cadence®

Unified Search User Guide

Product Version 23.1
September 2023

© 2023 Cadence Design Systems, Inc. All rights reserved.
Portions © Apache Software Foundation, Sun Microsystems, Free Software Foundation, Inc., Regents of the University of California, Massachusetts Institute of Technology, University of Florida. Used by permission. Printed in the United States of America.

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

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

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

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

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

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

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

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

Contents

1
Part Search 5
Accessing Unified Search 7
Accessing Unified Search From System Capture 7
Accessing Unified Search From Pulse Web Dashboard 7
Logging in to Unified Search Content Providers 7
Working With Unified Search 9
Searching for Parts 9
Performing Advanced Searches 11
Viewing Graphics in Part Details Pane 20
Adding Parts to BOM 23
Placing Parts 24
Working With Parts From Content Providers 26
Viewing Parts from Another Content Provider 26
Replacing External Vendor-Provided Parts in Designs 27
Part Manager Notifications for System Capture Users 28
Design Search for Reuse 30
Searching for Designs for Reuse 33

2
Configuring Unified Search 41
Project-Specific Configuration 41
Configuration for all Projects on a Machine 41
Locking Chiplets 42
Setting Filters as Project Default 44
Configuring Search Results Display 45
Using Custom Configuration File 46
Configuring Part Content Providers 48

? 2023 All Rights Reserved.

1 Appendix

Appendix 53
How Unified Search Uses Queries 53
Advanced Search Query Syntax 56
OrCAD Capture Libraries in System Capture 59
Adding Project-Specific Libraries to a Project 60
Troubleshooting 60

? 2023 All Rights Reserved.

1

Part Search

Using *Unified Search*, you can submit a free-text search for parts, based on categories, to find the components you need. Parametric search enables you to narrow down search results. While using Cadence schematic design tools, you can search for and add parts, symbols, footprints, 3D models, and other attachments, such as simulation models, from:

- Cadence-supplied or custom libraries

  These libraries are automatically installed during the Cadence installation setup. Custom libraries are libraries specific to your organization or created for your own use.

- Electronic component distributors or manufacturers, referred to as content providers, part providers, or external vendors

  Cadence provides parts from SamacSys and Ultra Librarian.

You can search for and place a manufacturer part number based on electrical parameters as well as supply chain information, such as cost and availability.

Parts from *Unified Search* are available for designers:

- In a single-user environment where all design and library data is managed on the same machine and designers are not connected to a central Pulse server
- In a multi-user environment, where designers connect to a central Pulse server using a remote URL

Display of Library Sources

Libraries supported in *Unified Search* are displayed in the following panes in the user interface:

- *Library* - Shows parts from one of the following sources:
  - Libraries configured in `Capture.ini` if you are working with a System Capture project with Capture libraries

    It is recommended that you run OrCAD Capture, configure the libraries and ODBC connection to the CIS database, and use the resulting `Capture.ini` file.

? 2023 All Rights Reserved.

- Libraries configured at a common location and referenced by the `CDS_SITE` environment variable

  This variable points to a location, which can be one of the following:

  - Cadence-supplied or custom libraries - These can be Design Entry HDL or OrCAD Capture libraries
  - Pulse-managed libraries, which are also referred to as remote libraries.

    To view Pulse-managed libraries, you must be connected to a central Pulse server using a remote URL.
  - Allegro Unified Libraries

    This is the native library format of Allegro System Capture. Parts from Allegro Unified Libraries are only available in the single-user environment.

- *My Parts* - Shows parts from project-specific libraries. This pane also displays parts placed from external part providers. This is because once placed in a design, an external part is cached in your local library and the library is automatically added to your design project.

  Note: The *My Parts* pane is not available if you are working with a Allegro Unified Library project.
- *SamacSys* and or *Ultra Librarian* if signed in to these external content providers.

The target audience for this user guide is:

- Designers in a single-user environment where design and library data is on the same machine
- Design project owners and designers in a multi-user System Capture environment who connect to a central Pulse server using a remote URL

  If you are a Pulse administrator of a multi-user System Capture environment, refer to *Allegro Pulse Configuration Guide*.

Related Topic

Allegro System Capture Pulse Guide

OrCAD Capture Libraries in System Capture

? 2023 All Rights Reserved.

Accessing Unified Search

In a multi-user environment, *Unified Search* can be accessed from System Capture and the Pulse web dashboard. In the single-user environment where design and library data is managed on the same machine, it can only be accessed from System Capture.

Accessing Unified Search From System Capture

To access *Unified Search* from System Capture, do the following:

1. Launch Allegro System Capture.
2. Open a project or create a new project.
3. Click the part icon to open the *Unified Search* pane.

Accessing Unified Search From Pulse Web Dashboard

To access *Unified Search* from the Pulse web dashboard, do the following:

1. Type `http://<Pulse access URL>:7100/projects` in a web browser.
2. Do one of the following:
   - Click *Search Part* for a global search of all parts in the Pulse primary node you are connected to.
   - To search for parts in a specific project, select the required project and click the *SEARCH* tab.

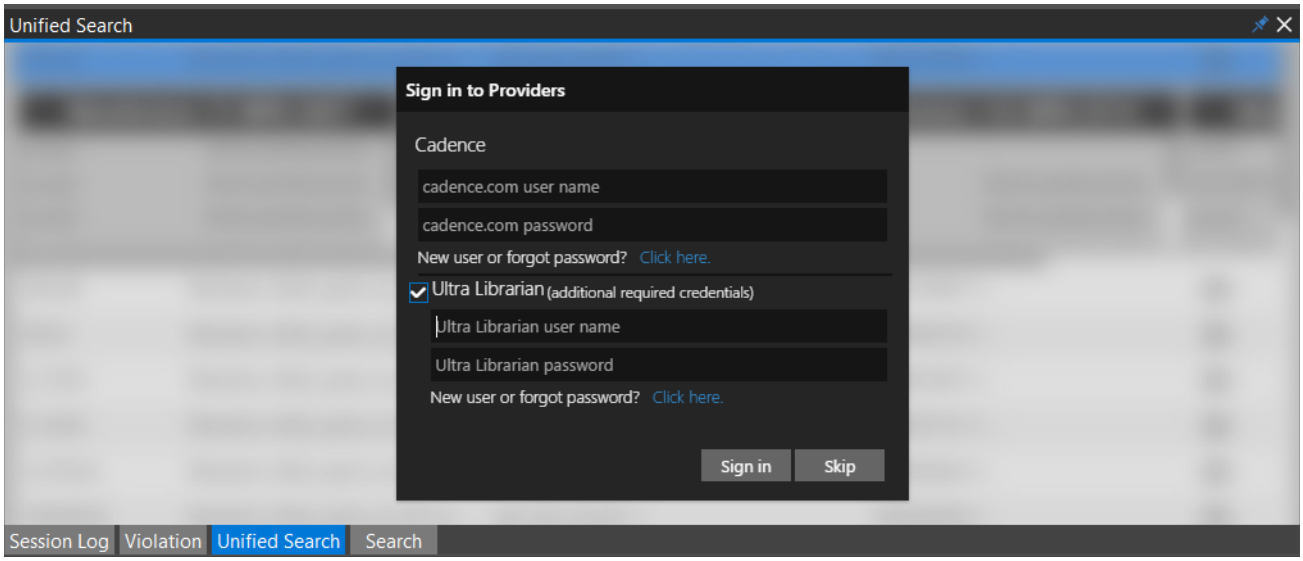
Logging in to Unified Search Content Providers

When you open or create a new project in System Capture or the Pulse web dashboard and navigate to the part search pane, the *Sign in to Providers* dialog box is displayed:

- for a first-time launch

? 2023 All Rights Reserved.

- if you are signed out of the *Unified Search* content providers.



To log in to *Unified Search*:

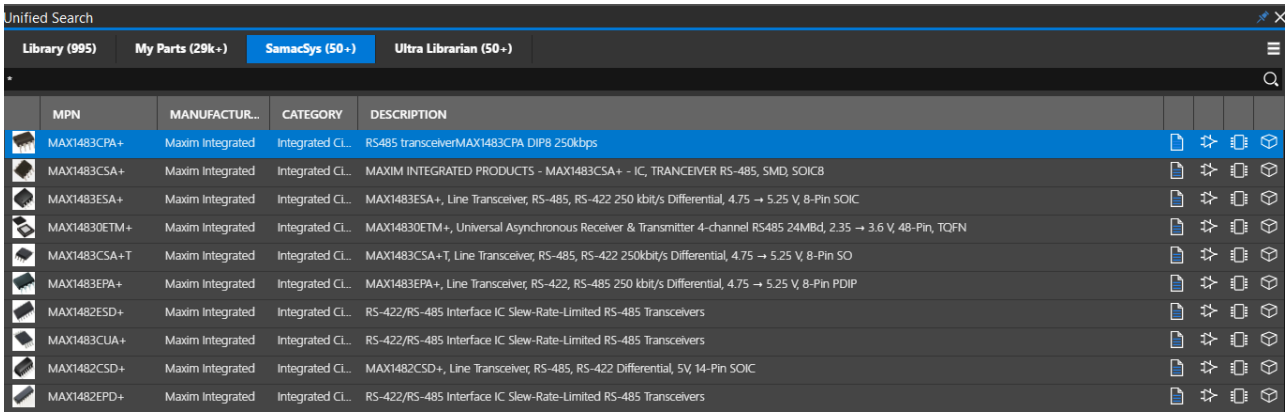
1. Provide your user credentials.

   You need an account on Cadence Support for both part providers. For Ultra Librarian, you must also have an Ultra Librarian account.

   If you do not want parts from SamacSys or Ultra Librarian, click *Skip* in the *Sign in to Providers* dialog box.

2. Click the *Sign in* button.

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



Note: *My Parts* is only displayed in System Capture. It is not available when *Unified*

All Rights Reserved.

Search is accessed from the Pulse web dashboard.

Related Topics

Working With Parts From Content Providers

Working With Unified Search

You can search for parts, perform an advanced search, place a part on the canvas, and add parts to the BOM. In the multi-user System Capture environment, you can also raise a new part request. For details on each task, see the following:

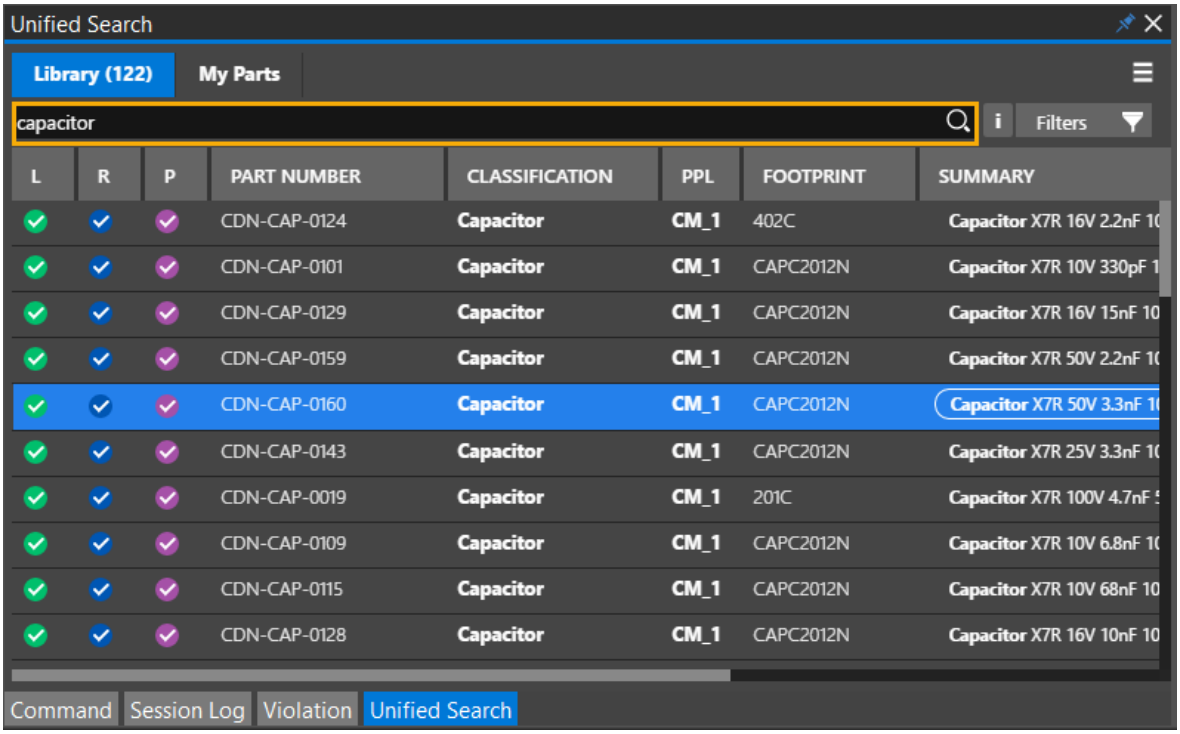
- Searching for Parts
- Performing Advanced Searches
- Viewing Graphics in Part Details Pane
- Adding Parts to BOM
- Placing Parts
- Working With Parts From Content Providers
  - Viewing Parts from Another Content Provider
  - Replacing External Vendor-Provided Parts in Designs

Searching for Parts

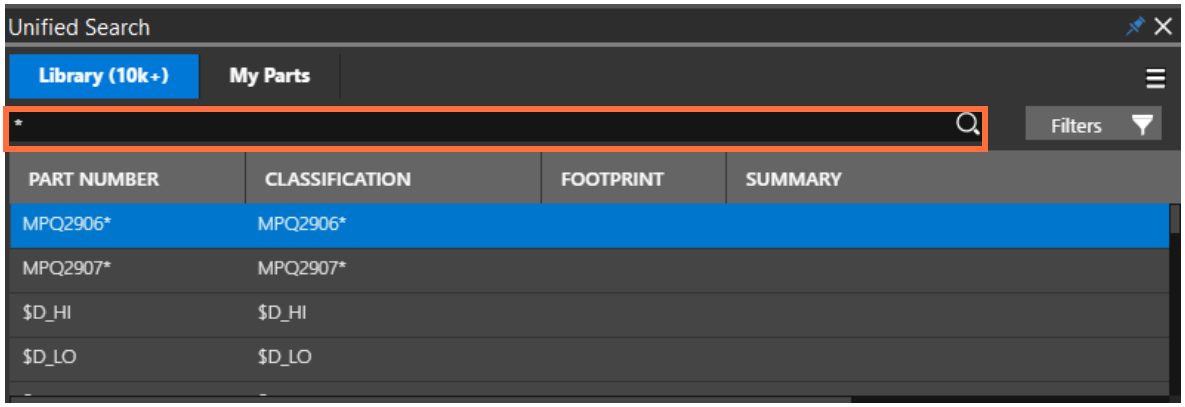
To search for parts in *Unified Search*, do the following:

? 2023 All Rights Reserved.

1. Perform a free-text search in the text box and click the search icon.



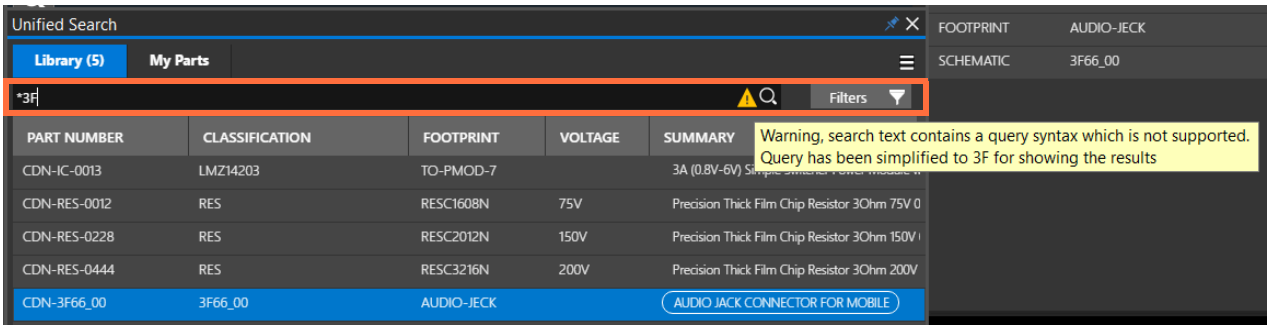
2. To view all parts in a project, use the asterisk symbol (*).



If wild card searches are enabled by the Pulse administrator, you can also use the asterisk symbol, that is a wild card, with a keyword. Wild card searches are only supported in the multi-user environment.

? 2023
All Rights Reserved.

A wild card or a keyword search in the single-user environment displays a warning icon as illustrated:

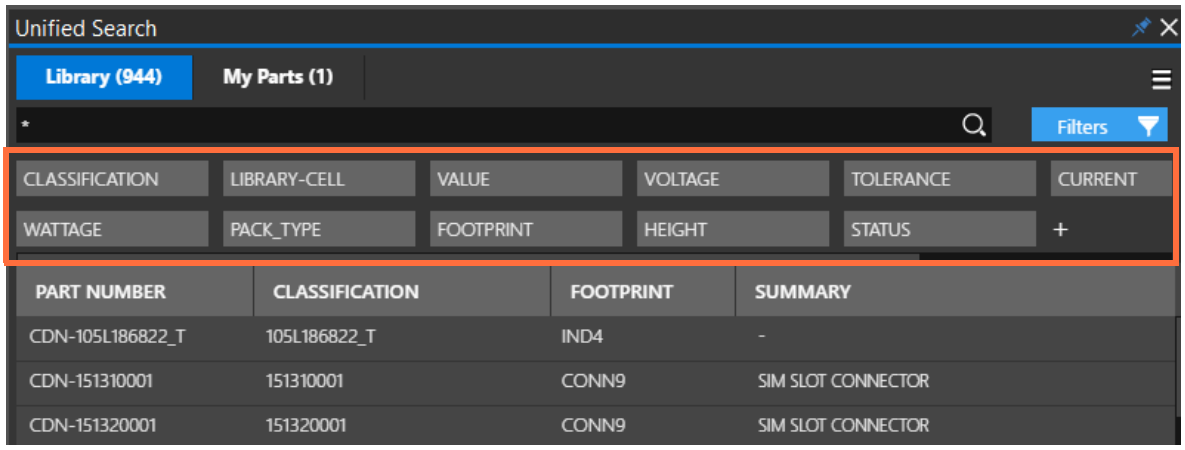


Performing Advanced Searches

In the multi-user environment, you can do an advanced search using a combination of filters and search queries, which can include Boolean operators. Advanced searches can only be done in the *Library* pane.

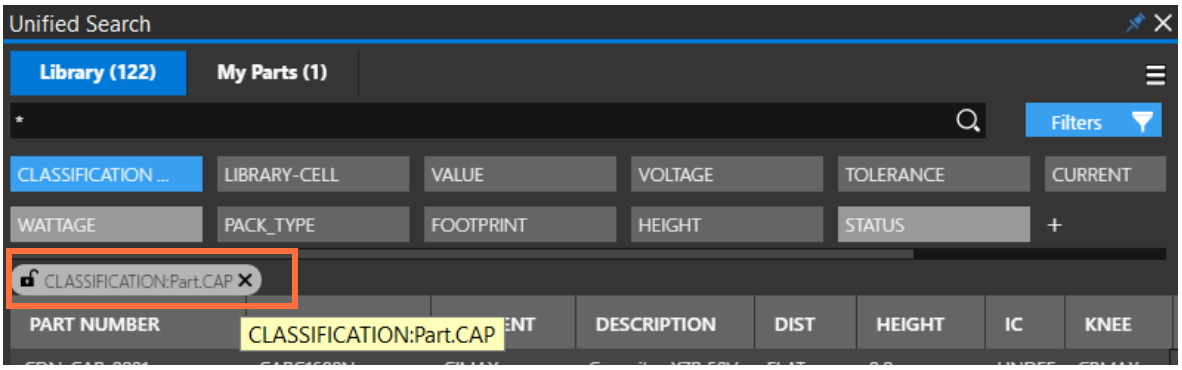
Note: In the single-user environment, an advanced search can only be done using filters or a combination of filters and free-text search.

A filter is a list of default part properties, such as *CLASSIFICATION*, *VALUE*, *VOLTAGE*, *WATTAGE*, *TYPE.*



? 2023
All Rights Reserved.

A chiplet is an applied filter. It is the combination of a filter and its value. For example, a combination of *CLASSIFICATION* and *CAP* is a chiplet.



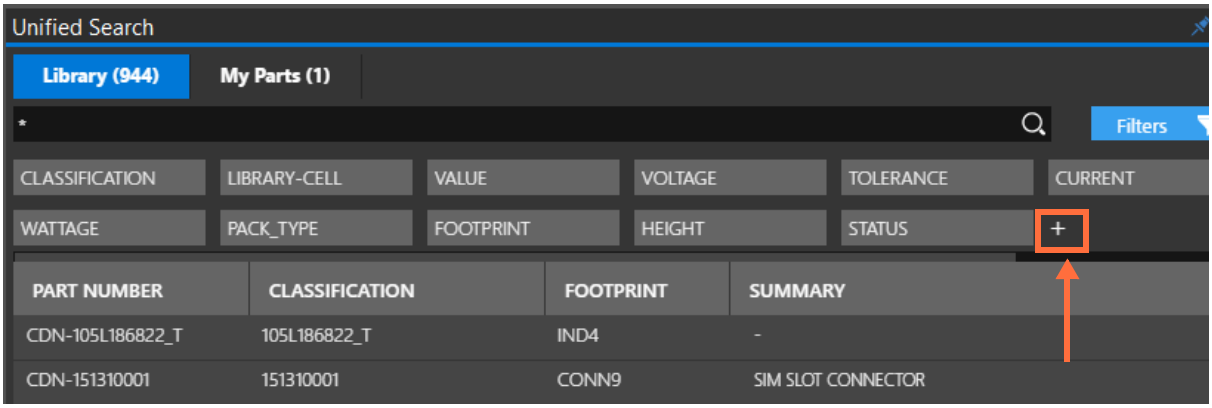
Using Filters for a Search

To perform an advanced search query, you can do the following:

1. Click the *Filters* button.

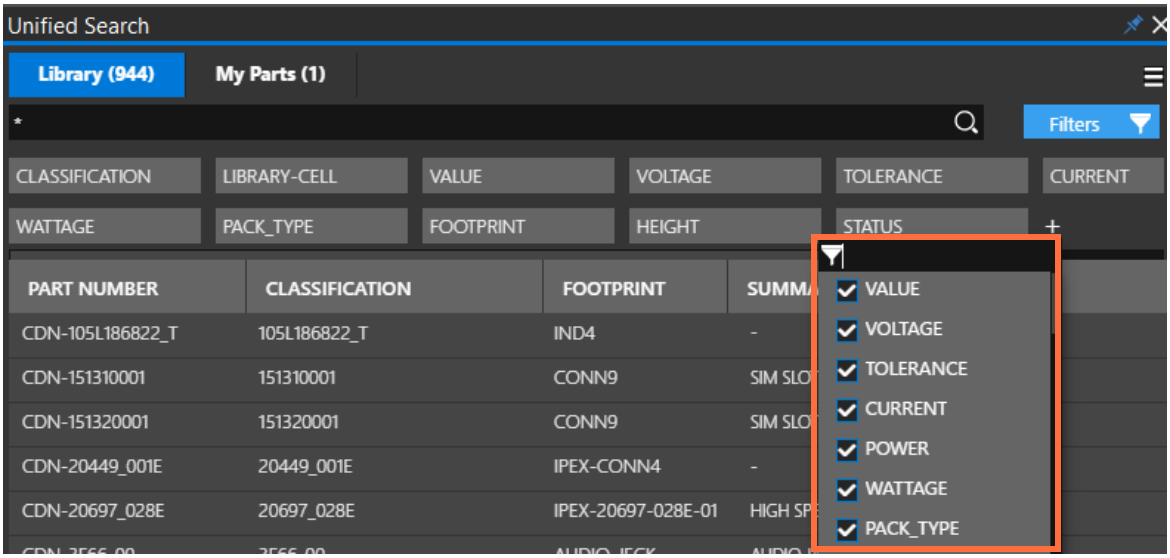
   A filter with default part properties, such as *CLASSIFICATION, VOLTAGE, TOLERANCE*, is displayed.

2. To add more filters, click the plus (+) icon next to the last default filter.



? 2023
All Rights Reserved.

3. Select the desired filter and click outside the drop-down list.



Important

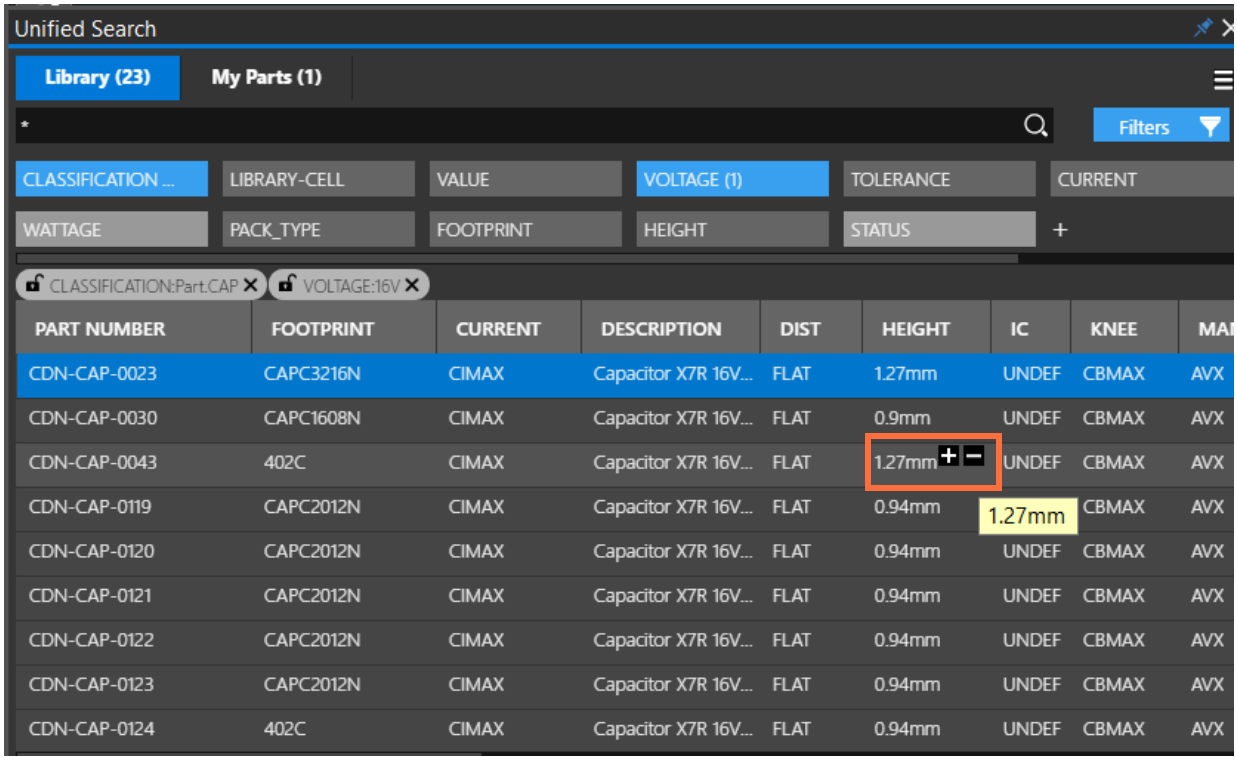
Only properties with letters, numbers, an underscore, or a space in the names are displayed in the filter list. Properties with names such as *BODY/SHELL_MATERIAL* are not displayed. To add such properties as a filter, search for them in the free-text search field.

4. To remove a filter, click the plus (+) icon again and deselect the required filter from the drop-down list.

? 2023
All Rights Reserved.

Using Filters in Search Result List

When you hover the mouse over any field in a part row, plus and minus icons are displayed.

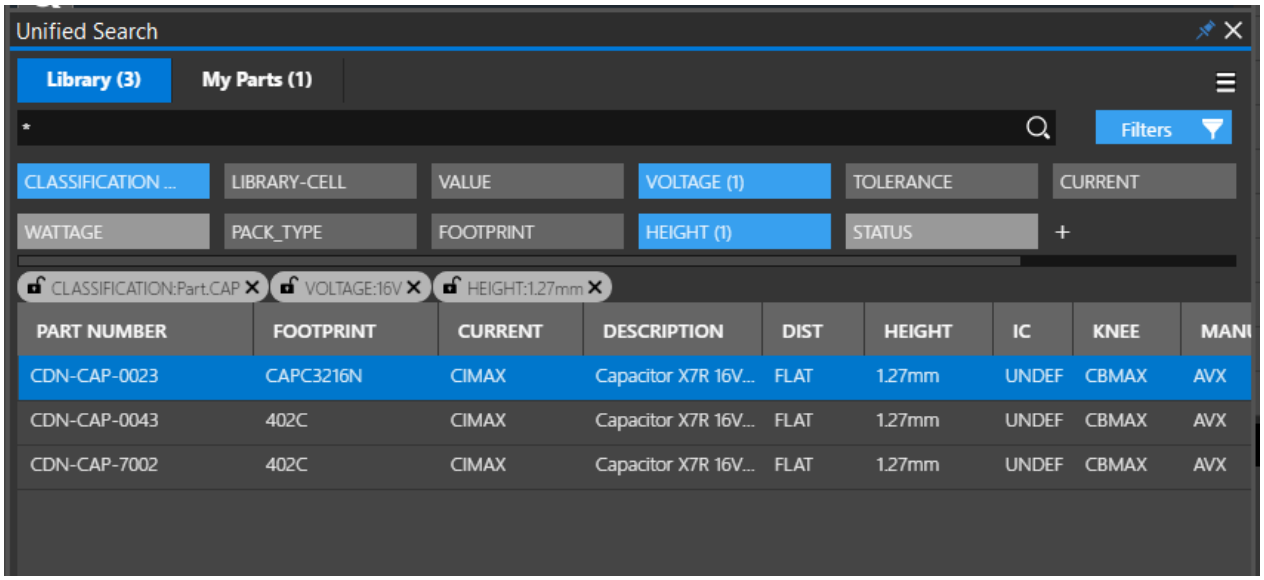


To use the filters in the search results list, do the following:

1. Click the plus (+) icon to add the selected value.

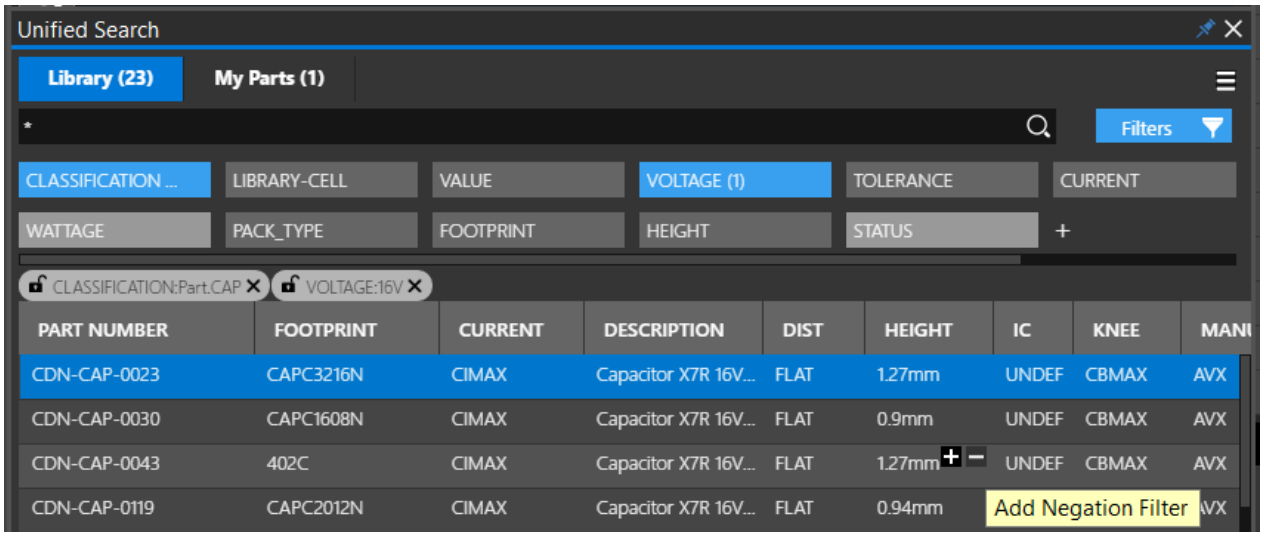
? 2023
All Rights Reserved.

For example, in the `HEIGHT` column, add `1.27 mm`. The chiplet is added and the search results are further modified to display capacitors whose height is `1.27 mm`.



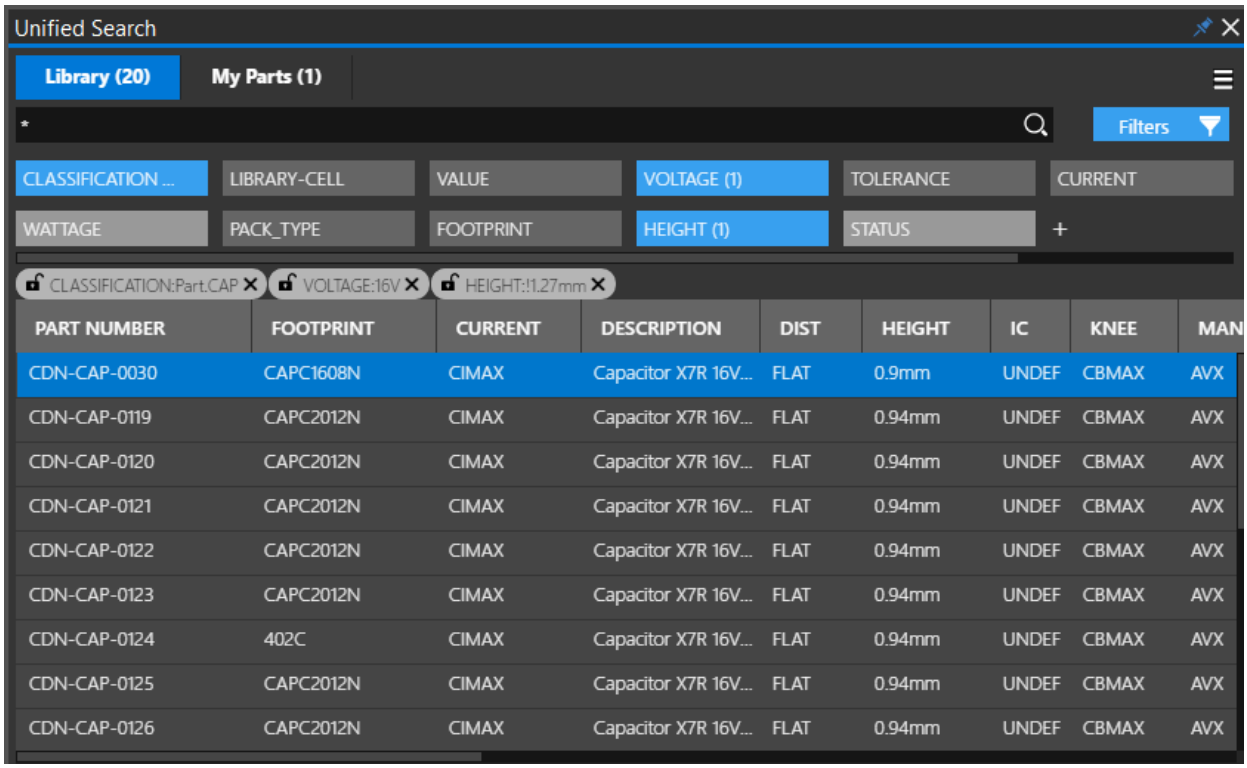
2. To exclude a selected value from a search, click the minus (-) icon.

This excludes all properties that do not match the selected value.



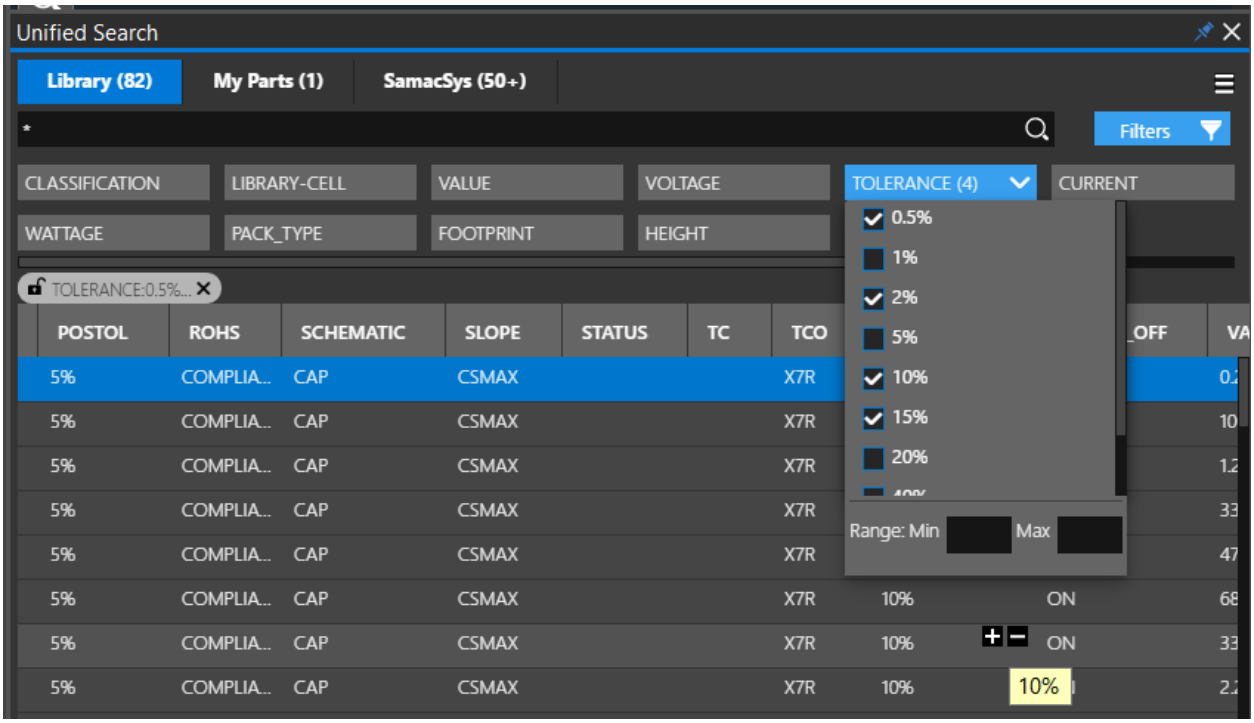
? 2023
All Rights Reserved.

For example, exclude `1.27 mm` from a search. All capacitors whose height is not `1.27 mm` are displayed.



You cannot mix positive and negative values. For example, if you select `TOLERANCE 0.5%, 2%, 10%, 15%` and select `-10%`, only capacitors whose tolerance is not `10%` are displayed.

All Rights Reserved.



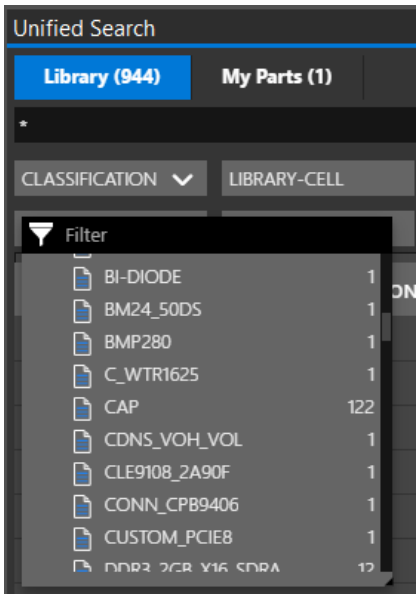
Using Chiplets

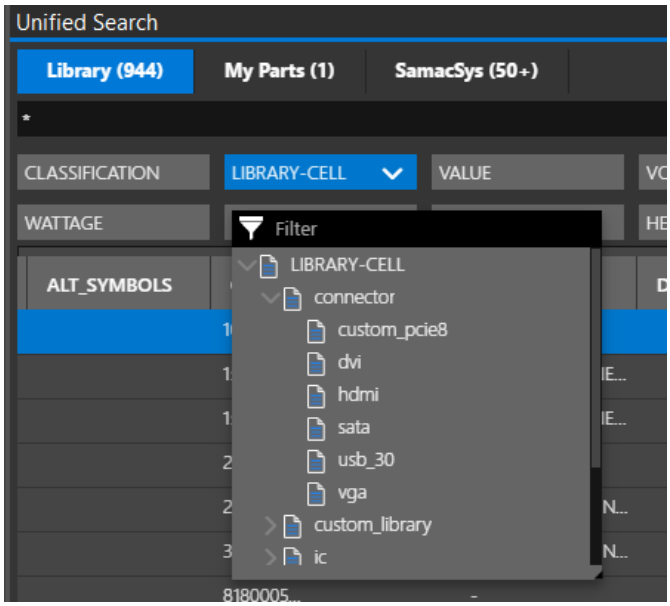
To use a chiplet in search, do the following:

1. Click the required filter in the list.

? 2023
All Rights Reserved.

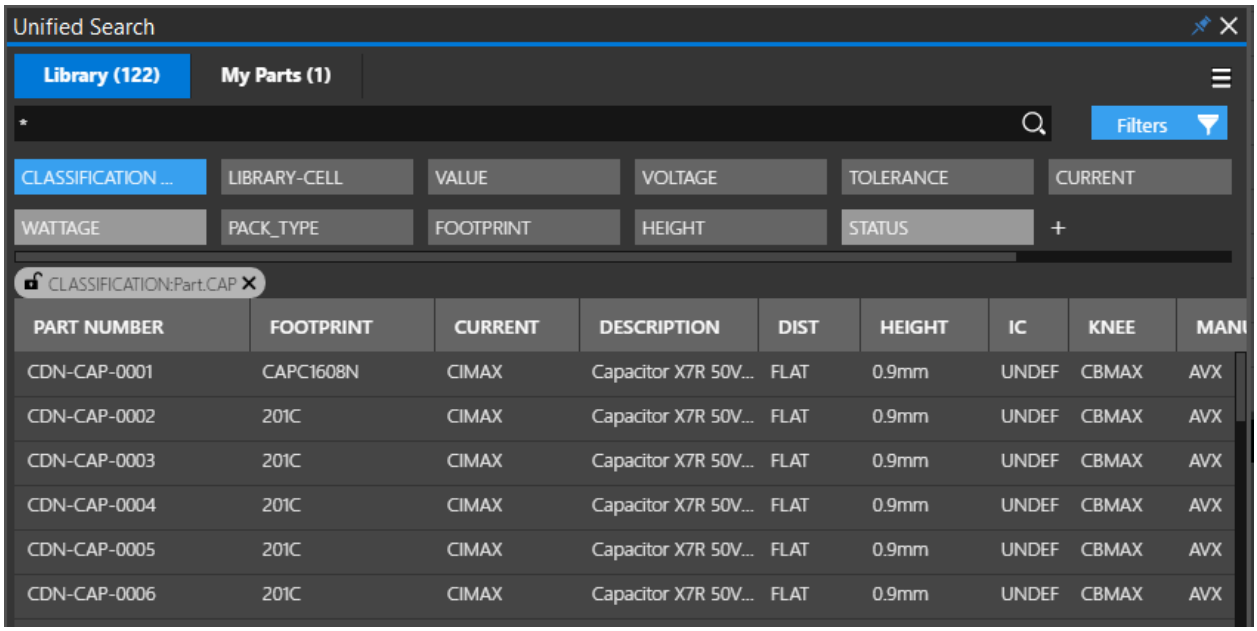
For example, select *CLASSIFICATION*. Or, for example, select *LIBRARY-CELL* to view a particular library.





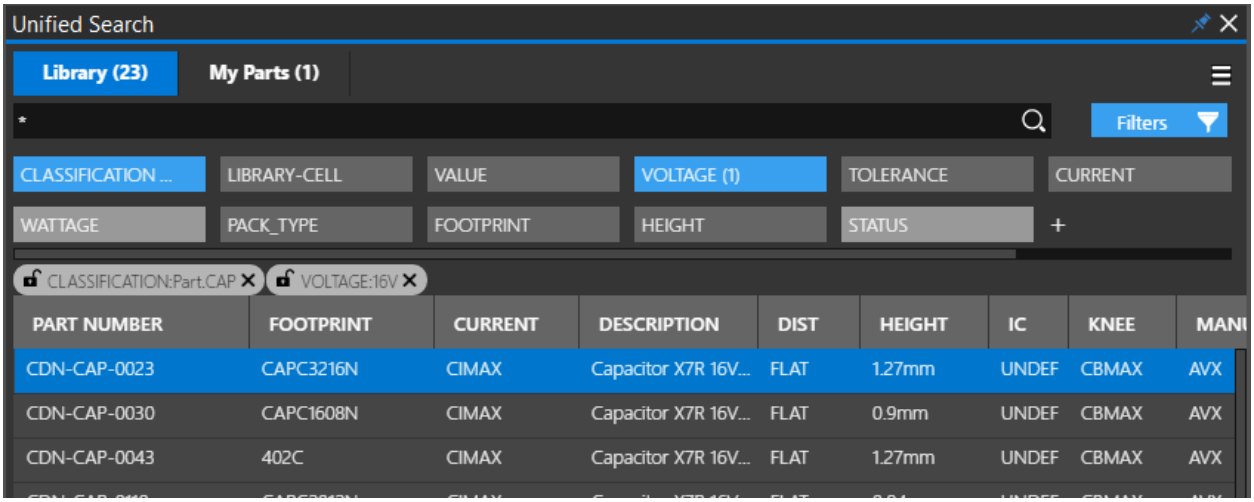
2. Select the desired value.

   For example, select *CAP* for *CLASSIFICATION*. The search results are narrowed down to capacitors only.



All Rights Reserved.

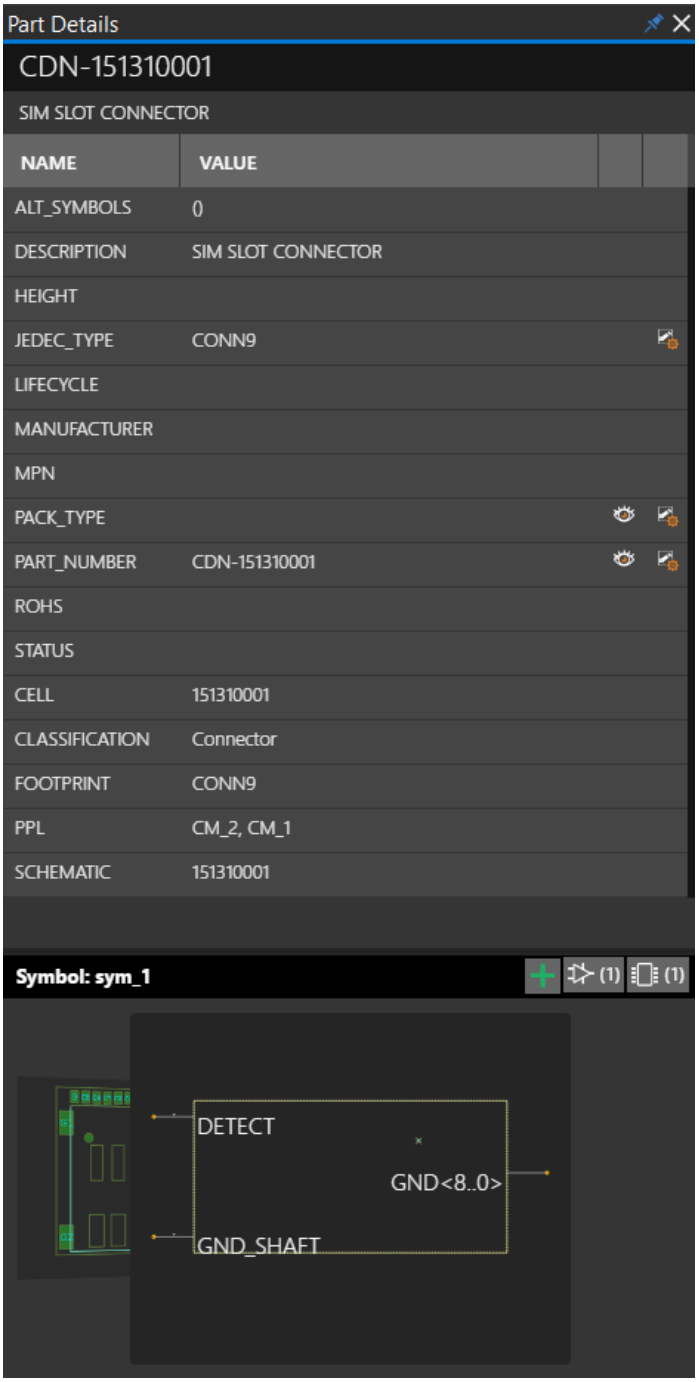
You can add as many chiplets as required.



? 2023
All Rights Reserved.

Viewing Graphics in Part Details Pane

Along with part properties, such as the JEDEC type, part number and description, schematic name, you can also view a part symbol and footprint in the graphics panel of the *Part Details* pane.

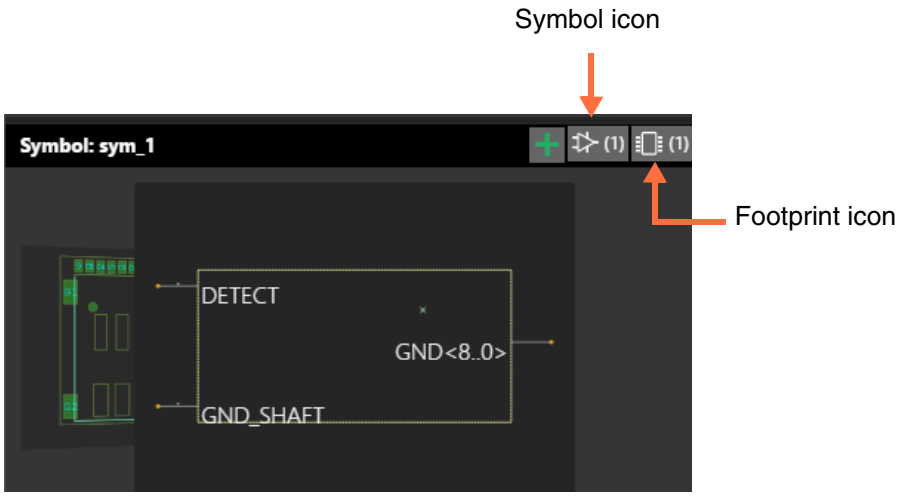


? 2023
All Rights Reserved.

To view a part symbol and footprint, do the following:

1. Click the symbol icon to view a symbol.

   The number within parentheses beside the symbol indicates the number of symbols available for the part.



2. Click the footprint icon to view footprints.

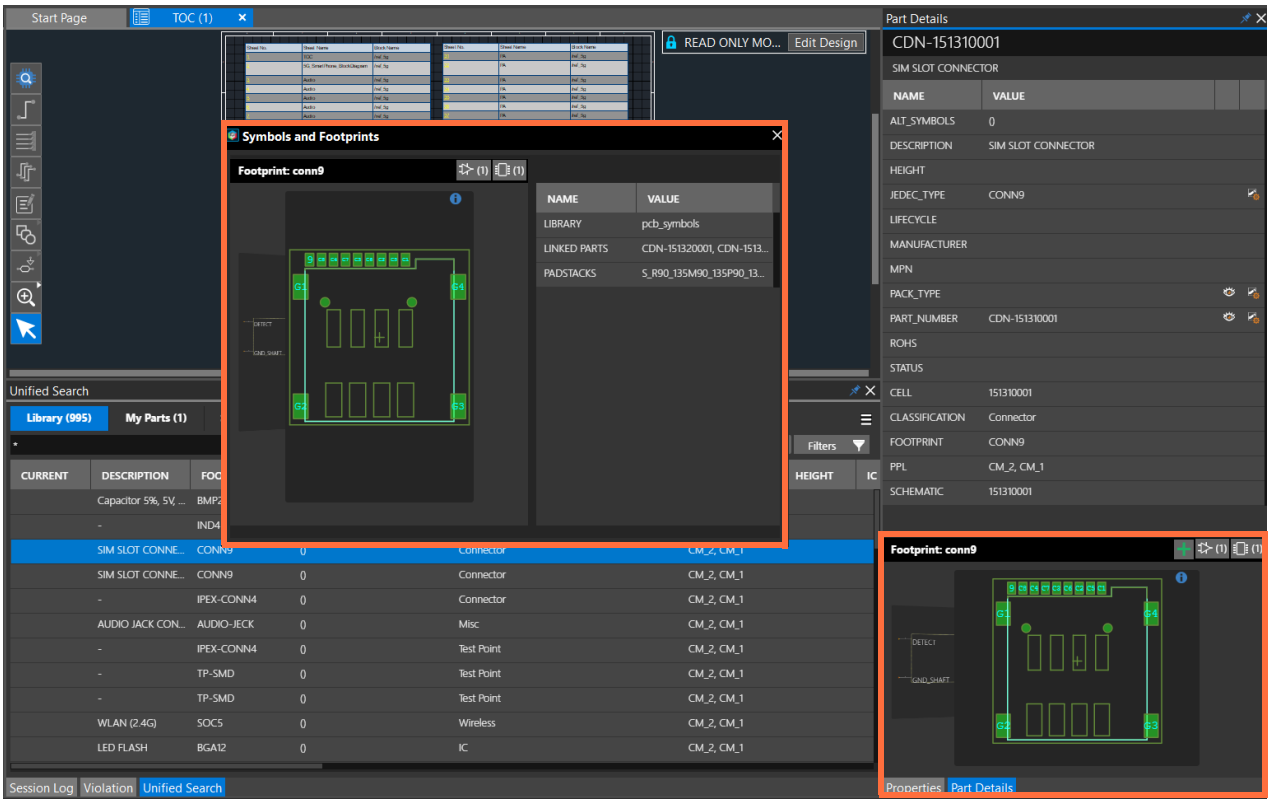
   The number within parentheses beside the symbol indicates the number of footprints available for the part. The number includes alternate footprints, if available.

   You can click through the carousel of symbol and footprint images.

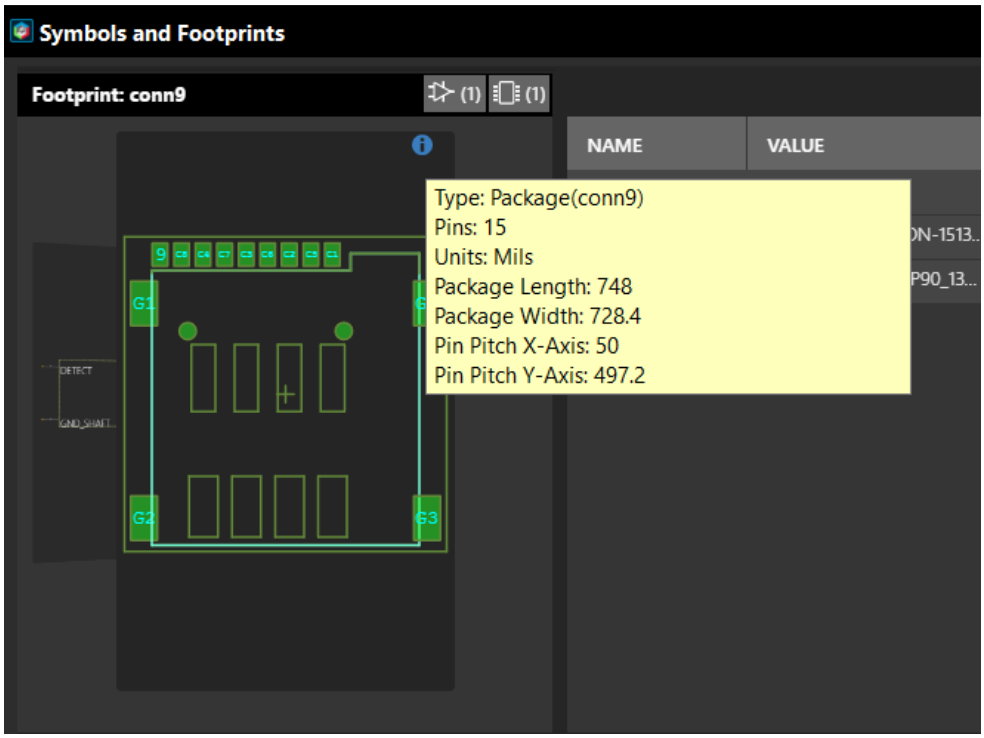
   If you specify a footprint directly in the model definition, that is, an `.olb` file, the library reader only reads footprints from the OrCAD CIS database. Such footprints are not displayed in the *Unified Search* results. For the information of such footprints to be displayed, footprints must be linked to parts in the OrCAD CIS database.

All Rights Reserved.

To see an expanded view of a symbol or footprint, click the symbol or footprint in the graphics panel.



? 2023
All Rights Reserved.



Adding Parts to BOM

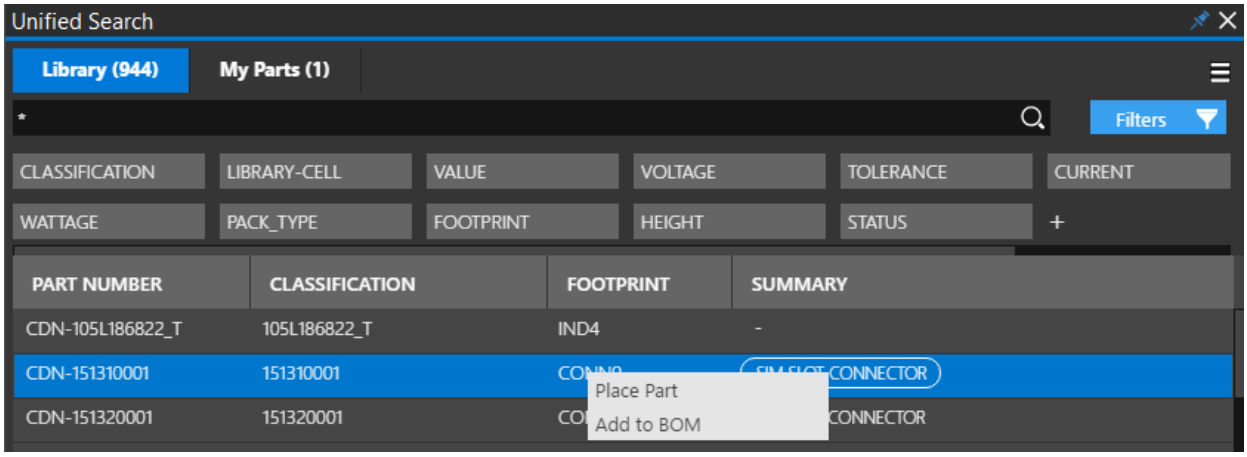
A part added to a design is automatically added to the BOM. The *Add to BOM* option allows you to add a part to the BOM without placing it on the canvas.

To add a part to *Live BOM*, do the following:

1. Select a part to view its properties.

All Rights Reserved.

2. Right-click and select *Add to BOM*.



An option to request a new part is also available if:

- ❑ New parts requests have been enabled by the Pulse administrator
- ❑ You work in the multi-user environment where you are connected to a central Pulse server using a remote URL.

Related Topic

Allegro System Capture Pulse Guide

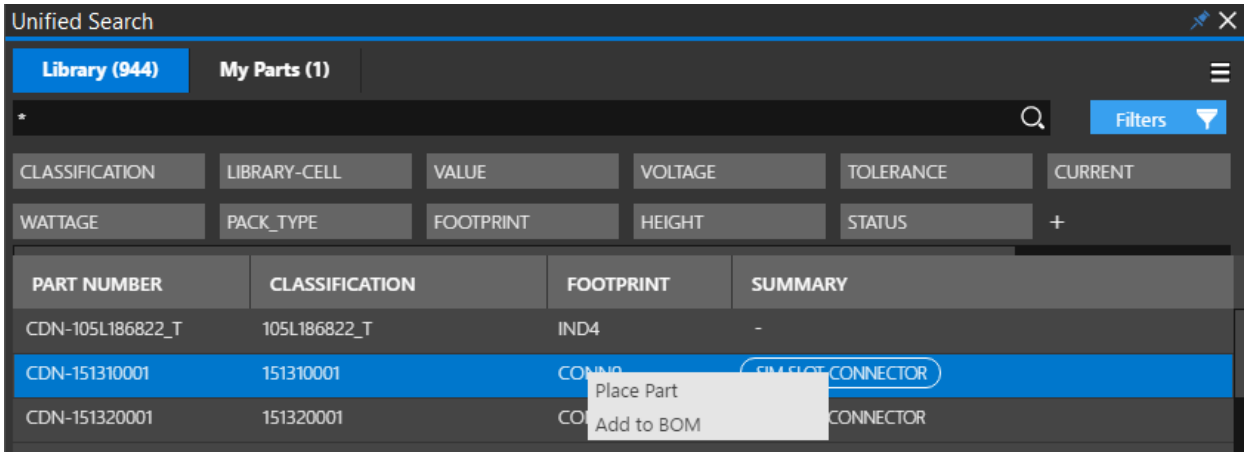
Placing Parts

To place a part on the canvas, do the following:

1. To decide whether to place a part, select it to view its properties.
2. Do one of the following:

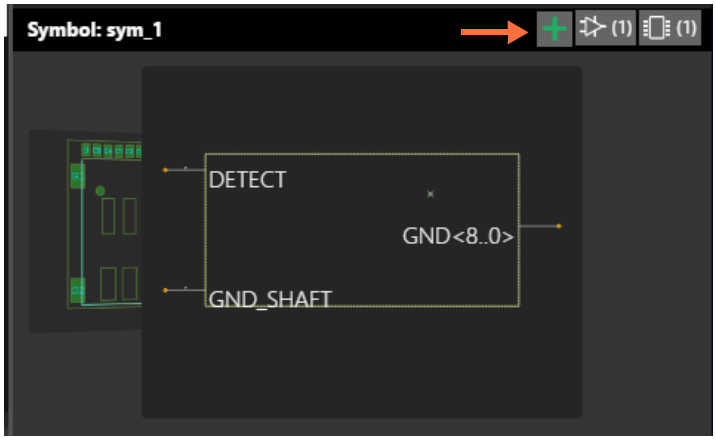
? 2023
All Rights Reserved.

- ❑ Right-click and select *Place part*.



If you work in the multi-user System Capture environment where you are connected to a central Pulse server using a remote URL, an additional option, to request a new part, is available.

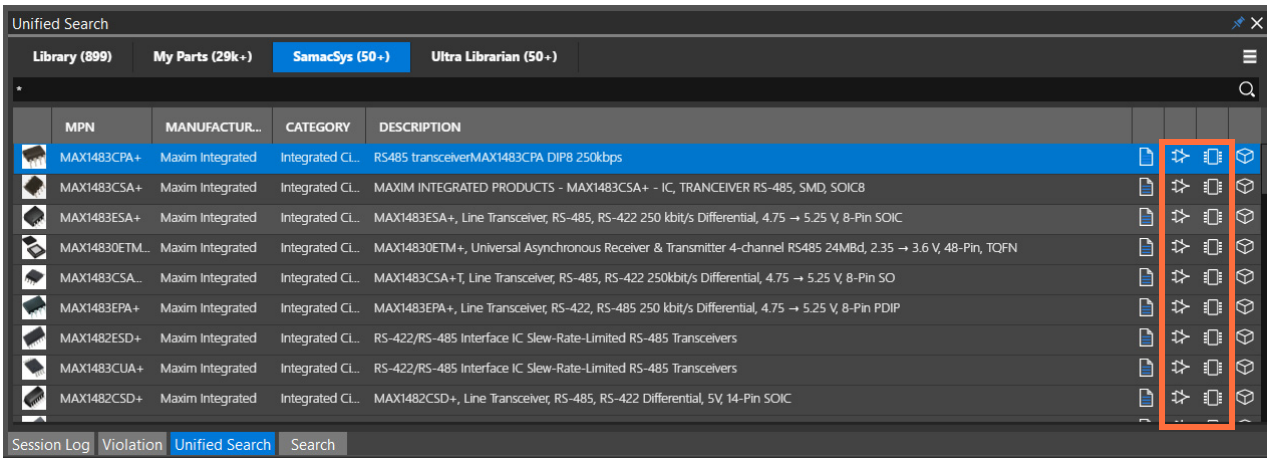
- ❑ Click the plus (+) icon in the graphics panel of the Part Details pane.



? 2023
All Rights Reserved.

Working With Parts From Content Providers

Unified Search provides parts from SamacSys and Ultra Librarian after you log in to these content providers. A part can be placed in a design only if it has a symbol and a footprint, which are indicated in *Unified Search* through icons as illustrated in the following image:



SamacSys and Ultra Librarian display the manufacturer part number (MPN). However, the following additional fields are displayed for Ultra Librarian:

- Distributor ID (*Distributor*)
- Distributor Part Number (*Distributor PN*)
- Information on the cost of a part

When you log in to a content provider, you can do the following tasks:

- Viewing Parts from Another Content Provider
- Replacing External Vendor-Provided Parts in Designs

Viewing Parts from Another Content Provider

If you log in to *Unified Search* after selecting only one external vendor, such as SamacSys, but now want to search for parts from Ultra Librarian, do the following:

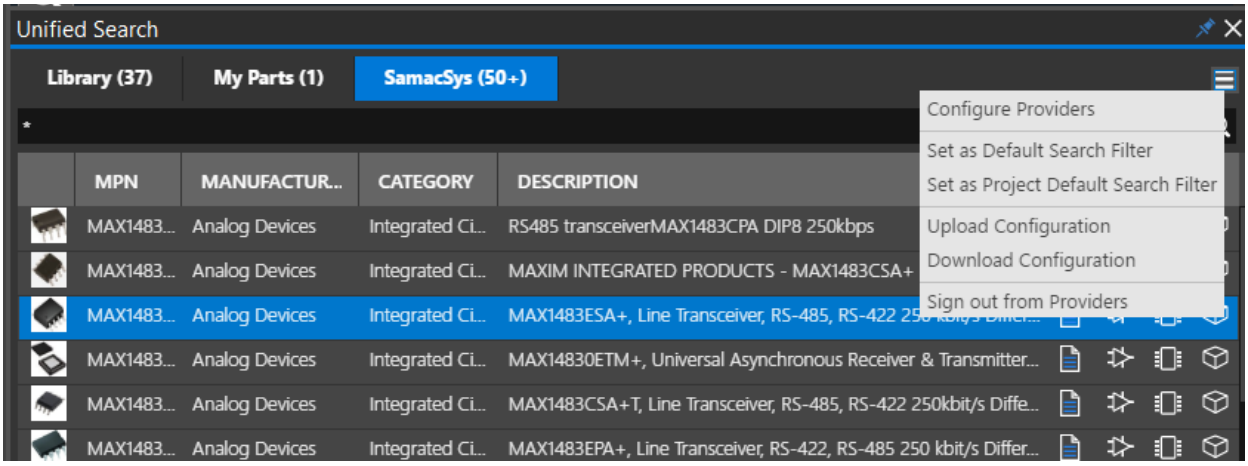
1. Click the hamburger menu (☰).
2. Select *Sign out from Providers*.

   The *Sign in to Providers* dialog box is displayed.

? 2023
All Rights Reserved.

3. Modify your selections as required and sign in to *Unified Search* again.

   Parts from SamacSys and Ultra Librarian cannot be copied and pasted in another design or imported into another design unless you include the libraries in the target design.



Replacing External Vendor-Provided Parts in Designs

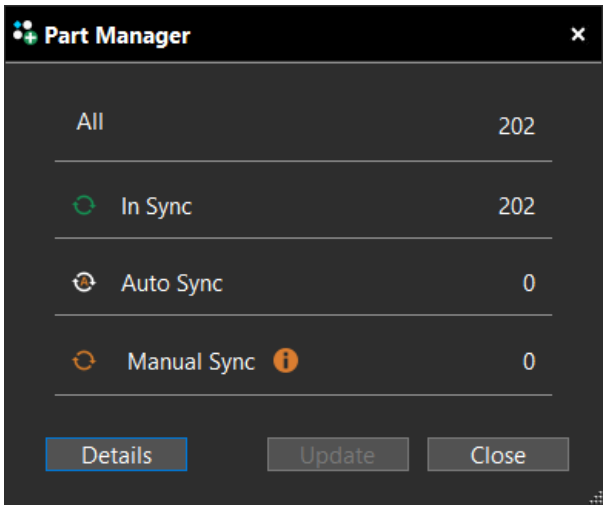
To replace a part in a design with an external vendor-provided part, do the following:

1. Right-click a placed part in the System Capture canvas.
2. Select *Replace*.
3. Use the *Find and Replace* dialog box that opens to select a part from the selected external vendor.

All Rights Reserved.

Part Manager Notifications for System Capture Users

Although Part Manager does not notify Allegro System Capture users of changes to parts by SamacSys or Ultra Librarian, external vendor-provided parts already placed in a System Capture design are always in sync with the library even if parts are modified by SamacSys or Ultra Librarian.



If you modify any part definition in the `downloaded_parts` library, Part Manager notifies you of those differences.

Placing External Vendor Part Footprints in PCB Editor

When parts from external content providers are placed in a schematic, the corresponding footprints are also downloaded to a folder, `allegropcb`, on the user's machine. This folder is in the same location as the `downloaded_parts` folder in the location defined in the *<HOME>* environment variable.

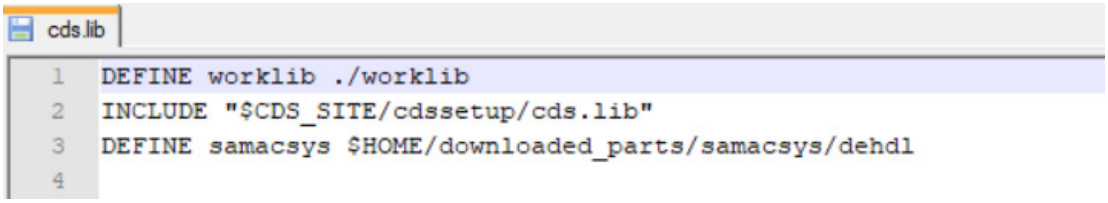
To place these footprints in a PCB Editor design, the local library must be available to PCB layout engineers.

Download Location for Content Provider Parts

When you place a part from an external content provider in a System Capture design, the part is downloaded as a zip file to a temp folder and is unzipped in a folder, `downloaded_parts`, in the location defined in the *<HOME>* environment variable.

? 2023
All Rights Reserved.

After an external part is placed in a design, it is cached in your local library and the library is automatically added to the `cds.lib` file of your design project.

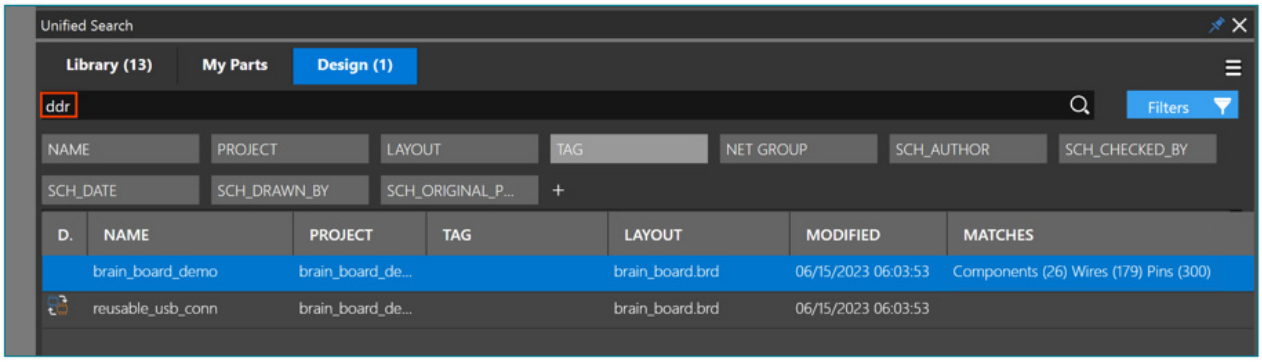


? 2023
All Rights Reserved.

Design Search for Reuse

System Capture designs can be imported into other System Capture designs from a local directory or from Pulse to enable easy reuse of designs, blocks, or pages. Designs cannot be imported across library types. For example, a DE-HDL library design cannot import OrCAD Capture design content.

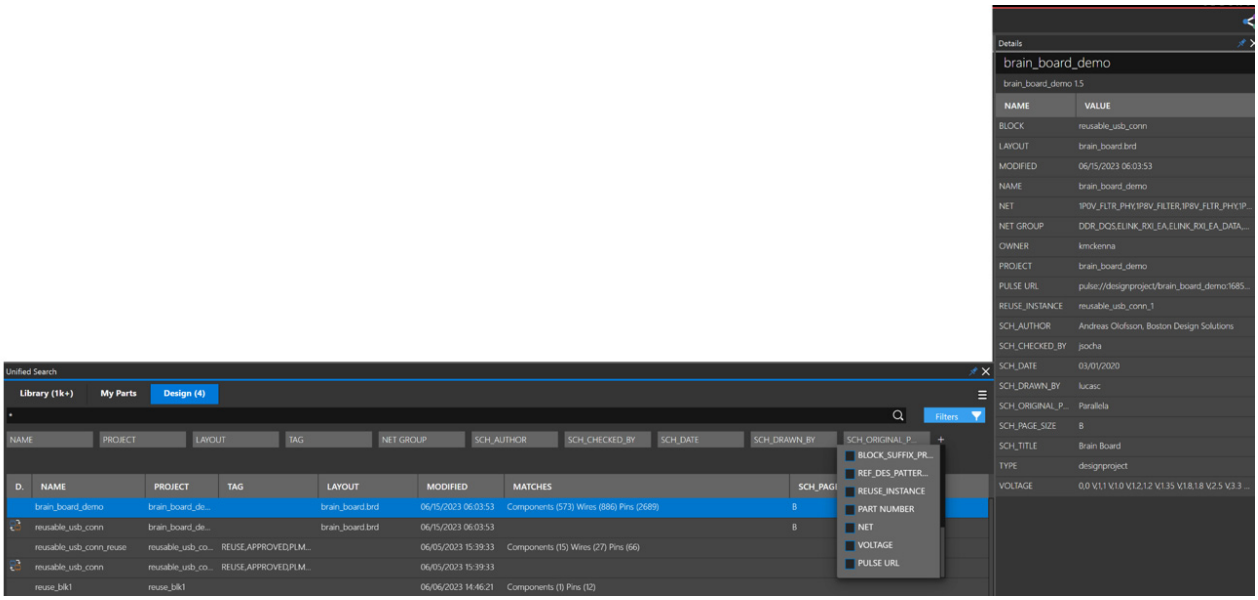
The advantage with the Pulse import is easy searchability using filters in *Unified Search,* access to the latest design data, notifications about changes in the source files, and a preview of the design to help you decide whether to reuse it.



The *Design* tab enables a detailed search of data within designs, and design metadata. Data within a design is searched for, as well as the names or metadata. For example, searching with the PART NUMBER facet displays design results where that specific part number is used.

? 2023
All Rights Reserved.

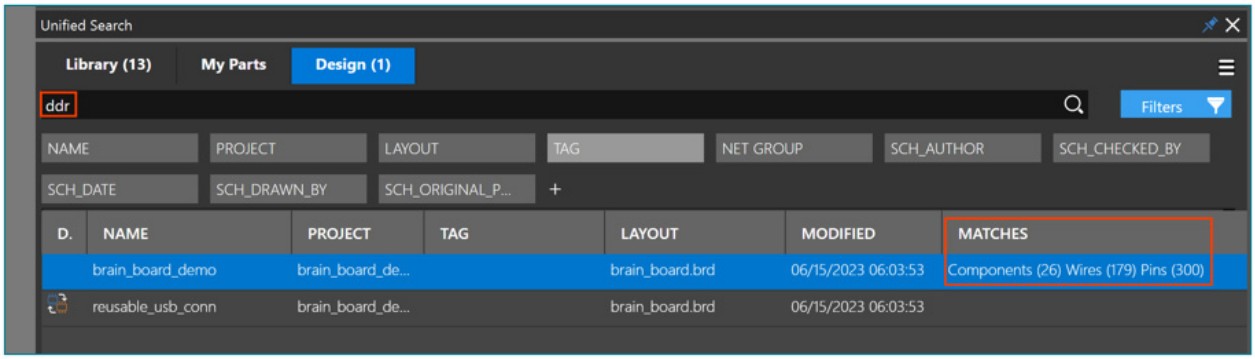
Search results display data only from the latest committed versions of the designs. You can also perform a parametric search of a design row in the *Details* pane.



Searchable data includes:

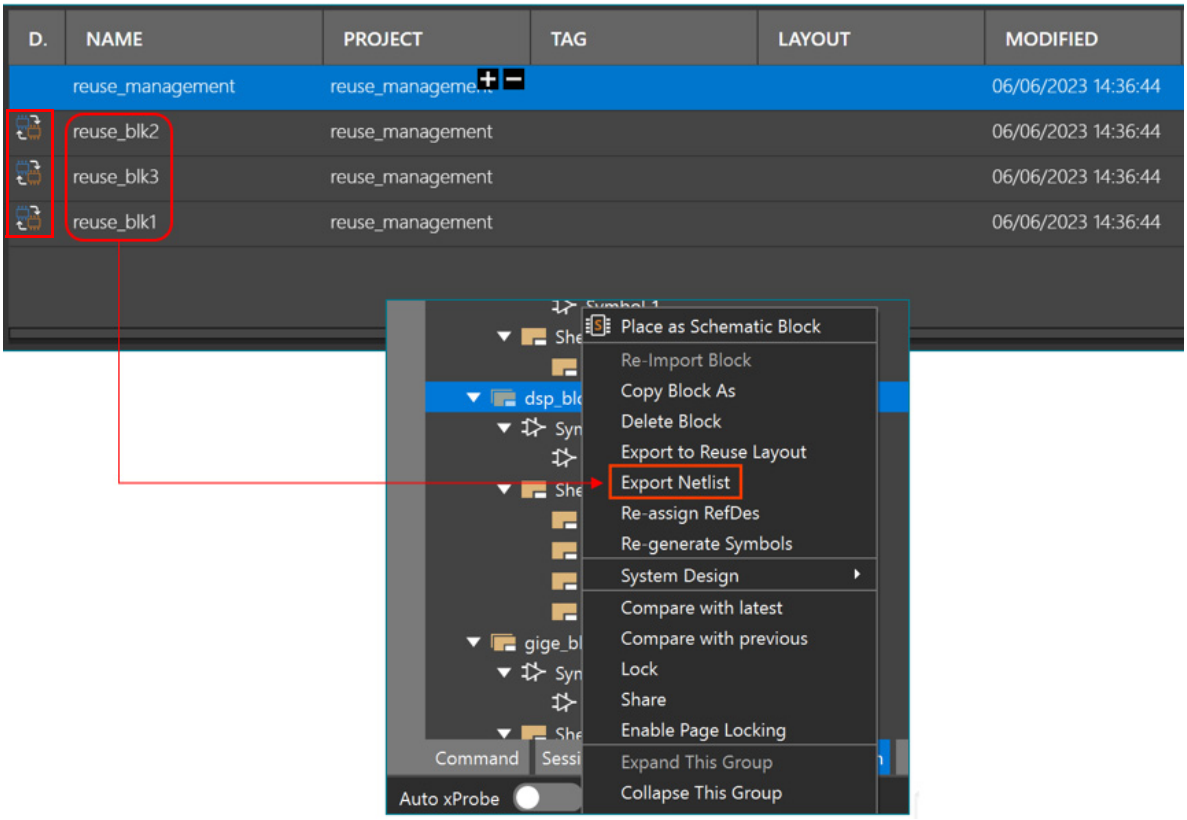
- Pulse metadata: Content in the project details form, version, the last modified date, design owner names, users with read-only access to a design, commit tags, comments, the Pulse URL
- Design information: Nets, net groups, buses, components, pins, voltages, CUSTOMVARs, variant names, block attributes, layouts associated with schematics

  Design information search results are aggregated into matches.

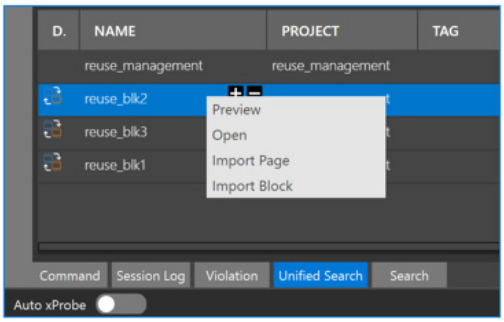


? 2023
All Rights Reserved.

To enable easy reuse of a design, blocks whose netlists are exported for reuse are identified with an icon in the first column. Exported blocks are always displayed below their parent root design.



With read-only access to a design, you can preview it, import blocks or pages, and open a design in a new System Capture instance. You can quickly copy and paste the required elements of one design into another between two System Capture instances.



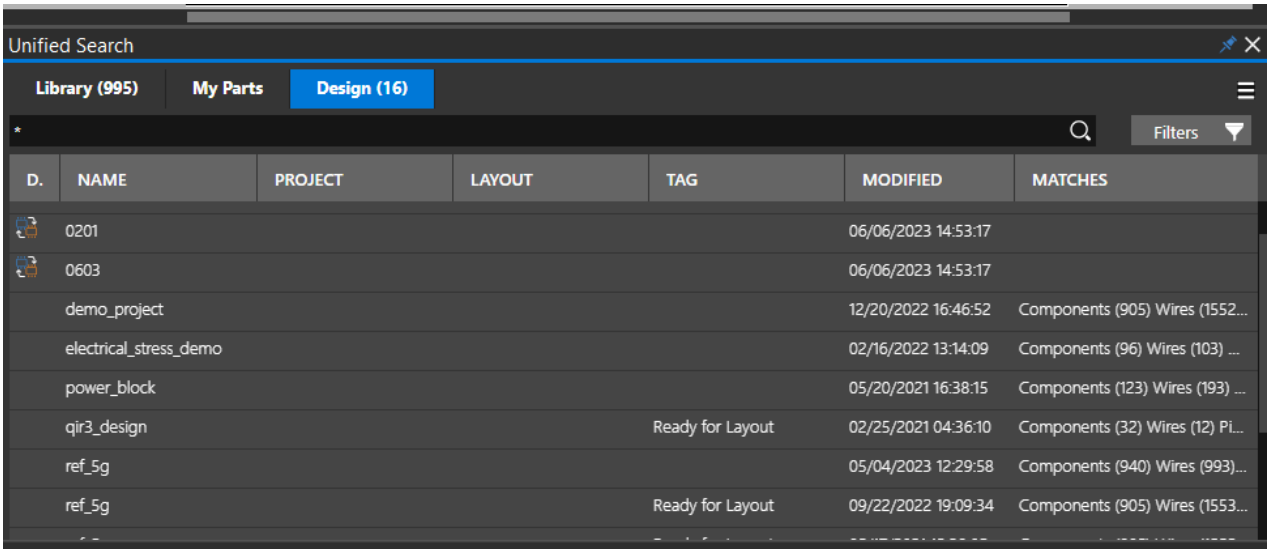
? 2023
All Rights Reserved.

Only project owners or users with whom a design is shared with read or write permission can import designs, blocks, or pages to reuse them.

You can also:

- Add facets and filter the results in the *Design* tab.
- Exclude results.
- Remove a design from the design list to hide it from future searches.
- Lock chiplets and set a filter or project as the default.
- Upload and download configuration for the *Design* tab.

  For details on these tasks, see Configuring Unified Search.



Searching for Designs for Reuse

To search for a design, do the following:

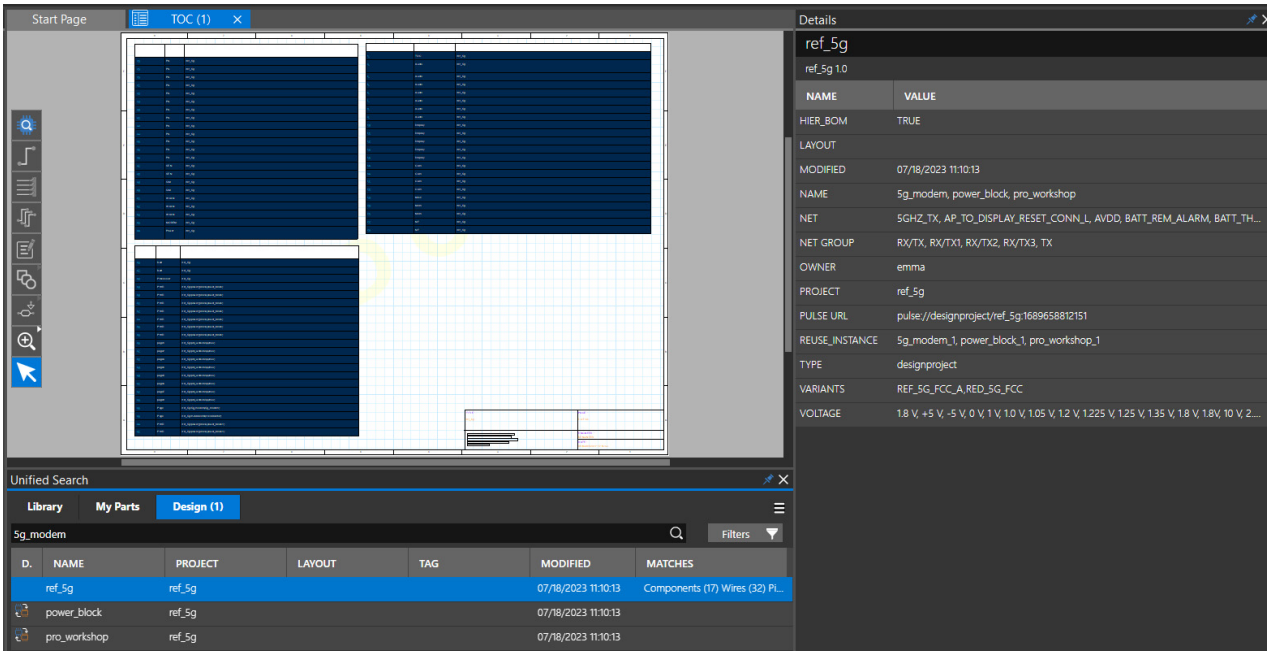
1. Open a project in System Capture.
2. Log in to System Capture.
3. Log in to *Unified Search*.
4. Click the *Design* tab.

? 2023
All Rights Reserved.

5. Use an asterisk to search for all root designs and all instantiated blocks marked for physical reuse in designs.
6. Select a row in the *Design* pane.

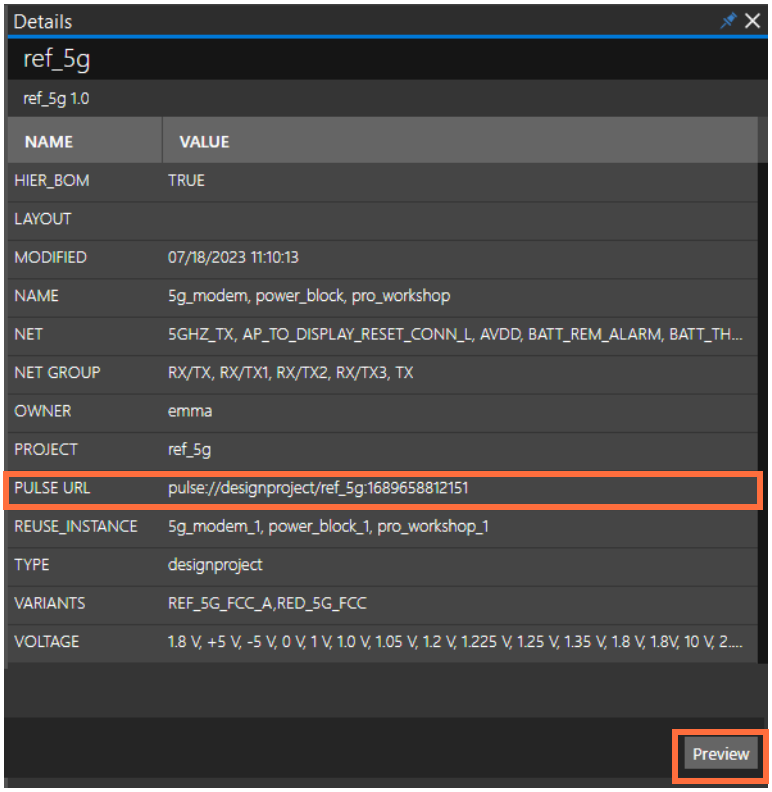
   The *Details* pane is displayed and shows all the block attributes, custom variables, variants, layouts, modules, blocks, nets, and voltages. Some properties, such as NET, VOLTAGES, are displayed in the *Details* pane but are not available in the search results.

   You can also view layouts linked to the designs and blocks.



? 2023
All Rights Reserved.

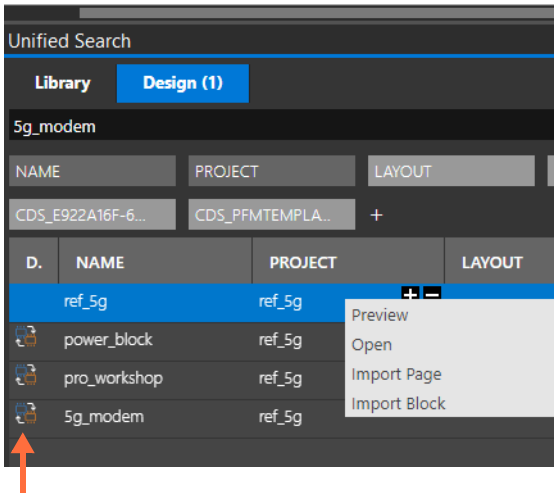
The Pulse URL of the design whose page or block you selected is also available in the *Details* pane. You can use this URL to quickly search for the design.



To import a page, block, or design, do the following:

? 2023
All Rights Reserved.

1. If you work in a team design environment, ensure that you have at least read permission for the block or page you want to import. You must have write permission for the destination design.



Blocks marked for physical reuse are indicated with the *Reuse Block* icon

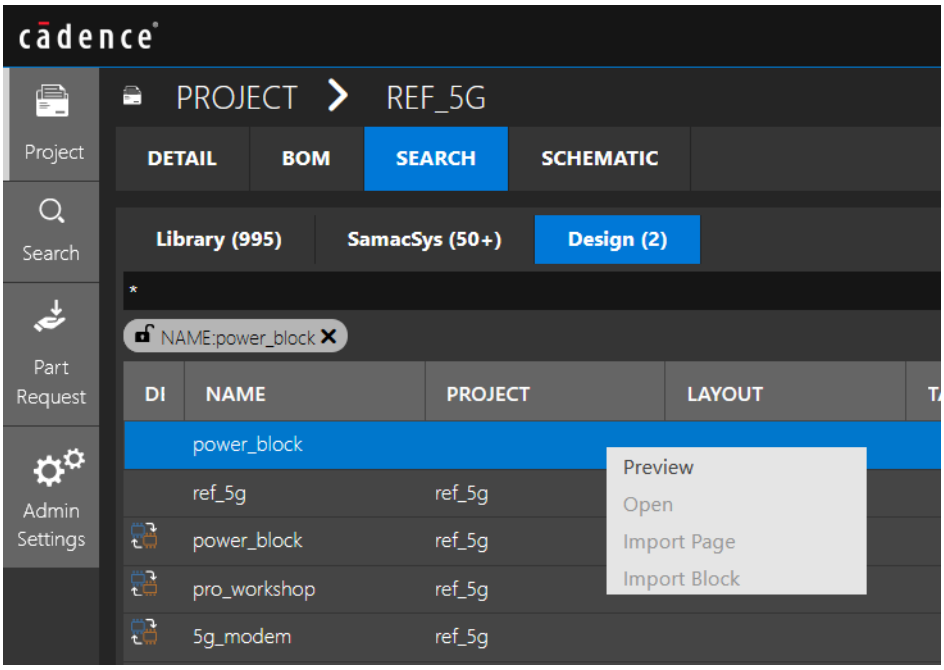
2. Right-click the required row and click one of the following to:

? 2023
All Rights Reserved.

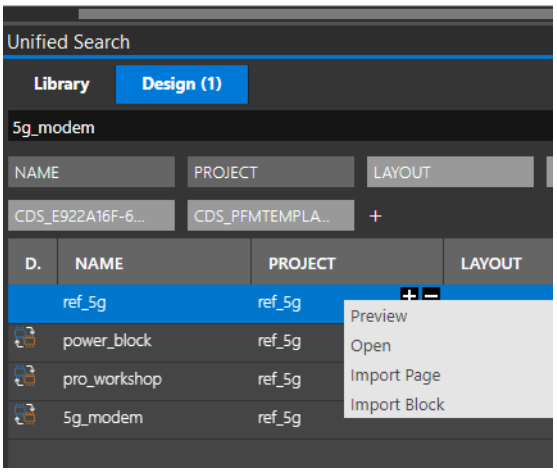
- *Preview* the design, block, or page to make an informed decision before importing it. The *Preview* search result includes a PDF with hierarchical navigation. You can right-click an item in the hierarchy navigation, and import a block or page.

? 2023
All Rights Reserved.

The *Preview* option is also available in the *Details* pane and in the *Search* tab of a project in the Pulse web dashboard.



- *Open* to view the design in greater detail. The design is opened in read-only mode. You can copy required circuitry from here to the destination design.
- *Import Page* or *Import Block*



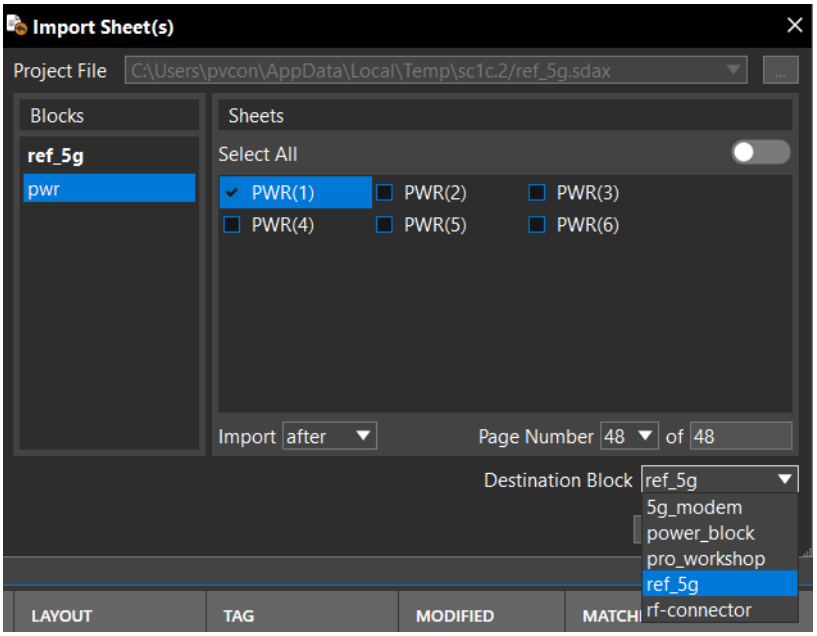
You can only import pages and blocks you own or that are shared with you.

? 2023
All Rights Reserved.

The *Import Sheet(s)* or *Import Block* dialog box is displayed. All the blocks and sheets in the source project of the selected sheet are listed.

3. If you are importing a sheet, do the following:

   a. Select the required items in the dialog box.



   b. Define where you want the page, block, or design imported in *Import* and *Page Number*.

   c. Select the block into which you want to import the selected items in *Destination Block*.

   d. Click *OK*.

4. If you are importing a block, do the following:

   a. Toggle off the *Read Only* button if you want to import the block with read-write permission.

   b. Click *OK*.

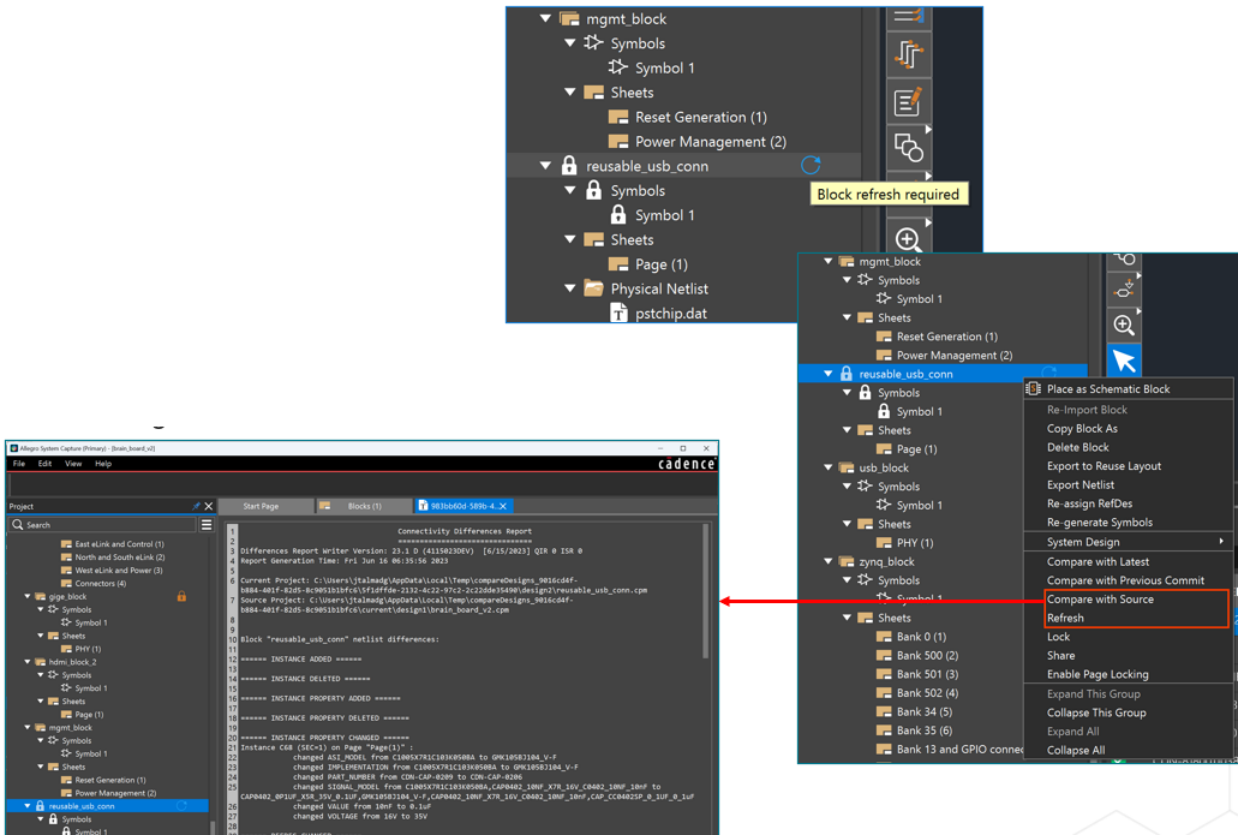
   Blocks that are modified in the source design after they are added to a destination design are displayed with a marker in the *Project* explorer tree.

   When a source design is committed and a change in it impacts a reused block, System Capture displays reuse update notifications for the blocks.

? 2023
All Rights Reserved.

You can view the differences between the source and the current block to assess the impact and decide whether to update your design.

To compare, right-click and select *Compare with Source*. Then click *Refresh* to update your design with the latest version of the reuse block. You can also compare the differences between two pages or blocks to decide whether to update your design. Ensure that you update any blocks that need updating before you synchronize blocks.



On clicking Refresh, all the local changes in the blocks are discarded and the blocks are reimported. You can sync read-only and read-write blocks. Sheets cannot be synchronized.

? 2023
All Rights Reserved.

2

Configuring Unified Search

Project-specific configuration in *Unified Search* can be done by:

- Pulse administrators of a multi-user System Capture environment
- Designers in a single-user environment where design and library data is on the same machine
- Design project owners who connect to a central Pulse server using a remote URL in a multi-user System Capture environment

Global configuration can be done by:

- Pulse administrators of a multi-user System Capture environment
- Designers in a single-user environment where design and library data is on the same machine

For details on configuring *Unified Search*, refer to *Allegro Pulse Configuration Guide*.

Project-Specific Configuration

To configure *Unified Search* for a specific project, see the following:

- Locking Chiplets
- Configuring Search Results Display
- Setting Filters as Project Default

Configuration for all Projects on a Machine

Designers in a single-user environment can configure *Unified Search* for all projects on a machine. This is done by modifying a configuration file. See the following:

- Using Custom Configuration File

? 2023 All Rights Reserved.

- Modifying Filter Display Order
- Configuring Contents of Summary Column in Part Search

Note: Wild cards and regular expressions cannot be configured and enabled in the single-user environment.

Locking Chiplets

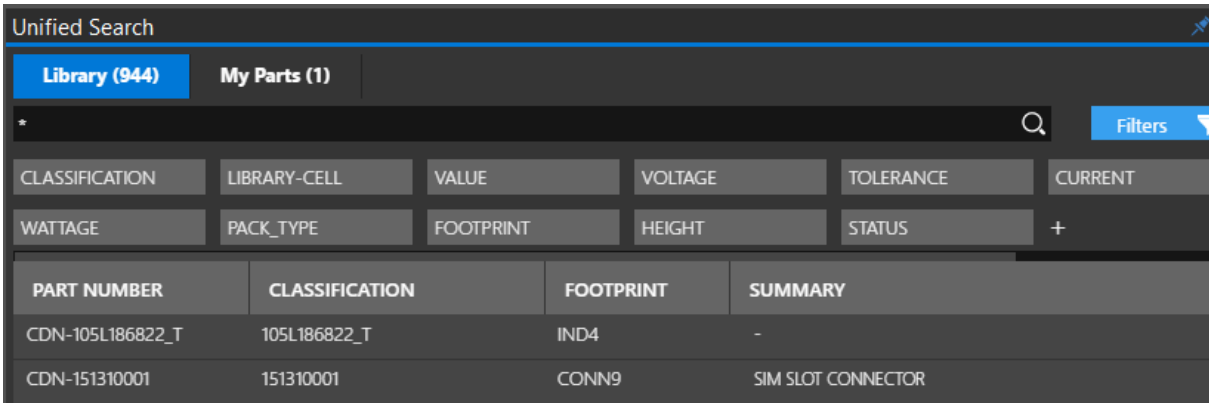
In the multi-user environment, you can lock a chiplet to restrict other designers from removing it and to make it a mandatory search criterion.

In a single-user environment, locking a chiplet ensures that it is always available in a specific project.

To lock a chiplet, do the following:

1. Click the *Filters* button on the top right of the part search pane.

   The filters are displayed.

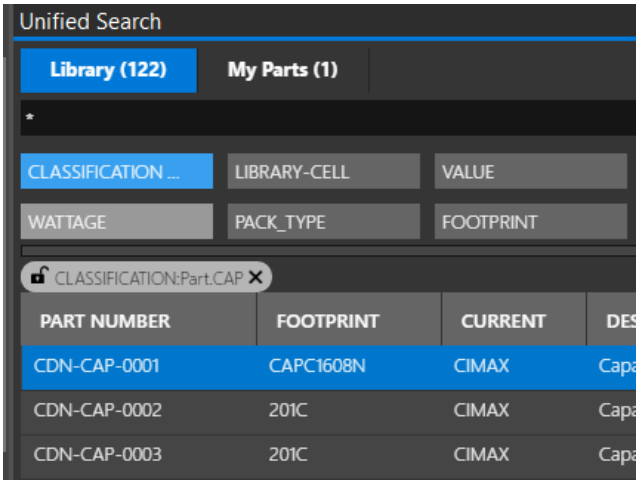


2. Click a filter. For example, click *CLASSIFICATION*.

? 2023
All Rights Reserved.

3. Select a filter value. For example, select *CAP*.

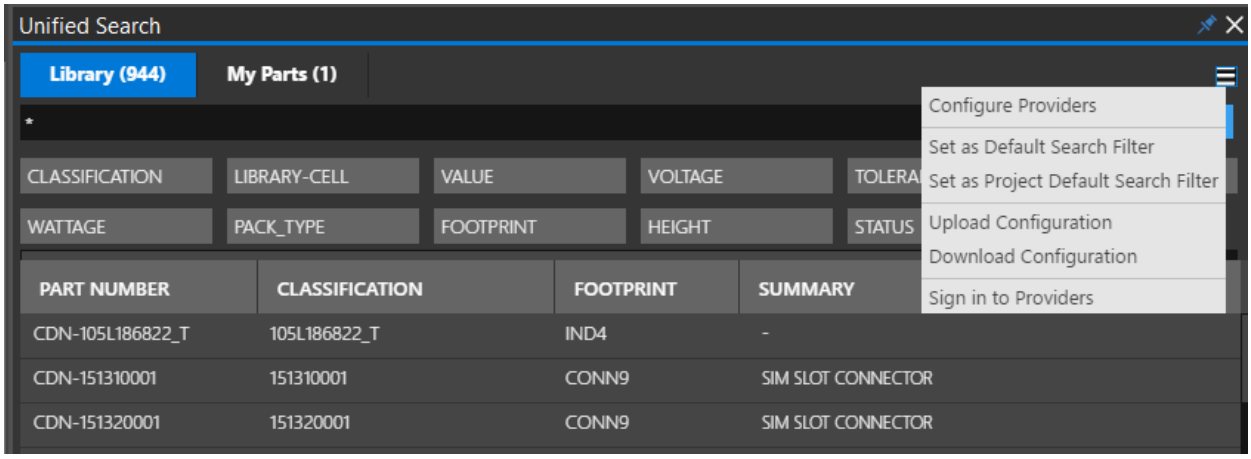
Make Mandatory icon



4. Click the *Make Mandatory* lock icon to restrict removal of a chiplet.

   This makes the chiplet a mandatory search criterion.

5. Click the hamburger menu on the top right of the *Unified Search* pane.



   Note: The *Upload Configuration* and *Download Configuration* options are only available to designers whose design and library data is on the same machine. These designers do not connect to a central Pulse server.

6. Select *Set as Project Default Search Filter*.

All Rights Reserved.

Setting Filters as Project Default

You can set a search query as the project default if you are a:

- Designer in a single-user environment

  For example, if you work on an automotive design and only want automotive grade parts, apply this default search filter to ensure that you do not accidentally select non-automotive parts.

- Design project owner in a multi-user environment

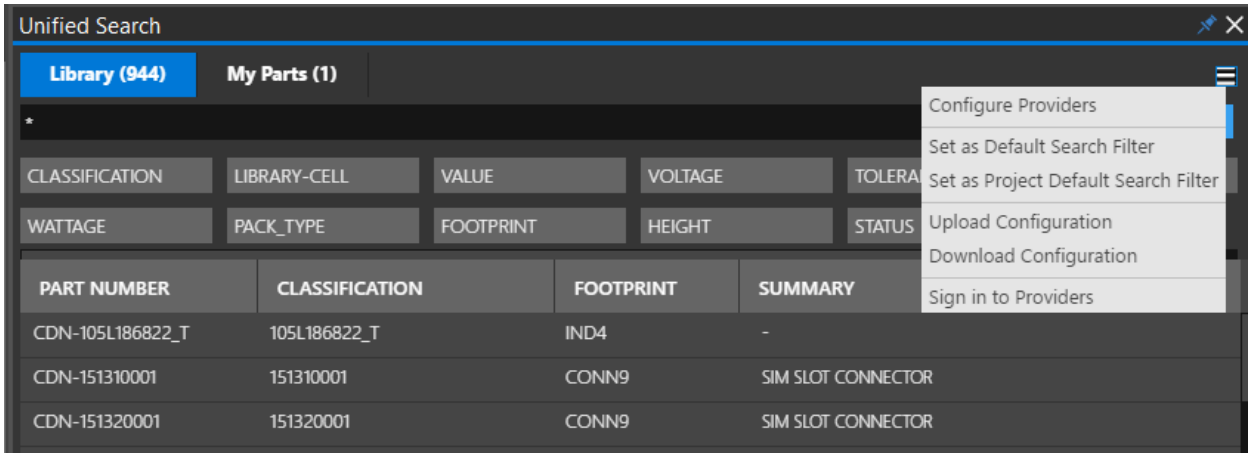
  This is helpful when a project is shared with others. For example, a locked chiplet cannot be edited or removed.

- Pulse administrator of a multi-user System Capture environment

  For details on configuring *Unified Search*, refer to *Allegro Pulse Configuration Guide*.

To set a filter as the project default, do the following:

1. Click the hamburger menu on the top right of the *Unified Search* pane.



   Note: The *Upload Configuration* and *Download Configuration* options are only available to designers whose design and library data is on the same machine. These designers do not connect to a central Pulse server.

2. Select *Set as Project Default Search Filter*.

? 2023
All Rights Reserved.

Configuring Search Results Display

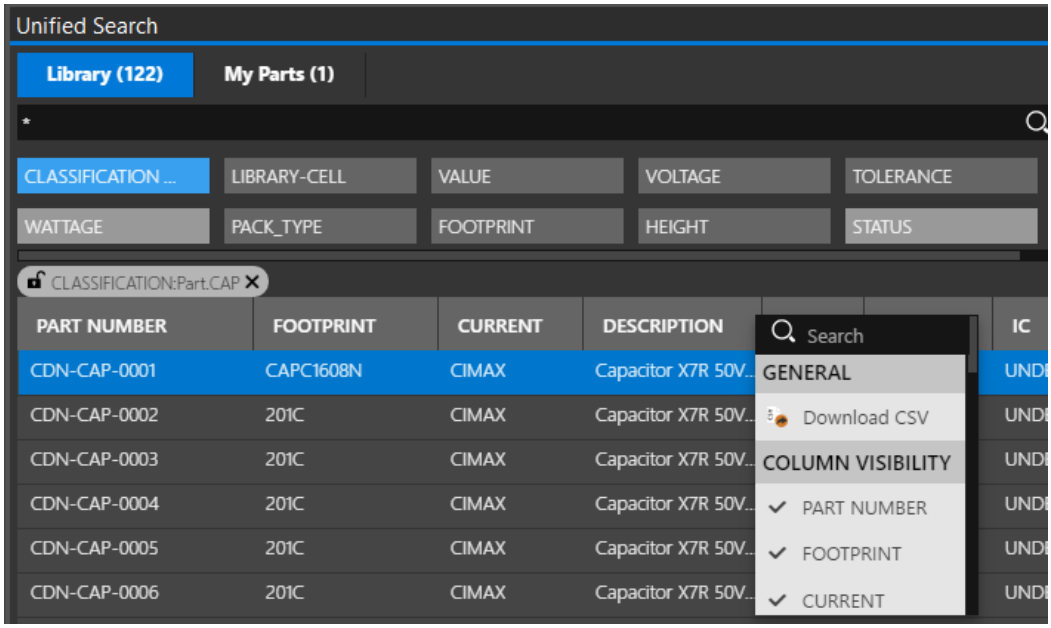
You can configure how you want the part search results to be displayed by:

- setting column visibility
- modifying column widths
- defining the column order

To configure the display of the search results, do the following:

1. Apply a filter or specify keywords to search for parts.
2. Click *Enter* or use the magnifier icon to search.

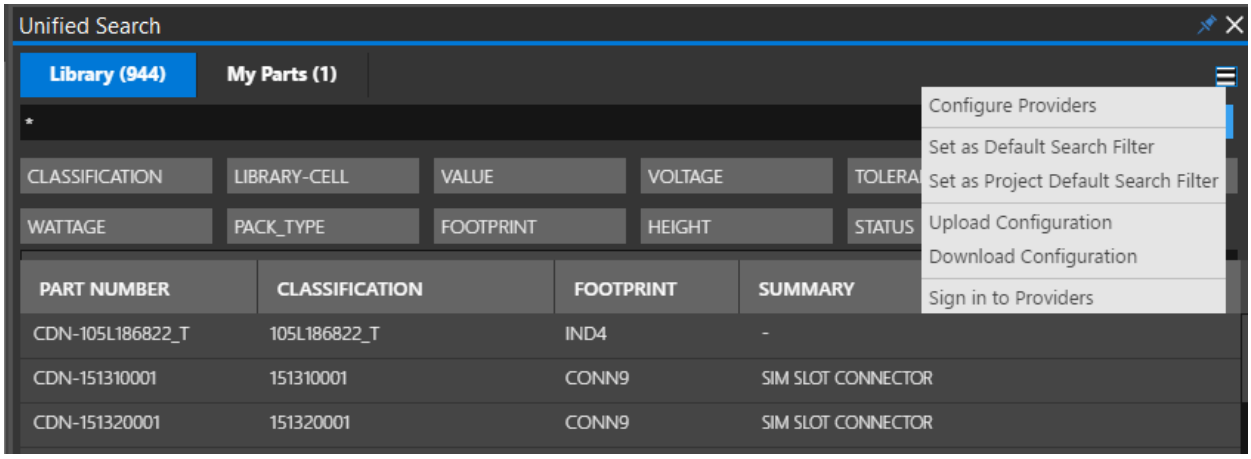
   Part search results are displayed.
3. In the search results table, right-click the column header you want to hide and deselect it.



   If you hide a column, it is hidden for all classifications, such as Capacitors, Connectors, Diodes, Amplifiers.
4. To modify the width of a column, click and drag the column header cell border until the column is the size you need.
5. To rearrange the order in which the columns are displayed, click and drag the selected column horizontally to the position that you want.

All Rights Reserved.

6. Click the hamburger menu on the top right of the *Unified Search* pane.



Note: The *Upload Configuration* and *Download Configuration* options are only available to designers whose design and library data is on the same machine. These designers do not connect to a central Pulse server.

7. Select *Set as Project Default Search Filter*.

Using Custom Configuration File

To configure *Unified Search* for all System Capture projects on a machine, you use the `search.config` file, which uses JavaScript Object Notation.

The steps to access the configuration file, download it, and upload it are the same regardless of the feature you want to configure.

To download and upload the configuration file, do the following:

1. Click the hamburger menu on the top right of the *Unified Search* pane.
2. Click *Download Configuration*.
3. Select a download location for the file.
4. Open the downloaded file in a text editor.
5. Copy the required section from the out-of-the-box `search.config` file, which is available at:

   `<Cadence installation directory>\tools\pcbdw\configs\unicorn`.

6. Modify the contents in the downloaded `search.config` file as required.

? 2023 All Rights Reserved.

7. Save the file.
8. Click the hamburger menu on the top right of the pane.
9. Click *Upload Configuration*.
10. Select the modified `search.config` file and click *Open*.

    The configuration is updated and the changes are applied to all existing and new projects created on the machine.

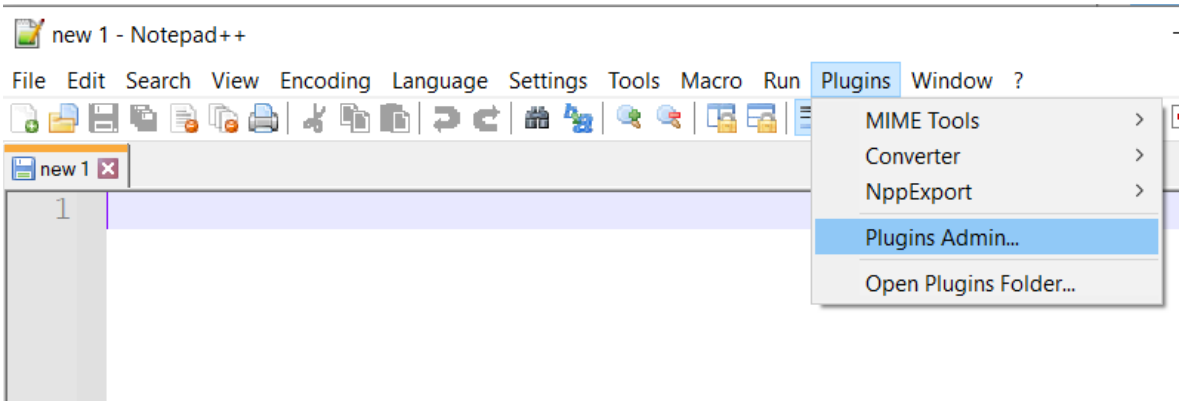
    If you are unfamiliar with JSON editing, consider going through the content at:

    https://www.json.org/json-en.html.

    Some text editors also provide JSON viewer plugins, which flag inconsistencies in the tags. For example, if you use Notepad++, you can enable the JSON viewer plugin by doing the following:

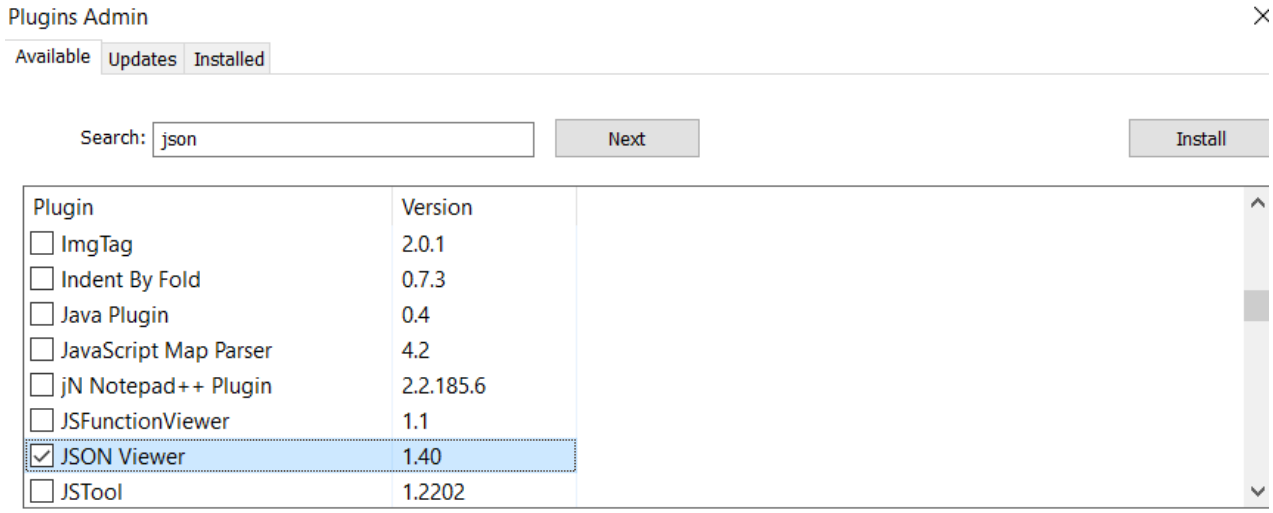
    a. Open Notepad++.

    b. Select *Plugins - Plugins Admin*.



? 2023
All Rights Reserved.

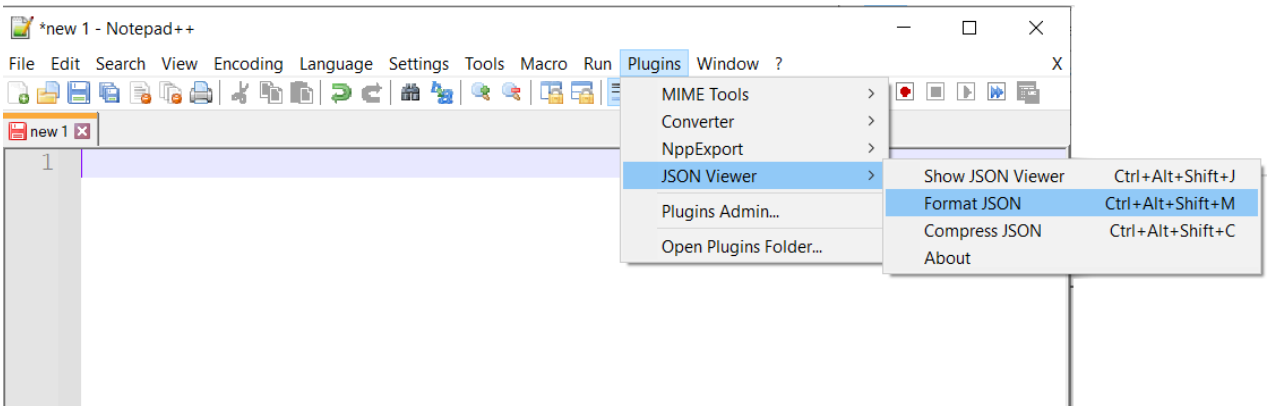
c. Search for JSON and select *JSON Viewer*.



d. Click *Install*.

Notepad++ closes and restarts automatically.

e. Modify the JSON as required and check for inconsistencies using the following option:



Configuring Part Content Providers

Pulse provides access to external parts from SamacSys and Ultra Librarian. If you do not require parts from these providers, you can disable one or both content providers. If both are disabled, you are no longer prompted to log in to the part content providers.

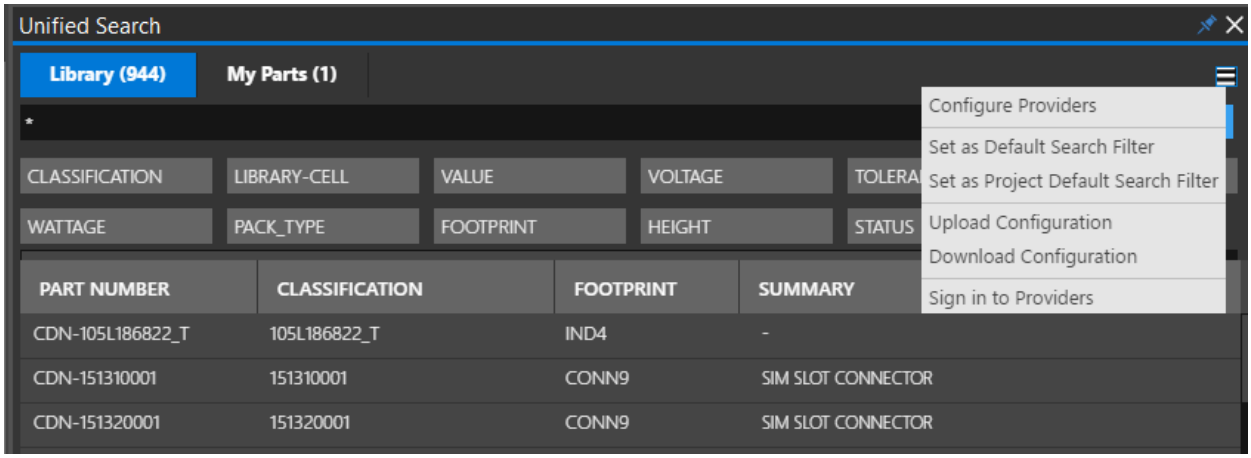
? 2023
All Rights Reserved.

You can also configure the following:

- whether designers can search for parts from a particular provider
- which properties should be annotated when placing a part in the System Capture canvas

To disable content providers, do the following:

1. Click the hamburger menu on the top right of the *Unified Search* pane.

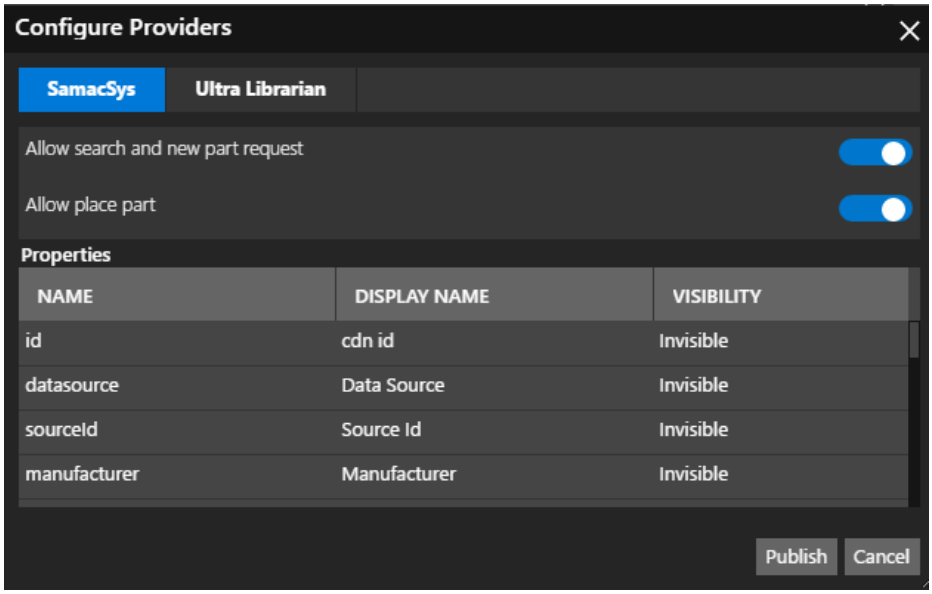


Note: The *Upload Configuration* and *Download Configuration* options are only available to designers whose design and library data is on the same machine. These designers do not connect to a central Pulse server.

2. Select *Configure Providers*.

? 2023
All Rights Reserved.

The Configure Providers dialog box opens.

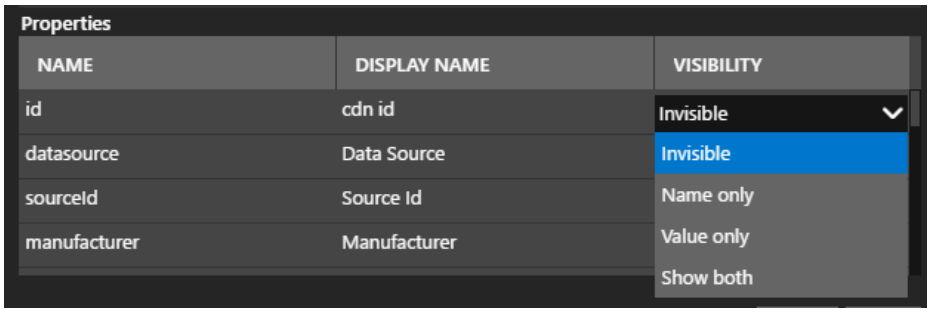


3. Toggle off the *Allow search part and new part request* button to make parts from a provider non-searchable.

   Note: Disabling new part requests for a provider is only applicable in the multi-user environment and can only be done by Pulse administrators.

4. Toggle off the *Allow place part* button to disable placement of parts from a provider on the Allegro System Capture canvas.

5. In the *VISIBILITY* column, define which properties should be visible on the canvas for a placed part in Allegro System Capture.



6. Click *Publish*.

   The new settings are published for the selected content provider.

All Rights Reserved.

Note: To reflect the modified settings, restart System Capture.

? 2023 All Rights Reserved.

? 2023 All Rights Reserved.

1

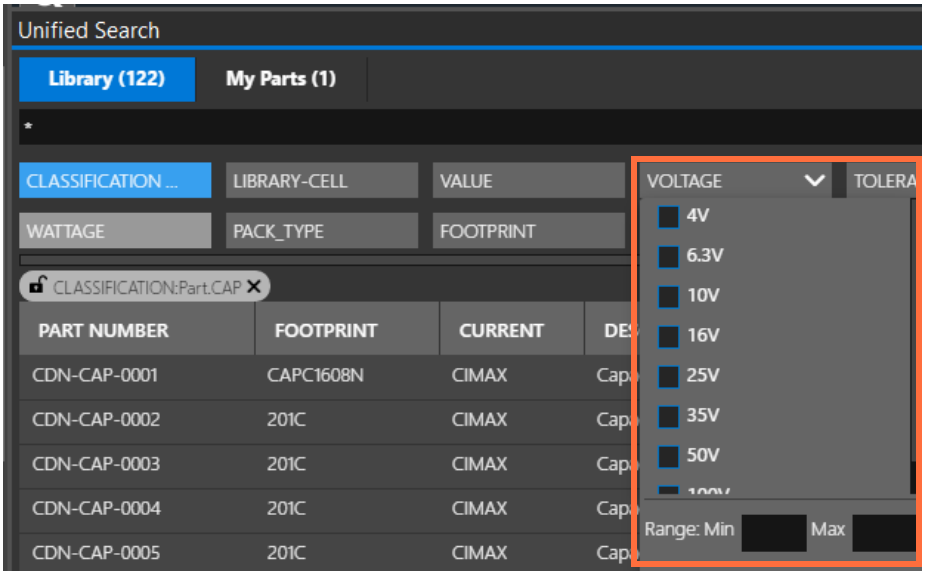
Appendix

This section provides information in a miscellaneous collection of topics.

- How Unified Search Uses Queries
- Advanced Search Query Syntax
- OrCAD Capture Libraries in System Capture
- Adding Project-Specific Libraries to a Project
- Troubleshooting

How Unified Search Uses Queries

Except *CLASSIFICATION* and *LIBRARY-CELL*, every filter has a multi-selection drop-down list from which you can select values. Numeric properties also have input fields to specify a range as illustrated in the following image:

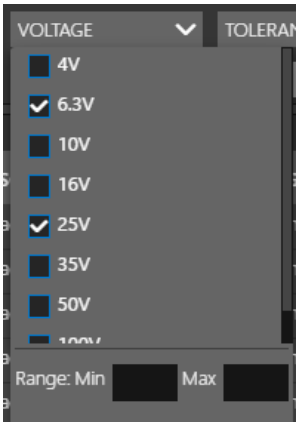


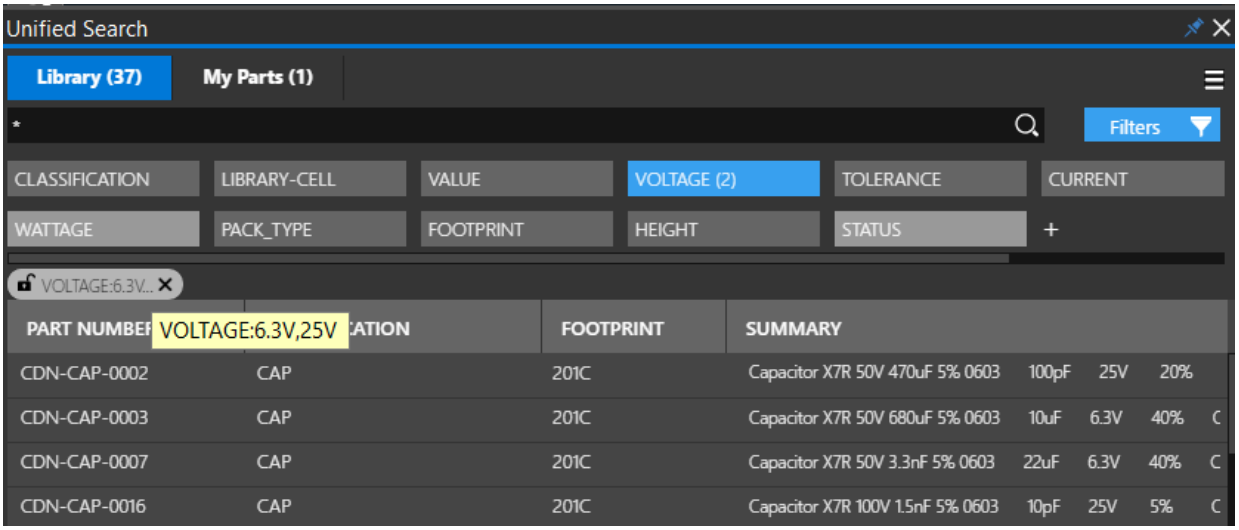
All Rights Reserved.

Supported Boolean operators are `AND`, `OR`, and `NOT`. When no operator is specified between query strings, the default Boolean operator is `OR`.

OR Operator

If you select more than one value for a filter, the selection or combination works with an `OR` Boolean operator. For example, in the following image, two values are selected.



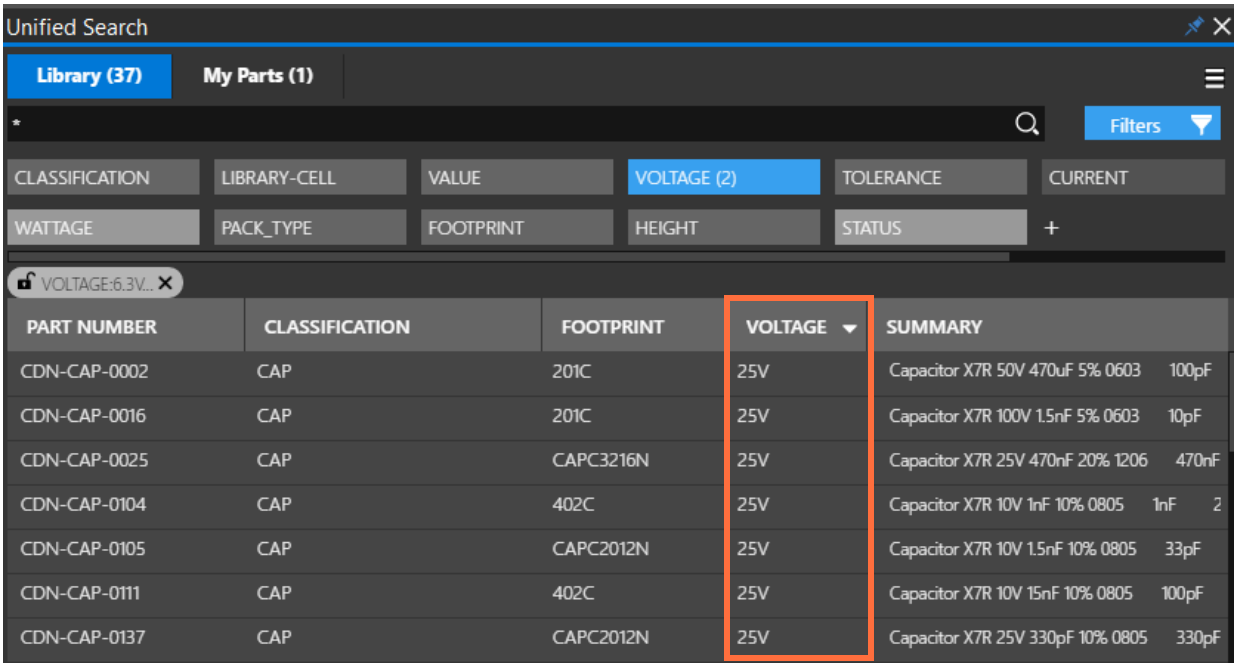


Observe the following:

- A chiplet, `VOLTAGE`, is added to the filter and the tool tip shows the selected values to which the `OR` operator is applied.
- The number of search results narrows down.
- The `VOLTAGE` filter shows the number of values selected for it, which is `2`.

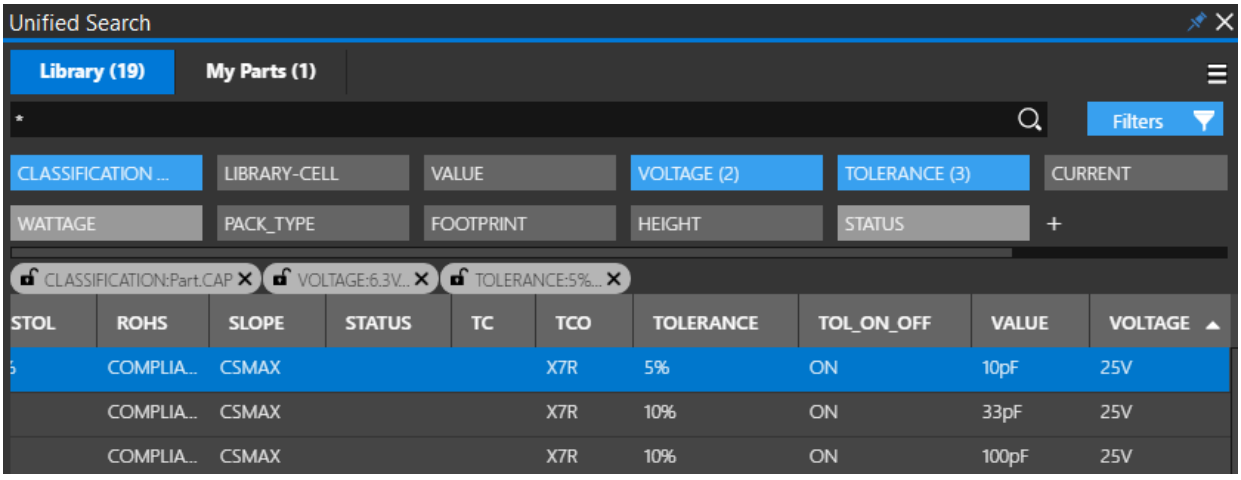
? 2023
All Rights Reserved.

You can sort the filtered results in ascending or descending order.



AND Operation

The AND operator is applied if you apply more than one chiplet to a filter. The AND operator is applied between different chiplets, not within a chiplet. Within a chiplet, OR is applied.



Observe that:

- The number of search results narrows down further the more chiplets you apply.

? 2023
All Rights Reserved.

- The *CLASSIFICATION*, *VOLTAGE*, and *TOLERANCE* filters are highlighted and show the number of values selected for these filters, which are `2` and `3`.

  Note: Because only one value can be selected for *CLASSIFICATION*, the number is not displayed.

Advanced Search Query Syntax

Advanced search means that you can search for parts using the `attribute:value` in the search text box of the *Library* tab in *Unified Search*. The queries are specified using attribute-specific name-value pairs.

- Syntax for creating queries:

  <attribute>:<value>

  Example

  `voltage:10V`

- Range symbols that can be used:

  `<, <=, >, >=`

- Boolean operators that can be used:

  `AND, OR, NOT`

  Note: The default Boolean operator is `OR` when no operator is specified between query strings.

  Example

  `capacitor AND voltage:>15V AND tolerance:>5%`

- For an exact match:

  Use the attribute value in quotation marks.

Note: Regular expressions are supported in advanced search queries but must be first enabled by an administrator. Wild card support can also be enabled by the administrator. However, by default, free text search performs a *text* search. As a result, many common searches are performed without the use of `*` or `?`. If you are a Pulse administrator, refer to Enabling Wild Card or Regular Expressions in Part Search.

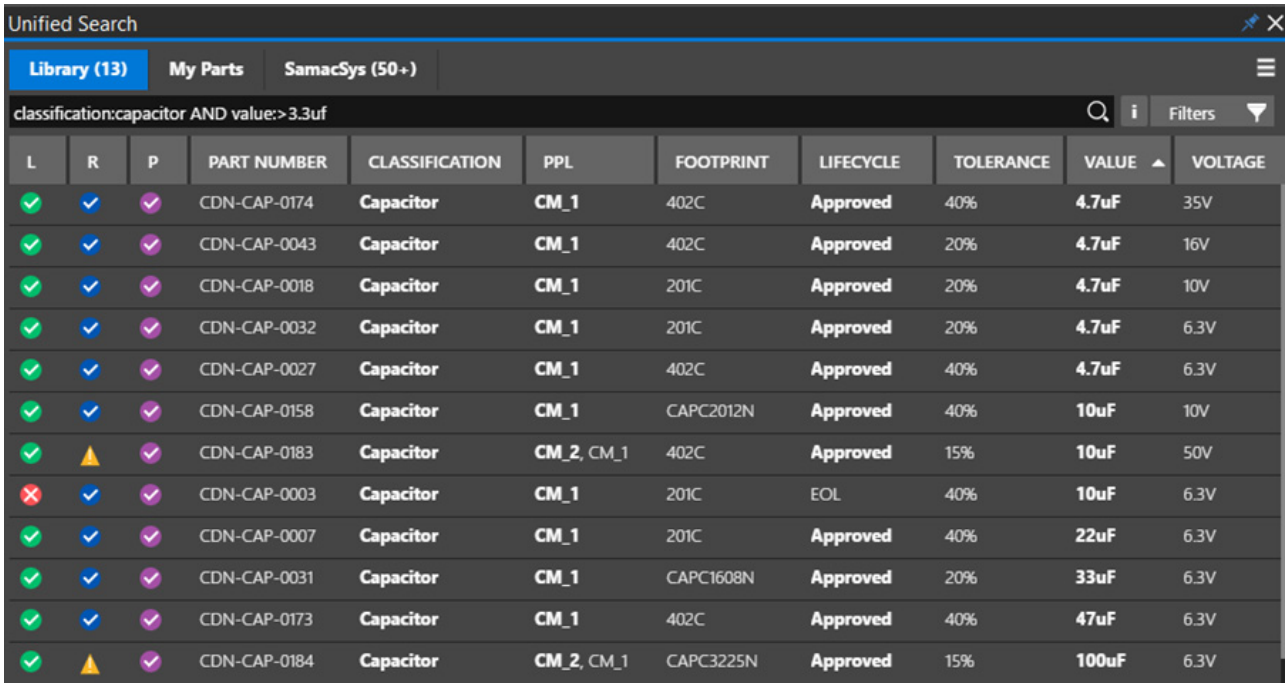
? 2023 All Rights Reserved.

Examples of Queries and Search Results

Tip

If you want to search for a string that has special characters, enclose them in double-quotes. For example, "`BODY/TYPE`".

- **`classification:capacitor AND value:>3.3uf`**

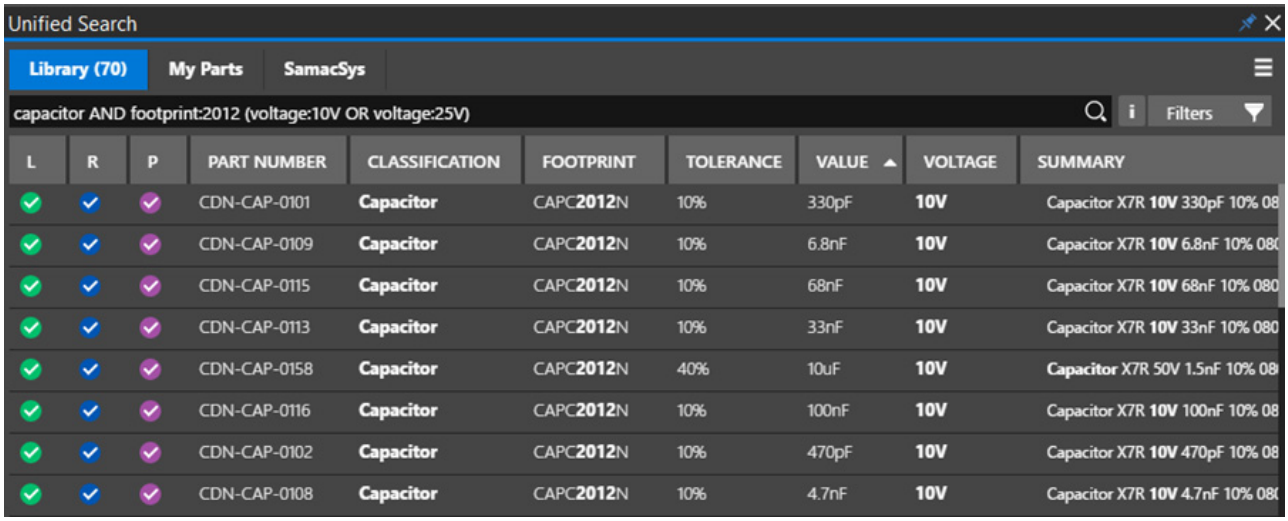


L	R	P	PART NUMBER	CLASSIFICATION	PPL	FOOTPRINT	LIFECYCLE	TOLERANCE	VALUE	VOLTAGE
			CDN-CAP-0174	Capacitor	CM_1	402C	Approved	40%	4.7uF	35V
			CDN-CAP-0043	Capacitor	CM_1	402C	Approved	20%	4.7uF	16V
			CDN-CAP-0018	Capacitor	CM_1	201C	Approved	20%	4.7uF	10V
			CDN-CAP-0032	Capacitor	CM_1	201C	Approved	20%	4.7uF	6.3V
			CDN-CAP-0027	Capacitor	CM_1	402C	Approved	40%	4.7uF	6.3V
			CDN-CAP-0158	Capacitor	CM_1	CAPC2012N	Approved	40%	10uF	10V
			CDN-CAP-0183	Capacitor	CM_2, CM_1	402C	Approved	15%	10uF	50V
			CDN-CAP-0003	Capacitor	CM_1	201C	EOL	40%	10uF	6.3V
			CDN-CAP-0007	Capacitor	CM_1	201C	Approved	40%	22uF	6.3V
			CDN-CAP-0031	Capacitor	CM_1	CAPC1608N	Approved	20%	33uF	6.3V
			CDN-CAP-0173	Capacitor	CM_1	402C	Approved	40%	47uF	6.3V
			CDN-CAP-0184	Capacitor	CM_2, CM_1	CAPC3225N	Approved	15%	100uF	6.3V

- **`capacitor AND footprint:2012 (voltage:10V OR voltage:25V)`**

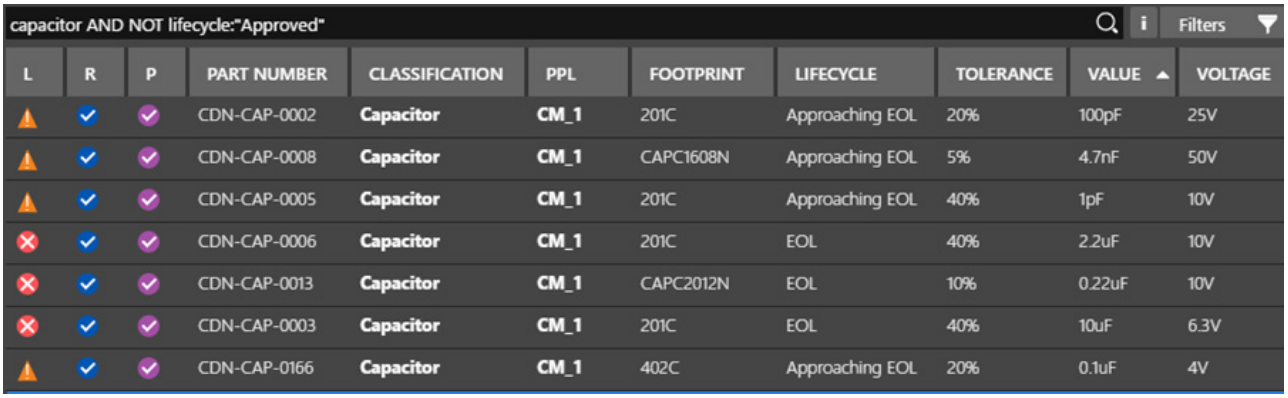
? 2023
All Rights Reserved.

```
capacitor AND footprint:2012 (voltage:10V voltage:25V)
```



L	R	P	PART NUMBER	CLASSIFICATION	FOOTPRINT	TOLERANCE	VALUE	VOLTAGE	SUMMARY
			CDN-CAP-0101	**Capacitor**	CAPC**2012**N	10%	330pF	**10V**	Capacitor X7R **10V** 330pF 10% 08
			CDN-CAP-0109	**Capacitor**	CAPC**2012**N	10%	6.8nF	**10V**	Capacitor X7R **10V** 6.8nF 10% 08(
			CDN-CAP-0115	**Capacitor**	CAPC**2012**N	10%	68nF	**10V**	Capacitor X7R **10V** 68nF 10% 080
			CDN-CAP-0113	**Capacitor**	CAPC**2012**N	10%	33nF	**10V**	Capacitor X7R **10V** 33nF 10% 080
			CDN-CAP-0158	**Capacitor**	CAPC**2012**N	40%	10uF	**10V**	**Capacitor** X7R 50V 1.5nF 10% 08
			CDN-CAP-0116	**Capacitor**	CAPC**2012**N	10%	100nF	**10V**	Capacitor X7R **10V** 100nF 10% 08
			CDN-CAP-0102	**Capacitor**	CAPC**2012**N	10%	470pF	**10V**	Capacitor X7R **10V** 470pF 10% 08
			CDN-CAP-0108	**Capacitor**	CAPC**2012**N	10%	4.7nF	**10V**	Capacitor X7R **10V** 4.7nF 10% 08(

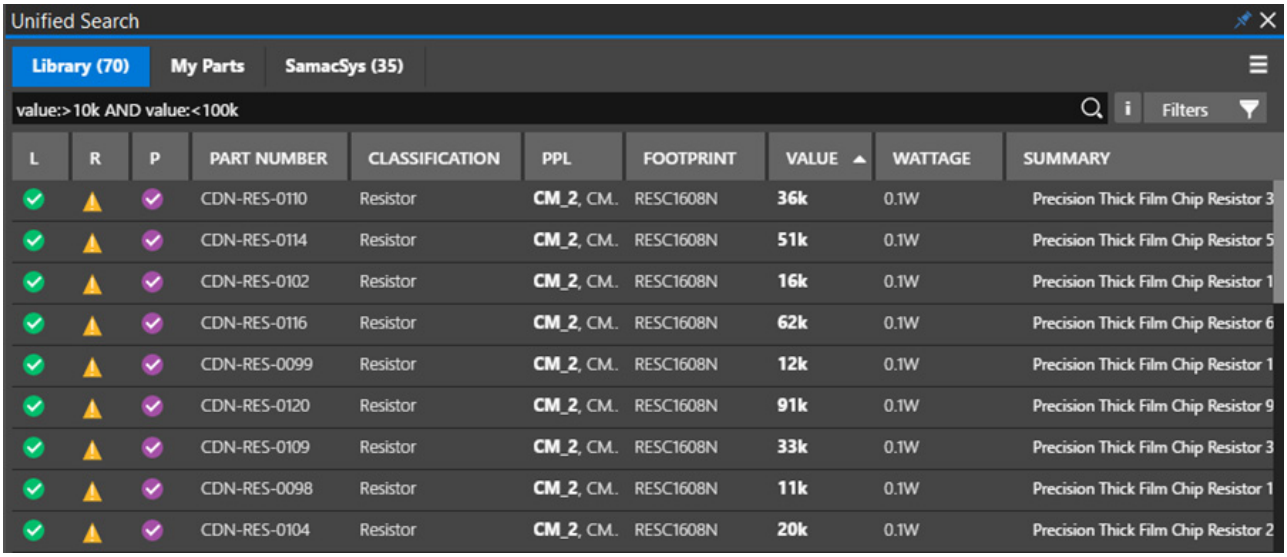
- **capacitor AND NOT lifecycle:"Approved"**



L	R	P	PART NUMBER	CLASSIFICATION	PPL	FOOTPRINT	LIFECYCLE	TOLERANCE	VALUE	VOLTAGE
			CDN-CAP-0002	**Capacitor**	**CM_1**	201C	Approaching EOL	20%	100pF	25V
			CDN-CAP-0008	**Capacitor**	**CM_1**	CAPC1608N	Approaching EOL	5%	4.7nF	50V
			CDN-CAP-0005	**Capacitor**	**CM_1**	201C	Approaching EOL	40%	1pF	10V
			CDN-CAP-0006	**Capacitor**	**CM_1**	201C	EOL	40%	2.2uF	10V
			CDN-CAP-0013	**Capacitor**	**CM_1**	CAPC2012N	EOL	10%	0.22uF	10V
			CDN-CAP-0003	**Capacitor**	**CM_1**	201C	EOL	40%	10uF	6.3V
			CDN-CAP-0166	**Capacitor**	**CM_1**	402C	Approaching EOL	20%	0.1uF	4V

? 2023
All Rights Reserved.

- **`value:>10k AND value:<100k`**



L	R	P	PART NUMBER	CLASSIFICATION	PPL	FOOTPRINT	VALUE	WATTAGE	SUMMARY
			CDN-RES-0110	Resistor	CM_2, CM.	RESC1608N	36k	0.1W	Precision Thick Film Chip Resistor 3
			CDN-RES-0114	Resistor	CM_2, CM.	RESC1608N	51k	0.1W	Precision Thick Film Chip Resistor 5
			CDN-RES-0102	Resistor	CM_2, CM.	RESC1608N	16k	0.1W	Precision Thick Film Chip Resistor 1
			CDN-RES-0116	Resistor	CM_2, CM.	RESC1608N	62k	0.1W	Precision Thick Film Chip Resistor 6
			CDN-RES-0099	Resistor	CM_2, CM.	RESC1608N	12k	0.1W	Precision Thick Film Chip Resistor 1
			CDN-RES-0120	Resistor	CM_2, CM.	RESC1608N	91k	0.1W	Precision Thick Film Chip Resistor 9
			CDN-RES-0109	Resistor	CM_2, CM.	RESC1608N	33k	0.1W	Precision Thick Film Chip Resistor 3
			CDN-RES-0098	Resistor	CM_2, CM.	RESC1608N	11k	0.1W	Precision Thick Film Chip Resistor 1
			CDN-RES-0104	Resistor	CM_2, CM.	RESC1608N	20k	0.1W	Precision Thick Film Chip Resistor 2

OrCAD Capture Libraries in System Capture

OrCAD Capture libraries (`.olb` files) and the OrCAD CIS database can be configured in System Capture in two ways:

- Capture.ini in the `HOME` directory
- Capture.ini at `CDS_SITE`

  Ensure that `CDS_SITE` has:

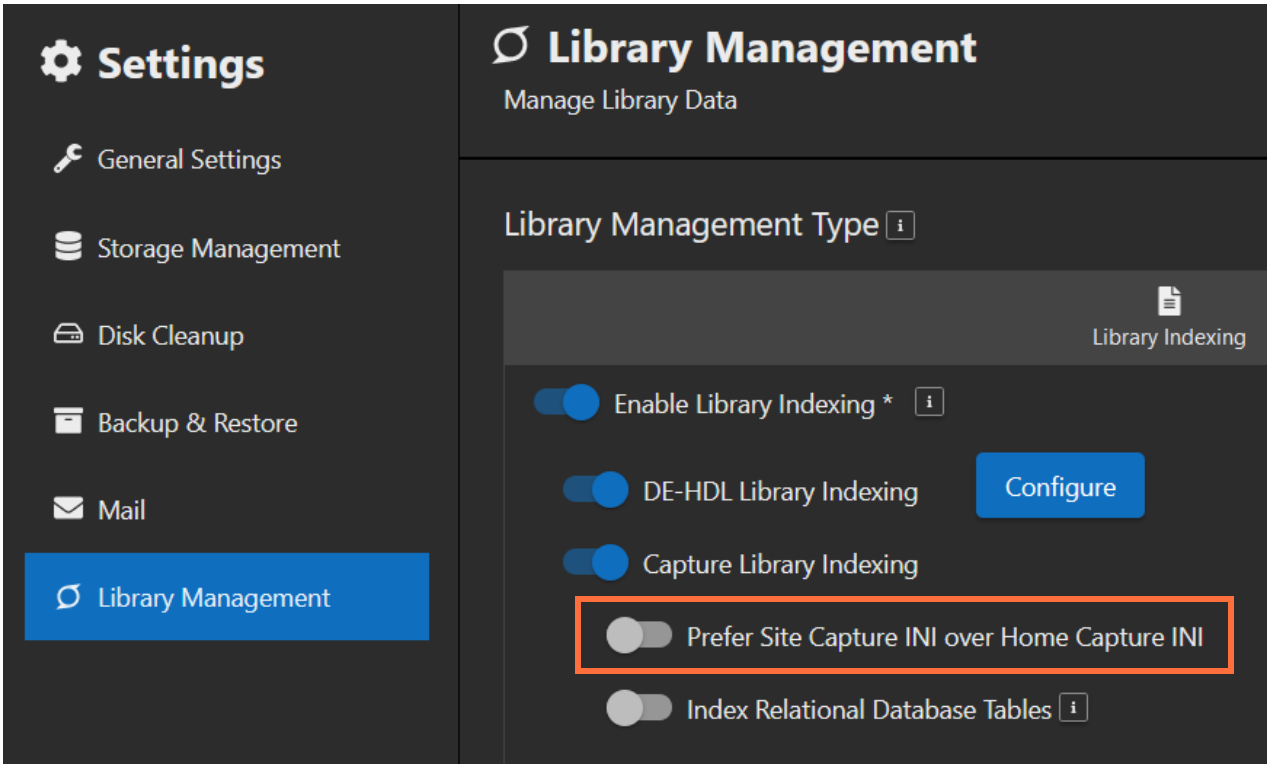
  - `$CDS_SITE/cdssetup/OrCAD_Capture/23.1.0/Capture.ini`
  - `$CDS_SITE/cdssetup/projmgr/site.cpm`

Depending on your requirement, do either of the following:

- Delete `Capture.ini` from the HOME directory

? 2023
All Rights Reserved.

- Enable the highlighted field in the Pulse Manager page:



Adding Project-Specific Libraries to a Project

The *My Parts* pane in *Unified Search* shows parts from project-specific libraries. This pane also displays parts placed from external part providers. This is because once placed in a design, an external part is cached in your local library and the library is automatically added to your design project.

To add a library to a project, do the following:

1. Select *Edit – Preferences* in System Capture.
2. Click *Available Libraries* to browse and select the libraries you want to add to the design.

Troubleshooting

- Cannot log in to search for parts from SamacSys or Ultra Librarian

? 2023
All Rights Reserved.

In certain countries, access to `pcb.cadence.com` might be geo-blocked because of copyright and licensing agreements with Cadence.

Currently, access to this site (`pcb.cadence.com`) is restricted for the following countries:

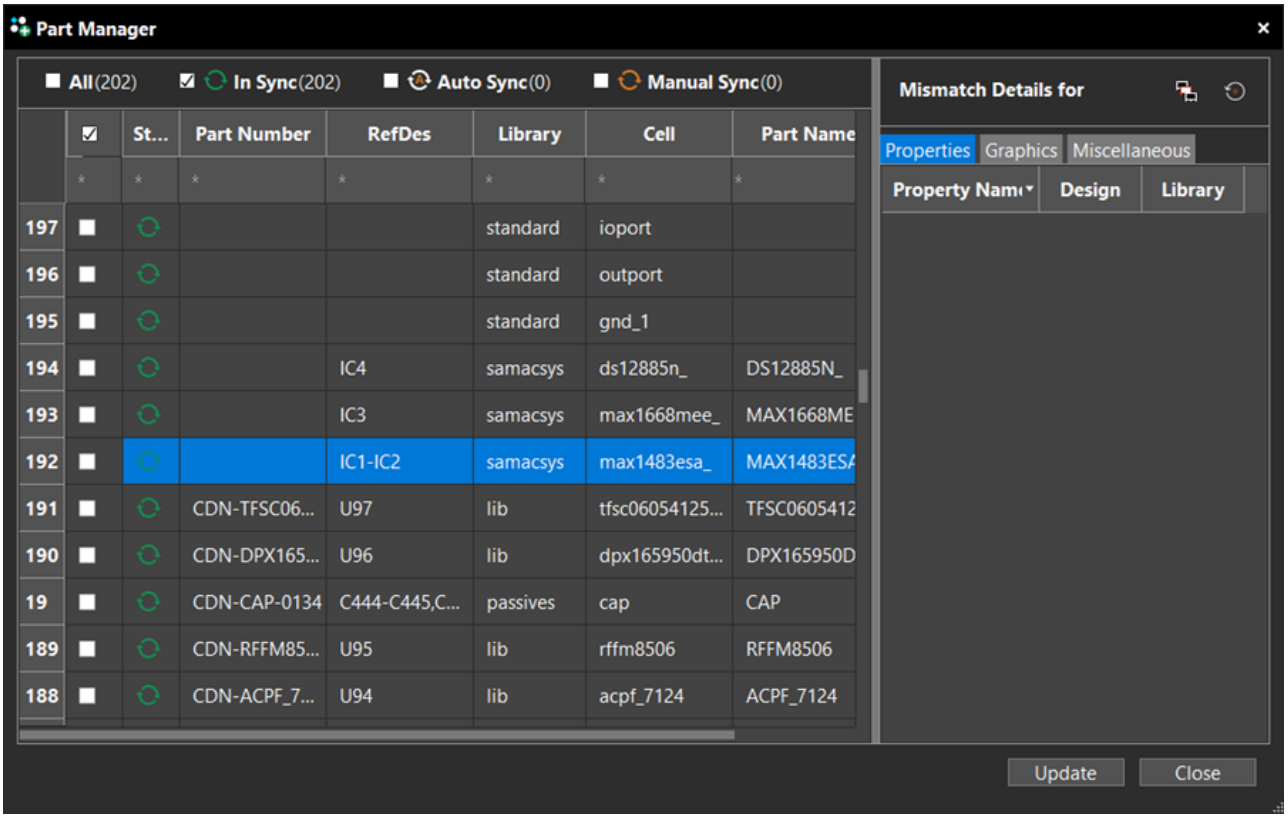
- Cuba
- Iran
- North Korea
- Sudan
- Syria

- The *Properties* tab in Part Manager is empty for external vendor-provided parts.

When parts from external part providers are placed in a design, they are cached in your local library and the library is added to your design project. However, in a team design environment, if you share such a design with others who do not have access to the local library that contains the external vendor-provided parts, these users see a blank *Properties* tab.

? 2023 All Rights Reserved.

This is because Part Manager cannot read external content provider data to which you do not have access to show the differences between it and the part in your design.



However, Part Manager displays the graphic and other properties for the part.

? 2023
All Rights Reserved.