cadence®

Allegro® X System Capture: What's New in Release 23.1

Product Version 23.1
September 2023

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

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

Allegro® Design Entry HDL 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.

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

Allegro® X System Capture: What's New in 23.1 5
Update Available Notification 7
Thermal Analysis and Celsius Integration 7
Topology Workbench Integration 8
New Analysis Modes Available in Design Integrity 9
MTBF (Mean Time Between Failure) Analysis 9
Power Topology Analysis 11
Adding Parts from External Content Providers to Libraries 11
Library Authoring Enhancements 13
Templates Introduced for Libraries 13
Creating Categories in a Library 13
Edit Multiple Parts in a Spreadsheet 14
Support for Custom Shapes and Pin Shapes 15
Enhanced Validation Checks Added 15
New Features in the Multi-User Environment 16
Managed Layouts in Project Explorer 17
Pulse Web Dashboard Updates 18
Pulse Status Icons 21
Comparing Versions 22
Working in the Offline Mode 24
Design Search and Reuse 26
AWR MWO-Allegro RF Design Flow 30
Variant Editor Enhancements 32
PDF/A Support Added 36
Updated Bill of Materials Dialog Box 37
Documentation Updates 39

© 2023 All Rights Reserved.

© 2023 All Rights Reserved.

Allegro® X System Capture: What's New in 23.1

This chapter contains the following sections:

- New features
  - Update Available Notification
  - Thermal Analysis and Celsius Integration
  - Topology Workbench Integration
  - New Analysis Modes Available in Design Integrity
    - MTBF (Mean Time Between Failure) Analysis
    - Power Topology Analysis
  - Adding Parts from External Content Providers to Libraries
  - Library Authoring Enhancements
    - Templates Introduced for Libraries
    - Creating Categories in a Library
    - Edit Multiple Parts in a Spreadsheet
    - Support for Custom Shapes and Pin Shapes
    - Enhanced Validation Checks Added
  - New Features in the Multi-User Environment
    - Managed Layouts in Project Explorer
    - Pulse Web Dashboard Updates
    - Pulse Status Icons
    - Comparing Versions
    - Working in the Offline Mode

© 2023 All Rights Reserved.

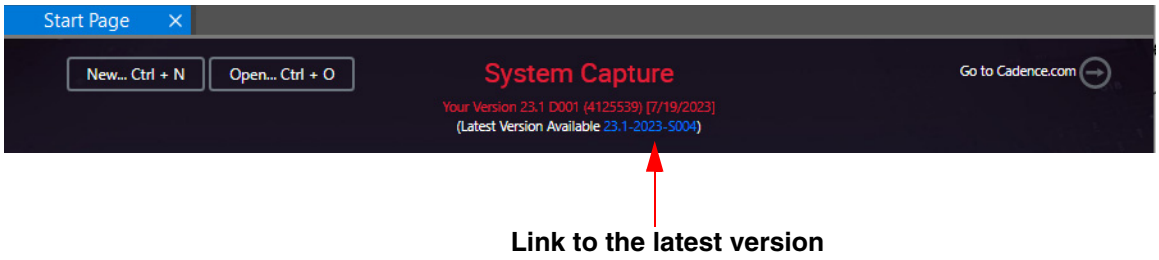
- Design Search and Reuse

- Changed features
  - AWR MWO-Allegro RF Design Flow on page 30
  - Variant Editor Enhancements on page 32
  - PDF/A Support Added on page 36
  - Updated Bill of Materials Dialog Box on page 37
- Documentation Updates on page 39

© 2023 All Rights Reserved.

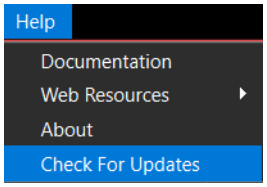
Update Available Notification

The *Start Page* shows the latest version available for System Capture. Click this link to download the newer version. If you are in an enterprise setup or do not have the required permissions, downloading and updating the installed version might require administrator support.



The version numbers in your installation might differ from those in the image.

A new *Help – Check for Updates* option is also added to check if a more recent update are available.



Thermal Analysis and Celsius Integration

Important

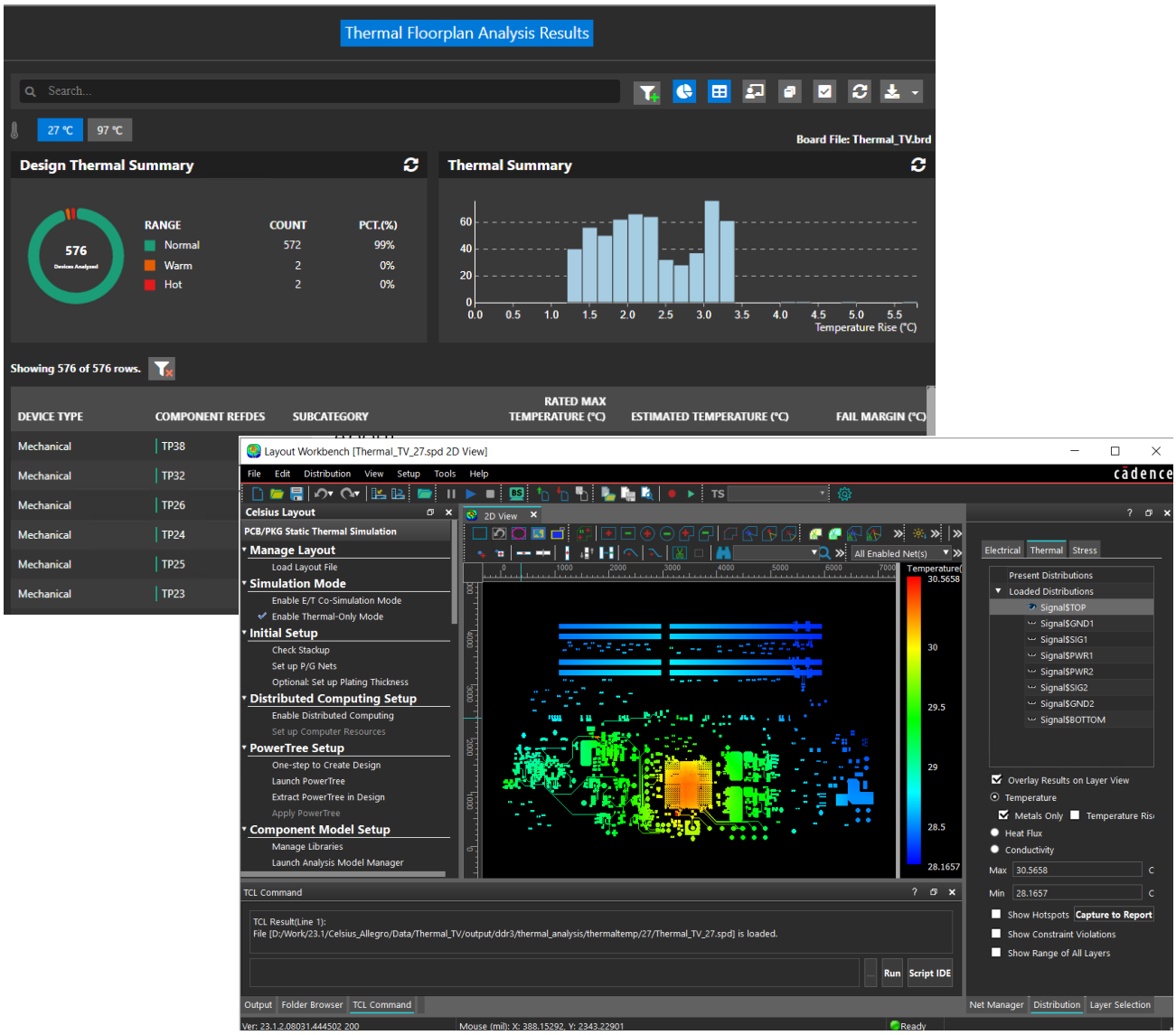
Requires a compatible Sigrity installation as listed in the *README.pdf* on *downloads.cadence.com*. System Capture 23.1 is compatible with Sigrity 23.1.

You can now estimate the life of a PCB design by generating a thermal floorplan for a board file in System Capture. This helps with improved placement of components based on the thermal map early in the design cycle. You can estimate the stress of schematic components based on the thermal map. You can launch Celsius Thermal Solver from the Electrical Analysis dashboard.

To access this functionality, create an environment variable called `SIGRITY_EDA_DIR` and

© 2023
All Rights Reserved.

set it to the *<path to Celsius executable file>*.



Related Documentation

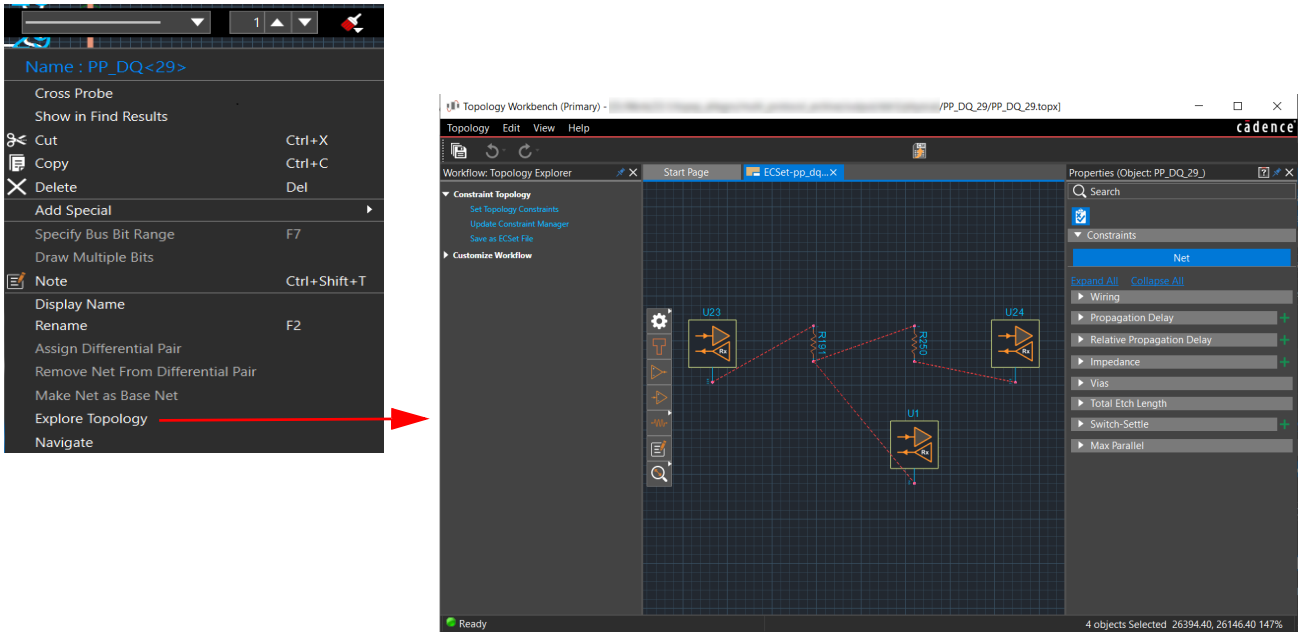
Thermal Analysis with Celsius Thermal Solver

Topology Workbench Integration

Topology Workbench has replaced Signal Explorer for analyzing the signal integrity of high-speed nets at the schematic, floorplan, and layout stages. All the Constraint Manager-based

© 2023
All Rights Reserved.

ECSet creation and modification flows and simulation flows, continue to be supported. You can launch Topology Workbench directly from System Capture Canvas for a single net, XNet, or from Constraint Manager.



Related Documentation

Extracting Topology in Topology Workbench

New Analysis Modes Available in Design Integrity

Design Integrity includes two new analyses:

- MTBF (Mean Time Between Failure) Analysis
- Power Topology Analysis

MTBF (Mean Time Between Failure) Analysis

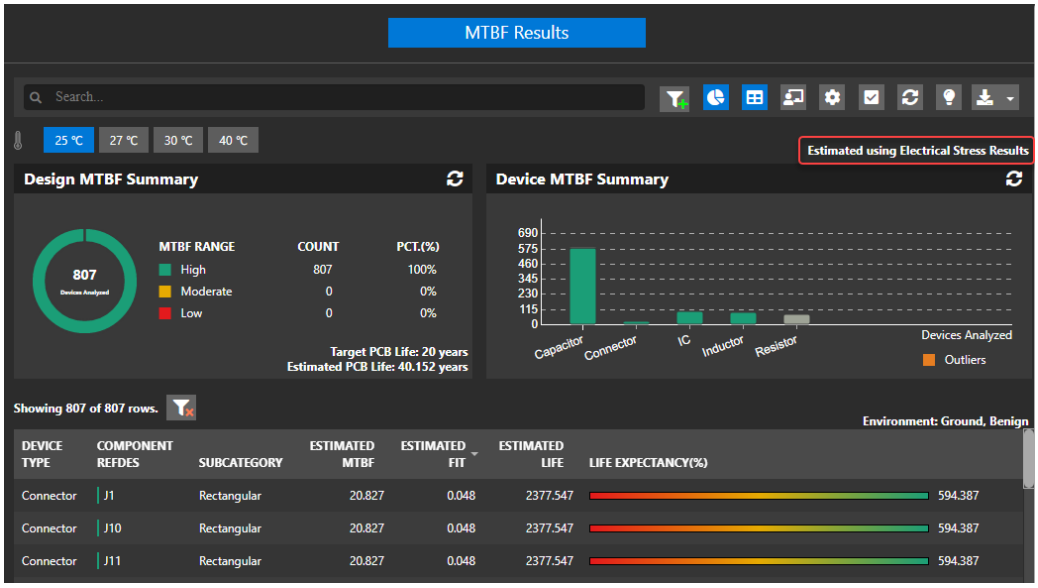
System Capture is enhanced to support MTBF analysis on a design to estimate the performance and safety of electrical, mechanical, and electro-mechanical parts. MTBF analysis is available with the following licenses:

- `Allegro X Designer`

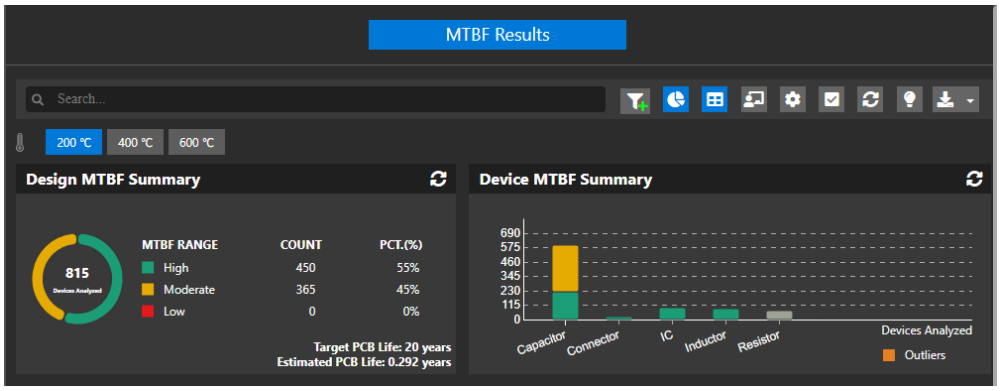
© 2023 All Rights Reserved.

- Allegro X Designer Plus
- Allegro X Venture

System Capture can calculate MTBF results with or without using the electrical stress data generated in the last run.



You can also configure the ambient temperatures used for MTBF analysis.



Related Documentation

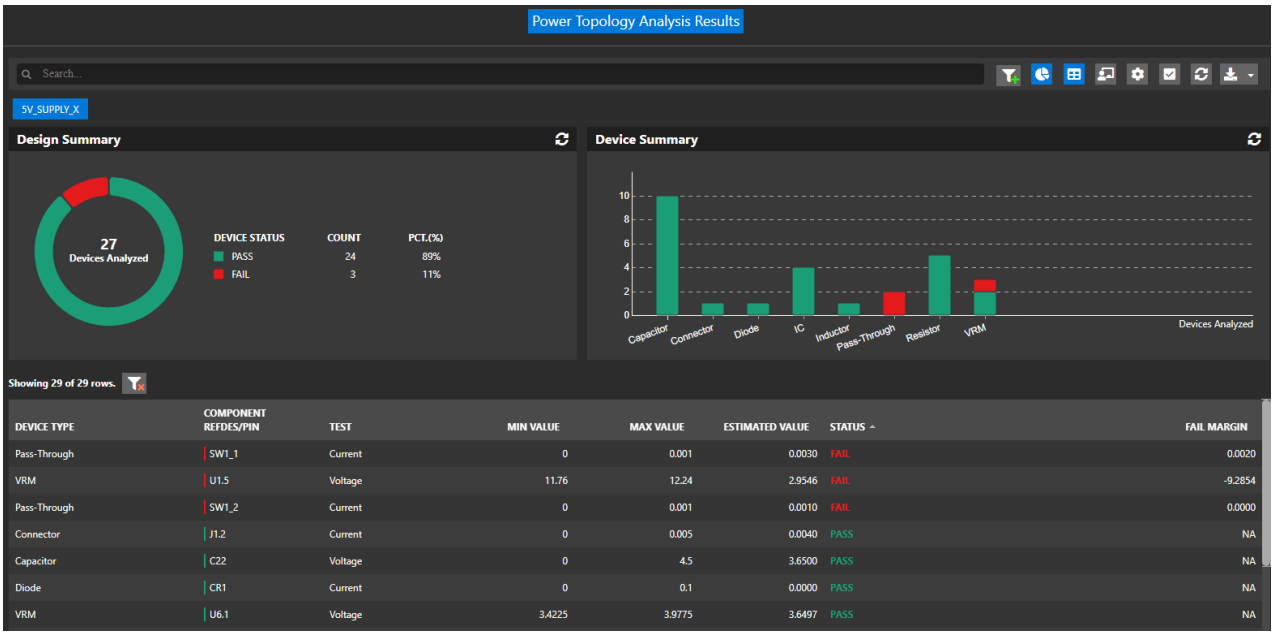
MTBF Analysis

© 2023
All Rights Reserved.

Power Topology Analysis

Power Topology analysis sets up a power distribution network on the schematic and estimates the DC power consumption by the PCB design components. This enables the verification of all the components in a design in the early stages of design development. Power Topology analysis is available with the following licenses:

- `Allegro X Designer`
- `Allegro X Designer Plus`
- `Allegro X Venture`



Related Documentation

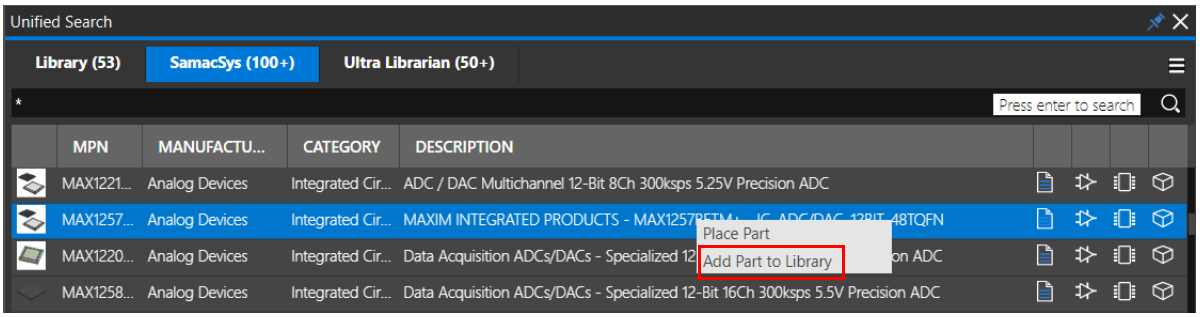
Power Topology Analysis

Adding Parts from External Content Providers to Libraries

Until the previous release, adding parts from external content providers, such as *SamacSys* or *Ultra Librarian*, downloaded parts to a provider-specific library and then placed them in

© 2023 All Rights Reserved.

designs. Now, you can add these parts to an existing library different from a provider-specific library, such as `local_part_lib`.



Related Documentation

Adding Parts from External Content Providers

© 2023
All Rights Reserved.

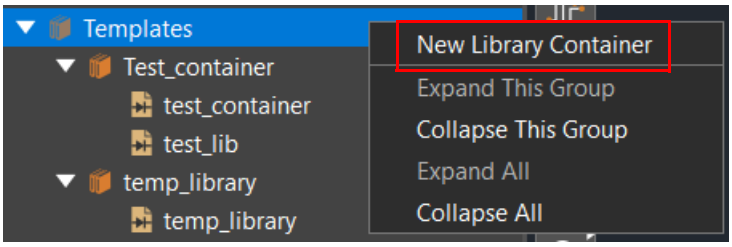
Library Authoring Enhancements

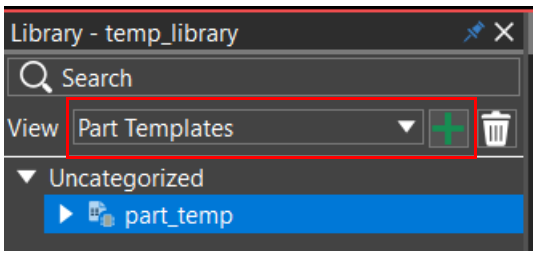
Allegro Unified library authoring includes the following new features:

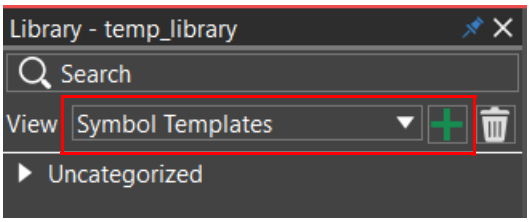
- Templates Introduced for Libraries
- Creating Categories in a Library
- Edit Multiple Parts in a Spreadsheet
- Support for Custom Shapes and Pin Shapes
- Enhanced Validation Checks Added

Templates Introduced for Libraries

You can create a new template library container under the *Templates* node, and create a new part or symbol template in this container. In addition to the default templates available for creating a part or a symbol, custom part and symbol templates are also available.





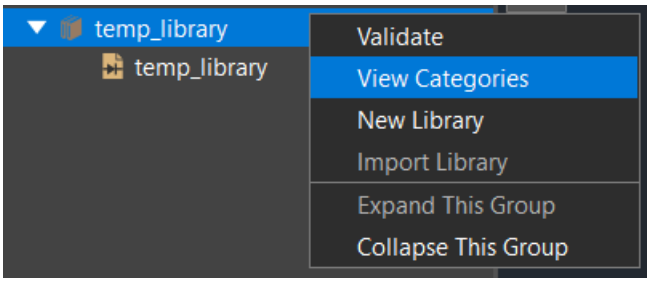


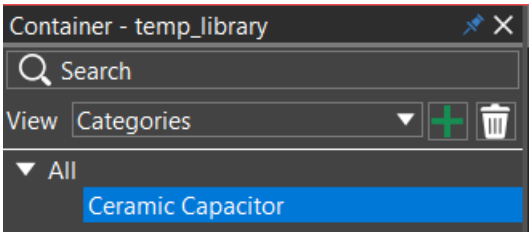
Creating Categories in a Library

New categories and subcategories in the template library improve component searching. For example, you can categorize a schematic model, resistor, as *Discrete*, which is further

© 2023
All Rights Reserved.

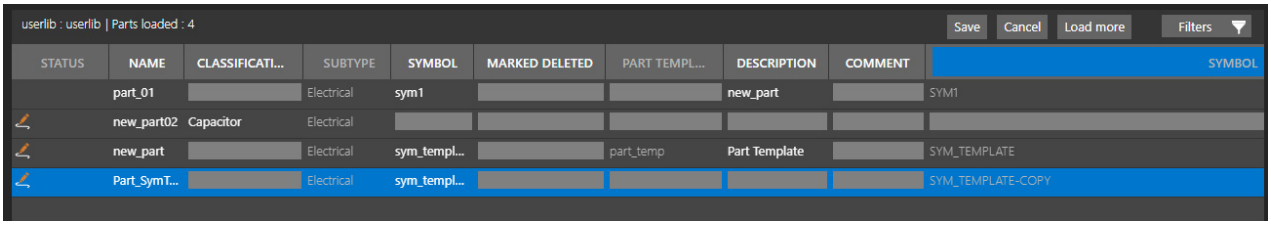
subclassified as *Resistor*. You can search for a keyword, *Discrete* or *Resistor*, and assign it to a part or a symbol.





Edit Multiple Parts in a Spreadsheet

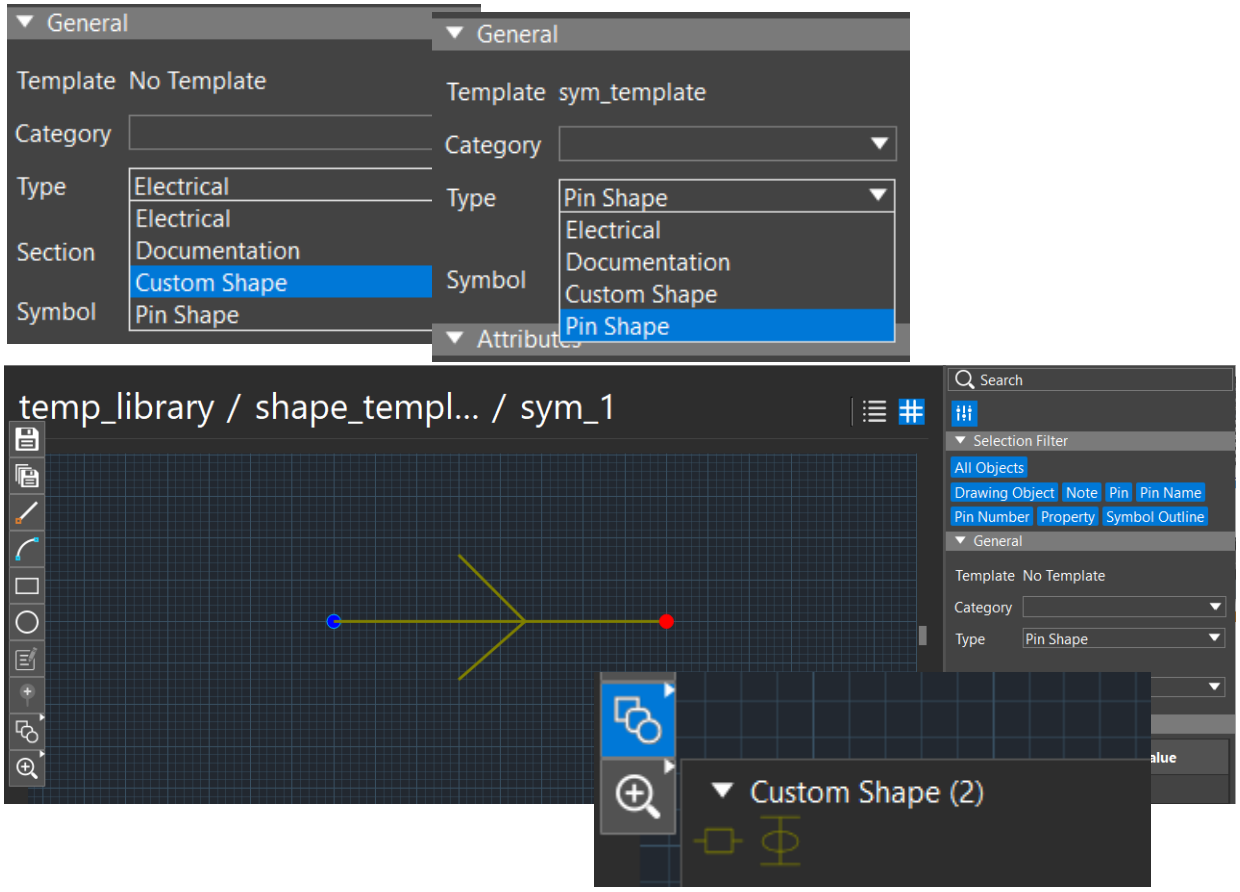
Modifying part names, descriptions, and properties is simpler in the spreadsheet view for part editing. In this spreadsheet view, you can edit multiple part properties, schematic models, footprints, simulation models, datasheets, and other columns.



© 2023
All Rights Reserved.

Support for Custom Shapes and Pin Shapes

You can create a new custom or pin shape under a template library and attach it to a pin or the complete symbol. This feature is available for projects based on Allegro Unified or DE-HDL libraries.

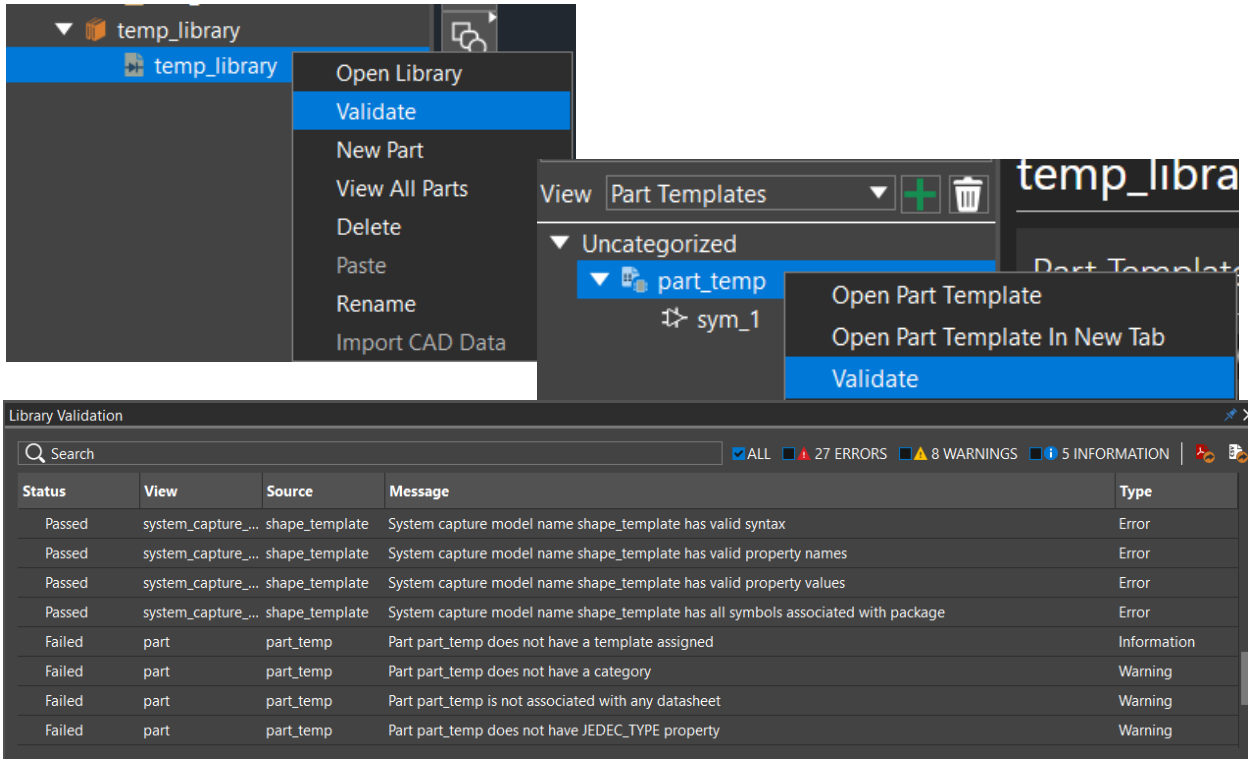


Enhanced Validation Checks Added

Validate libraries and data models, including parts, schematic models, footprints, and data sheets, to avoid common errors in a design with the *Library Validation* window. Libraries,

© 2023
All Rights Reserved.

parts, and schematic models are validated based on the rules defined in *Library Audit Settings*.



Related Documentation

Library Authoring and Editing in Allegro X System Capture

New Features in the Multi-User Environment

When connected to a remote Pulse server, the following additional features are available:

- Managed Layouts in Project Explorer
- Pulse Web Dashboard Updates
- Pulse Status Icons
- Comparing Versions
- Working in the Offline Mode

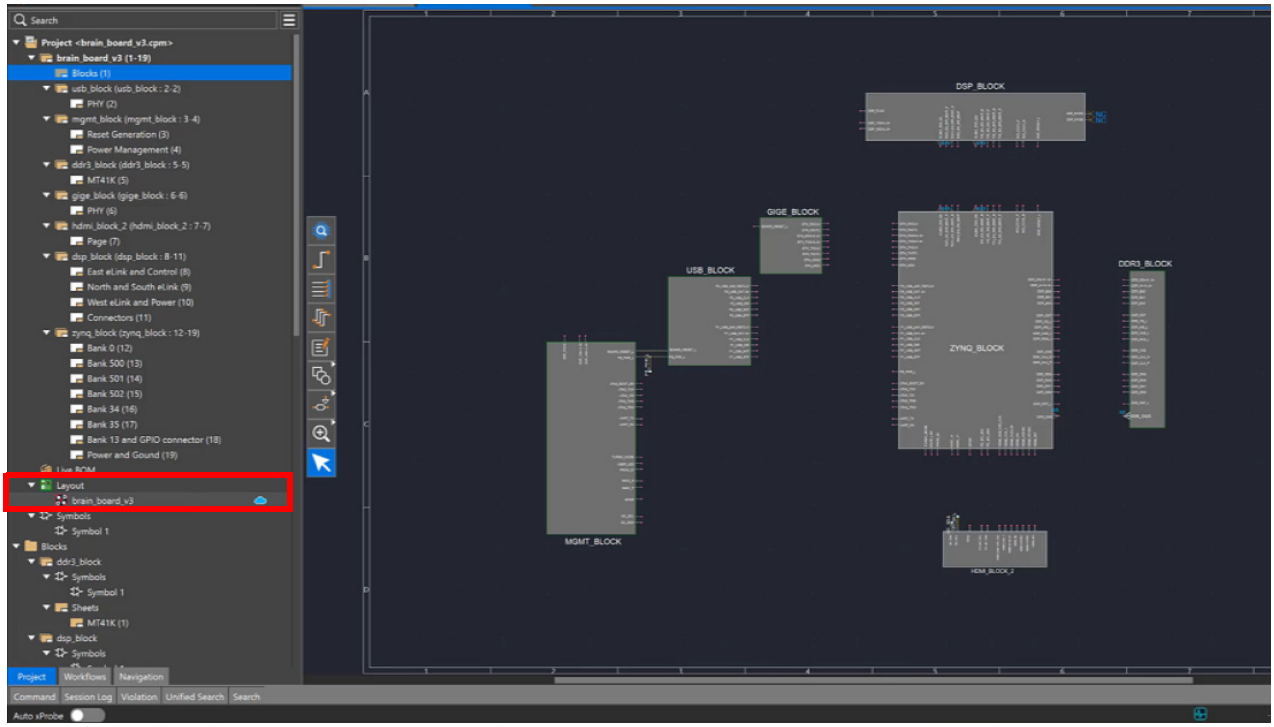
© 2023
All Rights Reserved.

- Design Search and Reuse

Managed Layouts in Project Explorer

System Capture users in a multi-user environment can stay updated with the latest changes in layouts associated with schematics.

Pulse-managed layouts include boards, packages, and modules. The layouts are displayed in *Project* explorer if you have read or edit permissions for the layout files.

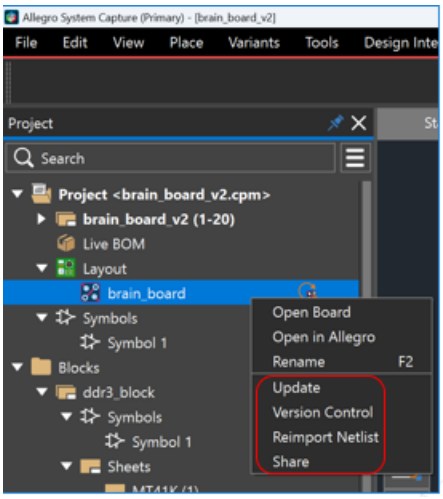


Tooltips displayed on hovering the mouse over layouts display the layout status, such as:

- Sync status with the associated schematic
- Whether a new revision is available
- Downloaded layouts that are removed from the design in the Pulse web dashboard

© 2023
All Rights Reserved.

Additional tasks are available from the context menu for Pulse-managed layouts, as shown in the following image.



Related Documentation

Design Data Management in Allegro X System Capture

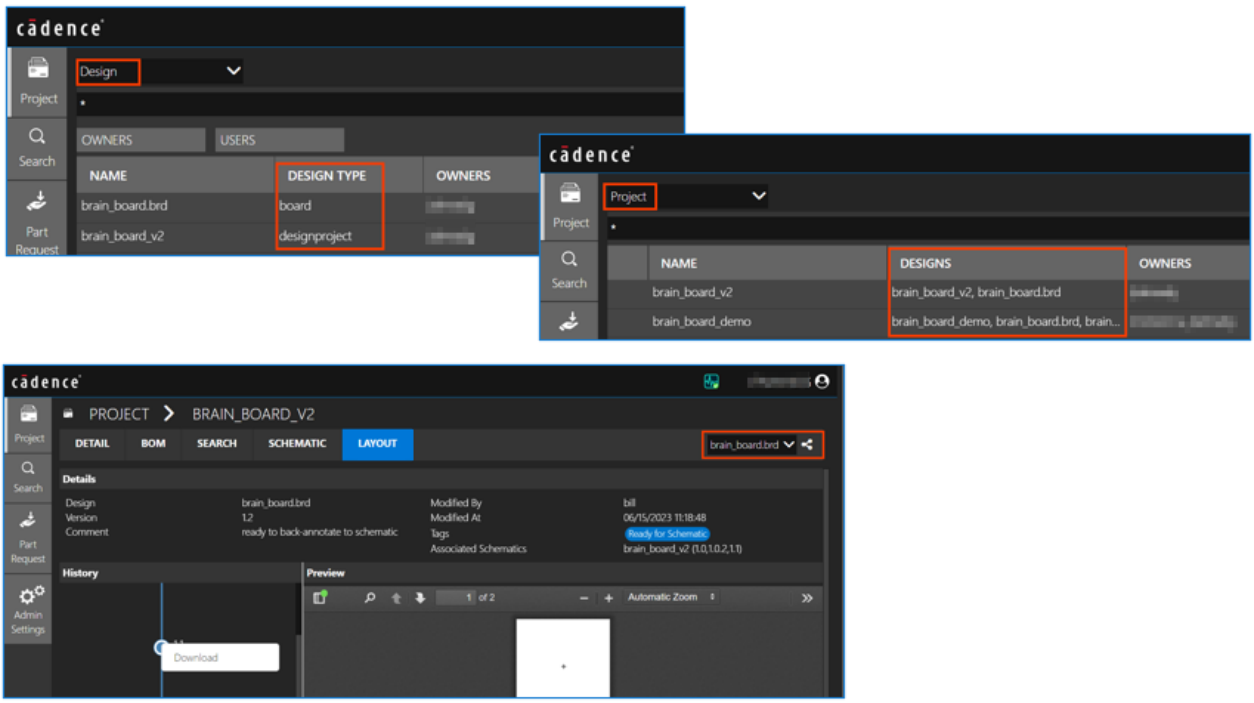
Pulse Web Dashboard Updates

Changes in the Pulse web dashboard are as follows:

- The *Part Search* tab on the left of the web page is renamed to *Search*.
- There is a *Project* and *Design* view. The *Project* view displays project containers and the designs contained within. The *Design* view shows all Pulse-managed designs, which

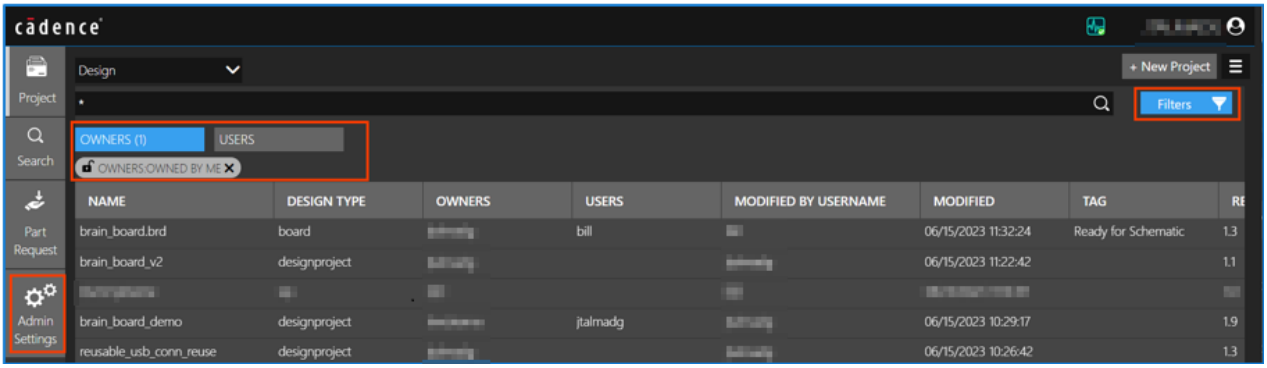
© 2023
All Rights Reserved.

includes schematics and layouts. This enables layout visibility in the *Layout* tab by double-clicking a project in the *Design* or *Project* list.



You can share a layout, view the layout versions, and download layout versions from *Version Control*.

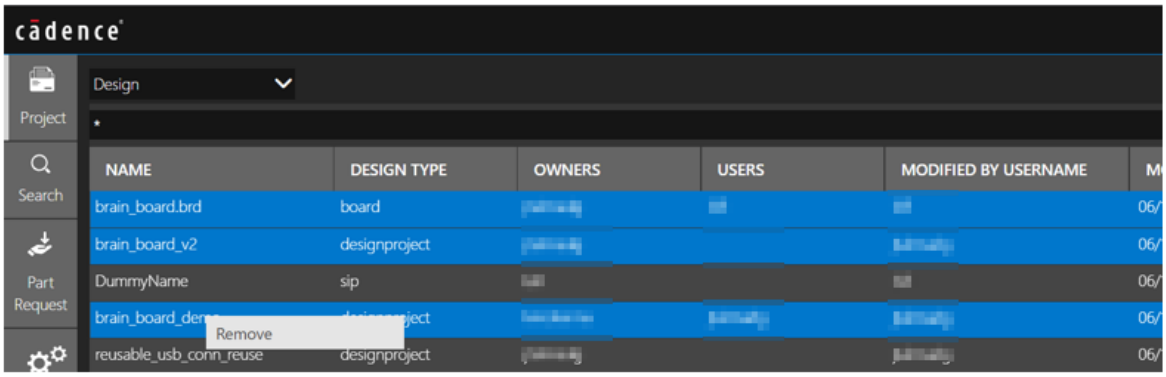
- The administrator tasks have moved from the user profile icon to the left side of the web page.



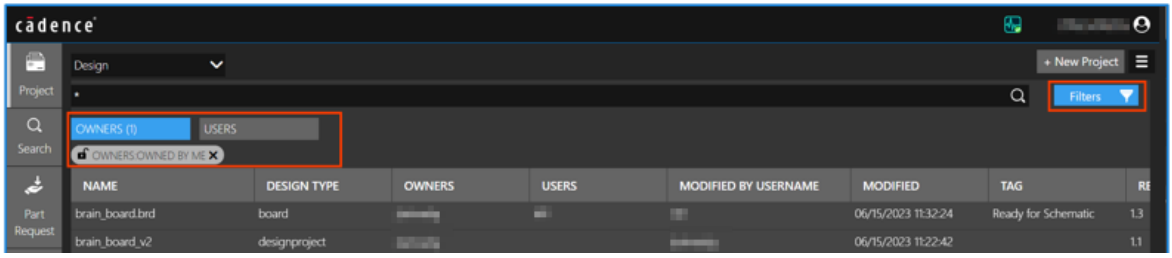
- Instead of clicking, you must now double-click to open a project or design from the list. Click selects a project or design instead of opening it.

© 2023
All Rights Reserved.

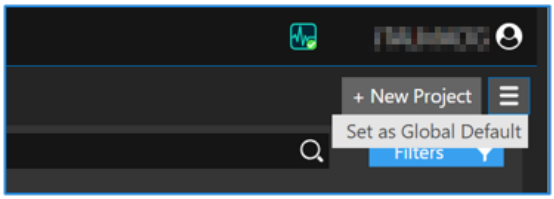
- Selecting and removing multiple entries simultaneously from the Design list is now supported.



- The *Owned by me* and *shared with me* filters enable easier viewing of projects and designs.

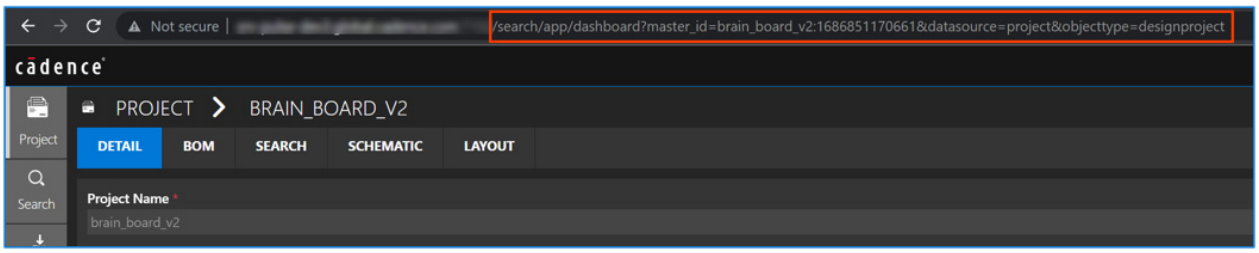


- Pulse administrators can save column visibility, the column order, and a filter for a project list as the global default.



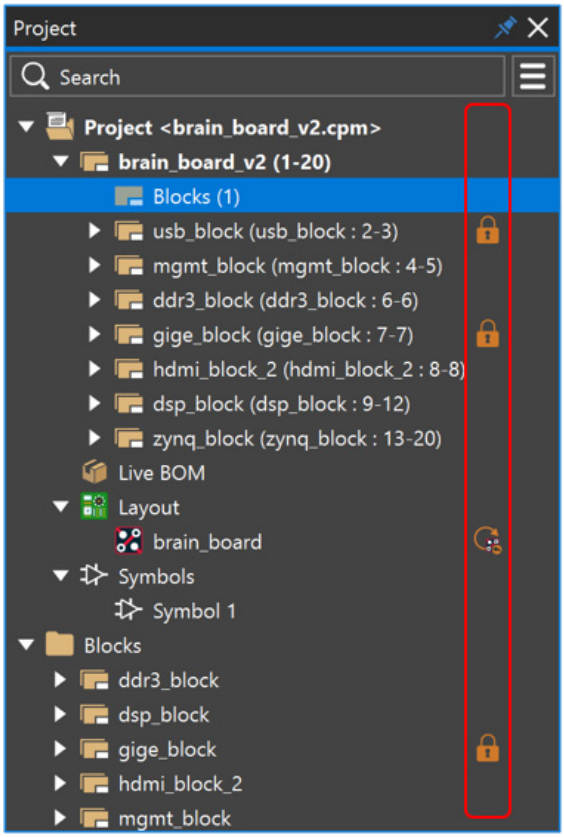
© 2023
All Rights Reserved.

- Project-specific URLs are now available for easy bookmarking. This simplifies opening a design or layout or sharing it.



Pulse Status Icons

The Pulse status icons, which provide notifications for team design, layout, and design reuse are now on the right side of the Project explorer pane. This enables easy viewing of the object type in the project.



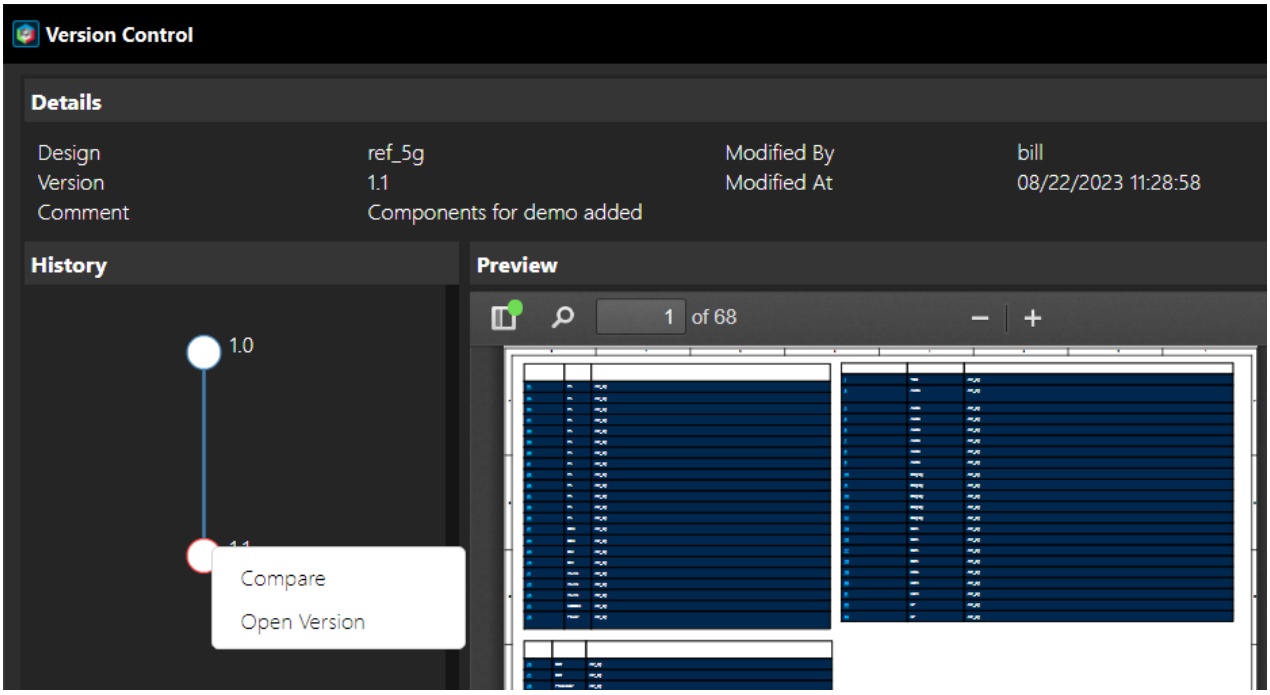
© 2023
All Rights Reserved.

Comparing Versions

To enable easier decision-making on whether to accept a team design update, you can now compare:

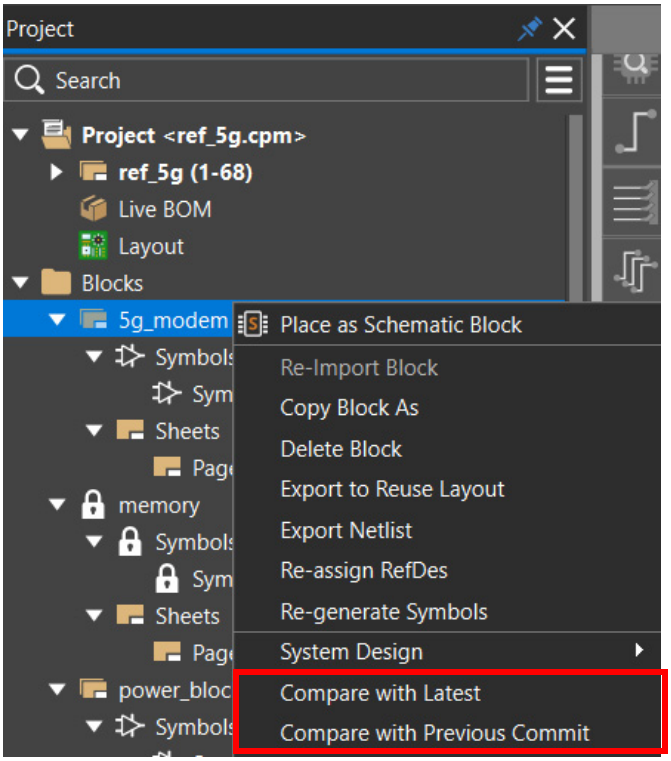
- Two design versions in *Version Control*.

  Right-click the first version and select *Compare* then click the second version.

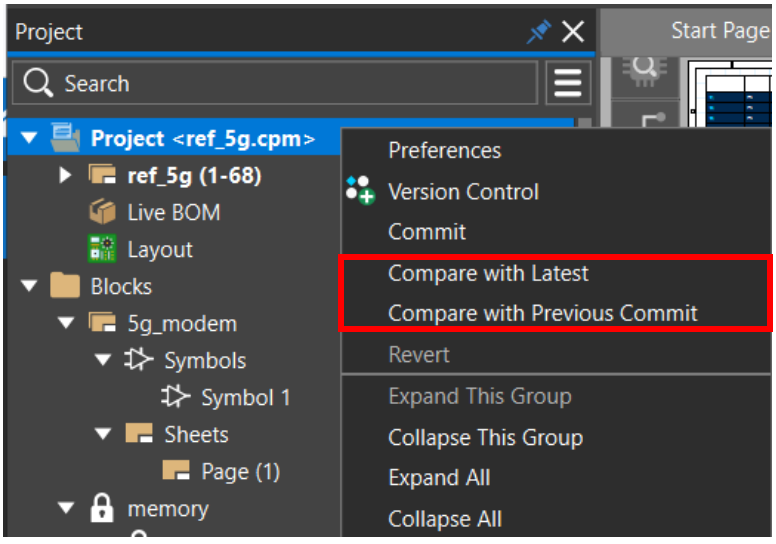


© 2023
All Rights Reserved.

- Individual blocks with the previous or latest commit versions.

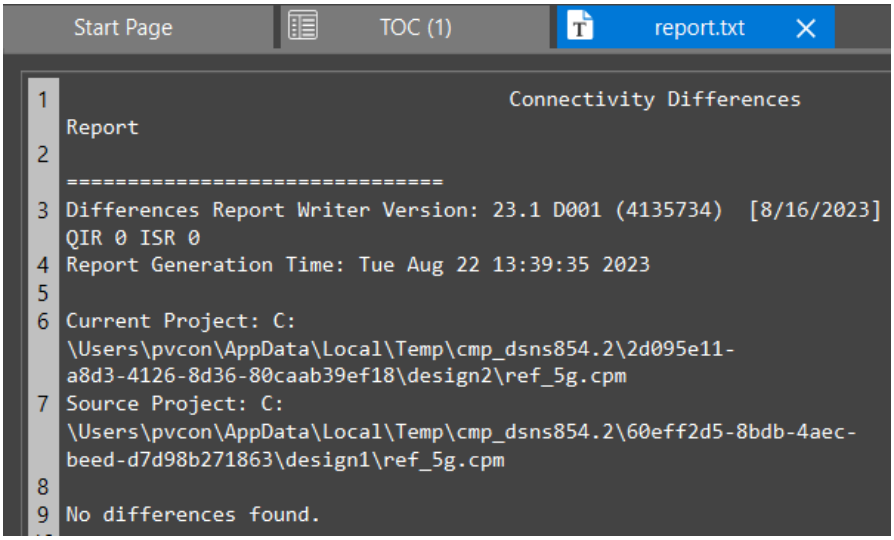


- The design with the latest version or any earlier version.



© 2023
All Rights Reserved.

The differences are displayed in a new tab.



Related Documentation

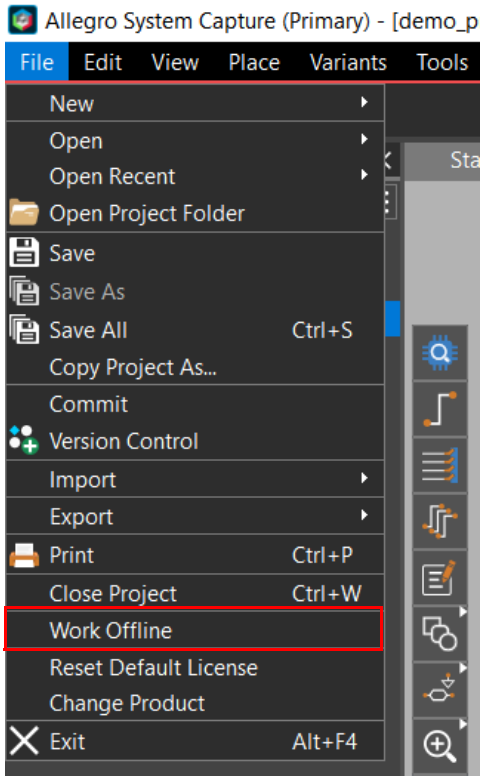
Comparing Designs

Working in the Offline Mode

If network connectivity is slow or the Pulse server is temporarily unavailable, users in a multi-user environment can disconnect from the Allegro Pulse server services and work in the

© 2023 All Rights Reserved.

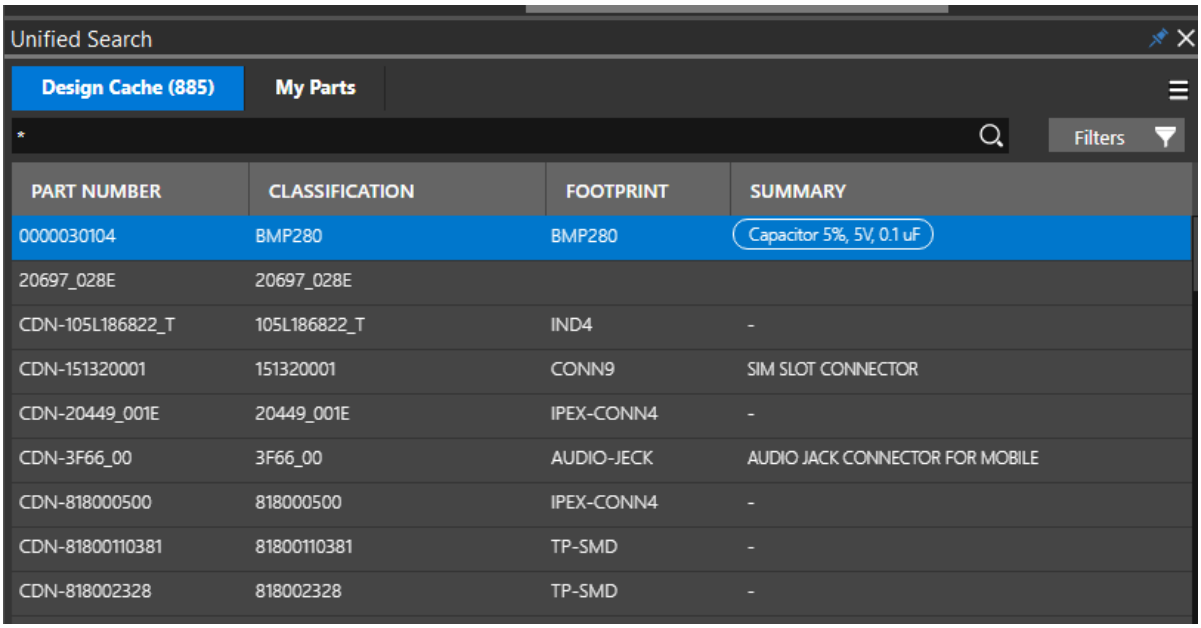
offline mode. This ensures uninterrupted work using parts in the design cache. You can connect to the server again, when available.



The offline mode is only supported for Allegro System Capture projects with Design Entry HDL or OrCAD Capture libraries.

© 2023
All Rights Reserved.

In the offline mode, you can only work on pages and blocks that you have explicitly locked before disconnecting from the Pulse server. Only parts cached in your local libraries are available in the offline mode in a *Design Cache* tab in Unified Search.



Related Documentation

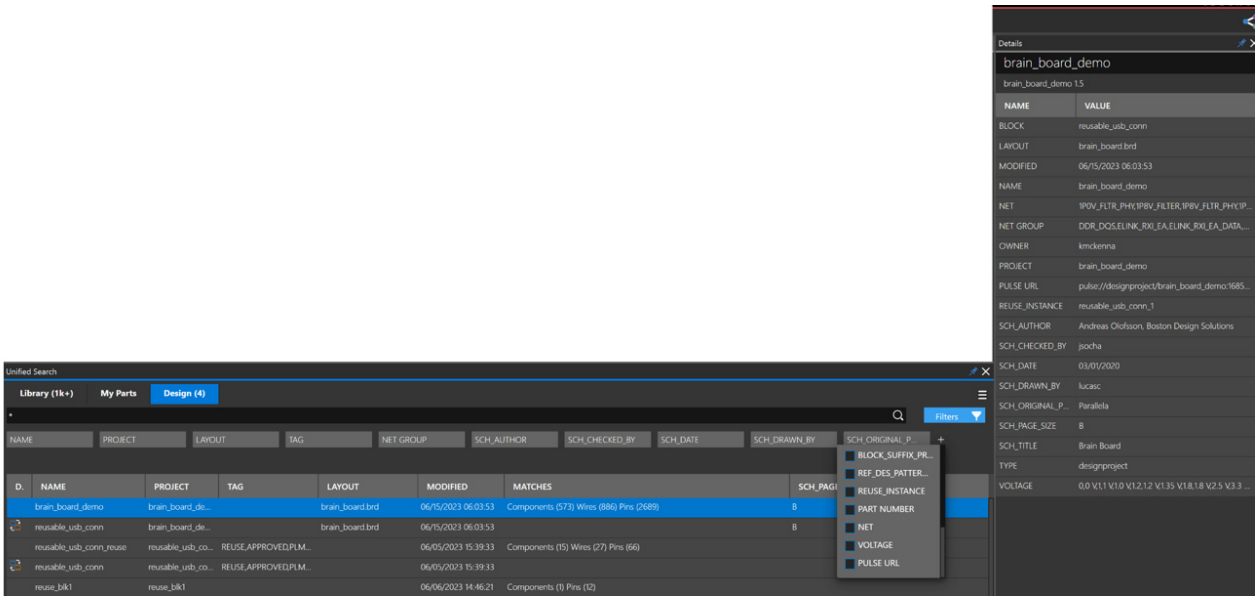
Disconnecting from Pulse Server

Design Search and Reuse

Unified Search supports searching data across designs and within design metadata. A new tab, *Design*, has been introduced for the multi-user environment. Search results display data

© 2023 All Rights Reserved.

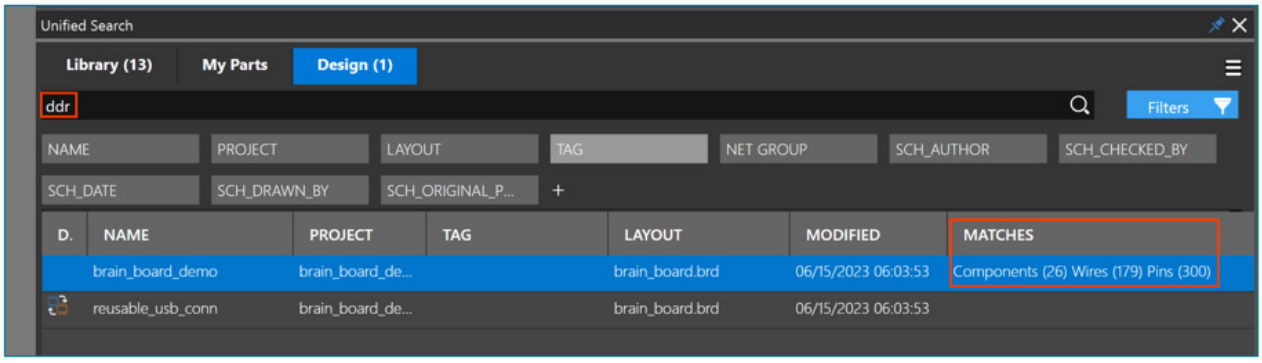
from the latest committed versions of the designs available on the shared server. You can also perform a parametric search of a design row in the *Details* pane.



Searchable data includes:

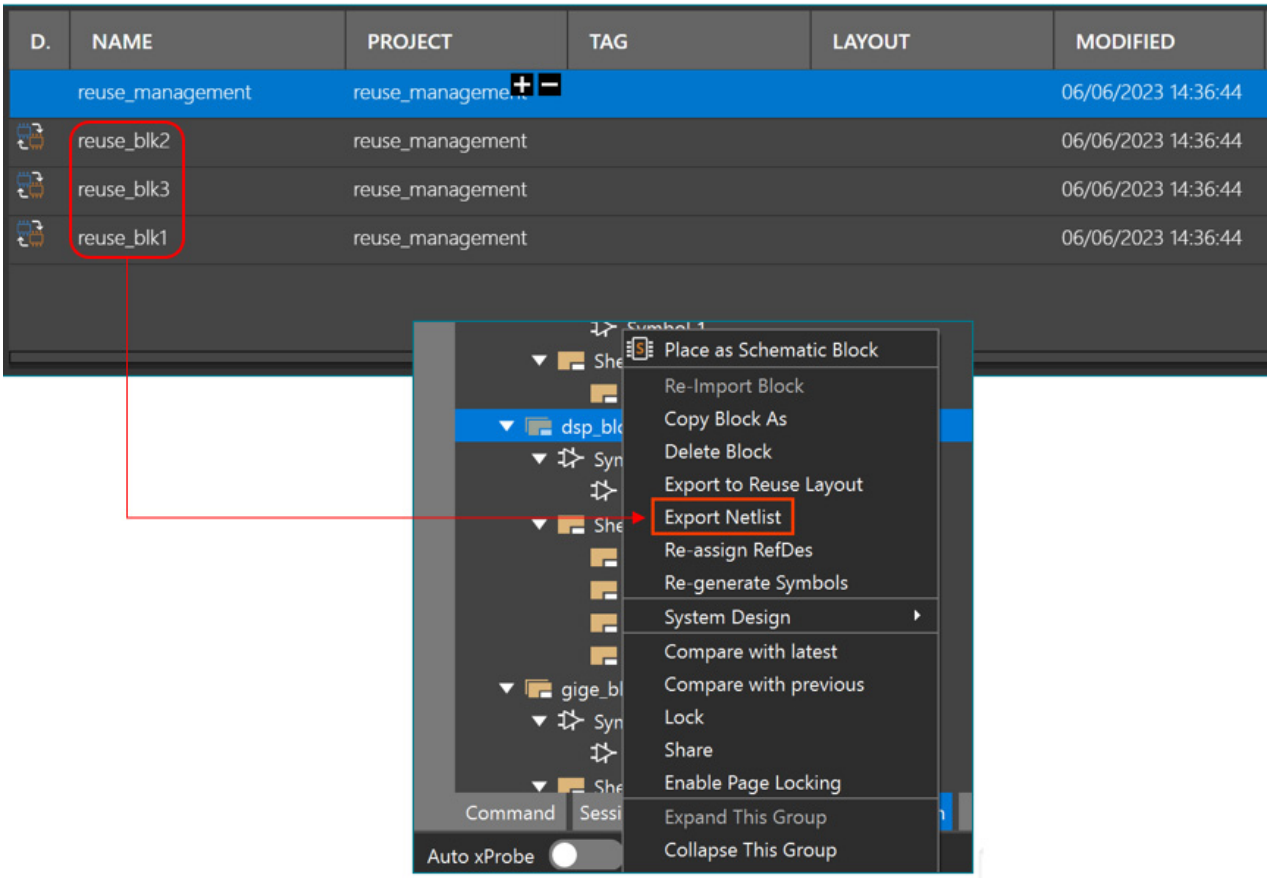
- Pulse metadata: Content in the project details form, version, the last modified date, design owner names, users with read-only access to a design, commit tags, comments, the Pulse URL
- Design information: Nets, NetGroups, buses, components, pins, voltages, CUSTOMVARs, variant names, block attributes,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

  Search results are aggregated into matches.

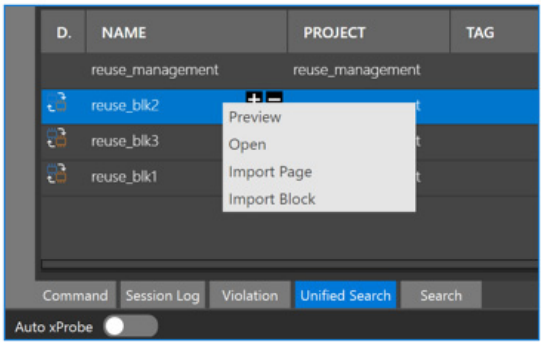


© 2023
All Rights Reserved.

To enable easy reuse of a design, blocks where netlists are exported for reuse are identified with an icon in the first column.



All the designs and projects that Pulse manages are displayed. You can only preview or import the designs that you have read or edit rights for.



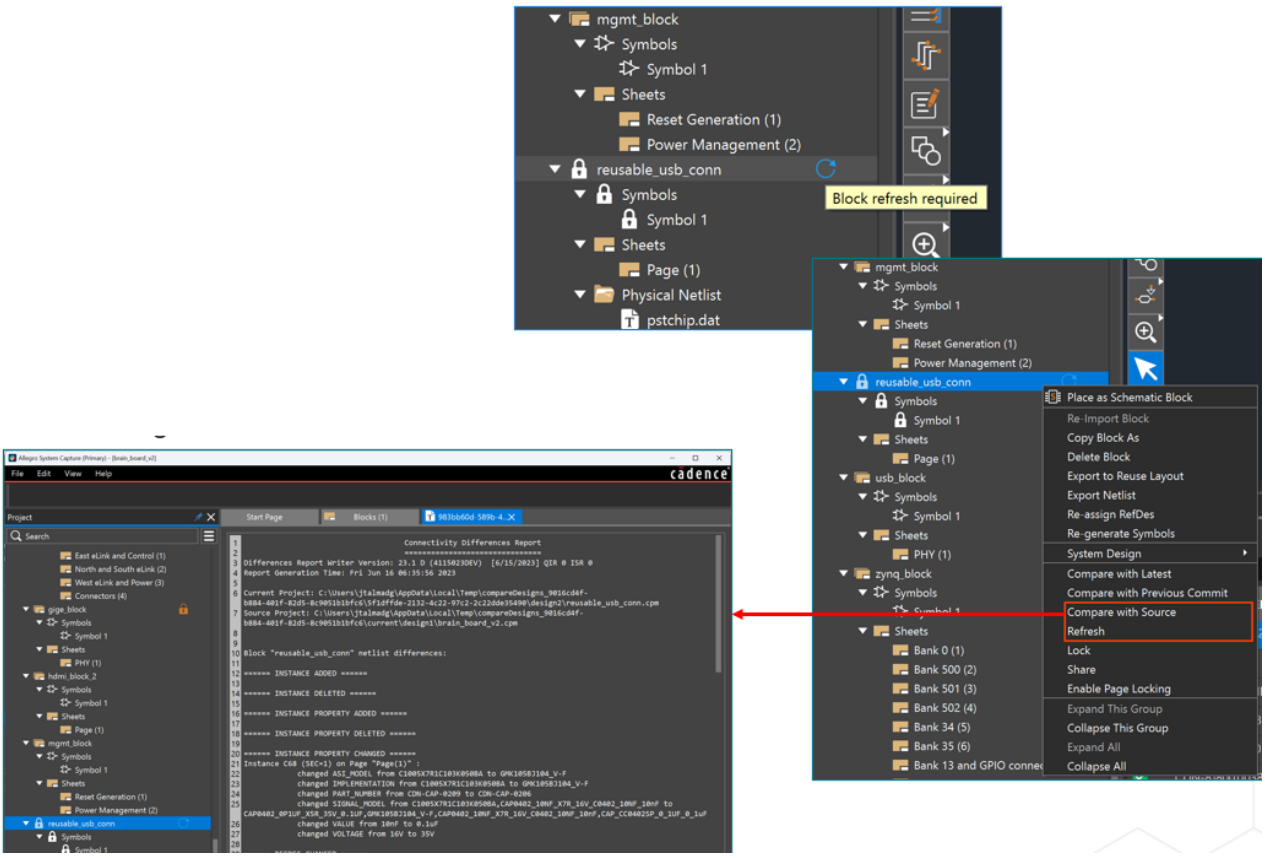
© 2023
All Rights Reserved.

If you have at least read-only access to a design or project, you can open the *Preview*, which includes a PDF with hierarchical navigation. You can right-click an item in the hierarchy navigation and import a block or page.

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

© 2023
All Rights Reserved.

assess the impact and click *Refresh* to update your design with the latest version of the reused block.



Related Documentation

Searching for Designs for Reuse

AWR MWO-Allegro RF Design Flow

Important

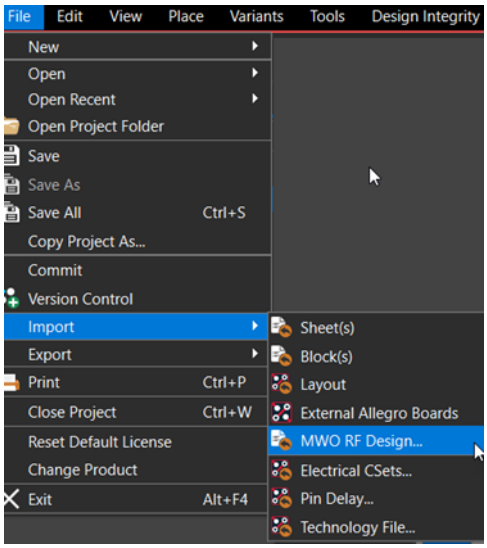
This functionality is currently a limited-release feature that is available only on Windows. To enable the flow, create an environment variable `MWO_ALLEGRO` and set it to `1`.

© 2023 All Rights Reserved.

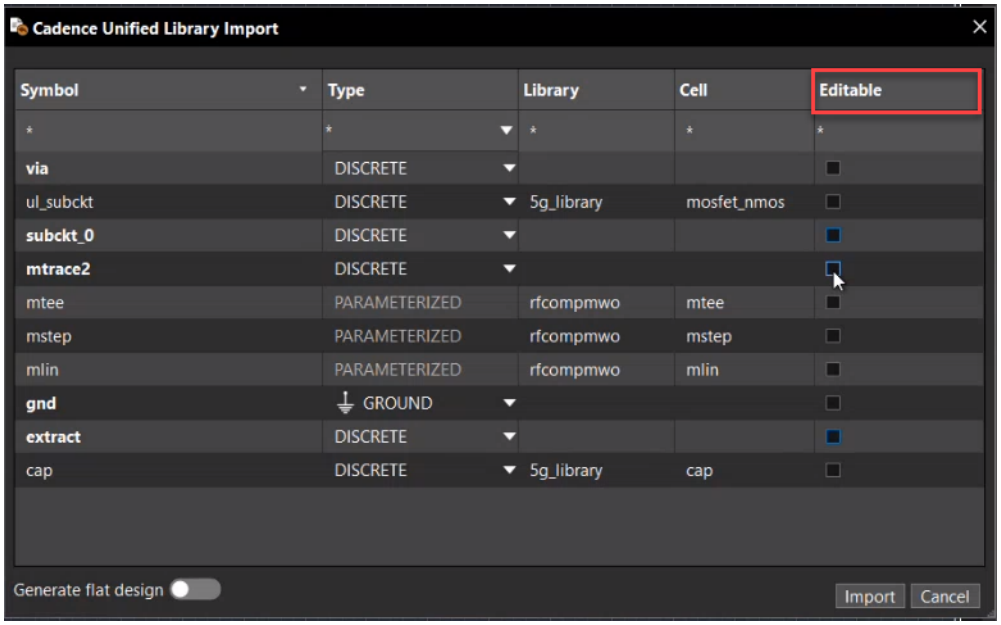
The integration of AWR® Microwave Office® (AWR MWO) and Allegro® PCB Design applications provides a way to create complex PCBs with RF design. Release 23.1 is compatible with the MWO 17.03 version.

Enhancements in this release include the following:

- Single container import from AWR MWO into System Capture.



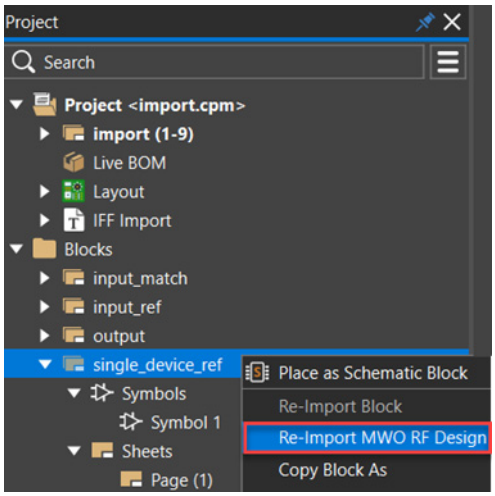
- Multi-primitives in the scope of components in the RFIP transfer flow.



© 2023
All Rights Reserved.

- Unmapped components from a single container are no longer locked and can be replaced with components from an Allegro library. This enables the use of split, symmetric, and asymmetric parts.
- Import RFIP into existing non-RF schematic.

  A single container can be imported into an existing schematic. This allows an RF block to be reused both in the logical design as well as in the layout.
- Re-import support is added for RF blocks.



Related Documentation

Importing RF Design in Allegro System Capture

Variant Editor Enhancements

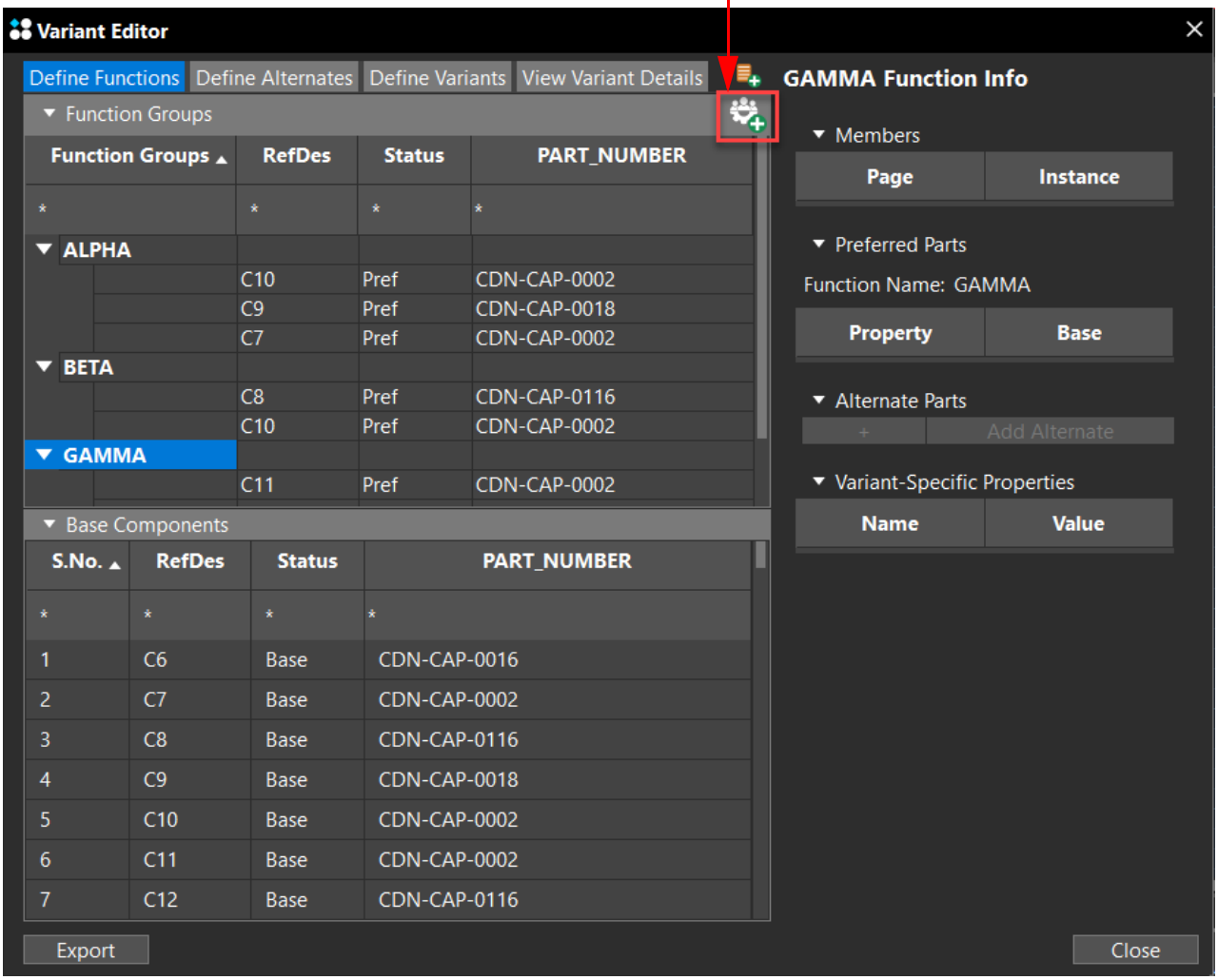
This release includes many enhancements in Variant Editor. In addition to viewing the variants summary, you can now perform the following tasks:

- Create function groups

© 2023
All Rights Reserved.

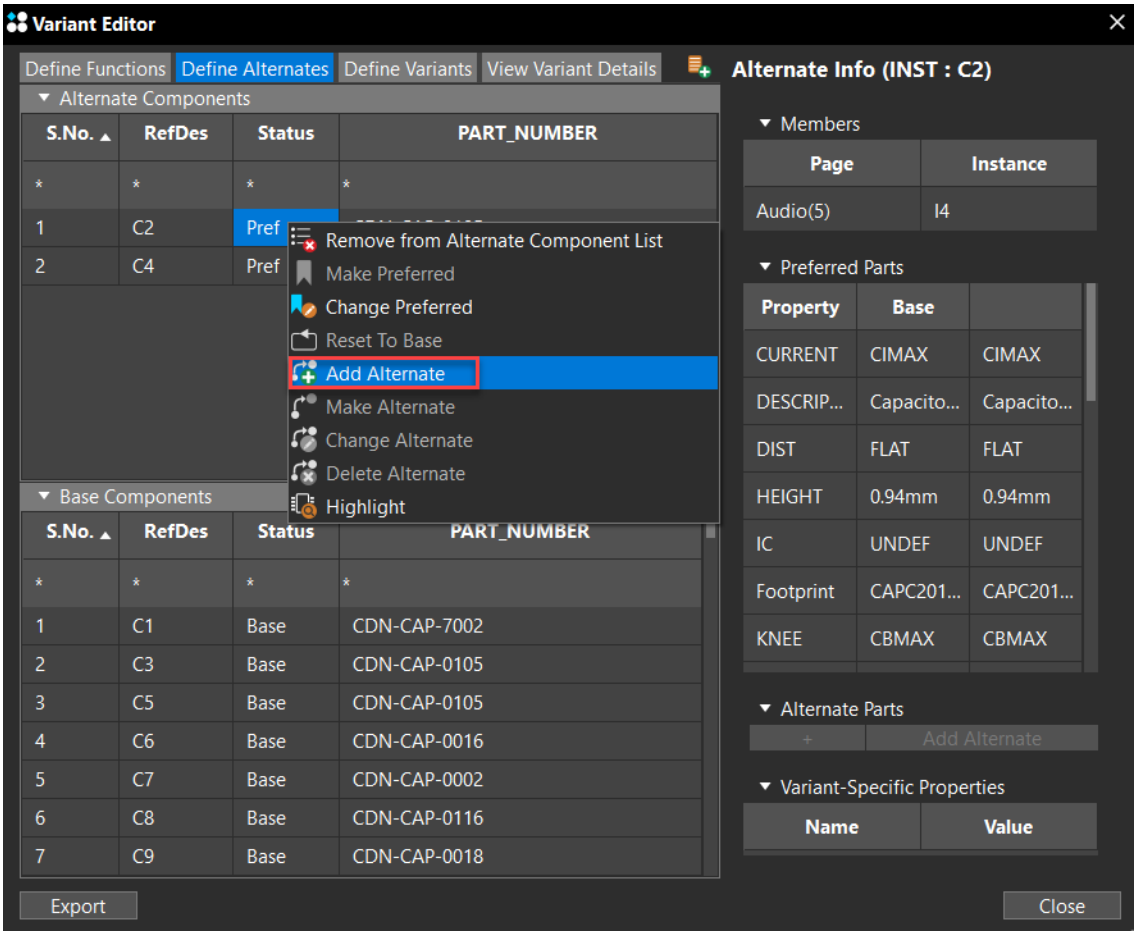
A function group is a collection of related components that can be added to a variant.

Create a Function Group



© 2023
All Rights Reserved.

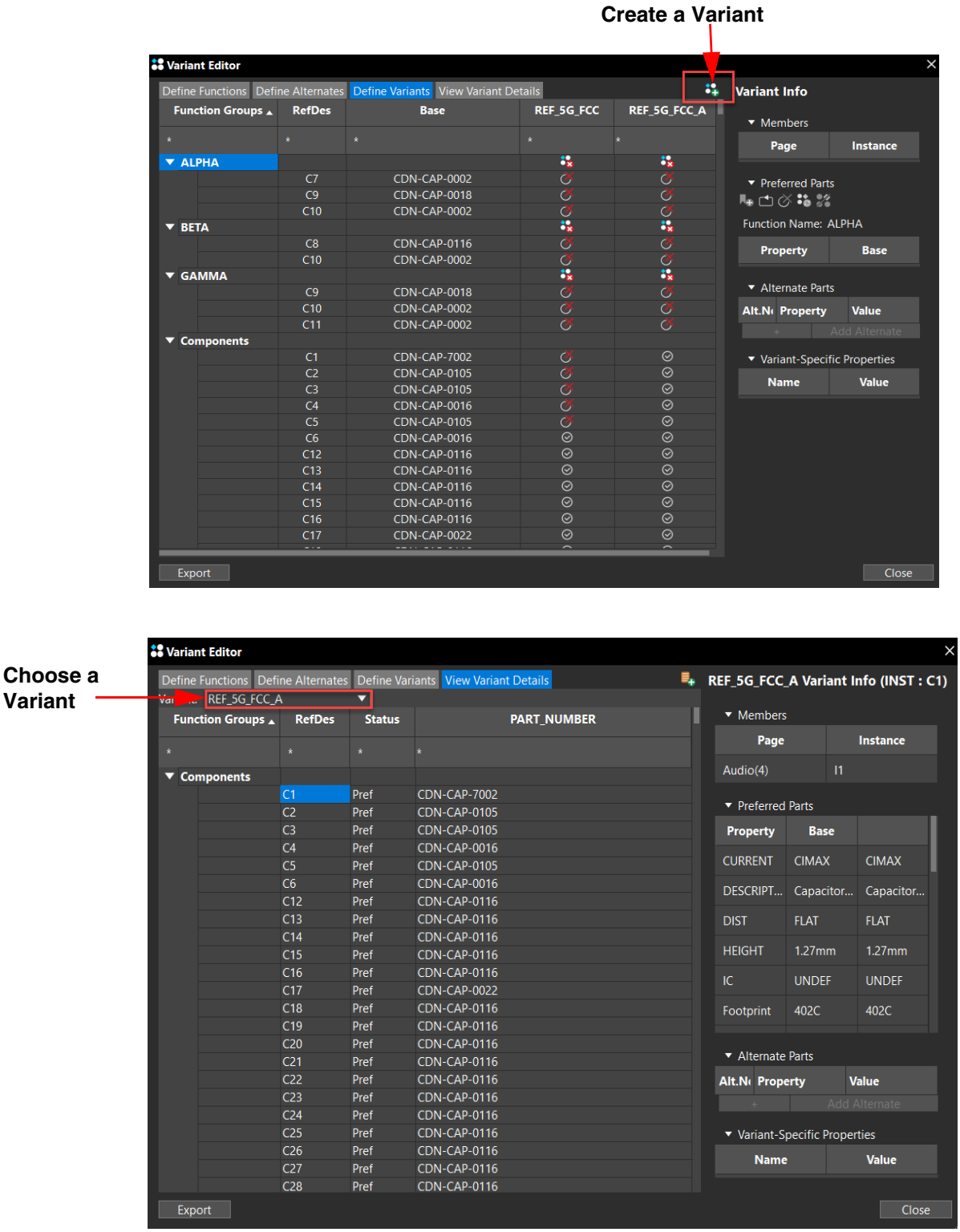
- Define alternate parts



- Define variants

© 2023
All Rights Reserved.

You can switch to a variant to view its details.

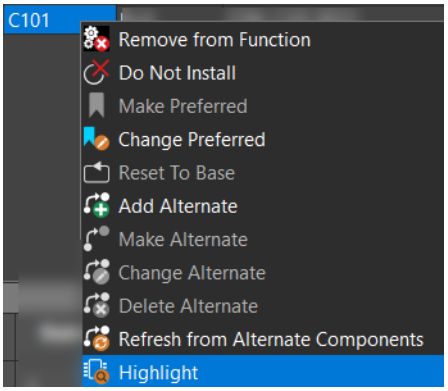


© 2023
All Rights Reserved.

- Column customization is introduced

  For example, you can filter content based on a property value and then work on that reduced data set for adding to a variant or function.

- Cross-probing components between the schematic and Variant Editor is introduced.



When you select an instance on the canvas, it is highlighted in Variant Editor. You can then add it to a variant or function.

Related Documentation

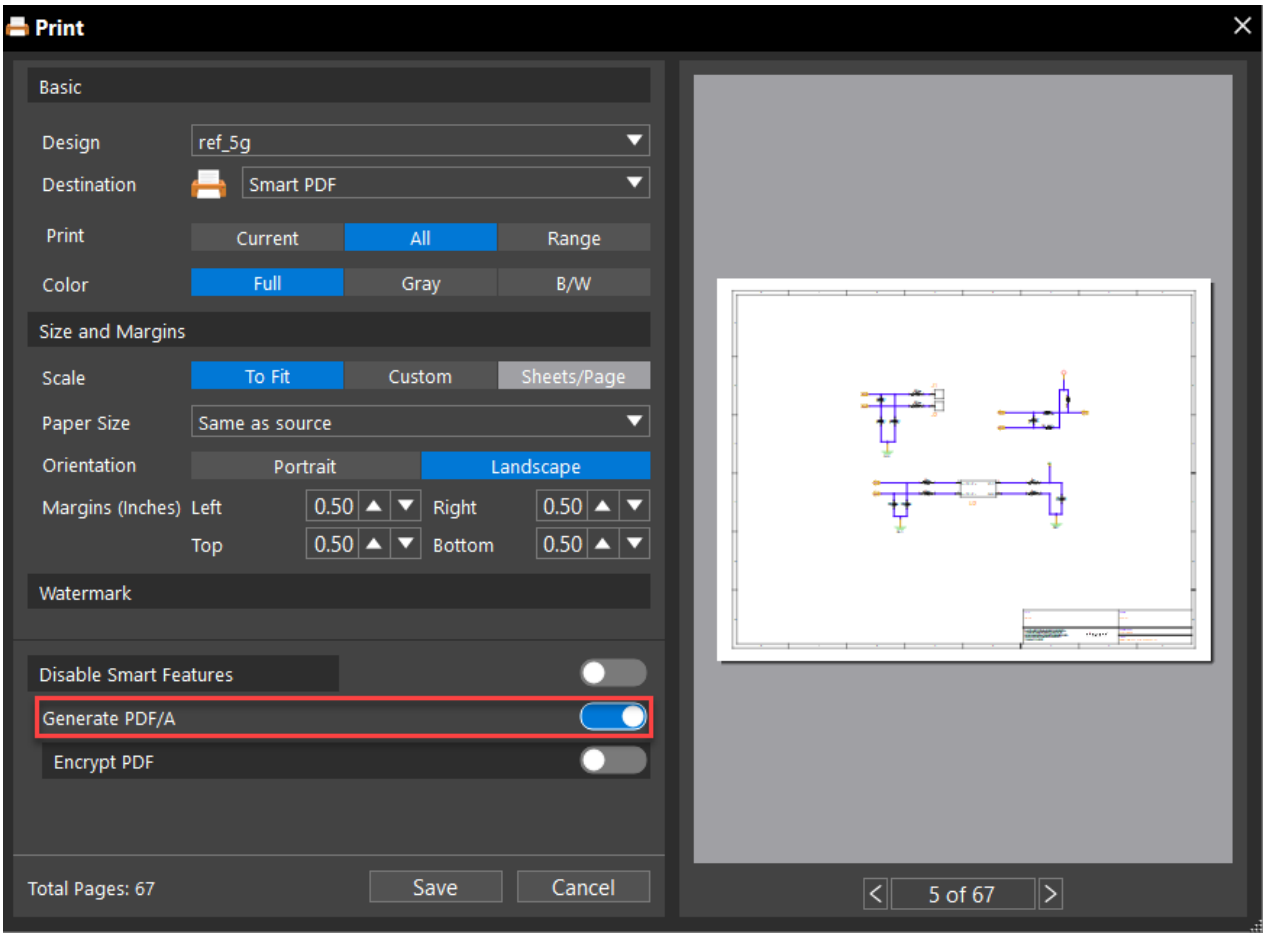
Variant Editor

PDF/A Support Added

System Capture now includes support for the PDF/A format when creating a SmartPDF for a design. PDF/A is an industry-standard format that embeds all the required font information within the document, making it more suitable for long-term archiving. A document that

© 2023 All Rights Reserved.

complies with the PDF/A format can be opened on any machine, not just those with the fonts used to create the PDF file.



Related Documentation

Create a PDF

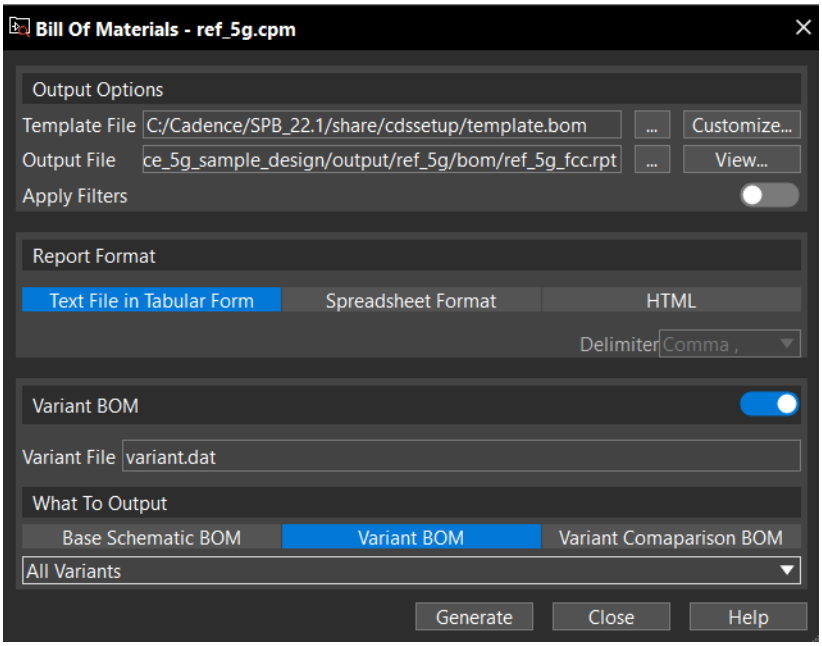
Updated Bill of Materials Dialog Box

The *Bill of Materials* dialog box UI has been updated. Here are some of the changes:

- Radio buttons replaced by tiles

© 2023
All Rights Reserved.

- Check boxes replaced by toggle buttons



Related Documentation

- Using Bill of Materials
- Bill of Materials Dialog Box

© 2023
All Rights Reserved.

Documentation Updates

Based on user feedback and to support new enhancements and features, the following guides have substantial changes:

- *Design Integrity and Analysis in Allegro® X System Capture*
- *Allegro® X System Capture User Guide*
- *Allegro System Capture Pulse User Guide*

  This guide has been renamed to *Design Data Management in Allegro X System Capture*.

- *Allegro System Capture Tcl Commands*

  The following commands have been added or updated:

  - `checkIfSyscapUpdateAvailable`
  - `crossprobeInTopology`
  - `dbGetActivePageSpath`
  - `dbGetSPathForActiveTab`
  - `dbSelectObjectById`
  - `downloadAttachment`
  - `dumpMenus`
  - `exportDashboardAsCsv`
  - `exportDashboardAsPdf`
  - `extractPTree`
  - `importBlock`
  - `MTBFPreferences`
  - `newProject`
  - `print`
  - `PTreePreferences`
  - `runMTBFAnalysis`
  - `runThermalAnalysis`

© 2023 All Rights Reserved.

- `variantEditor`
- `viewThermalMap`

© 2023 All Rights Reserved.