cadence®

Allegro® X User Guide

Product Version 23.1
September 2023

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

Contents

Allegro User Guide

The Allegro® Layout Editors Information Set

The information set for Cadence® Allegro® layout editor information set consists of fourteen books. These books explain features of the layout editors, Allegro® PCB Editor and Allegro® Package Designer Plus (APD Plus). If a feature is available in only one of the layout editors, either a note is provided mentioning the tool or the title of the topic is marked as such.

These books are available in *Cadence Help* under *Allegro User Guide.* All documentation is accessible from the *Help – Documentation* menu in the layout editors.

The following table lists book names and their descriptions. The order of books two through nine correspond to a typical PCB design flow.

Manual Name	Description
Getting Started with Physical Design	Describes the user interface of the layout editors. It also contains information on generic functions, setup and configuration information, and on the front-to-back flows.
Defining and Developing Libraries	Describes how to create libraries – a collection of graphic symbols – representing packages, mechanical elements, drawing formats, and custom pads and padstacks.
Transferring Logic Design Data	Describes how to transfer native and third-party design logic data to the back-end layout editors such as PCB Editor and Allegro Package Design Plus(APD Plus); and how to backannotate that data. Native logic transfer refers to the data derived from the Cadence schematic design capture tools, such as Allegro System Capture, Allegro Design Entry HDL, System Connectivity Manager, or Allegro PCB Design CIS.

Preparing the Layout	Describes the tasks involved in the preparation of a layout, before placing components in the design: ■ Defining the layout cross-section ■ Adding graphic elements ■ Editing layout padstacks and pad shapes ■ Creating interactive blind and buried vias ■ Creating and editing etch/conductor shapes
Creating Design Rules	Describes how to create and modify design rules for a design. The topics included are: ■ Design Rule Checking (DRC) ■ Properties ■ Constraints ■ Defining the layout cross-section
Placing the Elements	Describes how to place elements manually and automatically, and how to swap pins, functions, and components after placement.
Routing the Design	Describes how to perform basic and automatic routing in the physical designs and post-routing tasks. Topics included are: ■ Interactive routing ■ Automatic routing with Allegro PCB Router ■ Glossing to improve the appearance and manufacturability of a physical design.

Completing the Design	Describes tasks performed before sending a design out for fabrication: ■ Renaming reference designators ■ Running audits ■ Extracting information from your design ■ Generating coupons
Preparing Manufacturing Data	Describes the processes to create manufacturing data and the output files generated as a result of these processes: ■ Numerically controlled (NC) drills and routers ■ Silkscreen ■ Penplotting ■ Artwork (photocopying)
SKILL Reference	Describes the AXL (Allegro eXtension Language) use model, how to start AXL, and how to use each AXL function.
Working with RF PCB	Describes a unified design solution for complex mixed-signal projects. From the schematic, to layout, to manufacturing, a total front-to-back design flow that helps in streamlining entire RF design process.
High Density Interconnect (HDI)	Describes the key aspects of HDI requirements and the features supporting them, such as microvias, DRCs for same net and net-net conditions, unused inner-layer pad removal, rules for via tangency (vias touch but do not overlap) and stacking (coincident location of the adjacent layer vias), and dynamic filleting.

Allegro Timing Environment	Describes methodologies for solving timing relationships (differential phase, match groups and relative match groups) and other delay related constraints using the ATE (Allegro Timing Environment). This environment contains technology focused on visualizing and solving the delay issues.
Getting Started with Symphony Team Design	Describes team design methodologies for working in a concurrent and collaborative environment.
Allegro Layout Editor Environment Variables Reference	Describes user preference environment variables.
Allegro Layout Editor Design Parameters Reference	Describes layout editor design parameters.