cadence™

Allegro®
PCB Editor User Guide: Creating Design Rules

Series XL and GXL

Product Version 16.2
November 2008

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

Contents

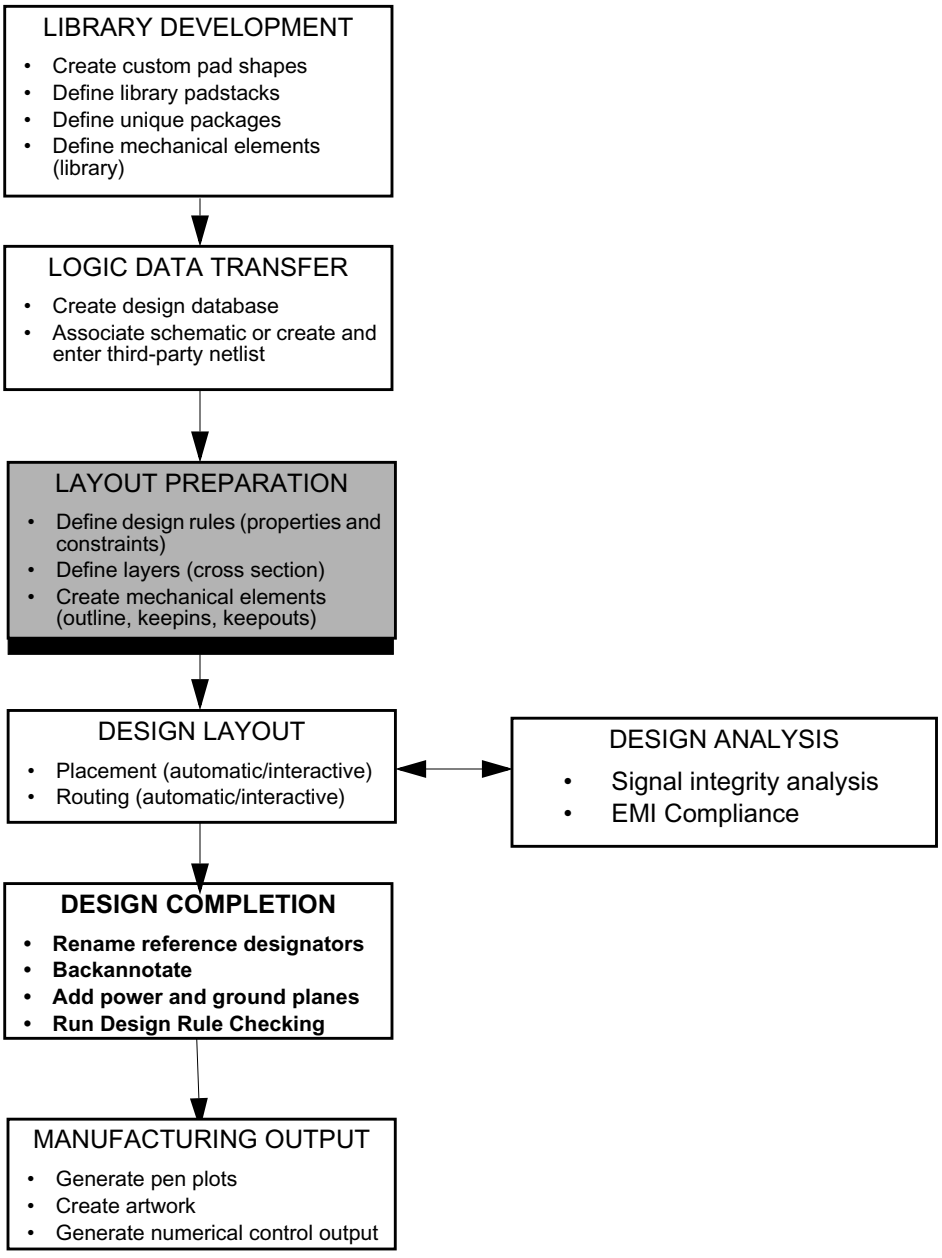
Preface

This document describes how to create design rules and prepare for creating a PCB design. These topics are included:

- Establishing and modifying the rules that govern how Allegro PCB Editor operates on design elements, specifically:
  - Design Rule Checking (DRC)
  - Properties
  - Constraints
- Defining the layout cross-section

You should set up design rules as part of your preparation for layout. The following figure illustrates where you would normally incorporate design rules in the design flow.

Design Rules in a Design Flow



1

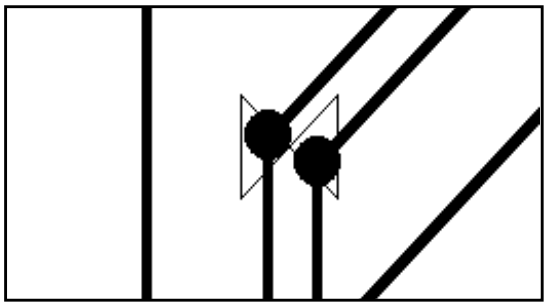
About Design Rule Checking

As part of preparation for layout, you should set up design rules. Allegro PCB Editor performs design rule checking with Design Rule Check (DRC) to ensure that the design conforms to specified properties and constraints you attach to individual design elements or assign globally to the entire design. (Properties are described in Chapter 2, “Working with Properties,” and constraints in Chapter 3, “Working with Constraints.”) Design rule checking identifies violations of physical design rules whenever you add an element or make any change to the design.

You can check for violations in real-time as you design (called online DRC error), or in batch mode (called batch DRC error). You may prefer the instant feedback of online DRC, at the expense of system performance, or you may prefer to use batch mode to improve system performance and decide to resolve violations later in the design process.

When Allegro PCB Editor detects a design rule violation, the offending design element is flagged with the appropriate design rule violation marker (bow tie), shown below.

Figure 1-1 Sample DRC Marker



DRC Modes

You control when DRC applies to each constraint by applying one of the following DRC modes:

On (or Always)	Run DRC for the constraint during interactive commands. This setting is for those checks you want to run while you modify design elements. The more constraints you check interactively, the slower your system performs during editing. Allegro PCB Editor checks the constraint when you set DRC to run online in the Status dialog box or when you choose *Tools – Update DRC* (`drc update` command). The dialog box, menu item, and command are detailed in the *Allegro PCB and Package Physical Layout Command Reference*.
Off (or Never)	Do not perform DRC for the constraint. Use this setting for checks that your design process does not require. This enhances interactive performance. **Note:** Positive shapes (static or dynamic) void to a Cadence default of `13 mil` (or design unit equivalent) when the DRC mode is set to *Never*.

You can also globally disable online DRC in the Status dialog box or in the Electrical Modes tab of the Analysis Modes dialog box, available by choosing *Analyze – Analysis Modes* in Constraint Manager.

The DRC mode settings you choose depend on the design complexity, CPU performance, and available memory. In general, Cadence suggests that you use online DRC for these checks:

- All spacing, physical, and global package checks
- Electrical
  - Stub length
  - Net schedule
  - Max via count
  - Max exposed length
  - Propagation delay

A DRC mode cannot be different for specific constraint areas nor for a particular constraint in a constraint set. It can be different for various ETCH subclasses. A single setting of the DRC mode applies to all instances of a constraint, such as *Line to Line Spacing* on an ETCH subclass.

DRC Status

DRC checks may become out of date when you change a constraint or property value. Allegro PCB Editor displays the status of the last DRC check in the Status dialog box, accessed by choosing *Display – Status (*`status` command).

DRC status can be:

Up To Date	All DRC checks (*On*) have been done and there have been no changes to any constraints or properties since the last full DRC update.
Out Of Date	One or more constraints set to the *On* DRC mode has been changed, properties have been changed, the DRC process was cancelled by a user command (`Control-C`, `Esc`, or the *Stop* button), or online DRC has been turned off globally in the the Status dialog box or in the Electrical Modes tab of the Analysis Modes dialog box, available by choosing *Analyze – Analysis Modes* in Constraint Manager.

Making DRC Errors Visible

Before running design rule checking, ensure that any DRC violations are visible.

1. Choose *Display – Color/Visibility* (`color192` command).

   The Color dialog box appears.
2. Choose *Stack-Up*.
3. Check that the *DRC* box is chosen for *All* (all layers).
4. Click *OK*.

Running Online DRC

Online DRC provides immediate feedback when you violate a design rule.

1. Choose *Display – Status (*`status` command).

   The Status dialog box appears.
2. Click the *On-Line DRC* button.
3. Click *OK*.

Running Batch DRC

Batch DRC allows you to do initial layout and then run DRC checking on the whole design at once. You can run batch design rule checking using any of the following:

- Within Allegro PCB Editor run *Tools – Database Check* (`dbdoctor` command) and enable the *Update all DRC (including batch)* option
- Launch the `dbdoctor_ui` command externally.

Note: It is recommended that you run batch DRC only for island checks and parallel and cross talk checks if the design is complex.

Updating DRC

If you turned off online DRC in the Status dialog box, you can reactivate it and run DRC simultaneously. Choose *Tools – Update DRC* (`drc update` command), described in the *Allegro PCB and Package Physical Layout Command Reference.*

Controlling the Display of DRC Markers

Allegro PCB Editor flags violations of design rules by displaying DRC marker(s), shaped like bow ties. DRC markers appear as non-filled outlines by default, as shown in Figure 1-1.

To display filled markers, do any of the following:

- Set the `display_drcfill` environment variable in the Display category of the User Preferences Editor choosing *Setup – User Preferences* (`enved` command), described in the *Allegro PCB and Package Physical Layout Command Reference.*
- Type `set display_drcfill` at the Allegro PCB Editor console window prompt.
- Enter `set display_drcfill` in your local Allegro PCB Editor environment file.

The default marker size is in user units (25 mils, for example). Change the size of the DRC marker in the in the *Display* tab of the *Design Parameter Editor*, available by choosing *Setup – Design Parameters* (prmed command).

Displaying Information About DRC Violations

DRC markers store the following information about a DRC:

- DRC class, subclass, and location
- Type of constraint set (spacing, physical, or electrical)
- Name of constraint set
- Constraint type being violated (for example, *Line to Thru Pin Spacing)*
- Data concerning first element in violation (type of element, location, refdes, if a package, and so on)
- Data concerning any second element in violation (type of element, location, refdes, if a package, and so on)

Viewing Information for a Specific DRC Marker

Before you can see information for a specific DRC marker, make sure that the visibility for the DRC layer class is turned on. For details, see Making DRC Errors Visible.

To see details about a DRC error:

1. Choose *Display – Element* (show element command).
2. In the Find Filter, make sure that *DRC errors* is checked.
3. Click on a DRC marker in the design.

   The elements associated with the DRC highlight, and the Show Element dialog box displays information about the chosen marker.

Displaying the DRC Error Report

You can display a report listing all DRC errors in the design.

1. Choose *Tools – Reports* (reports command).
2. Choose *Design Rules Check* from the list in the Reports dialog box.

3. Click *Report*.

Waiving Design Rule Check Errors

Often you may need to set aside a design rule to meet design requirements. Waive DRC allows you to flag these violations as acceptable, and attach an explanatory comment to ensure that those working with the design later on understand the rationale concerning the existence of any given DRC violation.

When you want to mark design rule violations as approved for the current design, Allegro PCB Editor lets you:

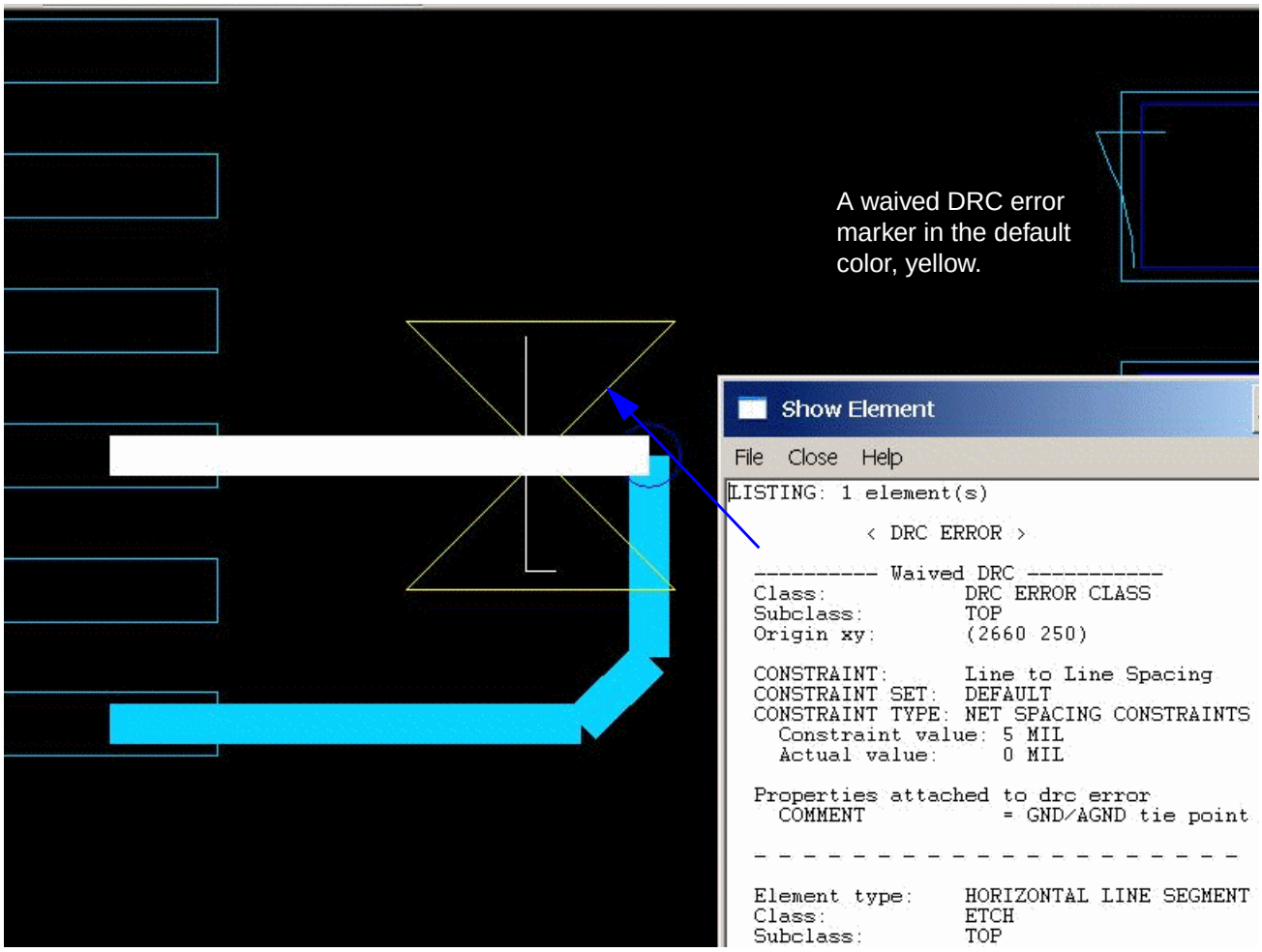
- Waive DRC errors either globally or by individual pick (`waive drc` command)
- Choose whether to display waived DRC errors in the design (`show waived drcs` and `blank waived drcs` commands)
- Assign a color to waived DRC error markers that differs from active DRC error markers in the design using *Display – Color/Visibility* (`color192` command)
- Restore waived DRCs errors to active status either globally or by individual pick (`restore waived drcs` and `restore waived drc` commands)
- Attach comments to waived DRC errors
- Generate a report that lists all waived DRC errors in the design

For more information about waiving DRC errors, see the `waive drc` command in the *Allegro PCB and Package Physical Layout Command Reference*.

Waiving a DRC Error Marker

When you waive a design rule violation, Allegro PCB Editor flags the design element with a waived design rule error marker (rotated bow tie), as shown in Figure 1-2. Waiving a DRC error updates the count of DRC errors and waived DRC errors in the *Status* tab of the Status dialog box.

Figure 1-2 Waived DRC Error Marker



Making Waived DRC Errors Visible

Important

By default, *Waived DRCs* is disabled in the *Display* tab of the *Design Parameter Editor*, available by choosing *Setup – Design Parameters* (prmed command).

Before a waived DRC error marker is visible in the design, you must enable the *Waived DRCs* check box in the Design Parameter Editor and DRC visibility in the Color dialog box. If the latter is not enabled, waived DRC errors are invisible, regardless of whether you enable the *Waived DRCs* check box in the Design Parameter Editor.

Allegro PCB Editor lets you show waived DRC errors when you:

- Enable the *Waived DRCs* check box in the Design Parameter Editor
- Use the `show waived drcs` command

For more information, see *Setup – Design Parameters* (`prmed` command) and *Display – Color/Visibility* (`color192` command) in the *Allegro PCB and Package Physical Layout Command Reference*.

Waived DRC Error Behavior

Waive DRC suppresses design rule violations reported by Allegro. In certain cases, waived DRC markers may become stale. With a stale waived DRC marker, the underlying design rule violation no longer exists. Performing a DRC update with *Tools – Update DRC* (`drc update` command) removes any stale waived DRCs.

For example, if you waived a via-to-via spacing violation and then move one of the vias beyond the via-to-via spacing requirement, a stale waive DRC occurs. The waived DRC appears until you perform a DRC update.

For disabled DRC checks, any associated waive DRCs are deleted as well.

Note: When waiving a differential pair DRC error, the DRC_ERROR_CLASS cline marker segments are hidden. External Waived DRC markers are never deleted.

Adding Comments to a Waived DRC Error

Allegro PCB Editor lets you attach a comment to a waived DRC error. The comment then appears in the Show Element dialog box information for that waived DRC error.

Generating the Waived DRC Error Report

You can generate a report listing all waived DRC errors in a design.

1. Choose *Tools – Reports* (`reports` command).
2. Double-click *Waived Design Rules Check Report* from the list in the Reports dialog box.
3. Click *Report*.

   The report appears with information on all waived DRC errors present in the design.

DRC Suppression

Sometimes, you must override the rules in your design. The following Boolean properties disable DRC checking. Cadence recommends that you apply these properties at the end of the design cycle because they override DRC checks.

- NODRC_COMPONENT_BOARD_OVERLAP
- NODRC_ETCH_OUTSIDE_KEEPIN
- NODRC_VIAS_OUTSIDE_KEEPIN

2

Working with Properties

You attach properties to elements in a design to instruct Allegro PCB Editor how to operate on those design elements. For example, the glossing router has no effect on a net with the NO_GLOSS property attached. Design Rule Check (DRC) adds markers to the design wherever violations occur. For details about DRC, see About Design Rule Checking. Allegro PCB Editor has more than 180 predefined property types, described in the *Allegro Platform Properties Reference*.

Properties also override design elements with attached constraints. For detailed information on constraints, see Working with Constraints.

You can attach properties:

- Directly to a design element in Allegro PCB Editor.
- To an Allegro Design Entry HDL or System Connectivity Manager schematic on your UNIX workstation that you import into Allegro PCB Editor. For additional information, see the *Transferring Logic Design Data* user guide in your documentation set.
- Through Allegro Design Entry CIS or a third-party netlist that you would import into Allegro PCB Editor. For additional information, see the *Transferring Logic Design Data* user guide in your documentation set.

You can also define your own property types (user-defined properties) by assigning each type a new name and telling Allegro PCB Editor what type of data the property represents (user units, integers, strings, or boolean) and to what types of elements the property can be attached (for example, nets, pins, or symbols).

Types of Values Used to Define Properties

Properties are defined by values of a type appropriate to their use. For example, the MIN_LINE_WIDTH property has a value giving the minimum line width expressed in user units. The NO_PIN_ESCAPE property has a boolean value—either true or false.

All properties have one of the following type of value:

STRING	LAYER_THICKNESS
INTEGER	IMPEDANCE
REAL	INDUCTANCE
DESIGN_UNITS	PROP_DELAY
BOOLEAN (true or false)	RESISTANCE
ALTITUDE	TEMPERATURE
CAPACITANCE	THERM_CONDUCTANCE
DISTANCE	THERM_CONDUCTIVITY
ELEC_CONDUCTIVITY	VOLTAGE

Types of Elements to Which You Can Attach Properties

CLINES (connect lines)	LINES
COMPONENT DEFINITIONS	NETS
COMPONENT INSTANCES	PIN DEFINITIONS
DEFINITIONS	PINS
DRCS	RECTANGLES
FIGURES	SHAPES
FRECTANGLES	SYMBOLS
FUNCTION	VIAS
FUNCTIONS	VOIDS
LAYOUT (a property attached to the entire design)	GROUPS

Extracting Property Values from Allegro PCB Editor into a Text File

You may need data about the properties in your Allegro PCB Editor design for processing with your own programs.The `extracta` command lets you extract property values from a design and place them into a text file. The command can extract both pre-defined and user-defined property values.

For example, you have defined a string property MY_PART_NUMBER, that can be attached to components. You would use `extracta` with the following command file to extract the reference designator and the MY_PART_NUMBER property value from each component in your layout.

```
#
#   This extract command file causes extraction of:
#   the user defined property MY_PART_NUMBER
#   and the reference designator
#   from each component in a layout.
#
COMPONENT
#
REFDES
MY_PART_NUMBER
#
END
```

If you had attached the property to a higher level element that owns the actual element (for example, the component definition of the component being extracted), `extracta` would still find the value and extract it.

For additional information, see the *Completing the Design* user guide in your documentation set.

Assigning Properties to Design Elements

Using *Edit – Properties* (`property edit` command), you can attach, edit, and delete properties on the following types of design elements:

Clines	Nets
Comps	Pins
Functions	Shapes
Groups	

Displaying Properties

Choose *Display – Property* (`show property` command) and use the Information tab on the Show Property dialog box to review properties attached to design elements or to show a property definition.

Use the *Graphics* tab on the Show Property dialog box to create text to visually identify one or more user-defined properties, for example, package height. Text location varies according to the type of property.

Property type	**Text location**
Component defined properties	Each placed symbol instance of that definition.
Lines, shapes, pins	Object.
Nets, XNets, diffpairs, buses	One of the pins.
Properties with or without values	Property name. For example, `<property_name>=<property_value>`

Property Inheritance

In Allegro PCB Editor, the FIXED property is an inherited property that you attach to one object (a parent), which then becomes inherited by other objects (children). When you apply an inherited property to an object, you define the behavior of that object. That behavior then defines the behavior of each child object; those objects that exist beneath it in a logical hierarchy (see Figure 2-1).

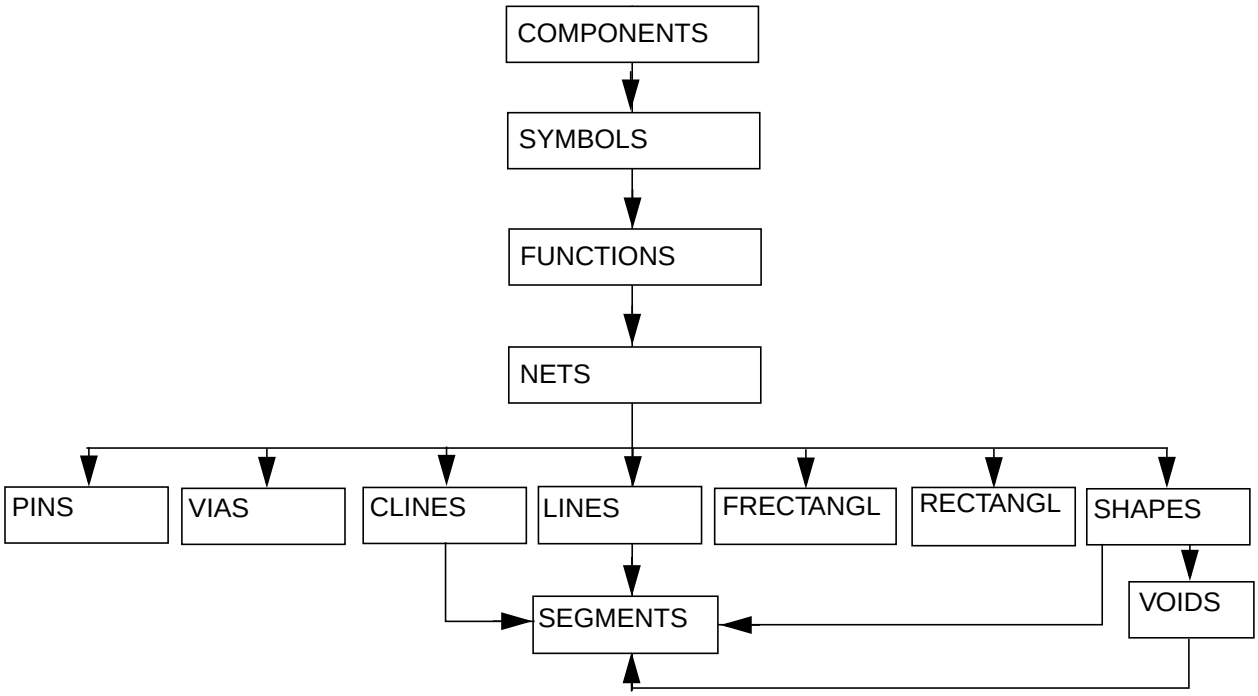
When you apply the FIXED property to a symbol, you prevent the movement or deletion of that symbol. Because it is an inherited property, you also prevent editing of the symbol children.

Design objects that can inherit the FIXED property are:

Connect lines	Nets	Symbols
Component instances	Rectangles	Symbol pins
Filled rectangles	Shapes	Vias
Lines		

Allegro PCB Editor searches objects for inherited properties along a pre-defined search path, shown in Figure 2-1. Notice that the precedence mirrors the display of objects found in the Allegro PCB Editor *Find* Filter.

Figure 2-1 Inherited Precedence



In Figure 2-1, you can see that a pin is part of a net, but is also part of a symbol. If you add the FIXED property to a net, you cannot move the symbol because the pins in the symbol inherit the FIXED property from the net.

Creating an Inherited Property

You create an inherited property with *Edit – Properties* (`property edit` command) in the same way that you attach any property to an object, using the procedure, Creating an Inherited Property, described in the *Allegro PCB and Package Physical Layout Command Reference*.

Displaying Inherited Properties on an Element

You can view inherited properties on a design element by choosing *Display – Element* (`show element` command), described in the *Allegro PCB and Package Physical Layout Command Reference*.

Changing Inherited Properties

You can modify inherited properties (add or remove) only at the parent object level. You cannot make changes at the child level. To make changes to an inherited property, choose *Edit – Properties* (`property edit` command), described in the *Allegro PCB and Package Physical Layout Command Reference*.

Creating and Editing User-Defined Properties

You can create and maintain properties of your own in Allegro PCB Editor designs. Then you can use these properties for reporting and data extraction.

You can display user-defined properties by using the *Filter* button in the Show Property dialog box (`show property` command) or any other Allegro PCB Editor function or report that displays property assignments.

Allegro PCB Editor recognizes a user-defined property assigned to an element during logic entry in Allegro Design Entry HDL or System Connectivity Manager if it has been defined in Allegro PCB Editor before executing backannotation. The definition must be an exact match of the name and pass edit checks of element type, units, and range.

In all other instances, Allegro PCB Editor recognizes user-defined properties only as information labels for your purposes. These properties do not affect automatic or interactive operations and do not generate DRC markers. You can create and use as many property definitions as you need without being concerned about their effect on Allegro PCB Editor.

Defining User-Defined Properties

You can define a property in two ways:

- Interactively, choosing *Setup – Property Definitions* (`define property` command), described in the*Allegro PCB and Package Physical Layout Command Reference.*
- In a technology file, described in the *Defining and Developing Libraries* user guide of your documentation set.

Editing User Properties

You can change a property definition, delete a property, or copy a current definition to a new property definition by choosing *Setup – Property Definitions* (`define property` command), described in the *Allegro PCB and Package Physical Layout Command Reference*.

Allegro PCB Editor lets you change and delete properties that are not currently in use and have not been defined through a technology file:

- This error message appears for a property that is currently assigned to elements:

  `Property <name> is currently associated with elements, cannot modify/delete property definition.`

- This error message appears for a property defined through a technology file:

  `Property <name> defined through a technology file, cannot modify/delete property definition.`

Storing Web Links as the Value of a Property

Allegro PCB Editor supports the storing of web links as the value of a user-defined property. You can use this feature to access a specification for the component to which the property is attached.

1. Choose *Setup – Property Definitions* (`define property` command) to define the property, as described in the *Allegro PCB and Package Physical Layout Command Reference.*
2. Choose *Edit – Properties* (`property edit` command) to add set the value of the property. In this case, set the value to be the complete browser path to the destination.
3. Choose the component and use the *Display – Element* (`show element` command) to access the web link.
4. Click on the web link.

Note: Ensure that you set the `allegro_html` environment variable in the User Preferences Editor. Use *Setup – User Preferences* (`enved` command) and click *UI*.

3

Working with Constraints

A constraint is a user-defined rule applied by Design Rule Check (DRC) to a physical element in a design. When you define and apply a constraint, Allegro PCB Editor adheres to that constraint during automatic and interactive processing and flags violations with DRC markers.

Cadence recommends using Constraint Manager to specify a variety of constraints. Choose *Setup – Constraints – Constraint Manager* (cmgr command) to access Constraint Manager. Constraint Manager provides worksheets for specifying Electrical, Physical, and Spacing constraints, as well as general DRC values and Property assignments. Basic Spacing and Physical constraints are called standard or default values.

Constraint Manager provides the following functionality:

- Creating topology files to use with electrical constraint sets
- Importing electrical constraint sets
- Assigning electrical constraint sets to buses, differential pairs, and XNets

For additional information on Constraint Manager, see the *Allegro Constraint Manager User Guide*. For detailed descriptions of the constraints, see the *Allegro Platform Constraints Reference*.

You control when DRC runs for a constraint by setting a DRC mode on that constraint. See "DRC Modes" on page 8 for definitions of each mode and tips for setting them.

Lower tiers of Allegro PCB Editor restrict access to certain types of constraints. For example, if you set electrical DRCs in a high-tier version of Allegro PCB Editor, and then open the design in a lower tier, Allegro PCB Editor preserves the constraint data, but may disable DRC. When you again open the design in the higher tier, DRC functions, but the DRC status is out-of-date. You need to run a DRC update to have the DRC markers reflect the current design state.

Types of Constraint Sets

Allegro PCB Editor organizes design rules, according to the behavior and type of element to which they apply, into the following pre-defined constraint sets:

Spacing	Constraints governing the spacing between elements on different nets (for example, *Line To Thru Pin Spacing*).
Same Net Spacing	Constraints governing the spacing between elements on the same net
Physical	Constraints governing physical construction of a net (for example, *Minimum Line Width* and *Allowed ETCH Layers*).
Electrical	Constraints governing electrical behavior of an entire net (for example, *Minimum Propagation Delay*) and differential pairs (for example, *Primary Gap*).
Design	Board-level constraints that are non-area and non-net in nature (for example, *Negative Plane Islands*). Constraints that flag a set of vias or pins with overlapping antipads or thermal pads that cut off a piece of a negative shape.

Allegro PCB Editor designs begin with default constraint sets (named DEFAULT) for spacing and physical constraints. However, electrical constraint sets do not have a default.

Allegro PCB Editor assigns nets without electrical constraint sets to the constraint set UPREV_DEFAULT in designs that you uprev to version 15.x.

You can edit the spacing and physical default constraint sets and specify where and to what elements each constraint applies. You can also assign height information to package symbol files (`.dra`) and to package keepin and package keepout areas of a board file (`.brd`) or substrate design file (`.mcm`).

Spacing Constraint Sets

A spacing constraint set (also called a spacing rule set) defines for each ETCH subclass the spacing between pairs of elements on the subclass and the controls for same-net checking.

When checking design rules, a DRC does not check pins (thru pins, test pins, smd pins) against route keepins or keepouts.

Note: Spacing Constraints are organized by net-to-net (*Spacing* domain) and same net (*Same Net Spacing* domain) in Constraint Manager.

For information on how to define a spacing constraint set, see the *Allegro Constraint Manager User Guide* and the *Allegro Platform Constraints Reference.*

Physical Constraint Sets

A physical constraint set defines rules that apply as you create the interconnections of an individual net. For example, physical constraints specify:

- Connect line (cline) widths to use in different areas and on different layers
- Allowable connections on a particular layer
- Acceptable vias to use with a particular net

This differs from the spacing constraint set, which specifies spacing rules between pairs of net elements.

Allegro PCB Editor uses these constraints to construct and check clines for minimum line width, maximum line width, minimum neck width, and maximum neck length. During routing, Allegro PCB Editor adds each cline segment using the minimum line width. When you choose *Neck* from the pop-up menu, Allegro PCB Editor adds the next cline segment with minimum neck width. A DRC places an error marker on a cline segment for any of the following violations:

- Less than minimum neck width
- Less than minimum neck width and longer than maximum neck length
- Greater than the maximum line width

You can also attach properties to a net that correspond to physical constraint sets. Property values override values in the physical constraint set.

For information on how to define a physical constraint set, see the *Allegro Constraint Manager User Guide* and the *Allegro Platform Constraints Reference.*

Electrical Constraint Sets

Electrical constraint sets contain rules controlling the electrical behavior of a net, bus or differential pair, for example, timing and noise tolerance.

Electrical constraints apply to all parts of a net, regardless of ETCH subclass or layout area. Therefore, constraint areas do not apply to electrical constraints, and there is no assignment table for them. Instead, when you assign an electrical constraint set to a net, the

ELECTRICAL_CONSTRAINT_SET property attaches to the net that is set to the name of that constraint set.

You can also attach properties to nets that correspond to electrical constraints. Property values override values in the electrical constraint set.

Note: Electrical constraints are not available in Allegro PCB Design L

For information on how to define an electrical constraint set in Allegro PCB Editor, see the *Allegro Constraint Manager User Guide* and the *Allegro Platform Constraints Reference.*

Design Constraints

Design constraints comprise board-level constraints for which you can run a DRC. Allegro PCB Editor considers placement keepins and keepouts design constraints, and as such they have a fixed spacing value of `0 mil`. A DRC considers ETCH elements and package symbols in violation when they touch or are outside/inside a keepin/keepout. Similarly, Allegro PCB Editor considers package to package placement a design constraint and has a fixed spacing value of `0 mil`.

Design constraints also let you identify potential alignment and spacing problems for soldermask openings within a symbol or pin or for pad soldermask to nearby pad or etch soldermask. The soldermask constraints are not area-dependent.

For information on how to define a design constraint set, see *Allegro Constraint Manager User Guide* and the *Allegro Platform Constraints Reference*.

The Soldermask Design Rule Check

External layer copper or etch that is not protected by a soldermask coating is considered exposed. Exposed copper can cause decay of the trace due to contamination buildup or acid cleaning as well as shorts in the design. Typically, using the symbol library, you build soldermask openings (on the soldermask layers) associated with the top- and bottom-side component pads.

The Soldermask Design Rule Check reports exposed copper or etch on the outer layers with DRC errors. It checks the TOP_COND layer with the Soldermask_Top layer in the Substrate Geometry and the Pin/Via classes of the design. It also checks the BOT_COND layer with the Soldermask_Bottom layer in the Substrate Geometry and Pin/Via classes of the design. Any copper or etch that infringes on a soldermask opening generates a DRC error.

Soldermask Design Rule Check Parameters

The Soldermask Design Rule Check supports these optional soldermask-to-copper or etch spacing parameters:

- Soldermask-to-pad and cline
- Soldermask-to-shape

To access the soldermask-to-copper or etch spacing parameters, choose *Setup – Constraints – Modes* (`cns modes` command, or alternately `cns design` command) from the menu and click the Design Constraints tab. Refer to the `cns design` command in the *Allegro PCB and Package Physical Layout Command Reference* for the description of the soldermask-to-copper or etch parameters found in the Design Constraints dialog box.

Generation of Soldermask DRC Errors

The following table summarizes the situations when soldermask DRC errors are generated.

Note: Constraint Manager displays the soldermask DRC errors in the DRC Design worksheet.

For....	**A Soldermask DRC occurs when...**
Pads	The pad is not entirely within the soldermask-to-pad and cline clearance. Overlap of the pad with the clearance results in a DRC error.
Clines	Any cline comes within the soldermask opening or the optional soldermask-to-pad and cline clearance. However, there are two exceptions. Allegro PCB Editor suppresses soldermask design rule checks on clines: ■ When the cline connects to a pad that has a soldermask opening. ■ When the cline has the FILLET property attached.
Shapes	Any copper or etch shape comes within the soldermask opening or the optional soldermask-to-shape clearance. Allegro PCB Editor suppresses soldermask design rule checks on clines when the shape connects to a pad that has a soldermask opening.
Lines and text	No soldermask DRC error.

The soldermask DRC marker appears on the DRC TOP_COND or BOT_COND subclass matching the copper or etch subclass object in violation. Allegro PCB Editor generates a single error for each cline, shape, or pad.

Soldermask DRC Text Markers

The soldermask DRC marker appears on the DRC TOP_ETCH or BOT_ETCH subclass matching the copper or etch subclass object in violation. Allegro PCB Editor generates a single error for each cline, shape, or pad. The following are soldermask DRC text markers:

- M-L: Soldermask to Cline
- M-V: Soldermask to Via
- M-P: Soldermask to Pin
- M-S: Soldermask to Shape

Differential Pairs

Allegro PCB Editor supports routing physical and electrical DRCs for edge-coupled differential pairs, that is, a pair of nets or XNets routed side-by-side on the same layers of a board. You can set up differential pairs as an electrical constraint set, or if you require a more robust line and space gap control, you can use a combination of physical and electrical constraint sets.

The following sections describe the basic flow for defining differential pairs:

- Designating Nets as Differential Pairs
- Assigning Electrical Constraint Sets to Differential Pairs
- Enabling a DRC

Designating Nets as Differential Pairs

You can create differential pairs in the following ways:

- Assign nets you want routed as differential pairs by choosing *Logic – Assign Differential Pair* (`diff pairs` command). You can assign nets individually or in groups using the net naming conventions.
- Import differential pair logic from a schematic by choosing *File – Import – Logic* (`netin` command).
- Designate nets in a signal model as differential pairs. Refer to Model and Library Management in the *Allegro PCB SI User Guide* for details.

Assigning Electrical Constraint Sets to Differential Pairs

You can assign electrical constraint sets (ECSets) to differential pairs at the net or XNet level or the differential pair object level.

If you assign ECSets at both the net or XNet level and the differential pair object level, a DRC uses the differential pair constraints at the differential pair object level.

If you assign different ECSets to each of the member nets or XNets, a DRC uses the ECSet with the most conservative settings:

- Smallest phase tolerance
- Smallest gaps and gap tolerances

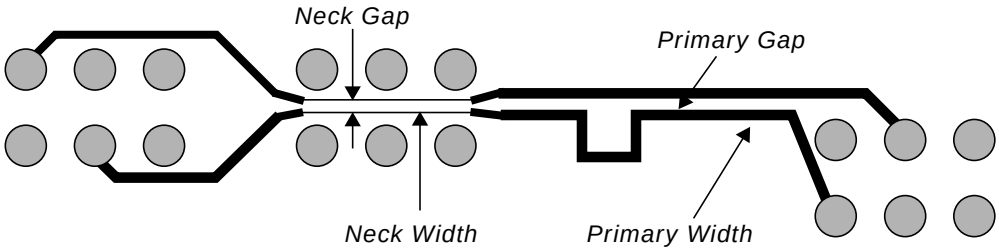
- *Gather control* set to *Include* takes precedence over *Ignore*

Notes:

- Properties override ECSet settings.
- When you set differential pair properties on nets, nearly all of these properties rise up to the XNet and differential pair group levels. The differential pair properties begin with DIFFP_ and correspond to the constraints in the *DiffPair Values* tab of the Electrical Constraints dialog box. The MIN_LINE_WIDTH and MIN_NECK_WIDTH properties (*Primary Line Width* and *Neck Width* in the Electrical Constraints dialog box), also rise up to the XNet and differential pair group levels.

Figure 3-1 shows the various parameters used for controlling a differential pair:

Figure 3-1 Controlling a Differential Pair



+*Tolerance* and -*Tolerance* determine when lines are uncoupled

Enabling a DRC

If not done in the previous step, enable a DRC for the differential pair constraints and update the DRC. See Setting DRC Modes for the Electrical Constraint Set in the *Allegro PCB and Package Physical Layout Command Reference.* Set *All Differential pair checks* to *On* and choose *Tools – Update DRC* (`drc update` command), so that the DRC markers reflect the current design state.

You can set and verify the following DRCs, shown in Figure 3-2, based on the parameters in Figure 3-1:

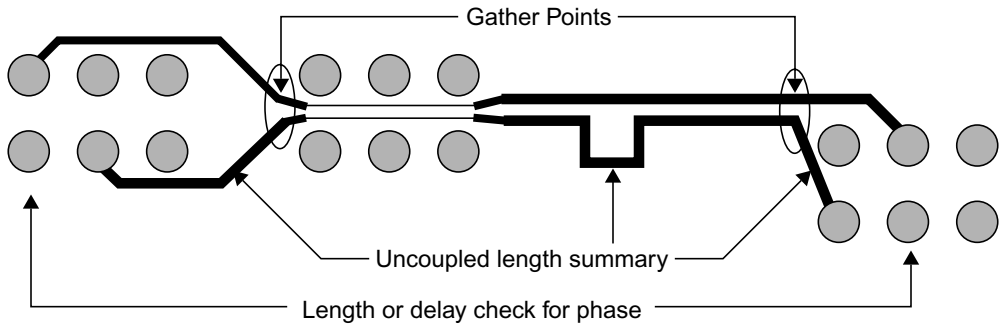
- Segment coupling (either primary gap or neck gap plus or minus tolerance, if defined).

  The length of uncoupled segments adds to the total of uncoupled length. This total is then compared to the *Max uncoupled length* constraint.

- Tolerance check comparison of the length or delay between corresponding driver/ receiver pairs of the two differential pair nets or XNets.
- Minimum segment-to-segment spacing.

  A minimum segment-to-segment spacing check is based on the specified differential pair minimum line spacing constraint or the *Line To Line* spacing in the Spacing rule set if a minimum spacing is not defined for the differential pair.

Figure 3-2 Setting and Verifying DRCs



Defining Differential Pairs by Layer

The following describes the basic flow for defining differential pairs by layer:

- Designating Nets as Differential Pairs on page 33
- Assigning Electrical Constraint Sets for Differential Pairs on page 33
- Creating a Physical Constraint Set on page 34
- Enabling a DRC on page 34

Designating Nets as Differential Pairs

See Designating Nets as Differential Pairs on page 31 for detailed information.

Assigning Electrical Constraint Sets for Differential Pairs

To define differential pair gap and neck width by layer, choose *Setup – Constraints – Physical* (`cmgr_phys` command). In the tree view, expand the *Physical Constraint Set* folder, then click the *All Layers* icon. Enter values under the *Differential Pair Gap - Primary* and *Differential Pair Gap - Neck* columns for each layer in the design.

Important

Differential Pair values that you enter in the Electrical worksheet of Constraint Manager (*Setup – Constraints – Electrical* or `cmgr_elec` command) will override the values you set in the Physical worksheet.

Note: You do not need to perform the following steps if all the differential pairs are using the same *Primary gap* and *Line width* values from the default Physical Constraint Set.

Creating a Physical Constraint Set

Create a physical constraint set in the Physical worksheet of Constraint Manager by choosing *Setup – Constraints – Physical* (`cmgr_phys` command). Enter values for *Differential Pair Gap - Primary*, *Differential Pair Gap - Neck*, *Line Width - Min*, *Line Width - Max*, and *Neck - Min Width*.

Enabling a DRC

See Enabling a DRC on page 32 for information.

Using Constraint Values in Routing and Checking Differential Pairs

When routing or checking differential pairs, Allegro PCB Editor determines the following:

- Whether to use a primary gap or a neck gap
- The primary line width
- Neck line width

Primary Gap or Neck Gap

Allegro PCB Editor uses *Neck gap* over *Primary gap* when the line width of the differential pair is less than the value of the *Primary line width*.

Special necking is implemented when the line width constraint equals the neck width constraint for nets, but the *Neck gap* is less than the *Primary gap*. A segment that comes within the neck gap plus the (+) tolerance to segments of its diff pair partner are considered necked for both the differential pair DRC check and the differential pair line width check.

- For the uncoupling check, the neck gap band is the defining coupled band.

- For the dynamic phase check, the neck gap band is the defining band for coupling/uncoupling events.
- For the neck length check, these segments are considered necked and must not exceed the neck length.

When using *Primary gap* during routing or checking, Allegro PCB Editor looks for the following in sequential order:

1. DIFFP_PRIMARY_GAP property value on the differential pair object.
2. *Primary gap* value in the ECSet.
3. *DiffPair primary gap* value in the Physical Rule set.

   If you did not define a value in this rule set, Allegro PCB Editor defaults to `0 mil`.

When using *Neck gap* during routing or checking, Allegro PCB Editor looks for the following in sequential order:

1. DIFFP_NECK_GAP property value on the differential pair object.
2. *Neck gap* value in the ECSet.
3. *DiffPair neck gap* value in the Physical Rule set.

Note: If Allegro PCB Editor does not find any of these values, Allegro PCB Editor uses the *Primary gap* value in the order described above to search for a primary gap value.

Primary Line Width

When determining the primary line width, Allegro PCB Editor looks for the following in sequential order:

1. *Primary line width* assigned to the differential pair object. You set this only in the Constraint Manager, or you can set the MIN_LINE_WIDTH property on individual nets, which bubble up to the differential pair object.
2. Differential pair *Primary line width* in the ECSet.
3. *Min line width* in the Physical Rule set.

Neck Line Width

Allegro PCB Editor determines the value of the neck line width in the following sequence:

1. *Neck width* assigned to the differential pair object. You set this only in the Constraint Manager, or you can set the MIN_NECK_WIDTH property on individual nets, which bubble up to the differential pair object.
2. Differential pair *Neck width* value in the ECSet.
3. *Min neck width* value in the Physical Rule set.

Note: When you define the primary gap or neck gap in a physical constraint set (differential pair by gap layer), Allegro PCB Editor always uses values from the constraint set that applies to the differential pair for points outside all constraint areas. Allegro PCB Editor does not use values from constraint sets assigned to the differential pair for points inside a constraint area.

Viewing DRC Violations for Differential Pairs

In addition to flagging nets that violate differential pair constraints with DRC markers, Allegro PCB Editor also marks the offending line segments with a highlighting color that is half the width of the line. To see these segments:

1. Choose *Display – Color/Visibility* (`color192` command).
2. In the Color dialog box, choose *Stack-Up*.
3. In the *DRC* column, enable the ETCH subclasses on which the differential pairs appear.
4. If necessary, modify the DRC subclass colors.
5. Click *OK*.
6. Choose *Display – Color Priority* (`color priority` command).
7. If a color you assigned to a DRC subclass is not at the top of the list, move it up. See Defining Color Priority in the *Allegro PCB and Package Physical Layout Command Reference* for instructions.

Transferring Logic from Older Schematics

You can transfer logic from 14.x schematics into a 15.x Allegro PCB Editor design. The differential pair properties on nets in the logic file connect to the 15.x elements in the following ways:

14.x Properties	Connected to these 15.x Elements
DIFFERENTIAL_PAIR	differential pair group object
DIFFP_LENGTH_TOL	DIFFP_PHASE_TOL property

14.x Properties	Connected to these 15.x Elements
DIFFP_2ND_LENGTH	DIFFP_UNCOUPLED_LENGTH property

To transfer logic, choose *File – Import – Logic* (`netin` command) and the `netrev` command, described in the *Allegro PCB and Package Physical Layout Command Reference*.

Layer Sets

Allegro PCB Editor lets you assign layer-set wiring rules to net-based objects to control impedance, shielding, or return path requirements. You place layer-set constraints on nets, XNets, differential pairs, or buses, ensuring adherence to wiring requirements by locking routes within the appropriate layer sets.

Topics covered in this section include:

- Defining Layer Sets
- Assigning Layer-Set Constraints
- Using Layer-Set Constraints in DRC Mode

The following definitions apply to layer sets:

Layer Set	Group of etch layers (also referred to as subclasses) applied to nets, XNets, diff pairs, or buses. You can assign one or more layer sets to an object.
Pin Escape	Series of blind, buried, or through-hole vias and clines that defines a path from an outer layer to an inner target routing layer.
Short side of an XNet	Shortest two-pin net of a two-net XNet.

Defining Layer Sets

Before you can apply a layer-set constraint, you must define a layer set. Using the Layer Sets dialog box, accessed by running the `define layersets` command or through the Electrical Constraints dialog box (*Setup – Constraints – Electrical constraint sets)*, you choose available layers and group them in a layer set. There is no limit to the number of layers that can belong to a layer set.

Assigning Layer-Set Constraints

After you define the layer set, assign the layer-set constraint to applicable etch objects using one of the following methods:

- Include a layer set in an ECSet.

A layer set can belong to multiple ECSets. For additional information on assigning a layer set to an ECSet, see the Select Layer Sets Dialog Box in the *Allegro PCB and Package Physical Layout Command Reference*.

- Attach the LAYERSET_GROUP property.

  For more information about this property, see LAYERSET_GROUP in the *Allegro Platform Properties Reference*.

- Use Constraint Manager to assign a layer-set constraint to a net, XNet, differential pair, bus, or ECSet.

  Cadence recommends using the *Electrical Constraint Spreadsheet*, available by choosing *Setup – Electrical Constraint Spreadsheet*, to manage layer-set constraints. For information about layer sets in the *Electrical Constraint Spreadsheet,* see Layer Sets in the *Allegro Platform Constraints Reference*.

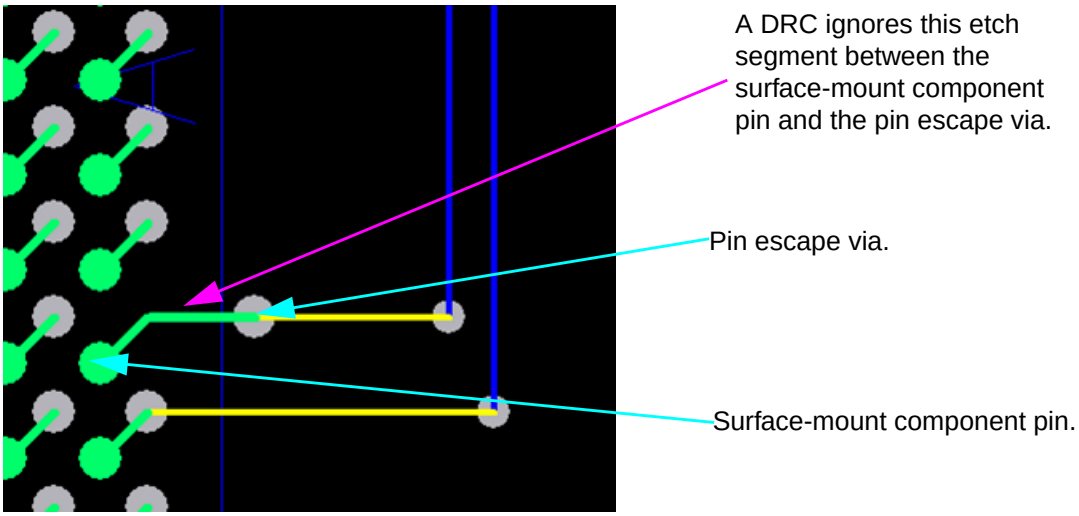
Using Layer-Set Constraints in DRC Mode

Enable design rule checking, so that the DRC markers reflect the current design state.

When checking layer-set constraints, Allegro PCB Editor determines the following:

- All nets of an XNet route on the same layer set to avoid a DRC error. However, a DRC violation does not occur when the short-side of an XNet runs on a surface-layer between two pins.
- The DRC on a bus is independent from other nets on the bus.
- If the cline is on the path from a surface-mount pin to a legal routing layer, either through a via or a staggered path that progresses towards the legal routing layer, no DRC errors occur. This is considered a pin escape, as shown in Figure 3-3.

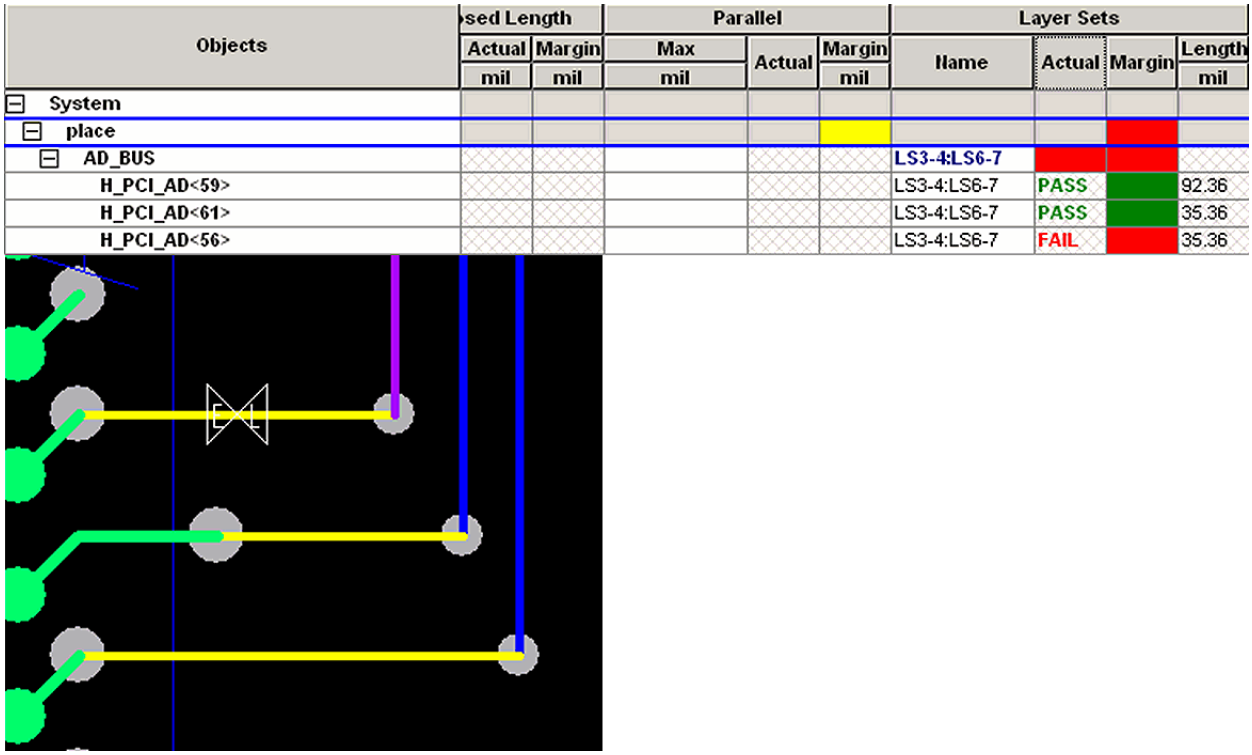
Figure 3-3 Example of Pin Escape Etch



Viewing DRC Violations for Layer-Set Constraints

When a layer-set constraint violation occurs, a DRC marker appears directly on the cline and status information appears in the *Electrical Constraint Spreadsheet*, as shown in Figure 3-4.

Figure 3-4 Example of a DRC Violation in Constraint Manager and Allegro PCB Editor



Objects	sed Length		Parallel			Layer Sets			
	Actual	Margin	Max	Actual	Margin	Name	Actual	Margin	Length
	mil	mil	mil		mil				mil
System									
place									
AD_BUS						LS3-4:LS6-7			
H_PCI_AD<59>						LS3-4:LS6-7	PASS		92.36
H_PCI_AD<61>						LS3-4:LS6-7	PASS		35.36
H_PCI_AD<56>						LS3-4:LS6-7	FAIL		35.36

In this example, a violation occurred because you cannot interchange the two layer sets (defined as LS3-4 and LS6-7), even though the purple cline exists on a subclass of the alternate layer set, LS6-7. To avoid this violation, you must route the purple cline on layer Sig_4V or route the yellow cline on one of the two layers of layer set LS6-7. Constraint Manager reports the actual results as a PASS/FAIL condition. The accumulated amount of etch length on non-layer set subclasses appears in the *Length* column of the Electrical Constraint Spreadsheet.

In addition to flagging nets that violate layer-set constraints with DRC markers, Allegro PCB Editor also marks the offending cline with a highlighting color. See Assigning Colors to Subclasses in the *Allegro PCB and Package Physical Layout Command Reference* for instructions.

Using Pin Delay

You can include pin delay in DRC calculations for DIFFERENTIAL PAIR PHASE TOLERANCE, PROPAGATION DELAY, and RELATIVE PROPAGATION DELAY by assigning the PIN_ DELAY property to component instance or definition pins. The PIN_ DELAY property specifies the time delay or length from the internal package connection to the pin's mounting layer. Use the PIN_DELAY property to manage interchip delay or die-to-die timing paths across a printed circuit board and thereby factor inter-package delay into timing requirements.

When pin delay is measured in time units, it is multiplied by the *Pin Delay Propagation Velocity Factor,* which is a constant used to convert from time to etch layer length units if you defined DIFFERENTIAL PAIR PHASE TOLERANCE, PROPAGATION DELAY, and RELATIVE PROPAGATION DELAY in time units. To factor pin delay into these DRC calculations, choose *Setup – Constraints – Modes*, then click the Electrical Options tab. In the *Pin Delay* group box, enable *Include in all Propagation Delays and DiffPair Phase checks* and enter a value for *Propagation velocity factor*. For more information, see the *Allegro PCB and Package Physical Layout Command Reference*.

A schematic- or a board-driven flow supports pin-delay values.

Schematic-driven Pin Delay Flow

In a schematic-driven flow, these pin-level delay values are defined as library properties that can be written to the Allegro Design Entry HDL, System Connectivity Manager, or Allegro Design Entry CIS library files sent to Allegro PCB Editor. You can use Allegro PCB Librarian XL to manually assign the PIN_DELAY property and values to symbol pins or automatically import the PIN_DELAY values through its *Import Wizard*, which supports Comma Separated Value (`.csv`) and Excel (`.xls`) file format options.

Packager-XL packages the design into `pst*.dat` files. The `pstchip.dat` file contains the default values of the PIN_DELA Y property, subsequently imported into Allegro PCB Editor using the latter's *File – Import – Logic* (`netrev` command).

You can use the PIN_DELAY property values in Constraint Manager, interactive routing in Allegro PCB Editor or PCB Router, automatic routing in PCB Router, and DRC verification in both Allegro PCB Editor and Allegro PCB Router. For more information on using pin delays in Allegro Constraint Manager and Allegro PCB Router, see *Analyze – Analysis Modes* in the *Allegro Constraint Manager Reference* and the *Allegro PCB Router User Guide*, respectively.

In Allegro PCB Editor, you can then optionally edit and override the PIN_DELAY values from the `pstchip.dat` file on component-instance pins. Only overrides are backannotated to the schematic.

Board-driven Pin Delay Flow

In a board-driven, pin-delay flow, you can export pin delay values from an external source using Allegro PCB Editor's *File – Export – Pin Delay* (`pin_delay out` command) and then import them to another design and annotate them to component instance pins using *File – Import – Pin Delay* (`pin_delay in` command) in Allegro PCB Editor. You can also use *Edit – Properties* (`property edit` command) to assign the PIN_DELAY property.

When you change the value of the PIN_DELAY property in Allegro PCB Editor and use *File – Export – Logic* (`feedback` command) to export modifications to Allegro Design Entry HDL or System Connectivity Manager, both Allegro PCB Editor-modified (instance) and Allegro Design Entry HDL or System Connectivity Manager-generated (definition) PIN_DELAY values come across in the `cmdbview.dat` file if you are using a Constraint-Manager-enabled flow. If you are using a flow without Constraint Manager, manual edits to the value of the PIN_DELAY property pass in the `pstxprt.dat` file.

Constraint Manager is an optional point to enter PIN_DELAY values and to edit those propagated from the `chip.prt` files. The Constraint Manager flow maintains any overrides of the PIN_DELAY property made in Constraint Manager or in Allegro PCB Editor, but does not backannotate them to the Allegro Design Entry HDL or System Connectivity Manager schematic. For more information on using the PIN_DELAY property in Constraint Manager, see *Analyze – Analysis Modes* in the Allegro Constraint Manager Reference.

Using Z Axis Delay

To more accurately account for delay in your designs, you can include the conducting portion of a via/pin (also known as Z Axis Delay) in DRC calculations for DIFFERENTIAL PAIR PHASE TOLERANCE, PROPAGATION DELAY, and RELATIVE PROPAGATION DELAY. Z Axis Delay includes any through-hole component or any hole with a depth value. The conducting portion of a via/pin comprises the thickness through the board from the placed symbol layer where a net enters a padstack, which may be a via or a through-hole pin, to the layer from which it exits.

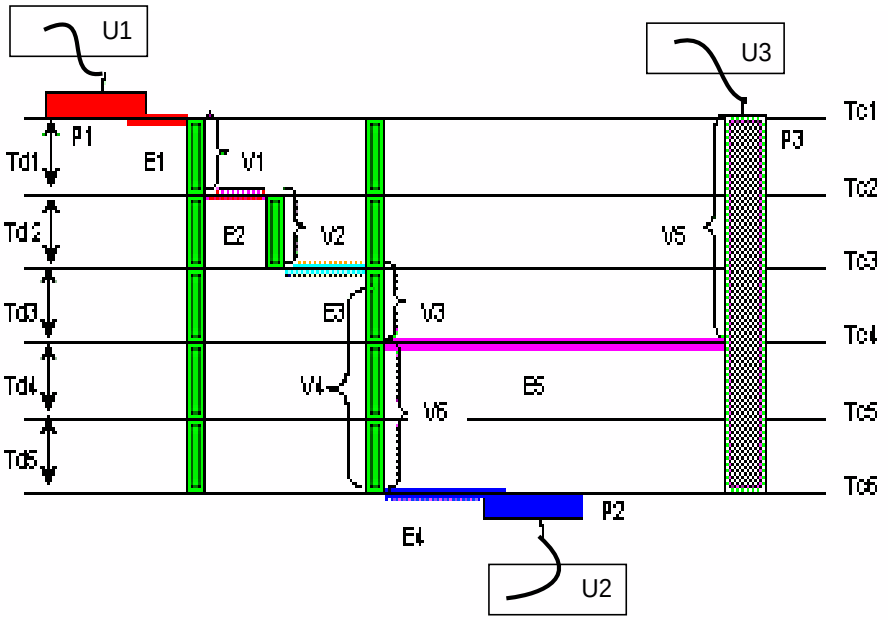
All layer dielectric and copper thickness lengths between the entry and exit layers are calculated for the conducting portion of a via/pin and are added to the overall net or pin pair length. Copper thickness for the entry and exit layers are excluded from the calculations. Surface mount vias, such as testpoints, have no effect on the total calculations.

When the conducting portion of a via/pin is measured in time units, it is multiplied by the *Z Axis Delay Propagation Velocity Factor,* which is a constant used to convert from time to etch layer length units if you defined DIFFERENTIAL PAIR PHASE TOLERANCE, PROPAGATION DELAY, and RELATIVE PROPAGATION DELAY in time units.

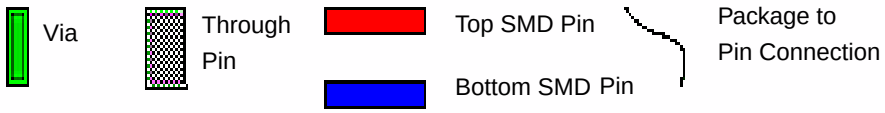
To factor the conducting portion of a via/pin into these DRC calculations, choose *Setup – Constraints – Modes*, then click the Electrical Options tab. In the *Z Axis Delay* group box, enable *Include in all Propagation Delays and DiffPair Phase checks* and enter a value for *Propagation velocity factor*. For more information, see the *Allegro PCB and Package Physical Layout Command Reference*.

Z Axis Delay Example

Assume the following stackup is defined in the *Layer Cross Section* dialog box, accessible by running *Setup – Cross-section* (`define lyrstack` command), described in the *Allegro PCB and Package Physical Layout Command Reference*:



Legend



Tdr = the thickness of the dielectric between layers l and l + 1
Tcr = the thickness of the copper on layer l
V1 = Td1; V2 = Td2; V3 = Td3
V4 = Td3 + Tc4 + Td4 + Tc5 + Td5
V5 = Td1 + Tc2 + Td2 + Tc3 + Td 3
V6 = Td4 + Tc5 + Td5

Total Length U1.1 to U2.1 = E1 + V1 +E2 +E3 +V4 +E4
Total Length U1.1 to U3.1 = E1 + V1 + E2 + V2 + E3 + E5 + V5
Total Length U2.1 to U3.1 = E4 + V5 + E5 + V5
Given U1.1 to U3.1 is the transmission line, stub length off of U2 = E4 + V6

Consider the following stackup and layer thicknesses, as outlined in the preceding figure, as an example.

Layer Type	Etch Subclass Name	Thickness
Conductor	Tc1	1.2 mils
Dielectric	Td1	5.0 mils
Plane	Tc2	1.2 mils
Dielectric	Td2	5.0 mils
Conductor	Tc3	0.7 mils
Dielectric	Td3	5.0 mils
Conductor	Tc4	0.7 mils
Dielectric	Td4	5.0 mils
Plane	Tc5	1.2 mils
Dielectric	Td5	5.0 mils
Conductor	Tc6	1.2 mils

The distance between two conductor layers, such as Tc3 to Tc4, comprises only the thickness of Td3, or 5.0 mils, excluding the thickness of the entry and exit conductor layers themselves.

For via 6, the conducting portion of the via is calculated as:

```
Td4 (5.0 mils) + Tc5 (1.2 mils) + Td5 (5.0 mils) = 11.2 mils
```

Setting Nets to Check Themselves for Crosstalk and Parallelism

To enable the nets in your design to perform a design rule self-check for crosstalk and parallelism (in addition to the checks the net makes against all other nets), you can turn on the feature using the *Objects - Create - Electrical CSet i*n Constraint Manager.

Enabling this command creates same net crosstalk records in your design database. Because such data is not supported in releases prior to 15.5.1, you must perform a database down rev in Release 15.2 to remove these objects. For releases earlier than 15.5.1, attempts to open designs containing same net DRC data produce an error message and the design does not open.