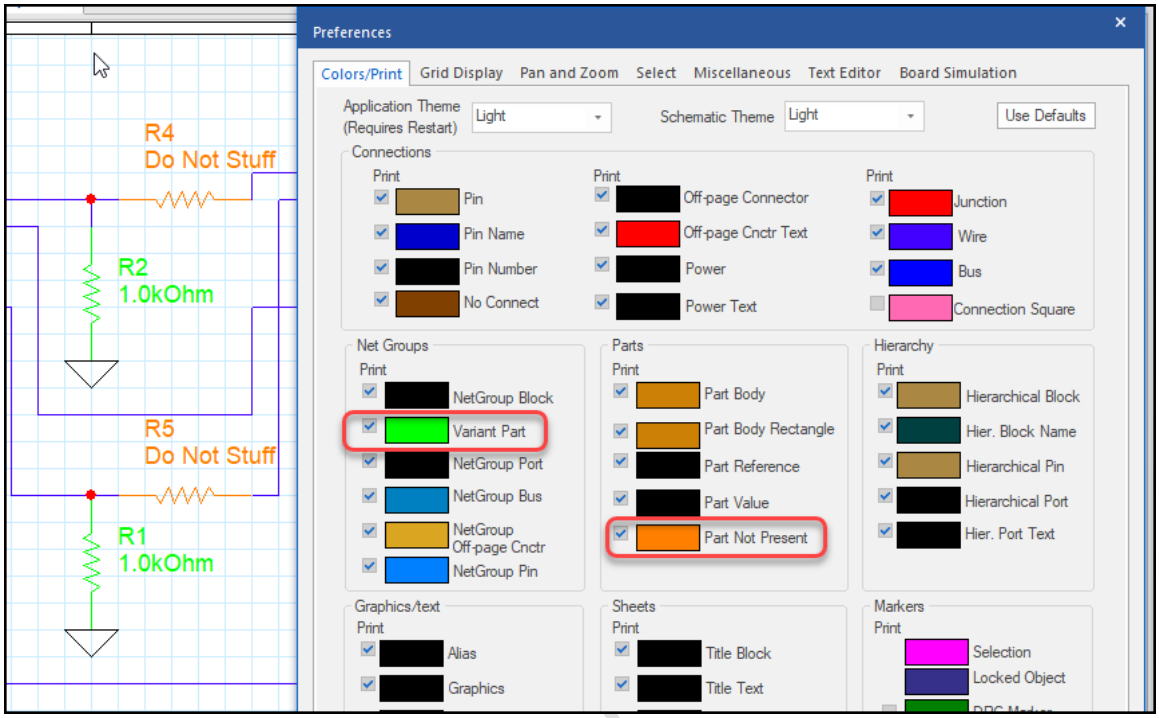
R2 and R3 are identified as Do Not Stuff components, while R4 and R1 are 10KOhm parts. Also, P1 is present in Variant1.



The colors for the variant parts are different than the other parts. You can choose how these parts will appear in the schematic.

5. Choose ***Options > Preferences***.

6. In the ***Colors/Print*** tab, change the ***Variant Part*** color to ***bright green***.
7. Change the ***Part Not Present*** color to ***orange***.
8. Click ***OK***.

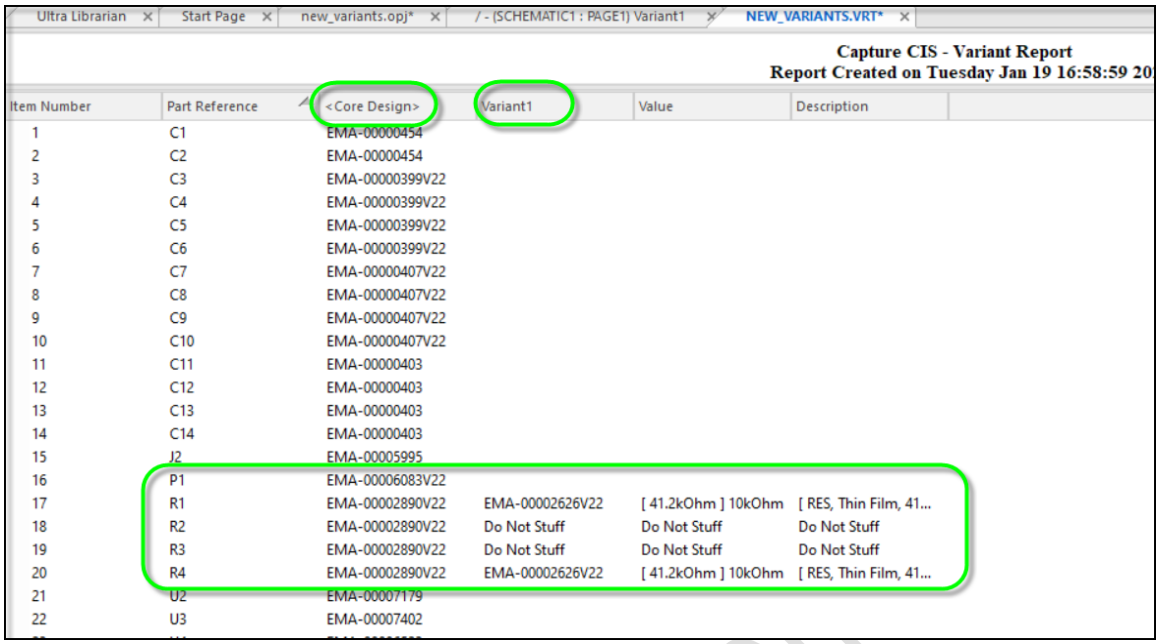


9. Select ***View > View Variant Mode*** and select the Core design to go back to the core schematic view.

Creating a Variant Bill of Materials

1. Select the Project Manager.
2. Select ***Reports > Variant Report***.
3. Remove the ***Description*** property from the ***Output Format*** section.
4. Select ***Variant1*** in the ***Variants*** area and click ***OK*** to run the report.

Optionally you could choose ***Reports > CIS Bill of Materials > Standard*** and select the settings along with the preferred variant, and export to an Excel file.



Item Number	Part Reference	<Core Design>	Variant1	Value	Description
1	C1	EMA-00000454			
2	C2	EMA-00000454			
3	C3	EMA-00000399V22			
4	C4	EMA-00000399V22			
5	C5	EMA-00000399V22			
6	C6	EMA-00000399V22			
7	C7	EMA-00000407V22			
8	C8	EMA-00000407V22			
9	C9	EMA-00000407V22			
10	C10	EMA-00000407V22			
11	C11	EMA-00000403			
12	C12	EMA-00000403			
13	C13	EMA-00000403			
14	C14	EMA-00000403			
15	J2	EMA-00005995			
16	P1	EMA-00006083V22			
17	R1	EMA-00002890V22	EMA-00002626V22	[41.2kOhm] 10kOhm	[RES, Thin Film, 41...
18	R2	EMA-00002890V22	Do Not Stuff	Do Not Stuff	Do Not Stuff
19	R3	EMA-00002890V22	Do Not Stuff	Do Not Stuff	Do Not Stuff
20	R4	EMA-00002890V22	EMA-00002626V22	[41.2kOhm] 10kOhm	[RES, Thin Film, 41...
21	U2	EMA-00007179			
22	U3	EMA-00007402			

This completes OrCAD CIS/CIP Usage Training.

Optional Videos on Variant Generation:

Schematic Capture Design Variants - Try OrCAD (vidyard.com)

https://resources.ema-eda.com/all-videos-2/variant-management