

# **Working in the Allegro® X Pulse Environment**

**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 PCB Editor 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. 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 DFAR52.227-7013 et seq. or its successor.

---

# Contents

---

<u>Data Management and Design Collaboration Using Allegro Pulse</u> .....	5
<u>Enabling Pulse Features</u> .....	5
<u>Pulse Features</u> .....	6
<u>Data Management Using Pulse</u> .....	10
<u>Design Sharing and Collaboration</u> .....	12
<u>Collaborative Editing</u> .....	14
<u>Concurrent Multi-User Editing</u> .....	15
<u>Layout Editor Connection to Pulse</u> .....	15
<u>Additional Configuration Options</u> .....	16
<u>Troubleshooting</u> .....	17
<u>Working with Pulse-Linked Layout and Schematic Designs</u>	19
<u>Creating Link in PCB Editor</u> .....	19
<u>Synchronizing Layout With Schematic in PCB Editor</u> .....	23
<u>Leveraging Pulse-Managed Module Design for Board Creation</u> .....	27
<u>Creating Modules for Pulse-Managed Schematic Blocks</u> .....	27
<u>Creating Board Using Pulse-Managed Modules</u> .....	30
<u>Managing Modules in Layout with Pulse Module Manager</u> .....	33

## Working in the Allegro X Pulse Environment

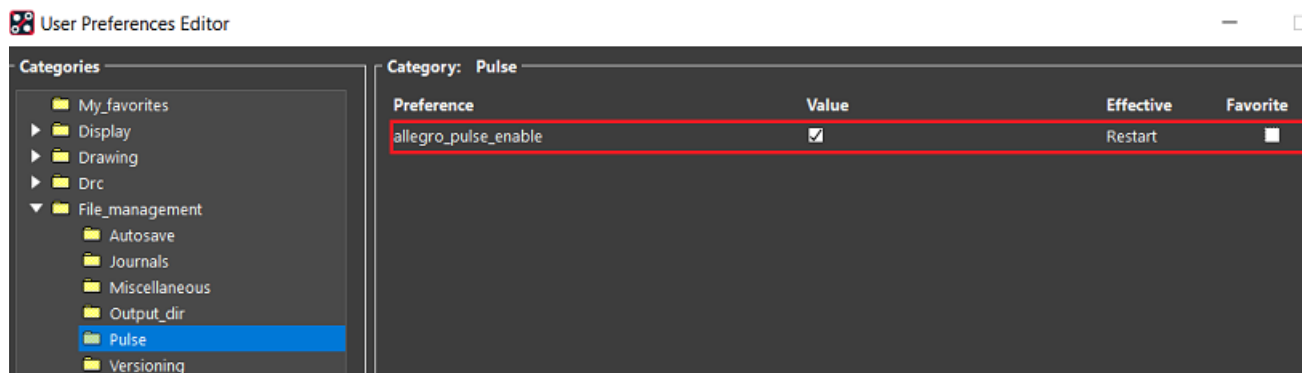
---

# Data Management and Design Collaboration Using Allegro Pulse

The native data management version control, and design sharing features in Allegro PCB Editor and Allegro X Advanced Package Designer provide an easy-to-use design sharing model familiar to anyone who has experience with mainstream Cloud-based file sharing applications. These features are backed by a robust data security infrastructure, which leverages the Pulse infrastructure as its backbone. The server (or servers), which provide encrypted communication (TLS 1.2) and design data security, via user management and authentication, can be integrated into a corporate single-sign-on solution.

## Enabling Pulse Features

To enable native data management features, set an environment variable `allegro_pulse_enable`, in the User Preferences Editor, and restart the layout editor.



Once set, the variable is saved in the `env` file and applies to subsequent sessions of the layout editors. To disable Pulse features, you can either unset the user preference variable, close the layout editor, or shut down Pulse. Upon relaunch of the layout editor, Pulse will not automatically start.

**Note:** The minimum hardware requirement to enable Pulse features is 8 GB of RAM and 4 CPU cores.

## Pulse Features

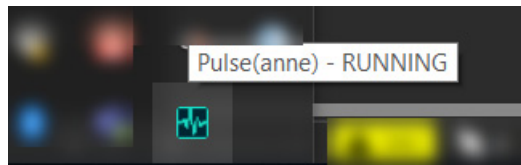
The Allegro layout editors with Pulse environment supports the following features.

### Version Control

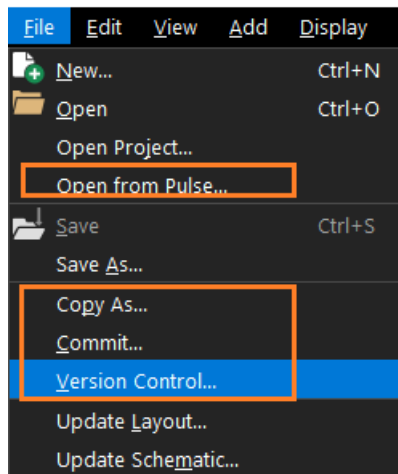
Native version control in Allegro PCB Editor and Allegro X Advanced Package Designer allows you to capture snapshots of designs at a point in time throughout the design process. Besides, it provides an easy way for designers to reference or revert to a previous version of the design. The version control features allow you to:

- Experiment with different placement or routing strategies
- Save the database before and after new netlist imports
- Save the database before and after constraint changes

If the `allegro_pulse_enable` variable is set, Pulse automatically starts and provides version control features when a layout editor is launched. The same Pulse icon in the system tray enables and controls versioning in all layout editor sessions and takes between 60-90 seconds to become active from the local installation.

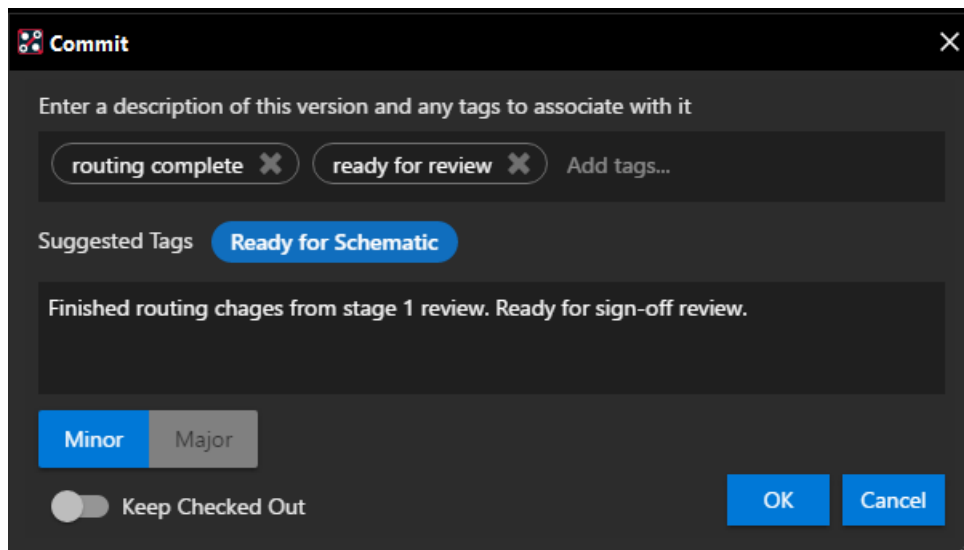


When Pulse is enabled, the following four commands are enabled in the *File* menu.



## Commit

You can generate a snapshot of the design at a point in time in the version control database using the commit command. The user interface provides options to specify tags and comments to identify the version of the design being created. A commit can be marked as *Major* or *Minor*, which controls the increment of the design version. For example, version 1.0 moves to 1.1 for a minor commit while the same version moves to 2.0 for a major commit. An additional option *Keep Checked Out* is available to keep the design checked out without unlocking the design after commit is successful.



**Note:** If you are working with System Capture as a schematic design application, you can leverage Pulse platform capabilities to synchronize your layout and the schematic. A pre-defined tag `Ready for Schematic` can be used when committing changes to the layout design. The addition of this tag notifies the schematic designer that the schematic is out of sync. For more information, see [Working with Pulse-Linked Layout and Schematic Designs](#).

## Version Control

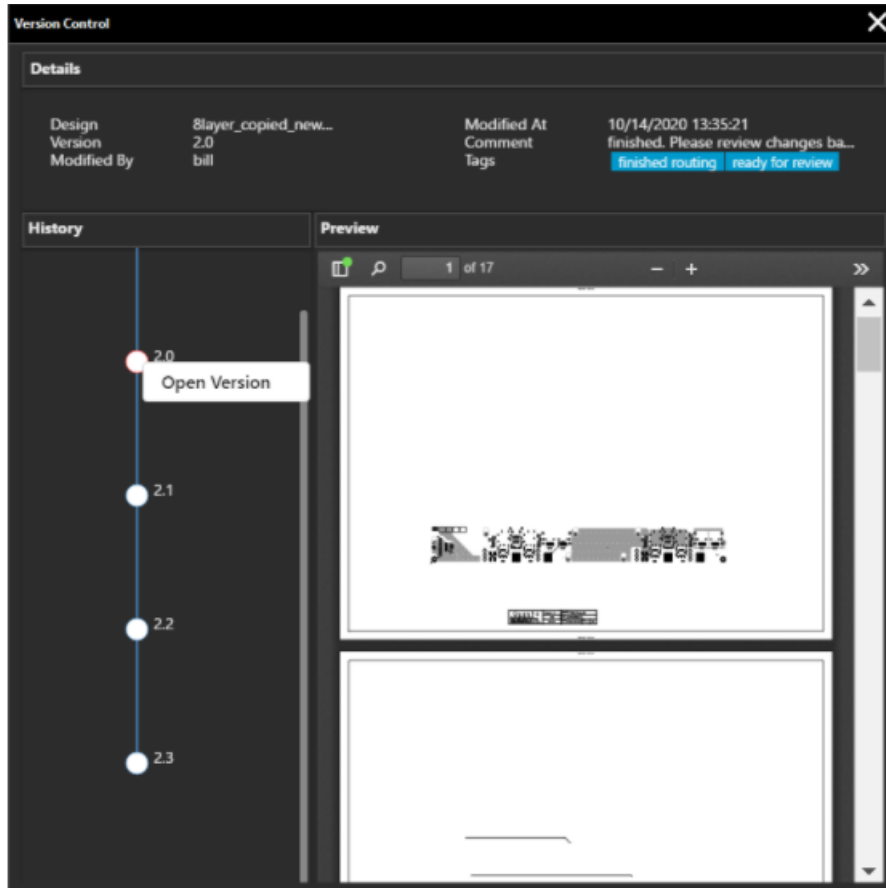
The version history of a design can be checked through the Version Control dialog. Each version of the design is represented as a node in the left pane. Selecting a node shows a preview of that version of the design in the right pane and relevant details on the section of the dialog. You can review any version of the design here without impacting the current version of the design that is loaded in the layout editor.

## Working in the Allegro X Pulse Environment

### Data Management and Design Collaboration Using Allegro Pulse

---

To open any of the previous versions in the layout editor, select the respective node, and right-click to choose *Open Version*.



Any version of the design can be locked for editing in a multi-user environment. If you are a design owner and acquired a lock, you can commit new changes based on this previous version. This action is equivalent to rolling back the design to a previous version. On the other hand, you can open the latest committed version from the same Version Control UI and move forward from there.

### Open from Pulse

Design versions already created can be browsed from the version control repository using the *File – Open from Pulse* command. You can view a list of available designs which are shared with you and open them in the layout editor. The command first looks for the selected design on the user's disk and if found, opens the local copy of the design in the layout editor. Else, it

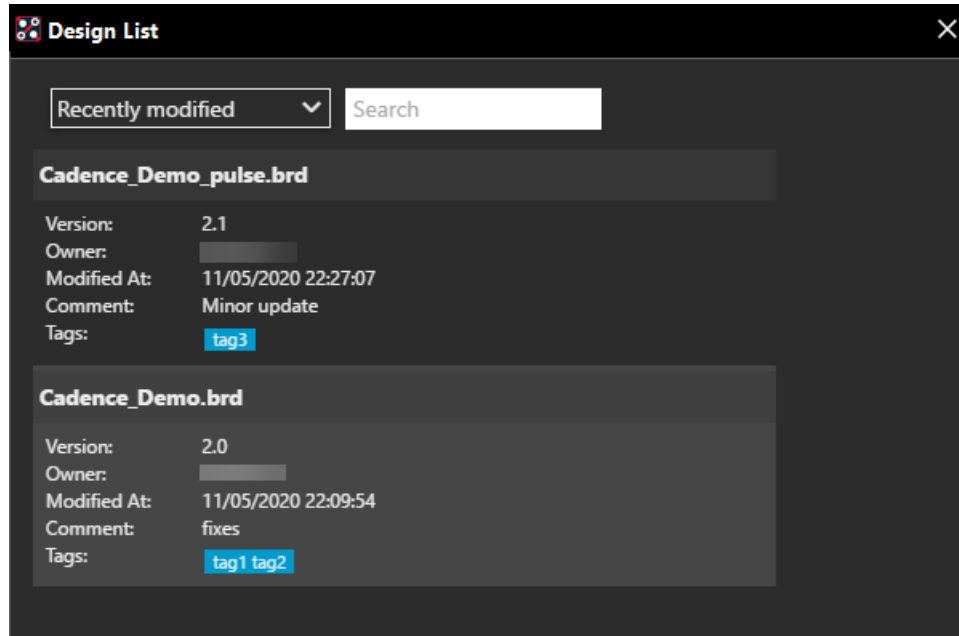


## Working in the Allegro X Pulse Environment

### Data Management and Design Collaboration Using Allegro Pulse

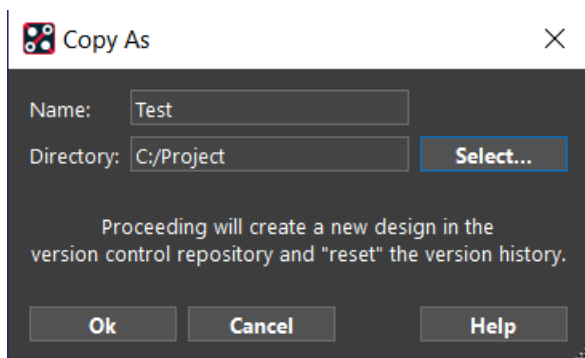
---

downloads the latest version of the design from the Pulse server and opens it in the layout editor.



### Copy As

Use this command to create a new design from an existing design with a version history in the version control system. Any existing design acts as a starting point for a new design. This command resets the version history of the new design and the first commit is saved as version 1.0.



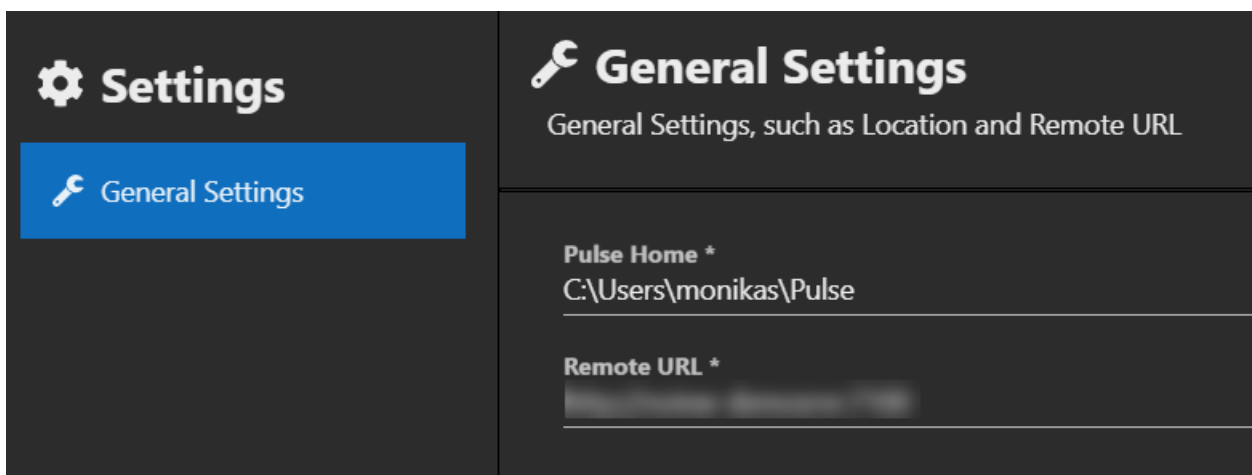
## Data Management Using Pulse

You can access native data management support through Pulse in two ways: single-user environment (using the individual system as the Pulse server) or multi-user environment (using a remote/central Pulse server). These two use modes provide scalable data management and version control, which includes basic version control for the individual user as well as collaborative, shared version control for larger design teams.

### Single-User Environment

In this mode, a version control system starts up and runs with the layout editor. By default, the data storage location for Pulse is the user's home folder. A sub-folder called Pulse (`$HOME/Pulse` on Linux and `%USERPROFILE%/Pulse` on the Windows platform) is created. The design versions, along with other operational data for the Pulse features, are stored in the Pulse directory.

To change the default storage location, right-click the Pulse icon in the system tray, and select *Manage*. In Pulse Manager, a web page that provides access to Pulse operations, you can modify Pulse Home to a new location for the storage of the design versions. For more information, refer to [Pulse Manager Settings Web Page User Interface](#) in *Design Data Management in Allegro X System Capture*.

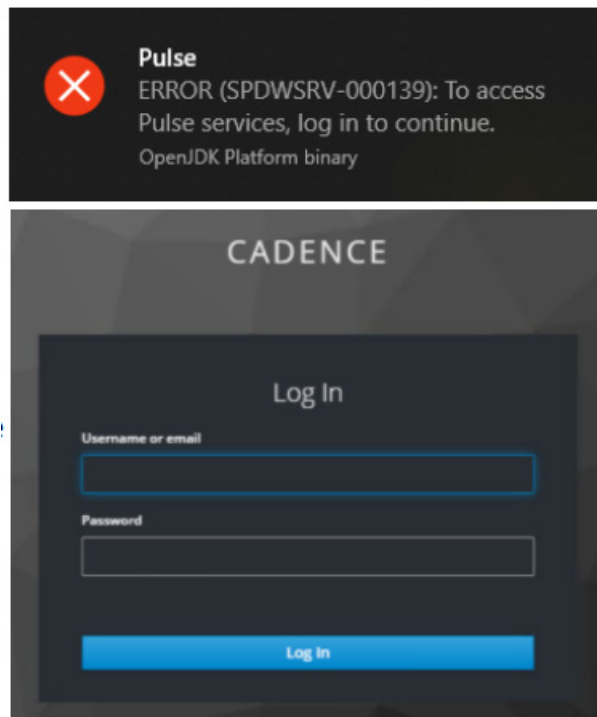


**Note:** After changing the Pulse home, the existing versions of the designs remain in the previous location and do not migrate automatically to the new storage location. However, you can manually move the design data to a new location. As a best practice, before using the native data management features, always identify a location with enough disk space.

### Multi-User Environment

Native data management features can also be configured to connect to a central Pulse server. Connecting to a Pulse server not only provides a robust central repository for design data but also enables new collaboration models for design teams to leverage. When connected to a Pulse server, teams can hand off design authoring responsibility directly within the layout editor. Teams can also share a design with stakeholders who may not be directly involved in design authoring but still require access to the design content.

In the multi-user environment, design versions are created and stored on a central server rather than on individual systems. As soon as you start the layout editor, you are prompted to log in to the Pulse server.



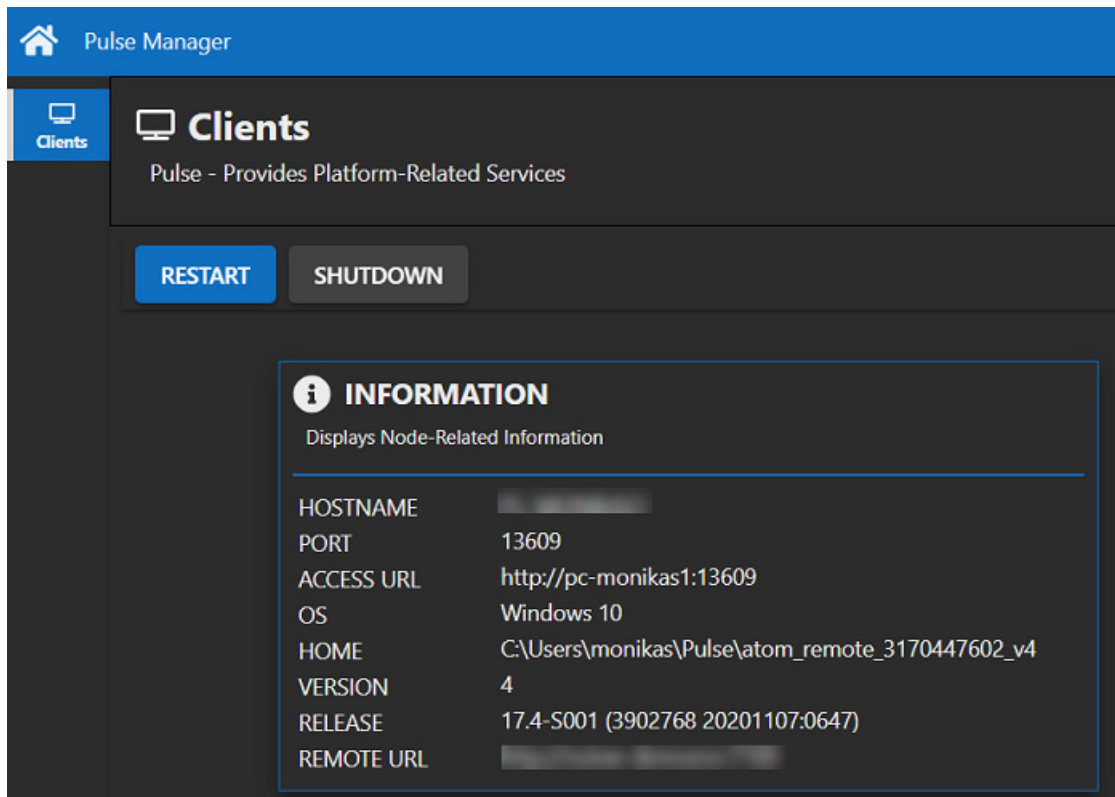
Logging in with authenticated credentials successfully enables the data management commands in the *File* menu.

## Working in the Allegro X Pulse Environment


### Data Management and Design Collaboration Using Allegro Pulse

---

The Pulse server can be set in the Remote URL field of the Pulse Manager settings or by using the `PULSE_REMOTE_URL` environment variable. For example, `PULSE_REMOTE_URL=http://pulse-server:7100`.



## Design Sharing and Collaboration

When connected to a Pulse server, you can share designs with other members of your team. The member sharing a design is automatically designated as the owner and can add additional members to work on the design using the Share Design dialog that is accessible through a toolbar icon  in the layout editor. Note that only the owner of the design can modify the design sharing configuration, including the list of members or groups and their roles.

Design owners can select specific members or a group to share the design with. While adding members or a group, owners can also select the permission level for each individual or group with whom a design is shared. The available permissions are:

- **Owner:** The owner has both read and write permission for the design and the ability to edit the team definition of the design. The owner can also forcefully release a lock held by another team member. Releasing the lock allows another member to lock (or check out) a design for editing. It is important to understand that any changes made by the









## Working in the Allegro X Pulse Environment

### Data Management and Design Collaboration Using Allegro Pulse

---

member whose lock was forcefully released will not be committed to the server and could be lost.

- **Read/write:** Members with read/write permission have read and write access to a design. They can lock the design for editing and commit their changes to the version history of the design. They can only view the team definition in the Share Design dialog but cannot modify permission settings.
- **Read-only:** Members with read-only permission only have read access to a design. A read-only icon in red color is displayed for Pulse in the status bar indicating this state. With read-only permission, members cannot acquire an edit lock for a design or commit any changes to the version control.

Design State	Icon
Locked by current user for editing	
Locked by another user	
Locked by another user + Update Available	
Unlocked + Update Available	
Lockable + Read Only	
Non-lockable + Read Only	
Lockable + Read Only + Update Available	
Non-Lockable + Read Only + Update Available	

Any design connected to the Pulse server is initially unlocked and can be modified without acquiring any lock. As soon as a user first commits a design, the design will be transitioned to a share-able design in Pulse. The Pulse server, in connection with the layout editor, manages an exclusive write lock. The user who holds the lock can make changes and commit those changes to the version history.

The Pulse design locking status is displayed in the layout editor when the design opens. The descriptions of the status icon are captured in the [Pulse StatusTable 1-1](#) on page 15. For quick reference, you can refer to tool-tip text for any icon. Each status icon has associated actions that depend on the permission settings as defined in the Share Design dialog.

The design owner has always the power to force the release of a lock when the member who acquired the lock is unavailable. This action allows another user to lock the design for editing and continue design work. In this case, any non-committed changes by the unavailable user remain only in the local copy of the design.

## Collaborative Editing

Pulse design sharing capabilities enable multiple users to access Allegro PCB and Package designs. It also facilitates one-editor-at-a-time collaborative design, which also provides protection against losing work due to concurrent design changes that can be challenging to merge together. A common design process known as follow-the-sun is easily achieved with the Pulse design-sharing features in the Allegro layout editors. Following sample flow can be adopted in the multi-user environment:

1. User A creates a design.
2. User A commits the design to create the initial version (1.0).
3. User A shares the design with other team members (including User B).
4. User B opens the layout editor, selects *File– Open from Pulse*, and selects the design shared by User A.
5. User B opens the design in the layout editor and can lock the design for editing.
6. User B changes and commits the design.

This action releases the edit lock and notifies other members of the design team of a new version of the design.

7. User A selects Update, which pulls down the new version User B has created.
8. User A can now lock the design for editing and continue authoring the layout design.

In addition to collaborative editing, additional users can be added to a design with read-only permissions. These users can open the design, view the version history of the design, and be notified of updates to the design made by other users. They cannot lock the design for editing or commit changes to the design to Pulse.

Adding read-only users enables team members who are not directly responsible for the layout design to stay in sync with design changes being made and provide early feedback. For

example, other layout designers who can provide design review feedback, schematic engineers responsible for the logic design, project managers, and manufacturing/DFM engineers can be added with read-only permissions.

## Concurrent Multi-User Editing








While design sharing through Pulse enables multi-user access to the design it does not directly enable concurrent multi-user design. Symphony Team Design can be used in concert with Pulse design sharing.

You can lock a design shared with Pulse for editing and start a Symphony server to host the design for others to join and edit concurrently. The user with the design lock maintains the lock throughout the Symphony collaboration session. Once the Symphony session is complete, the user can then commit the design to check-in the changes made by the team. After the user who initiated the Symphony session has released the lock (by committing the changes), another user can then acquire the lock for further editing of the design.

## Layout Editor Connection to Pulse

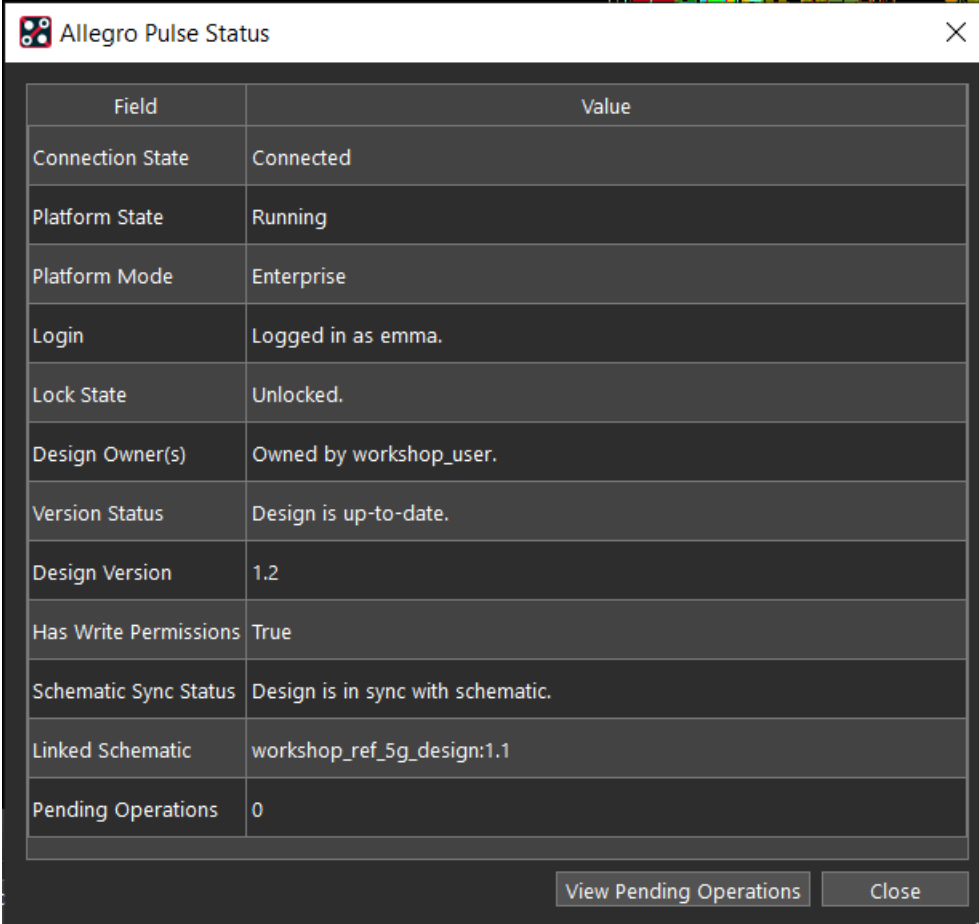
The status bar of the layout editor always displays the Pulse connection status through different icons. These icons give an insight into what is happening with the infrastructure and which actions are to be taken to continue the designing process. The following table shows the Pulse status and associated status bar icons:

**Table 1-1 Pulse Status**

Layout Editor's Connection to Pulse	Pulse's State	Icon
Connecting	Running	
Connecting	Running	
Disconnected	Running	
Connected	Maintenance	
Connected	Pending login	
Connected	Uploads pending	
Connected	Uploads pending + Maintenance	

### Design Status in Pulse

Details about the design and the connection to Pulse can be checked at any time by right-clicking the status icon. You can see the status of any pending operations, such as, a large design that is still being uploaded to the Pulse server.

A screenshot of the 'Allegro Pulse Status' dialog box. It features a title bar with the Allegro logo and a close button. The main content is a table with two columns: 'Field' and 'Value'. The table lists various system and design status metrics. At the bottom right, there are two buttons: 'View Pending Operations' and 'Close'.

Field	Value
Connection State	Connected
Platform State	Running
Platform Mode	Enterprise
Login	Logged in as emma.
Lock State	Unlocked.
Design Owner(s)	Owned by workshop_user.
Version Status	Design is up-to-date.
Design Version	1.2
Has Write Permissions	True
Schematic Sync Status	Design is in sync with schematic.
Linked Schematic	workshop_ref_5g_design:1.1
Pending Operations	0

### Additional Configuration Options

In addition to the remote URL setting, there are several settings and configurations available for Pulse. These include storage management, central logging, and backup/restore.

For more information, refer to Allegro Pulse Configuration Guide.



## Troubleshooting

### **Pulse commands (Commit, Version Control, Copy As) are not enabled in the File menu**

You can try one of the following solutions:

- Wait for some time. The commands are enabled after Pulse finishes initializing. It may take 90 seconds or more when running in embedded mode.
- Make sure Pulse is running (hover the mouse over the Pulse icon in the system tray). If Pulse is not in the running state, resolve any errors reported in Pulse Manager.
- Ensure that the installation is not being run from a UNC path on Windows. Pulse does not support running from a UNC path. Try running Pulse again using a mapped drive.
- Close the layout editor and restart Pulse. The Pulse commands become active after the Pulse finishes its initialization.

### **Design versions are not successfully being created**

There are two possible solutions for this:

- If network connectivity is interrupted while the design version is being uploaded to the Pulse server, the commit may fail. Retry committing the design.
- Save the design and close the layout editor. Restart Pulse and the layout editor.

### **Version Control or the Share Design form fails to display**

To view the Share Design and Version Control dialogs create at least one version of the design. Commit the design and try launching these dialogs again.

### **None of the solutions listed above worked and I need help**

Submit a case to Cadence including:

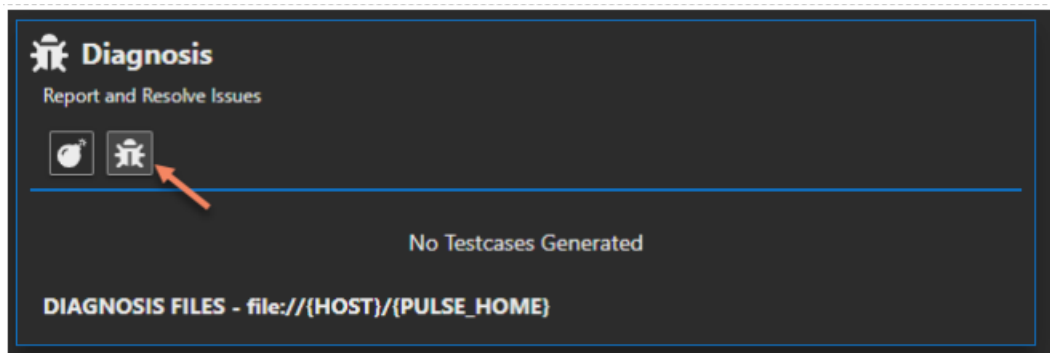
- A description of the problem or a video capturing the actions and issues.
- Relevant log information, which includes:
  - ☐ A zip file of the Pulse logs. To create a zip file with the logs, right-click the Pulse icon in the system tray, and choose Home. Zip the server/log folder
  - ☐ The Allegro journal file

## Working in the Allegro X Pulse Environment

### Data Management and Design Collaboration Using Allegro Pulse

---

- Any `allegro_<>.log` files created in the same directory as the journal file
- Create a test case using Pulse Manager. Click the *Generate* button and ask the administrator to retrieve the test case from the Pulse server. Provide this data to the Cadence support team for analysis.



---

## Working with Pulse-Linked Layout and Schematic Designs

---

When designing a PCB, it is important to keep the schematic and layout synchronized all the time, irrespective of whether you are creating a new layout or updating an existing layout or schematic. The task of syncing can be carried out by Pulse for the schematic and layout created in Allegro System Capture and Allegro PCB Editor, respectively. In the multi-user environment, both schematic and layout must be shared through the same Pulse server.

To sync the schematic and layout, Pulse establishes a link between two databases whenever designers commit a change in the design and apply `Ready for Schematic` or `Ready for Layout` tags. These tags trigger netlist regeneration that should be imported into either System Capture or PCB Editor to synchronize the databases. The import process, when started from Pulse, creates a link between schematic and layout. Pulse maintains this link and keeps track of all the versions of the schematic and layout marked for netlist updates during the design cycle. Multiple design linkage is supported, which means you can link multiple layouts with one schematic or vice-versa. Reimporting the latest versions of the schematic or layout ensures that they are perfectly in sync.

The versions of layout and schematic associated through Pulse cannot be disconnected. To create a new layout from the existing one, use the *Copy As* command in PCB Editor. Note that the copied layout acts as a new design and does not maintain the version history.

### Creating Link in PCB Editor

A layout, when opened in PCB Editor, by default, is not linked to any schematic. Importing schematic netlist from Pulse establishes a link between the schematic and the layout.

To create a link between a layout and schematic in PCB Editor, follow these instructions:

1. In PCB Editor, choose *File — Import — Logic/Netlist*.
2. In the Import Logic/Netlist dialog that opens, select *Import logic type* as *Design Entry HDL/System Capture*.

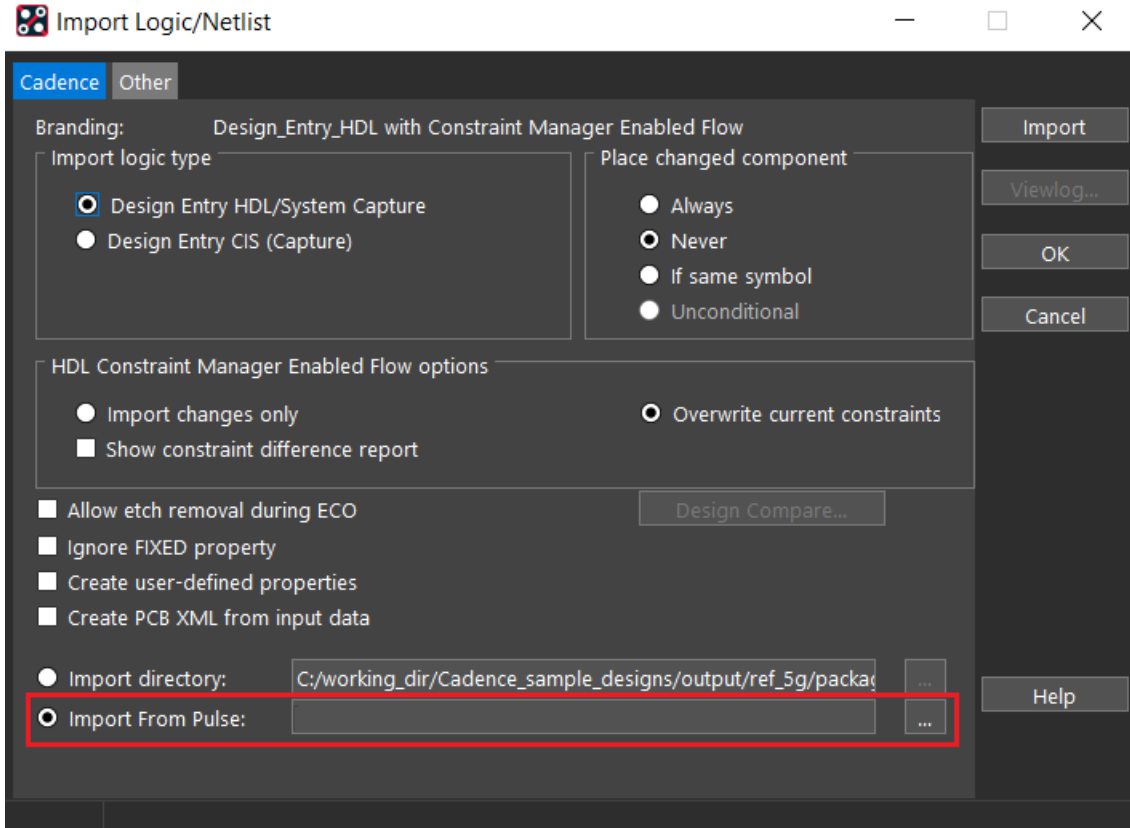
A new option *Import From Pulse* is displayed right below the *Import Directory* option.

## Working in the Allegro X Pulse Environment

### Working with Pulse-Linked Layout and Schematic Designs

---

3. Select the *Import From Pulse* radio button and click the browse button.

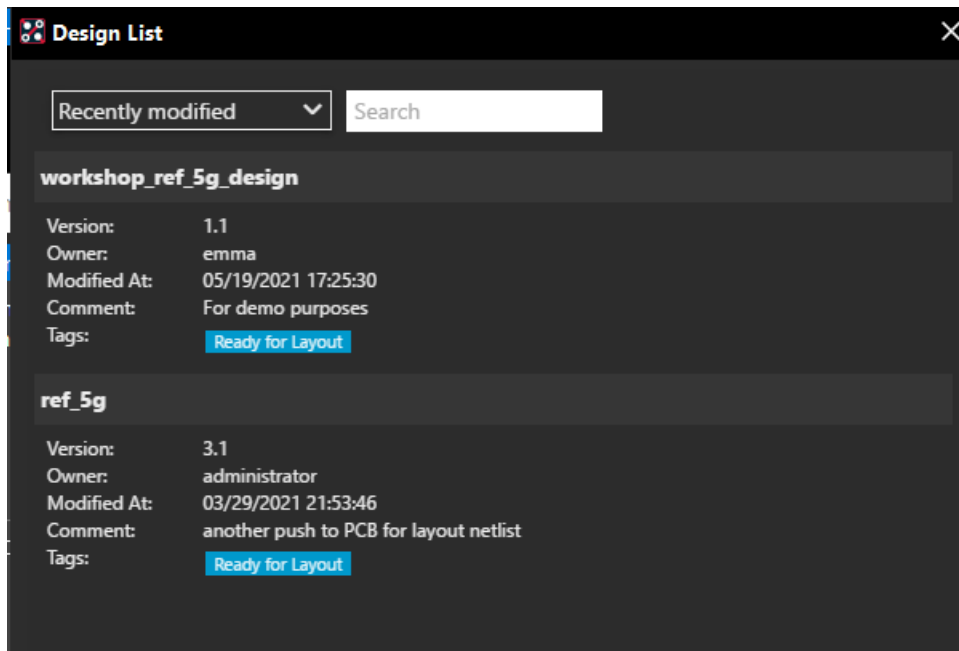


## Working in the Allegro X Pulse Environment

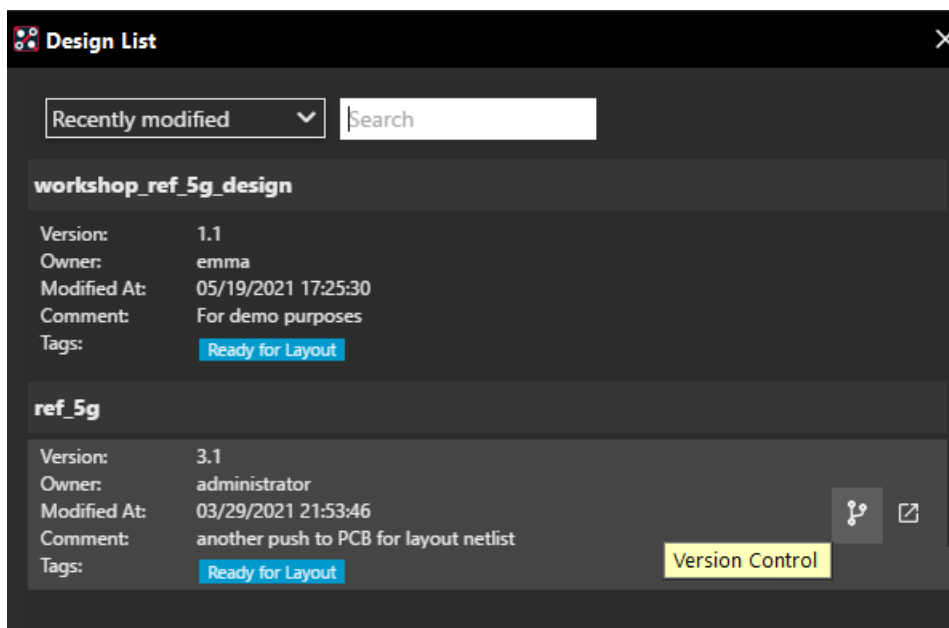
### Working with Pulse-Linked Layout and Schematic Designs

---

The Design List window opens and displays a list of schematic database names and versions shared with the layout designer and labeled as *Ready for Layout*.



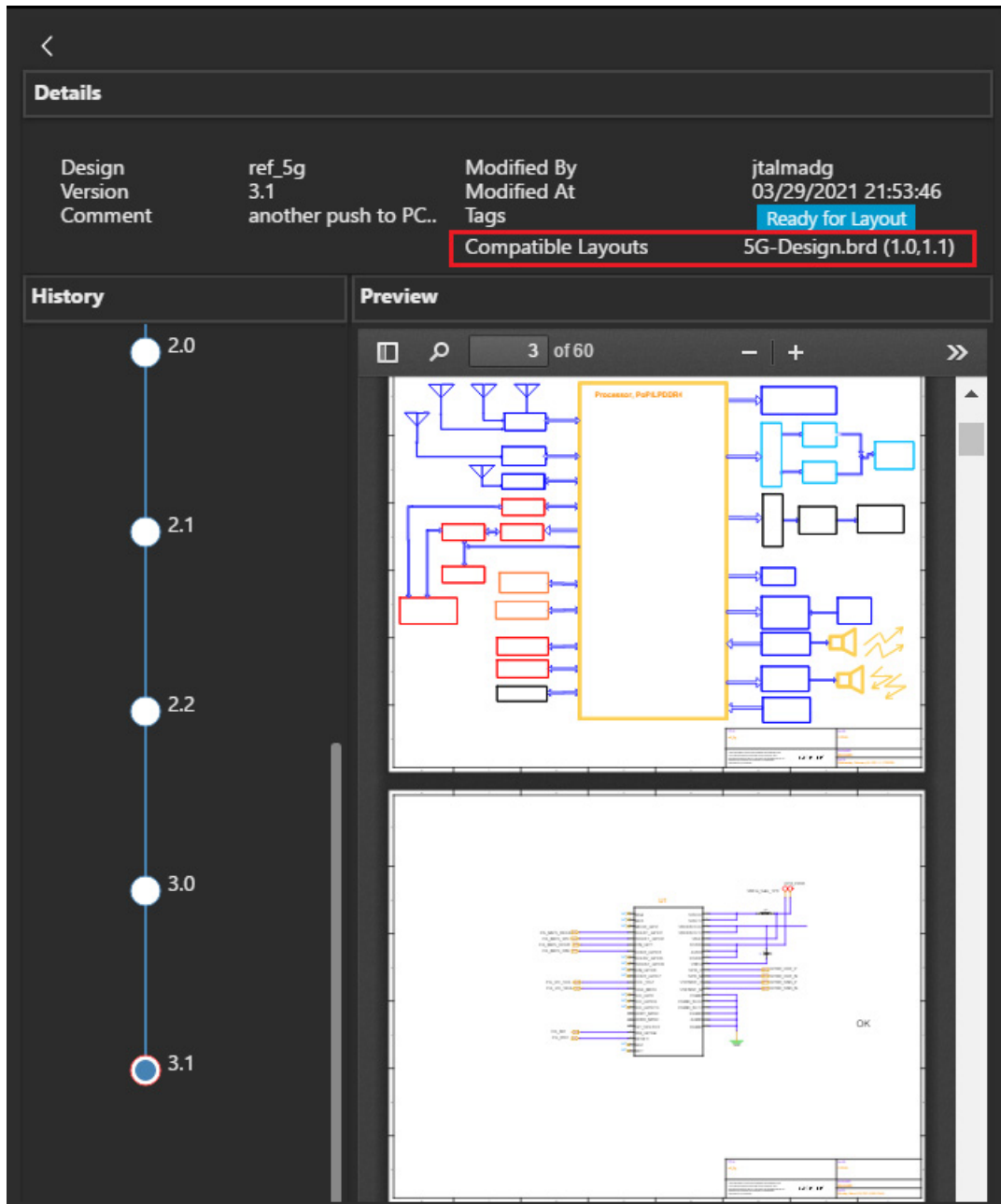
4. To select the schematic version to import, filter the list by *Recently modified*, *Design*, *Owner*, or type the name in the *Search* field.
5. Hover over a schematic and click the *Version Control* icon to preview the version graph.



## Working in the Allegro X Pulse Environment

### Working with Pulse-Linked Layout and Schematic Designs

The version graph shows *Compatible Layouts* for schematic versions tagged as *Ready for Layout*, and which have been either imported into the layout or have been back annotated to the schematic.



6. Go back and click the *Open* icon to launch System Capture to view schematic updates.
7. In the Design List window, double-click to select the schematic to import.

The *Import Form Pulse* field populates with the schematic name and version.

## Working in the Allegro X Pulse Environment

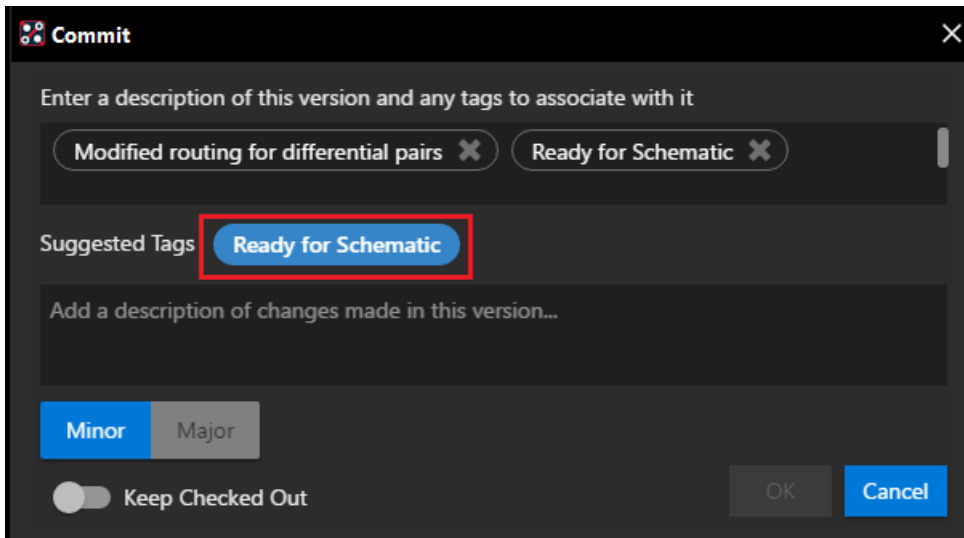
### Working with Pulse-Linked Layout and Schematic Designs

---

#### 8. Click on *Import*.

The import process downloads the netlist from the Pulse and updates the layout data. On success, Pulse establishes a link between the schematic and layout databases.

#### 9. Any addition or deletion of component and property, modification in the connectivity or reference designators required synchronization between layout and schematic. In such cases after modifying the layout, commit the changes with the *Ready for Schematic* tag.



The netlist is regenerated and uploaded into Pulse along with the version of the layout. The schematic designers who have permissions receive a notification indicating that the current schematic is out of sync and a newer version of the layout is available.

**Note:** *Read Only* permission is enough to share the netlist with schematic designers.

The schematic designer now re-imports the netlist from Pulse to be in sync with the layout. For more information, see [Linking Schematic and Layout for In-Design Notifications of Changes](#) in *Design Data Management in Allegro X System Capture* user guide.

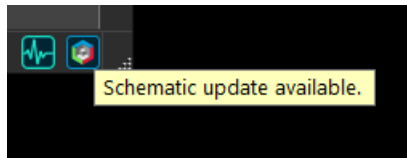
## Synchronizing Layout With Schematic in PCB Editor

Once Pulse establishes a link, for the subsequent schematic commits labeled as *Ready for Layout*, Pulse sends notifications to the layout designers. A new icon in the status bar of the PCB Editor represents that schematic update is available and the *Schematic Sync*

## Working in the Allegro X Pulse Environment

### Working with Pulse-Linked Layout and Schematic Designs

*Status* changes in the Allegro Pulse Status window. You can also view the schematic version available for update.

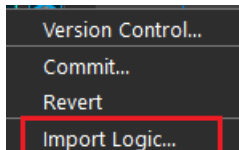


Allegro Pulse Status

Field	Value
Connection State	Connected
Platform State	Running
Platform Mode	Individual
Login	
Lock State	Unlocked.
Design Owner(s)	
Version Status	Design is up-to-date.
Design Version	1.1
Has Write Permissions	True
Schematic Sync Status	Schematic update available.
Linked Schematic	ref_5g:1.1 ( Latest : ref_5g:1.2)
Pending Operations	0

View Pending Operations Close

To import the latest schematic updates, right-click the Pulse icon and select *Import Logic*.

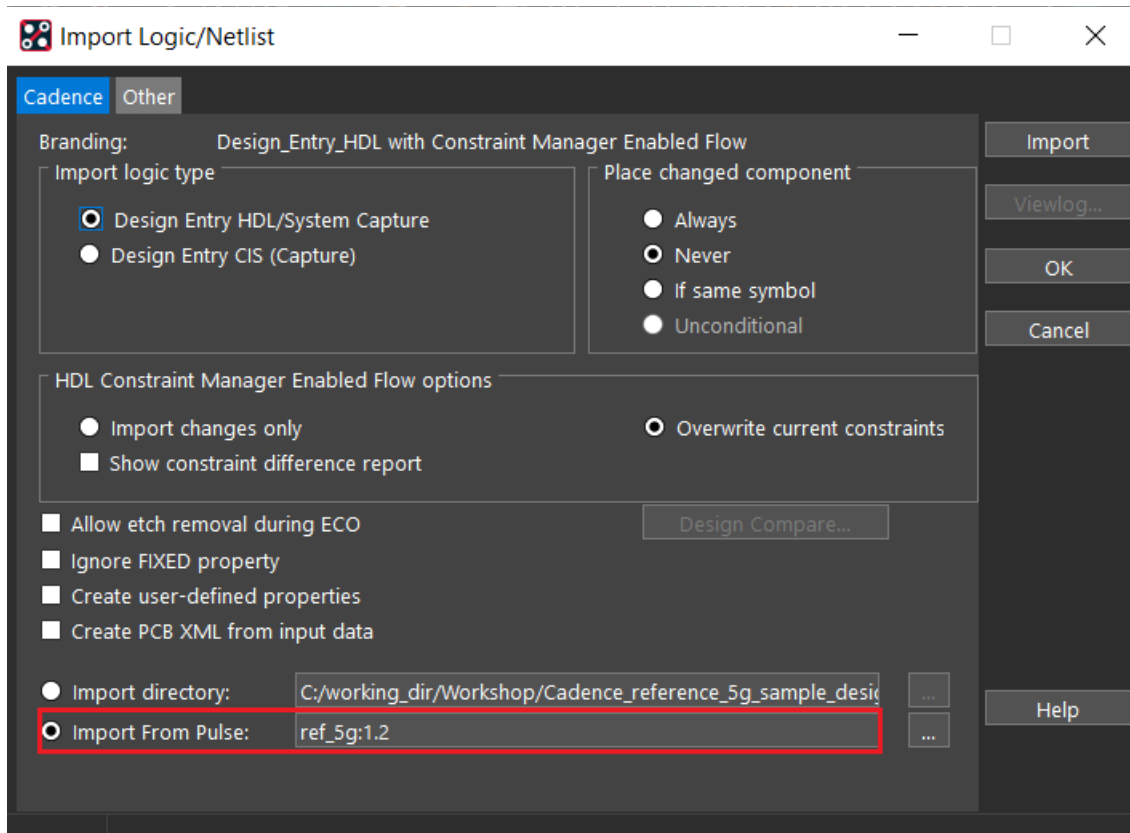




## Working in the Allegro X Pulse Environment

### Working with Pulse-Linked Layout and Schematic Designs

The *Import Logic/Netlist* dialog box opens and the *Import From Pulse* field is auto-populated with the latest version of the schematic. Click on *Import* to bring in the latest update from the schematic.



**Note:** The imported schematic is not instantly linked to the board. As a result, the data displayed by Allegro Pulse Status and Version Control differs. They are only synced after a commit is performed on the board.

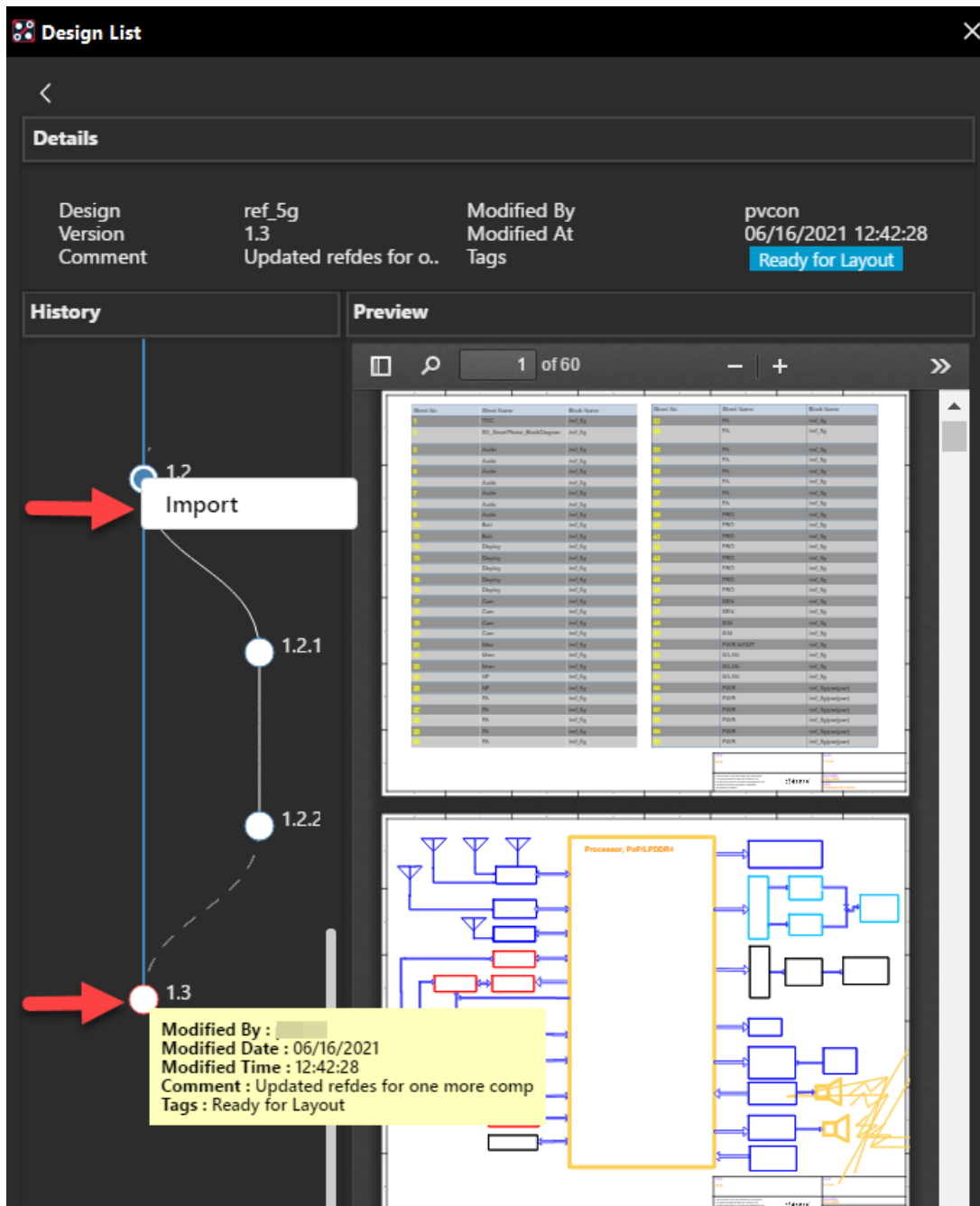
### Synchronizing Layout With the Previous Version of Schematic

Pulse stores all the versions of the schematic labeled as *Ready for Layout* tags. If required, the layout designer can sync the current version of the layout with a previous version of the schematic.

## Working in the Allegro X Pulse Environment

### Working with Pulse-Linked Layout and Schematic Designs

Open the Import Logic/Netlist dialog and click to browse for *Import From Pulse*. In the Design List window, choose a previous version of the schematic. In the version log under the *History* tab, select the version node and right-click to choose *Import*.



After import, the layout database switches to the version that was associated with the schematic just imported. This layout is, however, out-of-sync with the current version of the schematic.

## Leveraging Pulse-Managed Module Design for Board Creation

The Pulse-managed module design approach offers several advantages to PCB design teams. This approach divides the design effort among team members by enabling schematic and layout engineers to work on a project concurrently. It also offers the possibility to reuse a schematic block many times in a design. Additionally, it enables multiple layout designers to work simultaneously on different modules. If there are any changes to the module logic or layout after they are placed in a board design, these modules can be updated to the latest versions from the Pulse.

The managed module design approach facilitates faster time-to-market by dividing the design process into manageable steps. It makes it easier to detect and correct errors at earlier stages of the design cycle.

The managed modular design flow consists of the following steps:

1. Create a block in Allegro System Capture.
2. Upload block netlist to Pulse during commit.
3. Share the block with the layout designer.
4. Create a module layout in the Allegro layout editor by importing the block netlist from Pulse.
5. Commit the module.
6. Share the module layout with the board designer.
7. Import the schematic design netlist and place modules to create the board design.
8. Manage modules in the board using Pulse Module Manager.

For details on exporting and committing block-level netlists to Pulse, refer to [Pulse-Managed Block Reuse for Physical Designs](#).

## Creating Modules for Pulse-Managed Schematic Blocks

To create a new module that has been marked for reuse in the schematic, perform the following steps:

1. Create a new module using *File – New*.

Alternatively type `new` in the Command window.

## Working in the Allegro X Pulse Environment

### Working with Pulse-Linked Layout and Schematic Designs

---

The *New Drawing* dialog box opens.

2. Browse to a directory location to save the module.
3. Specify the name of the module.
4. Click *OK* in the *New Drawing* dialog box.

An empty module is created.

5. Import the module netlist using *File – Import – Logic/Netlist*.

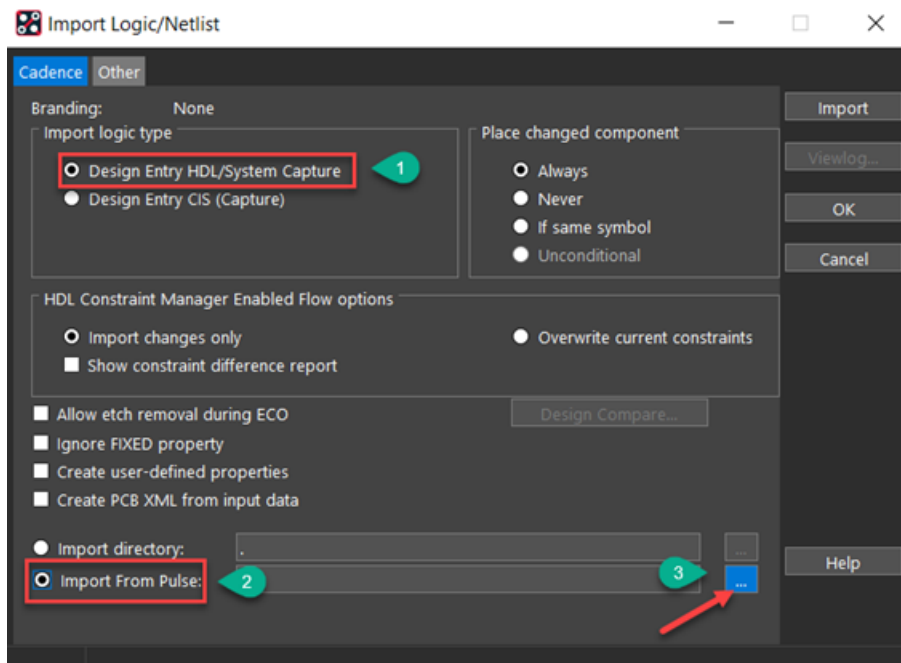
The *Import Logic/Netlist* dialog box opens.

6. In the *Import Logic/Netlist* dialog box, do the following:

- a. Open the *Cadence* tab.
- b. Enable *Design Entry HDL/System Capture* radio button.

A new option *Import From Pulse* is added to the bottom of the dialog box.

- c. Enable *Import From Pulse* and click the browse button to open the *Design List* dialog box that shows all the blocks that netlists are uploaded to Pulse.

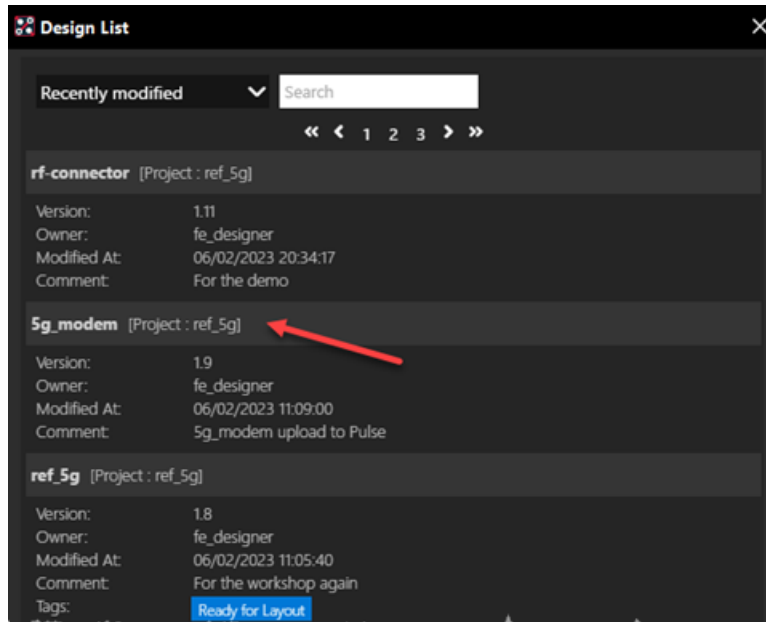


## Working in the Allegro X Pulse Environment

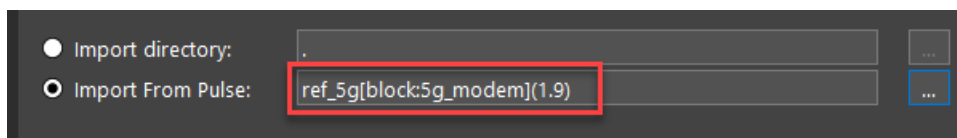
### Working with Pulse-Linked Layout and Schematic Designs

---

- d. In the *Design List* dialog box, select the schematic block for importing its netlist.



The *Import From Pulse* field displays the name and the version of the schematic design.



- e. Click the *Import* button.
7. Place the components of a module using any of the methods available in layout editor. To manually place the components, do the following:
- Click *Place – Manually*.
- The *Placement* dialog box opens.
- In the *Placement List*, select *Component by refdes*.
  - Select components from the list and place them in the design canvas.
8. Commit the module layout using *File – Commit*.

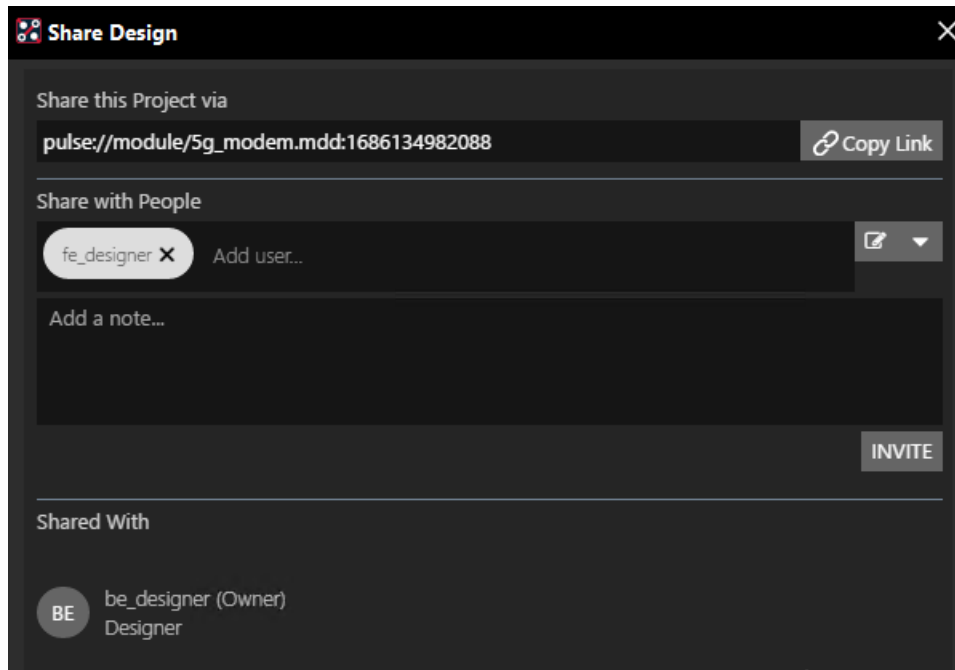
The module is saved and uploaded into Pulse for design management.

## Working in the Allegro X Pulse Environment

### Working with Pulse-Linked Layout and Schematic Designs

---

9. Optionally, click the *Pulse\_Share* toolbar icon to share the module in the Pulse with the designer creating the top-level board. Skip this step if the module and board layouts are being created by the same designer.



## Creating Board Using Pulse-Managed Modules

To create a board design using version-managed modules, the board designer must do the following:

1. Open Allegro layout editor and choose using *File – New* to create a new board.

Alternatively type `new` in the Command window.

The *New Drawing* dialog box opens.

2. Browse to a directory location to save the board.
3. Specify the name of the board.
4. Click *OK* in the *New Drawing* dialog box.

An empty board file is created.

5. Import the schematic design netlist using *File – Import – Logic/Netlist*.

The *Import Logic/Netlist* dialog box opens.

## Working in the Allegro X Pulse Environment

### Working with Pulse-Linked Layout and Schematic Designs

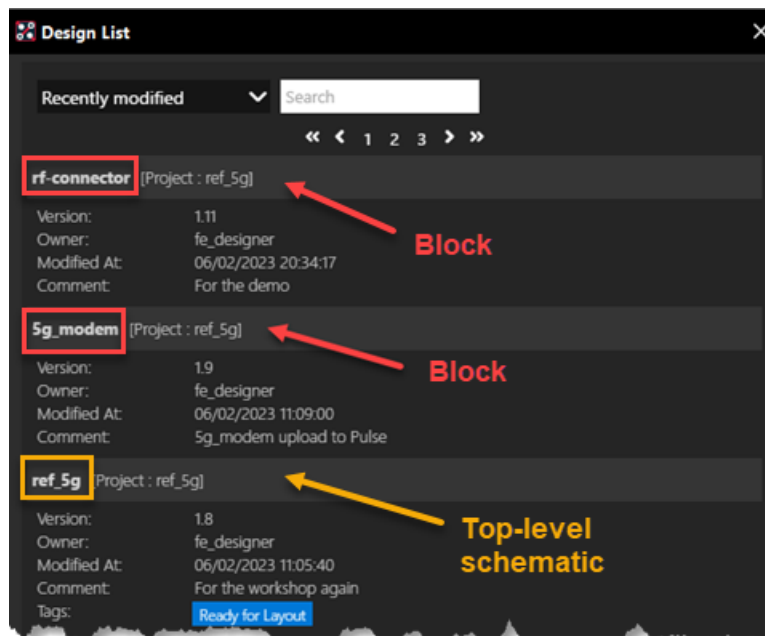
---

6. In the *Import Logic/Netlist* dialog box, do the following:

- a. Open the *Cadence* tab.
- b. Enable *Design Entry HDL/System Capture* radio button.

A new option *Import From Pulse* is added to the bottom of the dialog box.

- c. Enable *Import From Pulse* and click the browse button to open the *Design List* dialog showing list of schematic designs uploaded to the Pulse.
- d. In the *Design List* dialog box, select the latest version of the top-level schematic design that is available in the Pulse.

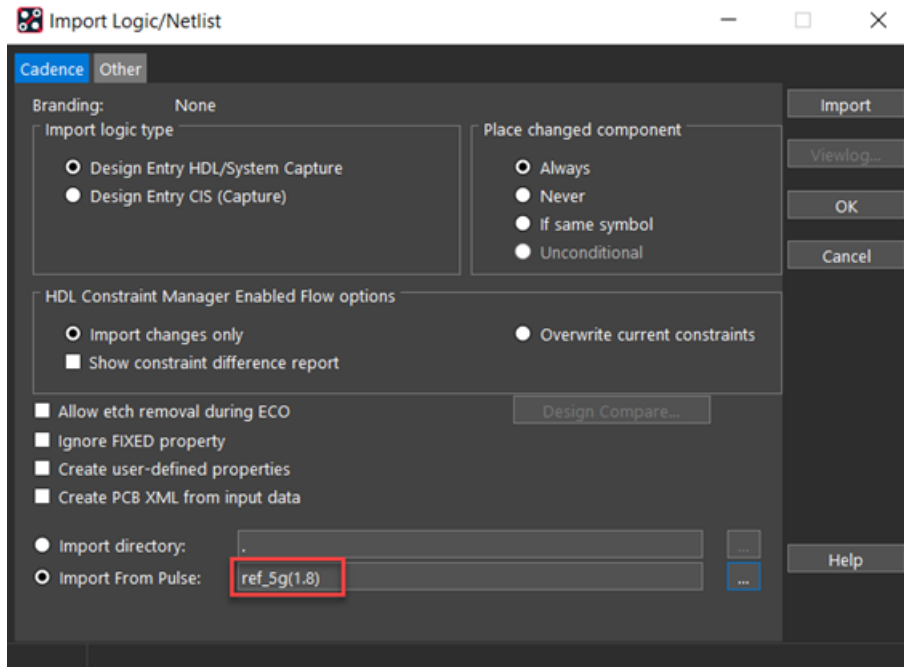


## Working in the Allegro X Pulse Environment

### Working with Pulse-Linked Layout and Schematic Designs

---

- e. Click the *Import* button.



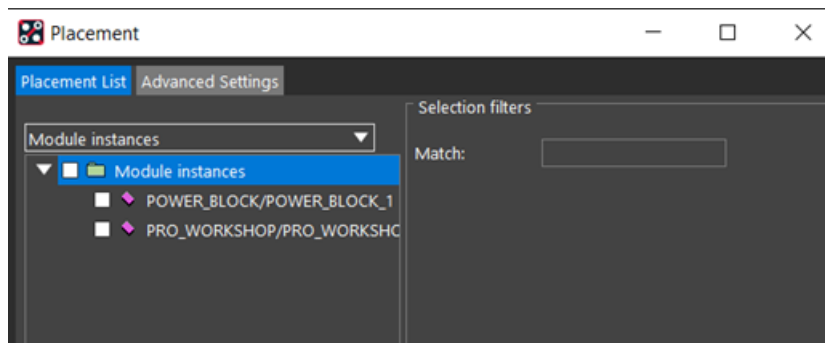
7. To place the modules, do the following:

- a. Click *Place – Manually*.

The *Placement* dialog box opens.

- b. In the *Placement List*, select *Modules Instances*.

The list automatically displays Pulse-managed modules that are shared with the designer.



**Note:** Pulse-managed modules take precedence over the same name modules found in the *modulepath* variable.



## Working in the Allegro X Pulse Environment

### Working with Pulse-Linked Layout and Schematic Designs

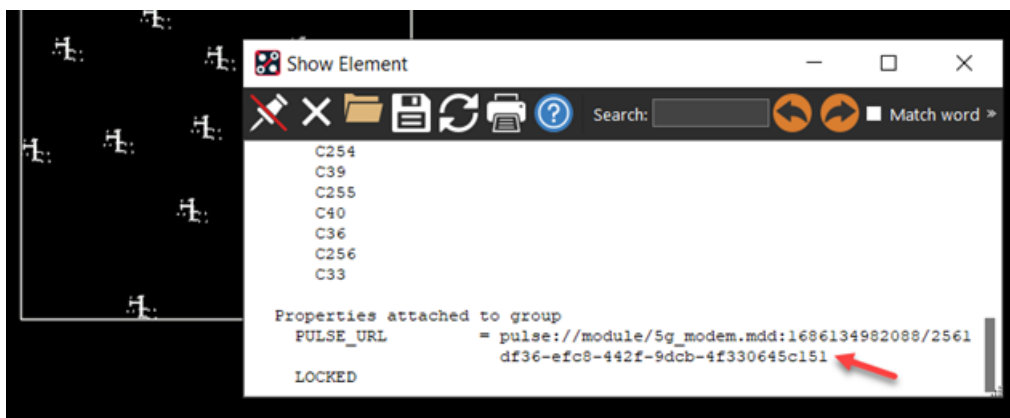
---

- a. Select a module from the list.

The the latest version of the module is loaded from the Pulse database and attached to the cursor.

- b. Click in the design canvas to place the module.
- c. Right-click the placed module and choose *Show Element*.

A PULSE\_URL property is attached to the module indicating that module comes from the Pulse.



- d. Select another module from the list and place in the design canvas.

**Note:** When a module that already exist in placed in the board design, the same version is loaded from the Pulse regardless of whether a newer version is available. After the placement, all module instances can be updated to the most recent version using Pulse Module Manager.

8. Place all the modules and commit the board design.
9. Optionally, click the *Pulse\_Share* toolbar icon to share the board in the Pulse with the top-level schematic designer. This step is required when changes made in the layout are to be back-annotated to the schematic using the *Ready For Schematic* tag.
10. This is still optional. Maybe clarify the layout designer would share with the top-level schematic designer here. The only reason this would be required is if the layout designer wanted to back-annotate to the schematic using Ready For Schematic tag

## Managing Modules in Layout with Pulse Module Manager

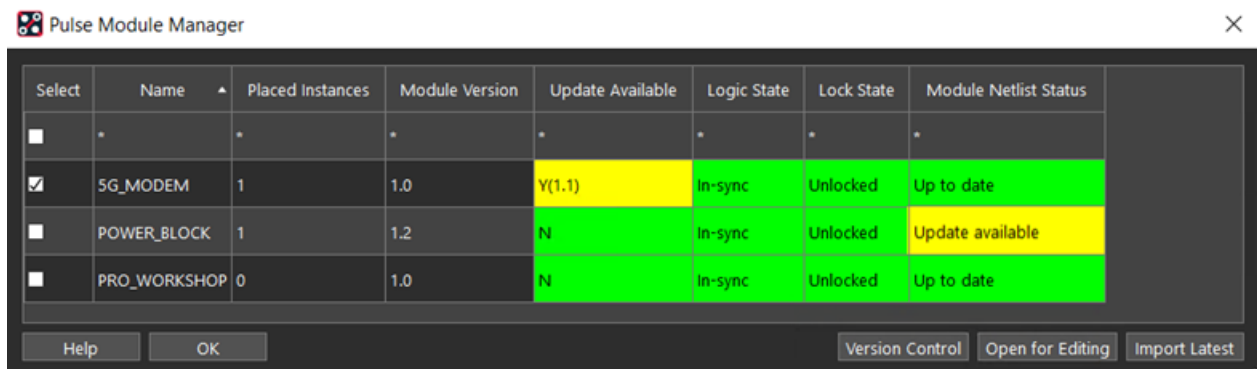
Modules placed in a board design that are marked for reuse in the schematic design can be managed using the *Pulse Module Manager* dialog box.

## Working in the Allegro X Pulse Environment

### Working with Pulse-Linked Layout and Schematic Designs

#### 1. Choose *Place – Pulse Module Manager*.

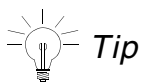
Alternatively, type `pulse_reuse_module_mgr` in the Command window.



The *Pulse Module Manager* displays list of all the version-manged modules related to the active board design. It also displays the count a module is instantiated in the layout and its status with respect to schematic and Pulse. If the module is in-sync with its schematic or Pulse version the status appears in green.

#### 2. Optionally, select a module and click *Version Control* button to rollback the version of the module with any of its previous versions available in the Version Control tree.

As a result the *Update Available* cell turns yellow and displays the latest version available in the Pulse.



#### Tip

Close and reopen *Pulse Module Manager* to view the latest status of modules.

#### 3. Optionally, select the module and click *Open for Editing* to launch a separate session of the layout editor for modifying the module database.

**Note:** Ensure to lock the module in Pulse and perform the Commit operation once you modify the module.

#### 4. Perform ECO operations on the module in the *Pulse Module Manager* as follows:

User Action	Status	Action Required
Layout driven: Module (.mdd) was modified in the layout editor and changes were committed to the Pulse.	<i>Update Available</i> cell turns yellow displaying the latest version of the module from the Pulse.	Click <i>Import Latest</i> to update the placed module instance with the latest version of the module that was last committed in Pulse.

## Working in the Allegro X Pulse Environment

### Working with Pulse-Linked Layout and Schematic Designs

---

Schematic driven:

Schematic block of the module was modified in System Capture. It's netlist was exported and changes were committed to the Pulse.

*Module Status Netlist* cell turns yellow and status changes to *Update Available*.

a. Click *Open for Editing* to update the module layout by reimporting the latest netlist, lock the module in Pulse, and commit the module.

b. Reload the *Pulse Module Manager* in the board design.

The *Update Available* cell turns yellow for the module and shows the last committed version in Pulse.

c. Select the module in *Pulse Module Manager*. Click *Import Latest* to update the module instance in the board design.