cadence®

Migration Guide for Allegro® X Platform Products

Product Version 23.1
September 2023

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

Contents

1

Allegro Platform Front-End Products

Designs created using the following products in release 22.1 are fully compatible with release 23.1 and no changes or upgrades are required:

- Allegro System Capture
- Allegro Design Entry HDL
- System Connectivity Manager
- Design Entry CIS
- Allegro PSpice Simulator

If you plan to migrate from an earlier release, refer to *Migration Guide for Allegro Platform Products - Release17.4-2019*.

2

Allegro X Platform Core Back-End Products

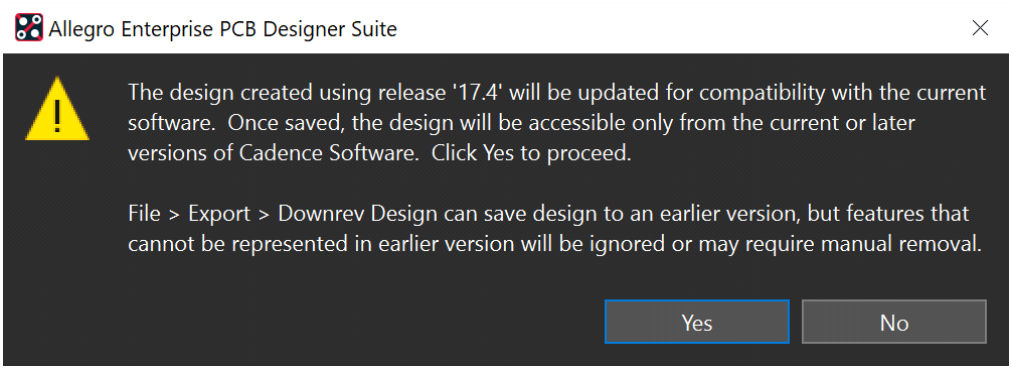
The following sections describe new or modified functionalities that impact existing designs or workflows for all the Allegro X layout and substrate products: Allegro X PCB Editor, Allegro X Advanced Package Designer (APD), and PCB SI. It contains the following topics:

Migration Impact on Database Compatibility

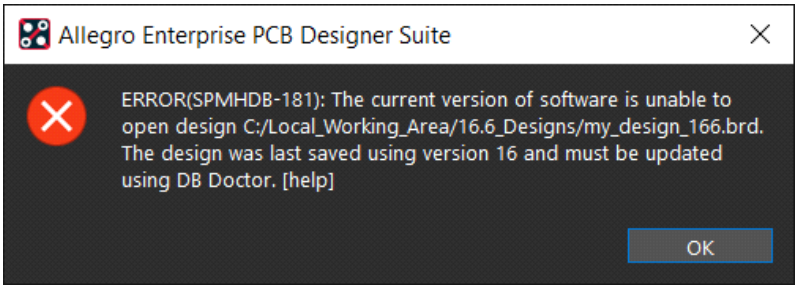
The following topics explain how database compatibility is impacted when migrating Allegro X layout editors to 23.1:

Design Database Compatibility

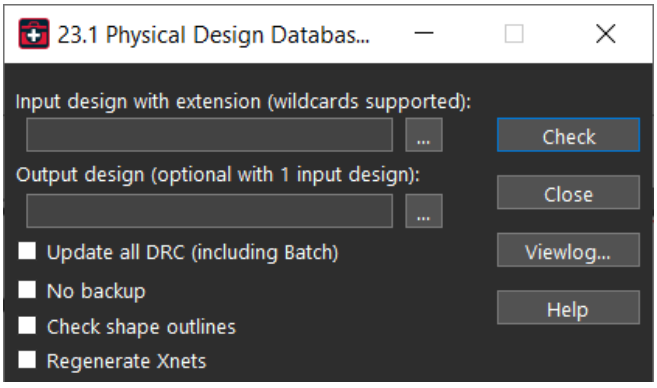
A layout design created in release 17.2 or later, when opened in release 23.1, is automatically upgraded to release 23.1 and saved in the 23.1 database format.



A design created in release 16.6 first needs to be upgraded before it can be opened in release 23.1.

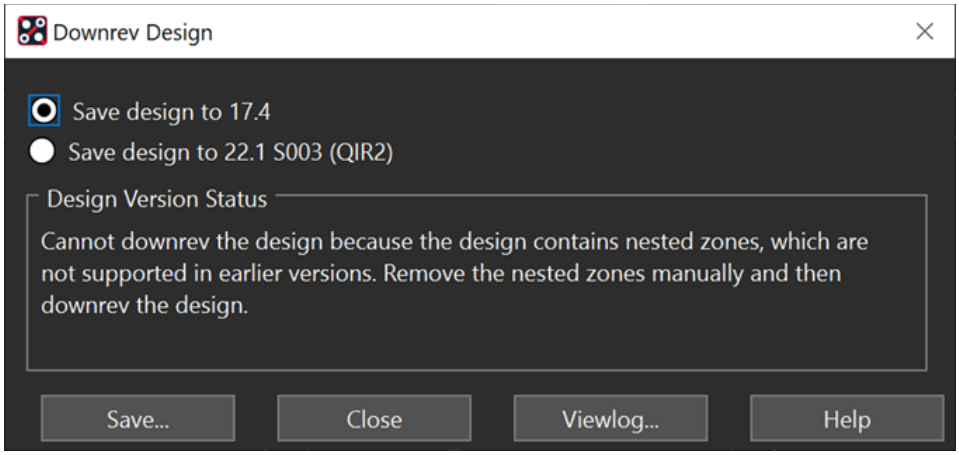


Use the PCB DB Doctor utility (`dbdoctor_ui.exe`) to upgrade the design to the 23.1 database format.



To roll back a 23.1 design database to an earlier version, use the `downrev` command or choose *File – Export – Downrev Design* from the main menu.

By default, the rollback operation saves the design to the 17.4 database format but if the design contains nested zones, it can only be saved to the 22.1 S003 (QIR2) database format.



Library Database Compatibility

Libraries created in release 17.2 or later can be used in 23.1 without any updates.

However, to use Allegro libraries created in release 16.6, you need to upgrade them to the 23.1 format using the PCB DB Doctor utility (`dbdoctor_ui.exe`).

To determine the database version of any symbol and padstack used in the library, use the batch DBSTAT command:

```
C:\My_EDA_Libraries\> dbstat  soic48w.dra
          soic48w.dra:  17.2 NT
```

Additionally, you can use the following batch DB Doctor commands to update libraries with a large number of symbols:

```
dbdoctor <wildcard> (Linux)
dbdoctor <file name> (Windows)
```

By default, the library symbols and padstacks saved in release 23.1 are saved in the 17.2 database format to maintain compatibility with earlier releases. However, in the following scenarios, the symbols and padstack will be saved in higher database formats:

- *Secondary Side Countersink/Counterbore* and *Actual Drilled Hole* values are 23.1 features. When a padstack with these features is saved, it is saved in the 23.1 database format. If these padstacks are used in a symbol, it is also saved in the 23.1 database format.

- *Hierarchal Route and Via Keepout shapes* defined on the *Outer_Layer, Inner_Signal_Layers* and *Inner_Plane_Layers* layers are 17.4 features. A symbol using these features is saved in the 17.4 database format.

Removing these features from symbols and padstacks will automatically save the library to the 17.2 database format.

17.2 Compatibility Mode Removed

The 17.2 compatibility mode is removed from release 23.1. The following environment variables are also removed from the current release:

- `database_compatibility_mode`
- `database_compatibility_new_design`

Migration Impact on Padstack Editor

The migration impact on Padstack Editor includes the following:

- Importing Padstack Data from XML File
- Pad Renamed to Regular Pad in Design Layer of Padstack Editor

Importing Padstack Data from XML File

An XML file with padstack version 1.1 or later can be imported only in release 23.1.

Pad Renamed to Regular Pad in Design Layer of Padstack Editor

The column name, *Pad* is renamed as *Regular Pad* in the *Design Layers* tab to be consistent with other instances in Padstack Editor. Update any script that refers to the old column name to use the new column name.

For example, in your scripts, you need to change:

```
QtSignal GuidedMaskLayersTab LayersTable itemSelectionChanged 1 Pad
```

to:

```
QtSignal GuidedMaskLayersTab LayersTable itemSelectionChanged 1 "Regular Pad"
```

3

Allegro Platform High-Speed Products

This section is valid for all Allegro-based high-speed products.

Designs created in release 22.1, 17.4-2019, or 17.2-2016 are fully compatible with release 23.1 and no migration tasks are required for the Allegro® high-speed products.

Read the Allegro X Platform Core Back-End Products chapter for changes that apply to the Allegro layout and substrate editors.

4

Allegro Pulse and Allegro EDM

This section describes the impact of changed functionality in release 23.1 on existing sites, databases, flows, and designs for Allegro Pulse and EDM, and all the other Pulse and EDM-based products.

- Workflows Migration
- Changes in Library Synchronization User Interface
- Migrating to Release 23.1

Workflows Migration

This release has a new documentation viewer, Cadence® Doc Assistant, which replaces Cadence Help. When migrating to this release, or if your organization uses customized workflows in Allegro X EDM Flow Manager, replace the name of the old help viewer executable, `cdnshelp`, with the name of the new viewer, `cda`. For example:

Old Command	New Command
`cdnshelp -openpage adwuidoc:adwuidoc_firstpage`	`cda -openpage adwuidoc:adwuidoc_firstpage`
`cdnshelp -search "dbadmin"`	`cda -search "dbadmin"`

Related Documentation

Customizing Flow Steps and Buttons

Changes in Library Synchronization User Interface

22.10 Library Synchronization for PTC Windchill	23.10 Library Synchronization for PTC Windchill
No *Preferences* page	*Preferences* page available
Container and *Soft Type* fields were in the *Connection* page	The fields have moved to the *Preferences* page. The *Container* Field name is now *Default PLM Container.* The *Soft Type* field name is now *Default PLM Part Type.*
New Part Request-related settings for PTC Windchill were in the *Update PLM* page.	New Part Request-related settings are now in the *Preferences* page for PTC Windchill. The *Update PLM* and *Update Pulse* pages now have only task configuration options.

Migrating to Release 23.1

If you work in a single-user Allegro System Capture environment, no migration tasks are required to move from release 22.1 to release 23.1.

If your company has a setup where designers connect to a remote Pulse server, ECAD or IT administrators need to migrate the Pulse server cluster. See Migrating Pulse Server Cluster for details.

Depending on your configuration, you need to perform the following, additional migration tasks:

- Migrating Unmanaged Libraries Indexed by Pulse
- Migrating Multi-Library Release Server

Migrating Pulse Server Cluster

When migrating to 23.1, Pulse runs a service that extracts all the System Capture designs stored in the Pulse server. This enables design data indexing.

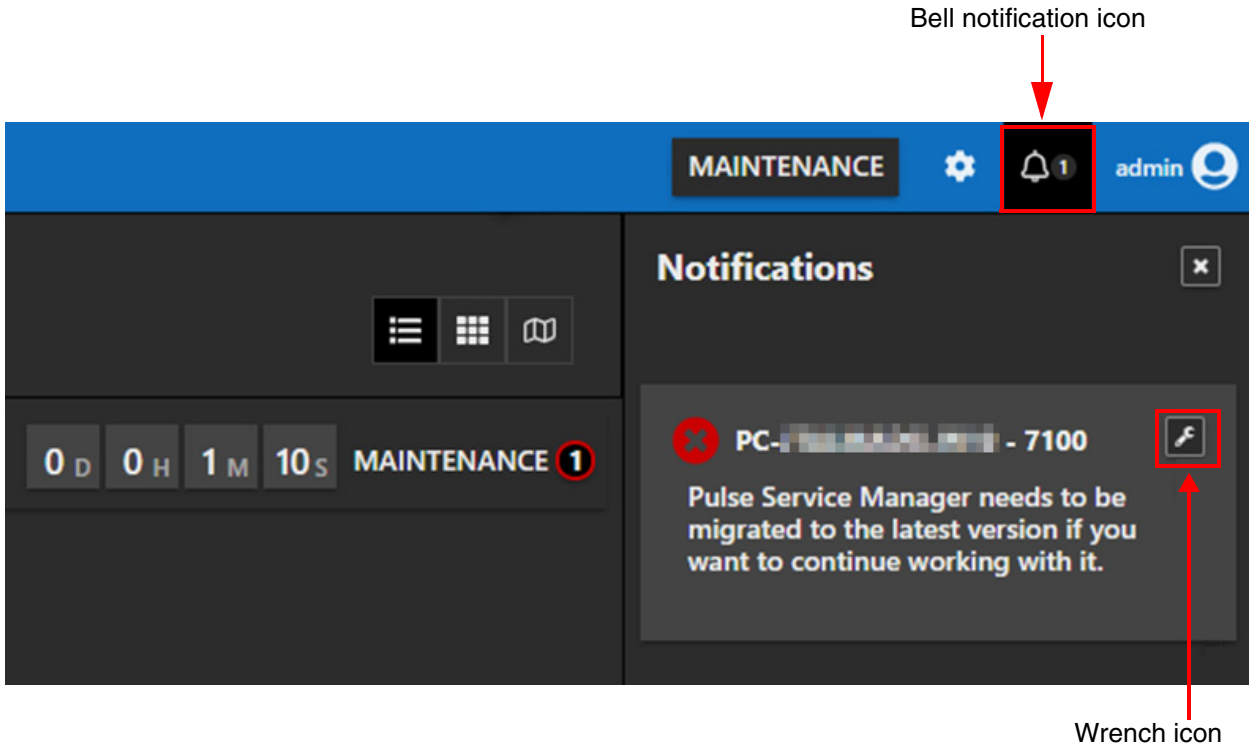
This task might take some time, which depends on the amount of data to be extracted. During the extraction process, the PDFs of newly saved and committed designs might be unavailable to designers in *Version Control*. To ensure smooth operations, it is recommended that Pulse administrators migrate the cluster during a scheduled downtime.

To migrate the Pulse server cluster, do the following:

1. Shut down the 22.1 Pulse primary node, if it is running.
2. Install release 23.1.
3. Start the 23.1 Pulse primary node.

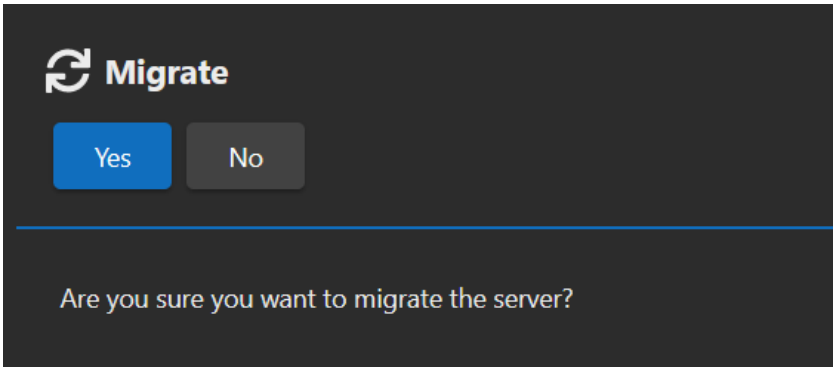
   To avoid migrating each node individually, start Pulse Service Manager on all nodes before you begin migrating to release 23.1.

   When the Pulse primary node starts, the Pulse Service Manager web page is displayed.
4. Log in to Pulse.
5. Click the bell notification icon.



   A message is displayed prompting you to migrate to the latest version.
6. Click the wrench icon.

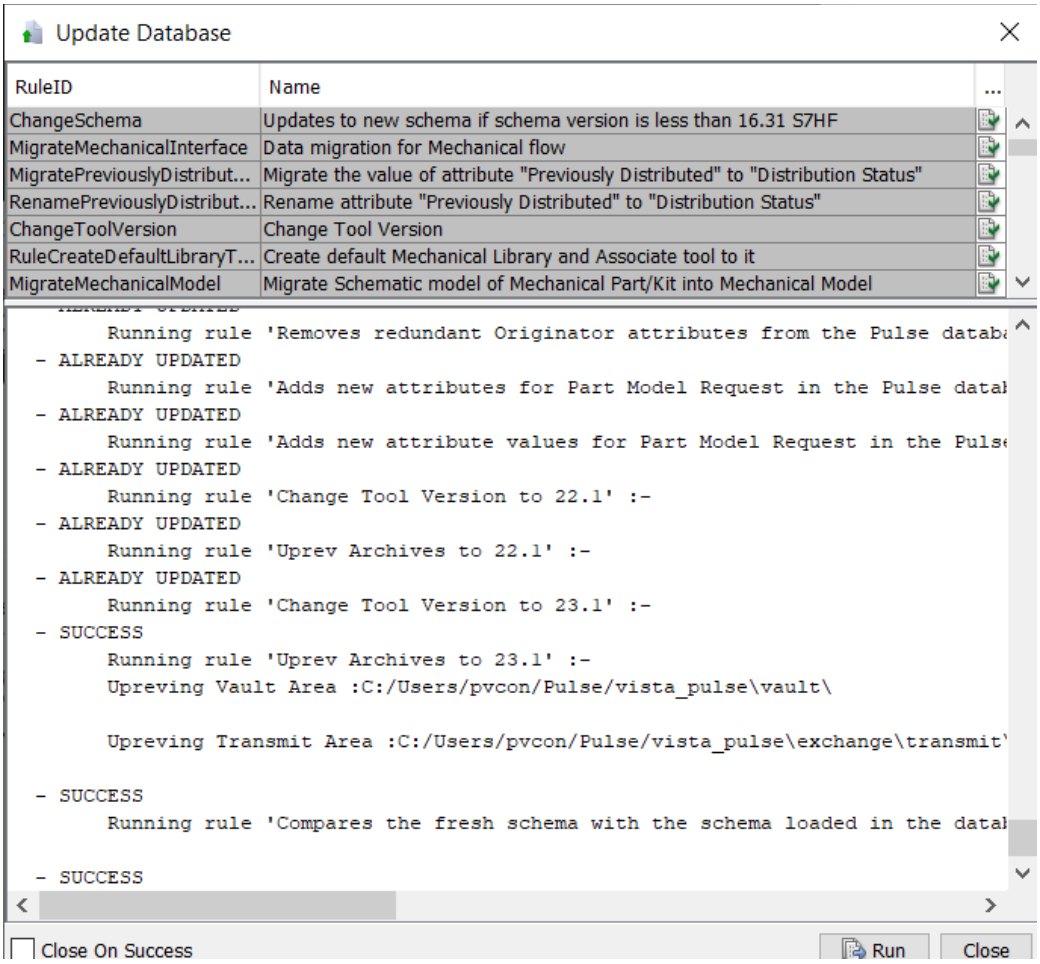
You are prompted to migrate.



7. Click *Yes* to migrate to the latest version.
8. Complete the following tasks to carry over any custom scripts or flows:
   a. Copy the 22.1 `<Allegro EDM Conf Root>` directory to the 23.1 setup.
   b. Modify custom scripts and flows as required.

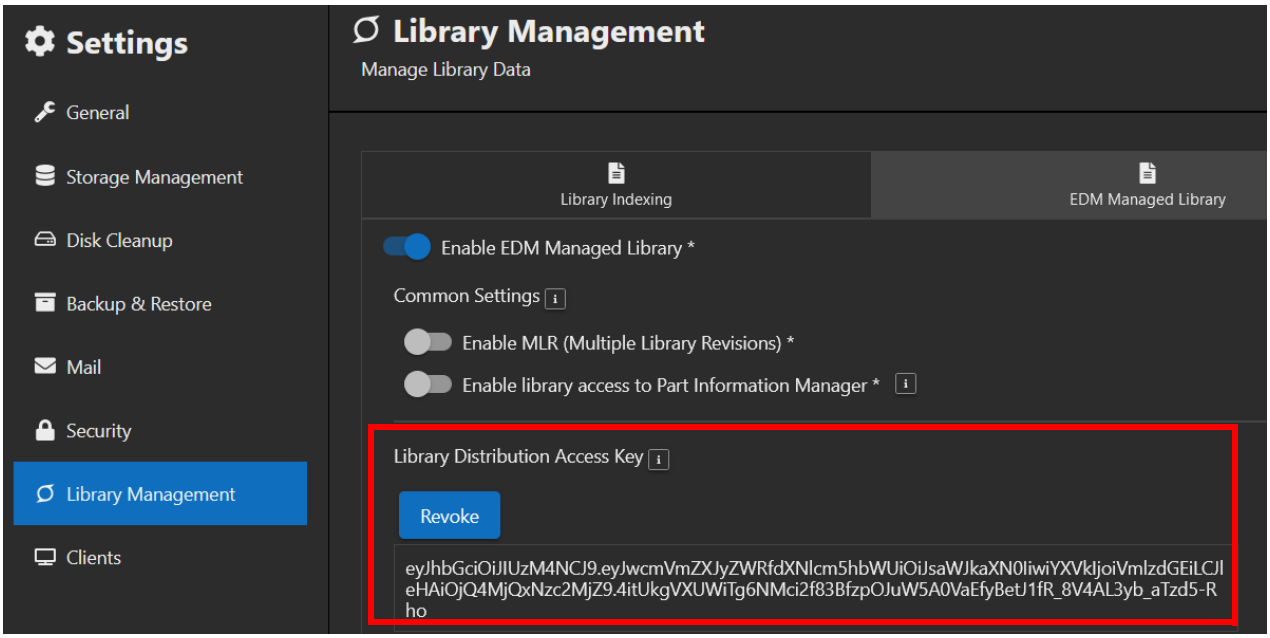
      For example, the `<company>_flow.env` file might have references to release 22.1. Modify these to 23.1.
9. Navigate to the location of the 23.1 `<startworkbench>.bat` file and open a Command Prompt window.
10. Run the batch file with the `adw_uprev` argument to ensure that the database schema is up to date with the latest release.

The `adw_uprev` utility indicates whether the process is successful. A further indication of a successful update is that there will be no uprev prompt when you launch Allegro EDM Database Editor or Allegro EDM Database Administration.



11. If you revoke an existing library distribution access key and generate a new access key, provide the regenerated key to all the designers who connect to the Pulse server.

Copy the value of the library distribution access key from the *Library Management* pane of the Pulse Service Manager page.



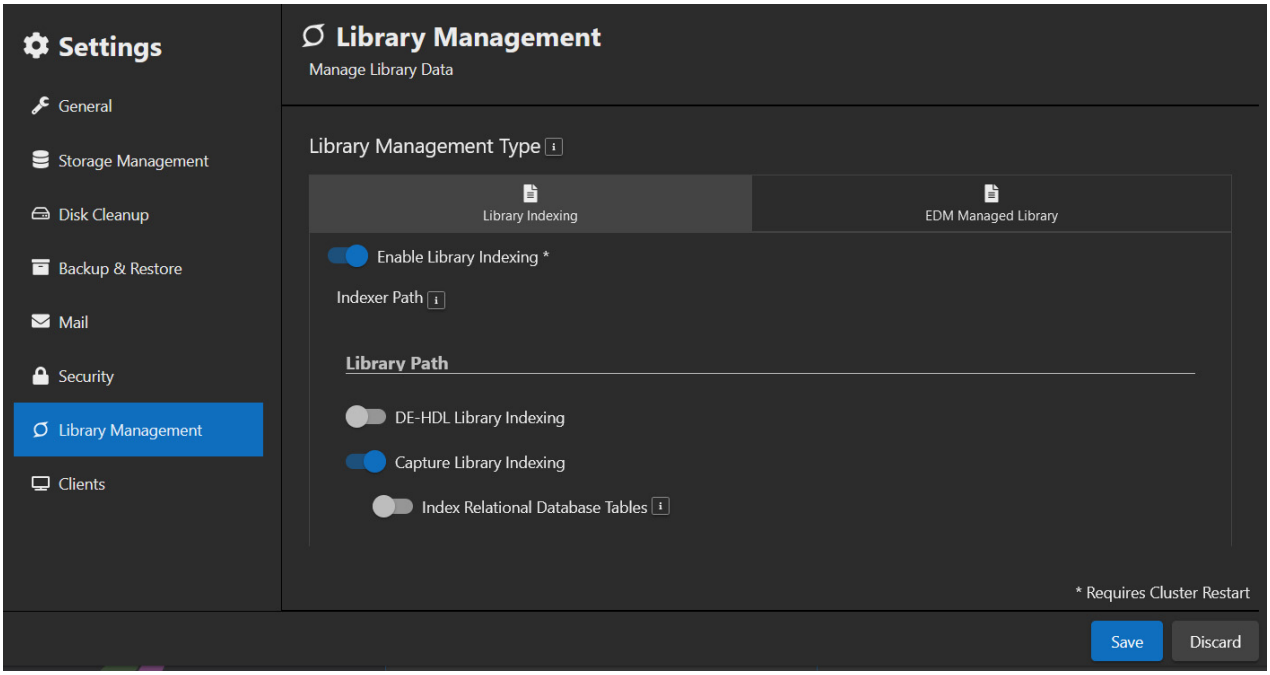
Migrating Unmanaged Libraries Indexed by Pulse

If you work with Design Entry HDL (DE-HDL) libraries that are not managed but indexed by Pulse in 22.1, no migration tasks are required to move to 23.1.

However, to migrate OrCAD® X Capture CIS libraries that are not managed but indexed by Pulse, do the following:

1. Navigate to *`$CDS_SITE`*`/cdssetup/OrCAD_Capture`.
2. Rename the `22.1.0` folder as `23.1.0`.
3. From the `23.1.0` folder, open `capture.ini` in a text editor.
4. Change `Version=22.1-2022` to `Version=23.1-2023`.
5. Access the Pulse Service Manager Settings page.

6. Select *Library Management* in the left pane.



7. Select *Enable Library Indexing*.
8. Specify *Library Path* as *`$CDS_SITE`*.

   The sub-directory that contains `capture.ini` must be under the *`$CDS_SITE`* directory.

   Note: It is recommended that you run OrCAD® X Capture CIS on the Pulse server, configure the libraries and ODBC connection to the CIS database, and then specify the resulting `capture.ini` in the Pulse server settings.

Important

OrCAD X library indexing is only supported on Windows.

Pulse uses the library location path to index and access all the configured libraries for parts.

9. Click *Save*.

Migrating Multi-Library Release Server

If all designers in your setup use Allegro System Capture, you can skip this section. It is expected that all System Capture designers point to the same Pulse remote server, which should ideally be on the latest release.

Read this section if your setup includes designers working on Design Entry HDL, and you have a multi-library release (MLR) server. In such a case, you can either:

- continue working with the MLR server, or
- migrate the MLR server to the latest release

The following tables list the possible MLR server setups, and the `client` in the tables refers to Design Entry HDL:

Possible MLR Server Setups

Existing Setup for Librarians	Existing Setup for Designers	Migration Tasks for Librarians	Migration Tasks for Designers	See...
17.4 library server	The client is on 22.1 and points to a 17.4 library server	Migrate the 17.4 library server to a 22.1 library server	The client is on 23.1 and points to a 22.1 library server	*Migration Guide for Allegro Platform Products - Release17.4 -2019.*
22.1 library server with 22.1 library data	The client is on release 22.1	Migrate the 22.1 library server to the 23.1 library server and update the data to 23.1	The client points to the 23.1 library server	Moving to the Latest Release
22.1 MLR server with 17.4 library data	The client points to the 22.1 MLR server	Migrate the 22.1 MLR server to 23.1 and update the library data to 23.1	The client points to the 23.1 MLR server	

Possible MLR Server Setups

22.1 MLR server with 17.4 library data	The client points to the 22.1 MLR server	Migrate the 22.1 MLR server to 23.1 and retain the library data on 17.4	The client points to the 23.1 MLR server	

Moving to the Latest Release

To migrate from the MLR server to a library server setup with the latest version, do the following:

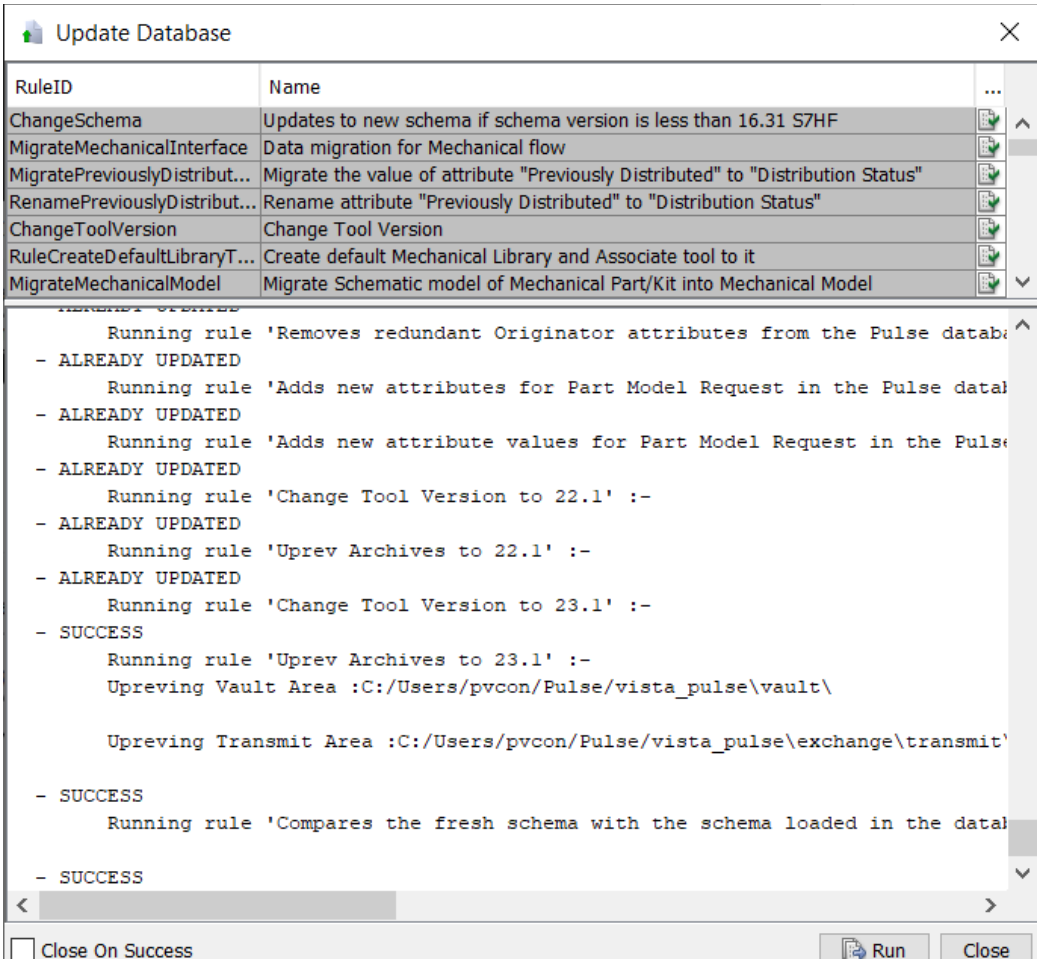
1. On the Pulse data nodes, set the remote URL value and point the data nodes to the Pulse primary node.
2. From the current primary library server, copy the contents of `$PCBDW_LIB_MLR/vault/model_*` to `$PCBDW_LIB/vault/model_*` of the 23.1 Pulse primary node.
3. From the current primary library server, copy the contents of `$PCBDW_LIB_MLR/exchange/transmit/model_*` to `$PCBDW_LIB/exchange/transmit/model_*` of the 23.1 Pulse primary node.
4. Edit `$PCBDW_LIB/distribution/env/lib_dist.ini` and ensure that the following values are set:
   - `genmodelhtml = on`
   - `mkdump = on`
   - `adwserver_install = off`
5. Edit `$PCBDW_LIB/distribution/env/fetch_dump.ini` and modify:

   ```
   urlRoot = file:///$env($PCBDW_LIB_MLR)/distribution/html/index.html
   ```

   to

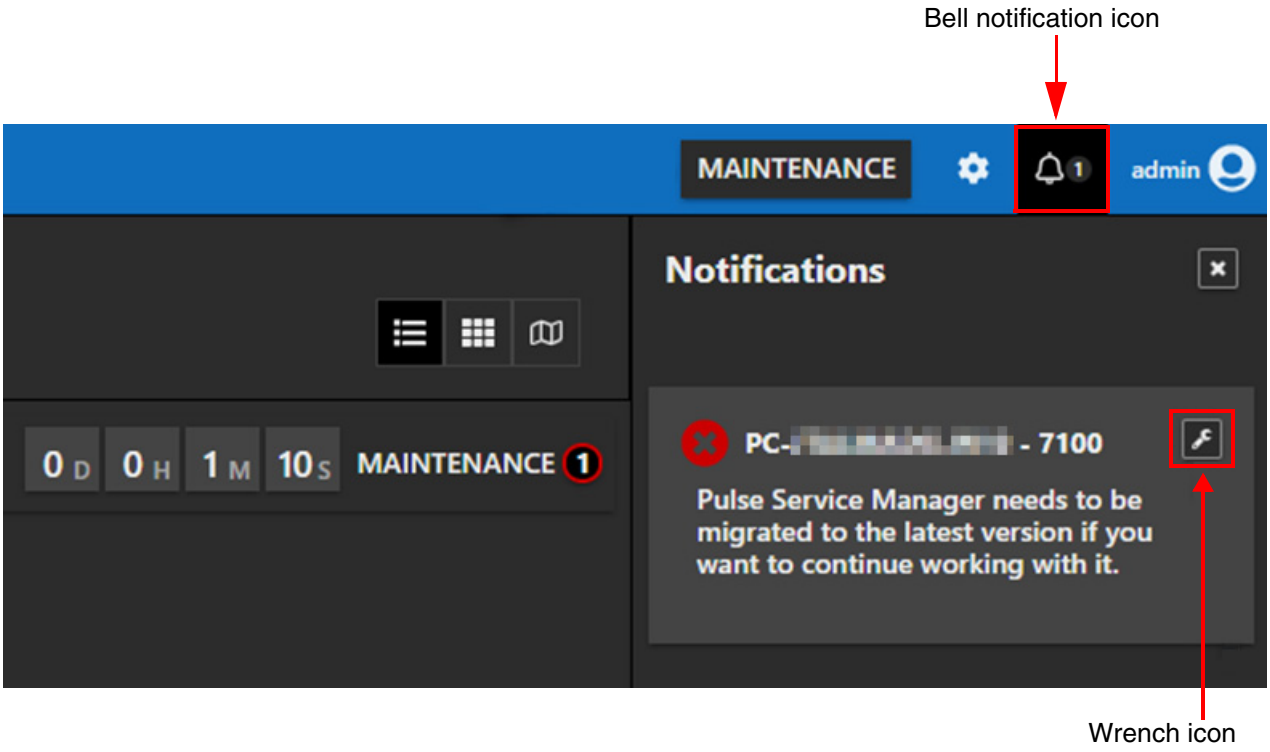
   ```
   urlRoot = file:///$env(PCBDW_LIB)/distribution/html/index.html
   ```
6. Navigate to the location of the 23.1 *<startworkbench>*`.bat` file and open a Command Prompt window.
7. Run the batch file with the `adw_uprev` argument to ensure that the database schema is up to date with the latest release.

The `adw_uprev` utility indicates whether the process is successful. A further indication of a successful update is that there will be no uprev prompt when you launch Allegro EDM Database Editor or Allegro EDM Database Administration.



8. Open the Pulse Service Manager page.

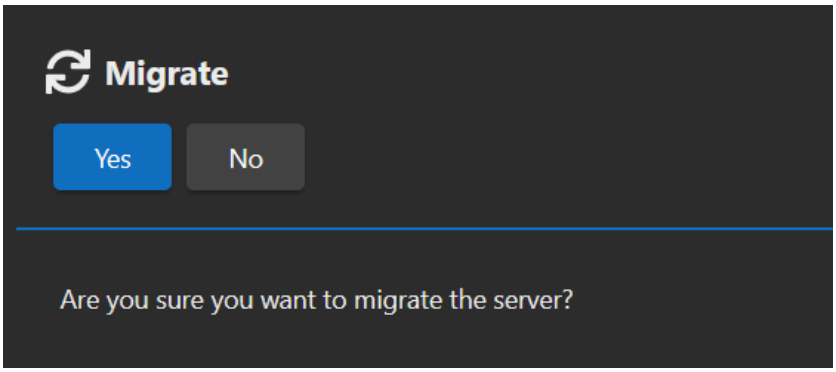
9. Click the bell notification icon.



A message is displayed prompting you to migrate to the latest version.

10. Click the wrench icon.

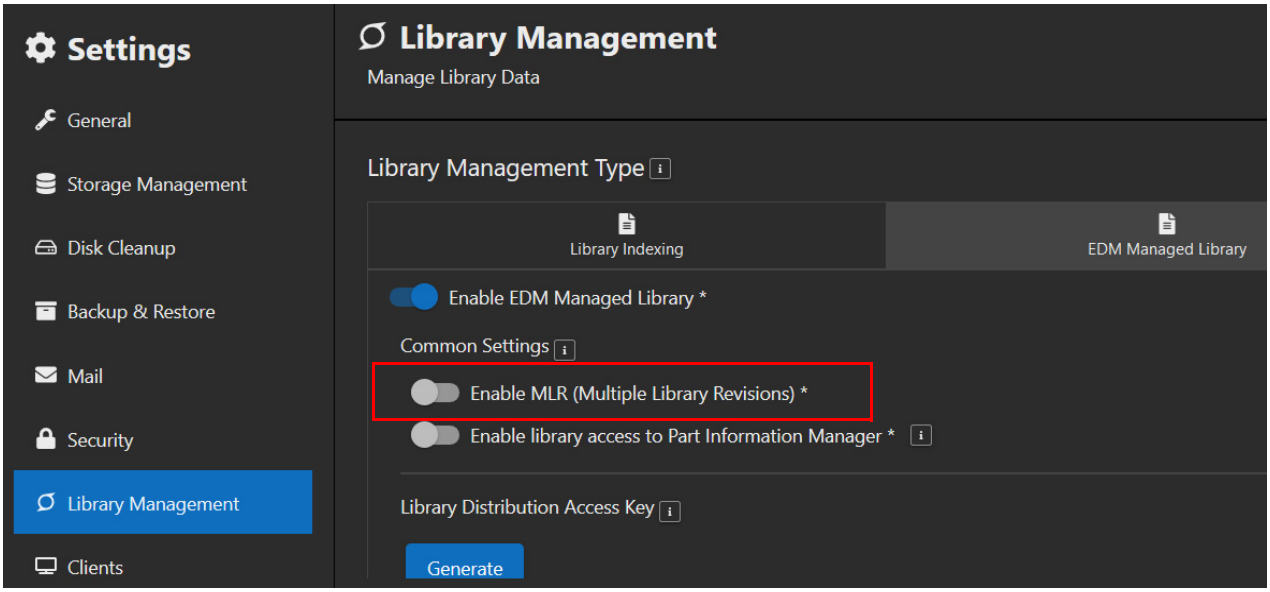
You are prompted to migrate.



11. Click *Yes* to migrate to the latest version.

12. Open the Pulse Service Manage *Settings* page.

13. Click *Library Management*.
14. Ensure that *Enable MLR (Multiple Library Revisions)* is deselected.



15. Navigate to the location of the 23.1 `<startworkbench>.bat` file and run the batch file.

    The Allegro EDM prompt is displayed.

16. Enter the `lib_dist` command at the Allegro EDM prompt.

    The PCB Editor models are re-installed and updated to the 23.1 model version on the 23.1 Pulse primary node.