cadence®

Allegro® EDM
Library Import User Guide

Product Version 23.1
September 2023

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

Contents

Preface

About This Guide

The *Allegro® EDM Library Import User Guide* explains the library import methodology and the procedures and tasks you need to perform for a successful library import.

Related Documentation

You can also refer the following documentation to know more about related tools and methodologies:

- For information on the new features, see *Allegro EDM: What's New in Release*.
- For learning how to use Database Administrator, see *Allegro EDM Database Administrator User Guide*.
- For learning how to use Database Editor, see *Allegro EDM Database Editor User Guide*.
- For learning how to use Library Distribution, see *Allegro EDM Library Distribution User Guide*.
- For learning how to migrate non-Allegro EDM designs and libraries into Allegro EDM, see *Allegro EDM Migration Guide*.

Related Tools and Flows

- For information on various PCB design working environments such as a team of designers working on a Design Entry HDL project, implementing FPGAs in designs, working with high-speed constraints, importing IFF files for radio-frequency designs, and reusing existing modules, see *Allegro PCB Design Flows*.
- For learning how to create and configure Design Entry HDL projects, see the *Allegro Project Manager User Guide*.

Typographic and Syntax Conventions

This list describes the syntax conventions used for this user guide:

`literal`	Nonitalic words indicate keywords that you must enter literally. These keywords represent command (function, routine) or option names.
`argument`	Words in italics indicate user-defined arguments for which you must substitute a name or a value.
|	Vertical bars (OR-bars) separate possible choices for a single argument. They take precedence over any other character.
`[ ]`	Brackets denote optional arguments. When used with OR-bars, they enclose a list of choices. You can choose one argument from the list.
`{ }`	Braces are used with OR-bars and enclose a list of choices. You must choose one argument from the list.

1

Getting Started with Library Import

Overview

Before you start using Allegro Library Manager, you need to migrate non-Allegro EDM library data into Allegro EDM-compliant libraries.

Generally, this task is only required when you move to the Allegro EDM environment for the first time. Importing non-Allegro EDM libraries into Allegro EDM is known as Library Import, and is usually done by site administrators and librarians.

This chapter explains the library import methodology. Chapter 2, "Working with Library Import," explains the procedures associated with the library import methodology.

Note: To use the design flows in Allegro EDM, design groups also need to migrate design projects into the Allegro EDM environment. This guide only explains the library import process. For information on how to migrate non-Allegro EDM projects into the Allegro EDM environment, see *Allegro EDM Migration Guide*.

Libraries in Allegro EDM

Allegro EDM supports the following:

- Logical design libraries
  - Schematic libraries and related information (e.g., classifications): These libraries are based on a library-cell-view (LCV) structure. Each part (cell) has several views, each of which describes a part in a unique way. This structure, created through Part Developer, must be in the logical library.
    - Schematic models, referred to as ConceptHDL Models in Allegro EDM
    - Block models
    - Standard symbols
  - OrCAD libraries

- Schematic models from `.olb` files

  Note: Temporary and mechanical parts, block models, standard symbols, and PSpice models are not supported.

- Physical design libraries, that is, Allegro PCB Editor libraries: include shapes, padstacks, and symbols (Footprints, Flash, Board, and Drawing), modules.
- Physical part information
  - Part table files (PTFs): These files contain the parametric information for individual parts and indicate the logical and physical symbol to use.
  - PPT Option Set - Physical part numbers and their attribute information for a design project are stored in a Part Table File (PTF). The default physical property settings and definitions for a part stored in a `ppt_optionset.dat` file is called an option set.
  - CIS database part information and the configuration file (`.dbc`)
- Datasheet models: Include various types of datasheet files, which are linked to parts.
- Simulation models - currently, only DML models are supported.

Allegro EDM library data supports additional information, such as versions, manufacturer, status (for example, preferred), and site-specific information. This information is stored in the Allegro EDM component database, and plays a key role in providing detailed part information.

When importing data into Allegro EDM, you might want to address incorrect, incomplete, or missing (e.g., for classification data for PCB Editor models) classification information in the library data that you are importing. You can do this by creating `.csv` files and specifying classification information for any model.

You can also add additional searchable properties in these CSV files, or specify datasheet to part mapping.

Allegro EDM Library Structure

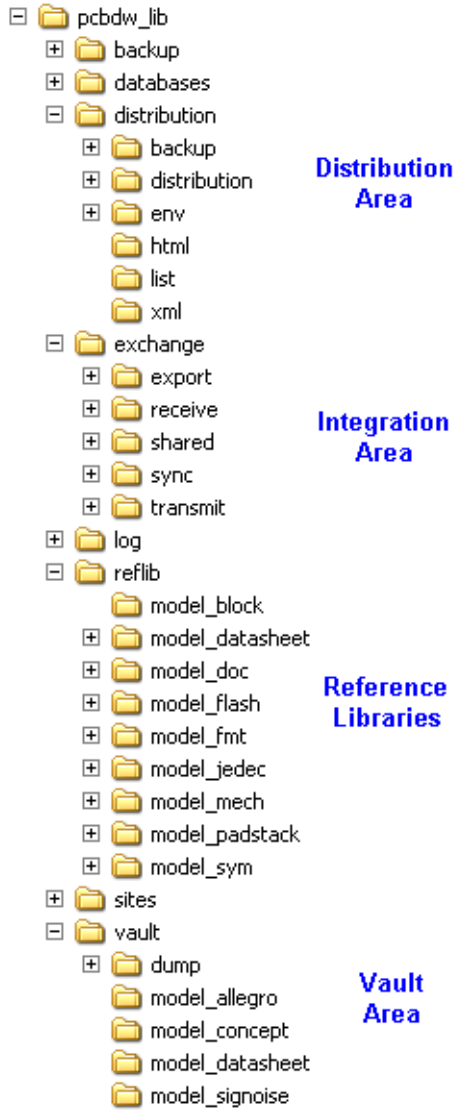
The physical locations of the Allegro EDM libraries follow a multi-tier structure. The directories include the following:

- **Work area**: That represents the work-in-progress area for the librarians. This area is the location of the library projects for library development.
- **Integration area**: That stores all the library components checked in by the librarian. This area is used during library synchronization, authoring, and distribution process.

- **Vault area:** That represents the location where all the archive files of the models are stored with entries of these models available in the Allegro EDM component database.

The `distribution` directory in the Allegro EDM Reference Library is used during the library distribution process. For information on the contents of this directory, see *Allegro EDM Library Distribution User Guide*.

Figure 1-1 Library Structure Defined by the PCBDW_LIB Environment Variable



You define integration and vault areas using the `PCBDW_LIB` environment variable. The reference libraries for a site are in the `reflib` directory.

Recommendations for Library Import

Before you import library data, read through the section that is relevant to the data that you want to import:

- DE-HDL Library Data Import Checklist
- DE-HDL and PCB Editor Library Data Import Checklist
- OrCAD Capture Library Data Import Checklist

Recommendations for all libraries:

- If you import partial data, ensure that there is no common data across data sets, such as common classifications or tables.
- Run the post-analysis utility in Library Import and check there are no missing models.

Recommendations for Datasheets

You can import and manage datasheet models in the Allegro EDM database, or manage them outside the Allegro EDM environment. Datasheet values can be specified in three different formats:

- Format A: URL or FTP path
- Format B: Mapped or UNC path
- Format C: The datasheet name with the file extension

If your existing Datasheet column has a combination of all these formats, and you want to manage datasheet models in Allegro EDM, you can split the Datasheet column into two columns: Datasheet_External and Datasheet. Maintain all the format A and B datasheet models in Datasheet_External and import them as properties and not as models.

Note: If you have duplicate datasheet models in different locations, import them as properties rather than as linked models.

If the datasheet models in the Datasheet column are in format B, you can do one of the following:

- Remove the UNC or mapped drive path from the value and maintain only the datasheet name and extension
- Specify the directory and datasheet file name with the extension (for example, `\datasheets\ABC.pdf`. In this case, the datasheet can only be imported as a property and not as a model.

DE-HDL Library Data Import Checklist

- Your library data follows the library-cell view (LCV) structure. This is the data structure supported by Cadence PCB tools.
- You have decided on the classifications to use.
  - You can derive classifications from the category (`.cat`) file for a library. A classification is a grouping of similar parts (usually by function) with common characteristics.
  - Ensure that the cells are classified according to the `.cat` file. If not, library import creates its own classifications. For more information, see *Library Explorer User Guide*.
  - If you want to classify models before running library import, you can import classifications available in the CSV format.
- You have to run the front-to-back verification (using `con2con -verify` command) successfully on the library to import.
- All library names are in lower case. If the library names are in upper or mixed case, after being imported into the Allegro EDM component database, the name will be converted in lower case.

DE-HDL and PCB Editor Library Data Import Checklist

- You have revised all the Allegro PCB Editor libraries to the appropriate SPB version (using the uprev tool).
- All the Allegro PCB Editor models require sub-models. For example, footprints should have the required pads, and the pads should have the required shapes and symbols.
- You must have the `ADW_generic.env` file before you run the library import tool. If it is missing, copy it from `<installation_directory>\adw_conf_root\@company_name@\@site_name@\pcb` to `<adw_conf_root>\<company>\<site>\pcb`.

OrCAD Capture Library Data Import Checklist

To ensure smooth import of OrCAD Capture CIS library data, read through the following:

- OrCAD Capture CIS is only available on the Windows platform. Enabling the Capture CIS schema in the Allegro EDM database for the library flow is only recommended on Windows.

- Import OrCAD CIS libraries after you import DE-HDL and PCB Editor libraries.
- Maintain each model in a single classification.

Tip

To simplify the tasks required after merging the Capture and Allegro databases, it is recommended that you first import OrCAD Capture CIS libraries along with all the required PCB Editor libraries into a fresh database and fix all the pre- and post-import errors.

Note: Before importing Capture data, also go through Recommended Sequence for Library Import.

Adjusting Memory Requirements for Library Import

If you have a large amount of library data to be uploaded, an out-of-memory error might occur. In such cases, do the following to adjust the memory:

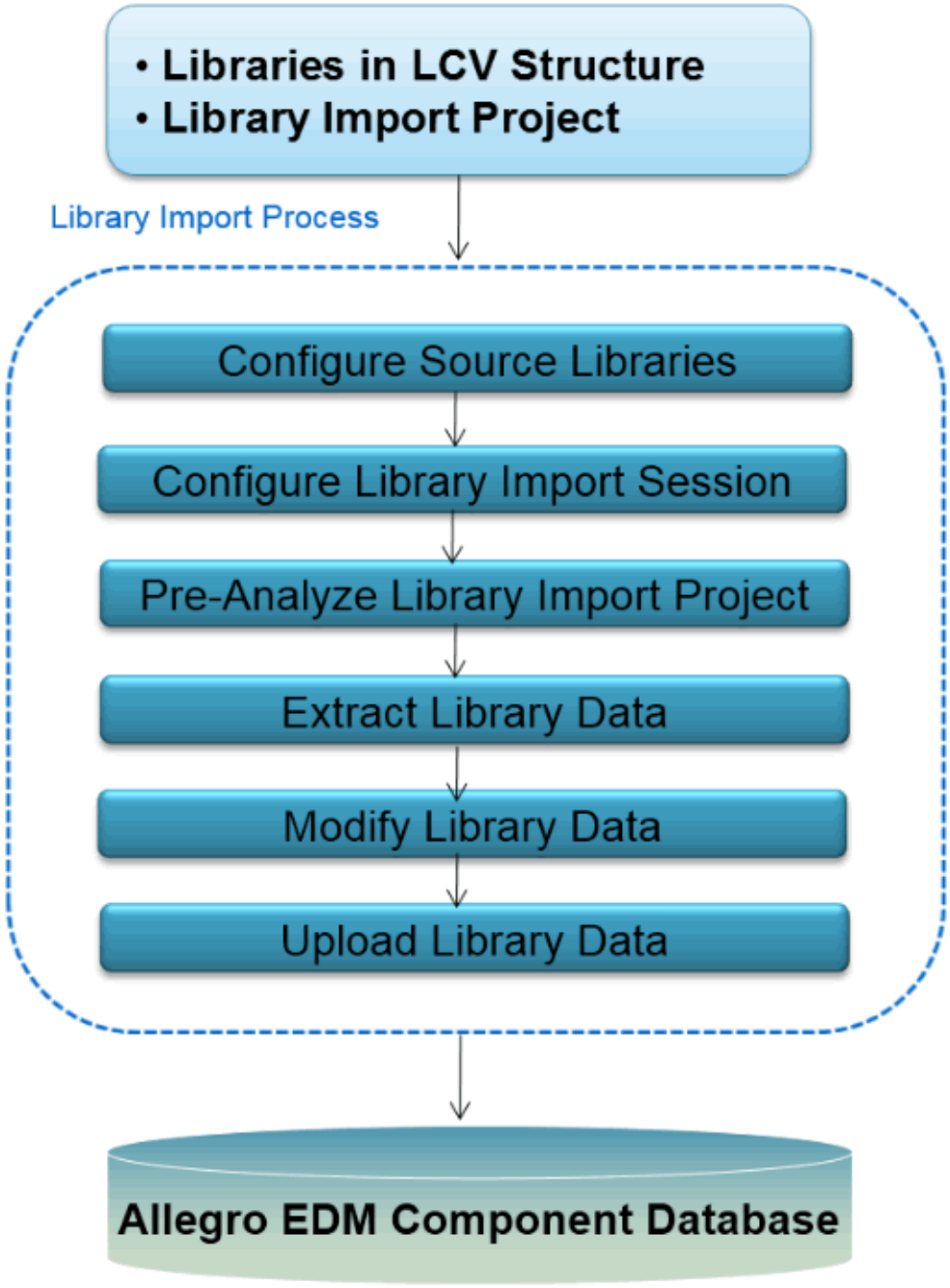
1. Launch Allegro EDM Configuration Manager.
2. Click *Set up or Manage Company & Site*.

   The *Set up or Manage Company & Site* tab displays the `workbench.ini` file.
3. On the left tree panel, choose *Allegro EDM Conf Root – <company> – <site> – Tool Configuration – Library Workbench – Library Import – libimport.ini*.
4. Modify the value of the directive `java_args`, that is, change the value of the numerical entry after `-Xmx` to any higher value depending on your requirement.

Library Import Process

A high-level library import process consists of the following major steps:

Figure 1-2 Library Import Process



1. **Creating a Library Project**: This step allows you to define a standard Cadence project containing libraries, part table files, tools, and views to import.
2. **Setting up Library Import Options**: In this step, you can define parametric information for the libraries to import. You can specify whether to import logical, physical, datasheets, and/or simulation models during the session. If you have a large number of libraries, you can configure sets of libraries to import in different sessions of the same project. For more information, see Understanding Library Import Sessions.
3. **Modifying the Data**: This step allows you to edit the library components before uploading them to the component database. For example, you can classify parts and models or can create relations for parts and models.
4. **Uploading to Database**: During this step, the library import process creates the necessary packaged formats (archive files) for the logical and physical library components. The metadata information of these components is stored in the Allegro EDM component database, while the components are physically stored in the reference area. The archive files are stored in the vault area as shown in Figure 1-1 on page 11.

Understanding Library Import Sessions

Enterprise design libraries are huge and their availability to design groups is critical for the design development cycle. Due to this, it is not practical to import all libraries in a single step. Furthermore, you might like to make some modifications to the non-Allegro EDM data before importing it. For example, you can introduce a new classification to categorize the parts.

Consequently, you need an import process that is flexible, robust, and helps you import libraries in a phased manner without impacting the design development. Allegro EDM supports the concept of *import sessions* to facilitate a phased transfer of data from legacy database into the Allegro EDM component database.

A library import session helps you capture *only* the import settings specific to the libraries to import. Some important settings include:

- What to import (such as logical, physical, datasheets, and simulation libraries)
- Where to import (physical locations and reference areas)

  Note: This information is specified while configuring the Allegro EDM Reference Library (`<PCBDW_LIB>`) using Configuration Manager.
- Part or model classification using category (`.cat`) and/or classification (`.csv`) files
- Part table file options

The Library Import Wizard helps you capture the settings for a library import session. Your complete database import can have as many sessions as you want. Moreover, you can save your import sessions and can reconfigure a previous library import session or can even delete a previous library import session.

Recommended Sequence for Library Import

Here is a step-by-step guide to a successful library import session:

1. Start by creating a new library project using Project Manager.

   This is a library project that is similar to the ones created for Part Developer (PDV).

   While creating the project, make sure that all the required objects (PTFs, category files, `.olb` files, DML, datasheets, PCB Editor models) are configured properly through the Configuration Manager wizard.

   Remember to set up the *Cell Level Physical Part Table File* and *Merge Physical Part Table File* options because every option is necessary and it drives how the data will be imported to the database.

2. After the project is created, open the project in Part Developer to check for any errors.

   Part Developer should be able to open the project without any errors. Proceed to the next step **only** if there are no errors.

3. Launch Library Import from Allegro EDM System Console.

4. Choose *Configure Import Options*.

   This launches the *Library Import Setup* wizard, where you:

   a. Import front-end, back-end, simulation, or datasheet libraries.

      You should select all options, if you have all kind of data in your libraries.

   b. Choose the required libraries from the logical libraries list.

   c. Provide the PSM and PAD paths.

   d. Provide Capture CIS related details.

   e. Provide the paths for Datasheet Model and SI DML Model.

   f. Select the corresponding file extensions that you want to import.

      You can also provide the category files path. This path is only necessary if you do not have library-level `.cat` files, and category files are stored in some other location.

5. Ensure that your data is compatible with Allegro EDM by clicking the *Library Consolidation* radio button. This opens the Library Consolidation — Options wizard, which allows you to consolidate your libraries before importing them into Allegro EDM.

6. Analyze the libraries to identify the properties that define the Part Number for the components.

   If you have multiple properties that specify a part number, you can provide them as comma-separated values. They will be used in the order in which they are given.

7. Specify the list of numeric properties. Again, you can provide multiple properties separated by a comma.

8. When both key and injected properties have the same names but different values in the PTF files of the source libraries selected for import, you can choose which property value should be added to the Allegro EDM component database. Select one of three options — both, only key, or only injected — to specify which value is to be used during the import process. Retain only one property unless it is really important to have the other property.

   After the wizard collects all this information and completes the configuration, the configuration settings are saved in the `<library_import_project_directory>\migration\default.conf` file.

9. Start the pre-analysis tool.

   This lists all the errors in the selected libraries. Each error needs to be reviewed and corrected in the libraries.

   To ensure that all the errors are corrected, run the pre-analysis tool again to rule out any more errors. This process should be repeated till all the errors are fixed.

   Proceed only after correcting the errors and verifying the pre-analysis report.

10. Verify all the project libraries using the front-to-back check utility.

    It is very important that libraries are front-to-back verified. Library Import marks all parts and models as released without verifying any information, so this front-to-back verification ensures that all libraries are verified.

    Proceed to the next step **only** after the data is verified.

11. Generate the XML by selecting the *Extract Library Data* option.

12. Open the project.

    You can now view its libraries. You need to ensure that all the data is available and searchable.

Library Import generates a temporary classification names for the Schematic models that were not categorized in the source `.cat` files. In addition, physical parts in PTFs that are linked to such uncategorized models are imported into auto-generated Part Classifications. You can change such auto-generated Schematic and Part Classifications using the manage classification options. For details, see Modifying the Data Before Upload.

13. After updating the classifications, click the *Save Xml* button ()in the toolbar to save the changes.

14. Start the upload of the project libraries to the server.

After the upload is completed, check the log file `<library_import_project_directory>\migration\MissingObjectReport.log` to see if there any errors or warnings in the file.

15. After the upload is complete, run the post-analysis tool.

In the Allegro EDM Library Migration window, choose *Tools — Generate Missing Object Report.*

This report generates the list of objects that are in the preliminary state in the database. The preliminary state implies that these objects were not found during the upload process. These objects can be imported by creating a separate library import project and then importing them separately.

The database is not usable by designers till all the objects are released and imported correctly.

2

Working with Library Import

This chapter explains the important procedures and steps you must follow to successfully complete library import. The major procedures in library import include:

- Launching the Library Import Tool
- Configuring Source Libraries
- Configuring Library Import Options
- Checking the Project
- Generating XML
- Opening Existing Projects
- Modifying the Data Before Upload
- Uploading the Data
- Verifying the Upload
- Post Library Import Tasks

Launching the Library Import Tool

To run the library import process:

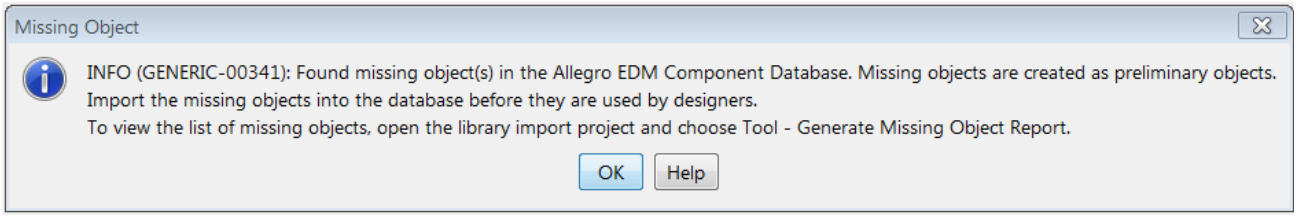
1. Type the following command in the Allegro EDM System Console.

   ```
   libimport
   ```

   The Login dialog box appears.

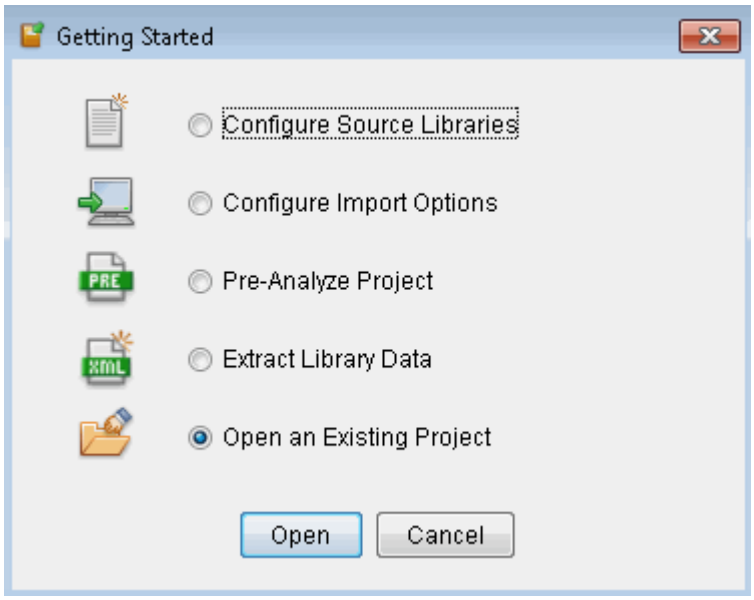
2. Enter the user name and password, and click *Login*.

   The following message appears only if there are preliminary objects in the database.



3. Click *OK*.

   The Getting Started dialog box appears.



   After you create a library project that contains the library, tools, and part table information, you can choose one of the following options in the Getting Started dialog box.

To be able to

- ❑ import legacy libraries for the first time into the Allegro EDM environment, select *Configure Source Libraries*.
- ❑ configure the library import session settings, select *Configure Import Options*.
- ❑ check the libraries for any errors, select *Pre-Analyze Project*.
- ❑ create an XML version of the source library data, select *Extract Library Data*.
- ❑ upload the libraries, select *Open an Existing Project*.

The library import projects are similar to design projects created using Allegro Project Manager Wizard. These projects have a `.cpm` file and follow the standard directory structure.

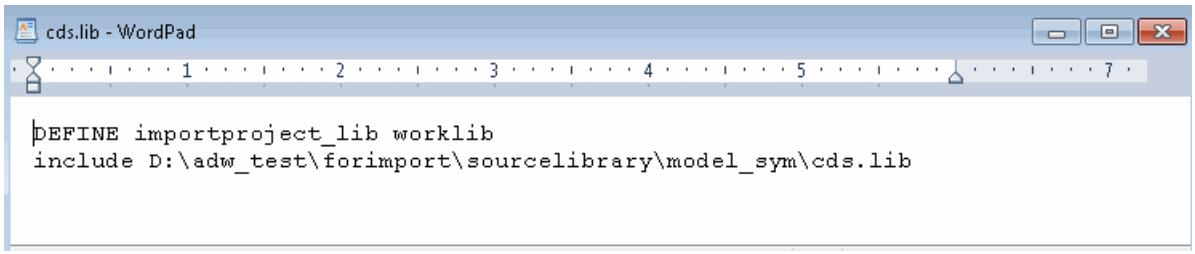
Configuring Source Libraries

After you have created a project that contains the library, tools, and part table information, you need to configure or reconfigure the path for source libraries and/or PTF files. To configure source libraries, perform the following steps from the Getting Started dialog box:

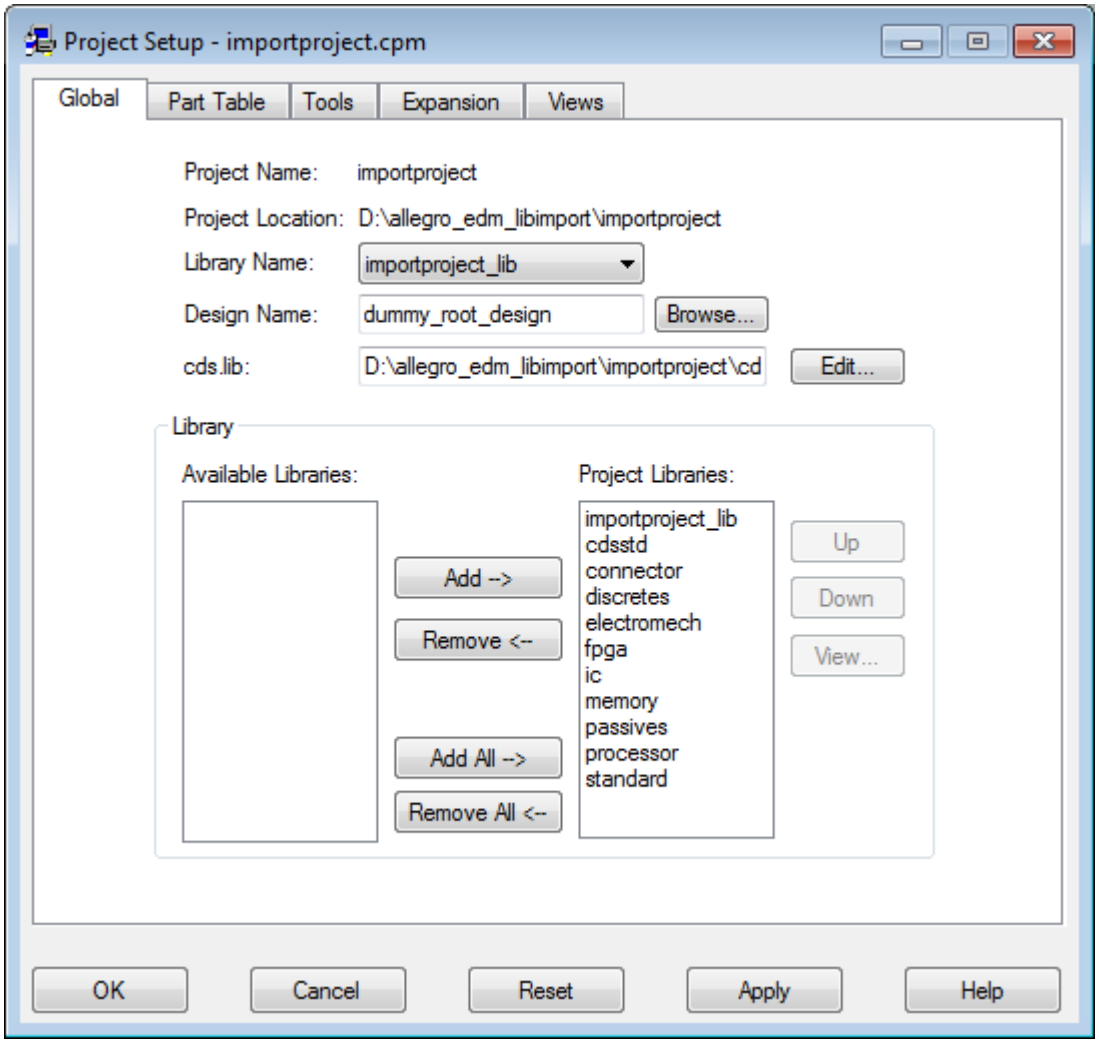
1. Choose *Configure Source Libraries.*
2. Click *Open*.

   The Select Project File (.cpm File) dialog box appears.
3. Browse for the *<library_import_project>*`.cpm` file and click *Open*.

   The Project Setup - <Library_Import_Project_Name> dialog box appears.
4. Click *Edit* to include the path for the source front-end libraries that you want to import.

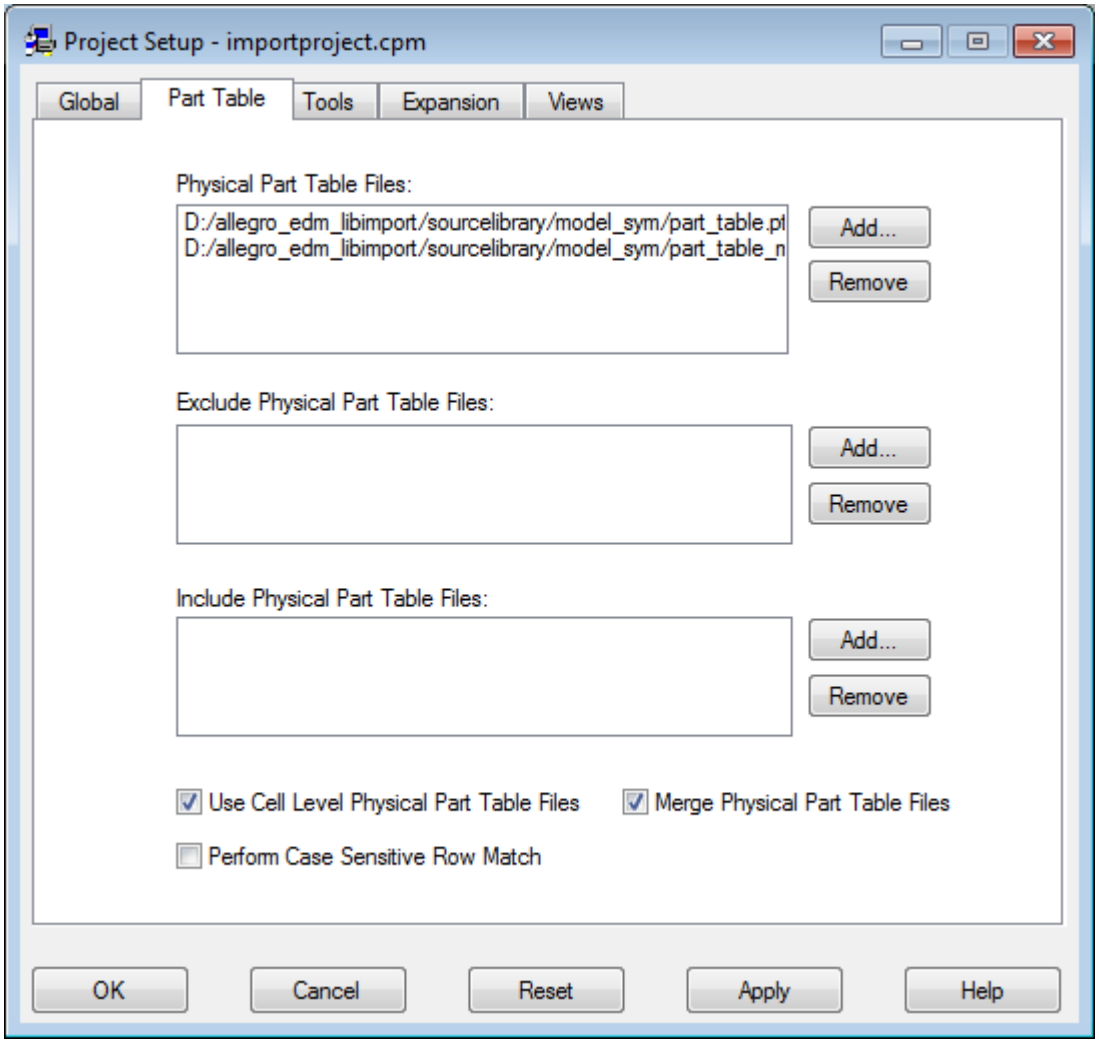


5. Choose the libraries from Available Libraries to add to Project Libraries.



6. Click *Apply*.
7. Click the *Part Table* tab.
8. Click *Add* to include the Physical Part Table files.
9. The Add Physical Part Table dialog box appears.
10. Click *File*.
11. Browse for the physical PTF file(s) and click *Open*.

12. Click *OK*.



13. Ensure that you select the following check boxes:

- *Use Cell Level Physical Part Table Files*
- *Merge Physical Part Table Files*

14. Click *Apply* and *OK*.

Configuring Library Import Options

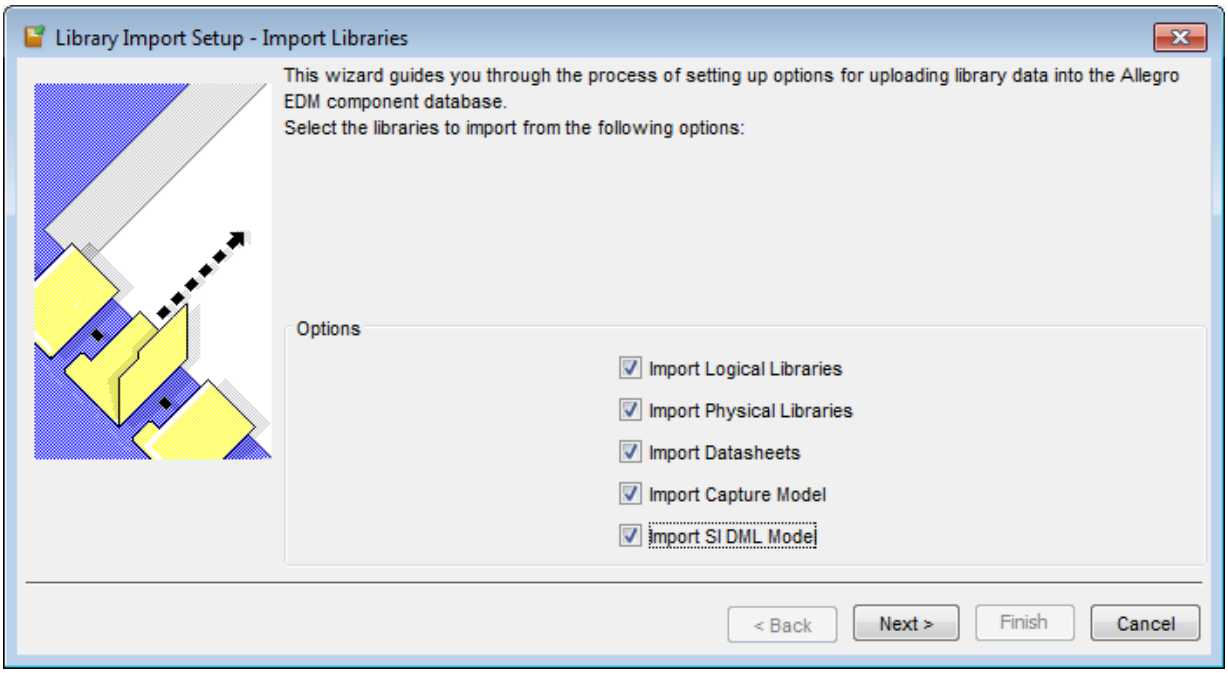
After you have configured the source libraries, you need to configure the project for library import options. To configure setup options for importing libraries, perform the following steps from the Getting Started dialog box:

1. Choose *Configure Import Options*.
2. Click *Open*.

   The Library Import Setup - Import Libraries wizard page appears.
3. In this dialog box, select:
   - *Import Logical Libraries* to import logical models.
   - *Import Physical Libraries* to import physical models.
   - *Import Datasheets* to import datasheets.
   - *Import Capture Model* to import OrCAD Capture CIS models
   - *Import SI DML Model* to import SI DML models.

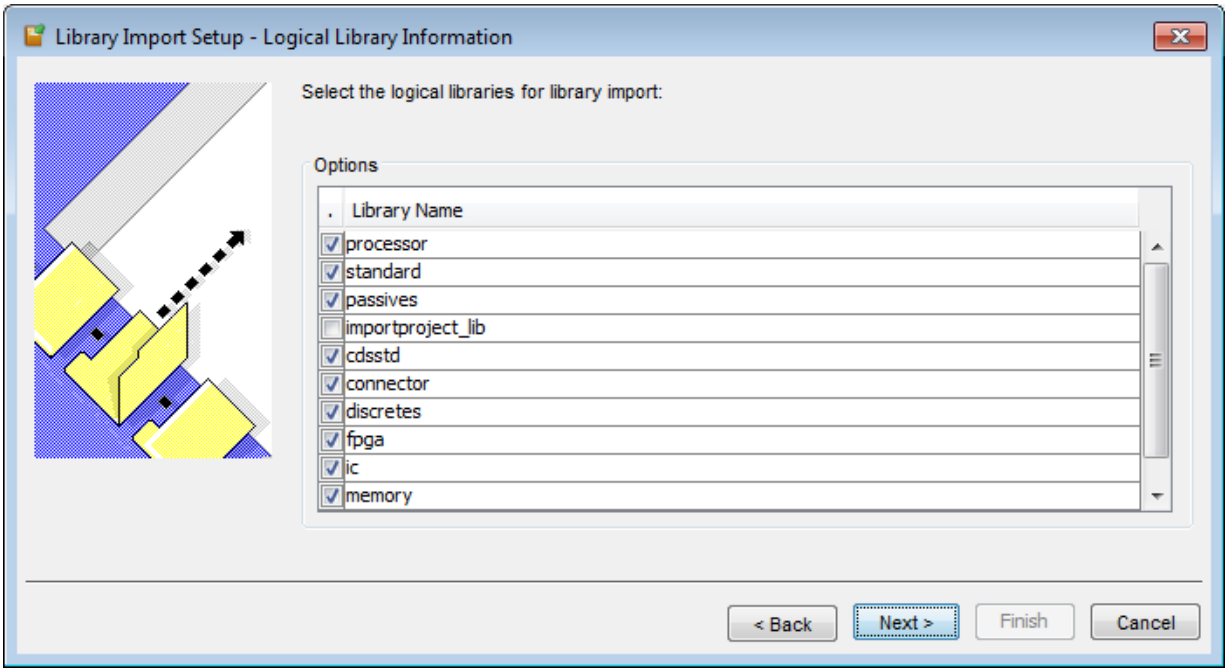
   The sequence of the screens that appears beyond this point in the wizard depends on the library components (logical, physical, datasheet, Capture, SI DML, or any combination of them) that you select.

To update the database and release the archive files from the integration area to the vault area, you must have both logical and physical models in the integration area.



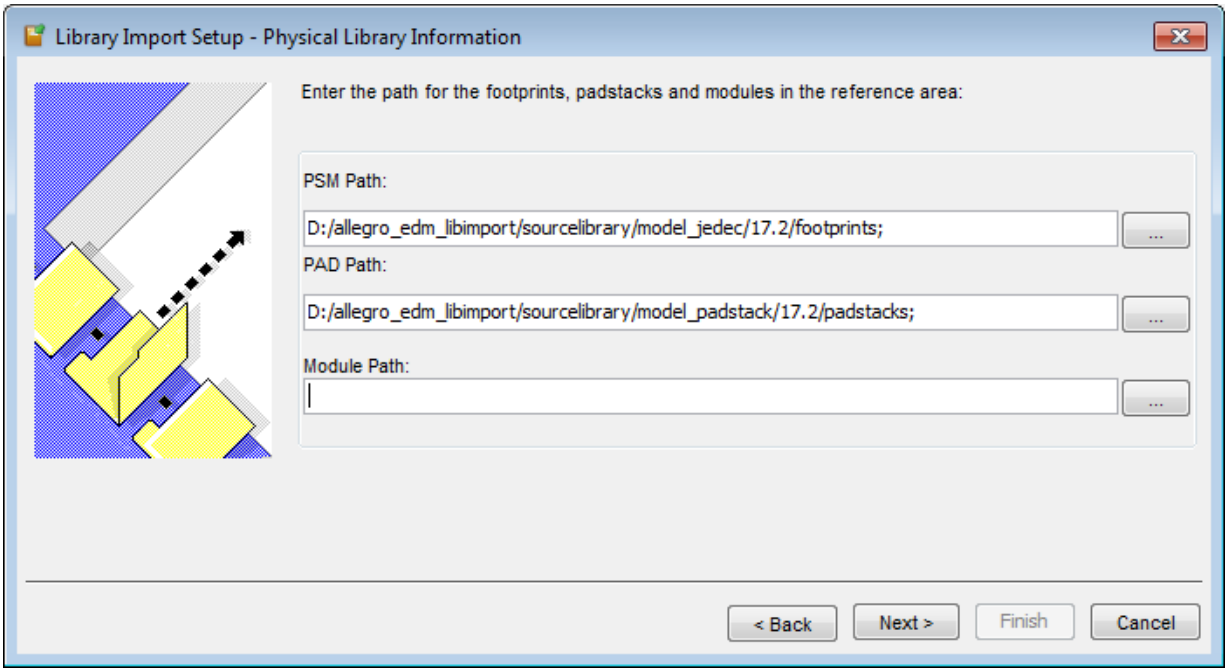
4. Click *Next*.

The Library Import Setup - Logical Library Information wizard page appears.



5. Clear the check box for the library import project library.
6. Click *Next*.

The Library Import Setup - Physical Library Information wizard page appears.



7. Specify the PAD, PSM, and Module paths. If you select *Import Physical Libraries* in the Import Libraries wizard page, you need to either select PSM/PAD path or Module path, or both in this page.

Field Name	Lets you...
PSM Path	Specify the absolute path of all the folders that have `.dra` files for the libraries being imported. You can enter the paths manually.
PAD Path	Specify the absolute path of all the folders that have `.pad` files for the libraries being imported. You can enter the path manually. You can have duplicate models in different libraries only in case of padstack models.
Module Path	Specify the absolute path of all the folders that have `.mdd` files for the libraries being imported.

Alternatively:

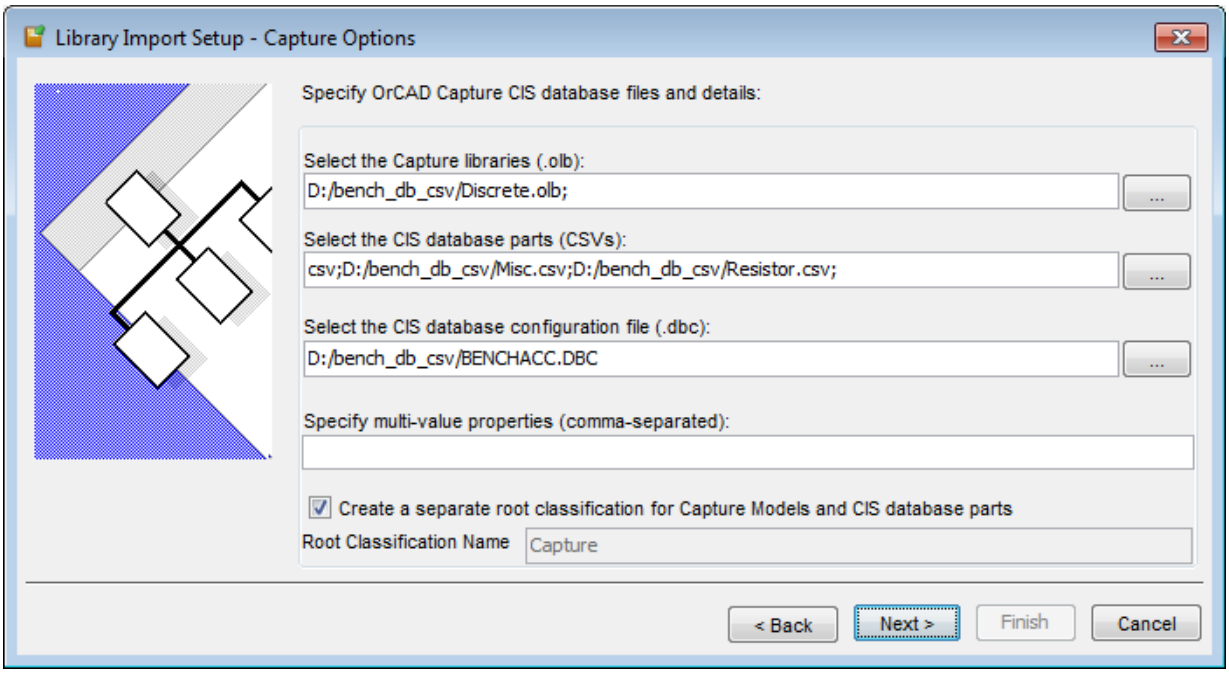
a. Click the Browse (...) button.

The Select Directories dialog box appears.

b. Specify the path and click *OK*.

8. Click *Next.*

The Library Import Setup - Capture Options wizard page appears.



a. In *Select the Capture libraries (.olb)*, browse and select the OrCAD Capture libraries that you want to import into Allegro EDM.

If you have standard symbols and block models, move them to a separate library, then update `capture.ini` accordingly. For details about `capture.ini`, see *OrCAD® Capture Quick Reference*.

b. In *Select the CIS database parts (CSVs)*, select the `.csv` files that contain information for each part that you want to import from the selected Capture library.

Ensure that the data is in the CSV format.

Note: Only string and numeric data types are supported.

c. In *Select the CIS database configuration file (.dbc)*, specify the CIS database configuration file you want to import to make use of your part database.

d. If Datasheet is a multi-value field in the CIS database and you want to import them as models, define the Datasheet property in *Specify multi-value properties (comma-separated)* before you start importing the columns into the EDM database.

For example:

```
Cap.pdf, Res.pdf
```

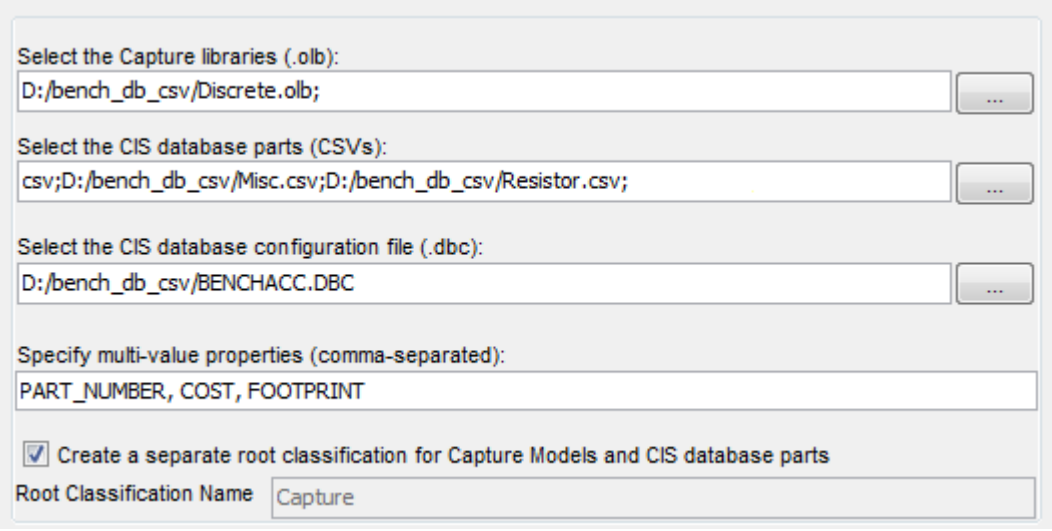
Note: You do not need to specify multiple values as properties for schematic part names or PCB footprints. Allegro EDM handles them by default.

Note: In the current release, the order of values in multi-value columns is not preserved.

e. Prior to importing Capture CIS data, decide whether you want the Capture CIS parts and classifications to be separate or merged. If you want Allegro EDM to create a separate root classification node for Capture data, select the check box. *Capture* is the default node name and cannot be modified.

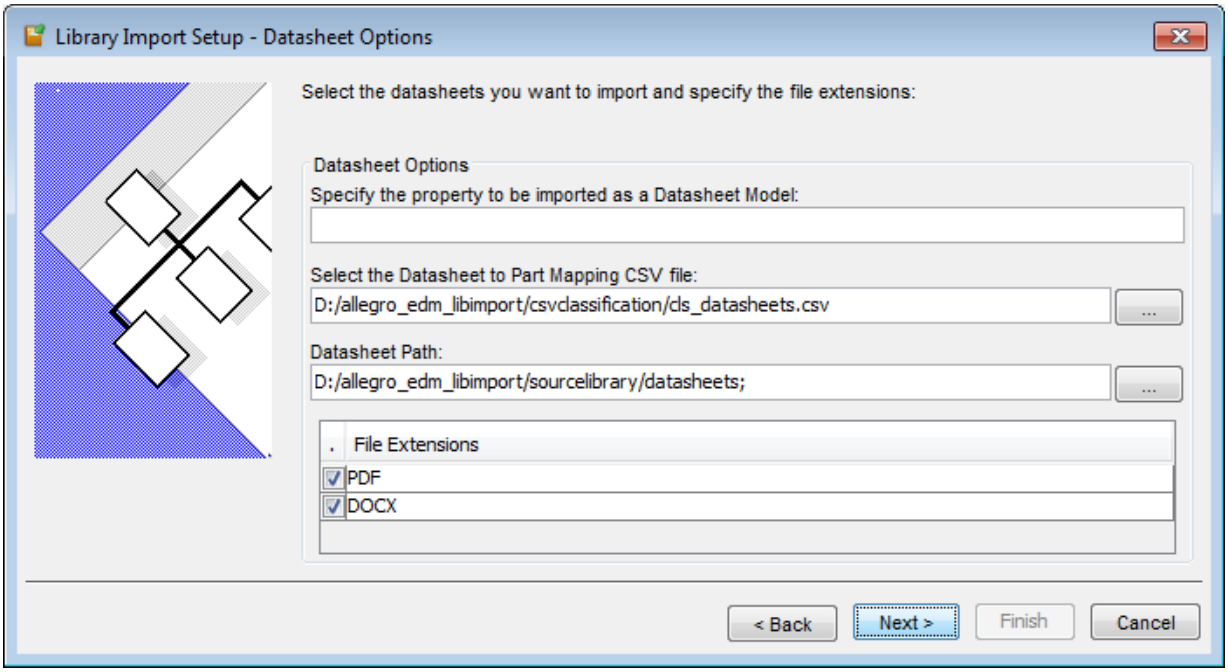
If you do not select the check box, the DE-HDL and Capture CIS classifications will be merged.

If you select this check box, all Capture classifications will be displayed under this node.



9. Click *Next.*

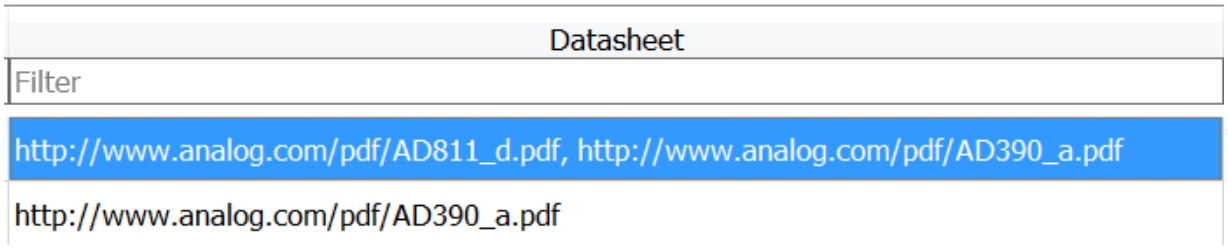
The Library Import Setup - Datasheet Options wizard page appears.



The Datasheet Options section provides two ways to create a part to datasheet relationship:

- In *Specify the property to be imported as a Datasheet Model,* specify the property name in the PTF or `.csv` file to be used to map a part to a datasheet. Ensure that:
  - the value of the property matches the datasheet name.
  - you associate only one datasheet with one part.

  You can specify multiple comma-separated values for the property names in this field as illustrated.



- In *Select Datasheet to Part Mapping CSV file*, specify the path to the CSV file that has the part to datasheet relationship.

If you need to specify the Block Part to Datasheet relationship, ensure that the corresponding CSV file contains the correct format for header.

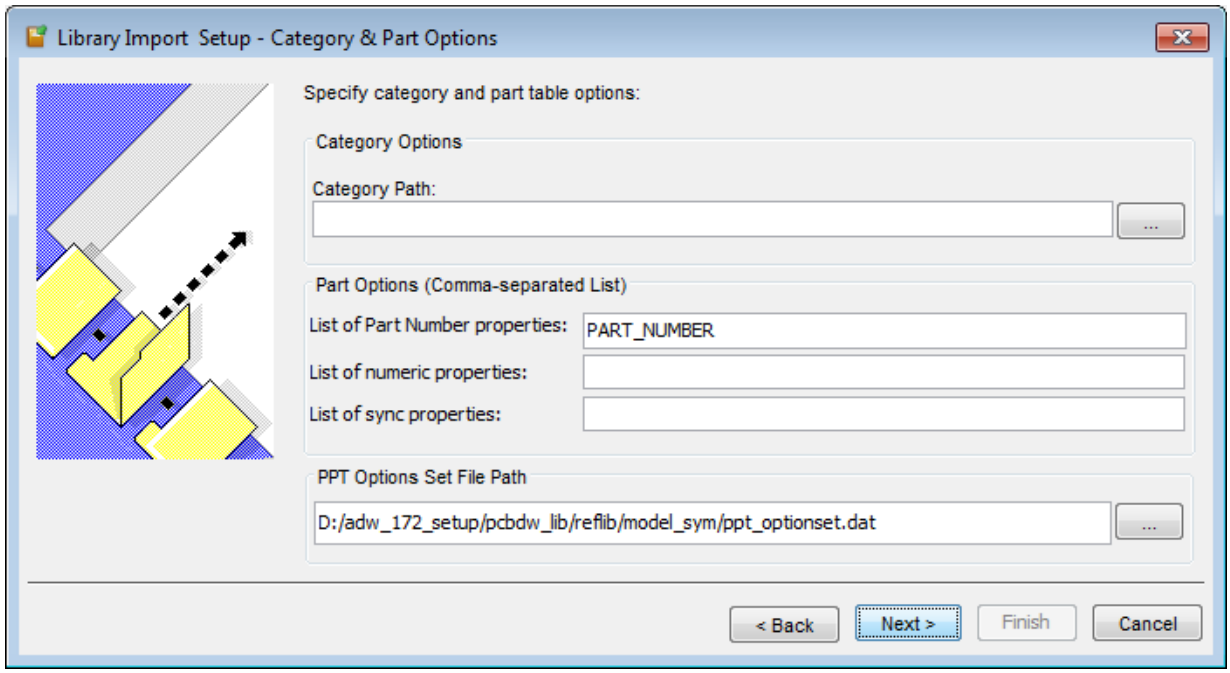
- In the *Datasheet Path* field, enter the absolute path of all the folders that have the datasheet files for the libraries being imported.

  Note: You can enter the path manually, or click the Browse button to specify the locations using the Select Directories dialog box.

- In the *File Extensions* section, select all the file extensions of the datasheet files to import. See Recommendations for Datasheets to manage datasheets in Allegro EDM.

10. Click *Next*.

The Library Import Setup - Category & Part Options wizard page appears.



- In the *Category Options* area, specify the path of the category file using the Browse (...) button.

  This information helps you classify the front-end models.

- In the *Part Options* area:

  - In the *List of Part Number properties* field, specify the name of the PTF column headers to use for identifying the parts. The default value for this field is `PART_NUMBER`.

- In the *List of Numeric properties* field, enter the name of the PTF column headers that contain numeric values frequently used for searching libraries.
- If you have duplicate part numbers, identify the sync properties by considering the database primary key property list. In the *List of Sync properties* field, specify the properties that make the row unique.

  If you specify `PART_NUMBER` as the sync property, duplicate part numbers will be ignored.

Note: If the sync property is a part number, only the first model in a multi-value column is considered.

Note: For all fields in the *Part Options* section, you can enter multiple properties (separated by semicolons).

11. In the *PPT Options Set File Path* field, click the Browse (...) button, navigate to the directory that has the PPT option set you want to import and click *Open*.

If you choose to import then upload a PPT option set that has been created by the librarian to the database, this ensures consistent annotation and visibility settings for part properties.

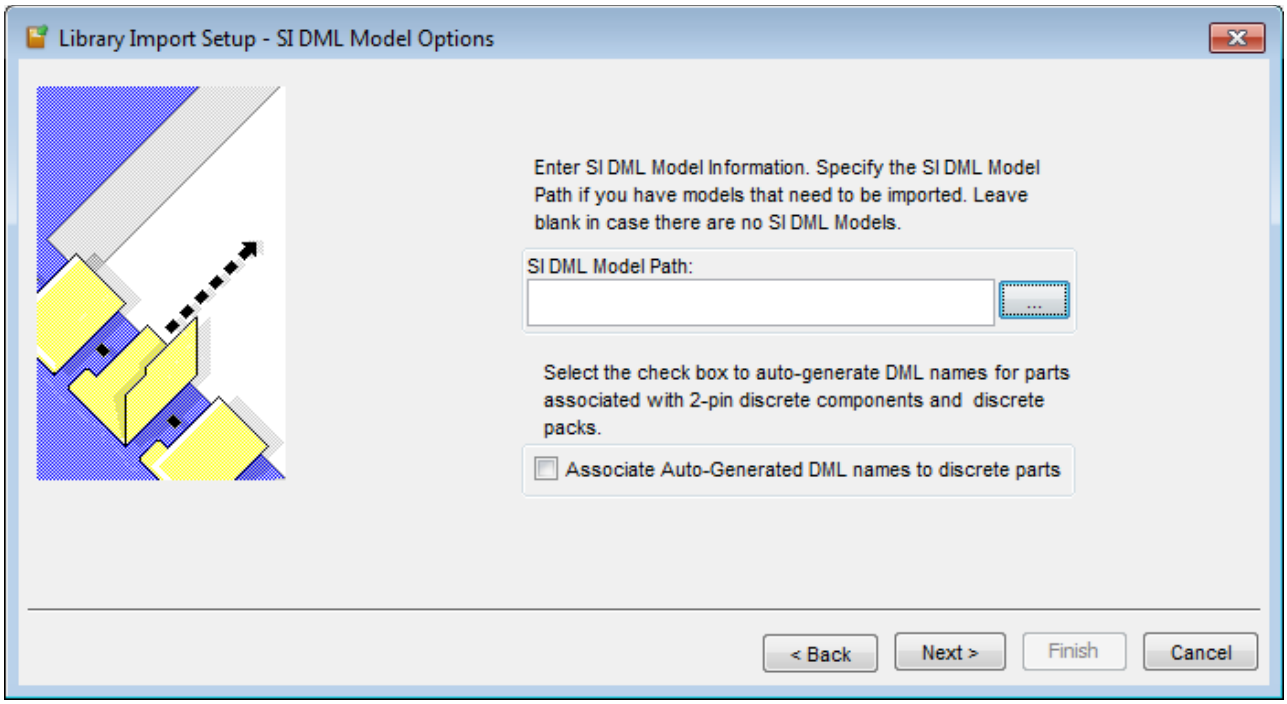
It also ensures that changes by the librarian to the option set are propagated to all designers in real time.

When this option set is uploaded to the database, Allegro EDM applies the settings in the option set file to all parts in the project. If there are parts for which settings have not been defined in the option set file, Allegro EDM applies the annotate to design and visibility attributes as follows:

- Injected property: *Annotate to Design* is set to No, and *Visibility* is set to Invisible. Both attributes can be edited using Allegro EDM Database Editor.
- Key Property: *Annotate to Design* is set to Yes, and *Visibility* can be set by the librarian using Database Editor.

12. Click *Next*.

The Library Import Setup - SI DML Model Options wizard page appears.



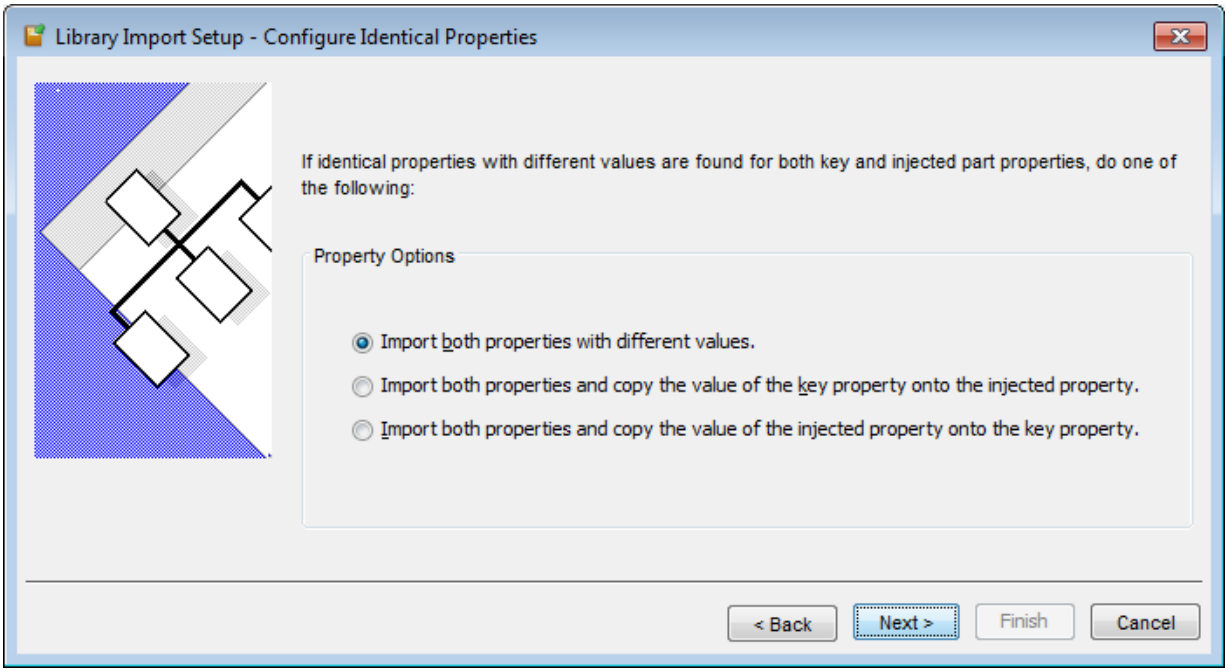
13. Specify the path of all the folders that have SI DML models for the libraries being imported.

Note: You cannot import simulation models, such as verilog, pspice, and ibis.

For information on when to use the *Associate Auto-Generated DML names to discrete parts* check box, see Auto Generation of SI DML Models for Discrete Parts.

14. Click *Next*.

The Library Import Setup - Configure Identical Properties wizard page appears.



In the PPT Option set file that you are importing, if there are identical key and injected part properties but with different values, do one of the following:

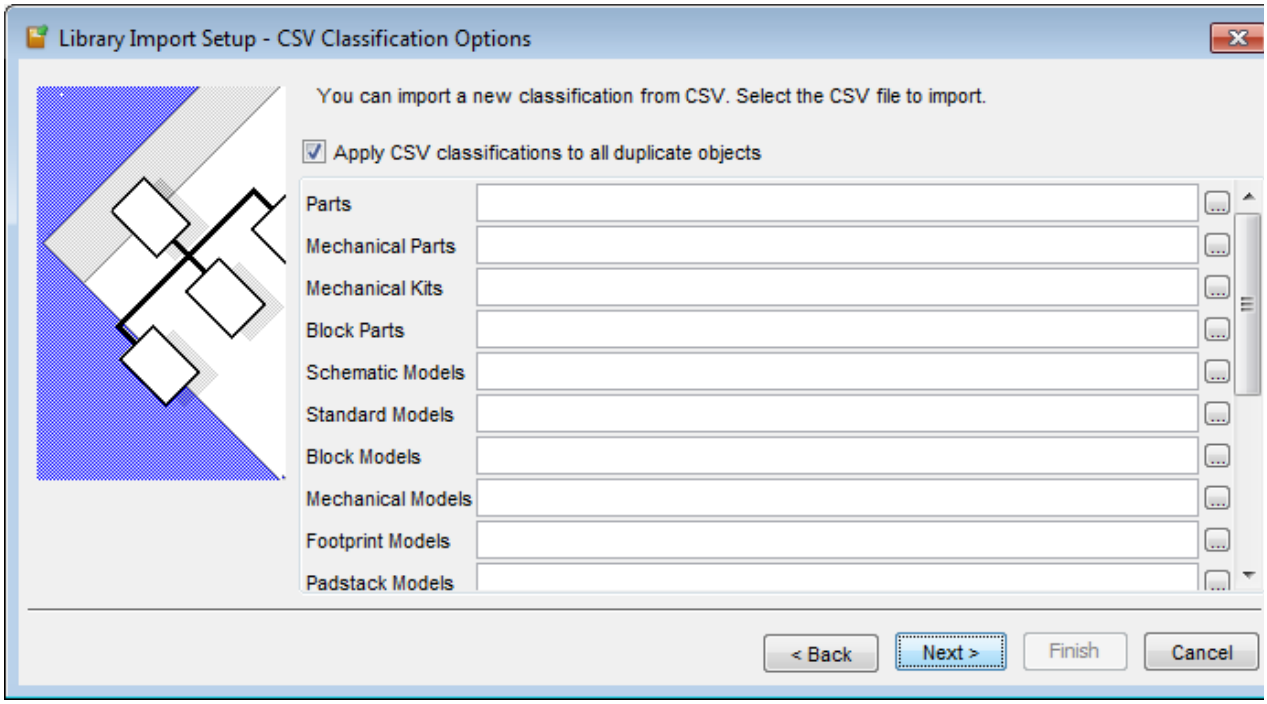
- ❑ Import the key and injected part properties with different values.
- ❑ Import both properties but overwrite the value of the injected property with the key property value.
- ❑ Import both properties but overwrite the value of the key property with the injected property value.

Ideally, you should retain only one property unless it is important to have the other property as well.

Note: If values for parts in key and injected properties have not been defined in the PTF, Allegro EDM does not mark these key and injected properties as Mandatory properties.

15. Click *Next*.

The Library Import Setup - CSV Classification Options wizard page appears.



Before running library import, you can import classifications available in the CSV format. Ensure that you do the following:

- Select the *Apply CSV classifications to all duplicate objects* check box. Selecting this option associates the related classification with all the duplicate objects. If you do not select this option, the duplicate objects are reported in the Pre-Analysis Report (`report.log`). If you upload the library data as such, the particular object from CSV is not included in the classification merge operation.
- Use the correct format for the header of these classification files.

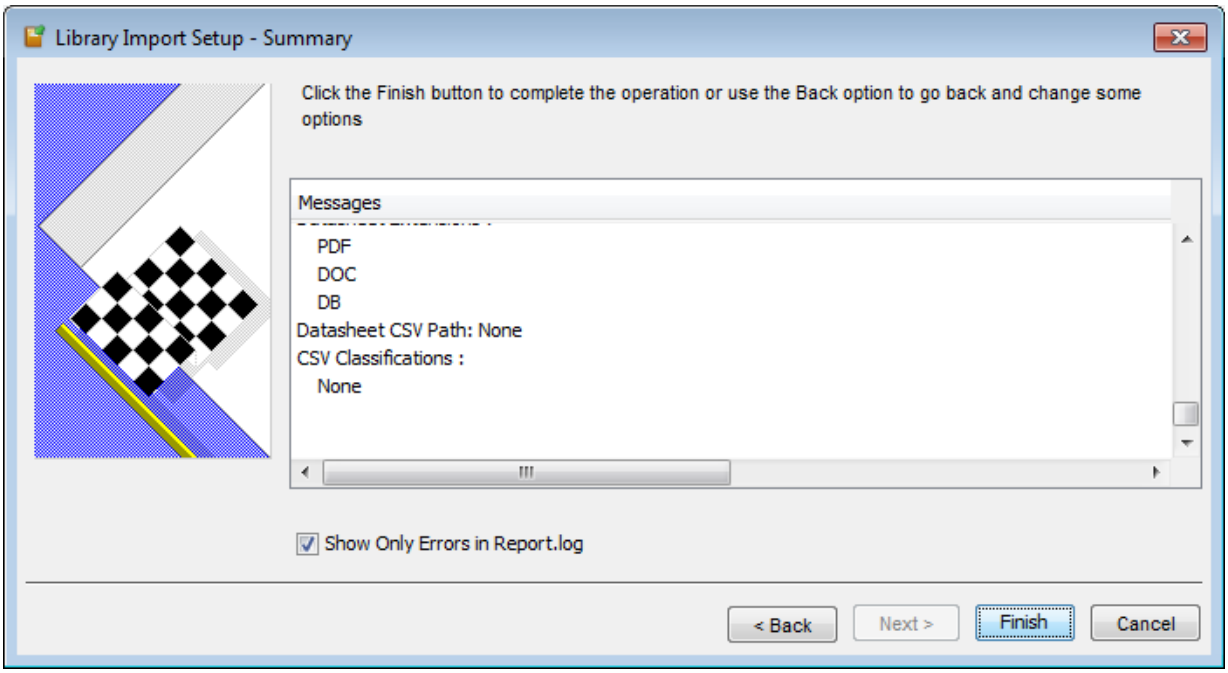
To classify padstack models which have the same name and are in two different libraries, the corresponding CSV file needs to have the library name before that padstack model. For example, if PAD1 is the name of the duplicate padstack model then in the CSV file, it should be specified as:

```
<library_1>: PAD1
```

```
<library_2>: PAD1
```

16. Click *Next*.

The Library Import Setup - Summary screen appears with all the selections made during the setup process.

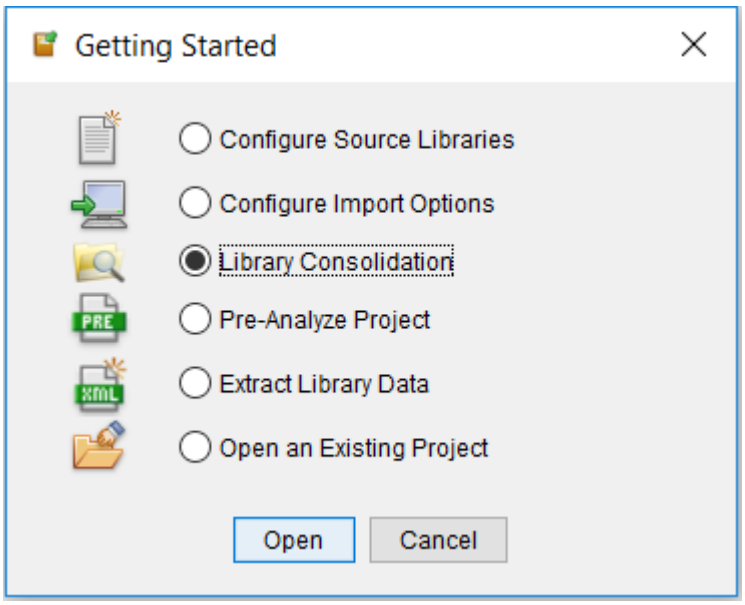


By default, Allegro EDM displays errors, warnings, and information messages in `Report.log`. If you want to only view errors, select the *Show Only Errors in Report.log* check box.

17. Click *Finish* to complete the setup.

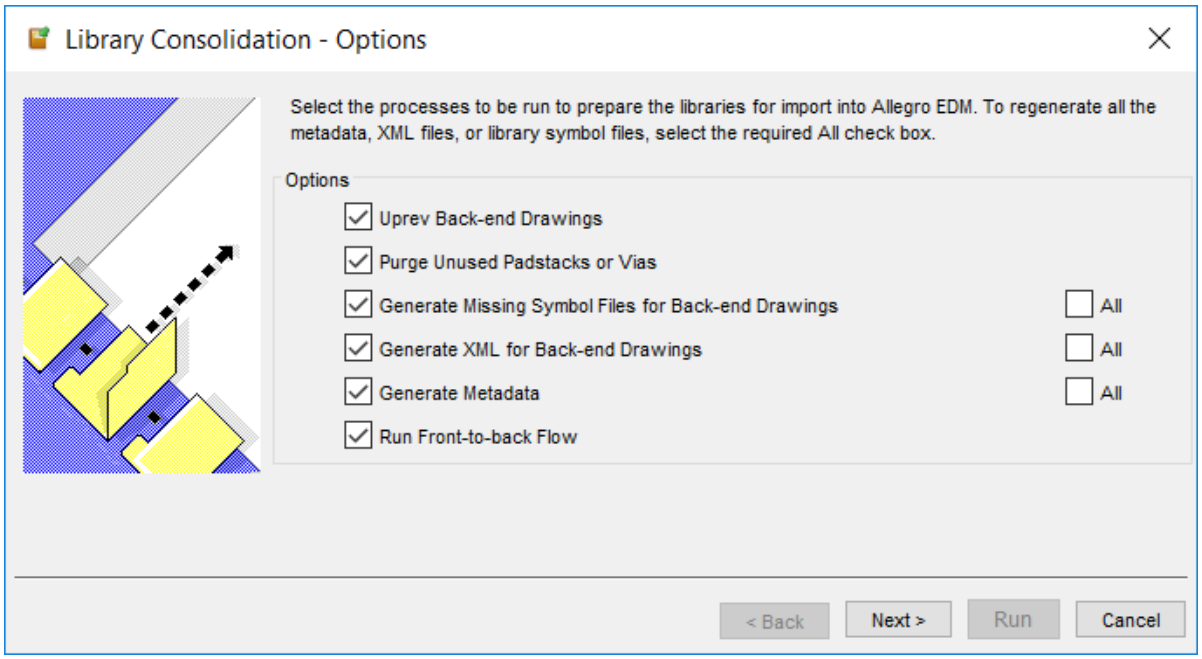
Consolidating Libraries Before Importing

1. Ensure that your data is compatible with Allegro EDM by clicking the *Library Consolidation* radio button.



This opens the Library Consolidation — Options wizard, which allows you to consolidate your libraries before importing them into Allegro EDM. You can choose to uprev the back-end models to the current version of Allegro EDM, purge unused padstacks or vias for footprints, generate missing symbol files for back-end models, generate the XML files used to display the footprint graphics in Allegro EDM Database Editor for back-end

models, generate metadata for the cells in your libraries, and run the front-to-back flow to verify all the front-end models.



2. Uprev the back-end models to the current version of Allegro EDM. The report will indicate the number of back-end models that have been uprev-ed.
3. Purge unused padstacks or vias for footprints.
4. Generate missing symbol files for back-end models. This will generate `.psm`, `.bsm`, `.fsm`, and `.ssm` files for back-end models. Allegro EDM will generate files only for those models that are missing in the source libraries or are outdated. You can also choose to regenerate all the XML files.
5. Generate the XML files used to display the footprint graphics in Allegro EDM Database Editor for back-end models. Allegro EDM will generate files only for those models that are missing in the source libraries or are outdated. You can also choose to regenerate all the XML files.

Important

Allegro EDM will generate files only for those models that are missing in the source libraries or are outdated. It does not reprocess the entire library every time you run the consolidation option.

Important

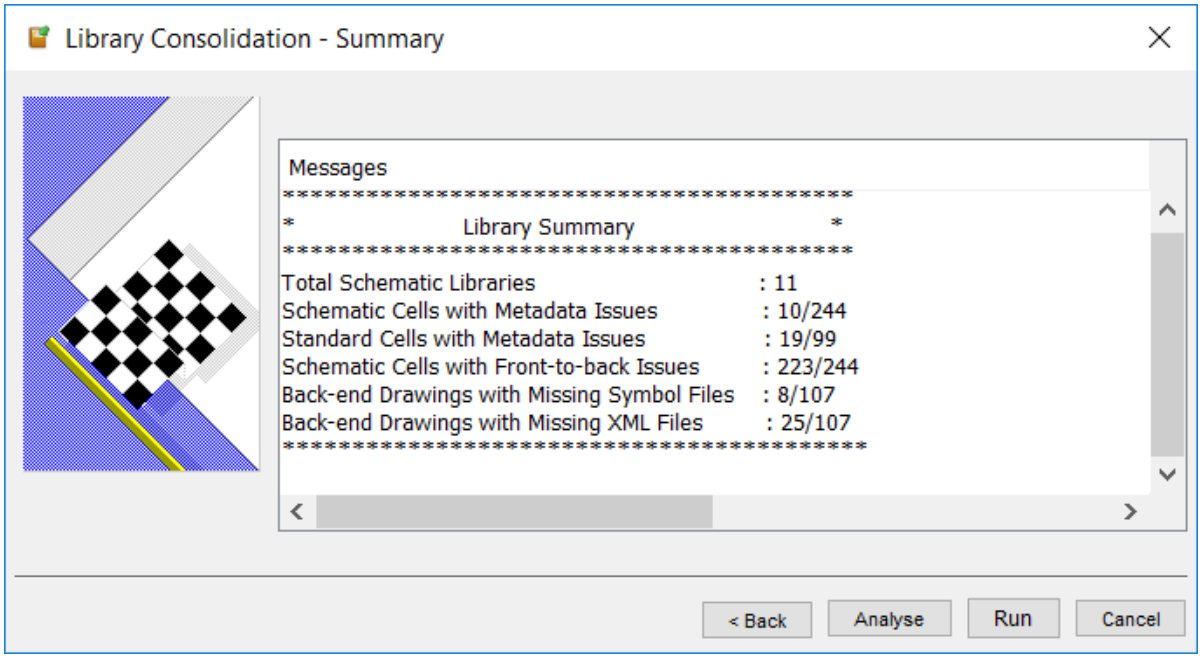
You can also choose to regenerate all the XML files. If you do not select the *All* check box, the consolidation process runs in incremental mode. For example, in the case of back-end models, if the timestamp of the XML or the compiled files is before that of the `.dra` files, the files are generated to incorporate the latest changes.

6. Generate metadata for the cells in your libraries. Allegro EDM will generate metadata only for those cells that do not have any. You can also choose to regenerate all the metadata. In this case, metadata from all the cells will first be deleted, and then regenerated.

   Note: If you regenerate all the metadata, all version history from the metadata will be removed and all cells will start at the 1.0 version.

7. Run the front-to-back flow to verify all the front-end models. Allegro EDM will create a report of models that have passed and failed the checks.

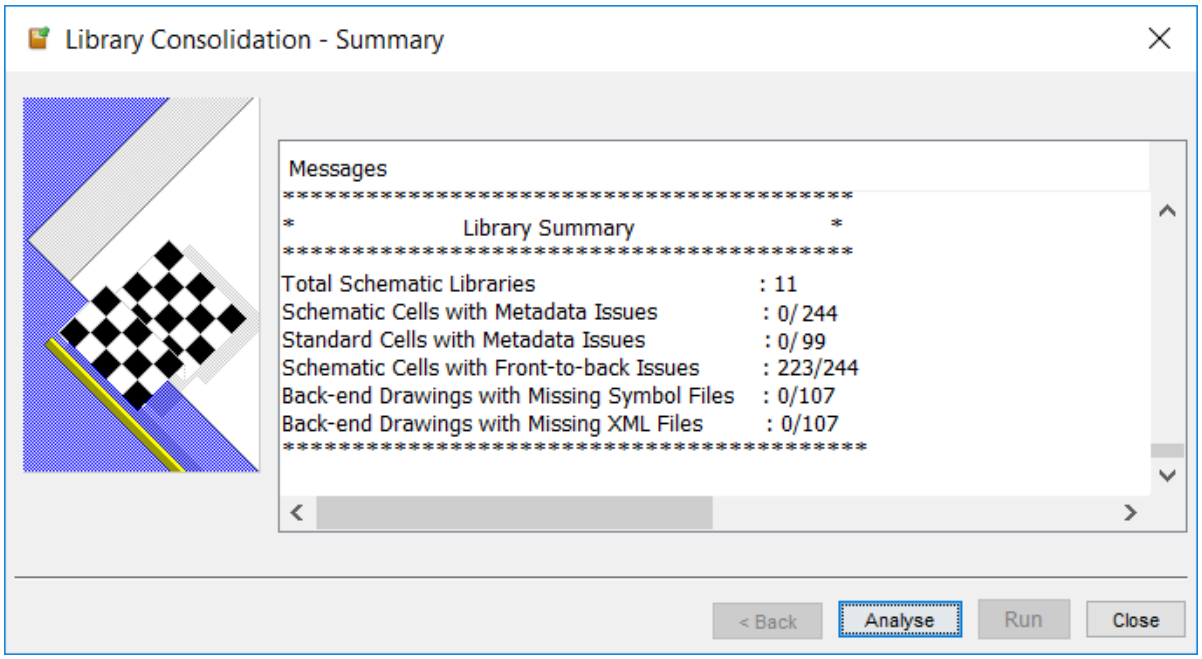
8. Click *Next*.



The Library Consolidation — Summary page appears with details about your libraries and the number of objects affected by each operation.

9. Click the *Run* button to consolidate your libraries.

10. After Allegro EDM executes all the operations, click the *Analyze* button to view the results.



As you can see, Allegro EDM has fixed some of the issues, while others might have to be manually fixed. For details of the changes that Allegro EDM made, see Report.log at `<EDM project>\migration`.

11. Click *Close* to return to the Getting Started dialog box.

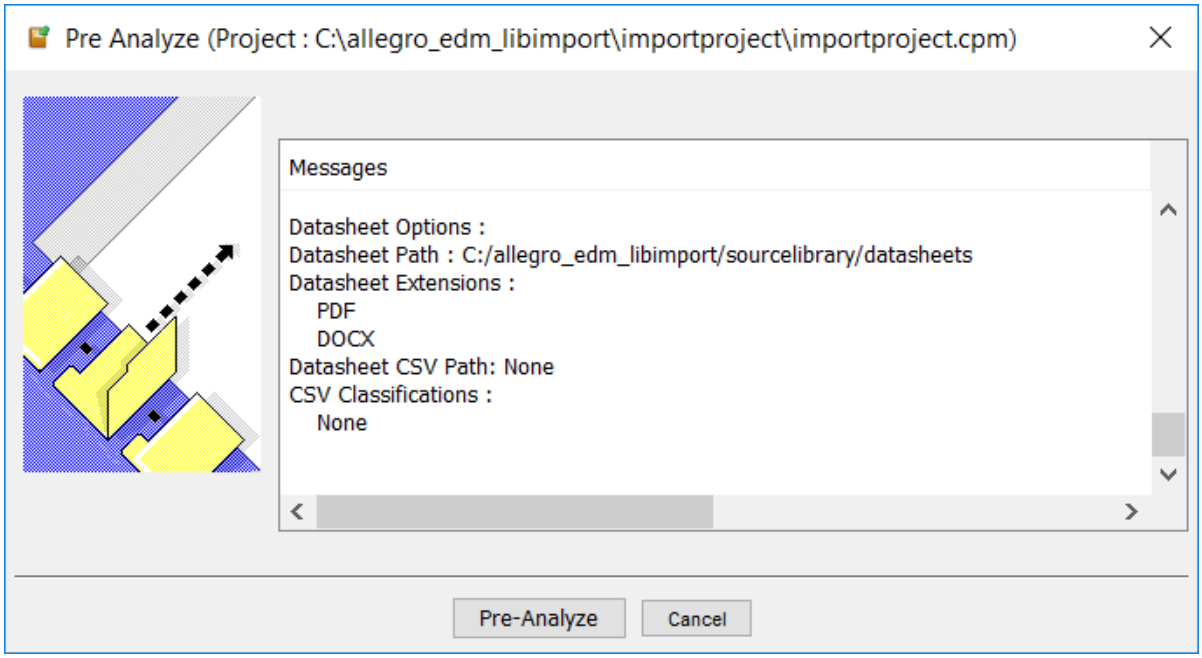
Checking the Project

After the library import project is configured, you are ready to import it. Before starting the import, you need to check the libraries for any errors using the pre-analyze tool. To check the libraries, do the following from the Getting Started dialog box:

1. Choose *Pre-Analyze Project*.
2. Click *Open*.

   The Pre Analyze (Project: `<Library_Import_Project_Path>`) screen appears.

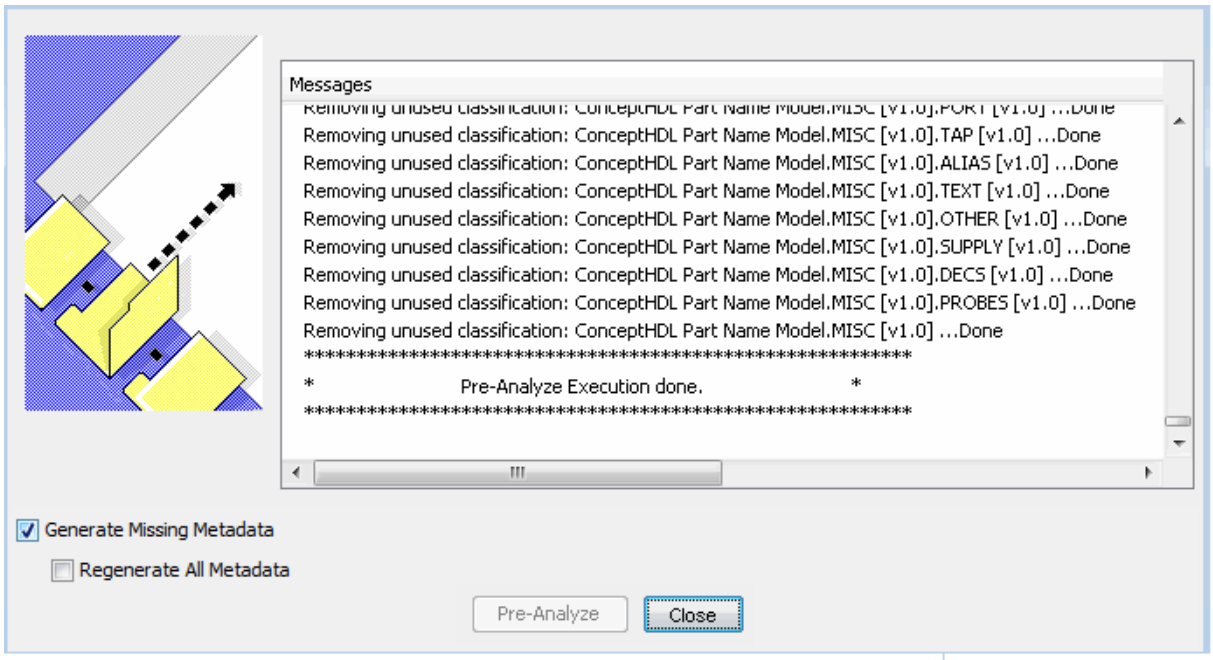
   Note: If you have launched the Library Import utility to directly perform pre-analysis, the Select Project File (.cpm File) dialog box appears. Select the `<project>.cpm` that you need to check and click *Open*.



3. Click *Pre-Analyze*.

   The progress of the analysis and results scroll up in the Messages section.

The `Report.log` file opens displaying the results of the pre-analysis.



4. Click *Close.*

Pre-Analysis Report

To view the `Report.log` file, navigate to:
<Library_Import_Project_Directory>`\migration`

Review all the reported errors and fix them in the libraries. The `Report.log` file also contains references to utilities or scripts that you can use to correct the errors. You need to run these scripts from the Allegro EDM System Console. To speed up the process, you can launch more than one prompt and run them simultaneously.

Note: It is recommended that you fix all the pre-analysis errors before importing the data.

After correcting the errors, run the pre-analysis tool again to identify remaining errors, if any. Repeat this until all the errors are fixed.

Generating XML

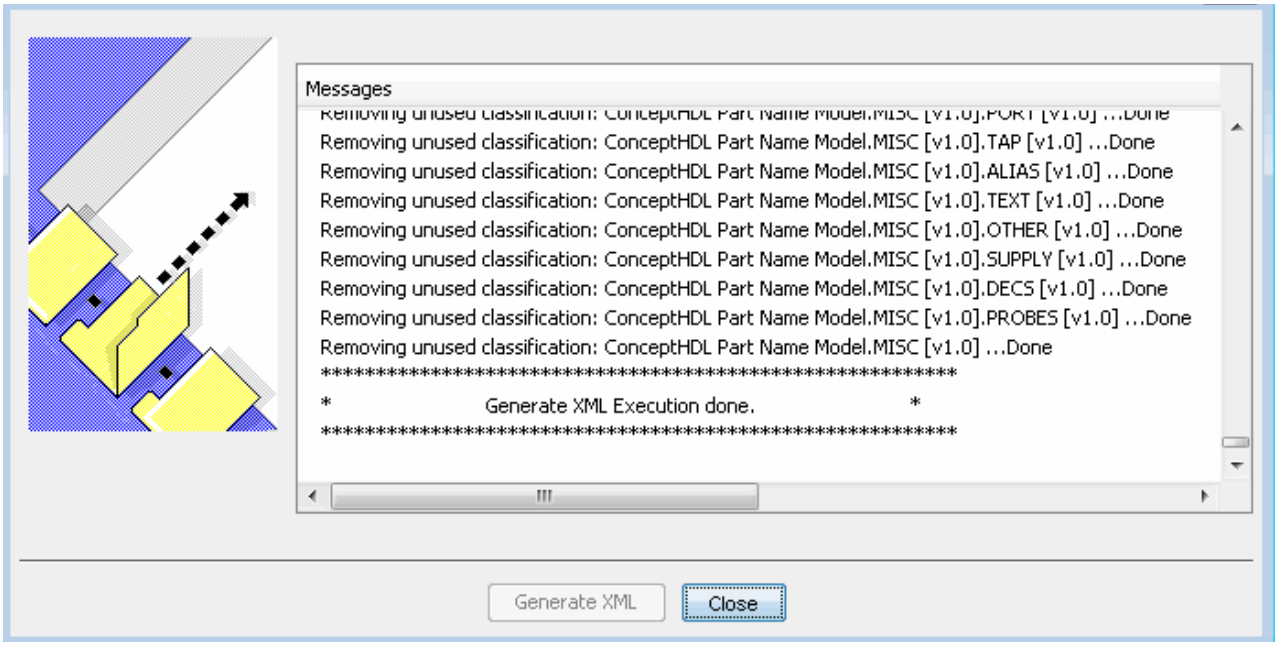
To generate the XML version of the source library data, do the following:

1. Click the *Extract Library Data* option in the Getting Started screen.

   The Generate XML (Project: `<Library_Import_Project_Path>`) screen appears.

2. Click *Generate XML*.

3. Click *Close* when the XML generation is completed.

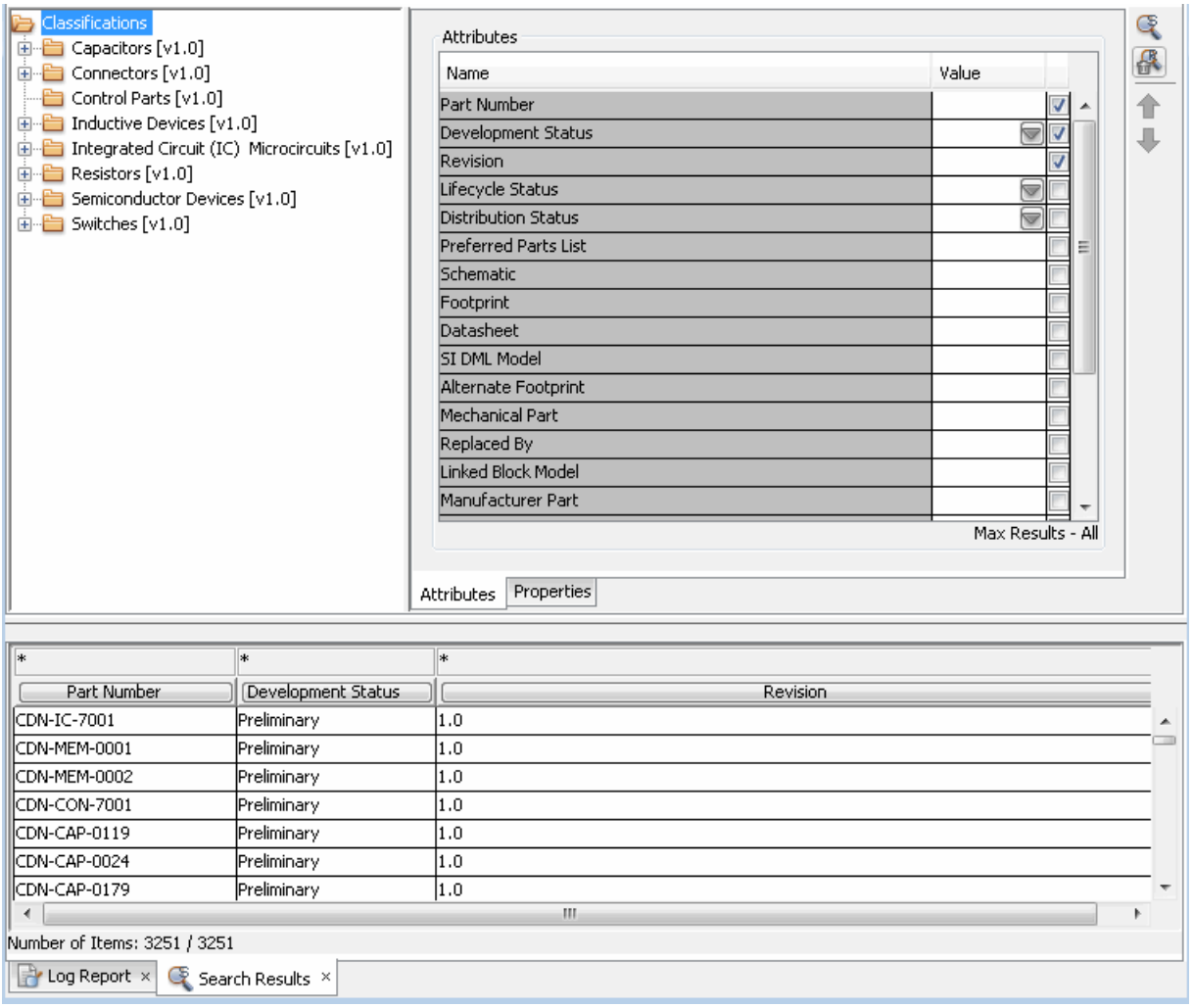


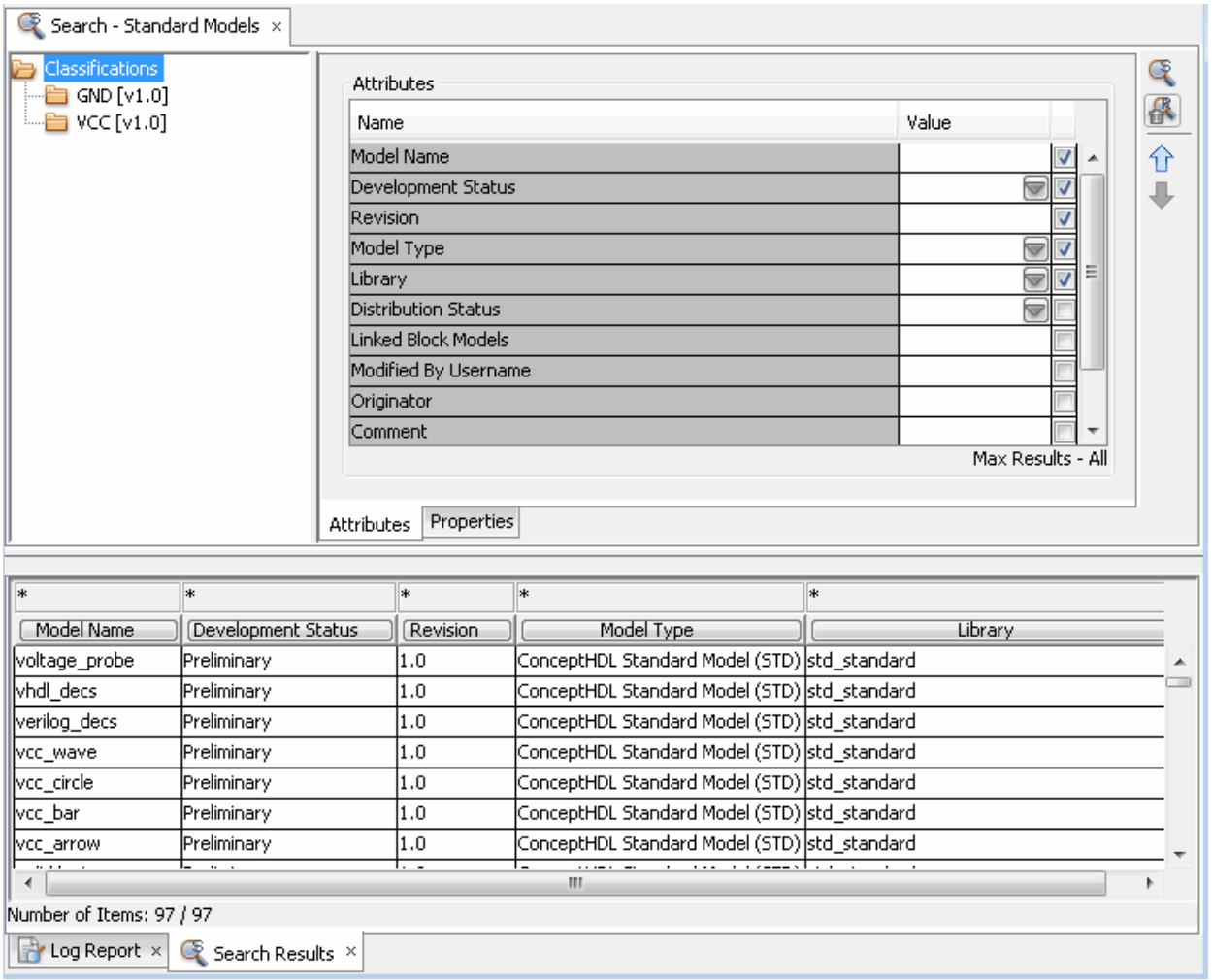
Opening Existing Projects

1. In the Getting Started dialog box, choose *Open an Existing Project* and click *Open*.

   The Allegro EDM Library Migration window appears with the project loaded.

The Select Project File (.cpm File) dialog box appears if you have not already selected the project file. Choose *<project>*.cpm, and click *Open*.





You can start making changes to the libraries. For more information on how to do this, see Modifying the Data Before Upload.

Modifying the Data Before Upload

The legacy part and model data specified for import may require modifications before you upload it to the database. Aligning your legacy data with the Allegro EDM data model helps you leverage the true benefits of product lifecycle management. For example, you may like to create classifications that facilitates part search and helps you move parts and models between appropriate classifications. Similarly, you can edit the part and model information such as attributes, searchable properties, and relations for efficient data management capabilities. These modifications can be made using the Library Import tool before uploading to the database.

Before uploading data, you can do the following:

- Manage part, model, and block part classifications. For more information on how manage the classification tree and the classification properties, see the *Working with Classifications* section in *Allegro EDM Database Editor User Guide*.
- View details of existing parts (mechanical parts and kits), block parts, models, libraries, and part and model classifications.
- Search for existing parts (mechanical parts and kits), block parts, models, and libraries by attributes, and searchable properties. For information on how to search for parts and models, see *Allegro EDM Database Editor User Guide*.
- Classify parts, block parts, and models
- Create associations

Important

The user interface of Library Import and some of its features are common to the Database Administrator and Database Editor tools. This guide only explains the unique features of Library Import. For other features, see *Allegro EDM Database Editor User Guide* and *Allegro EDM Database Administrator User Guide*.

Uploading the Data

When you have edited data in a library import session, you can upload the library data along with the changes to the database.

Note: You can access the data using the Database Editor tool only after you upload the edited data.

When importing Capture parts, check that they have all the required, associated models. If any models are missing, after merging the Capture parts with the Allegro EDM part, the Released status of Allegro EDM parts will not be modified despite the missing models in the Capture parts.

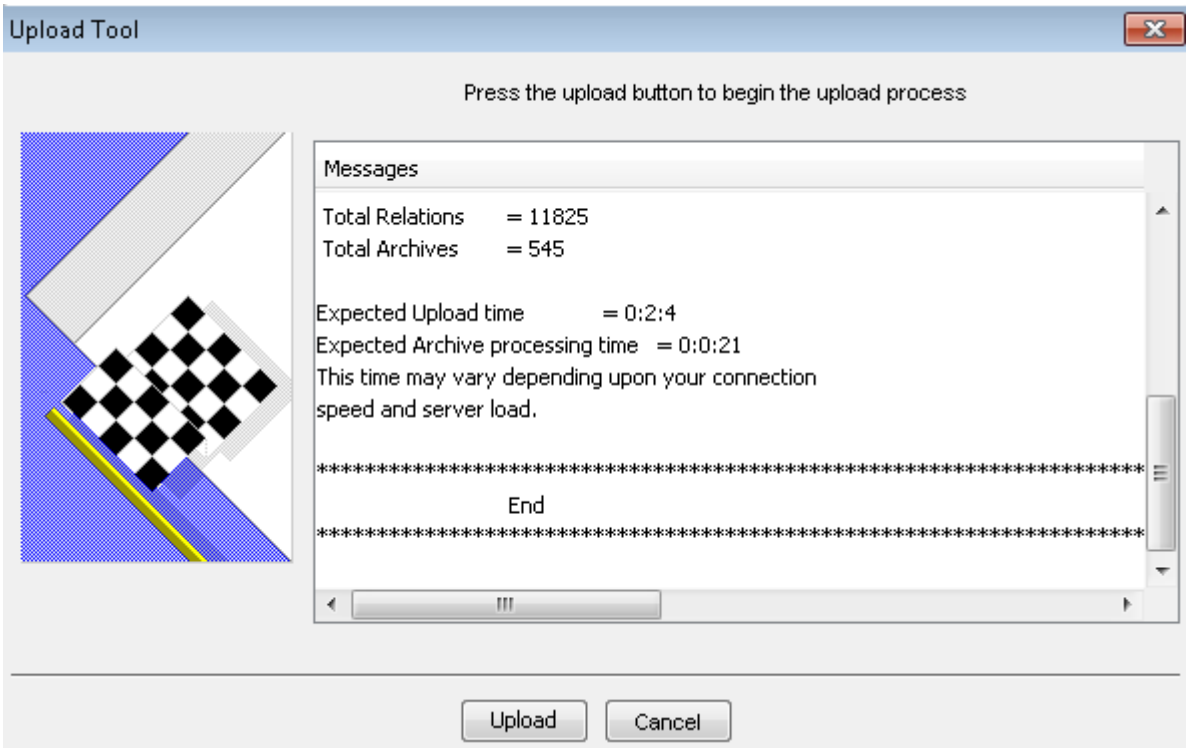
If there are conflicting values between common properties in the Allegro EDM and CIS database data for a particular part, maintain case sensitivity. The letter case of the property you import should be the same as that of the property in the Allegro EDM database.

If the values are not the same, Library Import appends `-Capture` to the Capture property. It is recommended that such a scenario be avoided to ensure conflicting information in the database.

To upload data, do the following:

1. In the Allegro EDM Library Migration window, choose *Tool – Upload Library Data* or click the Upload button () on the toolbar.

The Upload Tool dialog box appears with a summary of the data to be uploaded. It provides the expected time it will take to complete the upload operation along with details of the components to import.

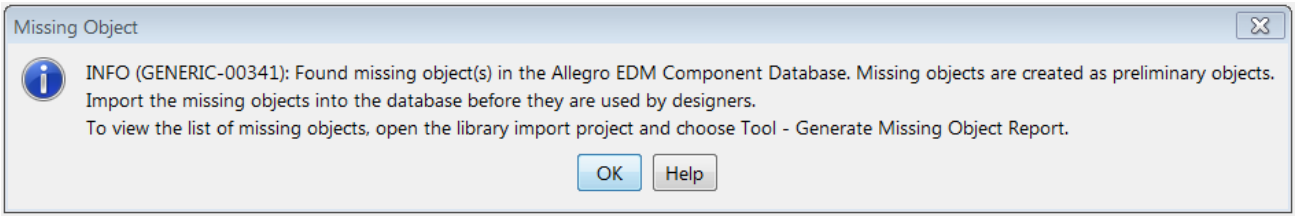


2. Click *Upload* to start uploading the data.

Important

The time to upload may vary depending on the amount of data to transfer and your server connection.

The following message appears if there are preliminary objects already in the database.



Click *OK*.

3. When the data is successfully uploaded, click *Close*.

Note: If there are any errors, they are displayed in the *Upload Tool* dialog box. Alternatively, you can see the log file at:
<Library_Import_Project_Directory>\migration\upload.log

Verifying the Upload

You should perform the following tasks after the upload process is complete:

- Post upload analysis
- Review errors and warnings

Post Upload Analysis

After you have ensured that the upload was successful, run the post analysis tool to check for objects that are in the preliminary state using *Tools – Generate Missing Object Report* in the Allegro EDM Library Migration window.

Ideally, this should not return any results, if the previous steps are performed. The results of this check are stored in a log file, *<Library_Import_Project_Directory>*\migration\MissingObjectReport.log, which you can open and review.

If you find some objects that are in the *Preliminary* state, that means that these models were not found during the upload process. You can import these models later by creating a separate library import project and then importing them separately.

Designers can use the database only after all the models are released and imported correctly.

Review Errors and Warnings

You should review all the error and warning messages after the upload process. To do so:

- Open this file and check for any errors or warnings:

  <Library_Import_Project_Directory>\migration\migration.err

- If there is an out-of-memory error, perform the steps listed in the Recommendations for Library Import section.

Post Library Import Tasks

After the library import is complete, you need to complete some tasks for your installation:

Note: If you imported Capture CIS libraries, also read Tasks after Capture CIS Data Import.

- Open Allegro EDM System Console and run `lib_dist`.

  The front-end libraries are added to `cds.lib` by the library distribution (`lib_dist`) process.

- To access back-end libraries that have been imported in the flows, you need to update the `PSMPATH`, `PADPATH`, and `MODULEPATH` entries in the `.env` files located at *<adw_conf_root>**<company>**<site>*\pcb.

  To generate the settings that are required to configure the `.env` files, do the following:

  a. In the Allegro EDM Library Migration window, choose *Tool – Generate Allegro Environment Settings*.

The `AllegroEnvSettings.txt` file is generated. Using the settings provided in this file, you need to update the `.env` files of your project workspace template(s) at *<adw_conf_root>**<company>**<site>*\pcb.

b. Open the required `<Allegro_EDM_Library_Flow>.env` and `<Allegro_EDM_Design_Flow>.env` file from `<adw_conf_root>\<company>\<site>\pcb` folder.

c. Copy the entries in the `Settings for Library Flow` section of the `AllegroEnvSettings.txt` file below the `set LIBPATH = $PCBDW_LIB/reflib` entry in the `<Allegro_EDM_Library_Flow>.env` file.

d. Copy the entries in the `Settings for Design Flow` section of the `AllegroEnvSettings.txt` file below the `set LIBPATH = $PCBDW_LIB/reflib` entry in the `<Allegro_EDM_Design_Flow>.env` file.

Alternatively, perform the following steps:

a. Launch Allegro EDM Configuration Manager.

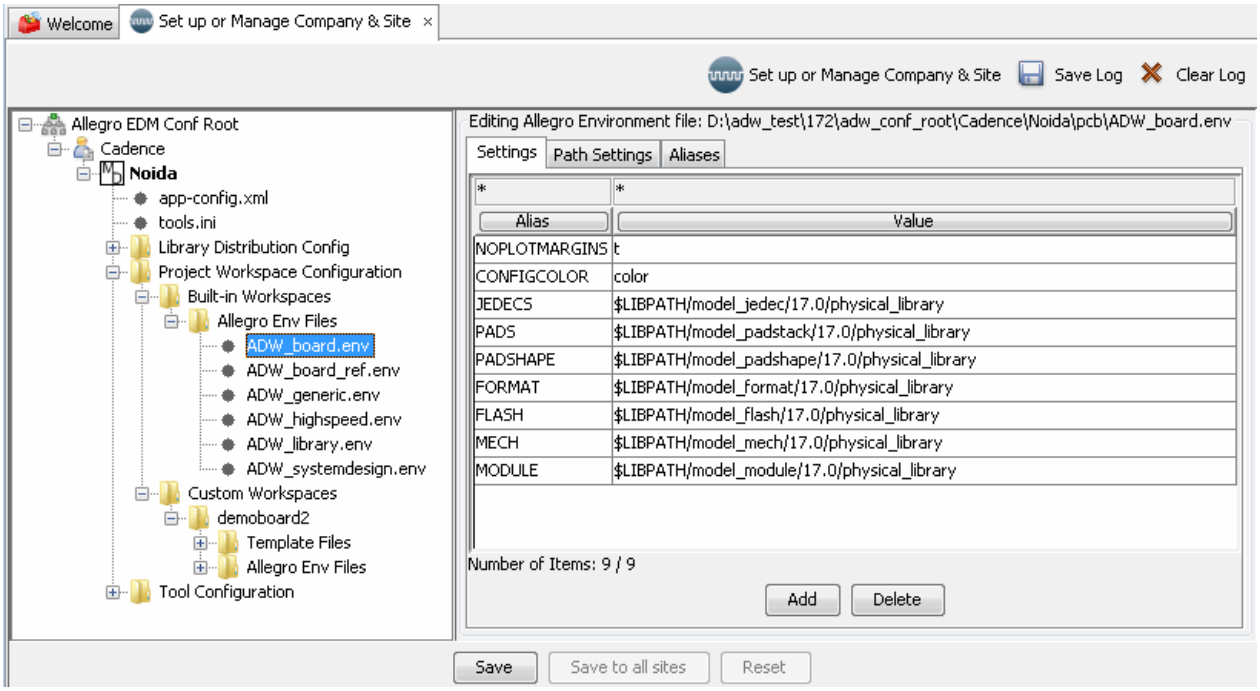
b. Click *Set up or Manage Company & Site*.

The *Set up or Manage Company & Site* tab displays the `workbench.ini` file.

c. On the left tree panel, choose *Allegro EDM Conf Root – <company> – <site> – Project Workspace Configuration – Built-in Workspaces – Allegro Env Files – ADW_<workspace_name>.env*.

For custom workspaces, navigate to *Allegro EDM Conf Root – <company> – <site> – Project Workspace Configuration – Custom Workspaces –*

<custom_workspace_name> – Allegro Env Files – ADW_<custom_workspace_name>.env.



d. In the right pane, edit the entries in the *Settings* tab.

e. Click *Save*.

- In the Allegro EDM Library Migration window, choose *Tool – Prerelease Parts Linked to Preliminary Models*.

  This option allows you to check the component database and mark as Pre Released all parts that are linked to a model in the Preliminary state. You need to perform this task, if before the upload process, you had set `SKIP_CHECK_PARTS_LINKED_PRELIM_MODELS = TRUE` in the `default.conf` file.

Tasks after Capture CIS Data Import

When you import Capture libraries into Allegro EDM, you need to do the following one-time tasks:

- After importing Capture CIS data into Allegro EDM, update the design cache and `capture.ini` accordingly.

  Reconfigure all the Capture library and PCB Editor footprint paths in `capture.ini` to point to the Allegro EDM Reference area. It is recommended that Capture users across

different geographies configure libraries using the same mapped drive so that designs created in one site can be used at other sites without a cache update for the library path.

For each OrCAD Capture design created prior to the Allegro EDM migration, choose *Tools – Utilities – Replace Path in Design Cache* to update the path in the Capture design cache.

- Note the following:
  - The SQLite database generated through the library distribution process is not password-protected.
  - EDM-specific metadata such as PPL, Lifecycle Status, Development Status is not available in the exported CIS database for use in Capture CIS.
  - Allegro EDM regenerates the database after each library distribution.

As part of your Capture setup, and to synchronize data with external systems, ensure that the Allegro EDM database is configured as the source CIS database. To integrate external systems with Allegro EDM, use Data Exchange.

Capture CIS import - Datasheet Models

If the value in the Datasheets column is imported as a model and is managed in Allegro EDM, configure the datasheet path as illustrated in the following example in OrCAD Capture:

```
[CIS Browse Directories]

Path1=D:\pcbdwlib\model_datasheets\

Path2=D:pcbdwlib\model_datasheets_cis/
```

Update the CIS properties using Part Manager so that all the changes related to datasheet values are updated.

3

Auto Generation of SI DML Models for Discrete Parts

Overview

The SPB core tools allow you to associate the SIGNAL_MODEL property on the discrete parts (2-pin components, resistors and capacitors pack discrete parts) with the auto generated SI DML model names.

Allegro EDM also provides the capability of assigning the SIGNAL_MODEL property value on the discrete parts with the auto generated SI DML model name. This helps the designer who is using the discrete parts with the task of auto-generation of SI DML models.

There are three ways that can enable you to generate SI DML model name:

- Using Library Import Wizard

  If you have large number of discrete parts in the Allegro EDM component database, then do the following:

  a. Associate the SIGNAL_MODEL property to the required schematic model classification using Database Editor (as an admin user).

  b. Configure Library Import setup to import SI DML models and choose the option to auto generate SI DML model name.

  c. Generate XML and upload the data using library import process.

  d. Run the library distribution command.

- Using Library Flow

  If you have only a selected number of discrete parts, then perform the following tasks using Database Editor:

  a. Associate the SIGNAL_MODEL property to the required schematic model classification.

b. Associate the SIGNAL_MODEL property to the related part classification.

c. Associate the discrete parts with the revised part classification.

d. Auto generate the SI DML model for the selected discrete parts.

e. Run the library distribution command.

For detailed information, see *Allegro EDM Database Editor User Guide*.

- Importing Library Data for the First Time

  If you are importing library data first time, then perform the following tasks:

  a. Add the SIGNAL_MODEL property to the PTF file of the discrete parts using PTF Editor.

  b. Configure Library Import setup to import SI DML models and choose the option to auto generate SI DML model.

  c. Generate XML and upload the data using library import process.

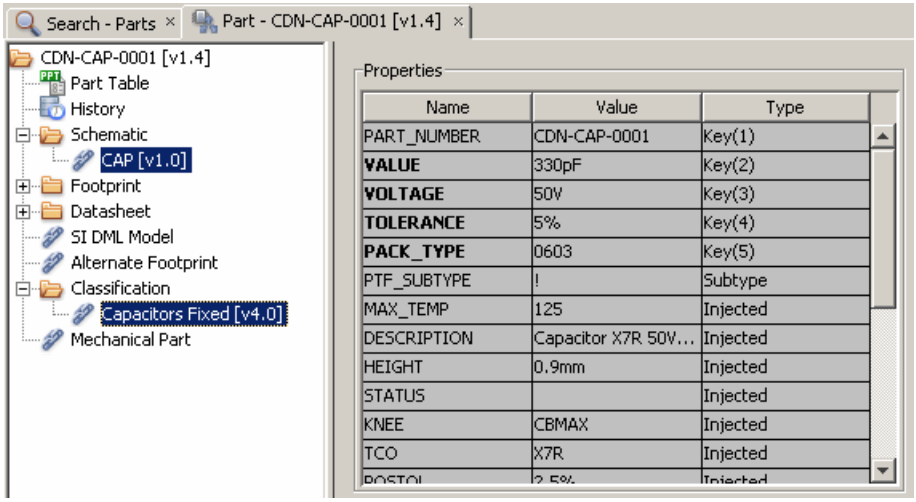
  d. Run the library distribution command.

Auto Generation of SI DML Model Using Library Import

Use Library Import when you need to auto generate the SI DML model name for all the discrete parts in the Allegro EDM component database. Consider the following example to understand the steps to follow.

For a set of discrete parts, including `CDN-CAP-0001[v1.4]`, the following associations exist:

- Schematic Model: `CAP[v1.0]`
- Schematic Model Classification: `Capacitors Fixed[v1.0]`
- Part Classification: `Capacitors Fixed[v4.0]`



Perform the following tasks to add the SIGNAL_MODEL property to the schematic model classification `Capacitors Fixed[v1.0]`:

1. Launch Database Editor (as an admin user).
2. Choose *File – Manage Model Classification – Schematic Model Classification*.
3. Choose `Capacitors Fixed[v1.0]` and click the Add button in the right pane to add to new property.
4. Enter the property name `SIGNAL_MODEL` and click *Create*.
5. Add the *Link To* value for *SIGNAL_MODEL* property as `SI DML Model.Model Name`.
6. Save the classification.

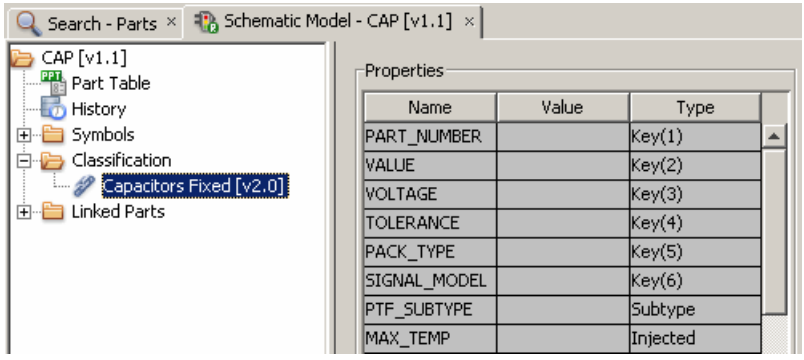
   The Update message appears asking you to select from the three options, Preserve, Check-out, or Release.
7. Click the required option. For this example, click *Preserve*.

   To learn about other options in the Update message, see *Allegro EDM Database Editor User Guide*.

   The Check-in Log for Model: `CAP` dialog box appears.
8. Enter the log details and click *Apply to All*.

   You will see the creation of revised:

   - Schematic Model Classification: `Capacitors Fixed[v2.0]`
   - Schematic Model: `CAP[v1.1]`



Next, you need to:

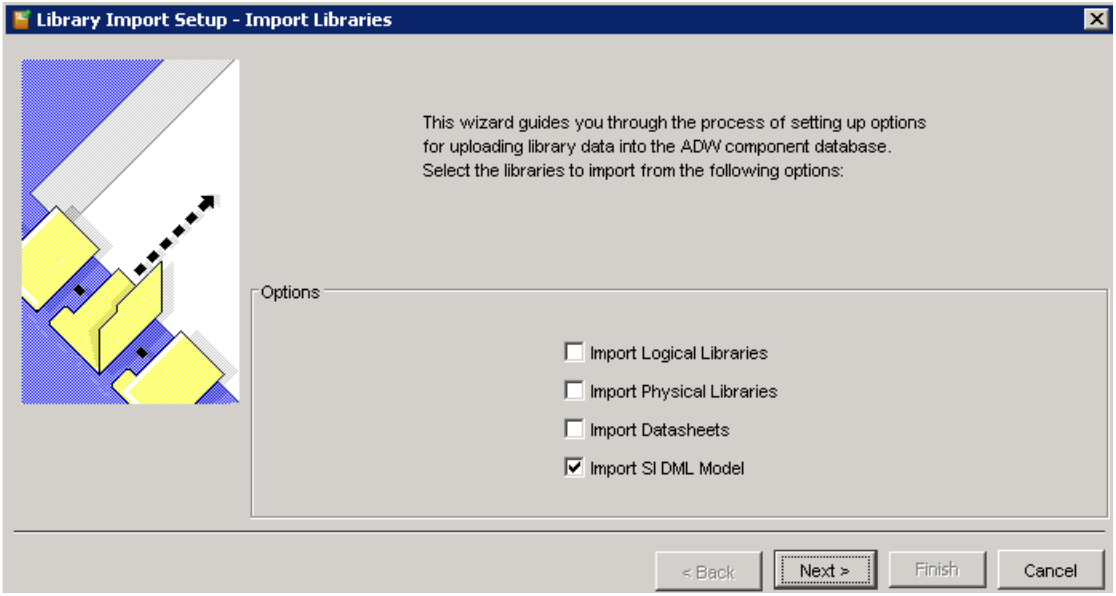
a. Add the SIGNAL_MODEL property on the related part classification `Capacitors Fixed[v4.0]`.

b. Auto generate the SI DML model for all discrete parts with part classification `Capacitors Fixed[v4.0]`.

This is done using the library import process by performing the following the tasks:

1. Launch Library Import.
2. Choose *Configure Import Options*.
3. Select any library project.
4. Click *Open*.

   The Library Import Setup - Import Libraries wizard page appears.

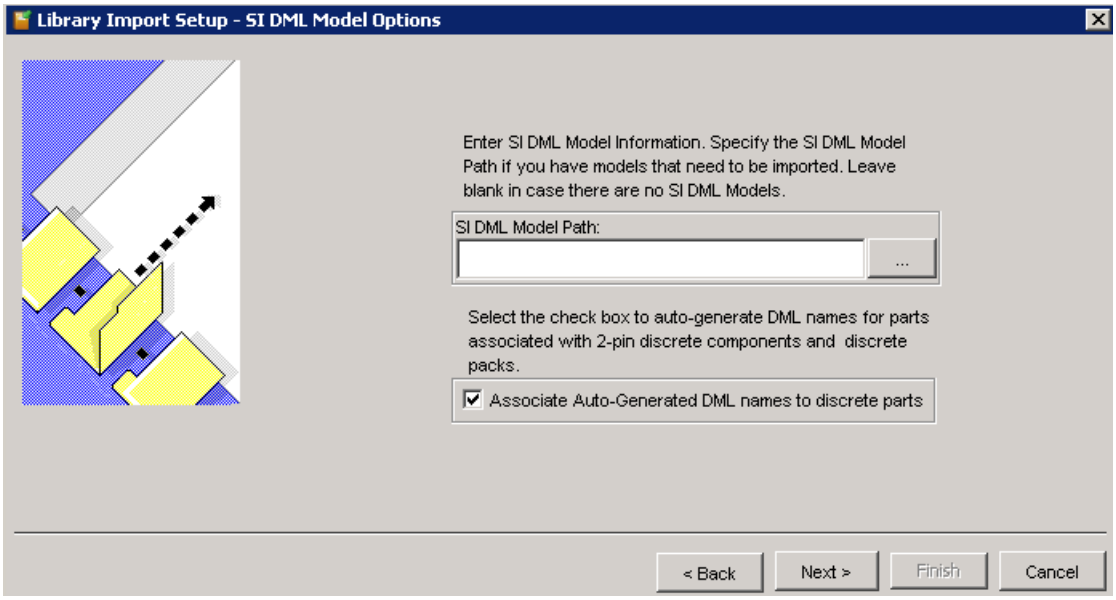


5. Select only *Import SI DML Model* to import SI DML models.
6. Click *Next*.

   The Library Import Setup - SI DML Model Options wizard page appears.

7. Delete the path populated in *SI DML Model Path*, if you have already imported the SI DML model libraries.

8. Select the *Associate Auto-Generated DML names to discrete parts* check box.



9. Click *Next*.

   The Library Import Setup - Configure Identical Properties wizard page appears. To configure the import options, choose one of the options from this wizard page.

10. Click *Next*.

    The Library Import Setup - CSV Classification Options wizard page appears.

11. Delete any paths specified for the CSV classification files. However, if you are importing the SI DML model libraries, and want to associate any classification, then specify the required path.

12. Click *Next*. The Library Import Setup - Summary page appears.

13. Click *Finish*.

14. Choose *Extract Library Data* in the Getting Started dialog box. The Generate XML (Project: `<Library_Import_Project_Path>`) screen appears.

15. Click *Generate XML*.

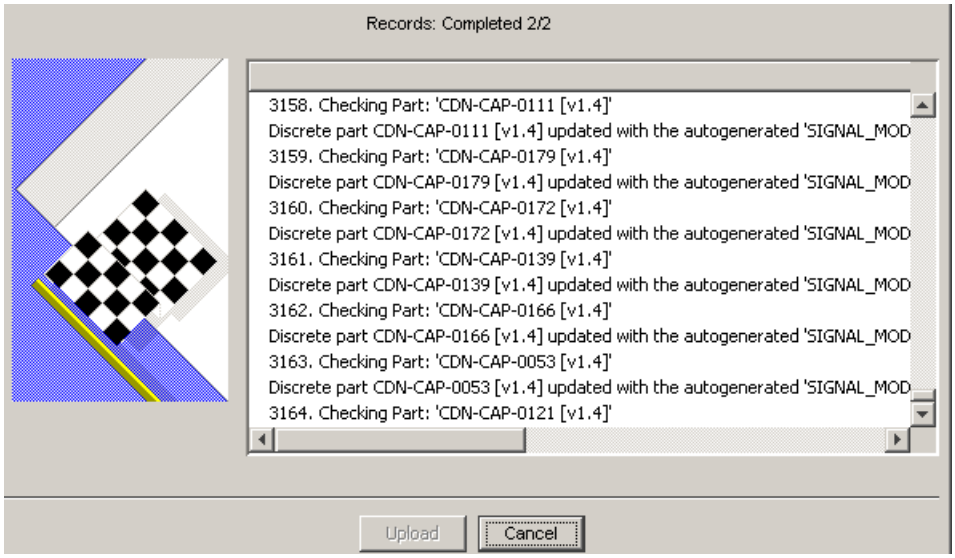
16. Click *Close* when the XML generation is completed.

17. In the Getting Started dialog box, choose *Open an Existing Project* and click *Open*.

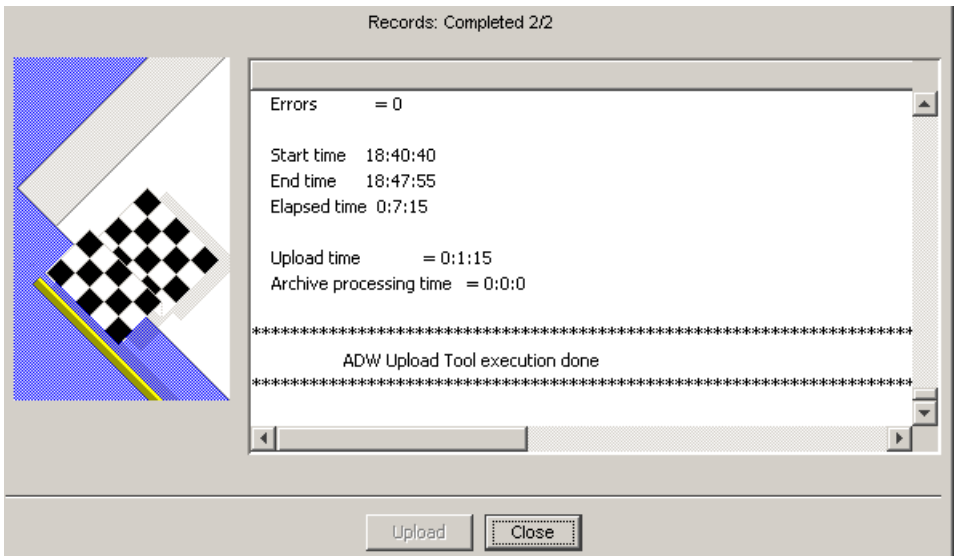
    The Allegro EDM Library Migration window appears.

18. Choose *Tool – Upload Library Data*. Alternatively, click  on the toolbar. The Upload Tool dialog box appears.

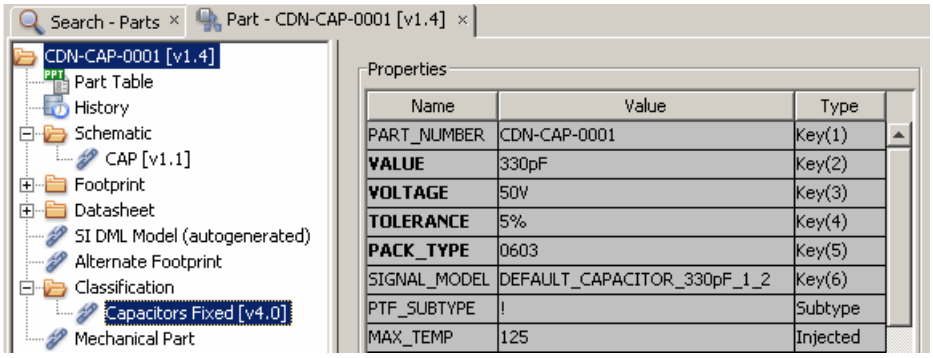
19. Click *Upload* to start uploading the data. The upload process shows the discrete parts that are getting associated with the auto-generated SI DML model name.



20. Click *Close* when the data is successfully uploaded.



This process modifies the part classification `Capacitors Fixed[v4.0]` to include the *SIGNAL_MODEL* property and populate its value on all the associated discrete parts including `CDN-CAP-0001[v1.4]`.



21. Run the library distribution command.

This process updates the `part_table.ptf` file to include the `SIGNAL_MODEL` property.

```
PART 'CAP'

:PART_NUMBER|VALUE|VOLTAGE|TOLERANCE|PACK_TYPE|SIGNAL_MODEL=MAX_TEM
P|HEIGHT|STATUS|KNEE|NEGTOL|DIST|SLOPE|TOL_ON_OFF|IC|CURRENT|MANUFA
CTURER|TC|DESCRIPTION|POSTOL|TCO|JEDEC_TYPE|M

'CDN-CAP-
0001'|'330pF'|'50V'|'5%'|'0603'|'DEFAULT_CAPACITOR_330pF_1_2'
('!')='125'|'0.9mm'|''|'CBMAX'|'2.5%'|'FLAT'|'CSMAX'|'ON'|'UNDEF'|'
CIMAX'|'AVX'|''|'Capacitor X7R 50V 330pF 5%
0603'|'2.5%'|'X7R'|'CAPC1608N'|'06035C331JAT2A'|'NON-
COMPLIANT':CDS_LW_PART_NUMBER='CDN-CAP-0001'
```

A

Library Import User Interface

This chapter details the following user interface components of the Library Import tool:

- Menus
- Library Migration Window
  - Panes
  - Tabs
- Dialog Box Help

Menus

- File Menu
- View Menu
- Search Menu
- Tool Menu
- Window Menu

File Menu

Command	Lets you
New – Library	Create a library in the library import project.
Open – Library	Open the library corresponding to the *Linked Model Type* selected.
Open – Part	View the existing parts available in the library import project.
Open – Block Part	View the existing block parts available in the library import project.
Open – Mechanical – Mechanical Part	View the existing mechanical parts available in the library import project.
Open – Mechanical – Mechanical Kit	View the existing mechanical kits available in the library import project.
Open – Model – <Model_Type>	View existing models available in the library import project. Models supported appear as submenus.
Project Option	Open the Getting Started window.
Manage Part Classification	Manage part classification in the library import project.
Manage Mechanical Part Classification	Manage mechanical part classification in the library import project.
Manage Block Part Classification	Manage block part classification in the library import project.

Command	Lets you
Manage Model Classification – <Model_Type> Model Classification	Manage model classifications. Models supported appear as submenus: ■ Standard Model Classification ■ Schematic Model Classification ■ Mechanical Model Classification ■ Block Model Classification ■ Footprint Model Classification ■ Padstack Model Classification ■ Flash Model Classification ■ Shape Model Classification ■ Board Model Classification ■ Format Model Classification ■ Module Model Classification ■ Datasheet Model Classification ■ SI DML Model Classification
Save	Save the changes made in the active tab into the library import project.
Save All	Save all the changes made in multiple tabs into the library import project. **Note:** *Save* and *Save All* commands are disabled in all the Search tabs.
Delete	Remove the selected object data from the library import project permanently. **Note:** The *Delete* command is disabled in all the Search tabs.
Exit	Close the Library Import.

View Menu

Command	Lets you
Search Results	Open or close the Search Results tab.
Show Logs	View the log details of any operation performed.
Toolbar	Control the visibility of the toolbar.
Refresh	Refresh the Search Results tab.

Search Menu

Command	Lets you
Search – Library	Search existing libraries in the library import project.
Search – Part	Search existing Parts in the library import project.
Search – Block Part	Search existing Block Parts in the library import project.
Search – Mechanical – Mechanical Part	Search existing Mechanical Parts in the library import project.
Search – Mechanical – Mechanical Kit	Search existing Mechanical Kits in the library import project.
Search – Model – <Model_Type>	Search existing Models in the library import project. Supported models appear as submenus.

Tool Menu

Command	Lets you
Upload Library Data	Upload the library data along with the changes to the database.

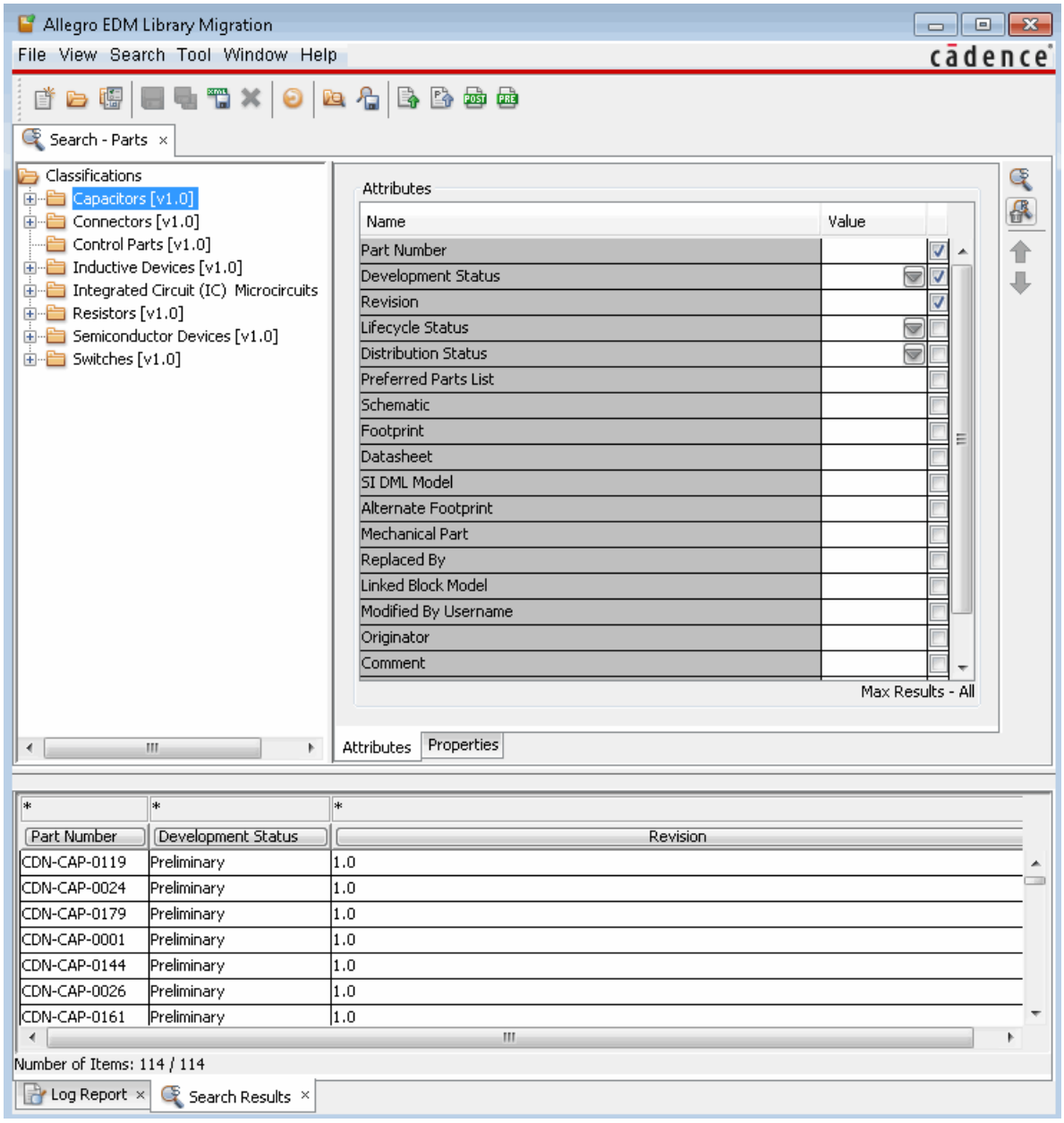
Command	Lets you
Prerelease Parts Linked to Preliminary Models	Check the component database and mark as Pre Released all parts that are linked to a model in the Preliminary state. This is a post-upload task. You need to perform this task if, before the upload process, you had set `SKIP_CHECK_PARTS_LINKED_PRELIM_MODELS = TRUE` in the `default.conf` file.
Generate Missing Object Report	View the `MissingObjectReport.log` file and check for all the parts with preliminary status, which were added to the database in the last upload session.
Generate Pre-Analysis Report	View the `Report.log` file and correct the errors in the current project's libraries as reported by the Pre-Analyze option.
Generate Allegro Environment Settings	Generate the Allegro environment settings required to be added or updated in the PCB Editor environment files. The `AllegroEnvSettings.txt` file is generated. Using the settings provided in this file, you need to update the `.env` files of your project workspace template(s) at *<adw_conf_root>**<company>**<site>*\pcb.

Window Menu

Command	Lets you
Single Detail Window	Control tab-based views in the window.
Recently Viewed	View the history of the tabs you have recently opened in the main window.

Library Migration Window

The interface of Library Migration is similar to Database Editor tools.



Many of the features available in these two tools, are available in Library Import. The main components of the window are:

1. Menu. See, Menus

2. Toolbar
3. Panes
4. Tabs

Panes

The panes are similar to those available in Database Editor and Database Administrator tools:

- Explorer pane
- Right pane

These panes are context-specific, and their content and visibility depend on the component database entity selected (part or model) and the type of operation being performed. For example, when you search for Models, the *Search - <Model_Name>* classifications hierarchy appears in the Explorer pane; when you choose a node in the classification hierarchy, then its attributes and properties appear in the right pane.

Similarly, when you view any part detail or model detail tab, then the explorer pane contains the existing relations for the part or model selected.

Explorer Pane

This pane provides a hierarchical view of part and model classifications. This hierarchical view may also represent relations associated with data entities, such as libraries.

Context-sensitive Pop-up Menus

The pop-up menus are similar to those available in Database Editor. Depending on the level of node you select in the explorer pane, the pop-up menu can help you:

- View the details of a classification using the *Show Details* command.
- Create and remove relations. It contains commands such as: *Associate <Relation_Name>* and *Associate <Relation_Name> from Tree*, *Delete*, and *Paste*.

  Note: Parent nodes contains the *Refresh* pop-up menu command.

Right Pane

This context-sensitive pane displays different views depending on the type of operation you perform. The active tab determines the view of the pane, which includes:

- The editable and non-editable views of the part and model data selected.
- When you perform a search, this pane allows you to specify search criteria for attributes and properties.

Note: You can use the Tab key to navigate to various editable fields in the right pane.

Tabs

The Library Migration window allows you to open as many tabs as you want to. A tab can be of the following types:

- Search Tab: This tab appears (below the main menu) when you choose to search for library data (or administrator data such as model type, library, and PPL).
- Details Tab: Part or model detail and related component data appears in the Detail tab. Whenever you open a part or a model, its detail appears in form of a *Part - <Part_Detail>* tab or *<Model_Type> - <Model_Detail>* tab, respectively.
- Search Results Tab: All the search results appear at the bottom of the Library Migration window. This pane also shows the number of results that match your search criteria at the bottom-left of the Search Results tab
- Log Report Tab: This tab appears before the *Search Results* tab, at the bottom and shows all the log details for each operation performed in Library Migration.
- <Object_Type> Classification Tab: This tab appears when you choose *File – Manage <Object_Type> Classification*.
- Edit All Tab: This tab appears when you select library data (parts and models) in the *Search Results* tab, and then choose *Edit All* pop-up menu option. This tab allows you to edit multiple parts or models.

Dialog Box Help

New Dialog Box

This dialog box appears when you create a new library.

Open Dialog Box

This dialog box appears when you view existing library data. This requires you to enter a name for the part or model, you want to view.

Associate <Relation_Name> Dialog Box

This dialog box allows you to choose a relation instance you want to associate with the selected relation.

Delete <Relation_Name> Dialog Box

This dialog box allows you to choose a relation instance to be deleted for the selected relation.

Add Property Dialog Box

This dialog box allows you to enter the name of a new property.

Delete Property Dialog Box

This dialog box allows you to enter the name of the property you want to delete.

Select <Object_Type> Dialog Box

This dialog box appears, when you try to open a duplicate object, which means that two or more objects exist with the same name. This dialog box contains a table, with all the objects with the same name. Select an object and click *OK* to view its details.

B

Standard Library Support

Overview

Component libraries that contain standard symbols are supported in Allegro EDM. You can configure and import these symbols as standard models into Allegro Library Manager.

- If you have symbols that were converted to standard models through the Allegro EDM uprev process (`adw_uprev` run while migrating to a newer release), run the library import process to import these new standard models:

  a. Configure standard models.

  b. Run pre-analysis.

  c. Upload library data.

- If you are using standard models in Allegro EDM projects, you need to update the design projects to manage these standard models.

  Perform the following tasks to update the design projects:

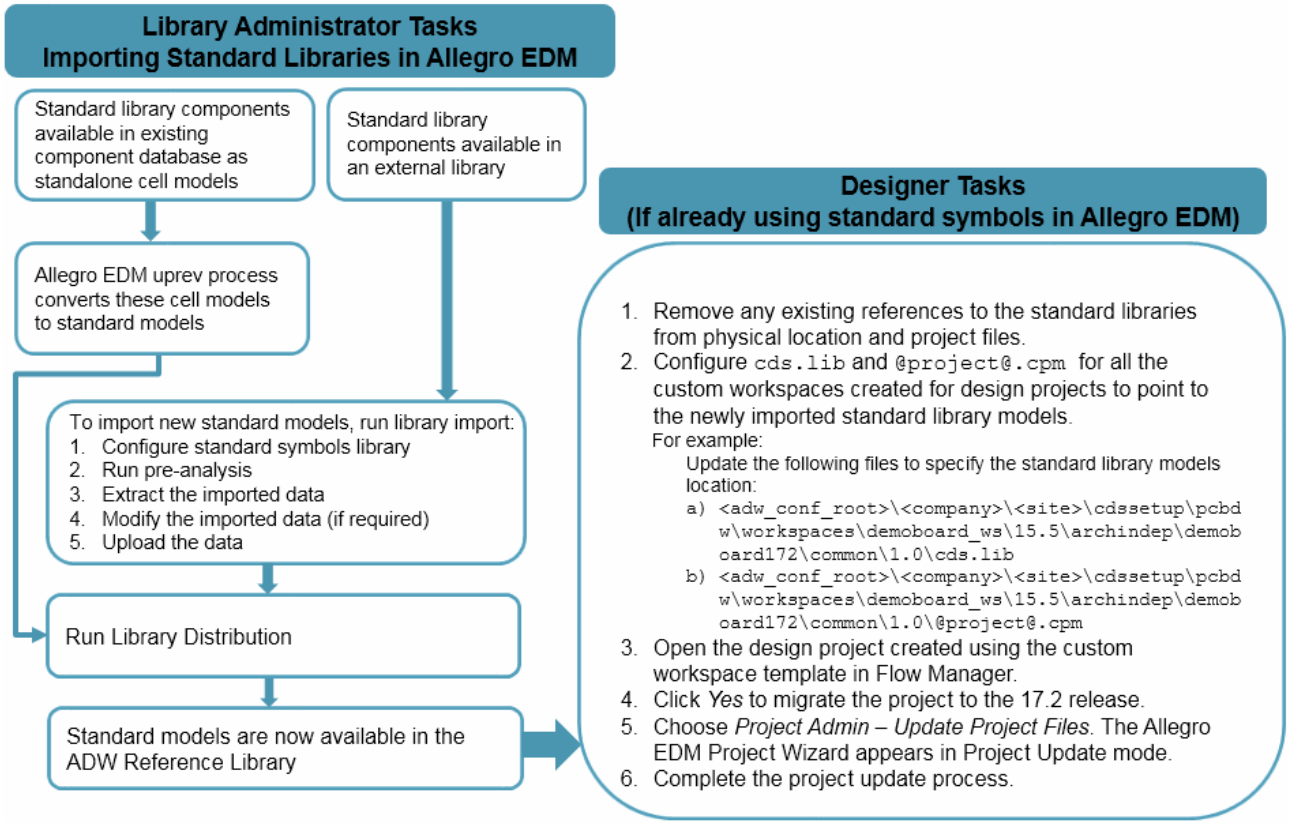
  a. Remove any existing references to the standard libraries from physical location and project files.

  b. Modify your project workspace templates.

  c. Migrate the existing designs.

Work Flow

The following diagram shows the standard library support flow.



Index

Symbols

A

B

C

I

K

L

M

O

U

V