

Allegro® X EDM Design Management User Guide

**Product Version 23.1
September 2023**

© 2023 Cadence Design Systems, Inc. All rights reserved.

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

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

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

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

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

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

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

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

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

Contents

<u>Preface</u>	5
<u>About This User Guide</u>	5
<u>Audience</u>	5
<u>Related Documentation</u>	5
<u>Cadence Web Resources</u>	6
<u>Related Tools and Flows</u>	6
<u>Typographic and Syntax Conventions</u>	6
<u>1</u>	
<u>Introducing Allegro Design Management</u>	7
<u>Overview</u>	8
<u>Allegro Design Management Benefits</u>	8
<u>Team Design Concepts</u>	9
<u>Design Management Flow for Flat and Hierarchical Designs</u>	11
<u>Setting up Allegro Design Management</u>	14
<u>Setting up Allegro Design Management</u>	14
<u>Preparing Designs for Design Management</u>	22
<u>2</u>	
<u>Enabling Projects for Design Management</u>	29
<u>Overview</u>	29
<u>Launching Allegro Design Management and Loading a Project</u>	30
<u>Enabling a Project for Design Management</u>	30
<u>Defining Users as LDI or PDI</u>	45
<u>3</u>	
<u>Joining a Design Project</u>	47
<u>Joining a Project</u>	48

4

<u>Working with Designs in Allegro Design Management</u>	1
<u>Integrator Tasks - Defining Project Management</u>	2
<u>Working with Flat Designs</u>	2
<u>Working with Hierarchical Designs</u>	3
<u>Creating Initial Design Hierarchy: Recommendations</u>	3
<u>Integrator and Designer Tasks</u>	4
<u>Checking Out a Design, Subdesign, or Page</u>	5
<u>Modifying a Design</u>	6
<u>Rolling Back Changes or Undoing Check Out</u>	10
<u>Checking In a Design, Subdesign, or Page</u>	11
<u>Changing the Root Design</u>	13
<u>Synchronizing with Shared Area</u>	13
<u>Releasing Check Out</u>	14
<u>Resolving Unauthorized Design Modifications</u>	15
<u>Deleting Objects from the Allegro Design Management Dashboard</u>	16
<u>Working with Flat Designs in Design Entry HDL</u>	23
<u>FAQs - Working with Managed Projects</u>	23
<u>Allegro Design Management User Interface in DE-HDL</u>	26

5

<u>Adding Derived Data to Allegro Design Management</u>	1
<u>Generating Default Derived Data</u>	1
<u>Derived Data Controls in the Allegro Design Management Policy File</u>	2
<u>Creating Custom-Derived Data Objects</u>	9
<u>Tasks Overview</u>	9
<u>Making Policy File Changes to a Managed Project</u>	17
<u>Updating Object Definitions in a Project</u>	18

6

<u>Cache-Enabled Projects</u>	1
<u>Additional Internal Global Objects</u>	1
<u>Allegro EDM Design Management Modes</u>	1
<u>Impact of Design Mode on Design Management Operations</u>	2

Allegro X EDM Design Management User Guide

<u>Join Allegro EDM Project</u>	2
<u>Schematic Check Out</u>	3
<u>Schematic Check in</u>	3
<u>Show Difference and Update</u>	3
<u>Reuse Blocks in Cache-Enabled Projects</u>	4

7

<u>Allegro Design Management User Interface</u>	1
<u>Dashboard Window</u>	1
<u>Allegro Design Management Start Page</u>	4

Allegro X EDM Design Management User Guide

Preface

About This User Guide

Allegro® Design Management User Guide explains how to create schematic designs and boards concurrently to reduce the overall design time. This guide is for Design Entry HDL users.

This guide describes the following tasks and their related concepts:

- Preparing the design hierarchy
- Setting up a project for a team of designers
 - Creating a shared area
 - Assigning team members to sub-blocks
- Monitoring concurrent design activities
- Managing data management tasks
- Integrating a design

Audience

The audience for this document includes:

- Integrators who initiate design management for a project and who, at the end of the design phase, merge all the subdesigns, resolve any clashes, and have master ownership of all the design components.
- Designers (engineers) who work on schematic and board files in Design Entry HDL.
- Site administrators who define site-level settings for Allegro Design Management.

Related Documentation

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

- To create custom project workspaces, refer to *Allegro EDM Flow Manager User Guide*.

Cadence Web Resources

For a list of Allegro EDM-related Known Problems and Solutions, visit [Cadence Online Support](#).

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, refer to *Allegro PCB Design Flows*.
- To learn how to create and configure Design Entry HDL projects, refer to *Allegro Project Manager User Guide*.

Typographic and Syntax Conventions

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

<code>literal</code>	Nonitalic words indicate keywords that you must enter literally. These keywords represent command (function, routine) or option names.
<i>argument</i>	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.

Introducing Allegro Design Management

Allegro Design Management provides two primary functionalities for Design Entry HDL users:

- work-in-progress (WIP) data management using Allegro Data Manager
- team collaboration also referred to as Team Design Option or TDO

Both functionalities are provided through an application called Allegro Design Management.

Allegro Data Manager

Using the same Part Information Manager as the Allegro Library Manager, Allegro Data Manager allows component searches based on parameters that are linked to your company's components database, promoting the use of approved and preferred parts. This helps to speed up component research time.

Because Allegro Design Management manages schematic and board files separately, it offers the ability to manage multiblock hierarchical, and flat designs, with multiple users concurrently modifying portions of the logical and physical design including secure shared areas for local and globally dispersed design teams.

Team Design

Team design in Allegro Design Management allows multiple design engineers or PCB designers to collaborate asynchronously in the development of a logical design or PCB. Schematic designs can be user-partitioned at the hierarchical block level, and assigned to specific members of a team, providing them with an isolated workspace for the development and verification of their partitions, blocks, or subdesigns.

A dashboard also allows users to view the state of various sub-designs, the current versions of a shared project versus a local working project, and a set of functions that allow the management of sub-designs.

Overview

Apart from supporting the bottom-up methodology, Allegro Design Management also supports a top-down methodology where a system can be partitioned into subsystems and reintegrated upon design completion.

Using Allegro Design Management, you can:

- Prevent unintended modification of a design (control access)
- Provide version control for all design changes
- Manage design modifications from multiple sources/sites
- Facilitate communication/notifications among the design team
- Maintain the design data at a central location
- Integrate all design components into a released design

Allegro Design Management Benefits

- Reduce schematic creation time.
- Share subdesign objects.
- Enable automated synchronization of a system.
- Manage individual files, such as board files, under a physical view.
- Manage board partitions to create independent regions.
 - Place and route physical subdesigns independently.
 - Refresh partitions back to the main board.
- Reduce design capture time.
- Collaborate across distributed teams.
- Manage peer-to-peer updates.

Team Design Concepts

Concepts in team design are as follows:

■ Shared Area

This is a common area for exchanging project data and is:

- ❑ populated after you enable a design project for team design. This process is often referred to as enable team design, or ETD.
- ❑ updated when a designer checks in or checks out an object.

Users have permissions on design objects as assigned by the integrator.

The shared area is a central location for which all team members have write permission so that they can save their work, that is, a file system.

■ Work Area

Each team member works on a copy of the project to ensure a stable view of the design. This local copy of the project is called the work area. This is created when a design team member joins the project.

- ❑ Each team member decides when a block is ready to be checked in; once checked in, the other users are notified.
- ❑ Team members are notified when a block is updated in the shared area.
- ❑ Team members control when to refresh the block submitted by another team member.

As the person who enables a design project for team design, the integrator automatically becomes a part of the project after enabling it for team design. As a result, the integrator does not need to join the project.

■ Design Objects

This includes project data that is classified under the following categories.

- ❑ **Schematic (Logical):** The contents of a block except the physical, package, and symbol view are a part of a schematic design object.
 - Schematic pages and constraints in flat designs
- ❑ **Layout (Physical):** This object consists of files under the physical view and thus may have multiple board files within one object. It may contain the following:
 - Board Files

Allegro X EDM Design Management User Guide

Introducing Allegro Design Management

- Partitions
- ❑ **Package:** All the contents of the packaged view under a block are part of this object.
- ❑ **Symbol:** All the contents of the block symbol view are part of this object.
- ❑ **Derived Data:** You can generate a BOM or PDF as derived data. You can also have additional derived data. For detailed information, see [Creating Custom-Derived Data Objects](#).
- ❑ **Global Objects:** Contains the files or folders that are not part of blocks or the design library.

The following global objects are modified by both, Cadence tools, and by Allegro Design Management users. If global objects are modified by Cadence tools, they are automatically checked out and checked in by Allegro Design Management.

- *project_settings*: Contains files such as `.cpm`, `cds.lib`, various Allegro EDM-specific files such as `.ini` files, reports, and log files.
- *shopping_cart*: This is available only with Allegro Data Manager.
- *design_cache*: Created only when a design is moved into the ECO mode for any `board` or `highspeed` type of project. This is available only with Allegro Data Manager.

Allegro Design Management also allows you to create and manage user-defined global objects. You need to explicitly check out these objects, modify them, and check them in. For example, *ecsets* is one such object.

Note: This is the default list of design objects and can be modified according to the organization's requirements.

Team design operations, such as check in and check out, are available for these objects.

■ Policy File

The policy file defines how a design management project is divided into design objects and what each design object contains, and how these objects are created and managed by Allegro Design Management. For example, it defines `sym_1` as a design object, which contains the symbol view of the block. Each design object can then be checked out and checked in by users.

The policy file also defines how shared design data, such as objects, commands, and rules for operations in Allegro Design Management, will be managed in the workspace.

Cadence provides three default policy files in `<Cadence installation directory>\share\cdssetup\sdm\policies`:

- ❑ `scm_policy.xml`
- ❑ `sda_policy.xml`
- ❑ `sdm_policy.xml`

These policy files are specific to the structure of the application you are using. For example, `scm_policy.xml` is meant for use with System Connectivity Manager. The `sda_policy.xml` is used with Allegro System Capture.

As an integrator, you will select the policy file to be used with your design project when you enable a project for team design. Administrators can define custom policy files depending on the requirements of their design projects.

■ Shared Data

Shared data is common design data that is used by all design engineers who are working as a team. It includes schematic libraries, highspeed models, site area, Electrical Constraint Sets (ECSets), and technology files.

Such design data can be managed by the integrator by defining them in Allegro Design Management as external global objects. Whenever this data is modified, design team members can access the updates.

Design Management Flow for Flat and Hierarchical Designs

When you create a design that will be worked upon in a team design environment, the usual flow is as follows:

1. As an integrator, you create a flat or hierarchical design using a design authoring tool, such as Design Entry HDL. For example, let us assume that this design project is called *Sample_Design_Project*.
2. Enable the project for team design to enable team collaboration. As the integrator, you will do the following when you enable a project for team design:
 - a. Define the shared area. For example, assume that you have named the shared area *Sample_Design_Project_SharedArea*.

After you define a shared area and complete the team design process, you will have two folders:

- Your own copy of the project. In this example, that would be *Sample_Design_Project*.
- *Sample_Design_Project_SharedArea*

Allegro X EDM Design Management User Guide

Introducing Allegro Design Management

This shared area project is what team members will join. Changes they make to their copies of the project, which they then check in, will be updated in this area.

b. Define a project team.

c. Specify the owners of blocks, pages, constraints, subdesigns, and so on. All of these are commonly referred to as design objects.

- Flat designs: Specify the owners of schematic pages and constraints.
- Hierarchical designs: Specify the owners of the schematic, blocks and subdesigns of the main design.

Note: In the documentation, the terms *managed projects*, *pages*, *blocks*, and *constraints* have sometimes been used to refer to objects in projects that have been enabled for team design.

3. Notify team members about the location of the design project and shared area. This allows these team members to join the project so that they can start working on the objects to which they have been assigned.

When you enable a project for team design, Allegro Design Management prompts you to notify team members of the location of the project and shared area. You will usually notify team members of these locations by e-mail.

4. Team members join the design project from Allegro Design Management.

As a team member, when you join a project that has been enabled for team design, you will specify a local work area.

On joining the project, Allegro Design Management fetches the design from the shared area into your local work area. Essentially, it downloads a copy of the project that is in the shared area into your work area.

5. Designers start to work on the pages that they own (flat designs) or blocks that they own (hierarchical designs).

Allegro Design Management automatically checks for changes in the shared area at a fixed interval of five minutes, which is customizable. You can manually check for updates too.

If changes are made to pages in a design by other users, Allegro Design Management indicates to you that modifications have been made to the design. You can then update your copy of the project so that the changes by other team members are reflected in your own copy of the project.

Allegro X EDM Design Management User Guide

Introducing Allegro Design Management

In the case of hierarchical designs, designs (schematic and physical) are created concurrently, limited only by the reuse, and team dynamics.

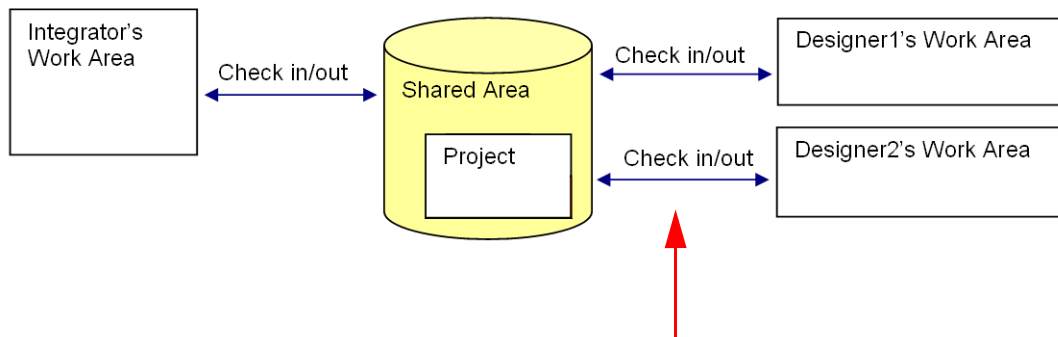
All data management tasks, such as check in, check out, refresh, undo check out, and update are fully supported in Allegro Design Management and are easily done from the Allegro Design Management interface.

6. After a design is completed, designers check in the design objects that they have been assigned and have modified.
7. After the design objects have been worked on and checked back into the shared area, the integrator updates the entire design to generate a physical netlist.

The integrator can always view the status of all the design project components in the Allegro Design Management user interface that acts as a dashboard for design activities.

After the system is integrated, the design is ready for packaging, layout, and manufacturing.

The following chart illustrates the design flow:



In the case of *flat designs*, pages or constraints are checked out and in.

For *hierarchical designs*, schematic, blocks and subdesigns are checked out and in.

Setting up Allegro Design Management

To work with Allegro Design Management, you first need to set it up. Typically, this is done by site administrators.

After a site administrator has set up Allegro Design Management, the integrator needs to prepare the design that you want to enable for design management.

Depending on your role, click on the following links to read the section that is relevant to you:

- Site Administrator: [Setting up Allegro Design Management](#)
- Integrator: [Preparing Designs for Design Management](#)

Setting up Allegro Design Management

When you first start working with Allegro Design Management, set `repository=ADWFileSysDAO` in `settings.ini` which is available at `<Cadence_installation_directory>\share\cdssetup\sdm\config`.

After you define the repository settings in `settings.ini`, do the following tasks:

- [Accessing the User List](#)
- [Defining Integrator Roles](#)
- [Updating Libraries and Designs](#)
- [Setting up a Project Shared Area](#)
- [Working with Generic Views](#)
 - [Defining Global Views](#)
 - [Defining Design-Specific Generic Views](#)
 - [Excluding the Generic View](#)

Accessing the User List

You need to enable Allegro Design Management to access the list of available users using one of the following methods:

- [LDAP Configuration](#)
- [User Assignment Template Configuration](#)

LDAP Configuration

Allegro Design Management employs the LDAP configuration as the default method to access the user list that Allegro Design Management uses for team members.

To enable Allegro Design Management access the list of users available in the LDAP server, you need to update the `ldap.config` file. Copy the `ldap.config` file from the Cadence installation directory

(`<Cadence_installation_hierarchy>\share\cdssetup\sdm\config`) to the following:

```
<CDS_SITE>\cdssetup\sdm\config\ldap.config
```

Now set the following details:

Note: You might require assistance from the IT department to get the required details:

- **Server:** This specifies the IP address of the LDAP server.
- **Port:** In most cases, it should remain the default, unless it has been changed in your organization.
- **Query:** This specifies the tree from where the LDAP data should be queried.
- **Filters:** It is important to filter the data returned by the LDAP query.

Depending on the operating system, such as RHEL or Ubuntu, you will need to modify the LDAP library path accordingly. For example, for Allegro Design Management to work on a Linux SUSE 11 SP2 (64-bit) machine, you might need to do one of the following:

- ❑ Create a link for `libldap-2.4.so.2` to `libldap.so` in the library area of the machine (`/usr/lib`). You might need to contact the administrator for this task.
- ❑ Specify the LDAP library path in the `ldap.config` file at the `$CDS_SITE` level as follows:

```
[server]
```

```
library=/usr/lib/libldap-2.4.so.2
```

User Assignment Template Configuration

Another method to access the user list is by using the `teamassignmenttemplate.xml` template file. Do the following to use this configuration method:

1. Copy the template file from

`<SPB_installation_hierarchy>\share\cdssetup\sdm\permissions to`

any of the following locations depending on whether you are working with Allegro Data Manager or Team Design, or both.

- ❑ Allegro(R) Design Authoring Team Design Option:
`<CDS_SITE>\cdssetup\sdm\permissions`

Note: You will need to create a folder named `sdm` and `permissions` within the `cdssetup` folder.

2. Customize this template file to specify the integrators and the design team members.
3. Set the following variable:

`LDAP_LOCAL_LOC = default` if you want to access the template file at the site as mentioned in step 1.

or

`LDAP_LOCAL_LOC = <team assignment template file path>`

For example: `LDAP_LOCAL_LOC=c:\team.xml`

Defining Integrator Roles

With Team Design and Allegro Data Manager, an integrator enables design management for a project and is automatically added to the project team.

LDIs and PDIs are assigned at the project level using the Project Management dialog box.

With the Team Design and Allegro Data Manager, to enable design management, you need to have your name in one or both of the following sections of the `ldap.config` file:

- `[logical_integrators]`
- `[physical_integrators]`

Allegro X EDM Design Management User Guide

Introducing Allegro Design Management

The following table explains these roles in detail.

Role	Configuration	Function
Logical Design Integrators	If your name appears in the [logical_integrators] section, you are an LDI.	<ul style="list-style-type: none">■ If you are an LDI or logical designer, then the default hierarchy will be <i>Logical Hierarchy</i>.■ The LDI has complete permission on logical objects as defined in the policy file.■ Only the LDI can assign team members for logical objects.■ The LDI can assign team members at the block level if there are no physical objects under that block.
Physical Design Integrators (PDI)	If your name appears in the [physical_integrators] section, you are a PDI.	<ul style="list-style-type: none">■ If you are a PDI or a physical designer, then, by default, you will see only the <i>Physical Hierarchy</i>.■ The PDI has complete permission on the physical objects defined in the policy file.■ Only the PDI can assign team members for physical objects.

Allegro X EDM Design Management User Guide

Introducing Allegro Design Management

Role	Configuration	Function
Both LDI and PDI	If your name is defined in both the sections, then you are both, an LDI, and PDI.	<ul style="list-style-type: none">■ If you are both, an LDI and PDI, or a logical and physical designer, then the default hierarchy as displayed in the dashboard will be <i>Working Design</i>.■ If you are both an LDI and PDI, you can assign team members for physical and logical objects.■ Only when you are both, an LDI and PDI, can you assign team members at the block level.■ If you are both an LDI and PDI, you have permission on all the global objects.

Updating Libraries and Designs

- A design (both, front-end and back-end) should be uprev'd to the current release before it is used for design management purposes.

For detailed information, refer to *Allegro Design Entry HDL User Guide*.

- If `_N` is used for differential pairs, set `ConceptSetup_Assertion_Read = *` in `CDS_SITE` to ensure `_N` is not treated as a low assertion character in the flow.

Setting up a Project Shared Area

The integrator needs to specify a location for sharing the design and assign the required read and write permissions for it. This location is called the Shared Area and is used to store the design and to be accessed by all the designers on the team.

Important

In addition, once a project has been enabled for design management, you must not modify the shared area manually.

Working with Generic Views

This section explains the design methodology for working with generic views.

A generic view is a user-defined folder at the project CPM level or within the 5x project structure. It maps to a design object that is used in the PCB design process, and can be defined in the following two ways:

- **Global View:** A global view is specified within the project, at the same level as the project CPM file.
- **Design-Specific View:** A design-specific view is specified in the project 5x structure and follows the library->cell->view architecture for storing data.

Team leads do the following:

1. Create the design project in their work-area, and then create the initial design hierarchy.
2. Identify any user-specific global generic objects and design-specific generic objects required in the design.

Important

To add these generic objects, you need to configure the out-of-the-box policy file. You can modify the policy file before or after enabling a project for design management. However, if you are making changes after a project has been enabled for design management, you need to explicitly check out the `project_settings` global object. It is therefore recommended that you define the generic views before enabling a project for design management.

Defining Global Views

The team lead identifies the global generic objects for a design project, for example, global constraint objects, physical and spacing classes, technology files, and ECsets for a design.

To specify a user-specific global generic object, you need to add its corresponding definition in the out-of-the-box policy file, which is located at:

`<SPB_installation_directory>\share\cdssetup\sdm\policies`

To customize this file, copy the file from the installation directory to the site area:

- If you are working with Team Design:
`<CDS_SITE>\share\cdssetup\sdm\policies`
- If you are working with Allegro Data Manager:
`<Allegro_EDM_conf_root>\<company>\<site>\cdssetup\sdm\policies`

Example

The out-of-the-box policy file allows you to manage a folder for global ECSets. The following figure shows how this global view is defined in the policy file.

```
<designObject label="ECSets" name="ecsets" type="ADW_ECSETS">
  <attachment>
    <includes>
      <include path="{projdir}/ecsets/*"/>
    </includes>
  </attachment>
  <tools>
    <tool args="" label="Edit ECSets" name="consmgr"/>
  </tools>
</designObject>
```

where:

- `label="ECSets"`: This indicates the entry you see in the *Type* column of the Allegro Design Management dashboard.
- `name="ecsets"`: This indicates the name of the new global generic object that you will see in the Allegro Design Management dashboard under the *Globals* section. Ensure that the value for *name* matches the folder name of the global generic object on the disk.
- `type="ADW_ECSETS"`: Each object has a type and for a global generic object, the type is user defined. If you want to define multiple global generic objects, ensure that the type for each object is unique.
- `includes` section: A design object is associated with some files. The `includes` section specifies all the data that will be included in the global generic object. The files defined in this section will form part of the object zip file.
- `tools` section: This section defines the context menu option you see when you right-click `ecsets` in the Allegro Design Management dashboard. In addition, it specifies the tool that will be used to launch files in the `ecsets` folder on the disk.

Note: When you launch Constraint Manager for a design from Design Entry HDL to make any changes, the design database is also modified.

Therefore, after enabling a project for design management, before you make any change using Constraint Manager, including creating new Electrical Constraint Sets (ECSets), you need to check out the schematic view.

Defining Design-Specific Generic Views

By default, Allegro Design Management manages the `sch_1`, `sym_1`, `packaged`, and `physical` views. You can also create design-specific generic views. To define a new design-specific generic view, you need to modify the policy file.

Example

To add a design-specific generic view that manages all the documents in a subdesign, you need to add its corresponding definition as a child of the subdesign in the policy file. In addition, you need to exclude this object from the `sch_1` object definition.

```
<designObject category="Logical" label="Document" name="doc_1" type="ADWGenericType">
  <attachment>
    <includes>
      <include path="${projdir}/${design_lib}/${design_name}/doc_1/*" />
    </includes>
  </attachment>
  <tools>
    <tool args="-proj,${projdir}/${cpm}" label="Open" name="wordpad" />
  </tools>
</designObject>
```

where:

- `category="Logical"`: This specifies the hierarchy that will be used to display a design-specific generic object. If you define the category as `physical`, the object will be displayed in the physical hierarchy.
- `label="Documents"`: This indicates the entry you see in the *Type* column of the Allegro Design Management dashboard.
- `name="doc_1"`: This indicates the name of new design-specific generic objects that you will see in the Allegro Design Management dashboard under the specified hierarchy. Ensure that the value for *name* matches the folder name of the design-specific generic object on the disk.
- `type="ADWGenericType"`: Each object has a type; the type for design-specific generic objects is `ADWGenericType`.
- `includes` section: A design object is associated with some files. The `includes` section specifies all the data that will be included within the design-specific generic object. The path of the folder is defined using the following parameters:
 - `projdir`: Project directory path

Allegro X EDM Design Management User Guide

Introducing Allegro Design Management

- ❑ `design_lib`: Name of library containing the blocks
- ❑ `design_name`: Name of the design or subdesign
- **tools** section: This section defines the context menu option you see when you right-click `doc_1`. In addition, it also specifies the tool that will be used to launch files in the `doc_1` folder on the disk.

The data for the `doc_1` view may not exist at the time of defining it in the policy file. Thus, after you make changes to the policy file, add the `doc_1` directory on the file system. This allows the team lead to assign team members to this view.

Excluding the Generic View

The default policy file has been set up in such a way that all views other than `packaged` and `physical` are included in the `sch_1` view for the design. The new, design-specific generic view is added and managed as an independent design object. Therefore, these views need to be excluded from the `sch_1` view. The following figure shows the `doc_1` view in the `excludes` section for `sch_1`.

```
<designObject category="Logical" label="Schematic" name="sch_1" type="ADWSchematic">
  <attachment>
    <includes>
      <include path="${projdir}/${design_lib}/${design_name}/*" />
    </includes>
    <excludes>
      <exclude path="${projdir}/${design_lib}/${design_name}/metadata/*" />
      <exclude path="${projdir}/${design_lib}/${design_name}/${packager_view}/*" />
      <exclude path="${projdir}/${design_lib}/${design_name}/${physical_view}/*" />
      <exclude path="${projdir}/${design_lib}/${design_name}/sym_*/" />
      <exclude path="${projdir}/${design_lib}/${design_name}/entity" />
      <exclude path="${projdir}/${design_lib}/${design_name}/${sch_view}/*.lck" />
      <exclude path="${projdir}/${design_lib}/${design_name}/${sch_view}/*.*,*" />
      <exclude path="${projdir}/${design_lib}/${design_name}/derived_data" />
      <exclude path="${projdir}/${design_lib}/${design_name}/doc_1" />
    </excludes>
  </attachment>
</designObject>
```

Preparing Designs for Design Management

After a site administrator has set up Allegro Design Management, you, as the integrator, need to prepare the design that you want to enable for design management.

To prepare a design, the following tasks are required:

- Physical Team Design
 - File-Level Management
 - Design Partition
 - Physical and Packaged Objects
- Working with Variants

Physical Team Design

This section covers the following topics:

- File-Level Management
- Design Partition

File-Level Management

There are times when multiple users work on different files under the physical view. In such cases, Allegro Design Management allows multiple users to check out the physical view. This option is available when you are enabling a project for design management in the Team Assignment page.

After enabling a project for design management, the option to allow multiple users to check out the physical view is available in the Project Management dialog box. When you select the *Manage files under physical view as separate objects* option, file-level management of the design is enabled. Files under the physical view can be managed as separate objects and multiple users can check out the physical view.

For example, a physical object may have multiple board files, which need to be managed individually. A PDI or a user who owns the physical view can select the *Manage files under physical view as separate objects* check box in the Team Assignment page when enabling a project for design management. Selecting this option will enable you to assign the board files to individual users, check in, and check out them to modify them as individual objects.

It is recommended that you leave the default option as is. Be aware however that the selection cannot be reversed.

However, if you do not select this option when enabling a project for design management, you have the option to select it later. To learn how to use this option, see [Enabling a Project for Design Management](#).

Design Partition

This section describes how Allegro Design Management allows you to manage design partitions in the physical object.

Assigning Partitions

As a PDI, you need to assign partitions to design team members. To assign partitions, do the following:

1. Open a project that has enabled for design management in Allegro Design Management.
2. Enable design management, if not done already.

The dashboard shows the following:

- ☐ the physical object containing the various partitions.
- ☐ users as assigned to the partitions in Workflow Manager in Allegro PCB Designer.

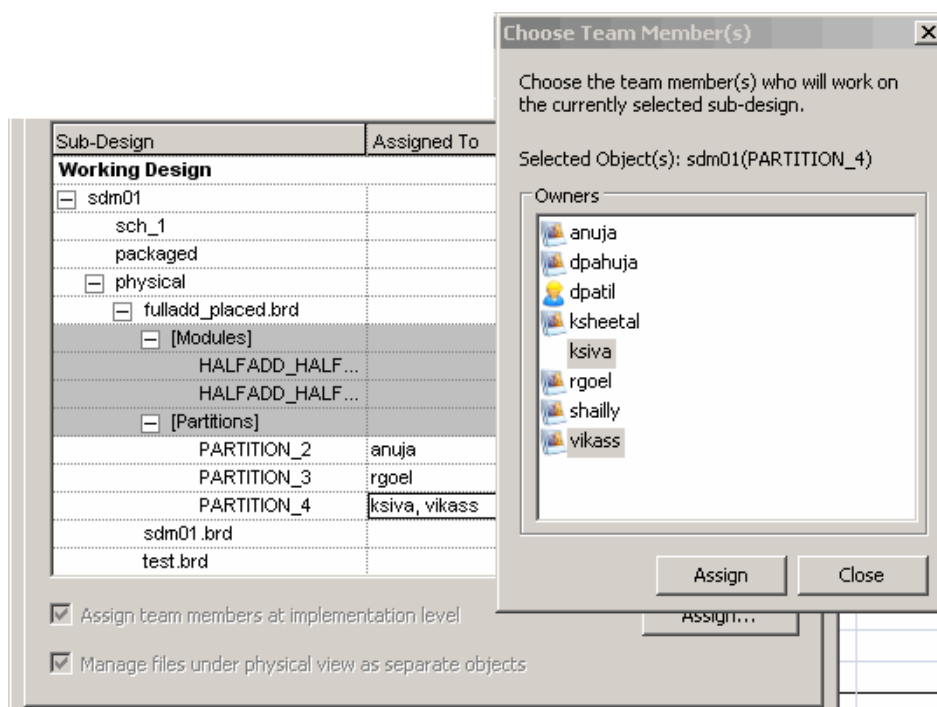
Note: There is no synchronization between users assigned in Workflow Manager and Allegro Design Management.

3. Choose *Design Management — Project Management - Team Assignment*. This is optional and is done in case you need to assign any other team member to a partition or

Allegro X EDM Design Management User Guide

Introducing Allegro Design Management

to add more team members to a partition. However, team assignment done in Allegro Design Management will not be reflected in Workflow Manager.



Importing Partitions

After a designer has made changes, you as a PDI need to import and integrate the partitions. Do the following to import and integrate partitions:

1. Synchronize all the partitions with the latest changes as made by the physical designers.
2. Check out the physical view of the design.
3. Open Workflow Manager in Allegro PCB Designer to import these partitions.
4. Save the master board file.
5. Check in the physical view.

Designer Tasks

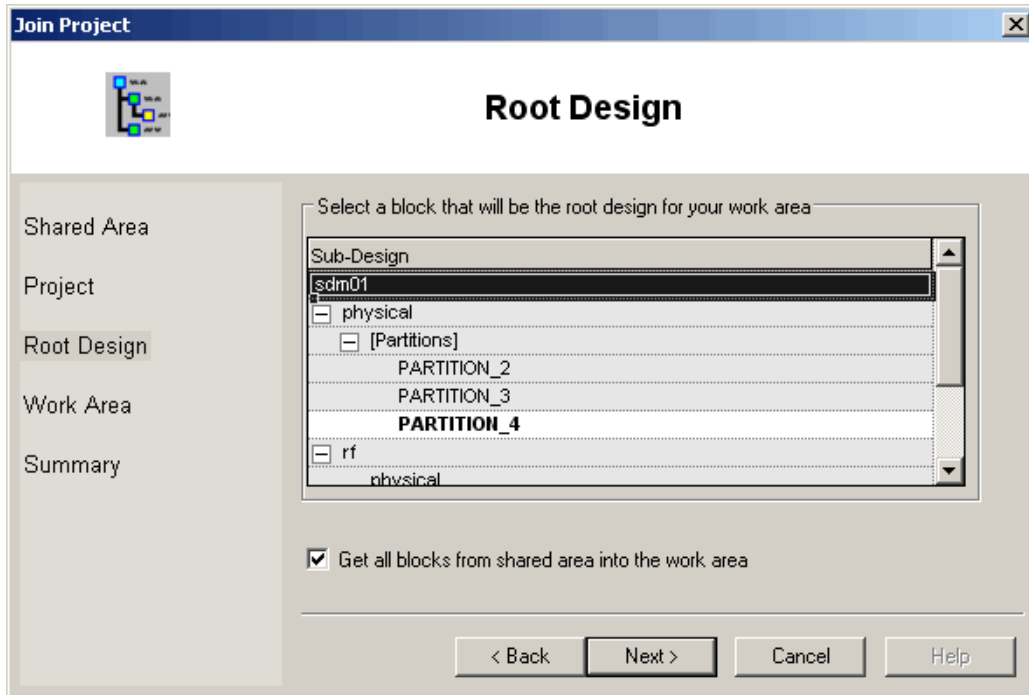
Steps to be performed by a designer who is assigned to a particular partition:

1. Join the project that has been enabled for design management in Allegro Design Management.

Allegro X EDM Design Management User Guide

Introducing Allegro Design Management

In our example, the Root Design page of the Join Project wizard shows that you have access only to PARTITION_4.



After you join the project, the dashboard appears.

2. Check out the partition assigned to you.
3. Open the partition in Allegro PCB Designer to work on it.
4. Save the changes made to the partition.

You can see the modified local version.

5. Check in the modified partition.

Physical and Packaged Objects

By default, Allegro Design Management identifies changes only in the root-level packaged and physical objects. If any lower-level block contains packaged and physical objects, by default, Allegro Design Management does not identify changes in such blocks. If you are working on a lower-level block, it is recommended that you set the block as the root block.

However, if you are working on a design and need Allegro Design Management to also identify packaged and physical changes in the lower-level blocks, you need to set

Allegro X EDM Design Management User Guide

Introducing Allegro Design Management

`monitor_package_physical_allblocks` to `TRUE` in the SDM section of the project CPM file.

Note: If this directive is set, Allegro Design Management performance will be affected during loading and when switching to Allegro Design Management after the save operation in other applications.

Working with Variants

By default, Allegro Design Management manages the variant view as a single object. When checking in multiple variant files, Allegro Design Management zips up the files as a single object and checks it into the shared area.

Allegro Design Management also supports different variant files as different objects. This allows different designers in a design team to work on different variants of a project in parallel.

To enable file-level variant support, that is, to manage multiple variant files as separate objects in Allegro Design Management, the integrator needs to modify the `<project cpm>` file by doing the following:

1. Open the `<project.cpm>` file in a text editor.
2. Locate the `START_SDM` section.
3. Add the following entry:

```
manage_view_files 'variant'

START_SDM
manage_file_as_objects 'TRUE'
manage_view_files 'variant'
team_design_project 'TRUE'
system_cpm_dir './.sdm/shadow_area'
END_SDM
```

4. Save and close the `.cpm` file.

Allegro X EDM Design Management User Guide

Introducing Allegro Design Management

All the variant databases will now be displayed as separate objects. The integrator can assign them to different users. Designers can now check out a variant database, open it in Variant Editor, modify and check the variants back in.

[-] vardsn1	2.0	2.0			
sch_1	1.0	1.0			
[-] variant	1.0	1.0			variant files created
kista.dat	1.0	1.0		abhay	
shanghai.dat	1.0	1.0		aileen	
montreal.dat	1.0	1.0		akryukov	
packaged	1.0	1.0			
[-] physical	1.0	1.0			
vardsn1.brd	1.0	1.0			
[-] mid1	1.0	1.0			
sch_1	1.0	1.0			
sym_1	1.0	1.0			

Important

If file-level variant support is required in all the projects, add `manage_view_files` 'variant' to the `<site cpm>` file as described in [step 1](#).

Enabling Projects for Design Management

This chapter covers the following topics:

- [Overview](#)
- [Launching Allegro Design Management and Loading a Project](#)
- [Enabling a Project for Design Management](#)
- [Defining Users as LDI or PDI](#)

Overview

To enable design management, you need to do the following:

1. Launch Allegro Design Management.
2. Load the design project.
3. Set up the project shared area.
4. Select the Enable Design Management option from Allegro Design Management.
5. Perform the steps in the Enable Design Management wizard.

Launching Allegro Design Management and Loading a Project

You can launch Allegro Design Management in one of the following three ways:

- From any non-Allegro EDM design project already open in Design Entry HDL, choose *Design Management — Enable Design Management*.
- From any non-Allegro EDM design project already open in Allegro Project Manager, choose *Tools — Team Design*.
- For an Allegro EDM project, launch Allegro Design Management from Allegro EDM Flow Manager.
 - a. Open any Allegro EDM design project.
 - b. Click *Allegro Design Management* in the Common Tools pane.

Depending on whether you work with Allegro Data Manager or Team Design, or both, and your repository settings, the following appears:

Table 2-1

Enabling a Project for Design Management

The integrator enables design management for a project and is automatically added to the project team.

Table 2-2 Integrator Roles to Enable Design Management

Allegro Design Management	Permission for ETD
Team Design	LDI or PDI or users with both permissions can enable design management.
Allegro Data Manager	Only an ECAD integrator, who is both an LDI and PDI, can enable design management.


Do the following to enable design management:

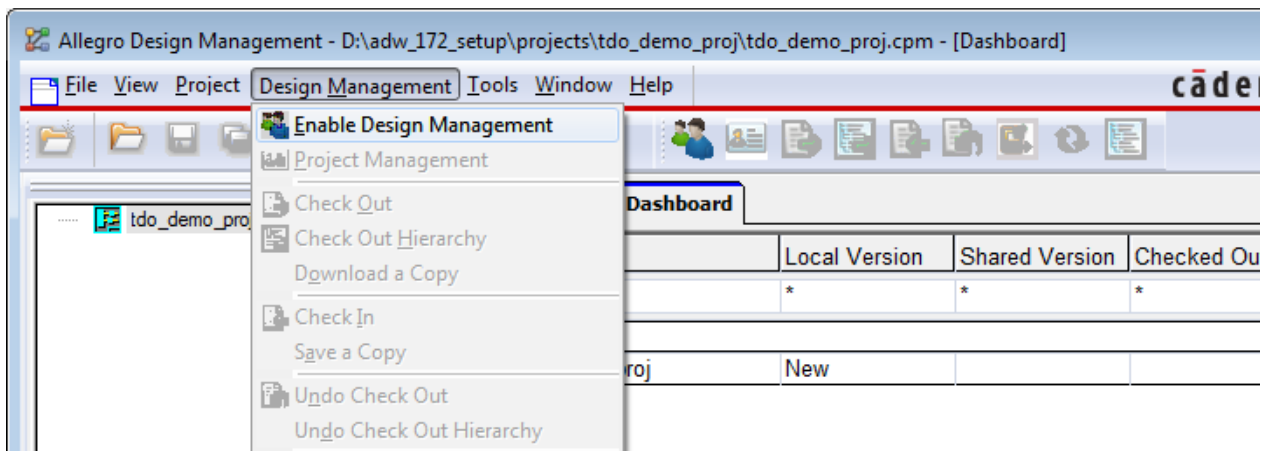
1. Launch Allegro Design Management and load the required design by following the steps in the Launching Allegro Design Management and Loading a Project section.

Allegro X EDM Design Management User Guide

Enabling Projects for Design Management

Note: If you want to enable design management for a site, workspace, or page, you must ensure that you select the same name as that specified during site/workspace/project creation.

2. Ensure that you have created the project shared area.
3. Do one of the following:
 - ❑ Choose *Design Management— Enable Design Management*.
 - ❑ Click the *Enable Design Management* icon () in the toolbar.



The Enable Design Management wizard starts. The *Shared Area* form appears.

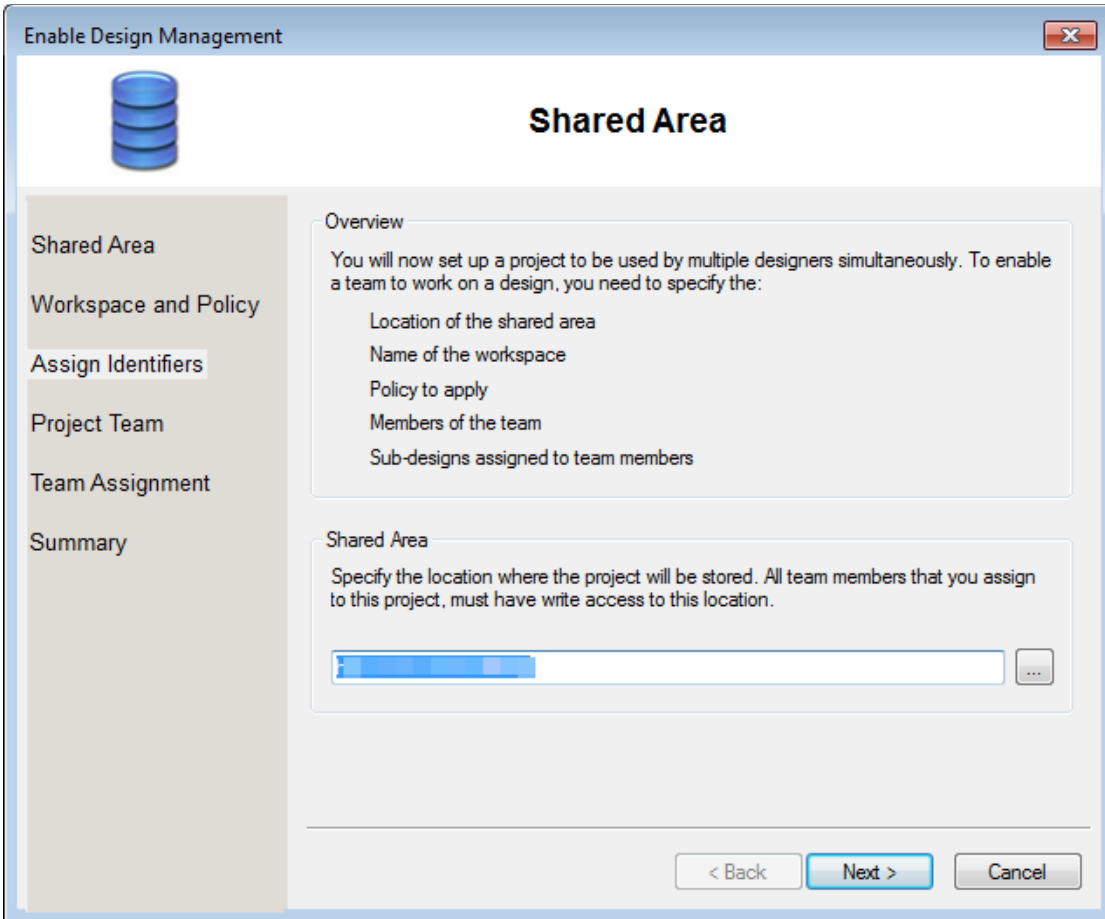
Shared Area Page

You need to provide a location where all the project files will be available. By default, you specify a shared area for each project that you enable for design management. You can also set a default shared area by using the `default_shared_area_path` entry in the `site.cpm` file.

Once this is set, the Shared Area field value is filled in automatically. You can change it if needed.

Allegro X EDM Design Management User Guide

Enabling Projects for Design Management



The screenshot shows a software window titled "Enable Design Management" with a close button in the top right corner. Inside the window, there is a blue database icon and a title "Shared Area". On the left is a sidebar with a list of options: "Shared Area", "Workspace and Policy", "Assign Identifiers", "Project Team", "Team Assignment", and "Summary". The "Shared Area" option is selected. The main area is divided into two sections. The top section, titled "Overview", contains the text: "You will now set up a project to be used by multiple designers simultaneously. To enable a team to work on a design, you need to specify the:" followed by a list of items: "Location of the shared area", "Name of the workspace", "Policy to apply", "Members of the team", and "Sub-designs assigned to team members". The bottom section, titled "Shared Area", contains the text: "Specify the location where the project will be stored. All team members that you assign to this project, must have write access to this location." Below this text is a text input field with a blue selection bar and a browse button (three dots) to its right. At the bottom of the window are three buttons: "< Back", "Next >", and "Cancel".

4. Click *Next*.

The *Workspace and Policy* form appears.

Enable Design Management

Workspace and Policy

Shared Area

Workspace and Policy

Assign Identifiers

Project Team

Team Assignment

Summary

Workspace

Shared area contains workspace(s), each of which can have one or more projects. Specify where to store the current project under the shared area.

demo_pro

Policy

Policy specifies how sub-design(s) and their child object(s) get managed and stored for this project. Additionally, policy can define the commands and rules to be run for operations, such as check in or check out.

HighSpeed_System

< Back Next > Cancel

Workspace is a container for storing projects in the shared area. It can contain more than one project. It acts as a system which is made up of multiple designs and boards.

By default, the workspace for the design project will be the same as the project name. You can choose the default name, select an existing workspace, or specify a new one. You can include only the following characters while specifying the workspace name:

abcdefghijklmnopqrstuvwxyz_0123456789

Note: By default, the *Policy* drop-down lists all entries in alphabetical order. If you want to specify a custom default policy file and lock it from the site-level setup, you can copy the policy file from the installation to the site. The site takes precedence over the local installation. If a policy file exists at the site, Allegro EDM does not read the installation policy file. If you copy the required policy file to the site, only that file will be listed in the *Policy* drop-down list.

5. Click *Next*.

The Project Team page appears if you are working with Team Design.

If you are working with Allegro Data Manager, the *Assign Identifiers* page appears, after which the Project Team page appears.

Assign Identifiers Page

Project Numbers

Assign Identifiers

Specify a number and display name for each of the following design objects. The server uses these numbers as unique identifiers for the design objects.

	Number	Description
Project	1	
Logical	1	
Physical	1	

< Back Next > Cancel

6. You can specify the values for the *Project*, *Logical*, and *Physical* numbers. This is an optional step.

The following table explains the fields in the Assign Identifiers form.

Table 2-3 Fields in Assign Identifiers Form

Field	Function
Project	This number indicates the project that you want to enable.

Allegro X EDM Design Management User Guide

Enabling Projects for Design Management

Field	Function
Logical	This number is the same as the Schematic Number field in the Allegro EDM Project Wizard that you specified while creating the design project.
Physical	This number indicates the object number for the layout or board for a project.

7. Click *Next*.

The *Project Team* page appears.

Project Team Page

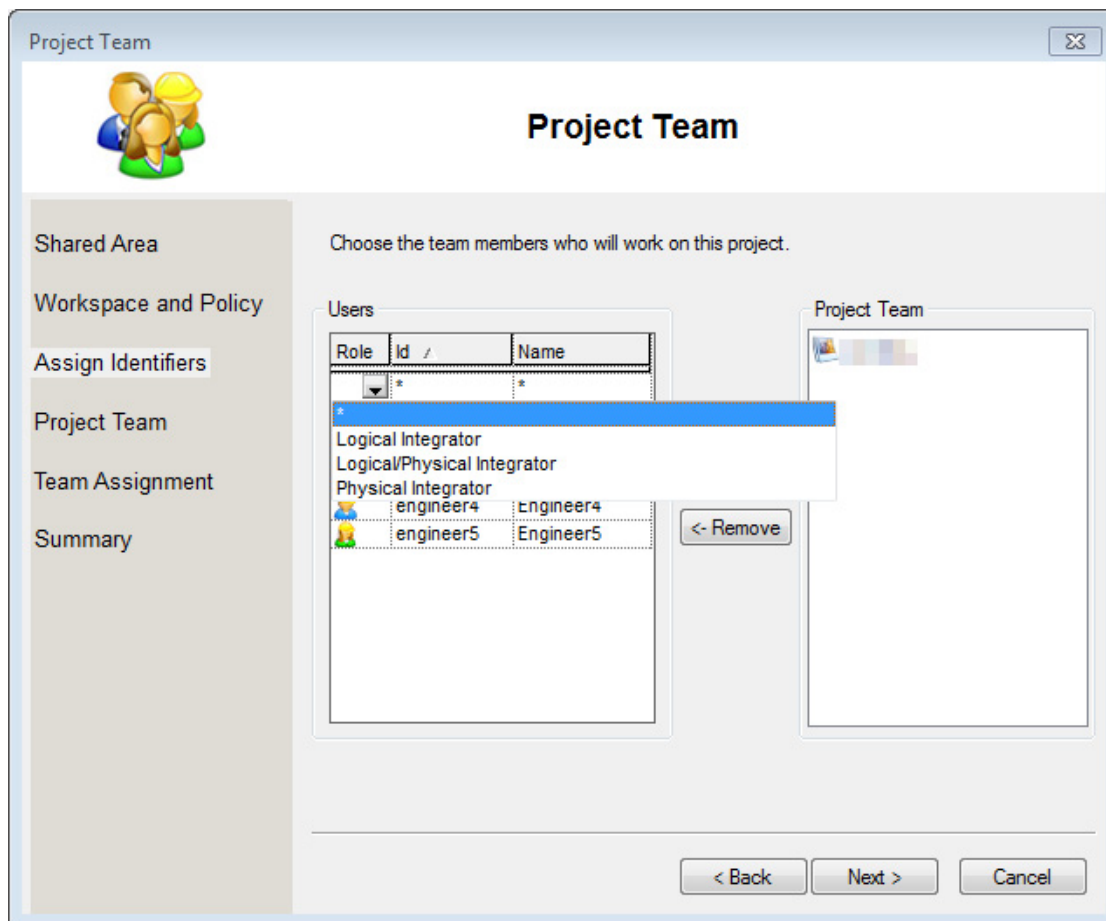
The *Project Team* page allows you to assign users to the design team.

Table 2-4

Allegro X EDM Design Management User Guide

Enabling Projects for Design Management

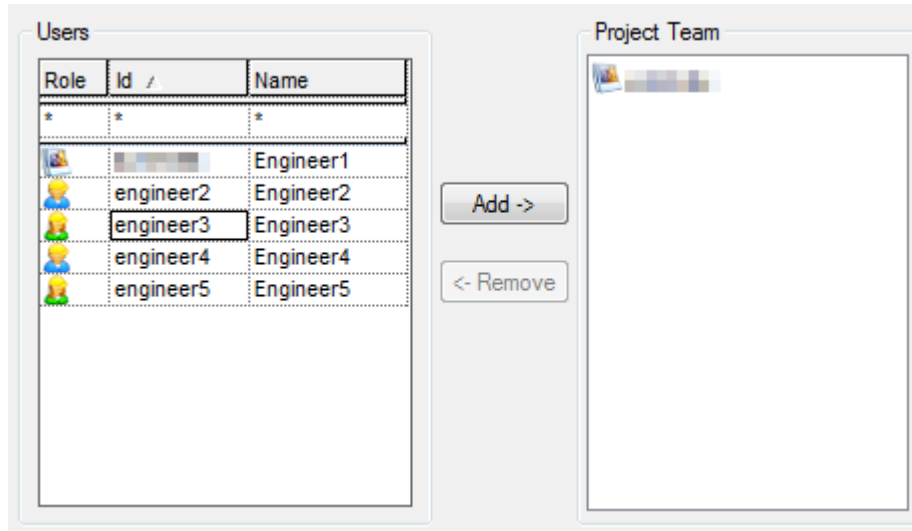
The *Users* list box contains three columns: *Role*, *Id*, and *Name*. You can also search for these using dynamic filtering below the column header.



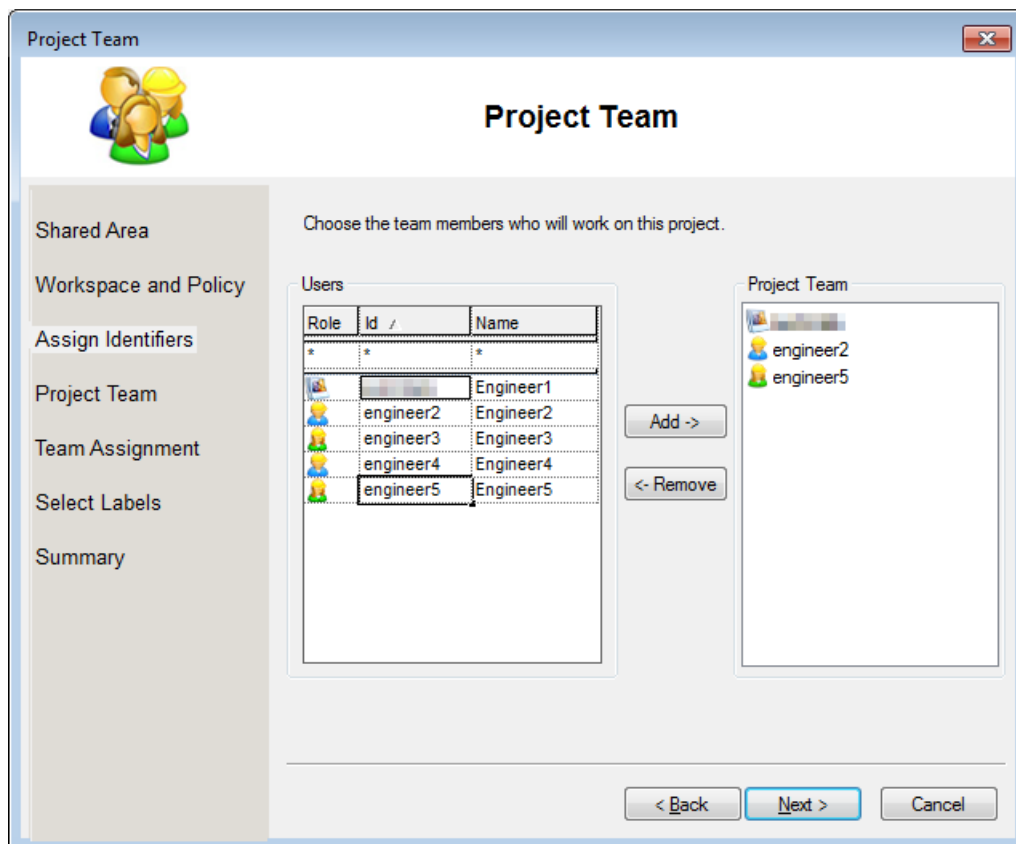
Allegro X EDM Design Management User Guide

Enabling Projects for Design Management

Note that each category of integrator is represented by a different icon.



8. Choose design team members from the *Users* list box and click *Add*.



Allegro X EDM Design Management User Guide

Enabling Projects for Design Management

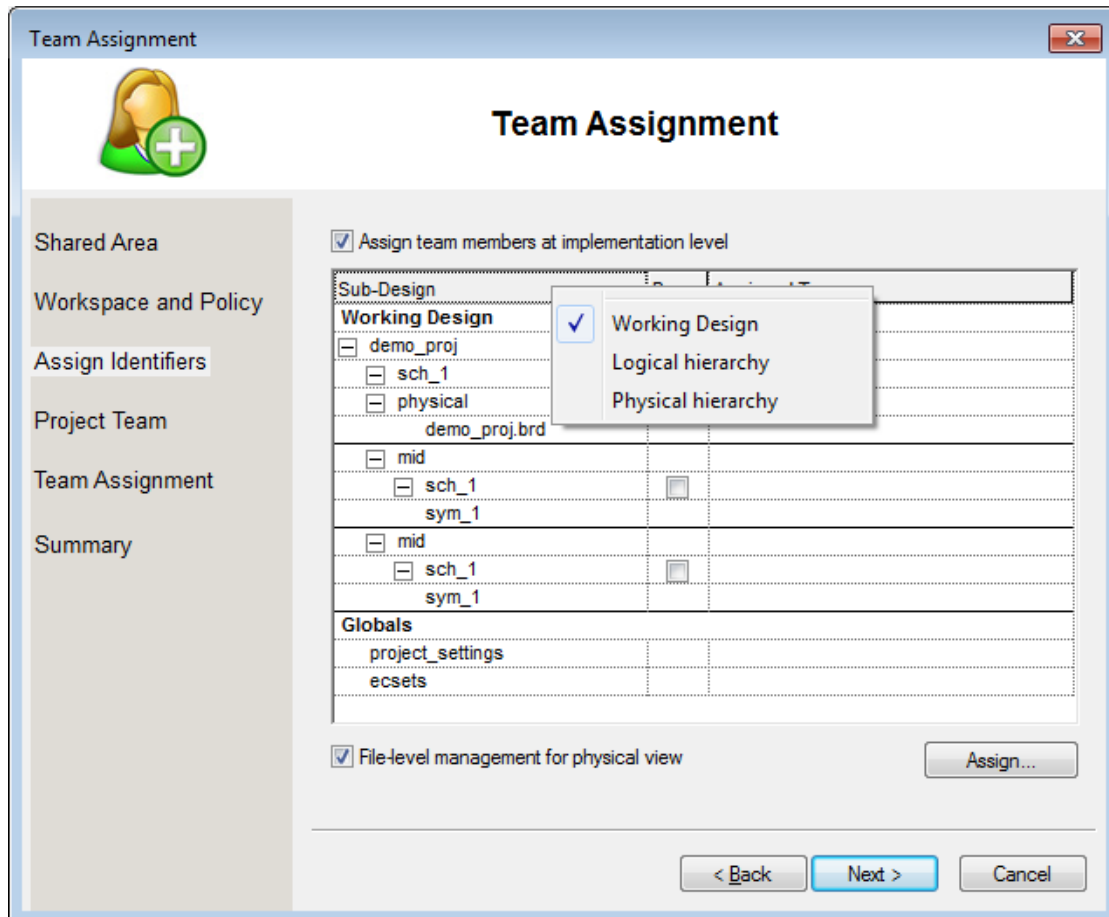
9. Change the role of the users in the *Project Team* list box, if required. For details on how to change the user role, see [Defining Users as LDI or PDI](#).

10. Click *Next*.

- ☐ If you are working with Allegro Data Manager, the Summary page appears. To read the instructions related to the Summary page, [click here](#).
- ☐ If you are working with Team Design, the Team Assignment page appears.

The Team Assignment page is used to assign project team members to a subdesign, page, or constraint.

The type of hierarchy displayed will depend on the role assigned to you. You can right-click the header and switch to any other hierarchy.



11. The *Assign team members at implementation level* option is used to assign team members at the object level.

If you clear this check box, you will see the view level of implementation.

Sub-Design	Pages	Assigned To
Working Design		
[-] demo_proj		
[-] mid		
[-] mid		
Globals		
project_settings		
ecsets		

- 12.** Using the *File-level management for physical view* option, Allegro Design Management enables file-level management and allows you to manage the files in the physical view as separate objects. This option is selected by default. It is recommended that you leave the default option as is. Be aware however that the selection cannot be reversed.

However, if you clear this check box, you can select this option even after enabling a project for team design by doing the following:

- Choose *Enable Design Management — Team Assignment*.
- In the *Team Assignment page*, select the *File-level management for physical view* check box.
- Click *OK*.

The files under the physical view are now checked in. To be able to assign any team member to these files, you need to choose *Enable Design Management — Team Assignment* again and click *OK*.

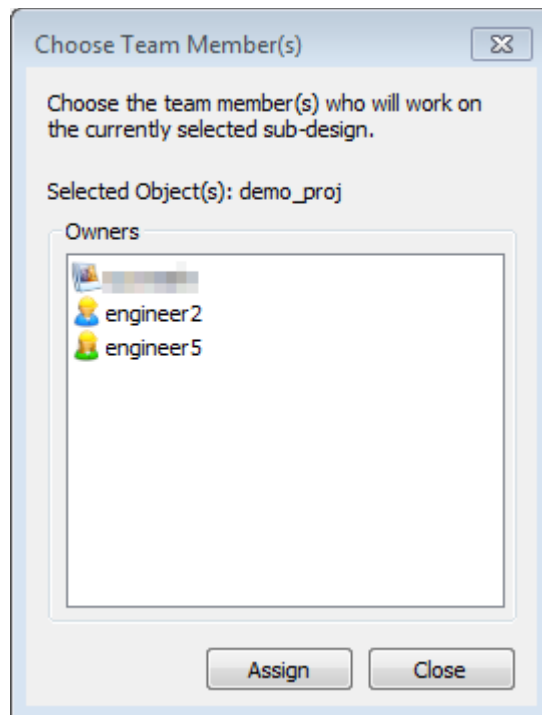
Important

If there are existing partitions or object-level assignments, both these options are selected and disabled, and you will see the object view of the hierarchy only.

13. Assign a user for a subdesign by doing the following:

- a.** Click the subdesign to be assigned.
- b.** Click *Assign*.

The Choose Team Member(s) window appears.

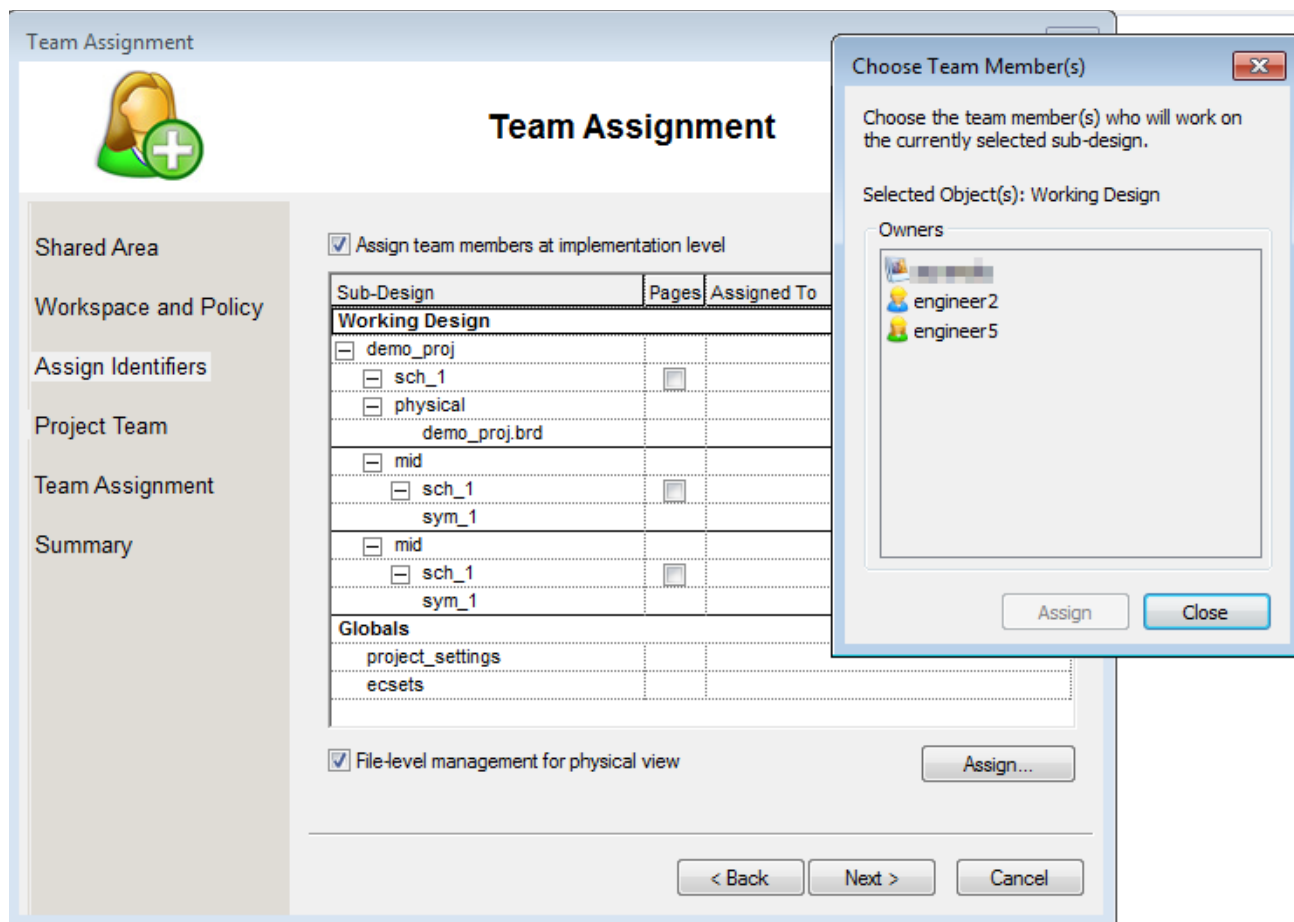


- c.** Choose a team member. You can assign more than one team member for a subdesign.
- d.** Click *Assign* in the Choose Team Member(s) dialog box.

Allegro X EDM Design Management User Guide

Enabling Projects for Design Management

The team member is now the owner of the subdesign. To assign more than one designer to a block, keep the CTRL key pressed, click the names of the designers who will own the block, and then click *Assign*.

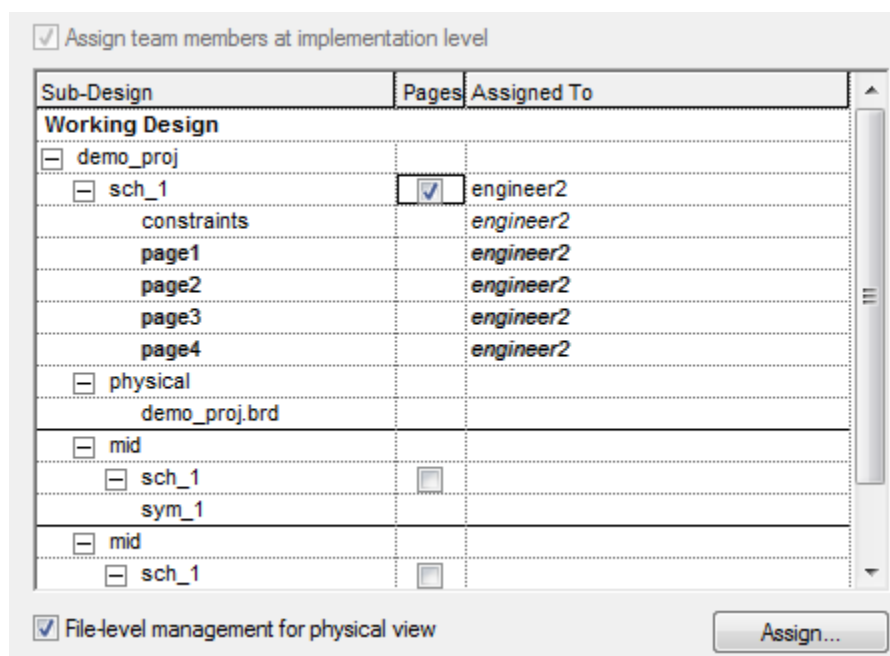


- e. Assign team members to all other subdesigns and global objects, if required.
- f. Click *Close* in the Choose Team Member(s) window.

When assigning a subdesign to a team member, you can choose to enable page-level management for flat designs. Page-level management allows you greater control over version management and allows users to work in their own workspaces.

Enabling Page-level Management

To enable page-level management, select the *Pages* checkbox next to the subdesign that you are assigning to a team member. You can choose to assign different pages in a schematic to different users in the project team.



Removing User Assignments

To unassign a user from a subdesign or page, do the following:

- a. Click the subdesign or page from which you need to remove user assignment.
- b. Click *Assign*.

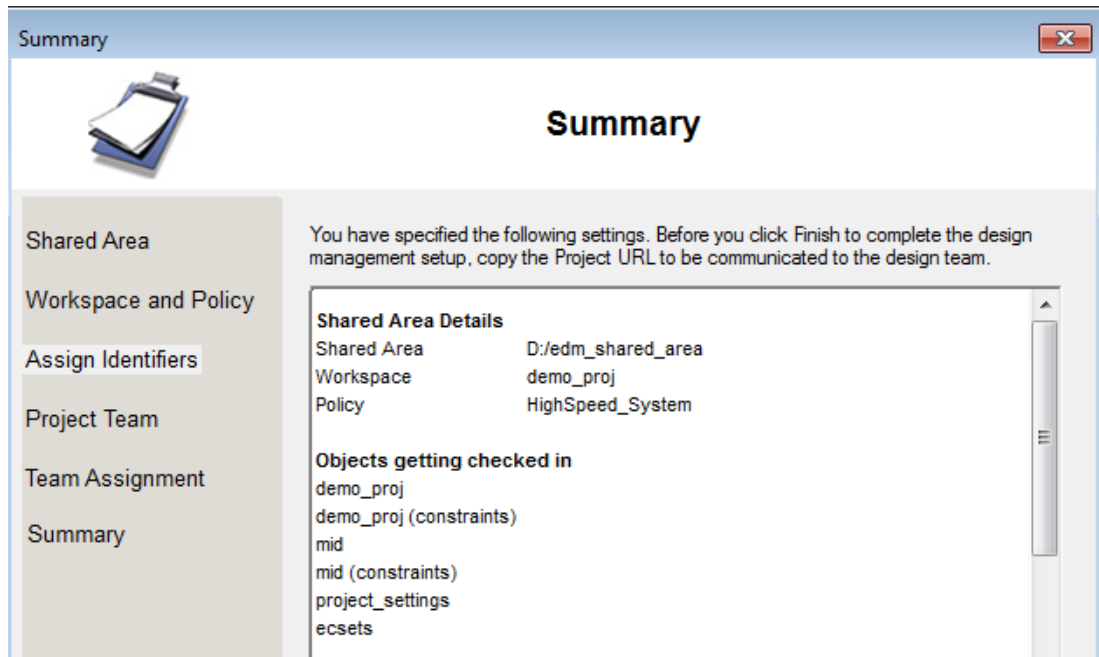
The Choose Team Member(s) window appears with the assigned users selected.

- c. Click in a blank area in the Choose Team Member(s) window then click *Assign* in the Choose Team Member(s) dialog box.

Note: If you assigned more than one team member to a subdesign or page, click on the member that you want to retain and click *Assign*. The other team members are unassigned.

14. Click *Next*.

The Summary page appears.



Summary Page

15. In the Summary page, do the following:

- Verify the objects assignments in the *Summary* page.
- Copy *Project URL* to be able to send this to the designers who will work on this project.

16. Choose *Finish*.

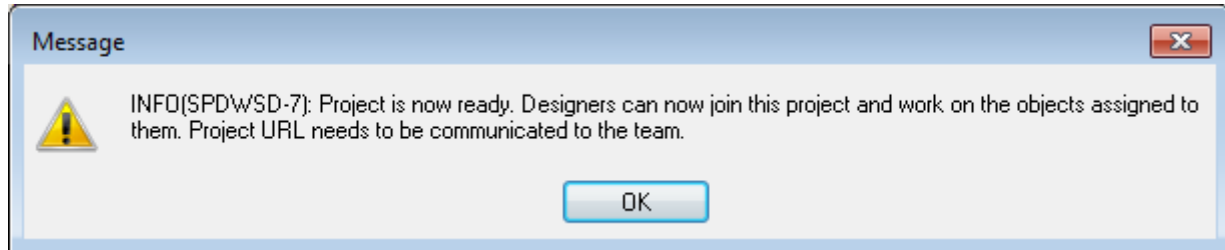
When you enable design management for a project, the following happens:

- ☐ Subdesigns and pages are assigned to team members and can be used by designers by joining the project.
- ☐ The selected design project is now in the shared area.

Allegro X EDM Design Management User Guide

Enabling Projects for Design Management

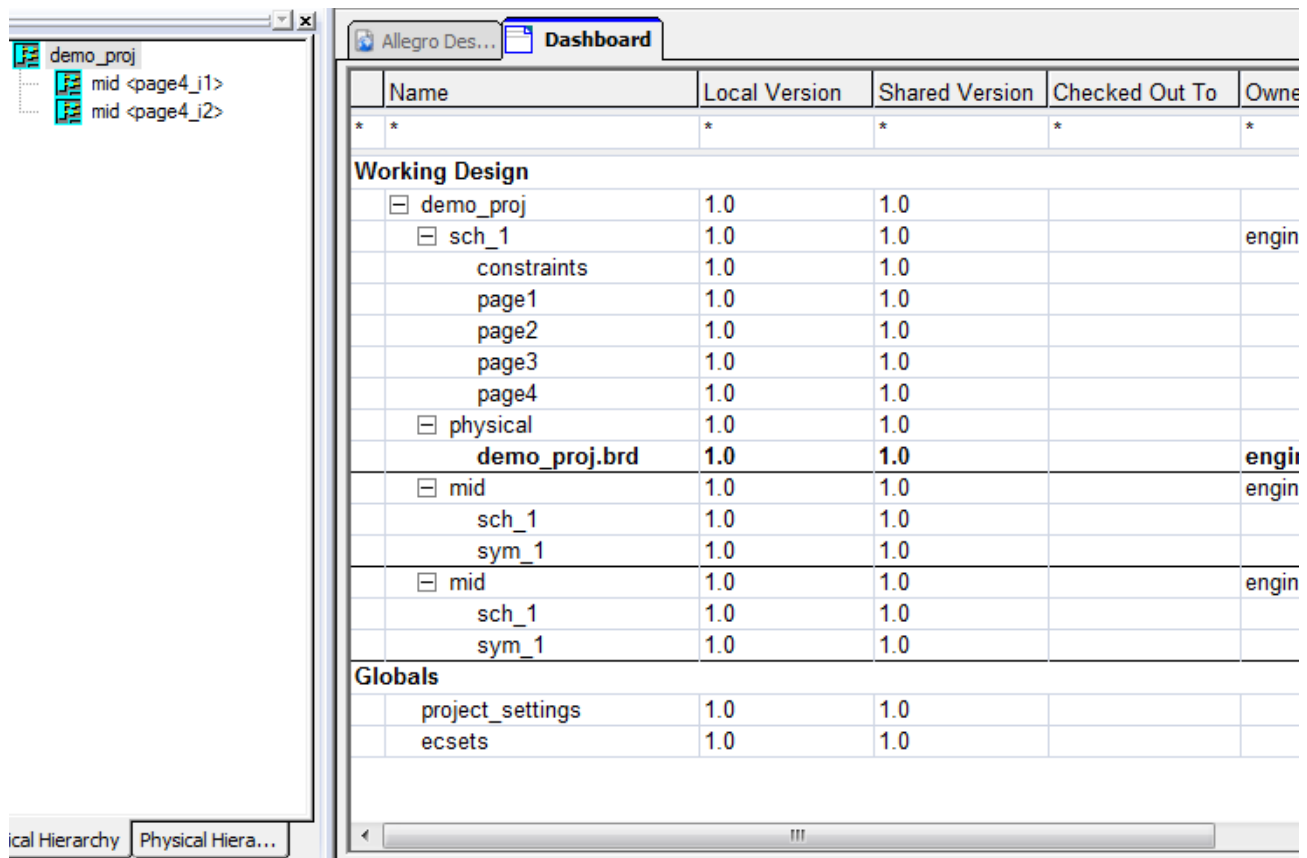
A message appears to indicate the next steps to be performed by the integrator.



17. Click *OK*.

The Enable Design Management wizard closes.

The dashboard now shows the local and shared versions, and the assigned owners for the blocks in the design.



Name	Local Version	Shared Version	Checked Out To	Owner
Working Design				
demo_proj	1.0	1.0		
sch_1	1.0	1.0		engin
constraints	1.0	1.0		
page1	1.0	1.0		
page2	1.0	1.0		
page3	1.0	1.0		
page4	1.0	1.0		
physical	1.0	1.0		
demo_proj.brd	1.0	1.0		engin
mid	1.0	1.0		engin
sch_1	1.0	1.0		
sym_1	1.0	1.0		
mid	1.0	1.0		engin
sch_1	1.0	1.0		
sym_1	1.0	1.0		
Globals				
project_settings	1.0	1.0		
ecsets	1.0	1.0		

After you set up the shared area and enable design management, being the integrator, you need to communicate the following to team members:

- Location of the project in the shared area as a project URL
- Blocks assigned to each team member

Defining Users as LDI or PDI

To be able to define a role for a project team member in Allegro Design Management, you **must** be a Logical Design Integrator (LDI) and a Physical Design Integrator (PDI). Project team members can be defined as a Logical Design Integrator, Physical Design Integrator, or both.

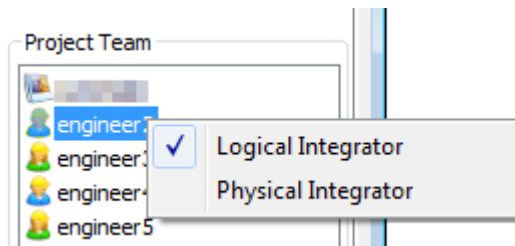
The two integrator roles are responsible for the following:

- Logical design integrator - front-end design
- Physical design integrator - back-end design

You can define these roles for users during the process of enabling a project for design management, or after you enable the project.

If you want to define the role of a user while enabling a design project for design management, do the following:

1. In the Project Team page, when assigning a team member to a design object, right-click the user whose role you want to define.



2. Select the required role.

If you want to define the role of a user after enabling a design project for design management, do the following:

1. Launch Allegro Design Management.
2. Choose *Design Management — Project Management*.

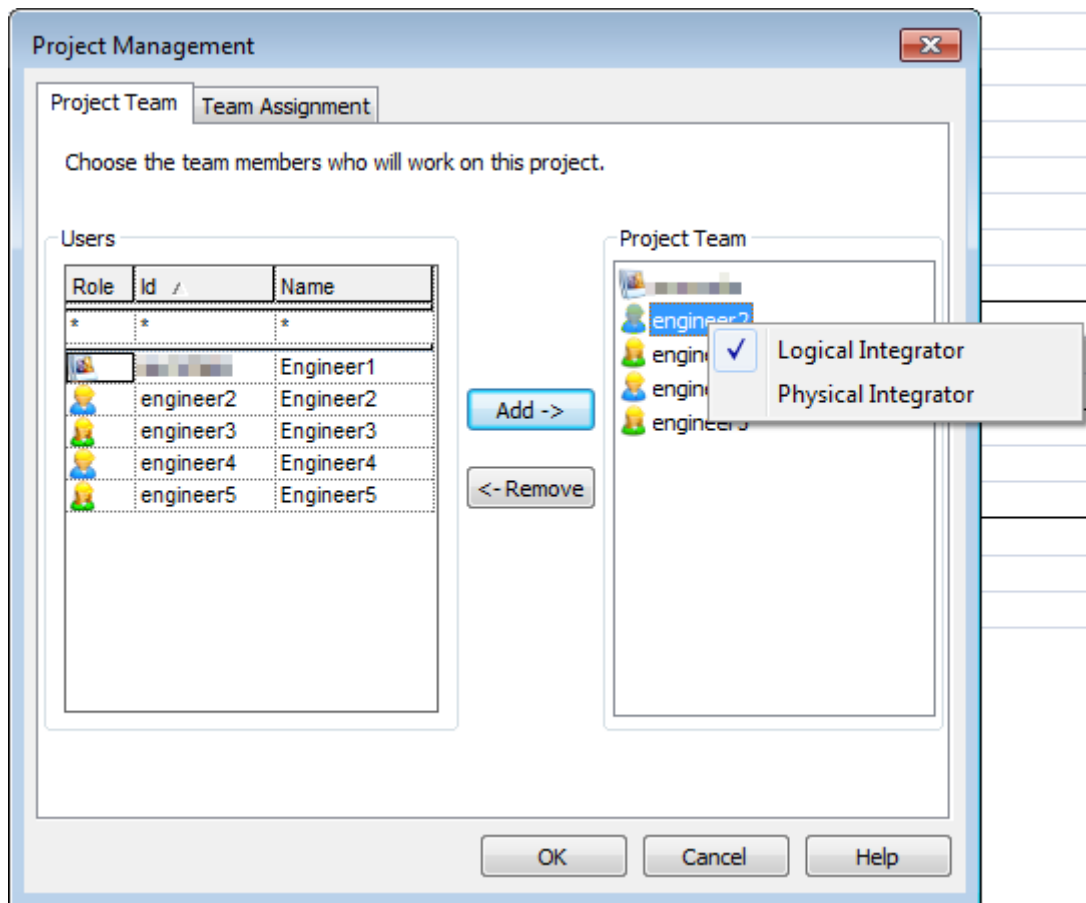
The Project Management dialog appears.

3. Click *Project Team*.

Allegro X EDM Design Management User Guide

Enabling Projects for Design Management

4. In the Project Team section, right-click the user whose role you want to define.



5. Select the required role and click *OK*.

Joining a Design Project

After the integrator has set up the shared area and assigned ownership rights for subdesigns, designers access the project and start work on the subdesigns they own.

To start work on a design project, the following sequence of tasks is performed:

- Each designer joins the project.
- Allegro Design Management creates a work area for each design team member.
Each designer works on a local copy of the project to ensure a stable view of the design. This local copy of the project is called the work area.
- The designer checks out the required design, subdesign, block, or page and begins modifying it.

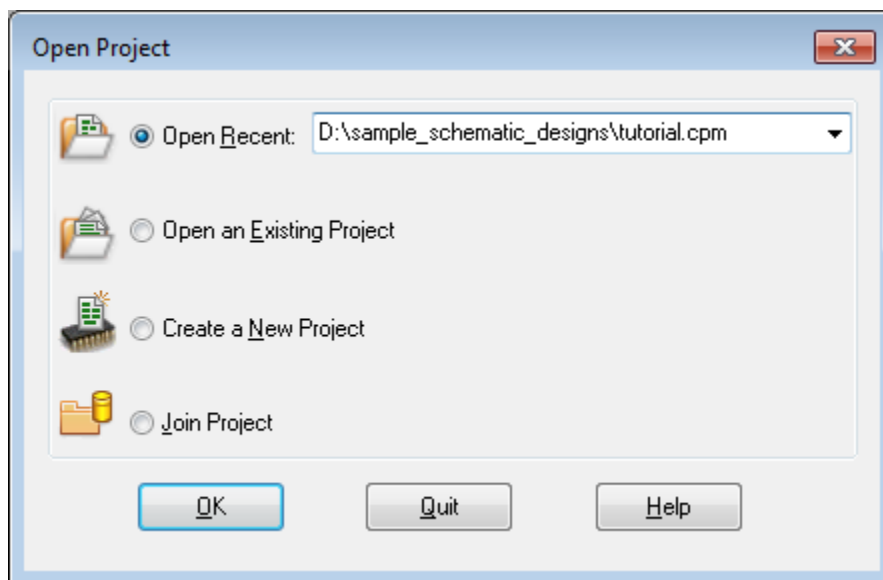
When you check out a design, subdesign, or page, an icon appears next to the design name on the dashboard. This enables all team members to monitor the status of the designs, subdesigns, and pages in a project. When you modify a subdesign or page and check it in, the version number is incremented. You can also add comments when checking in the subdesigns or pages.

Joining a Project

To join a project, do the following:

1. Launch Allegro Design Management in one of the following ways:

- ❑ If you are working with a non-Allegro EDM project, choose *File — Join Project* from Project Manager or select the *Join Project* radio button from the Design Entry HDL welcome page.



The Allegro Design Management window appears.

- ❑ If you are working with an Allegro EDM project, open Allegro EDM Flow Manager.
Click *Join Project* from the Allegro EDM Welcome Page or choose *File — Join* from a project tab.

The Join Project wizard starts. The *Shared Area* form appears.

Allegro X EDM Design Management User Guide

Joining a Design Project

The Join Project wizard starts. The *Shared Area* form appears.

The screenshot shows a window titled "Join Project" with a close button in the top right corner. Inside the window, there is a blue database icon on the left. The main area is titled "Shared Area". On the left side of the main area, there is a vertical list of steps: "Shared Area", "Project", "Root Design", "Work Area", and "Summary". The "Shared Area" step is currently selected. The main content area is divided into two sections. The top section is titled "Overview" and contains the text: "To join a project and work as part of a team, specify the:" followed by a list: "Shared area", "Workspace", and "Project details (name, root design, type and location)". The bottom section is titled "Shared Area" and contains the text: "Specify the shared area where the design managed project is stored. To directly navigate to the project, use the Project URL provided by the integrator". Below this text is a text input field with a blue highlight and a small button with three dots. At the bottom of the window, there are three buttons: "< Back", "Next >", and "Cancel".

2. In the Shared Area page, specify the Project URL to where the project shared area is located as communicated by the integrator.

3. Click *Next*.

In the Project page, the *Project* form displays the workspaces and projects available to you depending on the Project URL specified in the *Shared Area* section.

4. Choose the project to join.

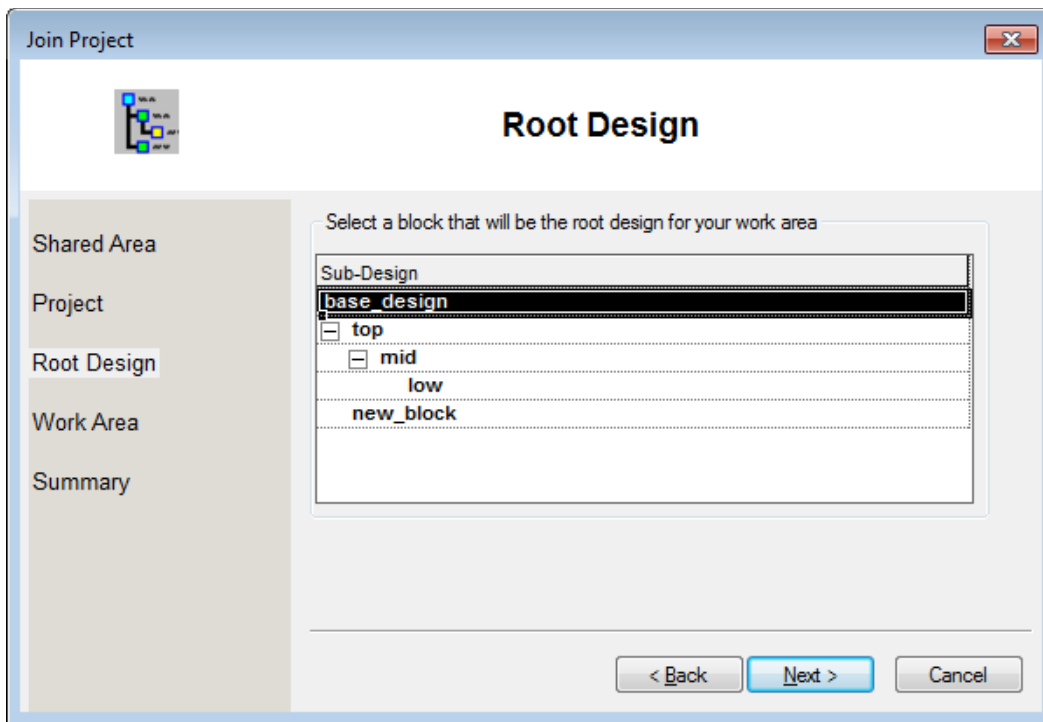
5. Click *Next*.

The *Root Design* form displays the subdesigns assigned to this user. Subdesigns that are not available to the current user have a shaded background.

Allegro X EDM Design Management User Guide

Joining a Design Project

6. If you want to choose a different block as the root design, select that subdesign. You have an option to get only the selected subdesigns and their children. However, it is recommended that you get all the subdesigns.



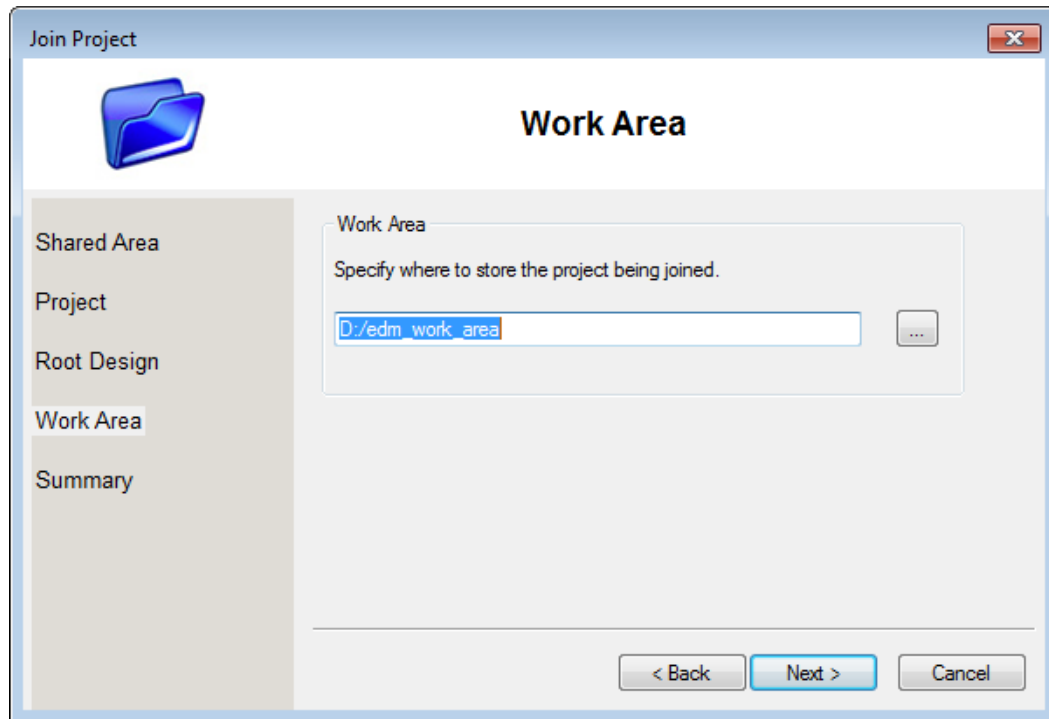
Entries in bold for object names indicate that you have the required write permission.

7. Click *Next*.

Allegro X EDM Design Management User Guide

Joining a Design Project

The *Work Area* form appears.



Join Project

Work Area

Shared Area

Project

Root Design

Work Area

Summary

Work Area

Specify where to store the project being joined.

D:/edm_work_area

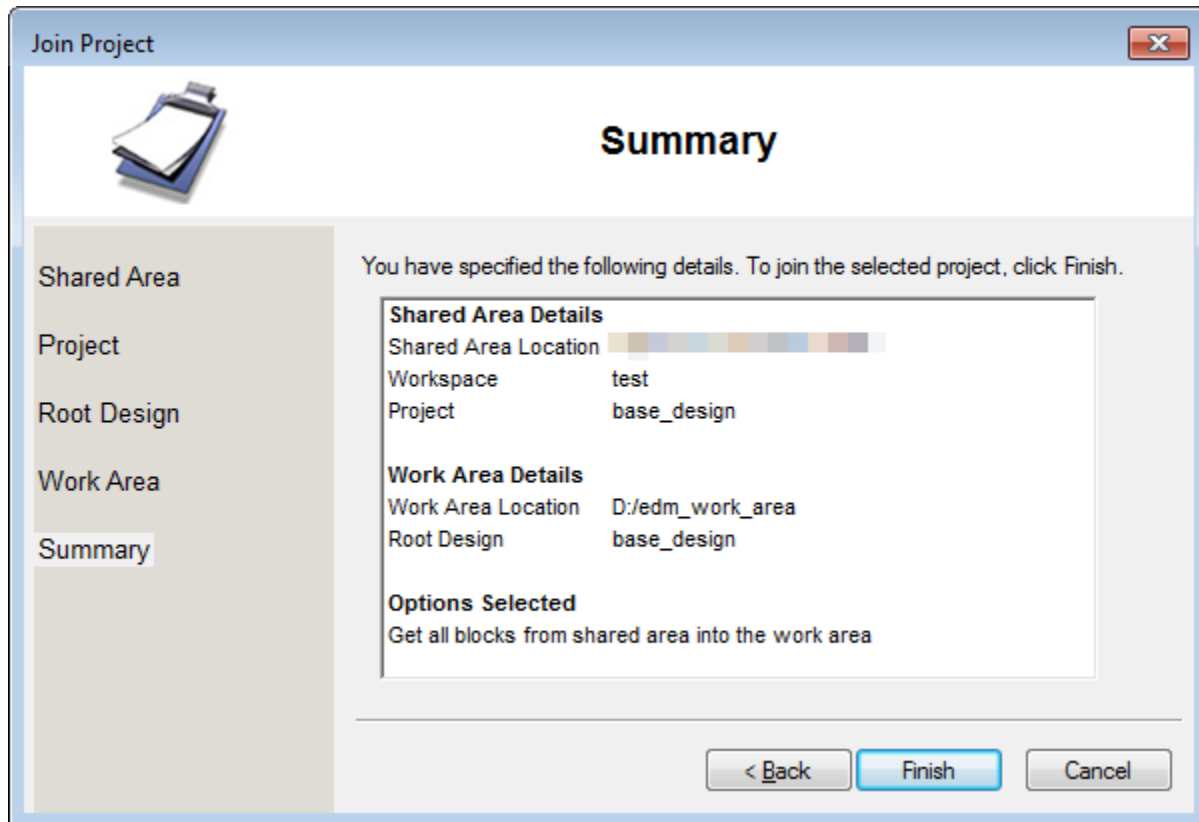
< Back

Next >

Cancel

8. Specify the work area folder. This is the location where the project will be created locally.
9. Click *Next*.

The *Summary* page appears.



10. Click *Finish*.

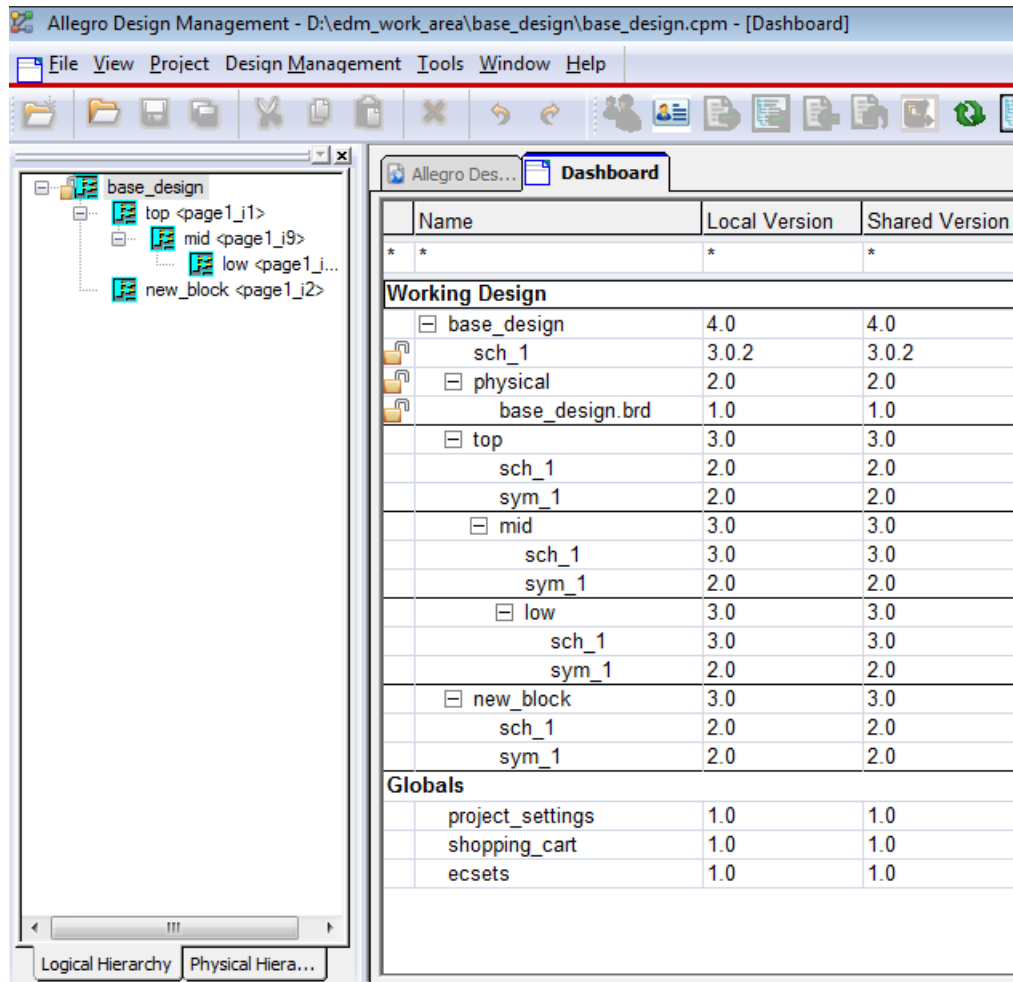
The local work area is created. A message appears to indicate that you have joined the project and prompts you to make the joined project the active project.

11. Choose your option.

Allegro X EDM Design Management User Guide

Joining a Design Project

The dashboard is updated and the subdesigns are now available in the local work area.



Note: In the Allegro EDM Flow Manager, there will now be three tabs: the Welcome page, the project that you opened from the Flow Manager, and the project that you have just joined.

Allegro X EDM Design Management User Guide

Joining a Design Project

Working with Designs in Allegro Design Management

The tasks you perform when working with designs in Allegro Design Management depend on your role, such as integrator, or designer. To navigate quickly through the following sections, click on the link that is relevant to your role:

- [Integrator Tasks - Defining Project Management](#)

- ☐ [Working with Flat Designs](#)
 - [Assigning Team Members for Flat Designs](#)
- ☐ [Working with Hierarchical Designs](#)

- [Integrator and Designer Tasks](#)

Some operations are common for both, flat, and hierarchical designs. They are as follows:

- ☐ [Checking Out a Design, Subdesign, or Page](#)
- ☐ [Modifying a Design](#)
 - [Adding New Blocks](#)
- ☐ [Rolling Back Changes or Undoing Check Out](#)
- ☐ [Checking In a Design, Subdesign, or Page](#)
- ☐ [Changing the Root Design](#)
- ☐ [Synchronizing with Shared Area](#)
- ☐ [Releasing Check Out](#)
- ☐ [Resolving Unauthorized Design Modifications](#)
- ☐ [Deleting Objects from the Allegro Design Management Dashboard](#)

- [Working with Flat Designs in Design Entry HDL](#)

■ Creating Initial Design Hierarchy: Recommendations

Integrator Tasks - Defining Project Management

Using the page-level management feature for a design, you can work with distributed flat designs. The primary difference in defining project management for flat, and hierarchical, designs is in team assignment.

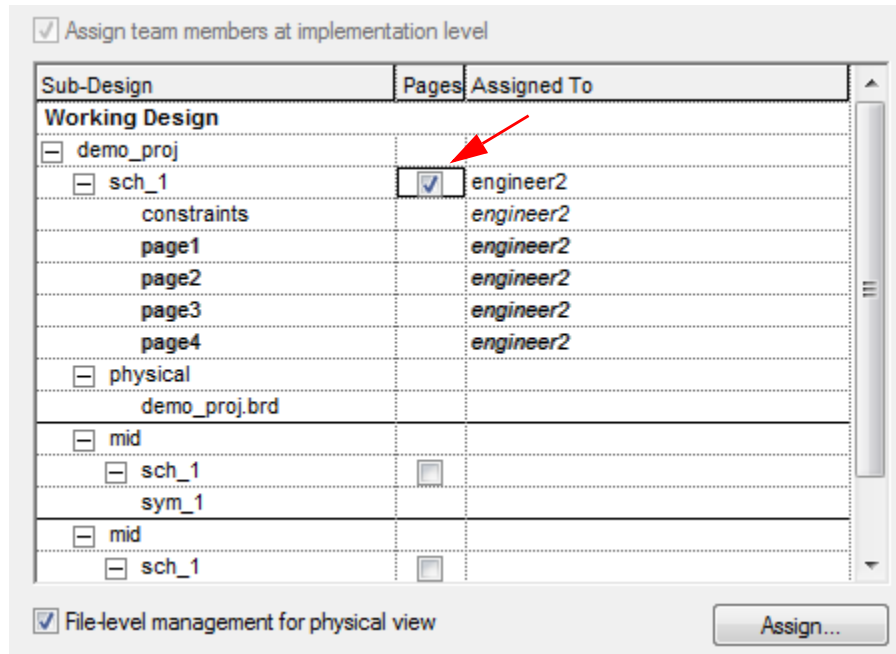
Working with Flat Designs

When you define project management for a flat design, you can choose to enable page-level management when assigning a subdesign to a team member. Page-level management allows you greater control over version management and allows users to work in their own workspaces.

Assigning Team Members for Flat Designs

To enable page-level management, select the *Pages* checkbox next to the subdesign that you are assigning to a team member. You can choose to assign different pages in a schematic to different users in the project team.

Note: A schematic can be assigned to a user and its corresponding constraint view can be assigned to another user.



Note: Page-level management is not supported in cache-enabled projects. As a result, the *Pages* checkbox will not be visible when working with cache-enabled projects.

Working with Hierarchical Designs

When you define project management for a hierarchical design, instead of assigning pages and constraints to team members, only schematic, blocks and subdesigns can be assigned.

Creating Initial Design Hierarchy: Recommendations

To gain maximum benefit from Allegro Design Management, approach a design at a conceptual level as a collection of blocks and interconnects. This enables the following:

- Main system design
- Interface definition
- Block interconnections
- Multi-level subdesign hierarchy

■ Creation of blocks and ports

When preparing a design hierarchy, do the following:

1. Create a design project
2. Create an initial design hierarchy
3. Define global power signals for the design
4. Import reuse blocks from a library/design
5. Assign subdesign_suffix to reuse blocks
6. Define packaging settings for reuse blocks
7. Validate IBIS and DML models
8. Update DML Device Index from DML Models
9. Set up the search path for IBIS models
10. Set up CM Units and Precision
11. Set up and import global ECsets

Before creating the project, the team should also review the following:

- Design hierarchy
- Reuse blocks
- Global ECset names
- Naming convention for physical and spacing classes
- Define the board units and precision across team members

Integrator and Designer Tasks

The Allegro Design Management user interface offers tooltips and icons to help you perform data management tasks. This section provides detailed steps for routine design tasks.

Note: Design management tasks can be performed from within Design Entry HDL as well. For details, refer to [Design Management Menu in Design Entry HDL](#).

Checking Out a Design, Subdesign, or Page

To check out a design, subdesign, or page, do the following from the dashboard or hierarchy tree in the left pane:

1. Right-click the design, subdesign, or page to be checked out.

There are two checkout options:

- ☐ Check Out
- ☐ Check Out Hierarchy - only available for a design or subdesign

Depending on your requirements and rights, you can check out the design, the design along with all its subdesigns, or a page.

Check Out can apply at any level in the design, whether it is the root, subdesign, or page. and Check Out Hierarchy is only applicable at the design level or subdesign level and includes checking out all the child objects of the selected design.

Important

If you intend to auto-generate models for discrete devices in the schematic, you need to check out the schematic *and* physical views. This is to ensure that `devices.dml` is writable.

The `devices.dml` file is the default working signal integrity (SI) device model library for a project. SI models for two-pin discrete devices are, by default, stored in this file, unless you have specified another file as the working SI model library for the project.

2. Choose *Check Out*.

The Checkout window appears.

You can choose which designs to check out, or click *OK* to check out all the subdesigns.

3. Click *OK*.

The dashboard is updated.

- ☐ An unlocked icon is displayed for checked out designs.
- ☐ The designer's ID is displayed in the *Checked Out To* column.

In addition, icons also appear in the hierarchy tree. This shows that the designer has checked out these objects and can modify them.

Modifying a Design

To modify a design, do the following:

1. Check out a design.
2. Double-click a design node in the hierarchy tree in the left pane, or in the dashboard, right-click a design object and choose *Open*.

Note: This step is not required if you are modifying a design from within Design Entry HDL.

Allegro Design Entry HDL or Allegro PCB Editor launches depending on the object you are working on.

3. Modify the design.
4. Save the design. This marks the object as modified in Allegro Design Management.

You can use Allegro Design Management and Design Entry HDL simultaneously. This is because all changes are local and in the current work area. To push these changes into the project's shared area, designers need to check in designs.

Note: If you make changes in a sub-block that are related to interface signals, ensure that you modify the parent block accordingly. Allegro Design Management does not capture such changes.

Allegro Design Management reflects the current state of the design hierarchy at all times. When blocks are added or deleted, these changes are updated in the hierarchy tree in the left pane and the dashboard.

Note: In Allegro EDM 16.6 QIR 4 onwards, if you import a block from a team-design enabled project into a non-Allegro EDM project, the view version number may not be the same as that displayed in the Allegro Design Management dashboard.

Save a Copy as Draft

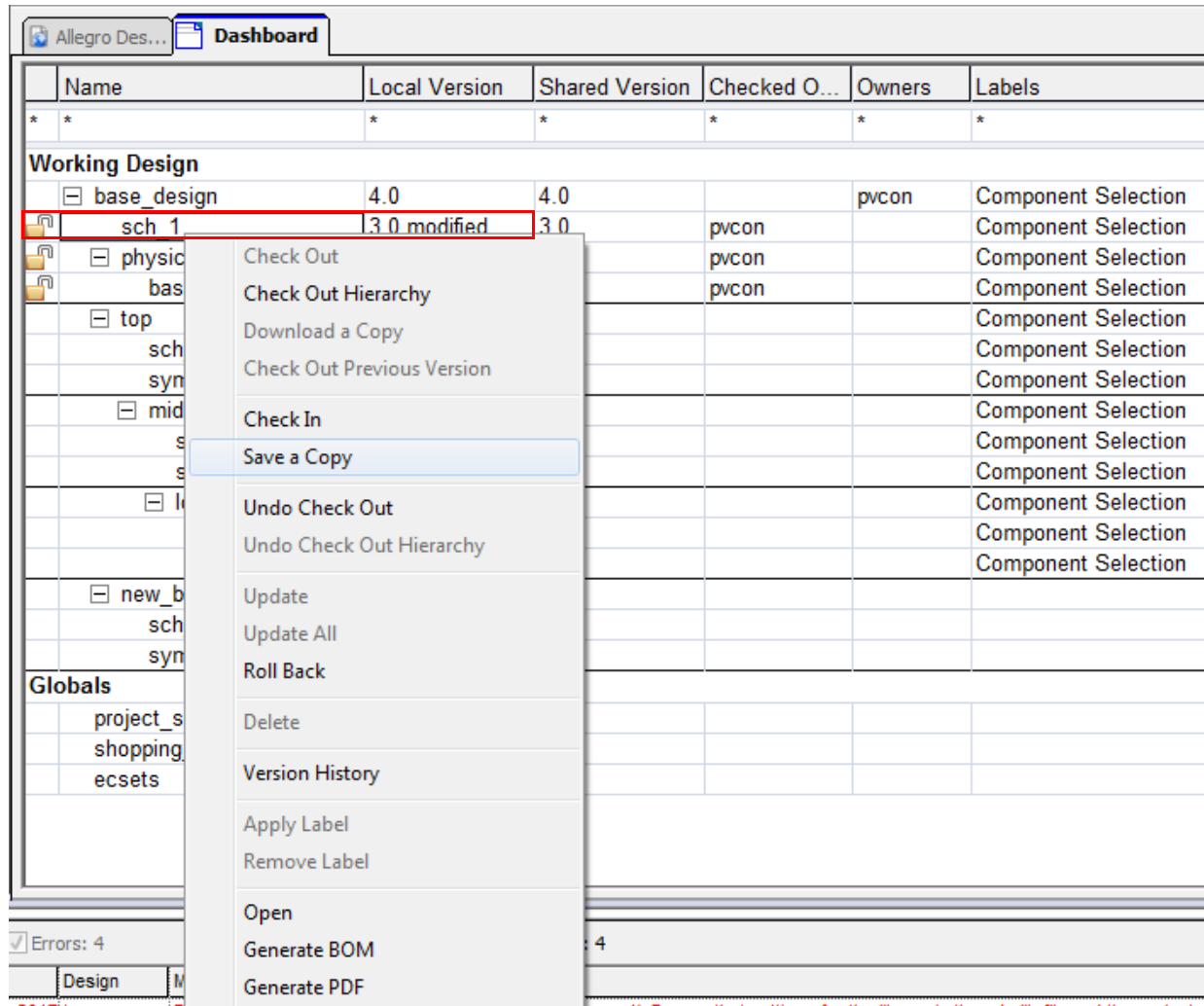
After designers check out an object and modify it as required in DE-HDL, they can save data without checking it in. For example, you can save design objects. This has dual benefits: safe storage, and, when required, easy retrieval of the data by administrators or integrators even in the absence of designers.

This feature is also useful if you do not want an object to be available to the entire project team at a particular point in time. In such cases, rather than check in an object, you can choose to save a draft of it for safe storage or to share it with an integrator.

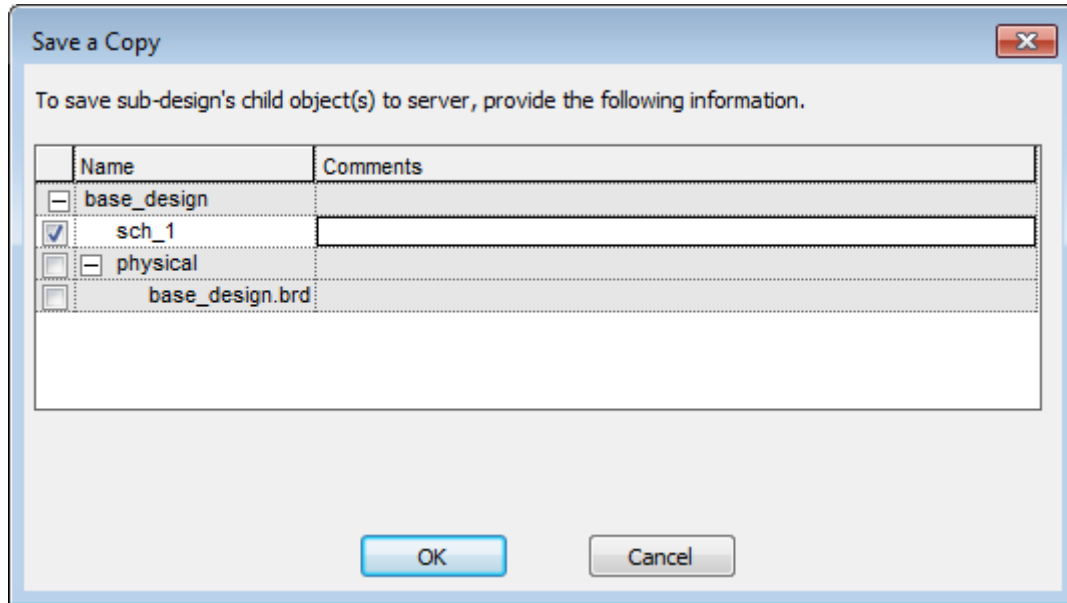
Allegro X EDM Design Management User Guide

Working with Designs in Allegro Design Management

To save data, right-click on a design object in the Allegro Design Management dashboard and choose *Save a Copy*.



The Save as Copy dialog appears.



1. Select the required sub-design's child objects.
2. Specify comments if required, and click *OK*.

After you save a copy of the data, the version number of the object changes in the dashboard.

	Name	Local Version	Shared Version
*	*	*	*
Working Design			
<input type="checkbox"/>	base_design	4.0	4.0
<input checked="" type="checkbox"/>	sch_1	3.0.3	3.0.3
<input type="checkbox"/>	physical	2.0	2.0
<input type="checkbox"/>	base_design.brd	1.0	1.0

Important

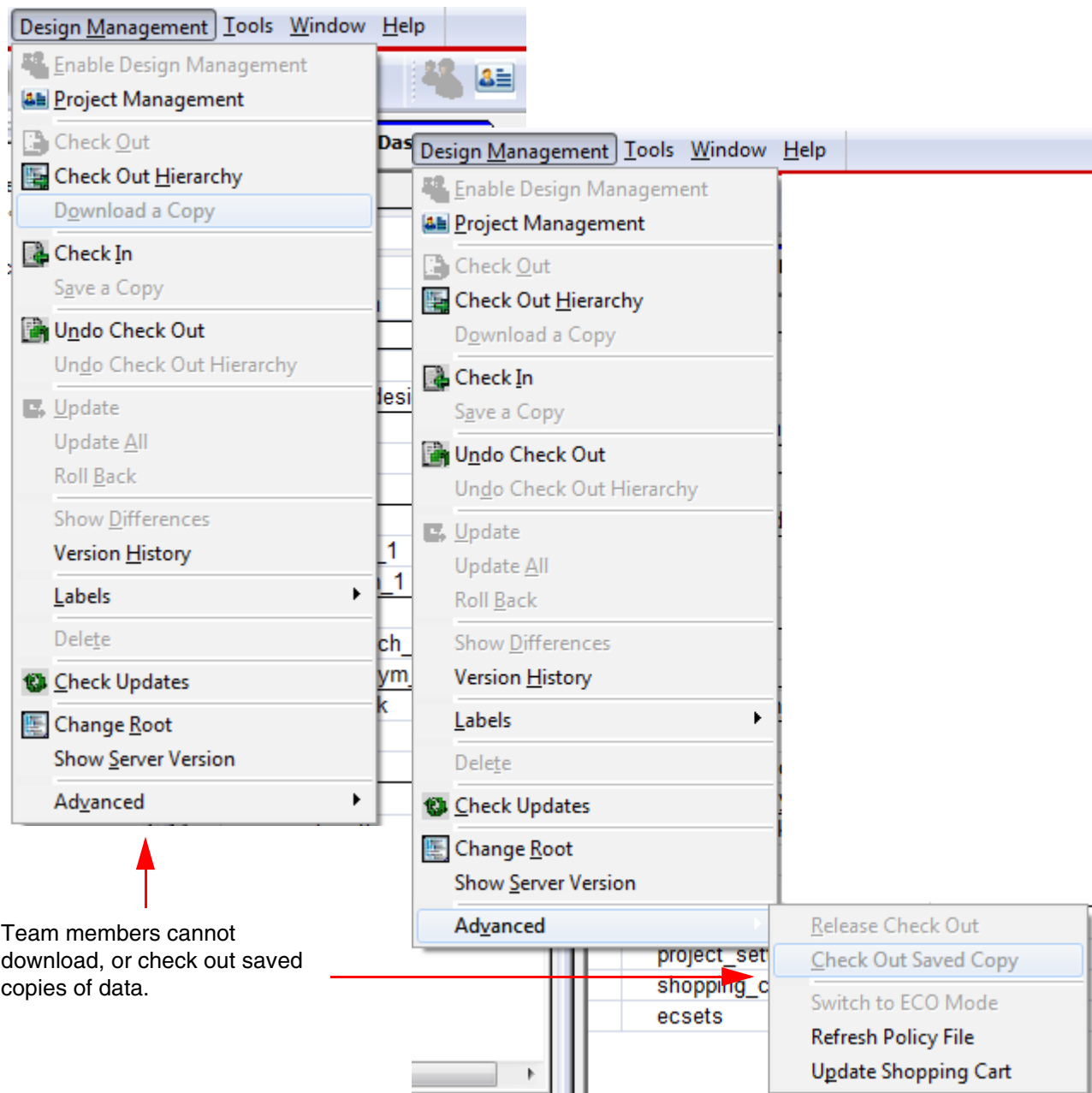
The changed version number is not visible to team members.

After a copy of data has been saved, the integrator can download a copy of the data, and can also check it out if required.

Allegro X EDM Design Management User Guide

Working with Designs in Allegro Design Management

Team members cannot download or check out a copy of saved data. These options are disabled as illustrated:

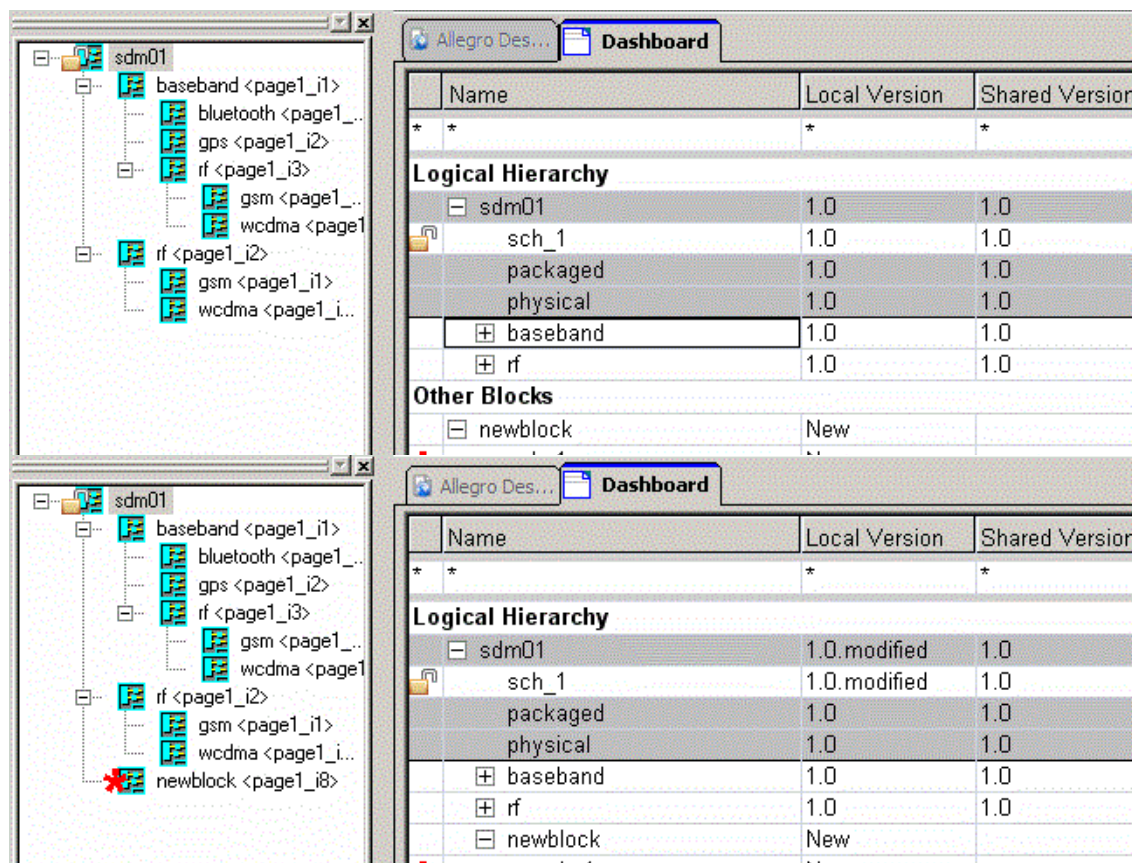


Adding New Blocks

To add new blocks, do the following:

1. Check out the required design.
2. Open the design in Allegro Design Entry HDL.
3. Add a new block.
4. Save the schematic.

The *Logical Hierarchy* and *Dashboard* now reflect the change depending on whether you have added the block within the hierarchy or outside it (other blocks):



Rolling Back Changes or Undoing Check Out

After you check out a design, subdesign, or page, and make changes, you have three options—you can roll the changes back, you can undo the check out, or you can check in the changes you have made.

When you roll the changes back, the changes you made to the design, subdesign, or page after checking it out are discarded but the object is still checked out. However, when you undo

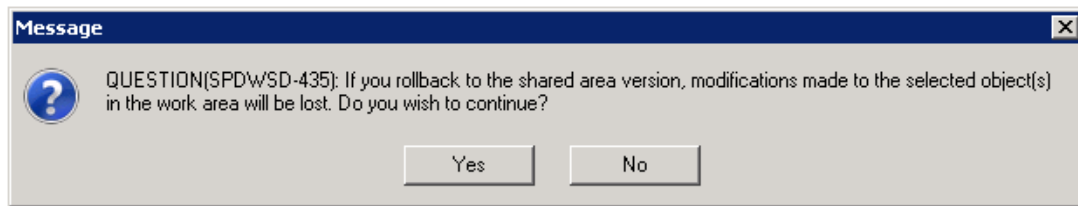
the checkout of a design, both, the local changes you made, as well as the checkout, are cancelled.

Rolling Back Changes

To roll back changes, do the following:

1. Right-click a checked out design, subdesign, or page or multiple objects.
2. Choose *Roll Back*.

A message is displayed.



Undoing Check Out

If you need to undo a check out, do the following:

1. Right-click a checked out design, subdesign, or page or multiple objects.
2. Choose *Undo Check Out*.

This returns the design hierarchy to its previous state and cleans up the work area.

Checking In a Design, Subdesign, or Page

To check in a modified design object, do the following from the dashboard:

1. Right-click the modified design.
2. Choose *Check In*.

The check in window appears displaying the modified objects being selected. If you select an object that is not modified, it will not be checked in.

3. You can add a comment in the *Comments* field. This step is optional.

Allegro X EDM Design Management User Guide

Working with Designs in Allegro Design Management

By default, comments are not mandatory. However, if required, you can make comments mandatory. To do so, set the `checkin_comments_mandatory` directive in the `START_SDM` section to `TRUE` in the site or project-level `.cpm` file. The default value of this directive is `FALSE`.

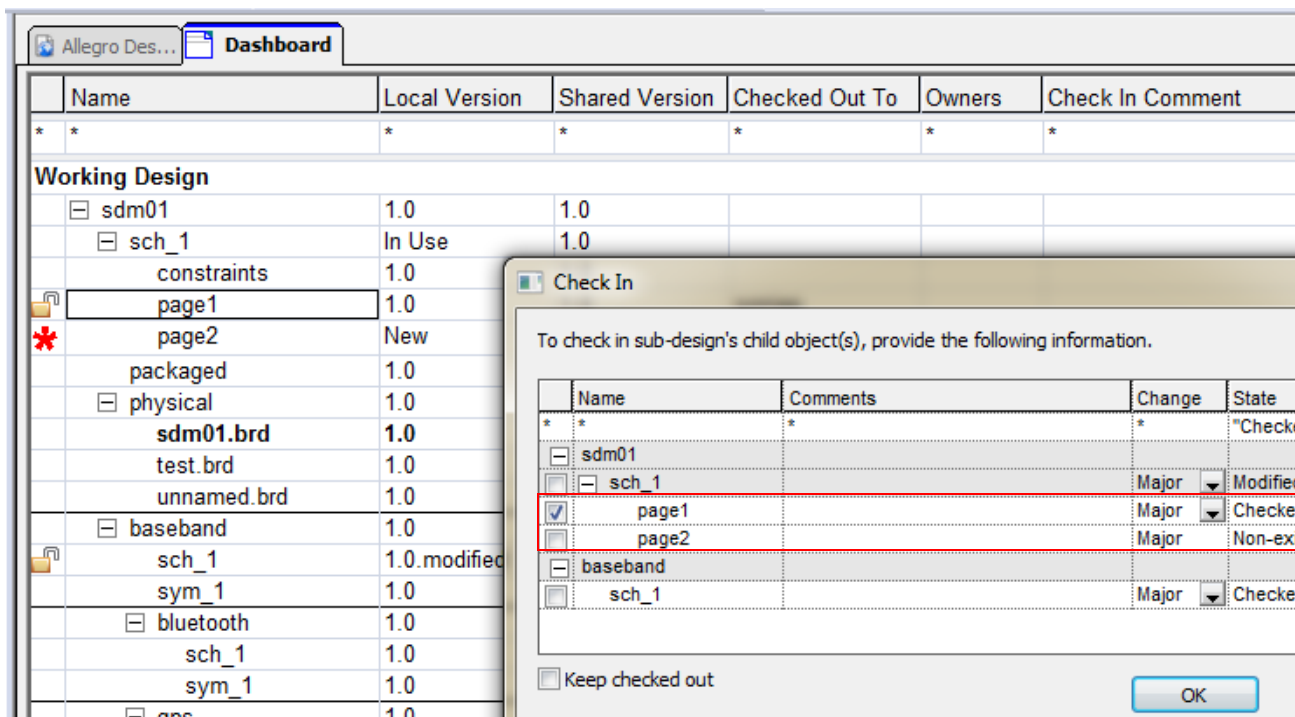
4. Select the check-in type as *Major* or *Minor*.

The default check-in type in Allegro Design Management is *Major*. If you want to modify the default check-in type, use a directive, `checkin_change_type`, in the site or project-level `.cpm` file under the `START_SDM` section.

Even if you set the default check-in type as *Minor*, you can choose *Major* in the *Change* column.

Important

The default check-in type for a new page or object is *Major*, and this cannot be modified regardless of the directive setting.



5. Click *OK*.

The entry for the design in the dashboard changes:

- ☐ The checked out icon is removed

- ☐ *Local Version* is incremented.
- ☐ *Shared Version* becomes the same as *Local Version*.
- ☐ The check in comment is also displayed.



Tip

You can check out, check in, roll back, or undo a checkout for more than one design object at a time. To select more than one object, either select the first block and keep moving the mouse to the last block, or select the first block, press the Shift key and select the last block.

Changing the Root Design

If you create a physical reuse block, you might need to change the root design. You cannot change the root design when working from within Design Entry HDL.

1. To set a root design for your blocks, do either of the following:
 - ☐ Choose a design node in the *Logical Hierarchy* tab. Right-click the selected node and choose *Set As Root Design*.
 - ☐ Click the *Change root design* icon on the toolbar.
 - ☐ Choose *Design Management — Change Root*.
The *Select the new root design* window appears.
2. Choose the library.
3. Choose a cell that will be the new root design.
4. Click *OK*.

Synchronizing with Shared Area

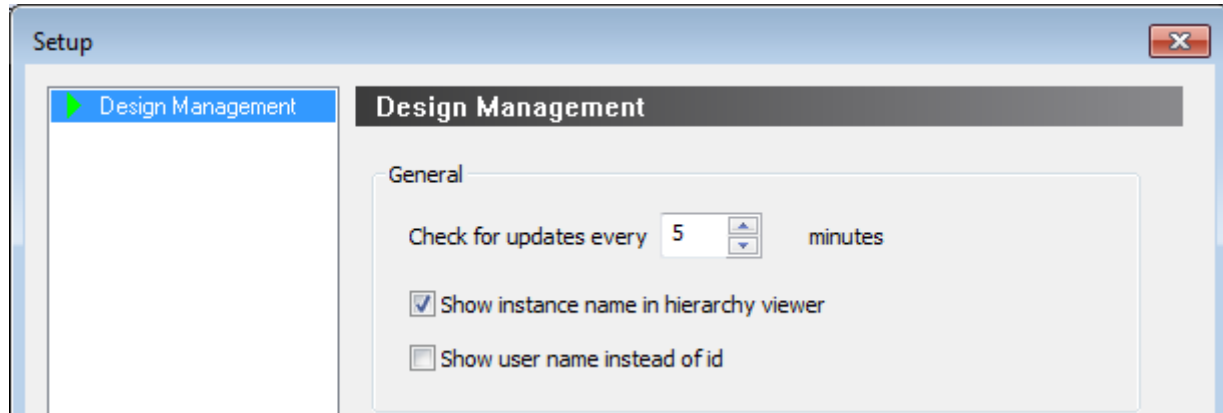
Allegro Design Management automatically notifies changes in the shared area at a specified time interval. By default, this time interval is five minutes.

Note: This feature is not available for the team design environment in Design Entry HDL.

To modify this interval, do the following:

1. Choose *Project — Settings*.
2. Choose *Design Management* in the left pane.

3. Specify the time interval in the *General* group box.



If you need to view the new changes in the shared area before a notification from Allegro Design Management, choose *Design Management — Check Updates*.

A modified and checked-in object is represented by an exclamation symbol corresponding to that object. To get the latest object from the shared area, do one of the following:

- Right-click the changed object and choose *Update*.
- Click *Resolve* in the *Violations* tab.

Releasing Check Out

As an integrator, you can use the *Release Check Out* option to release the check-out done by any other user. You may need to do this, for example, if a team member is on leave or has left the team. The LDI can only release the check-out of logical objects; the PDI can release the check-out of physical objects.

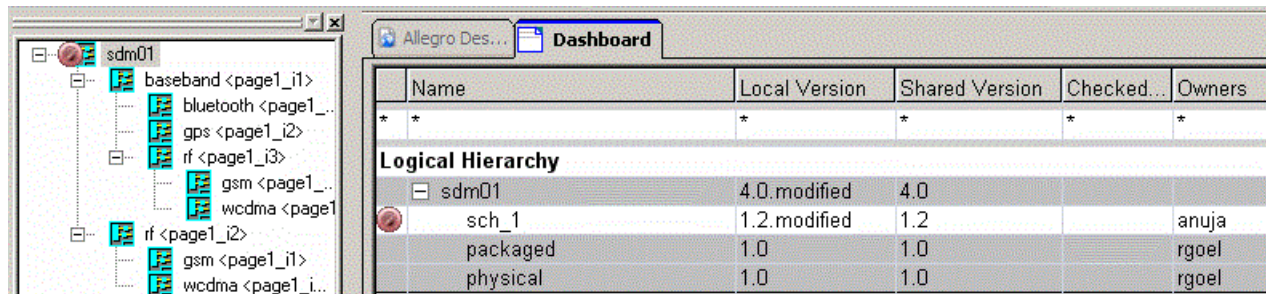
To release the check-out, do the following:

1. Select the object for which you want to remove the check-out.
2. Choose *Design Management — Advanced — Release Check Out*.

The check-out is removed.

Resolving Unauthorized Design Modifications

Although not recommended, designers may sometimes modify a drawing without checking out the design. In such cases, Allegro Design Management highlights this unauthorized modification.



Allegro Design Management recognizes that the design has been modified locally without being checked out, and thus marks the edited object as *Hijack*, that is, an unauthorized modification. This is true even for designs not assigned to a user.

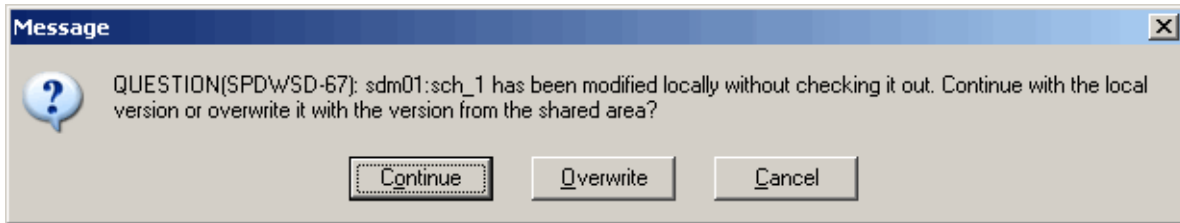
Note: When Allegro EDM itself modifies an object without first checking out the object, EDM marks the status as *In Use*. When a user modifies an object without first checking it out, Allegro EDM marks the status as *Hijack*.

To resolve such cases, you can either roll back the changes that you made or you can check the design out:

1. Right-click the design.
2. Do one of the following:
 - ☐ Choose *Roll Back* if you want to roll back the changes. In this case, the changes you made to the design are discarded.
 - ☐ Choose *Check Out*.

The Check Out option is available only to the assigned owner. You cannot check out a design assigned to another designer. So if you have made changes to a design that is not assigned to you, you might lose your changes.

Allegro Design Management indicates that a newer local version exists, that is, the design has been modified locally without being checked out.



- a. Click *Continue* to save the changed local version.
- b. Right-click and check in the design.

Deleting Objects from the Allegro Design Management Dashboard

You can delete unused logical blocks and physical board files from the Allegro Design Management dashboard. Remember the following when deleting in Allegro Design Management:

- Deleted objects ***cannot be recovered***. They are permanently deleted from the disk as well as from the Allegro Design Management database.
- When you delete an object, ***all current and older versions are deleted***.
- Once a user performs a delete operation,
 - ☐ all affected files are removed from that user's work area.
 - ☐ other users are notified of the mismatch between their work areas and the project's shared area through the dashboard. A deleted object is indicated in the dashboard in red with the following text: "Deleted from shared area" Users need to pull in the shared area changes into the work area.

Deleting a Logical Block from the Allegro Design Management Dashboard

When deleting blocks from Allegro Design Management:

- you can delete only uninstantiated blocks in the design. Any block that is currently in use in the design cannot be deleted. Only blocks in the *Other Objects* section can be deleted.
- when you delete a block, its child blocks are not deleted.

For example, deleting Mid that contains Low will not delete Low.

Allegro X EDM Design Management User Guide

Working with Designs in Allegro Design Management

- Any new block that you add in DE-HDL before you check in the design, cannot be deleted by Allegro Design Management. It should be deleted from DE-HDL.

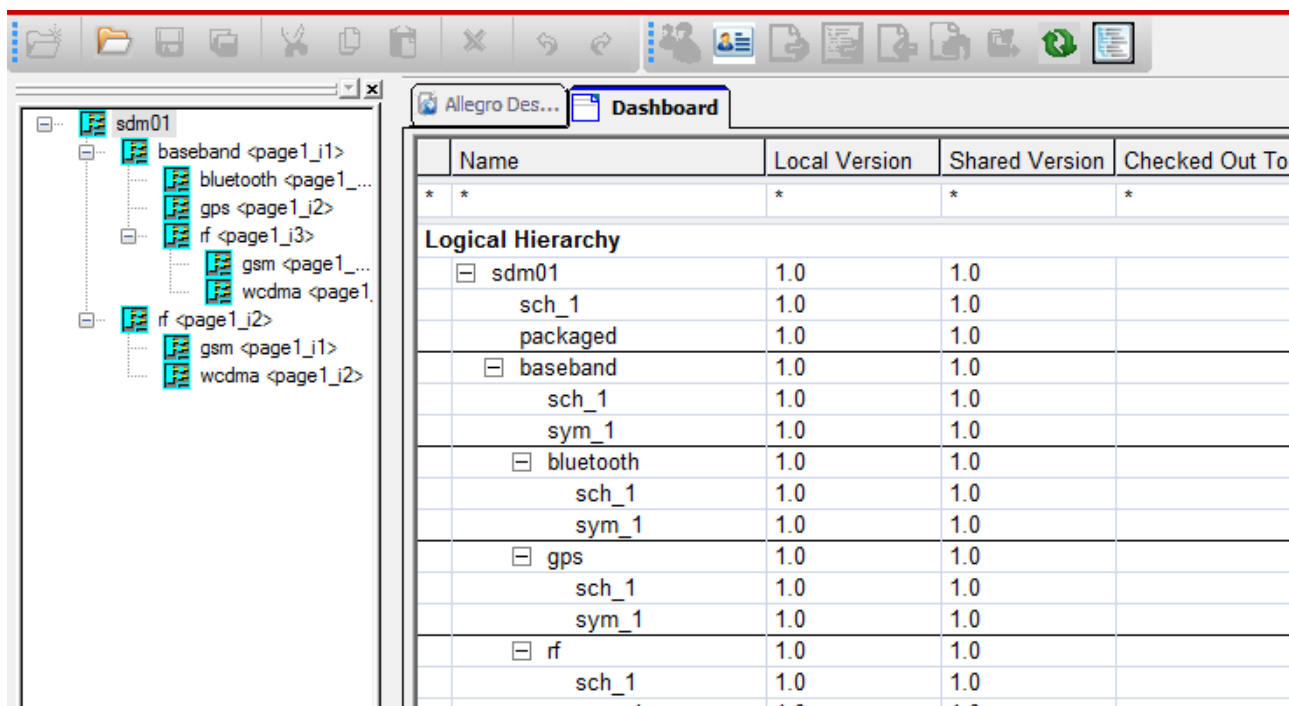
As you see, the block is still in use in the design For details, see the following:

- ❑ [Uninstantiating a Block](#)
- ❑ [Removing the Uninstantiated Block](#)

Uninstantiating a Block

To remove the `baseband` block from the design, do the following from the Allegro Design Management dashboard:

1. Check out the schematic view, such as `sch_1`.






Right-click `sch_1` and choose *Check Out*.

Allegro X EDM Design Management User Guide







Working with Designs in Allegro Design Management

The schematic is checked out. You can now make changes to it in DE-HDL.

Logical Hierarchy					
	sdm01	1.0	1.0		rganguli
	sch_1	1.0	1.0	rganguli	
	packaged	1.0	1.0		
	baseband	1.0	1.0		
	sch_1	1.0	1.0		
	sym_1	1.0	1.0		

2. Select a license, if prompted.
3. Right-click the block `baseband` and choose *Delete*.
4. Choose *File — Save All*.
5. Close DE-HDL.
6. Check in `sch_01`.

The dashboard is updated and the `baseband` block along with its sub-blocks is moved to the *Other Blocks* section.

	packaged	1.0	1.0		
	rf	1.0	1.0		rgoel
	sch_1	1.0	1.0		
	sym_1	1.0	1.0		
	gsm	1.0	1.0		
	sch_1	1.0	1.0		
	sym_1	1.0	1.0		
	wcdma	1.0	1.0		
	sch_1	1.0	1.0		
	sym_1	1.0	1.0		
Other Blocks					
	baseband	1.0	1.0		
	sch_1	1.0	1.0		
	sym_1	1.0	1.0		
	bluetooth	1.0	1.0		vikas
	sch_1	1.0	1.0		
	sym_1	1.0	1.0		
	gps	1.0	1.0		
	sch_1	1.0	1.0		
	sym_1	1.0	1.0		
Globals					
	project_settings	1.0	1.0		
	ecsets	1.0	1.0		

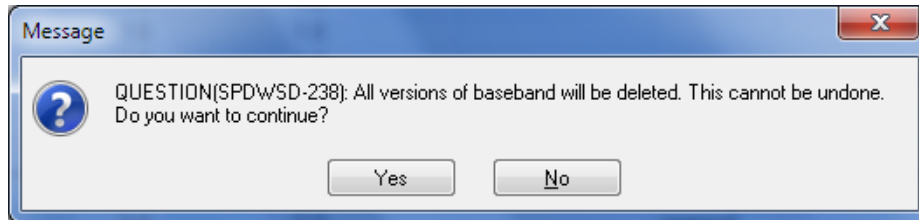
You can now remove the `baseband` block.

Removing the Uninstantiated Block

Now that the block is no longer in the design, it can be removed from Allegro Design Management.

1. Right-click the `baseband` block, and choose *Delete*.

Allegro Design Management checks that the block is not in use and is not open in any application, and then prompts you for confirmation.



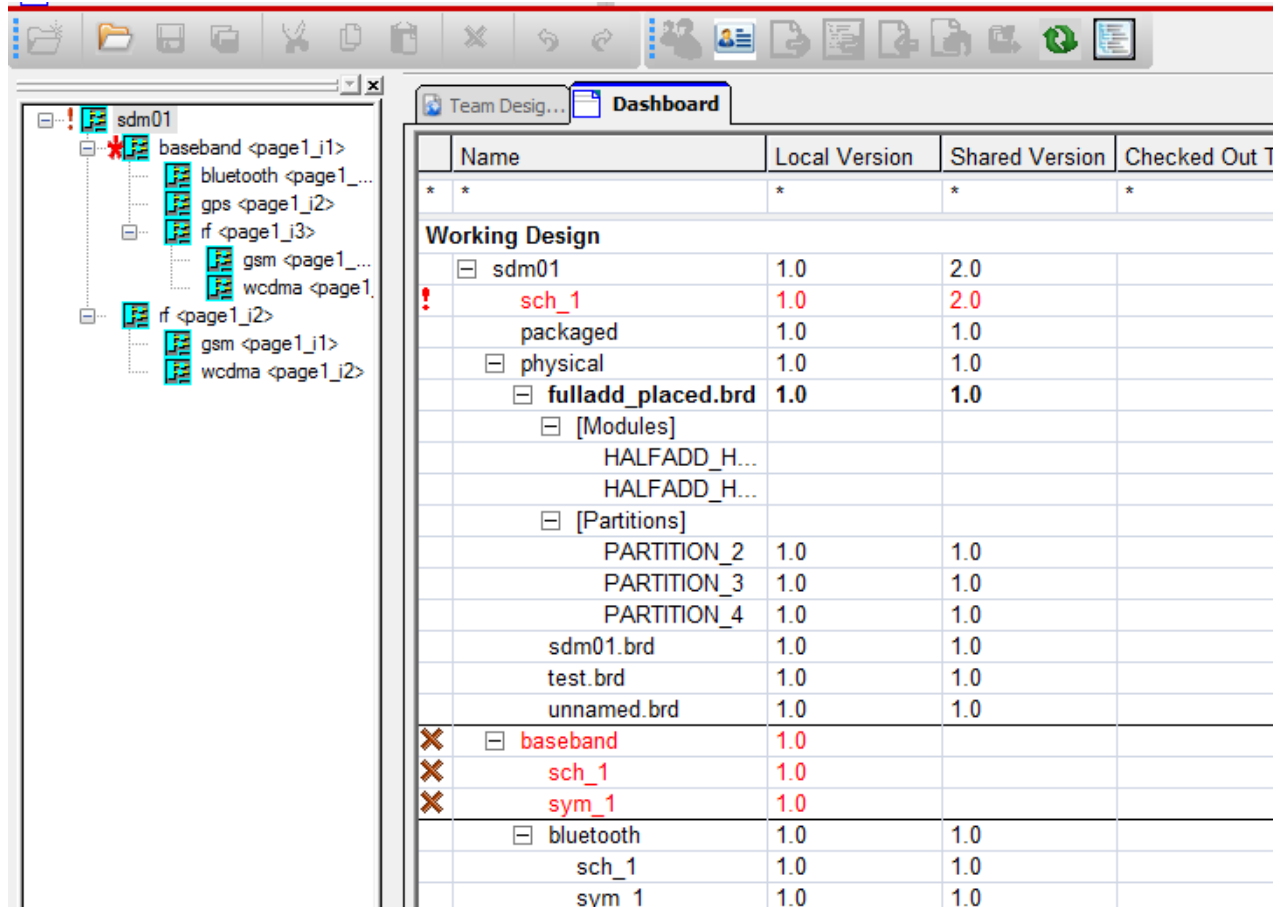
2. Click Yes.

The block is removed from the dashboard of the user who deleted it.

Allegro X EDM Design Management User Guide

Working with Designs in Allegro Design Management

All the other team members will see the block as deleted from the shared area and the new version number of `sch_01`.



The dashboard shows the following:

- `sch_1` has a new version.
- `baseband` block and its views are no longer available in the shared area.

3. Choose *Design Management — Check Updates*.

4. Right-click each entry that has an icon (denoting that it has changes) and choose *Update*.

Important

Exercise caution **when adding a new block with the same name as that of a deleted block**. Once a block is deleted, and all work areas are updated, only then should you add a block. If team members have not updated their work areas, the newly-created block may not be seen correctly.

Deleting a Physical File

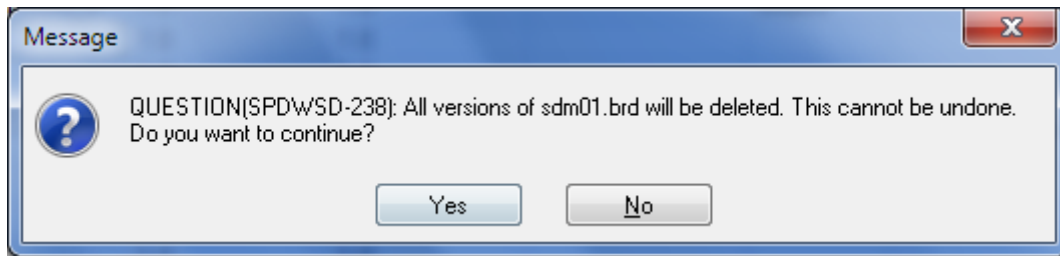
Remember the following when deleting physical files:

- You **cannot delete** the master board file. The master board file is shown in bold.
- Any physical file that is deleted is completely removed. You cannot recover the file or any version.
- Deleting a board file removes all its partitions.

To delete a physical board file, do the following:

1. Identify the file to delete.
2. Right-click the board file and choose *Delete*.

You will be prompted for confirmation.



3. Click Yes.

The board is removed from the project and the dashboard is refreshed.

Allegro X EDM Design Management User Guide

Working with Designs in Allegro Design Management

The dashboard of the other users shows that there has been a change in the shared area and the work area needs to be updated.

The screenshot displays the Allegro X EDM Design Management interface. On the left, a project tree shows 'gsm <page1_i1>' and 'wcdma <page1_i2>'. The main area is divided into two sections: 'Working Design' and 'Physical Hierarchy'.

Working Design Table:

Design	Version	Version	User
sdm01	2.1	2.2	rganguli
sch_1	2.0	2.0	
packaged	1.0	1.0	
physical	1.1	1.2	
fulladd_placed.brd	1.0	1.0	
[Modules]			
HALFADD_H...			
HALFADD_H...			
[Partitions]			
PARTITION_2	1.0	1.0	anuja
PARTITION_3	1.0	1.0	rgoel
PARTITION_4	1.0	1.0	vikass
test.brd	1.0		
unnamed.brd	1.0	1.0	
rf	1.0	1.0	rgoel
sch_1	1.0	1.0	
sym_1	1.0	1.0	
physical	1.0	1.0	
rf.brd	1.0	1.0	
gsm	1.0	1.0	
sch_1	1.0	1.0	
sym_1	1.0	1.0	
wcdma	1.0	1.0	
sch_1	1.0	1.0	
sym_1	1.0	1.0	

Physical Hierarchy Table:

Design	Version	Version	User
sdm01	2.1	2.2	rganguli

Below the tables, a message log is displayed with the following entries:

Type	Message	Time	Design
Error	Failed to load component cell 'cap.r:sym_1'. Ensure that settings for the library in the cds.lib file	31-Mar-201 4 15:48:02	sdm01
Warn	WARNING(SPDWSD-4): Version mismatch found for sdm01:physical. Version in shared area is or continue with the local version.	31-Mar-201 4 16:38:03	
Warn	WARNING(SPDWSD-10): sdm01:physical:test.brd has been deleted and does not exist in the sl	31-Mar-201 4 16:38:03	
Warn	WARNING(SPDWSD-4): Version mismatch found for project_settings. Version in shared area is or continue with the local version.	31-Mar-201 4 16:38:03	

The interface also includes tabs for 'Hierarchy', 'Physical Hierarchy', 'Violations', and 'Command Window'. The 'Violations' tab is currently active, showing the message log.

4. Click *Resolve* for each of the messages in the *Violations* tab.

The dashboard and work area are both updated with the latest status of the project as in the shared area.

Working with Flat Designs in Design Entry HDL

When you work with managed projects in Design Entry HDL, remember a few points.

Note: In the documentation, the terms *managed projects*, *pages*, *blocks*, and *constraints* have sometimes been used to refer to objects in projects that have been design management-enabled.

FAQs - Working with Managed Projects

■ **Can I edit a managed page in Design Entry HDL if the page has not been checked out?**

If a page is not checked out, it cannot be edited in DE-HDL. It is a read-only page.

■ **How can I insert a new page?**

To insert a page between the schematic hierarchy, you must check out the complete schematic view. To insert a page at the end of the schematic, you do not need to check out the schematic.

■ **How do I delete a managed page?**

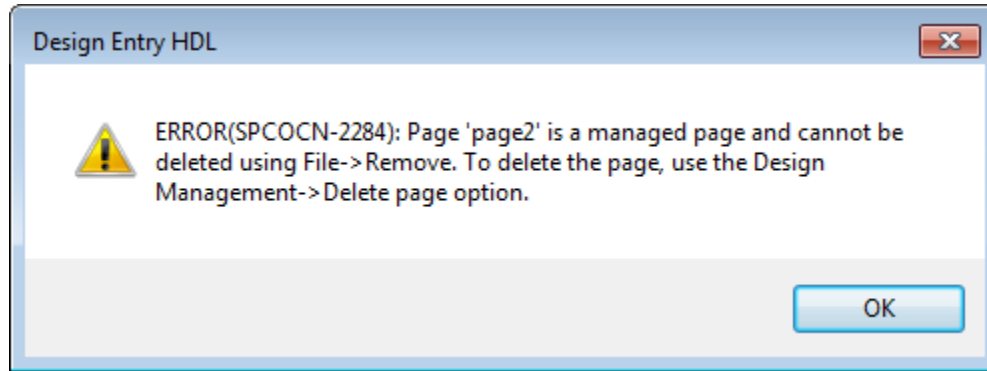
If you are assigned to a schematic, you can delete a managed page by doing the following:

- ❑ In the DE-HDL Hierarchy Viewer, right-click on a page, and select *Design Management — Delete* from the pop-up menu.
- ❑ In DE-HDL, select a page, then choose *Design Management— Delete* from the main menu.
- ❑ From the DE-HDL Allegro Design Management dashboard, select a page. right-click and choose *Delete* from the pop-up menu.

The page will be deleted from the shared area.

Note: You cannot use the Design Entry HDL delete feature to delete a managed page as the page is locked in Allegro Design Management. Doing so will result in an error as

illustrated:



■ **How do I edit constraints in a managed project?**

Constraints can only be edited by one user at a time even if a constraint has been assigned to multiple users. To edit a managed constraint, check out the constraint, edit it, then check it in.

■ **How do I add or edit a sync constraint?**

Check out a page, assign a sync constraint to a wire (for example, a differential pair), then launch Allegro Constraint Manager with Design Entry HDL. The constraint will be visible in the DE-HDL Constraint Manager. You can then check in the page.

■ **How do I assign models to discrete devices in a managed page?**

To assign a model to a discrete device on a managed page, you must ensure the following:

- ☐ the page must be checked out
- ☐ the physical view must also be checked out

Note: If you are assigned to a page but not to the physical view, you cannot assign models to devices on that managed page.

■ **How do I use and manage auto-generated models for discrete devices in an Allegro Design Management-managed Design Entry HDL (DE-HDL) design?**

When schematic designers generate models for discrete devices in DE-HDL, this leads to the creation of the `devices.dml` file in the physical view.

In managed designs, schematic designers might not have the permission to check out the physical view. As a result, model generation might fail in DE-HDL. It might also cause the physical view to be marked as having undergone an unauthorized modification.

Allegro X EDM Design Management User Guide

Working with Designs in Allegro Design Management

Further, when the schematic is checked in with the applied models, the `devices.dml` file is not checked in because the physical view was not checked out.

To address this, when preparing the project for team design, you, as the integrator, need to define the following directives in the `GLOBAL` section of the project `.cpm` file:

- ❑ `SI_DML_WORKING_LIB ' ../models/devices.dml '`
- ❑ `SI_MODEL_PATH ' ../models' (append this path to the existing SI_MODEL_PATH)`

Also complete the following:

- a. Create a folder called `models` (as a sibling to the `sch_1` view) under each design block in the project for which model assignment is to be exercised.
- b. Remove “.” from `SI_MODEL_PATH`.
- c. Remove any existing stale `devices.dml` file from the physical folder.
- d. Modify the Allegro Design Management policy file, which is at the site level to exclude any `devices.dml` files from the physical object definition:

```
<exclude path="${projdir}/${design_lib}/${design_name}/${  
${physical_view}/devices.dml*" />
```

Note: This task will require write permission, and is usually performed by the ECAD Administrator.

With these tasks completed, after the design is team-design enabled and team members start working on their sub-blocks, auto-generated model information is stored in the `devices.dml` file under the `models` directory of the same block. For example, the information will be stored in the `<Allegro EDM project installation directory>/worklib/MID1/models/devices.dml` file.

When updating a sub-block, the integrator will see the updated `devices.dml` file under the sub-block. As a result, the model assignment launched on the original root design will display all the models applied for the instances under the sub-block.

As the integrator, if you intend to merge the model information at the root-design level, that is, the `devices.dml` file, you need to check out the root schematic view, run the model assignment in DE-HDL, and apply the changes.

■ How do I package a managed design?

To package the design, you must check out the package view, if any.

Note: If you are not assigned to the package view, you cannot package the design.

■ How do I import sheets into a managed design?

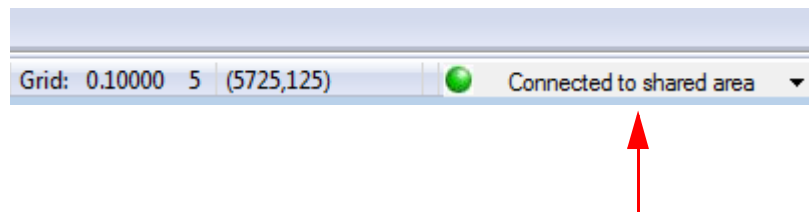
To import a page between the schematic hierarchy, you must check out the complete schematic view. To import a page at the end of the schematic, you do not need to check out the schematic.

For details about working with non-design management enabled designs, see the Design Entry HDL documentation.

Allegro Design Management User Interface in DE-HDL

The Design Management menu option, and other interface elements related to design management in Design Entry HDL are available only if you have the Team Design or Allegro Data Manager.

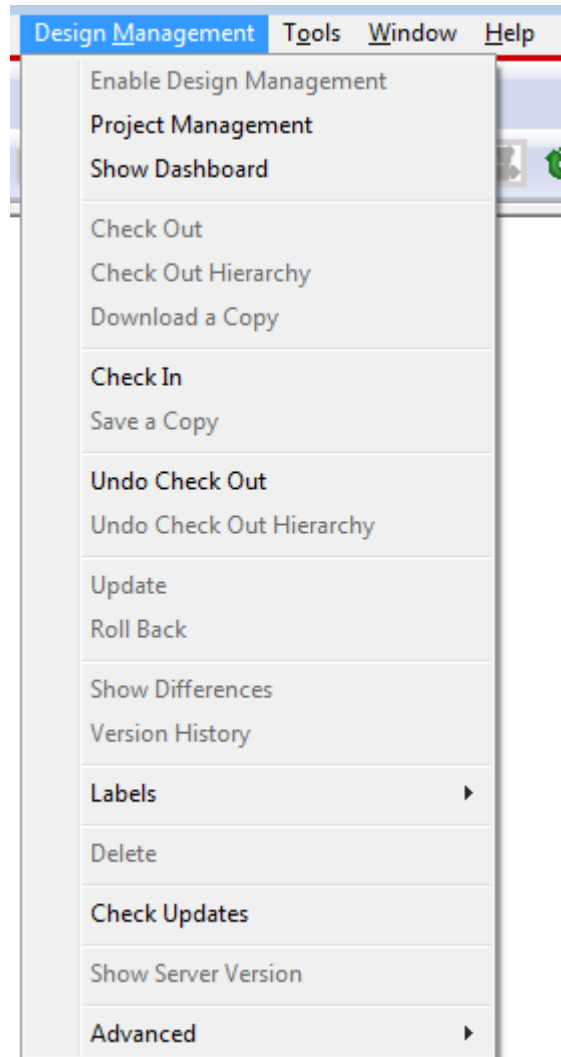
The following icon and status are displayed in the bottom-right corner of the DE-HDL screen when you work with a managed design:



Allegro X EDM Design Management User Guide

Working with Designs in Allegro Design Management

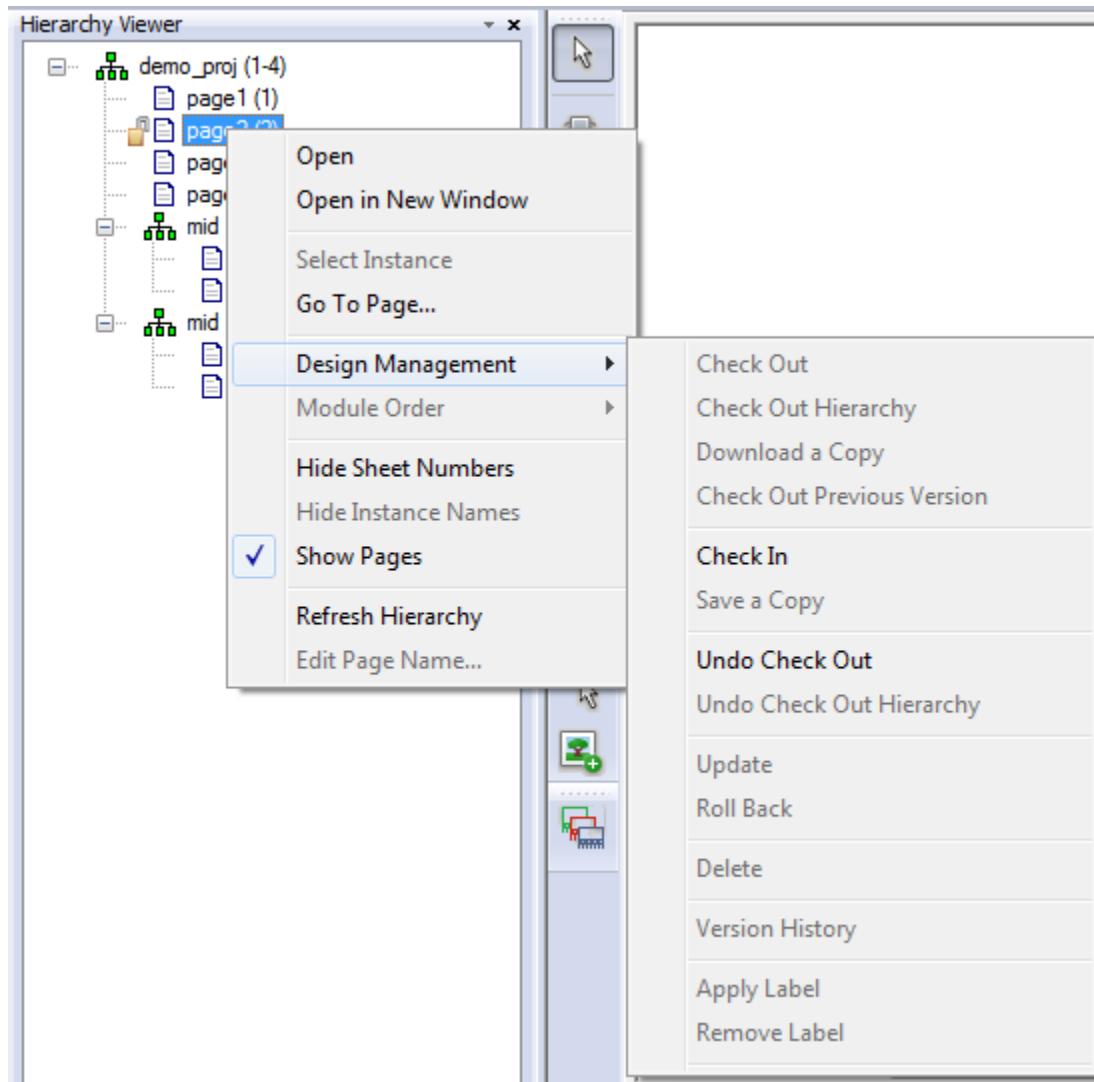
Allegro Design Management options in Design Entry HDL can be accessed from the toolbar as illustrated:



Allegro X EDM Design Management User Guide

Working with Designs in Allegro Design Management

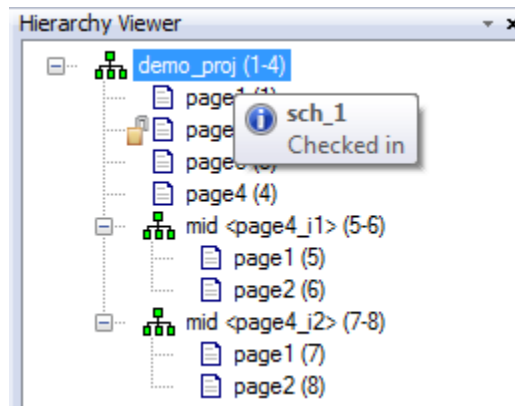
Allegro Design Management popup menu options in Design Entry HDL are as follows:



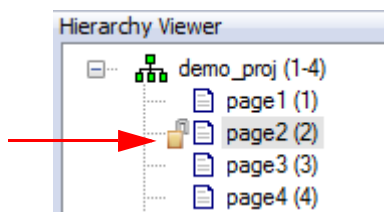
Allegro X EDM Design Management User Guide

Working with Designs in Allegro Design Management

The status of a managed page in Design Entry HDL is indicated by a tooltip when you hover over a page in the Hierarchy Viewer.



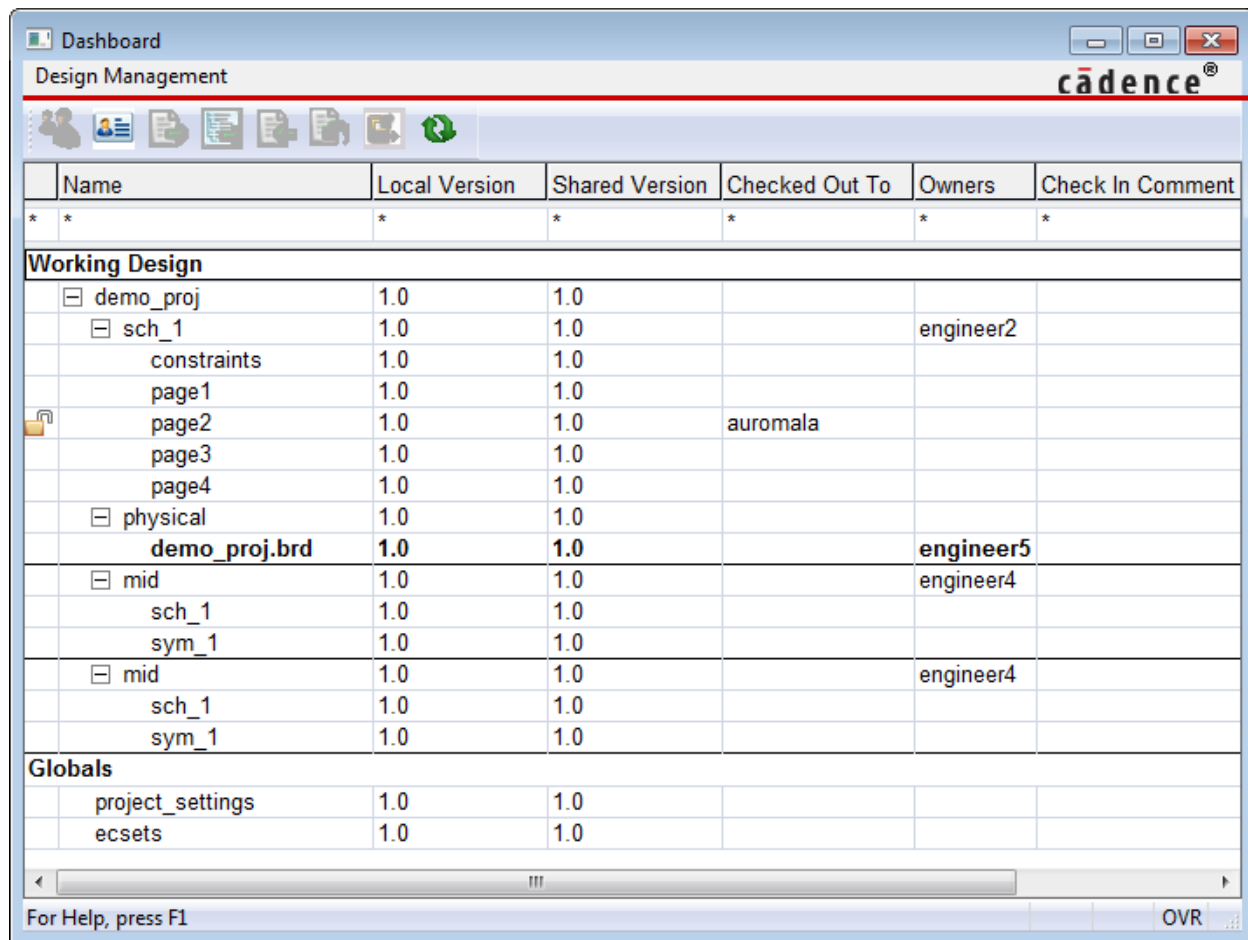
A checked-out page is indicated by an unlock icon:



The dashboard of a managed design in Design Entry HDL is as follows:

Allegro X EDM Design Management User Guide

Working with Designs in Allegro Design Management



The screenshot shows the Allegro Design Management interface. At the top is a 'Dashboard' window with the 'Design Management' title bar and the Cadence logo. Below the title bar is a toolbar with icons for design management. The main area contains a table with the following columns: Name, Local Version, Shared Version, Checked Out To, Owners, and Check In Comment. The table is divided into three sections: 'Working Design', 'Globals', and a footer section. The 'Working Design' section lists various design components like 'demo_proj', 'sch_1', 'constraints', 'page1', 'page2', 'page3', 'page4', 'physical', 'demo_proj.brd', 'mid', 'sch_1', and 'sym_1'. The 'Globals' section lists 'project_settings' and 'ecsets'. The footer section contains the text 'For Help, press F1' and an 'OVR' button.

Name	Local Version	Shared Version	Checked Out To	Owners	Check In Comment
* *	*	*	*	*	*
Working Design					
[-] demo_proj	1.0	1.0			
[-] sch_1	1.0	1.0		engineer2	
constraints	1.0	1.0			
page1	1.0	1.0			
page2	1.0	1.0	auromala		
page3	1.0	1.0			
page4	1.0	1.0			
[-] physical	1.0	1.0			
demo_proj.brd	1.0	1.0		engineer5	
[-] mid	1.0	1.0		engineer4	
sch_1	1.0	1.0			
sym_1	1.0	1.0			
[-] mid	1.0	1.0		engineer4	
sch_1	1.0	1.0			
sym_1	1.0	1.0			
Globals					
project_settings	1.0	1.0			
ecsets	1.0	1.0			
For Help, press F1					
OVR					

If you are assigned to a block,
those block rows are displayed

Design Management Menu in Design Entry HDL

Design Management – Enable Design Management

Use this option to enable a design for team management or for version control only. This is done by an integrator.

Design Management – Project Management

Opens the Project Management dialog. When enabling, or after enabling a project for design management, you can make a user a Logical Integrator, Physical Integrator, or both using this dialog.

In this dialog, you can also define whether multiple users can check out the physical view of the design.

When you select the Manage files under physical view as separate objects checkbox, this enables file-level management and allows you to manage files under the physical view as separate objects.

Design Management – Show Dashboard

Use this option to display the dashboard in Allegro Design Management. The dashboard window shows the entire design and the state of all its subdesigns, their status, and team member ownership.

Design Management – Check Out

After the integrator has set up the shared area and assigned ownership rights for subdesigns in a project, you, as the designer, can access the project and start work on the subdesign you own.

Use this option to check out the required design objects such as subdesigns, or blocks when working with hierarchical designs or pages, or constraints when working with flat designs.

Design Management – Check Out Hierarchy

Depending on your requirements and rights, you can check out just the design or the design along with all its subdesigns.

Check Out and Check Out Hierarchy can apply at any level in the design, whether it is root or a subdesign. Check Out hierarchy includes checking out all the child objects of the selected design.

Design Management – Download a Copy

An integrator in a team design environment can download a saved copy of a design object that was checked out and modified. The copy is downloaded from the shared area.

Design Management – Check In

After the integrator has set up the shared area and assigned ownership rights for subdesigns in a project, you, as the designer, can access the project and start work on the subdesign you own.

Use this option to check out the required design objects such as subdesigns, or blocks when working with hierarchical designs or pages, or constraints when working with flat designs.

Design Management – Save a Copy

In a team design environment, save a copy of the design object that you checked out and modified. The copy is saved to the shared area.

Design Management – Undo Check Out

Undo the checkout of a design object in a team design environment. This returns the design hierarchy to its previous state and cleans up the work area.

Design Management – Undo Check Out Hierarchy

Depending on your requirements and rights, you can check out just the design or the design along with all its subdesigns. After checking out the hierarchy, you can undo the checkout.

Design Management – Update

Update merges the shared area changes with the local work area.

Design Management – Roll Back

Rolls back changes that are made to a design that was not checked out when working in a team design environment.

Designers may sometimes modify a drawing without checking out the design. In such cases, Allegro Design Management highlights this unauthorized modification.

In such cases, you can roll back the change.

Design Management – Show Differences

When working with a cache-enabled project, this option displays cell-level changes between two versions of a schematic. For example, if you check out a schematic, add a new cell, then check the schematic in, using this option will display the differences between the two schematic versions.

Design Management – Delete

Deletes a managed page from the shared area.

Design Management – Check Updates

Displays changes in the shared area before a notification from Allegro Design Management when working in a team design environment.

Design Management – Advanced – Release Check Out

As an integrator, you can use the Release Check Out option to release the check-out done by any other user. You may need to do this, for example, if a team member is on leave or has left the team. The LDI can only release the check-out of logical objects; the PDI can release the check-out of physical objects.

Design Management – Advanced – Check Out Saved Copy

In a team design environment, check out a copy of a design object that is modified and saved in the shared area.

Design Management – Advanced – Switch to ECO Mode

Switches a design to ECO (ready for release) mode when working in a team design environment.

For cache-enabled project types such as board and highspeed, Allegro Design Management manages additional, internal global objects—shopping_cart and design_cache—using Library Revision Manager (LRM). Allegro Design Management operates in two types of modes: Work in Progress (WIP) and ECO.

The design_cache object is created when the integrator switches the design mode to ECO.

Design Management – Advanced – Refresh Policy File

Fetches all the changes from the site level when working in a team design environment.

In case your project has object definitions that are not there in the site-level policy file, do not use the Refresh Policy File option. Doing so will overwrite the local policy file and objects defined based on the older policy file may no longer work.

Design Management – Advanced – Update Shopping Cart

Updates the shopping cart. The shopping cart is only available in the data management mode.

Adding Derived Data to Allegro Design Management

Derived data, that is, information that is derived from an ECAD design, can comprise derived items and datasets. You can add derived data for schematic designs and physical files to Allegro Design Management. The policy file defines how a project that has been enabled for design management is divided into design objects and what each design object contains. By default, two kinds of derived data options are available in the policy file:

- BOM
- PDF

You can configure the Cadence-supplied policy file according to your requirements to generate custom-derived data that can be shared and managed with Allegro Design Management. This chapter covers the following topics:

- [Generating Default Derived Data](#)
- [Derived Data Controls in the Allegro Design Management Policy File](#)
- [Creating Custom-Derived Data Objects](#)

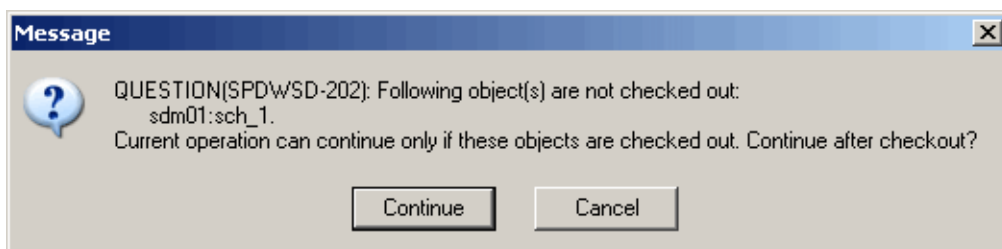
Generating Default Derived Data

To generate the derived data for a schematic or physical object, do the following:

1. Launch Allegro EDM Flow Manager.
2. Join the project that has been enabled for design management, or, if you are the integrator, open a design management-enabled project.
3. Right-click the design or subdesign for which you want to generate the derived data (for example, for *sch_1*).
4. Choose *Generate PDF* (for example) or any derived data that you need.

Ensure that Allegro Design Entry HDL does not have any unsaved changes; unsaved changes will not be captured while generating the PDF.

A message appears to indicate that you cannot generate derived data unless you check out the object.



5. Click *Continue*.

The derived data, *LogicalPDF*, appears with an icon next to it to indicate that it is a new object.

	Name	Local Version	Shared Version
*	*	*	*
Logical Hierarchy			
	sdm01	3.0.modified	3.0
	sch_1	2.0	2.0
*	LogicalPDF	New	
	packaged	1.0	1.0
	physical	2.0	2.0

6. Check in the design.

This ensures that derived data is also checked in.

Important

Whenever you modify the source object, the derived data should be regenerated.

Derived Data Controls in the Allegro Design Management Policy File

The Allegro Design Management policy file, `sdm_policy.xml`, is located at:
<SPB_installation_directory>\share\cdssetup\sdm\policies

Allegro X EDM Design Management User Guide

Adding Derived Data to Allegro Design Management

To customize this file, copy the file from the installation directory to the site area:
<Allegro_EDM_conf_root>\<company>\<site>\cdssetup\sdm\policies

You define derived data for a particular type of design object. The following example describes how the policy file has been customized to generate a PDF file as the derived data.

Example of a default derived data object: PDF File

An example of derived data defined for the `Schematic` design object is a PDF file. This derived data is defined under the following entry:

```
<designObject category="Logical" label="Schematic" name="sch_1"
type="ADWSchematic">
```

For the `Schematic` design object, the following is to be specified:

- Definition of the derived data
- Command to run to generate the derived data

Definition

The following entries under the `Schematic` design object constitute the definition of derived data:

```
<derivedObject label="PDF" name="LogicalPDF"
type="ADWDerivedDataLogicalPDF">

  <attachment>

    <includes>

      <include path="{projdir}/{design_lib}/{design_name}/
        derived_data/sch.pdf"/>

    </includes>

  </attachment>

  <tools>

    <tool args="{projdir}/{design_lib}/{design_name}/
      derived_data/sch.pdf" label="Open" name="AcroRd32"/>

  </tools>
```

</derivedObject>

where:

- `label="PDF"`: This indicates the entry you see in the *Type* column of the Allegro Design Management dashboard, which is *PDF*
- `name="LogicalPDF"`: This indicates the name of the derived data object that you will see in the Allegro Design Management dashboard, which is *LogicalPDF*
- `type="ADWDerivedDataLogicalPDF"`: Each object has a type; the type for a derived data object starts with `ADWDerivedData`.
- `includes` section: A design object is associated with some files. In case of the PDF derived data, there is only one file whose path is defined in the `includes` section.
- `tools` section: This section defines the context menu option you see when you right-click the derived data object.

Command

You need to specify the command that generates the required derived data. The command is specified in the source object (`sch_1` in this example). To generate a PDF, the command that needs to run on the design object is:

```
publishpdf -proj <projpath> -cell <cell> -view sch_1 -all -file  
<outfilepath>
```

Note: Ensure that the command is available in the PATH.

This command is specified in the `tools` section as follows:

<tools>

```
<command args="-proj,${projdir}/${cpm}, -cell,${design_name}, -  
view,${sch_view}, -all, -file,${projdir}/${design_lib}/  
${design_name}/derived_data/sch.pdf" label="Generate PDF"  
name="publishpdf" type="ADWDerivedDataLogicalPDF"/>
```

</tools>

where:

- `label="Generate PDF"`: Defines the context menu option that appears when you right-click the schematic object to generate the derived data

Allegro X EDM Design Management User Guide

Adding Derived Data to Allegro Design Management

- `name="publishpdf"` : Defines the command that runs when you select this menu option

- `type="ADWDerivedDataLogicalPDF"` : Each object has a type; the type for a derived data object starts with `ADWDerivedData`. Thus, the syntax for `type` is as follows:

`ADWDerivedData<user_defined_derived_data_name>`

- `args`: Defines the arguments for this command. The arguments are separated using a comma.

For detailed information, see [Tools Section of the Policy File](#).

Both, the definition and tool sections, use certain Cadence-defined variables and environment variables. Here is a list of Cadence-defined variables used in the policy file:

Table 5-1 Cadence-Defined Variables for Policy File

Variable Name	Description
<code>projdir</code>	Project directory path
<code>cpm</code>	Name of the project cpm file
<code>sch_view</code>	Name of the schematic object
<code>design_lib</code>	Name of the library containing the blocks
<code>design_name</code>	Name of the design or subdesign
<code>cdslib</code>	Name of <code>cds.lib</code>
<code>refcdslib</code>	Name of <code>ref_cds.lib</code>
<code>atdm</code>	The <code>atdmdir</code> directory as defined by the <code>ADWCONFIGDIR</code> directive in the CPM file under the <code>ADW</code> section
<code>cachedir</code>	Refers to the flatlib directory
<code>history</code>	Refers to the history folder as defined using the <code>history_folder</code> directive in the CPM file under the <code>CACHE</code> section
<code>callout_file</code>	Refers to the callout file as defined using the <code>last_callout_file</code> directive in the CPM file under the <code>BOMHDL</code> section
<code>packager_view</code>	Refers to the packaged object

Allegro X EDM Design Management User Guide

Adding Derived Data to Allegro Design Management

Variable Name	Description
<code>physical_view</code>	Refers to the physical object
<code>root_design</code>	Refers to the root-level design object of the project
<code>master_file</code>	Name of the master file as defined in the <code>master.tag</code> of the object. For example, if <code>test.brd</code> is the master file then <code>test</code> is the name of the master file.
<code>master_file_extension</code>	Extension of the master file as defined in the <code>master.tag</code> of the object. For example, if <code>test.brd</code> is the master file, then <code>.brd</code> is the extension name.

To use these environment variables in the policy file, use the following syntax:

```
${<variable_name>}
```

For example: `${HOME}`

To use custom variables in the policy file, use the following syntax:

```
$cpm{section/directive}-artwork
```

```
$cpm{<variable_name>}
```

For example:

```
$cpm{customvar/pdf_data}
```

Tools Section of the Policy File

This section defines the various commands and tools you can launch for a design object. It has two types of elements:

- Command
- Tool

Command

These are non-GUI applications that do not require any user input (for example, a script that generates derived data for an object.)

The syntax of this element is as follows:

Allegro X EDM Design Management User Guide

Adding Derived Data to Allegro Design Management

```
<command args="command_arguments" label="display_name_to_launch
command" name="name_of_executable_file"
type="type_of_design_object"/>
```

where:

- **args:** Defines the arguments of the command. The arguments are separated by commas.
- **label:** Defines the context menu option that appears when you right-click the object.
- **name:** Defines the command that runs on the object.
- **type:** Is defined only for derived data. Each object has a type; the type for a derived data object starts with `ADWDerivedData`. Thus, the syntax for `type` is:

```
ADWDerivedData<user_defined_derived_data_name>
```

The command element can be extended to include a condition that is validated before running the command. For example, the following condition defines that the command element is run only for the root-level design:

```
cond="${design_name},==,${root_design}"
```

Only two operators are supported for defining a condition: `==` and `!=`

An example where a command contains a condition:

```
<command args="-proj,${projdir}/${cpm},-nographic,-f,SS, -
delim,${COMMA}, -o,${projdir}/${design_lib}/${design_name}/
derived_data/logical.bom" cond="${design_name},==,${root_design}"
label="Generate BOM" name="bomhdl" type="ADWDerivedDataLogicalBom"/>
```

Tool

The tool element defines the GUI applications that you want to launch for an object.

The syntax is as follows:

```
<tool args="command_arguments" label="display_name_to_launch
command" name="name_of_executable_file"/>
```

The syntax of tool and command is the same. However, tool does not support type and condition.

For example:

Allegro X EDM Design Management User Guide

Adding Derived Data to Allegro Design Management

- `<tool args="-proj,{projdir}/{cpm}" label="Open design" name="concepthdl"/>`

This command opens a project in Design Entry HDL.

- `<tool args="$action{file_select}" label="Open" name="notepad"/>`

This command opens all the files selected in Notepad.

Creating Custom-Derived Data Objects

Allegro Design Management contains default derived data objects with fixed names. These are LogicalBOM, PhysicalBOM, VariantBOM, and LogicalPDF. As a site administrator, you might require additional derived data objects, especially for board files with names that you define. The names can be user defined or controlled through some variables. For example, your setup might have the same project number associated with manufacturing data.

This section shows you how to

- use project cpm variables or directives to identify the derived data name so that they are easier to search.

You can use any of the variables used in the CPM, or use custom variables.

- Add a suffix or prefix the derived data with the derived data type. For example, `$cpm{section/directive}-artwork`
- Include multiple files in a derived data object
- Exclude specific files from a location from the derived data object

Tasks Overview

The main tasks in setting up custom derived data objects are as follows:

Task	Done by
1. Customize the policy file for adding new derived data objects.	Allegro EDM Administrator
2. Add custom variables to the cpm from Design Entry HDL.	Integrator
3. Open the project and enable design management.	
4. Generate derived objects - ensure that the location is the same as that specified in the policy file; otherwise, it will not be recognized	Designer/owner
5. Check in the new objects.	

Example: Adding Custom Derived Data Object

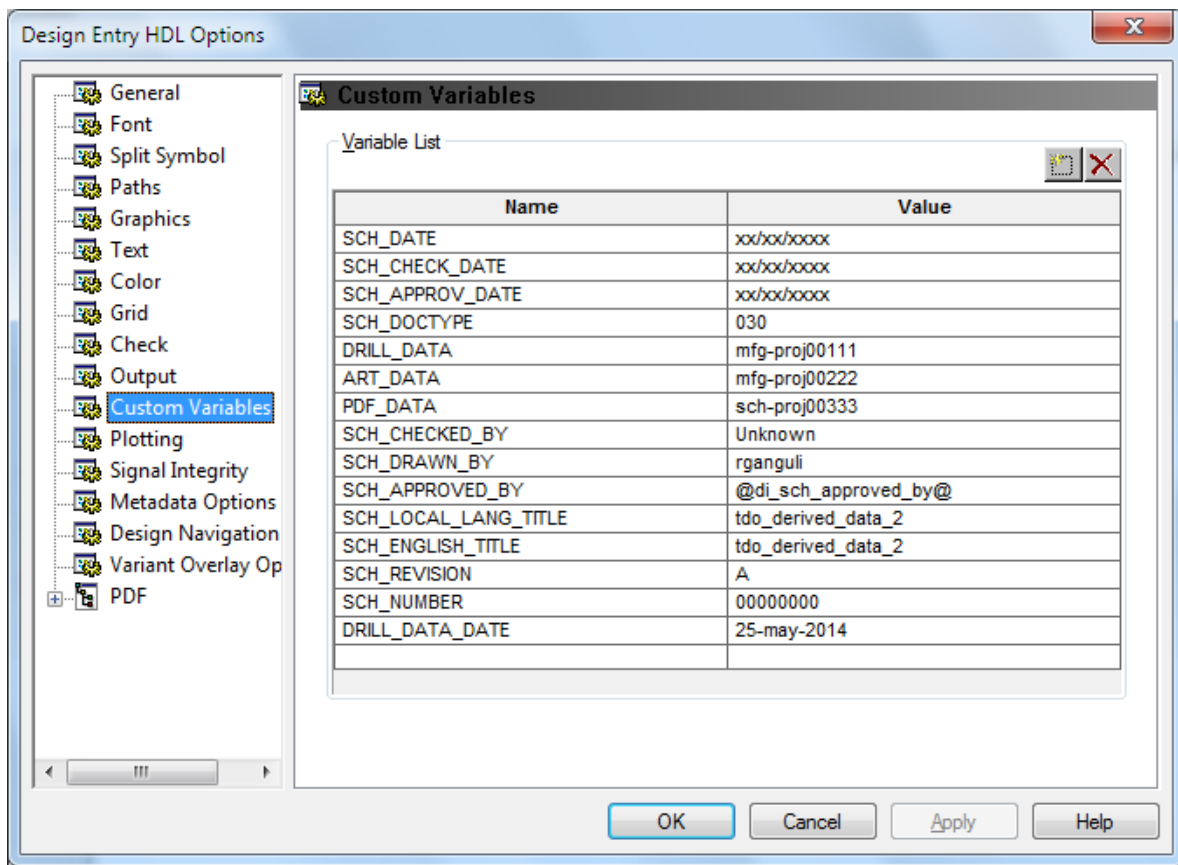
In this section, artwork for a board will be added as a custom derived data object to the Allegro Design Management dashboard. The tasks are divided based on who will be performing them.

Allegro EDM Administrator Tasks

These tasks require Allegro EDM administrator permissions.

1. Identify the CPM variables to use in the derived data.

For example, custom variables.



The CPM file has a section for custom variables. For example:

```
START_CUSTOMVAR
```

```
drill_data 'mfg-proj00111'
```

Allegro X EDM Design Management User Guide

Adding Derived Data to Allegro Design Management

```
art_data 'mfg-proj00222'
pdf_data 'sch-proj00333'
END_CUSTOMVAR
```

Although custom variables are the most commonly-used CPM variables for this task, you can use any section in the project file.

2. Identify the files to include in the derived data.

```
D:\projects\edm_derived_data\worklib\edm_derived_data\
physical\be_data\*.art
```

3. Create a site-level policy file, if needed.

As a starting point, you can copy the *sdm* folder from:

```
<SPB_installation_directory>\share\cdssetup
```

to:

```
<Allegro_EDM_conf_root>\<company>\<site>\cdssetup
```

4. Open the policy file in a text editor:

```
<Allegro_EDM_conf_root>\<company>\<site>\cdssetup\sdm\policies\
sdm_policy.xml
```

5. Add an entry in the policy file for a new derived data object.

a. Label, Name, and type of the derived object

b. File(s) to include in the derived data.

c. File(s) to exclude in the derived data

You need to specify the location and the file names. You can use variables and wildcards for the files to include as well as exclude.

d. Application to open the derived data object in

```
<derivedObject label="Art" name="art-$cpm{customvar/art_data}"
type="ADWDerivedDataArt">
```

```
<attachment>
```

```
<includes>
```

```
<include path="${projdir}/${design_lib}/${design_name}/
${physical_view}/be_data/${master_file}/*.art"/>
```

```
</includes>

<excludes>

    <exclude path="${projdir}/${design_lib}/${design_name}/
    ${physical_view}/be_data/${master_file}/*.txt"/>

</excludes>

</attachment>

<tools>

    <tool args="$action{file_select}" label="Open"
    name="notepad"/>

</tools>

</derivedObject>
```

6. Save the policy file.

In case you are creating a new policy file, you should also change the policy label. This will help when you are selecting the policy file in the Enable Design Management command. For example, you can set

```
<policy label="New_Policy" schemaVersion="2.2">
```

Integrator Tasks

These tasks are done by the logical/physical integrator.

1. Open the project in Allegro Design Management.
2. Enable Design Management.

Designer Tasks

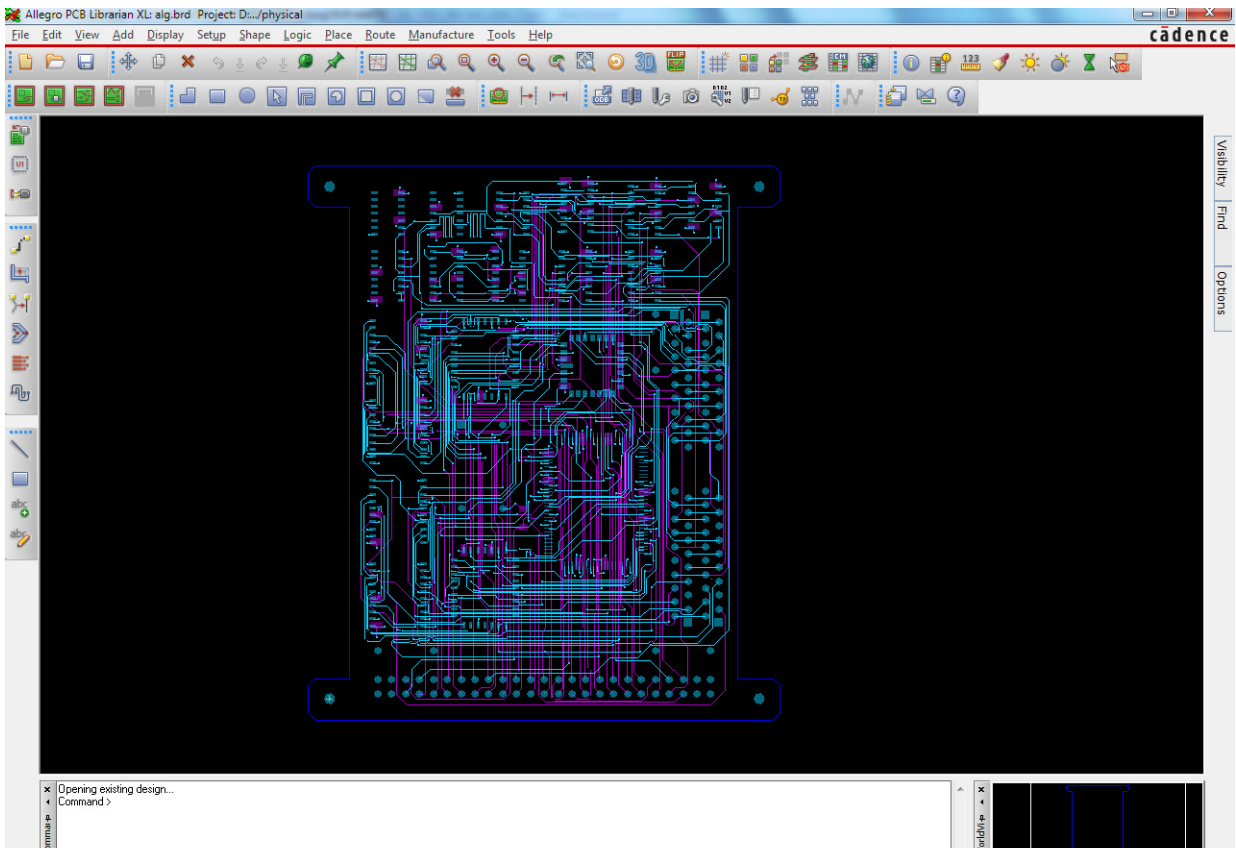
You can now check out the board for which you require derived data. For example, `alg.brd`

1. Right-click the board and choose *Check Out*.
2. Right-click the board and choose *Open*.
3. Choose the license.

Allegro X EDM Design Management User Guide

Adding Derived Data to Allegro Design Management

The board opens in PCB Editor.



4. Change the default output folder for the manufacturing information to match the location of the derived data specified in the policy file.

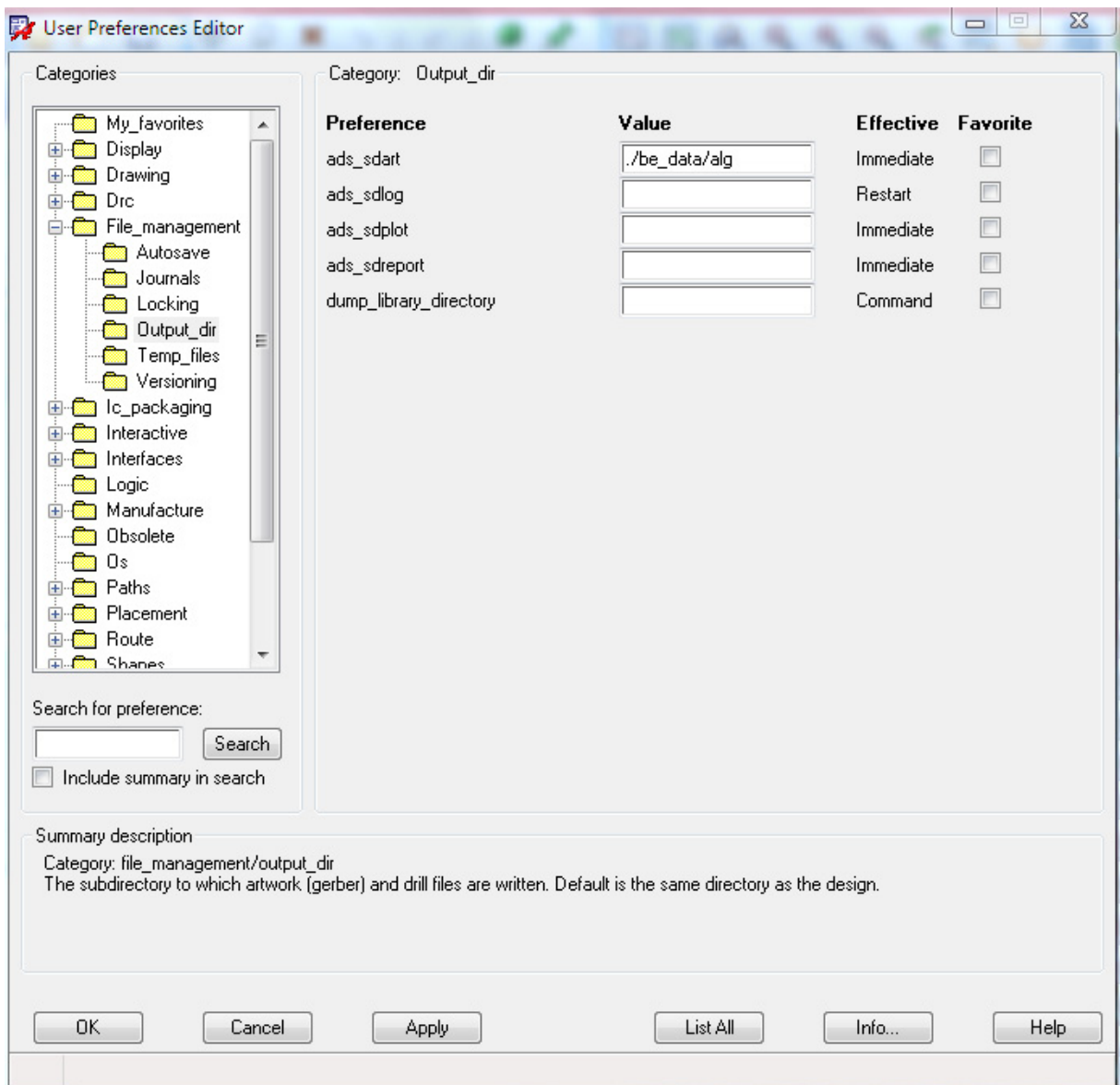
Ensure this location is the same as the one specified by the administrator in the policy file (refer to [step b](#)).

- a. Choose *Setup — User Preferences*.
- b. Open *File_management — Output_dir*.

Allegro X EDM Design Management User Guide

Adding Derived Data to Allegro Design Management

- c. Specify the location for the *ads_sdart* field.

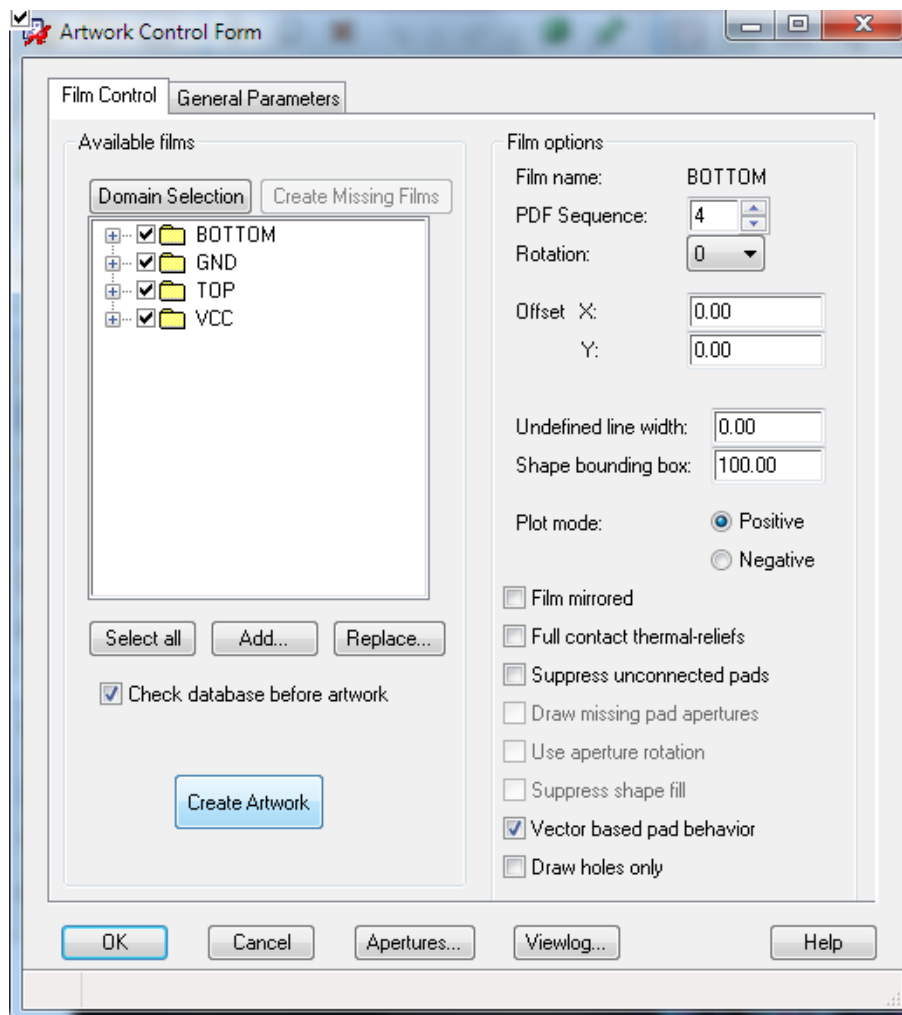


- d. Click *OK*.

You can now generate the artwork.

5. Choose *Manufacture — Artwork*.
6. Choose the layers.

7. Click *Create Artwork*.









8. Click *OK*.

Allegro X EDM Design Management User Guide

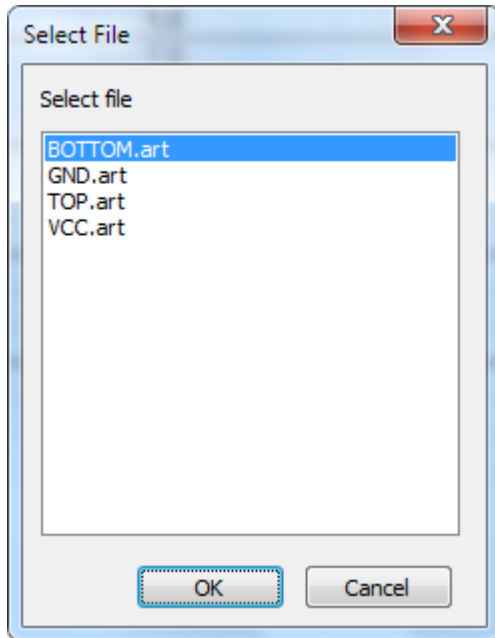
Adding Derived Data to Allegro Design Management

The Allegro Design Management dashboard is updated with the newly-created artwork as derived data.

	Name	Local Version	Shared Version	Checked Out To	Owners	Check In Comment
*	*	*	*	*	*	*
Working Design						
	[-] tdo_derived_data_2	2.0	2.0			
	sch_1	1.0	1.0			
	[-] physical	2.0.modified	2.0			
	[-] alg.brd	1.0.modified	1.0			
	[-] [Derived Data]					
	art-mfg-proj00222	New				
	tdo_derived_data_2.brd	1.0	1.0			
	[-] top	1.0	1.0			
	sch_1	1.0	1.0			
	sym_1	1.0	1.0			
	[-] low	1.0	1.0			
	sch_1	1.0	1.0			
	sym_1	1.0	1.0			
	[-] mid	1.0	1.0			
	sch_1	1.0	1.0			
	sym_1	1.0	1.0			
Globals						
	project_settings	2.0	2.0			
	ecsets	1.0	1.0			

You can right-click and open the derived data object.

9. Select which of the layer's artwork will be opened in the specified application.



10. Check in the derived data object.

The artwork generated for the board is now available to all the other members of the project.

Making Policy File Changes to a Managed Project

Note that in the previous example, the policy file was first modified to include new derived data objects, and then the project was enabled for team design. In situations where custom derived data is required for a project that is already design management-enabled, you need to modify the policy file:

1. Check out *Globals — Project Settings*.
2. Make the required changes.
3. Check in the policy file.

Updating Object Definitions in a Project

The policy file defines the objects managed by Allegro Design Management. At times, the administrator makes changes to the policy file at the site-level and a project in progress might need those changes. To get the object definition changes, the project integrator can refresh the policy file.

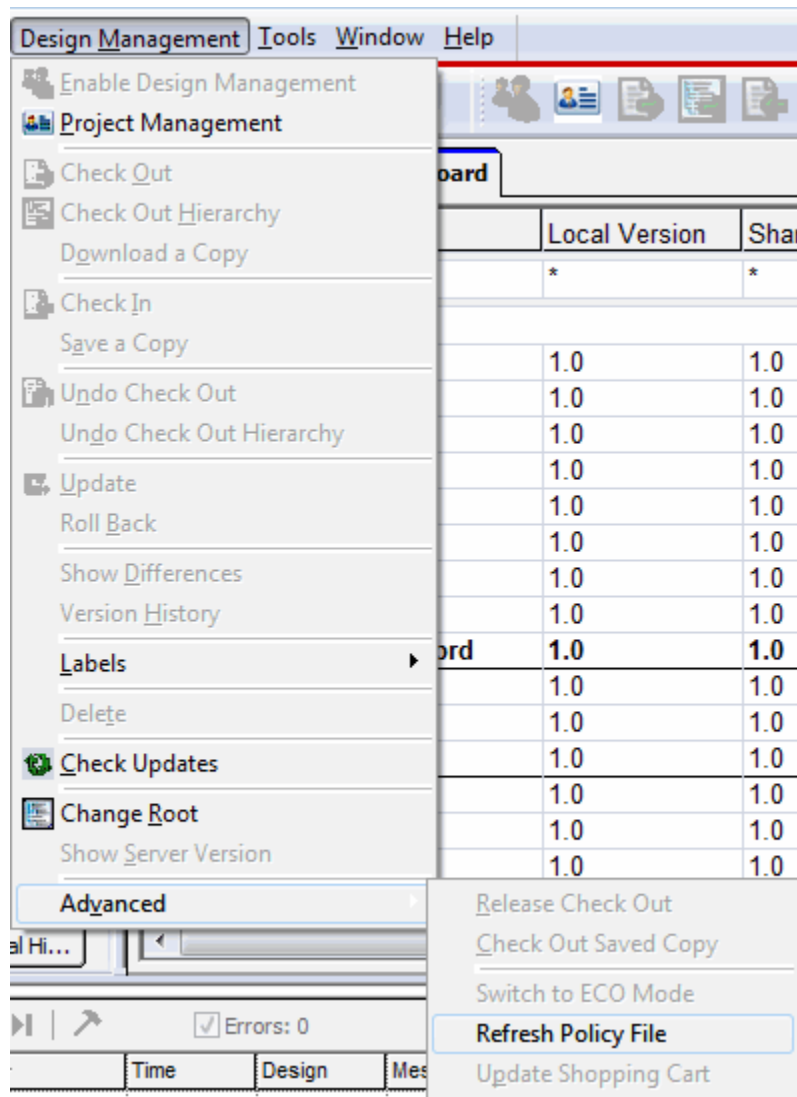
Important

Refreshing a policy file fetches ***all*** the changes from the site level. In case your project has object definitions that are not there in the site-level policy file, do not use the *Refresh Policy File* option. Doing so will overwrite the local policy file and objects defined based on the older policy file may no longer work.

Allegro X EDM Design Management User Guide

Adding Derived Data to Allegro Design Management

To refresh the policy file, choose *Design Management — Advanced — Refresh Policy File*.



Allegro X EDM Design Management User Guide

Adding Derived Data to Allegro Design Management

Cache-Enabled Projects

For cache-enabled project types, such as board and highspeed, Allegro Design Management manages additional, internal global objects—*shopping_cart* and *design_cache*—using Library Revision Manager (LRM). Allegro Design Management operates in two types of modes: Work in Progress (WIP) and ECO (ready for release).

Additional Internal Global Objects

The *shopping_cart* and *design_cache* objects are additional internal global objects in the team design flow.

The *design_cache* object is created when the integrator switches the design mode to ECO. You can then synchronize the work area with this new *design_cache* object.

The integrator may need to explicitly check out and check in these objects, in the following cases:

- When you want to create or update your shopping cart without adding components to the design.
- When the design is in the ECO mode and there is a new version of a component available in the Allegro EDM reference library. To get the latest version, launch LRM from Flow Manager.

Allegro EDM Design Management Modes

An Allegro EDM design, after being enabled for design management, can function in two modes:

- Work in Progress (WIP)

In this mode, designers create or modify their assigned designs/blocks, and designs change rapidly.

At this stage, components are used from the Allegro EDM reference library. Any changes in the version of these components in the Allegro EDM reference library need to be incorporated in the design. This ensures that the version of the components used in the design is always synchronized with the reference library.

If there is a mismatch in the component version used in the design and the one in the Allegro EDM reference library, the mismatch is reported by LRM during schematic check in. Use LRM to resolve this mismatch by getting the latest component version from the Allegro EDM reference library.

■ **ECO Mode (ready for release mode)**

After a designer completes all the required modifications, and the design is ready to be released, the integrator changes the mode from WIP to ECO. At this point, a new object, *design_cache*, is created in the shared area. The *design_cache* object:

- ❑ contains the versions of components used in the design
- ❑ isolates the design from component version changes in the Allegro EDM reference library

The integrator needs to decide which changes and whether or not to take from the Allegro EDM reference library.

Switching to ECO can be done only after you have checked in all the objects. The integrator is responsible for resolving all version/component conflicts with the Allegro EDM reference library, if there are any.

Impact of Design Mode on Design Management Operations

Based on the design mode, you will see the following impact on design management operations when working with cache-enabled projects such as board or highspeed:

Join Allegro EDM Project

When a designer joins an Allegro EDM project, the following occurs:

- In the WIP mode, a copy of the components used in the design is created from the Allegro EDM reference library.
- In the ECO mode, a copy of the components used in the design is created from the shared area.

Schematic Check Out

There is no change. The behavior is the same as in non-Allegro EDM-Design Management.

Schematic Check in

When a designer checks in any schematic object, global objects such as *shopping_cart* and *project_settings* may also be revised if they have changed.

For example, a designer checks out a schematic object and adds a few symbols that are not already used in the design. This will result in a change of the *shopping_cart* object. This may also result in the use of a new library in the design. In such cases, these global objects are marked as *In Use*. When the designer checks in the schematic object, both the global objects can also be checked in.

Note: When Allegro EDM itself modifies an object without first checking out the object, EDM marks the status as *In Use*. When a user modifies an object without first checking it out, Allegro EDM marks the status as *Hijack*.

Handling Reference Library Updates in Allegro EDM-Allegro Design Management Flow

While a designer is working on a design, the reference library may change. In such cases, the designer may have to update symbols or components. When a schematic is checked in, the LRM rule is run to ensure consistency between the local design and the Allegro EDM reference library.

- In the WIP mode, if a designer has used an old version of a symbol in the design and the latest version is available in the reference library, LRM is launched during the check in. The designer needs to resolve the conflicts between the two versions by running the update operation. Selecting *Cancel* will cause the check-in operation to fail.
- In the ECO mode, LRM runs between the designer's local cache and the shared area cache. In this case, LRM will not be launched and the integrator will have to resolve the conflicts by running the standalone version of LRM and then perform the check-in operation.

Show Difference and Update

When a schematic object has been updated, the designer can run the Show Difference option to see the differences between the previous and current version of the object. On running Show Difference, the LRM window comes up in read-only mode and displays the additional

symbols or parts that are now in the new version of the block. It also highlights conflicts, if any, in the symbol version used in the design and the ones available in the reference library.

Running Update fetches the latest design from the shared area. If the design is in the WIP mode, and you have saved the design but then modified it again, you may want to want to discard your changes and revert to the draft copy of the design. Use the Rollback menu option in such cases. If not, the symbols in the design are updated from the shared area.

Note: Allegro Design Management does not monitor reference library changes. Therefore, if there are components used in design that are out of sync with the reference library, Allegro Design Management will show their State as checked in. If required, you need to run LRM outside Allegro Design Management and have the block updated with the latest library changes.

Reuse Blocks in Cache-Enabled Projects

Depending on requirements, designers can reuse blocks in different ways.

- As the designer, you simply want to use a block that exists in another project and do not intend to modify it

In this case, you can reference the block in your project. To reference the block, import it as a read-only block. With this, you have the advantage of being informed of changes to the source block and can re-import the block if needed. For details, refer to the *Re-importing Read-only Blocks* section of *Allegro Design Entry HDL User Guide*.

Note: When you work with blocks in cache-enabled projects, you must import the blocks into the destination project. This ensures that all the cells in the block library are cached.

- As the designer, you want to reuse an existing block and edit it based on your requirements

For this, you can import the block as a writable block into the Allegro EDM project using Design Entry HDL. For details on importing designs, refer to *Allegro Design Entry HDL User Guide*.

- As the designer, you want to edit a block and have it in your reference library so that you are aware of any changes to the block.

In this case, designers can submit the block that they want to reuse to the librarian who can release it. For details about the reuse block flow, see *What is the Block Flow* in *Allegro EDM Administrator Reference Guide*.

The advantage with the block being in the reference library is that if the block is modified in the library, LRM prompts designers to update the block if it is in their projects.

Allegro Design Management User Interface

This section describes the Allegro Design Management interface:

- [Dashboard Window](#)
- [Allegro Design Management Start Page](#)

Dashboard Window

The dashboard window shows the entire design and the state of all subdesigns, their status, and ownership. It contains the following sections:

Hierarchy Tree

- *Logical Hierarchy* displays the schematic design in a hierarchical form. It shows all the blocks along with their schematic objects. By default, an LDI or users assigned to logical objects will see this hierarchy.
- *Physical Hierarchy* displays the physical design in a hierarchical form. It shows all the blocks and their physical objects such as packages, physical blocks, and partitions. By default, a PDI or users assigned to physical objects will see this hierarchy.

Dashboard

The *Dashboard* displays the following:

- Hierarchy Type: This can be the following:
 - *Working Design* hierarchy (containing logical and physical objects)
 - *Logical Hierarchy*
 - *Physical Hierarchy*

- Details of the design in the hierarchy, such as:
 - ❑ Names of subdesigns, pages, and global objects
 - ❑ Current version number of each object in the local work area
 - ❑ Version number of each subdesign and page in the shared area
 - ❑ ID of the designer who has checked out the subdesign or page
 - ❑ Designers assigned to the design
 - ❑ Comments, if added, for the checked-in subdesigns and pages

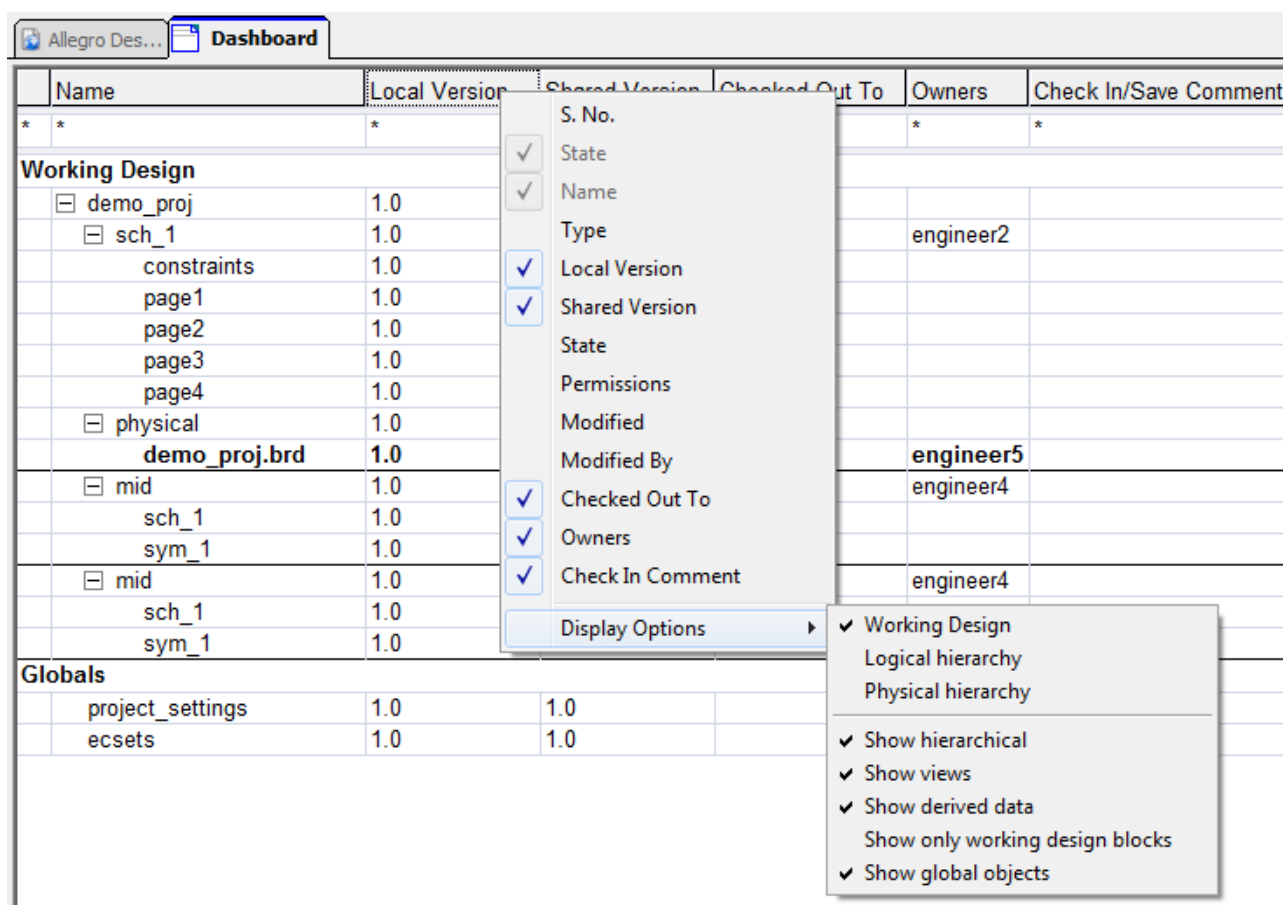
Dashboard Customization

When you launch Allegro Design Management, the *Dashboard* automatically displays a particular view depending on the role of the user who has logged in.

Allegro X EDM Design Management User Guide

Allegro Design Management User Interface

You can right-click the dashboard header to see various options that you can use to customize the dashboard view. The following figure shows these options.



Note: The view options — *Working Design*, *Logical hierarchy*, and *Physical hierarchy* — you choose in the Allegro Design Management *Dashboard* are only valid for the current session.

If you want to preserve these settings between sessions, set any one of the following directives in the `START_SDM...END_SDM` section of the site or project `.cpm` file:

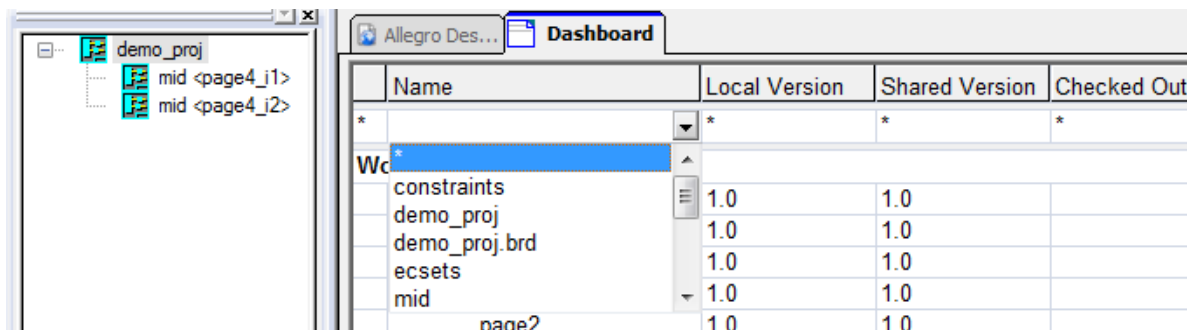
- ☐ `dashboard_show_logical_hierarchy`
- ☐ `dashboard_show_physical_hierarchy`
- ☐ `dashboard_show_working_design`

Valid values for these directives are 'Show'|'Hide' or 'Yes'|'No'.

Allegro X EDM Design Management User Guide

Allegro Design Management User Interface

- You can see information related to specific design objects by using dynamic filtering below the dashboard header as shown in the following figure:



- To see the complete name of the designer who has been assigned to or has checked out the subdesign, do the following:
 - a. Choose *Project — Settings*.
 - b. In the *General* group box, select the *Show user name instead of id* option.
 - c. Click *OK*.

Information Pane

This pane, at the bottom of the dashboard window, contains the following three tabs:

- *Session Log*
- *Violations*
- *Command Window*

The first two tabs show the messages and the command window is used for entering commands. You can use the *Resolve* button in the *Violations* tab to correct any reported violations.

Allegro Design Management Start Page

The Allegro Design Management Start Page appears when you do one of the following:

- Launch Allegro Design Management.
- Click the tab for the start page.
- Choose *Help - Display Start Page*.

From the start page, you

- Open an existing project
- Join an existing project

Open an Existing Project

To open an existing project, do any of the following:

1. Click *Open an Existing Project* in the start page.
2. Choose *File - Open Project*.

The Open dialog box appears.

3. Select the project file (.cpm) and click *Open*.

The project is opened in Allegro Design Management.

Join an Existing Project

See [Joining a Design Project](#) for the procedure.

Menu Bar

The menu bar in Allegro Design Management has the following menus.

- File
- View
- Project
- Design Management
- Tools
- Window
- Help

Session Log Tab

The Session Log window displays the details of the current session. These details include the information about opened design files and packaged designs.

To open the Session Log window, do one of the following:

- Click the *Session Log* tab.
- Choose *View — Session Log*.
- Press *Ctrl + Alt + L*.

Violations Tab

The Violations window displays error, warning, and information messages. To highlight the object causing the error, warning or informational message, double-click on the row for the message.

To open the Violations window, do one of the following:

- Click the *Violations* tab.
- Choose *View — Violations*.
- Press *Ctrl + Alt + V*.

Command Window Tab

The Command Window tab is a console to enter TCL Shell commands.

To open the Command Window, do one of the following:

- Click the *Command Window* tab.
- Choose *View — TCL Shell*.