cadence®

Virtuoso Design Intent User Guide

Product Version ICADVM20.1
October 2020

Printed in the United States of America.

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

Contents

Preface

This document describes Virtuoso® Design Intent available in Virtuoso Schematic Editor XL and Virtuoso Layout Suite XL.

Design intent is a collaboration tool that uses annotations as a means for schematic designers to capture and communicate design goals in Schematics XL and provide layout designers with the freedom to decide how to implement those goals in Layout XL.

Design intents in schematic are stored in the schematic view and in the layout database for layout. Using the process of syncing, the design intent data is shared between the schematic view and layout database.

This user guide is aimed at schematic and layout designers of integrated circuits and assumes that you are familiar with:

- Virtuoso Schematic Editor XL (Schematics XL)
- Virtuoso Layout Suite XL (Layout XL)
- Virtuoso Unified Custom Constraints

This preface contains the following topics:

Scope

The functionality described in this guide can be used only in advanced nodes and methodologies releases (for example, ICADVM18.1, ICADVM20.1).

Licensing Requirements

For information on licensing in the Virtuoso design environment, see *Virtuoso Software Licensing and Configuration Guide*.

Related Documentation

What's New and KPNS

- *Virtuoso Design Intent What's New*
- *Virtuoso Design Intent Known Problem and Solutions*

Installation, Environment, and Infrastructure

- *Cadence Installation Guide*
- *Virtuoso Design Environment User Guide*
- *Virtuoso Design Environment SKILL Reference*
- *Cadence Application Infrastructure User Guide*

Technology Information

- *Virtuoso Technology Data User Guide*
- *Virtuoso Technology Data ASCII Files Reference*
- *Virtuoso Technology Data Constraints Reference*
- *Virtuoso Technology Data SKILL Reference*

Virtuoso Tools

- *Virtuoso Layout Suite XL: Basic Editing User Guide*
- *Virtuoso Layout Suite XL: Connectivity Driven Editing*
- *Virtuoso Schematic Editor User Guide*
- *Virtuoso Unified Custom Constraints User Guide*

- *Virtuoso Unified Custom Constraints Configuration Guide*
- *Virtuoso Unified Custom Constraints Getting Started Guide*
- *Virtuoso Unified Custom Constraints SKILL Reference*

Additional Learning Resources

Video Library

The Video Library on the Cadence Online Support website provides a comprehensive list of videos on various Cadence products.

To view a list of videos related to a specific product, you can use the *Filter Results* feature available in the pane on the left. For example, click the *Virtuoso Layout Suite* product link to view a list of videos available for the product.

You can also save your product preferences in the Product Selection form, which opens when you click the *Edit* icon located next to *My Products*.

Virtuoso Videos Book

You can access certain videos directly from Cadence Help. To learn more about this feature and to access the list of available videos, see Virtuoso Videos.

Rapid Adoption Kits

Cadence provides a number of Rapid Adoption Kits that demonstrate how to use Virtuoso applications in your design flows. These kits contain design databases and instructions on how to run the design flow.

To explore the full range of training courses provided by Cadence in your region, visit Cadence Training or write to training_enroll@cadence.com.

Note: The links in this section open in a separate web browser window when clicked in Cadence Help.

Help and Support Facilities

Virtuoso offers several built-in features to let you access help and support directly from the software.

- The Virtuoso *Help* menu provides consistent help system access across Virtuoso tools and applications. The standard Virtuoso *Help* menu lets you access the most useful help and support resources from the Cadence support and corporate websites directly from the CIW or any Virtuoso application.
- The Virtuoso Welcome Page is a self-help launch pad offering access to a host of useful knowledge resources, including quick links to content available within the Virtuoso installation as well as to other popular online content.

  The Welcome Page is displayed by default when you open Cadence Help in standalone mode from a Virtuoso installation. You can also access it at any time by selecting *Help – Virtuoso Documentation Library* from any application window, or by clicking the *Home* button on the Cadence Help toolbar (provided you have not set a custom home page).

For more information, see Getting Help in *Virtuoso Design Environment User Guide.*

Customer Support

For assistance with Cadence products:

- Contact Cadence Customer Support

  Cadence is committed to keeping your design teams productive by providing answers to technical questions and to any queries about the latest software updates and training needs. For more information, visit https://www.cadence.com/support.
- Log on to Cadence Online Support

  Customers with a maintenance contract with Cadence can obtain the latest information about various tools at https://support.cadence.com.

Feedback about Documentation

You can contact Cadence Customer Support to open a service request if you:

- Find erroneous information in a product manual
- Cannot find in a product manual the information you are looking for

- Face an issue while accessing documentation by using Cadence Help

You can also submit feedback by using the following methods:

- In the Cadence Help window, click the *Feedback* button and follow instructions.
- On the Cadence Online Support Product Manuals page, select the required product and submit your feedback by using the *Provide Feedback* box.

1

Getting Started with Design Intent

Virtuoso Design Intent complements the Virtuoso Schematic Editor XL and Virtuoso Layout Suite XL applications by providing a method for the schematic designer to capture and communicate their design goals on existing objects within a design and to provide the layout designer the freedom to decide how those goals are implemented. Design goals can be defined and discussed, implementation restrictions resolved, and decisions agreed and recorded to prevent duplication of effort during design reuse.

The schematic designer can specify the design intent for an object by recording editable notes, referred to as annotations, directly in the schematic design. Each design intent is displayed on the canvas with an accompanying glyph which acts as a marker and facilitates access to detailed information about the design intent. Each annotation can hold key constraint requirements and detailed instructions for individual objects.

Design intents in Schematics XL are stored in the schematic view and in the layout view for Layout XL. The process of syncing shares the design intent data from the schematic view to the layout view, and vice versa.

After syncing, the annotations can be easily identified on the layout canvas and in the Navigator assistant. The layout designer can begin implementing each intent as specified, reporting the current implementation stage for each intent, and adding progress notes as required.

Design Intent complements the existing constraints flow. Capturing the schematic designer's requirements at a higher level, enables them to communicate their requirements to layout engineers without overlapping their roles. By using Design Intent to capture design goals, constraints can be used to focus purely on defining the specific rules that are required to satisfy and implement the designer's original intent.

Design intent is not a replacement for Constraints, so if you already use constraints, you can continue to use them as a basis to implement the Design Intent requirements.

By regularly syncing in the layout view, schematic designers can be kept up to date on the implementation progress of each annotation and respond to any queries or comments recorded by the layout designer. Finally, the schematic designer can sign off a design when the original goals have been satisfactorily implemented into the design.

The progress of all the design intent implementation on a design can be checked using a high level report generated from either Schematics XL or Layout XL.

This section covers the following topics:

- The Design Intent Flow
- Benefits of using Design Intent
- Design Intent Interaction between Schematics XL and Layout XL
- Features Specific to Design Intent

The Design Intent Flow

The flow of Virtuoso Design Intent is contained within Virtuoso Schematics XL and Virtuoso Layout XL and requires interaction between the schematic and layout designers working together on a design.

1. Schematic designer has some intent to convey for an object or group of objects (for example, device matching requirements, noisy/sensitive nets, high currents, voltage drops, pin information). They capture their design goals on the Create Design Intent form using a combination of text notes and predefined custom property profiles. (The property profiles contain frequently used design intent specific properties that formalize design goals e.g. add shield, add guard ring, etc.)
2. The design intent is saved to the schematic and is displayed as an easily identifiable, colored annotation on the canvas. Objects with annotations are easy to identify in the Navigator as the existing icon is overlaid with a special design intent icon overlay. On the canvas, design intent annotations on objects are displayed with a name, glyph and shape.
3. The design intent is synced to Layout XL and from then on design intent changes are updated on the design, visible in both Schematics XL and Layout XL.
4. Layout designer can clearly identify the objects with design intents and begins implementing each intent. Using the Edit Design Intent form, they can update the current implementation stage and add implementation notes or queries to communicate back to the schematic designer.
5. By regularly syncing, the schematic designer is updated on the implementation progress of each design intent in the design. Using the Edit Design Intent form, they can respond to any queries or comments recorded by the layout designer, adapting the intent if required and ultimately signing off on the implementation of their intended design.
6. Progress can be checked at any point using a high level summary report generated from either Schematics XL or Layout XL.

Benefits of using Design Intent

Design intent offers the following benefits:

- Design intents are directly created and edited on design objects within the canvas using right-click menu items and colored annotations. Saves the schematic designer from having to design and set up each object individually. Using annotations allows them to focus on capturing their design intent without having to create physical constraints on the design.
- The Navigator displays design intent information within its own specific category and also on individual objects associated with a design intent using including colored overlays, tooltips and right-click menu items.
- A tooltip summarizing the design intent on a group of objects and its implementation status, is displayed by hovering on a design intent in the Navigator.
- Key information can be accessed from an annotation's accompanying glyph on the canvas to display an info balloon that can be pinned to the canvas.
- Allows a seamless flow to be developed by means of regular syncing between the schematic and layout views. Syncing also updates the easily identifiable annotations between the schematic and layout canvases.
- Design intent is transferred to the layout view for implementation, allowing layout designers the freedom to decide how to physically implement the intended design.
- Encourages collaboration between the schematic and layout designers to produce effective designs. The layout designer is able to update the design intent status in Layout XL and add implementation notes to communicate with the schematic designer about any issues that need to be resolved. The schematic designer can change the specification if necessary and ultimately sign off on the implementation.
- Facilitates checking on the current progress of all the design intent on a design using a high level report generated from either Schematics XL or Layout XL.

Design Intent Interaction between Schematics XL and Layout XL

Virtuoso Design Intent is run between Schematics XL and Layout XL. Generally, design intents are created in Schematics XL where they are stored in the schematic view. They are then transferred (using syncing) to Layout XL, where they are stored in the layout view.

To enable seamless syncing when working with design intents, the same up to date version of a cell must be open in both schematic and layout views. If working in Schematics XL, the corresponding layout view must also be open (read-only is acceptable) and vice versa if working in Layout XL. As might be expected, many of the options are disabled when in read-only mode.

Design intent can be launched directly from existing schematics. If your organization uses constraints, there are specific requirements that you need to consider. For more details, see Using Design Intent with Constraints.

Using Design Intent with Constraints

The storage of constraints differs between schematic and layout. In the schematic, constraints created in Schematics XL are stored in an OpenAccess database (`hierDesign.oa`) that resides in the constraint view. In the layout, the constraints are stored in the layout database itself.

Design Intent complements the constraints flow in your organization by allowing schematic designers to capture their design goals in Schematics XL without actually creating constraints so a separate constraint view is not created. The design intent is synchronized to Layout XL, where the layout designer has the freedom to decide how to physically implement and achieve the design intent using constraints within Layout XL. The responsibility of creating constraints moves from Schematics XL to Layout XL.

To ensure that a constraint view is not created when transferring constraints from Layout XL to Schematics XL using the *Update Schematic/Layout Constraints* button, all design intents implemented using the constraint manager must have their constraints included in the design intent template (physical constraint). This ensures that device correspondence information is maintained during an *Update Schematic/Layout Constraints* transfer and that implementation constraints are not transferred.

Prerequisites

It is recommended that before implementing design intent into a design, the schematic and layout designer agree on the type of information to be shared within a design intent and that custom profiles are set up. For more details, see Defining Property Profiles.

Features Specific to Design Intent

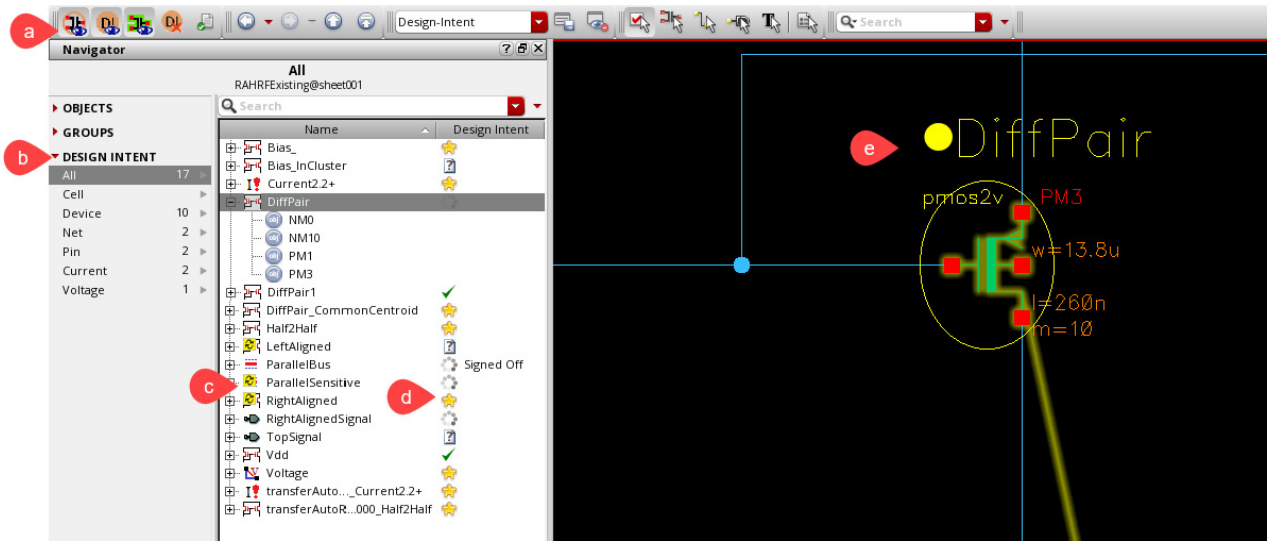
Design intent has specific features available using the *Design-Intent* workspace which is comprised of the Design Intent toolbar, the canvas, and the Navigator assistant. In Layout XL, it also includes the addition of the Constraint Manager assistant. The features range from carrying out specific actions on an object or design intent to toggling their visibility on the canvas. Each of these features is described in detail in the following sections.

- Design-Intent Workspace
- Design Intent Toolbar
  - Interacting with Design Intent in the Navigator Assistant
  - Context-Sensitive Menu Options in Navigator
  - Design Intent Icons in Navigator
  - Implementation Status of Design Intent
  - Tooltips on the Navigator
- Constraint Manager
- Interacting with Design Intent on the Canvas
  - Context-Sensitive Menu Options on Canvas
  - Annotations and Glyphs
  - Design Intent Info Balloon
  - Haloing
  - Syncing

Design-Intent Workspace

The Design Intent workspace is designed to help you create and manage design intents using the Navigator assistant and on the canvas.

The workspace comprises the following elements:



a Design Intent Toolbar	lets you toggle the display of design intent with a single mouse click.
b Navigator assistant	shows the *Design Intent* category in the *Summary* pane of the Navigator assistant. There is a set for each design intent category and the number of design intents within each set is displayed. When you select an object in the Navigator, the object is highlighted on the canvas in Schematics XL and Layout XL. It also halos any objects that are also members of the same design intent across both applications. For more details, see Interacting with Design Intent in the Navigator Assistant.
c Sync icons	alerts designer to any design specification or implementation status changes. For more details, see Design Intent Icons in Navigator.
d Implementation Status icons	indicates the current implementation status for each design intent. For more details, see Design Intent Icons in Navigator.
e Annotations	holds key design intent requirements, detailed instructions for individual objects, and can easily be identified on the canvas.

Design Intent Toolbar

The design intent toolbar contains the following icons, which are available in both write and read-only modes:

Icon	Command	Lets you...
	*Annotation Show/Hide*	Toggle the display of the annotation shape on the canvas. For more details, see Annotations. **Note:** This is not available in Layout XL.
	*Glyph Show/ Hide*	Toggle the display of the annotation name and glyphs on the canvas. For more details, see Glyphs.
	*Halo Show/ Hide*	Toggle the halo functionality in the Navigator and on the canvas. For more details, see Haloing.
	*Delete all DI in Cellview*	Remove all the design intent from the cellview. For more details, see Deleting Design Intent on Objects. **Note:** This option is not available in Layout XL.
	*Report*	Generate a high level report from Layout XL or Schematics XL to report on the design intent within a design. For more details, see Running Design Intent Reports.
	*Sync All Design Intent*	Enabled only if there are design intent changes to be synced, syncs all the design intent between the schematic and layout cellview. The tooltip lists the individual design intents that shall be synced. For more details, see Syncing Design Intent.

Interacting with Design Intent in the Navigator Assistant

The Navigator shows a hierarchical representation of all the devices, nets, and pins in the design. The design intent within a cellview can be viewed using the following categories in the *Summary* pane of the Navigator:

- *Objects* lists each object in the design. Each object associated with a design intent has an overlay over its normal icon.

  When you select an object in the Navigator, the object is highlighted on the canvas in both applications. It also halos any objects that are also members of the same design intent.

- *Design Intent* lists each design intent within the cellview, grouped by set. When a set is expanded, each design intent within the cellview for that category is listed. An icon is associated with each category of design intent.

  Selecting a set, opens the *Details* pane where each design intent within the cellview for that category is listed. Immediately below this, a hierarchical representation of the instances where those design intents are associated is listed. Selecting the set *All*, lists a hierarchical representation of all the design intent within the cellview.

  When you select a design intent in the Navigator, all the associated objects are haloed in Schematics XL and Layout XL.

The *Design Intent* column is a resizable and sortable column displayed by right-clicking over the column header area. The column displays the implementation status of each Design Intent. If displaying *Objects* in the Navigator, the column also displays the name of the design intent. If no design intents are associated with an object, the column is left blank.

More information on design intent features is available in the following sections:

- Design Intent Icons in Navigator
  - Implementation Status of Design Intent
- Context-Sensitive Menu Options in Navigator
- Tooltips on the Navigator

For general information on the functionality of the Navigator Assistant, refer to *Virtuoso Schematic Editor User Guide*.

Design Intent Icons in Navigator

For the *Design Intent* category, the Navigator tree displays the associated design intent icon for the category of each design intent.


Cell DI


Device DI


Net DI


Pin DI


High Current DI

Max Voltage Drop DI

In the Navigator tree for the *Objects* category, any objects associated with a design intent have a design intent overlay displayed on their object icon. If a design intent requires to be synchronized, the design intent overlay is replaced with the Sync icon  .

	*DI applied*	A design intent has been associated with the object. On creating a design intent, the member objects of the design intent are displayed with a design intent overlay. The color of the overlay is the same used to identify the annotation on the canvas and is set as part of the *Style* when Creating Design Intents.
	*Sync*	The design intent associated with the object has been amended and changes require to be synced between the schematic and layout cellviews.

Implementation Status of Design Intent

The following icons displayed in the *Design Intent* column indicate the current implementation status for the design intent. The implementation status is updated by the layout designer in the Create Design Intent form.

	*none*	No progress has been reported for the associated design intent in Layout XL.
	*review*	This status type is automatically associated when a design intent is created or an object is added or removed from a design intent in Layout XL. Design intents that have been created or amended in this way require to be reviewed by the schematic designer and validated before they are implemented into the design. For more details, see Reviewing and Editing a Design Intent Created in Layout XL.
	*in-progress*	Work is in progress for the associated design intent.
	*issues*	There is an issue regarding the implementation of the associated design intent.
	*failed*	The design intent cannot be implemented.

	*complete*	The design intent has been implemented into the design and is ready to be checked and signed off in Schematics XL. For more details, see Verifying and Signing Off a Design Intent.

Note: Hovering over any of the status icons displays a tooltip containing details of the design intent.

Context-Sensitive Menu Options in Navigator

A context-sensitive menu is accessed by right-clicking anywhere on the Navigator and choosing *Design Intent*. The menu options displayed vary depending on the selections made on the Navigator from which the menu is called.

The options are as follows:

Create Design Intent	Create a new design intent on the selected objects. For more details, see Creating Design Intents.
Edit Design Intent*	View/amend existing design intents associated with the selected objects using the Edit Design Intent form. For more details, see Editing a Design Intent.
Add to Design Intent*	Add the selected objects to an existing design intent. For more details, see Adding Objects to a Design Intent.
Remove Object from Design Intent*	Remove the selected objects from a design intent. For more details, see Removing Objects from Design Intents.
Delete Design Intent*	Delete the selected design intent by removing it from all associated objects. For more details, see Deleting a Design Intent. **Note:** This option is disabled in Layout XL. In Schematics XL, when multiple objects are selected that are associated with multiple design intents, the submenu option *Delete All Design Intent on Selected Objects* is also displayed. This removes all the associated design intents from the selected objects. For more details, see Deleting a Design Intent.

Delete All Design Intent in Cellview	Delete all design intent from the cellview. For more details, see Deleting All Design Intent in Cellview. This option is also available by right-clicking anywhere on the canvas in Schematics XL only.
Sync...*	**Sync –** if the objects selected in the Navigator have design intents associated that require to be synced, the individual design intents are listed with the prefix “Sync –”. Only the design intent for the selected menu item is synced between the schematic and layout cellviews. For more details, see Syncing a single design intent. **Sync All Selected Design Intent** when the objects selected in the Navigator have multiple design intents associated that require to be synced, the option to *Sync All Selected Design Intent* is displayed. Only the design intents associated with the selected objects are synced between the schematic and layout cellviews. **Sync All Design Intent** syncs all the design intent in the design between the schematic and layout cellviews. For more details, see Syncing Design Intent.

* If the selected object has multiple design intent options available, a slider option is displayed providing access to each design intent.

Selecting a design intent in the Navigator halos all the members of that design intent on the canvas.

Tooltips on the Navigator

Hovering the cursor over the *Name* column in Navigator displays a tooltip containing general information on the object. Hovering the cursor over the *Design Intent* column displays a tooltip showing only the associated design intent information for that object, including the status of implementation.

Interacting with Design Intent on the Canvas

On creating a design intent, in addition to the features added in the Navigator, a design intent annotation is created on the canvas as an easily identifiable name, glyph and shape. Each annotation holds key design goals, instructions, implementation plans and status for associated objects.

More information on design intent features is available in the following sections:

- Context-Sensitive Menu Options on Canvas
- Annotations
  - Glyphs
- Design Intent Info Balloon

Context-Sensitive Menu Options on Canvas

A context-sensitive menu is accessed by right-clicking anywhere on the canvas and choosing *Design Intent*. The menu options displayed vary depending on where on the canvas the menu is called from, and the selections that are currently made.

Right-clicking on an annotation's glyph, displays a menu with options available specific to that design intent and the selections that are currently made.

The options are as follows:

Create Design Intent	Create a new design intent on the selected objects. For more details, see Creating Design Intents.
Edit Design Intent*	View/amend existing design intents associated with the selected objects using the Edit Design Intent form. For more details, see Editing a Design Intent. ■ This option is also available when right-clicking on a design intent's glyph.
Add to Design Intent*	Add the selected objects to an existing design intent. For more details, see Adding Objects to a Design Intent. ■ This option is also available by selecting an object and right-clicking on the required design intent's glyph.
Remove Object from Design Intent*	Remove the selected objects from a design intent. For more details, see Removing Objects from Design Intents. ■ This option is also available by selecting an object and right-clicking on the required design intent's glyph.

Delete Design Intent*

Delete the selected design intent by removing it from all associated objects. For more details, see Deleting a Design Intent.

- This option is also available when right-clicking on a design intent's glyph.
- In Layout XL, this option is only available by right-clicking on the required design intent's glyph.

In Schematics XL, when multiple objects are selected that are associated with multiple design intents, the submenu option *Delete All Design Intent on Selected Objects* is also displayed. This removes all the associated design intents from the selected objects. For more details, see Deleting a Design Intent.

Delete All Design Intent in Cellview

Delete all design intent from the cellview. For more details, see Deleting All Design Intent in Cellview.

- This option is also available by right-clicking anywhere on the canvas in Schematics XL only.

Sync...

Sync – if the objects selected in the canvas have design intents associated that require to be synced, the individual design intents are listed with the prefix "Sync –*"*. Only the design intent for the selected menu item is synced between the schematic and layout cellviews. For more details, see Syncing a single design intent.

Sync All Selected Design Intent when the objects selected in the canvas have multiple design intents associated that require to be synced, the option to *Sync All Selected Design Intent* is displayed. Only the design intents associated with the selected objects are synced between the schematic and layout cellviews.

Sync All Design Intent syncs all the design intent in the design between the schematic and layout cellviews. For more details, see Syncing Design Intent.

* If the selected object has multiple design intent options available, a slider option is displayed providing access to each design intent

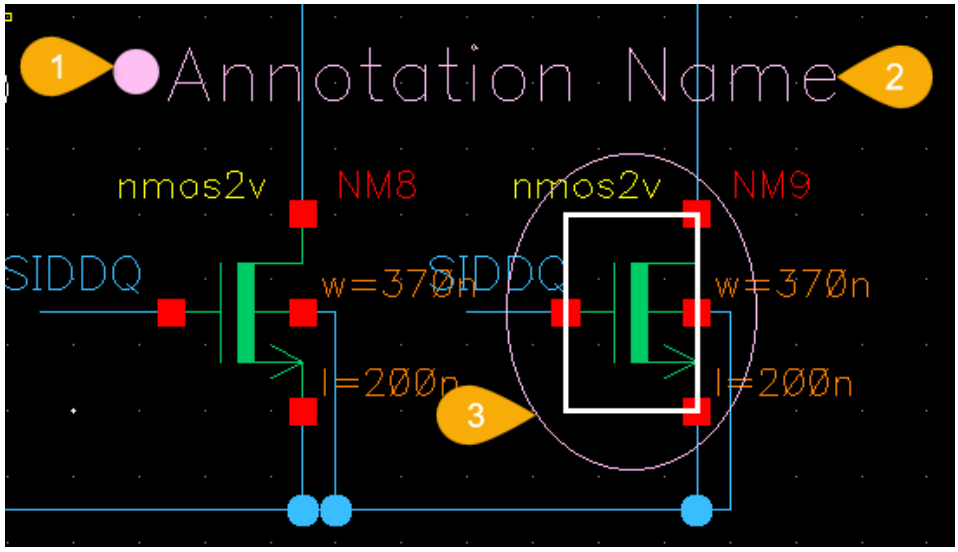
Note: With the exception of *Edit Design Intent*, these options are all disabled in Schematic read-only mode. The *Edit Design Intent* option is enabled but the form displayed is read-only.

In Layout read-only mode, only *Sync All Design Intent* and *Edit Design Intent* are available. The *Edit Design Intent* option is enabled but the form displayed is read-only.

Annotations

Annotations (notes) are created and edited directly on the design to be shared across the schematic and layout canvases. Each annotation can hold key design intent requirements, detailed instructions for individual objects, and can easily be identified on the canvas.

An annotation on the canvas is displayed as a glyph, annotation label, and shape in Schematics XL and as a glyph in Layout XL:



1. Glyph — Symbol on the canvas used to denote a design intent annotation. They are displayed as large dots beside each design intent annotation label.
2. Annotation Label — Label displaying the unique annotation identifier.
3. Annotation Shape — Outline of the design intent on the canvas. The shape can outline the individual members or be a single bounding box around all the member objects.

The display of annotation items can be toggled using the relevant button on the Design Intent Toolbar.

Design intent annotations have customizable style options to make them distinguishable from other design intents on the canvas. The style options are set and edited on the Create Design Intent form and the color selected is used for each element of the design intent across the schematic and layout canvases. The design intent name, font, and font size of each annotation label can also be amended using the Property Editor or the Edit Object Properties form. This excludes annotation labels for design intent categories *High Current* or *Max Voltage Drop* which display the object name rather than the design intent name so they cannot be amended in this way.

In Schematics XL, the position of the glyph and label can be moved as required to improve visibility on the canvas. For more information on annotation functionality, see Interacting with Design Intent on the Canvas.

The display of the annotation items can be toggled using the buttons on the Design Intent Toolbar.

Glyphs

The glyph color is defined in the *Style* section of the Create Design Intent form. In Schematics XL, the size of the glyph is controlled by the design intent's font size which is also specified in the Create Design Intent form. In Layout XL, it is fixed to a size proportional to the size of the device with which the design intent is associated with and so is unable to be adjusted.

Clicking a glyph displays a Design Intent Info Balloon which displays details of the design intent. The design intent info balloon also has an *Edit* button which directly opens the Edit Design Intent form where you can edit the design intent.

Right-clicking on a glyph displays a context-sensitive menu. The options available vary depending on the selections that are currently made. For more details, see Interacting with Design Intent on the Canvas.

Hovering on a glyph halos all the members of that design intent in the Navigator and on the canvas in Schematics XL and Layout XL and also in the *Constraint Manager* assistant in

Layout XL. Haloing on a glyph halos the design intent members even if haloing has been turned off using the Design Intent Toolbar.

Design Intent Info Balloon

Design intent info balloons are accessed by clicking on a glyph and they display directly on the canvas.

Each info balloon contains details of the design intent including the profile properties and notes. In Layout XL, the *Implementation Status* and *Notes* are also displayed.

Clicking *Edit*, opens the Edit Design Intent form where you can edit the design intent or update its implementation status.

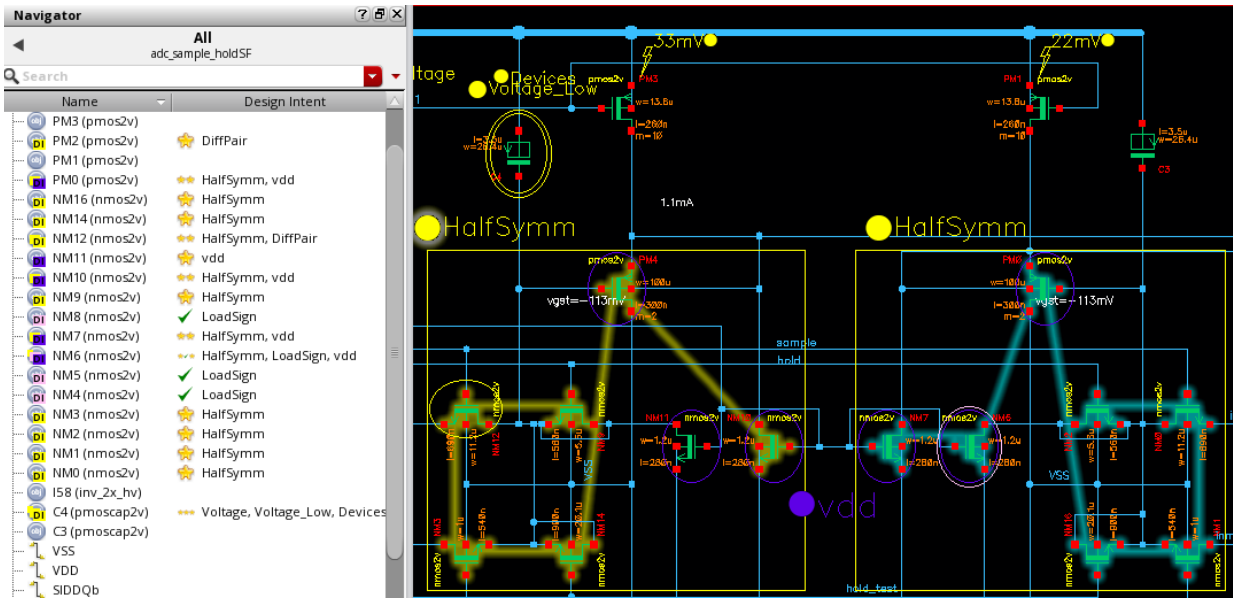
You can pin and reposition an info balloon as needed.

Haloing

Selecting a design intent or object in the Navigator or on the canvas, or hovering on a glyph, halos its members in the Navigator and on the canvas in Schematics XL and Layout XL, and in the *Constraint Manager* assistant in Layout XL.

Selecting multiple items in the *Constraint Manager* results in multiple design intent members being haloed in the canvas simultaneously. A different color is used for each design intent.

Haloing is controlled by toggling the button  on the Design Intent Toolbar for each application.



Note: Hovering on a design intent glyph continues to halo the design intent members even if the halo visibility has been switched off.

Syncing

For schematic and layout designers to work collaboratively on a shared design, it is crucial that the design intents in the schematic and layout views of a design are kept in sync. Regular syncing creates a seamless flow between Schematics XL and Layout XL and enables schematic and layout designers to communicate with each other to produce designs without unwanted electrical side-effects.

You can either sync an individual design intent or all design intent to update any design intent changes in the schematic and layout views to ensure that both systems are synced. See Syncing Design Intent for more details.

The following types of update result in the changes being marked in the other system with a Sync icon appearing in the Navigator beside each of the affected design intents and the Sync button on the Design Intent toolbar becoming enabled.

- Changing annotation colors
- Updating Profile or property settings?
- Adding design notes in the Edit Design Intent form
- Adding or removing design intent members

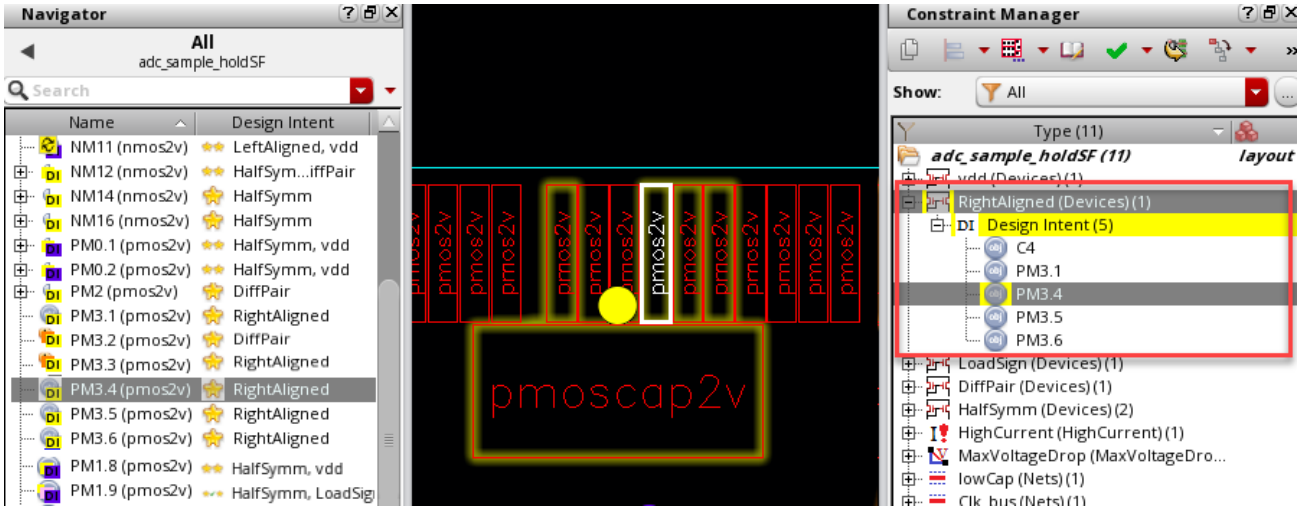
For more details, see Syncing Between Schematic and Layout Views.

Constraint Manager

Design Intent allows schematic designers to capture their design goals in Schematics XL without actually creating constraints. The design intent is transferred to Layout XL, where the layout designer is responsible for physically implementing the design intent in the design. The responsibility for creating constraints moves from Schematics XL to Layout XL.

Important

Creating the constraints directly in Layout XL means that constraints are stored in the layout view. To ensure that a constraint view is not created when syncing from layout to schematic, you must ensure that all the constraints on a design are physical constraints. This ensures that device correspondence information is maintained during the transfer.



Design intents are grouped together in the Constraints Manager. You can create constraints on the layout view using the *Constraint Manager* assistant. The constraints created are supported by the various physical design applications. For more details, see Adding a Physical Constraint to an Existing Design Intent.

Selecting a design intent in the *Constraint Manager* halos the design intent on the canvas across both Layout XL and Schematics XL.

For more information on how to use constraints and the *Constraint Manager*, see the Constraint Manager Assistant in the Virtuoso Unified Custom Constraints User Guide.

2

Working with Design Intent

The following functionality is covered in this section:

- Creating Design Intents
- Syncing Design Intent
- Editing a Design Intent
- Adding Objects to a Design Intent
- Copying Objects with Design Intents
- Removing Objects from Design Intents
- Deleting a Design Intent
- Generating Reports on Design Intent
- Verifying and Signing Off a Design Intent

Creating Design Intents

You can create a design intent using the Create Design Intent form. Access the form by right-clicking the required object in the Navigator or on the canvas and choose *Design Intent – Create Design Intent*.

The type of object selected in the Navigator or on the canvas determines the type of design intent that can be created. The design intent categories are *Cell DI*, *Device DI*, *Net DI*, *Pin DI*, *High Current DI*, and *Max Voltage Drop DI*.

	Cell	Device	Net	Pin	HighCurrent	MaxVoltageDrop
Cell	•					
Instance		•				
Net			•		•	•
Pin				•	•	•
InstTerm					•	•

Note:

- The Net category can also be used for global nets.
- The categories HighCurrent and MaxVoltageDrop can both be created for an instance terminal, but an instance terminal can only be associated with one from each category.

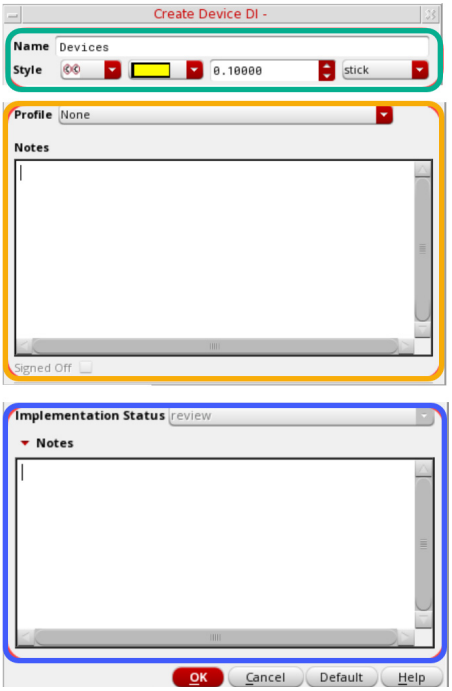
Each category type has a design intent icon associated with it. For more details, see Design Intent Icons in Navigator.

The following sections provide information about creating design intents:

- Sections within the Create/Edit Design Intent form
- Creating a Design Intent
  - Creating a HighCurrent Design Intent
  - Creating a HighCurrent Design Intent for Multiple-bit Instances
  - Creating a MaxVoltageDrop Design Intent
  - Creating a Cell Design Intent for a Cellview
  - Creating a Device Design Intent for Half Cells
  - Creating a Design Intent in Layout XL
  - Creating Design Intents in the Constraint Manager

Sections within the Create/Edit Design Intent form

The Create Design Intent form and the Edit Design Intent form are divided into three sections. The upper and middle sections of the form are visible in both applications, but only the upper section can be completed in Schematics XL and Layout XL.



The upper section lets you provide a unique identifier for the design intent and defines a style so that it can be identified on the canvas and distinguished from other design intents. This section is available in Schematics XL if creating or editing a design intent, but it is only available in Layout XL when creating a design intent.

The middle section enables schematic designers to capture the requirements for the design intent using notes and properties. This section of the form is only editable in Schematics XL.

The lower section enables the layout designer to capture their implementation strategy and feedback progress and status to the schematic designer. This part of the form is only editable in Layout XL.

Note: The lower section of the form is not visible on the Create Design Intent form in Schematics XL.

The notes recorded in the middle and lower sections of the form capture the comments, queries, and responses between the schematic and layout designers to develop a tracked conversation around the specification and implementation of the design intent. The design notes (Schematics XL) are initially used to record the design goals, and the implementation

notes (Layout XL) are used to add notes regarding the implementation. During the implementation, any issues or queries can be recorded in the implementation notes and the responses from the schematic designer recorded in the design notes. The communication is useful as a record and the information entered is used within Generating Reports on Design Intent.

Note: The information entered in the Create/Edit Design Intent forms is also included in the tooltip displayed when you hover in the Navigator and in the info balloon displayed when you click the glyph on the canvas in Schematics XL and Layout XL.

Creating a Design Intent

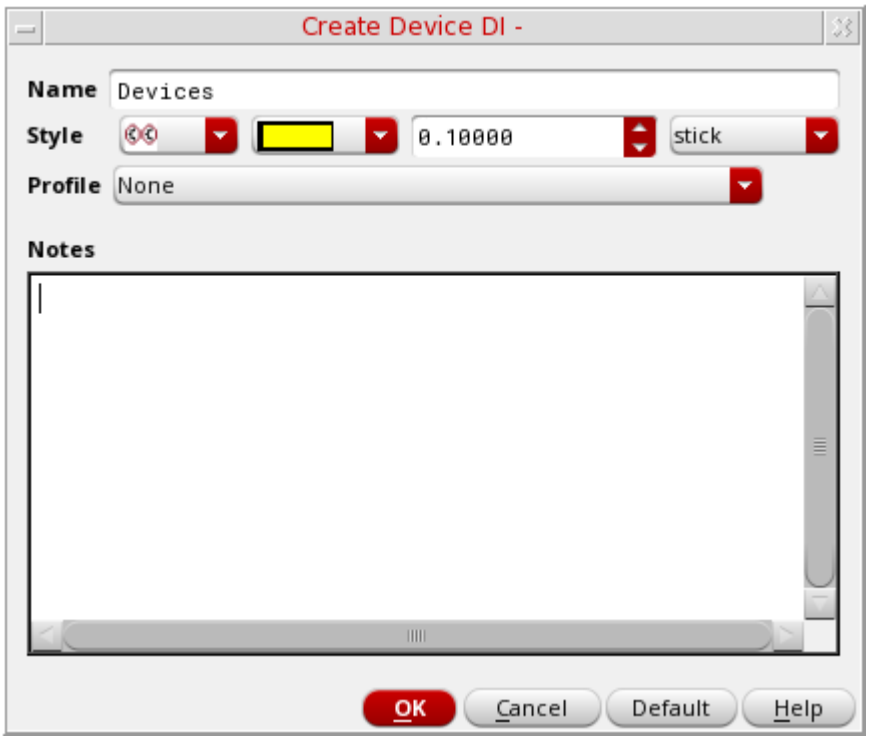
Note: The options available on the Create Design Intent form differ between Schematics XL and Layout XL. To create a new design intent in Layout XL, see Creating Design Intents in Layout XL.

To create a design intent in Schematics XL:

1. Right-click one or more objects in the Navigator or on the canvas and choose *Design Intent – Create Design Intent*.

The type of object selected determines the design intent options available. A context-sensitive submenu is displayed for those objects with multiple design intent categories.

The Create Design Intent form is displayed.

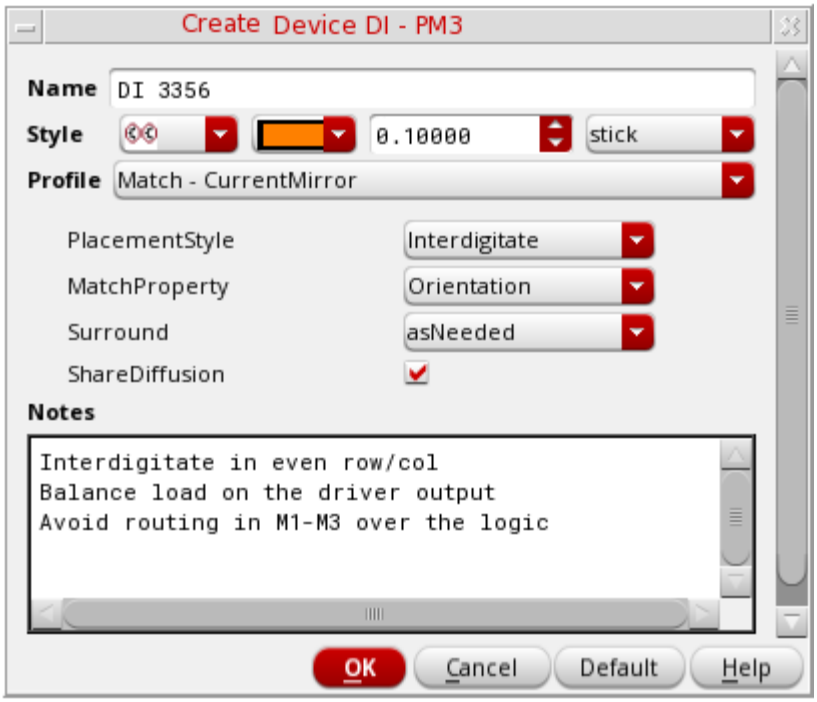


Note: The options displayed on the form vary to the type of object, category and profile selected.

2. Specify a *Name* for the design intent.
3. Define a *Style* to identify the design intent annotation and distinguish it from other design intents by selecting a shape, color, font size and font. The selections made identify the design intent annotation on the canvas and are reflected on the associated design intent icons in the Navigator.
4. (Optional) In Schematics XL, define property requirements by selecting a pre-configured *Profile*. Additional selection options are displayed specific to the profile selected.

   If displayed, use the *Show/Hide Disabled* button to toggle the display of the selected property options.

5. In Schematics XL, define property requirements by in the *Notes*. Add any additional notes for the layout designer about the purpose of the design intent, its properties and to record any special considerations for implementation.



6. Click *OK*.

   An annotation is displayed on the canvas and in the Navigator, the object icon is displayed with a design intent icon DI overlaid and a design intent annotation highlights the object on the canvas.

   In Layout XL, the affected objects are immediately displayed with a sync icon as a prompt to Sync Design Intent.

The function `ciTemplateCreateDI` can also be used to create a design intent using SKILL. For more details about the SKILL functions available for Design Intent, see Design Intent Functions in *Virtuoso Unified Custom Constraints SKILL Reference*.

Note: Global nets in schematics differ to normal nets as they make connections across the design hierarchy without using pins and wires. When a *Net*s design intent is created for a global net, the global net and the new design intent annotation are automatically created on the canvas at the top level of the design in Schematics XL, regardless of where in the hierarchy the global net is located. When the layout view is generated, pins are created in Layout XL for the global nets in the schematic. For more details, see Pins and Pin Names in the *Virtuoso Layout Suite XL User Guide*.

Creating a HighCurrent Design Intent

To create a design intent for a net, pin, or instance terminal with currents:

1. Right-click a net, pin, or instance terminal in the Navigator or on the canvas and choose *Design Intent – Create Design Intent – HighCurrent*.

   Note: An instance terminal can only be associated with a single HighCurrent design intent.

2. Specify a name for the design intent.

   For the category *HighCurrent*, the name given for the design intent is only displayed in the Navigator. On the canvas, the current value is displayed as the annotation for each member instance terminal, for example `5A`. The annotation for the reference terminal has an additional `(REF)` to signify that it is the reference terminal. For example `5A(REF)`.

3. In the *Style* field, *Arrow* is preselected for the category *HighCurrent*. If required, specify a color, font size, and font to identify the design intent annotation on the canvas.

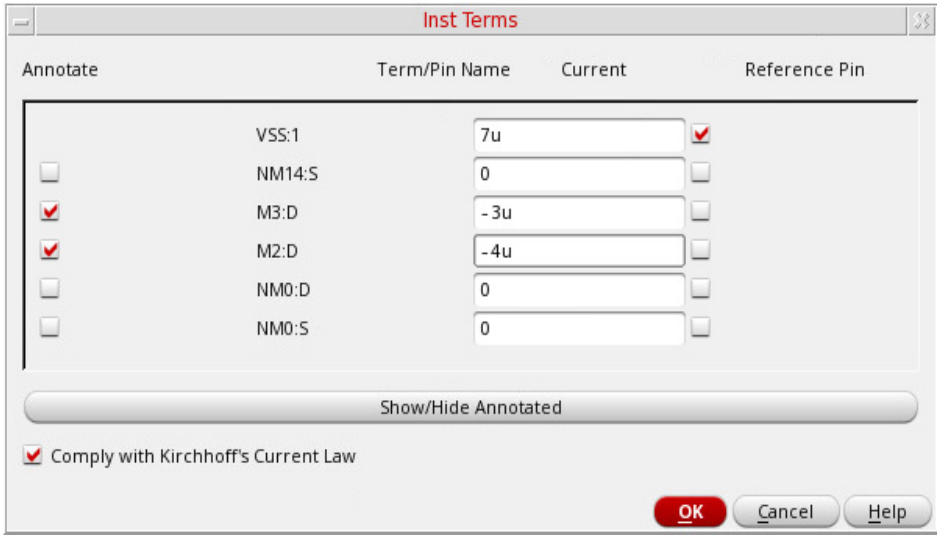
   Note: *Current* is the only profile available for the HighCurrent design intent category and must not be overridden with a user-defined profile.

4. Define the requirements for the following pre-defined profile properties:

   - *Reference (I)*: the total current on a net.
   - *Reference Pin*: the pin through which the current flows in or out.

5. (Optional) Click *Setup Pin/InstTerm Current*s.

The Inst Terms dialog box opens. It lists all instance terminals available on the net and lets you define the current for each instance terminal to be sourced.



6. (Optional) Select the *Annotate* check box to create an annotation and a glyph on the canvas for the required terminals. The current value is used as the annotation for each terminal.
7. (Optional) Click *Show/Hide Annotated* to toggle the display of the instance terminals selected for annotation in the Inst Terms dialog box.
8. (Optional) Select the *Comply with Kirchhoff's Current Law* check box to ensure that the sum of all currents entering and exiting a node is zero.

   For example, to comply when `7u` is the sum of all currents, the reference current must be `-7u` to give a total current of `0`. Conversely, if the sum of all currents is `-3u`, the reference current must be `3u`.
9. Click *OK*.

   If you attempt to close the Inst Terms dialog box with the *Comply with Kirchhoff's Current Law* check box selected and the total current does not equal zero, a warning message is displayed.

   - Click *OK* to automatically adjust the value of the reference current to match the sum of the nodes for the reverse current (`+` or `-`).
   - Click *Cancel* to manually amend the values and distribute the remaining current.
10. In the Create Design Intent form add any notes to assist in the implementation of the design intent and click *OK*.

A HighCurrent design intent is created on the canvas, and the annotation label displays the total current.

Important

When the design intent is transferred from Schematics XL to Layout XL, the callback `ciDiPostTransferHighCurrent` is run to split the current evenly between the mfactored members in Layout XL.

Related Topics

Create Design Intent (form reference)

Edit Design Intent (form reference)

Creating a HighCurrent Design Intent for Multiple-bit Instances

Modifying Sample Profiles and Property Definitions

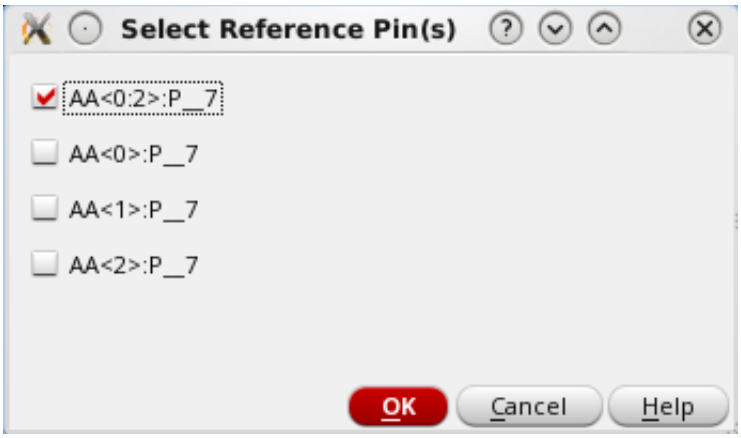
Creating a HighCurrent Design Intent for Multiple-bit Instances

Note: An instance terminal can be associated only with a single HighCurrent design intent.

To create a design intent for a multiple-bit pin or instance, or for a single-bit instance with currents connected to a multiple-bit instance terminal or pin:

1. Right-click the net, pin, or instance terminal in the Navigator or on the canvas and choose *Design Intent – Create Design Intent – HighCurrent*.

If you are creating the design intent for a multiple-bit pin connected to multiple-bit instance terminals, the Select Reference Pin(s) form appears listing the pins that are available for selection.

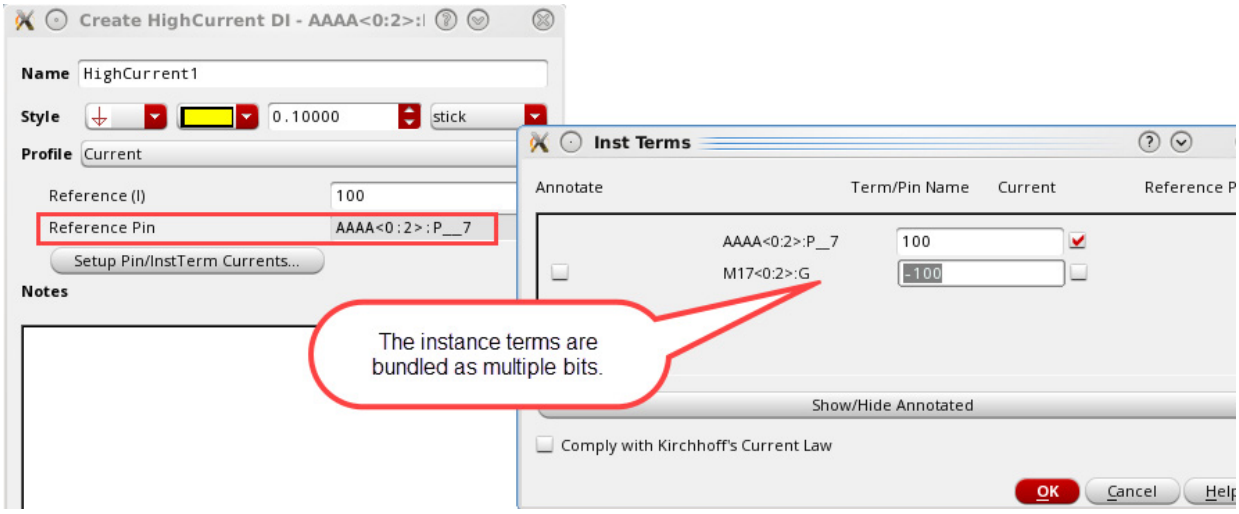


2. Select whether the reference pin for the design intent is to be the bundled multiple-bit pin or a single-bit pin and click *OK*.
3. Specify a name and style for the design intent.
4. In the *Profile* field, *Current* is the only profile available for the HighCurrent design intent category and must not be overridden with a user-defined profile. Specify the requirements for the following pre-defined properties:
   - *Reference (I)*: the total current on a net.
   - *Reference Pin*: the pin through which the current flows in or out.
5. Click *Setup Pin/InstTerm Current*s.

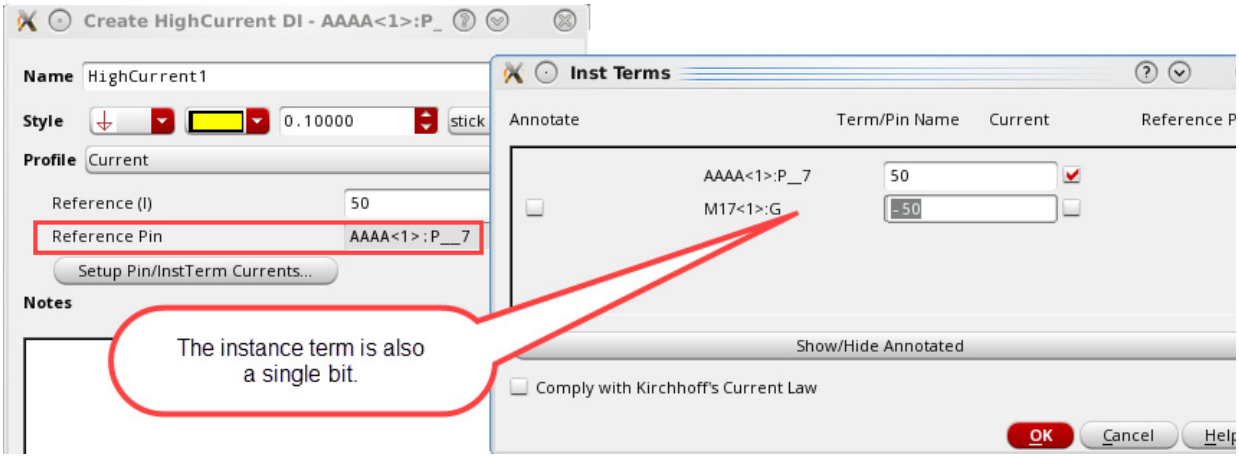
   The Inst Terms dialog box opens. It lists all instance terminals available on the net and lets you define the current for each instance terminal to be sourced.
6. Enter the current value for the required instance terminals or pins.

   For multiple-bit pins connected to multiple-bit instance terminals, the terminals listed are determined by what you selected on the Select Reference Pin(s) form. If you specified that the reference pin is a multiple-bit instance, the instance terms are bundled as multiple bits. The current value specified applies equally to each instance term within the

bundle. A current of `100uA` would be allocated to each of the instance terms `M17<0>:G`, `M17<1>:G`, and `M17<2>:G`.

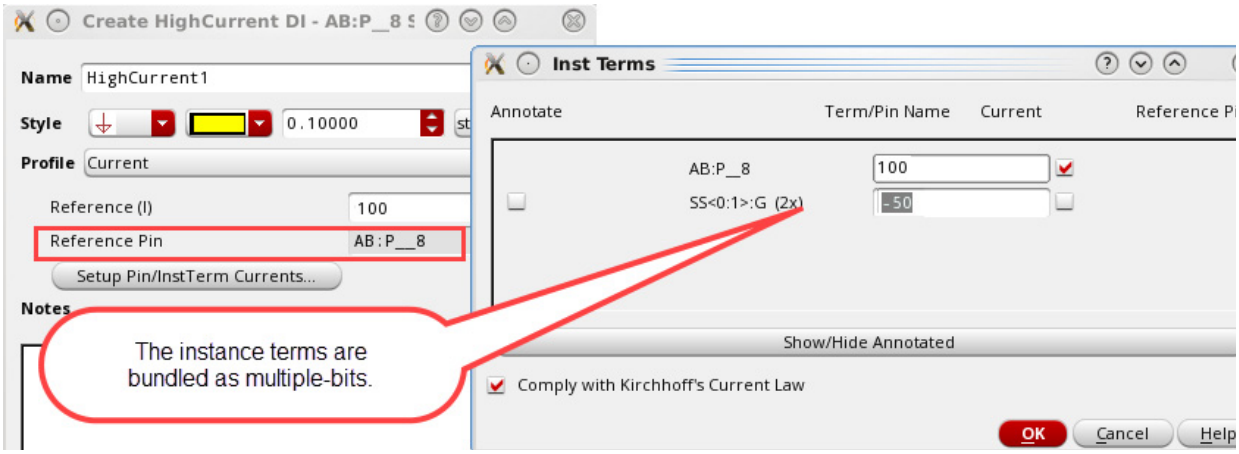


If you specified that the reference pin is a single-bit instance, the instance term is also single bit. The *Current* value applies to the single-instance term.



For single-bit pins connected to multiple-bit instance terminals, the reference pin is the single-bit pin. The instance terms are bundled as multiple bits. The *Current* value you

specify is shared between each instance term within the bundle. A current of `100uA` would be allocated as `50uA` to `SS<0>:G` and `50uA` to `SS<1>:G`.



7. (Optional) Select the *Annotate* check box to create an annotation and a glyph on the canvas for the required terminals. The current value is used as the annotation for each terminal.

8. (Optional) Click *Show/Hide Annotated* to toggle the display of the instance terminals selected for annotation in the Inst Terms dialog box.

9. (Optional) Select the *Comply with Kirchhoff's Current Law* check box to ensure that the sum of all currents entering and exiting a node is zero.

   For example, to comply when `7u` is the sum of all currents, the reference current must be `-7u` to give a total current of `0`. Conversely, if the sum of all currents is `-3u`, the reference current must be `3u`.

10. Click *OK*.

    If you attempt to close the Inst Terms dialog box with the *Comply with Kirchhoff's Current Law* check box selected and the total current does not equal zero, a warning message is displayed.

    - Click *OK* to automatically adjust the value of the reference current to match the sum of the nodes for the reverse current (`+` or `-`).
    - Click *Cancel* to manually amend the values and distribute the remaining current.

11. In the Create Design Intent form add any notes to assist in the implementation of the design intent and click *OK*.

    A HighCurrent design intent is created on the canvas, and the annotation label displays the total current.

Related Topics

Create Design Intent (form reference)

Edit Design Intent (form reference)

Creating a HighCurrent Design Intent

Modifying Sample Profiles and Property Definitions

Creating a MaxVoltageDrop Design Intent

To apply a design intent to a net, pin, or instance terminal with voltages:

Note: An instance terminal can only be associated with a single MaxVoltageDrop design intent.

1. Right-click a net, pin, or instance terminal in the Navigator or on the canvas and choose *Design Intent – Create Design Intent – MaxVoltageDrop*.
2. Specify a *Name* for the design intent.

   Note: For the category *MaxVoltageDrop*, the design intent name specified is only displayed in the Navigator. On the canvas, the voltage is displayed as the annotation for each member pin/instance terminal, for example `12V`. The annotation for the reference pin has an additional `(REF)` to signify that it is the reference pin/terminal, for example `12V(REF)`.
3. In *Style*, the *Lightning Flash* is preselected for the category *MaxVoltageDrop*. If required, specify a color, font size, and font to identify the design intent annotation on the canvas.
4. (Optional) In *Profile*, *Voltage* is the only profile available for this design intent category and must not be overridden. For more details, see Modifying Sample Profiles and Property Definitions. The *Voltage* profile has the following properties:

   Max Voltage Drop defines the maximum voltage drop between the *Reference Pin* and the instance terminals/pins.

   Reference Pin is the pin from which the maximum voltage drop is measured by all the other instance terminals/pins on a net. To select a different reference pin or to amend the instance terminals/pins enabled, click the ...button to open the Inst Terms form. This form lists all the instance terminals/pins available on a net and if enabled, each is checked against the maximum voltage drop for the reference pin. To change the reference pin, select the *Reference* check box for the required instance terminal/pin.

You can specify different voltages for each instance terminal/pin using positive or negative variations, but if the voltage specified at *Max Voltage Drop* is amended, the voltages associated with the enabled instance terminals/pins are also overridden.

5. Add any additional notes to assist in the implementation of the design intent.
6. Click *OK*. A MaxVoltageDrop design intent is created on the canvas, the annotation label displays the name of the object.

Important

(ICADVM18.1 Only) When the design intent is transferred from Schematics XL to Layout XL, the callback `ciDiPostTransferMinMaxVoltage` is run to propagate the design intent properties Min Voltage, Max Voltage, Signal Type, Power Sensitivity, and Ground Sensitivity on the associated design intent objects.

Creating a Cell Design Intent for a Cellview

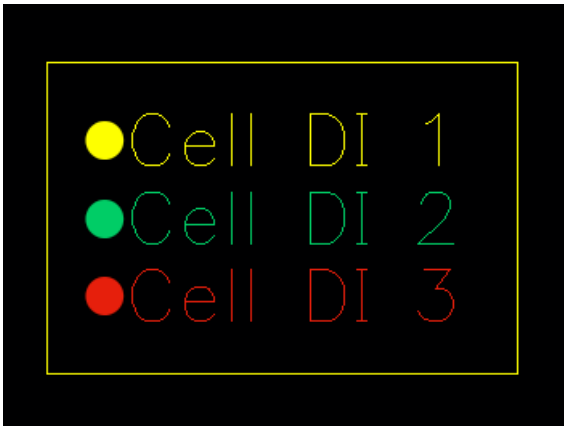
To create one or multiple *Cell* design intents to specify design instructions that are intended for the entire cellview, not just for individual objects:

1. Ensuring that no objects are selected, right-click the canvas, (or in the Navigator, right-click the cell) and choose *Design Intent – Create Design Intent - Cell*.

   The Create Design Intent form is displayed.
2. Specify a *Name* for the design intent.
3. Define a *Style* to identify and distinguish the design intent from other cell design intents by selecting a color, font size, and font. The default shape *Design Notes Legend* is already preselected.
4. (Optional) Define property requirements by selecting a pre-configured *Profile*. Additional selection options are displayed specific to the profile selected.

   If displayed, use the *Show/Hide Disabled* button to toggle the display of the selected property options.
5. Define property requirements in the *Notes* and add any additional notes for the layout designer about the purpose of the design intent, its properties and to record any special considerations for implementation.

6. Click *OK*.



The annotation for the new cell design intent is automatically created within a rectangular box positioned on the lower left of the canvas in Schematics XL.

This box can be repositioned as required. Any subsequent cell design intents created are also displayed within this box, wherever it is positioned.

Note: Cell design intents are not displayed on the Navigator in Schematics XL. In Layout XL, the new cell design intents are displayed on the Navigator and on the *Constraint Manager*.

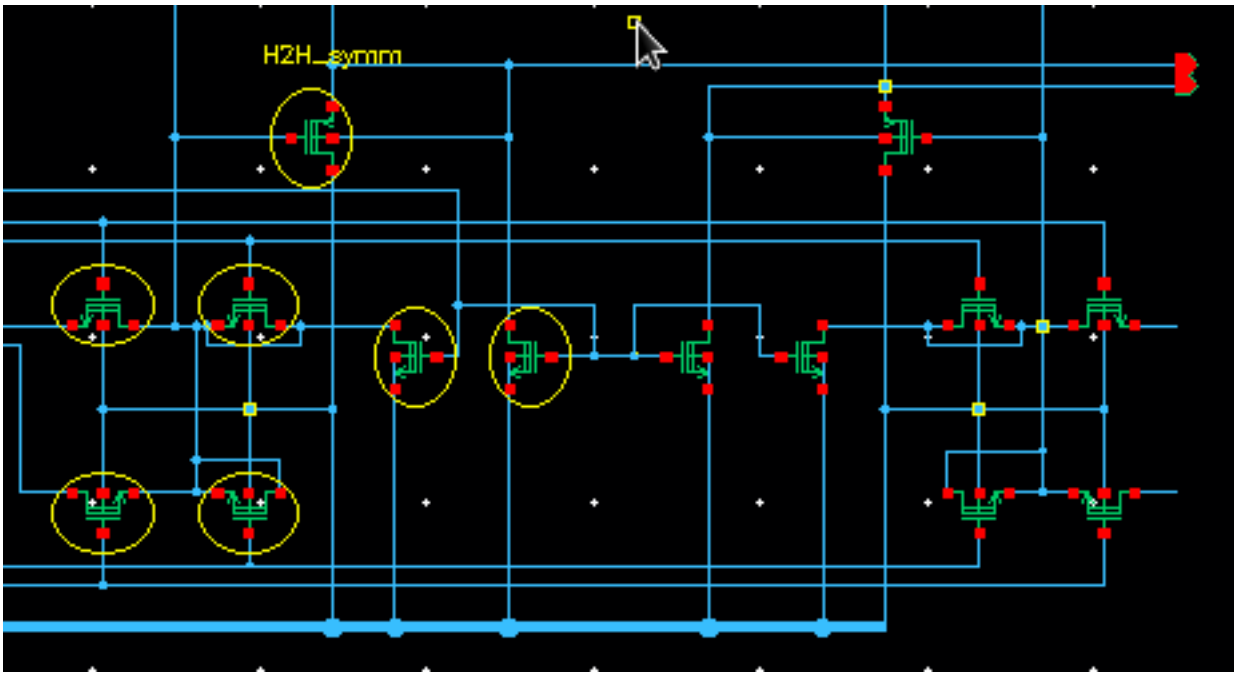
Creating a Device Design Intent for Half Cells

To apply a design intent to two groups of devices that should be laid out symmetrically:

1. Select the objects for the first half in the Navigator or on the canvas.
2. Right-click and choose *Design Intent – Create Design Intent – Devices*.
3. Specify a *Name* for the design intent.
4. Define a *Style* to identify the design intent annotation on the canvas.
5. Select the *Profile*, *Half to Half*.

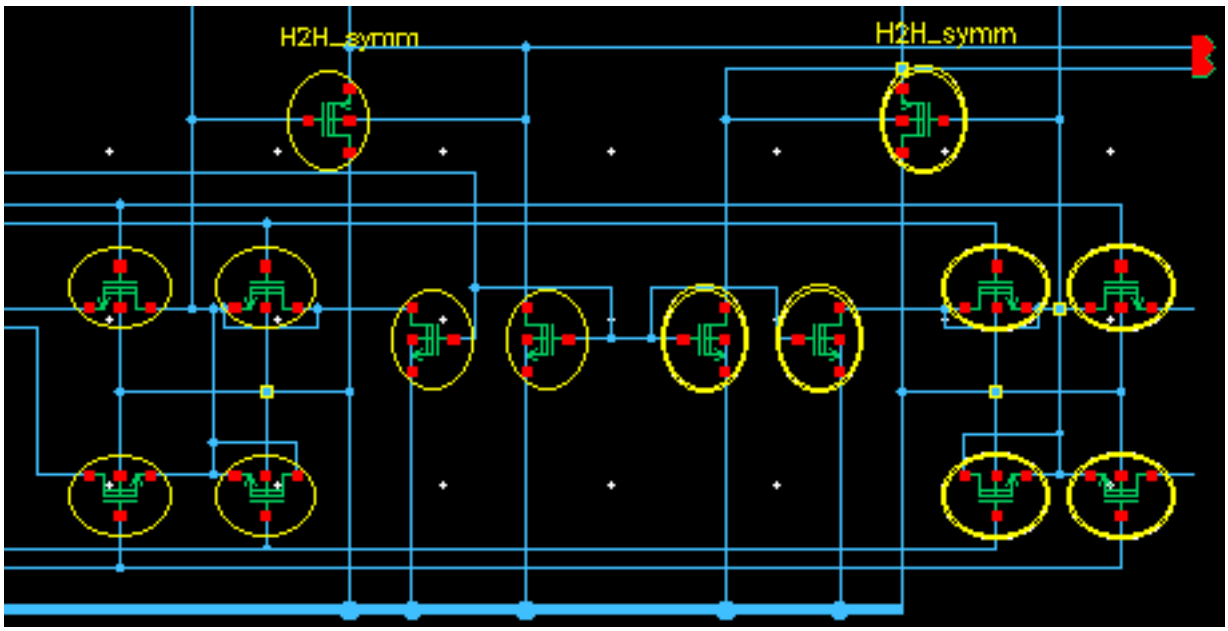
Additional selection options are displayed specific to the Half to Half profile.

6. Select and enter the property options for the design intent.
7. Add any additional notes to assist in the implementation of the design intent.
8. Click *OK*. A device design intent annotation is created on the canvas for the first half.



9. Select the objects for the second half.
10. Right-click and choose *Design Intent – Add Objects to Design Intent*.

11. Select the second half of the device to be created. Device design intent annotations are created on the canvas for the second half.



Note: Selecting an object on either half of the device creates halos over the corresponding objects in each half.

In Layout XL, the affected objects are immediately displayed with a sync icon as a prompt to Sync Design Intent.

Creating Design Intents in Layout XL

The process for creating design intents in Layout XL differs as all design intents must be reviewed and validated within Schematics XL before they are fully integrated into the standard implementation process.

The following sections provide information about creating design intents in Layout XL:

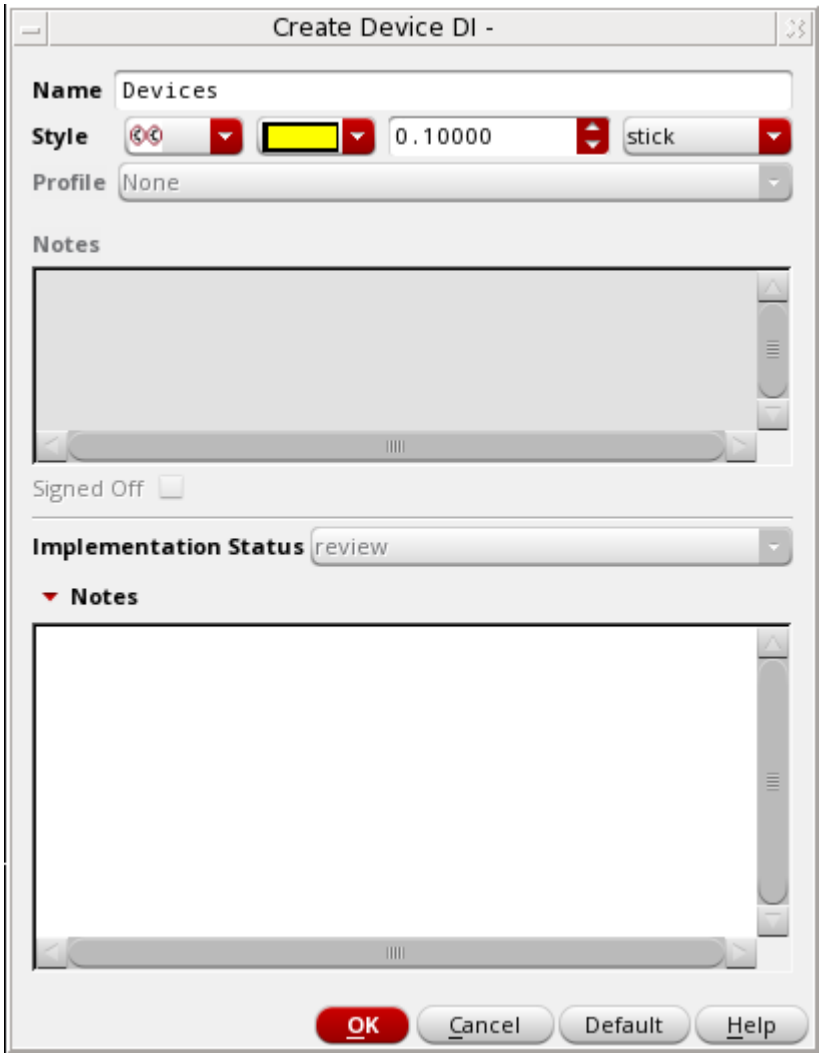
- Creating a Design Intent in Layout XL
- Creating Design Intents in the Constraint Manager

Creating a Design Intent in Layout XL

Note: The options available on the Create Design Intent form differ between Schematics XL and Layout XL. To create a new design intent in Schematics XL, see Creating a Design Intent.

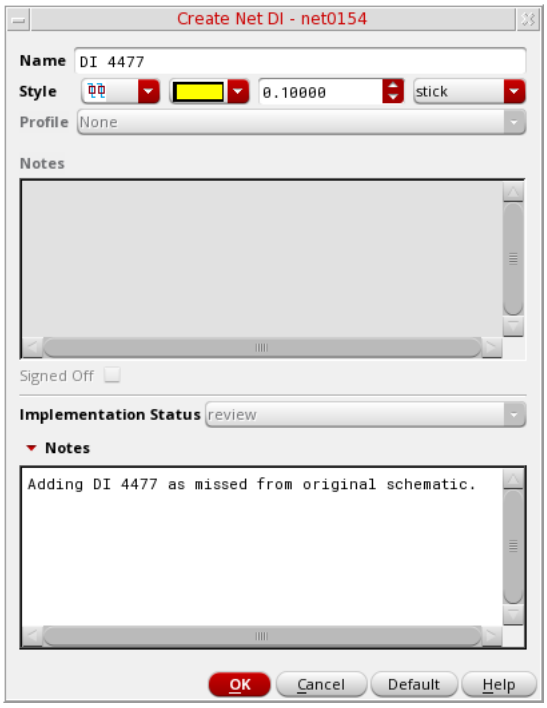
1. Select an instance in the Navigator or on the canvas in Layout XL.
2. Right-click and choose *Design Intent – Create Design Intent*.

The Create Design Intent form is displayed.



3. Specify a *Name* for the design intent.
4. Define a *Style* to identify the design intent annotation on the canvas.
5. In the lower section *Notes*, add any information regarding the implementation and the purpose of the design intent for the schematic designer. The *Notes* in the upper section

of the form are unavailable. An *Implementation Status* of `review` is set automatically and cannot be changed.



6. Click *OK*.

   In Schematics XL, the affected objects are immediately displayed with the `review` status icon and a sync icon as a prompt to Sync the design intent. For more details, see Reviewing and Editing a Design Intent Created in Layout XL.

Creating Design Intents in the Constraint Manager

Important

All design intents implemented using the constraint manager must have their constraints included in the design intent template (physical constraint). This ensures that when syncing design intent or transferring from Layout XL to Schematics XL using the *Update Layout Constraints* button, a constraint view is not created.

To create a design intent from the Constraint Manager:

1. In the *Constraint Manager* assistant in Layout XL, select the object (instance, net, pin, or terminal) for which you want to create the design intent.
2. From the *Constraint Creation* menu, choose the *Design Intent* submenu.

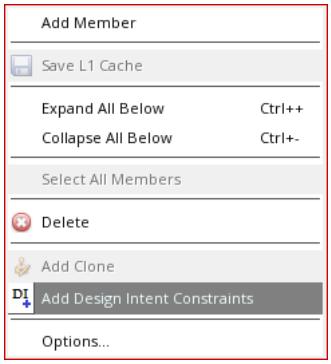
3. Choose the design intent category that is to be created for the selected object.

   Note: Only the categories that can be created for the current selected object are available.

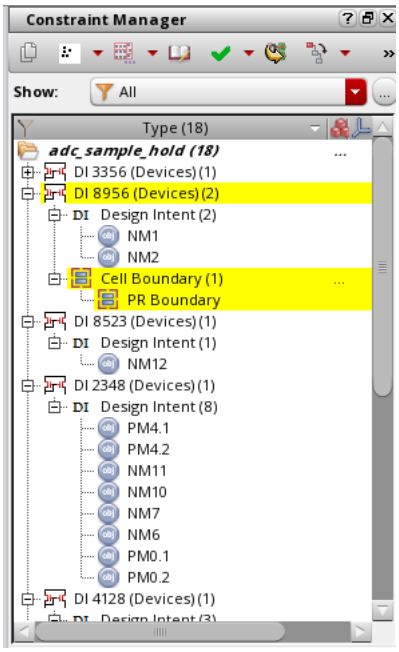
   The Create Design Intent form is displayed. For details on how to complete the form, see Creating a Design Intent (in Schematics XL) or Creating a Design Intent in Layout XL.

4. Select the object and the constraint to be added in the *Constraint Manager* assistant.

5. Right-click and from the menu displayed, choose *Design Intent – Add Design Intent Constraints*.



The constraint is added to the design intent and its associated objects.



Note: Design intents implemented using constraints do not generate a sync indication.

For more information on how to use constraints and the *Constraint Manager*, see the Constraint Manager Assistant in the Virtuoso Unified Custom Constraints User Guide.

Syncing Design Intent

Syncing updates design intent changes in the schematic and layout views and ensures that design intents in both systems are fully synchronized.

Important

To enable seamless syncing when working with design intents, the same up to date version of a cell must be open in both schematic and layout views. If working in Schematics XL, the corresponding layout view must also be open (read-only is acceptable) and vice versa if working in Layout XL.

The following sections provide information about syncing design intents:

- Checking Updates Before Syncing A Design Intent
- Syncing Between Schematic and Layout Views
  - Syncing in Layout XL
  - Syncing in Schematics XL
  - Syncing a single design intent
  - Syncing All the Design Intent
- Recognizing Amended Design Intents after Syncing

Checking Updates Before Syncing A Design Intent

Any design intent changes made on the schematic or layout views result in the changes being marked in the other system by a Sync icon  appearing in the Navigator beside each of the amended design intents. These icons are temporary and draw attention to those design intents that have been updated and that require to be synced. The *Sync* button on the Design Intent toolbar is also enabled.

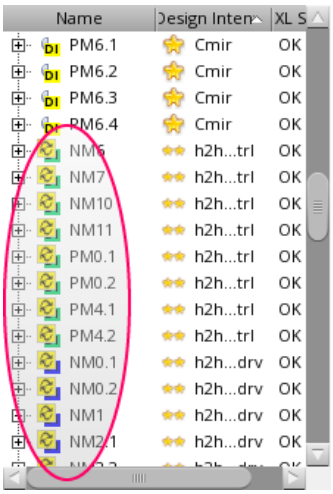
You can review individual design intent updates before syncing by visiting the Edit Design Intent form for each design intent with a Sync icon and reviewing the relevant section's Notes. If you are reviewing changes made in Layout XL, any updates will have been made to the lower section of the form. Changes made within Schematics XL will have been made to the upper and middle sections of the form.

After reviewing the updates for the design intents, you can sync the view.

Note: In Schematics XL you can choose to sync an individual design intent. This lets you review multiple design intents by checking and syncing each design intent, one at a time.

Syncing Between Schematic and Layout Views

To sync a schematic or layout view, you must have the same cellview open in both Schematics XL and Layout XL.



Any design intent changes made on the schematic or layout views result in the changes being marked in the other system with a Sync icon appearing in the Navigator beside each of the amended design intents. These icons are temporary and draw attention to those design intents that have been updated and that require to be synced. The *Sync* button on the Design Intent toolbar is also enabled.

Important

The *Update Schematic Constraints* option on the Constraint Manager assistant also syncs any design intent changes, but it primarily ensures that the constraints in the schematic are in sync with the layout, by pulling in all/selected constraints in the layout into the constraint view. It should therefore only be used if all design intents implemented using the constraint manager have their constraints included in the design intent template (physical constraint). For more details, see Using Design Intent with Constraints.

Syncing in Layout XL

The following changes and updates in Schematics XL result in a sync being required in Layout XL:

- Creation of a new design intent
- Changes to the annotation style
- Updates to the profile or properties
- Updates in the design notes section
- Addition/removal members of a design intent
- Deletion of a design intent

When an update is made to a design in Schematics XL, the *Sync* button on the Design Intent toolbar is enabled and a sync notification is displayed beside each affected object in the Layout XL Navigator. You can perform a sync on a single design intent or on all the design intent within a design.

Syncing in Schematics XL

The following changes and updates in Layout XL result in a sync being required in Schematics XL:

- Creation of a new design intent
- Changes to the implementation status
- Addition or amendment of implementation notes
- Addition/removal of members of a design intent

Note: Implementation constraints within a design intent are not transferred to the schematic when a design intent is synced.

When an update is made to a design in Layout XL, the *Sync* button on the Design Intent toolbar is enabled and a sync notification is displayed beside each affected object in the Schematics XL navigator. You can perform a sync on a single design intent or on all the design intent within a design.

Syncing a single design intent

You can review a large design update that impacts multiple design intents by individually checking and syncing each design intent, one at a time.

To review and update an individual design intent,

1. Ensure that the corresponding view is visible in Layout XL or Schematics XL. (This can be read-only.)
2. If required, review the updates for each design intent or object with a sync notification displayed. Right-click and choose *Design Intent – Edit Design Intent* to open the Edit Design Intent form. Add notes if required and click *OK*.
3. Right-click the object associated with the design intent in Navigator or in the canvas (or if in Layout XL, right-click the design intent glyph) and choose *Design Intent – Sync...* and choose *Sync* – for the required design intent.

   Only the selected design intent is synced between the schematic and layout cellviews. The sync icon is removed indicating that this design intent is now up to date.
4. Repeat for any remaining objects with sync notifications displayed in the Navigator.

Syncing Selected Design Intent Only

To review and update only the selected design intent in a design,

1. Ensure that the corresponding view is visible in Layout XL or Schematics XL. (This can be read-only.)
2. If required, review the updates for each design intent or object with a sync notification displayed. Right-click and choose *Design Intent – Edit Design Intent* to open the Edit Design Intent form. Add notes if required and click *OK*.
3. In the Navigator or on the canvas, right-click and choose *Design Intent – Sync... – Sync All Selected Design Intent*.

   The design intent for the selected objects are synced between the schematic and layout cellviews and their sync icons are removed. The *Sync* button on the toolbar will remain enabled until all the design intent in the design is up to date.

Design intents implemented using constraints do not generate a sync indication.

Syncing All the Design Intent

To review and update all design intent in a design,

1. Ensure that the corresponding view is visible in Layout XL or Schematics XL. (This can be read-only.)
2. If required, review the updates for each design intent or object with a sync notification displayed. Right-click and choose *Design Intent – Edit Design Intent* to open the Edit Design Intent form. Add notes if required and click *OK*.

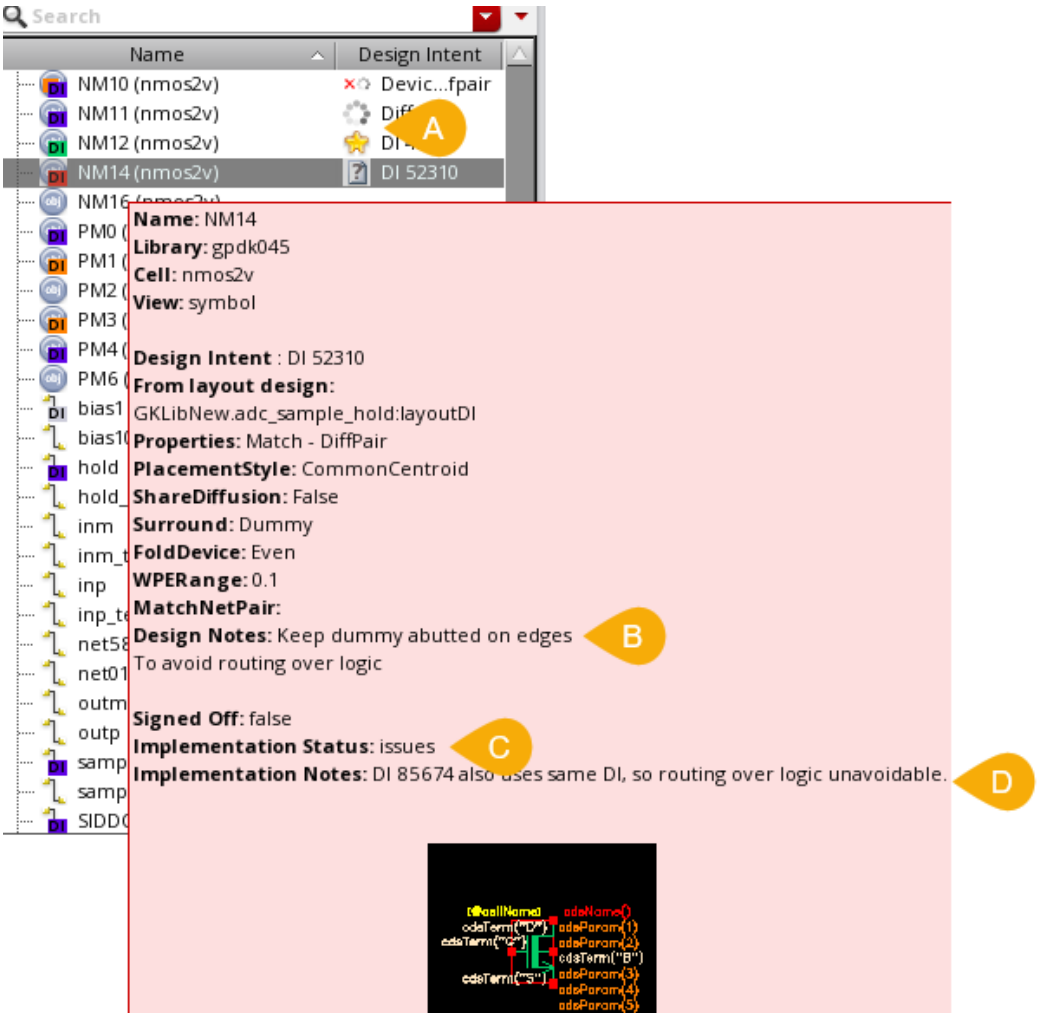
3. In the Navigator or on the canvas, right-click and choose *Design Intent – Sync... – Sync All Design Intent*, or click on the *Sync* button on the toolbar.

   All design intent in the design is synced between the schematic and layout cellviews. The sync icons are removed and the *Sync* button on the toolbar is disabled, indicating that the design is now up to date.

Note: Design intents implemented using constraints do not generate a sync indication.

Recognizing Amended Design Intents after Syncing

On syncing a design, the sync notification is removed from recently amended and updated design intents. The updated design intents that require attention are then identifiable by the implementation status icons on the Navigator or by hovering over a design intent to display a tooltip.



A Implementation status icons indicate the current status of each design intent.

B Shows the schematic designer's design goals and any additional comments are listed.

C Shows the current implementation status for each design intent.

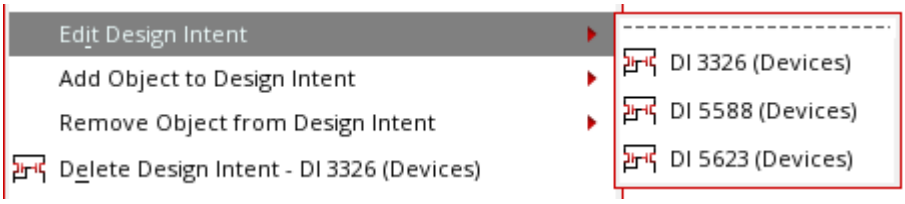
D Shows the layout designer's progress in implementing the design intent and any additional comments are listed.

Editing a Design Intent

You can edit a design intent using the Edit Design Intent form. Access the form by right-clicking the required object in the Navigator or on the canvas:

- right-click the required object and choose *Design Intent – Edit Design Intent* option.
- click the object's glyph and click *Edit* on the Design Intent Info Balloon that is displayed.
- click the object's glyph and double-click on the Design Intent Info Balloon that is displayed.

If an object is associated with more than one design intent, a submenu is displayed listing the different design intents available to be edited.



The following sections provide information about editing design intents:

- Editing a Design Intent in Schematics XL
- Editing a Design Intent in Layout XL
- Adding a Physical Constraint to an Existing Design Intent
- Reviewing and Editing a Design Intent Created in Layout XL

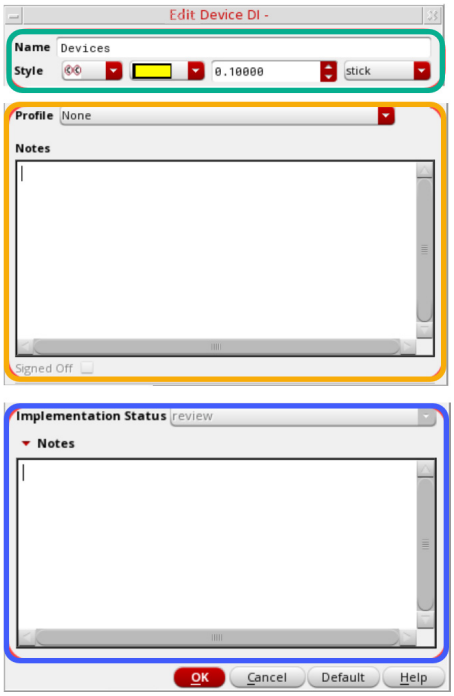
Editing a Design Intent in Schematics XL

1. Open the Edit Design Intent form from the Navigator or the canvas.

   A submenu is displayed if there are multiple design intents available for the selected objects.

2. Choose the design intent to be edited.

The form is divided into three parts: the upper and middle sections are completed in Schematics XL.



3. Edit the *Name*, *Style* or *Profile* on the upper section of the form as required.
4. Amend the *Notes* in the middle section if required.
5. Review the *Implementation Status* and accompanying *Notes* in the lower section of the form. If a response is required to assist the layout designer in implementing the design intent, enter any additional notes.
6. If the *Signed Off* option is enabled it means the design intent has an implementation status of `complete` or `review`.

   If the status is `review`, the design intent was created in Layout XL. See Reviewing and Editing a Design Intent Created in Layout XL for more details.

   If the status is `complete`, review the original design intent that was requested against any recorded notes and the actual outcome. If the design intent is considered to be satisfactorily implemented into the design, click the *Signed Off* option.
7. Click *OK*.

   In Layout XL, the affected objects are immediately displayed with a sync icon as a prompt to Sync Design Intent.

The functions ciTemplateChangeDIProfile and ciTemplateUpdateDIProps can also be used to edit a design intent using SKILL. For more details about the SKILL functions available for Design Intent, see Design Intent Functions in *Virtuoso Unified Custom Constraints SKILL Reference*.

Note: The design intent name, font, and font size of each annotation label can also be amended using the Property Editor or the Edit Object Properties form. This excludes annotation labels for design intent categories *High Current* or *Max Voltage Drop* which display the object name rather than the design intent name so cannot be amended in this way.

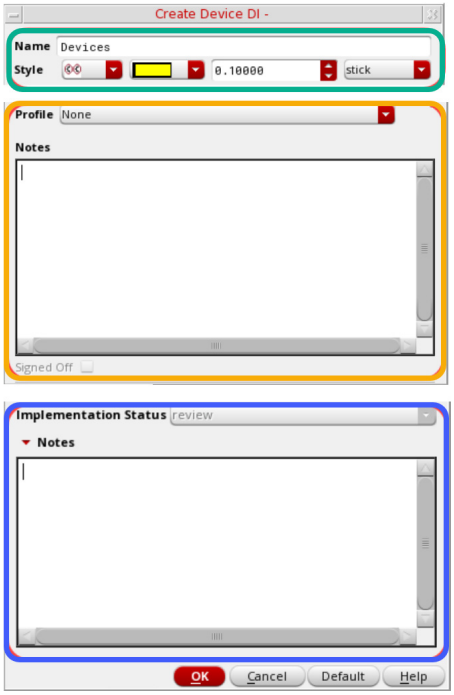
Editing a Design Intent in Layout XL

1. Open the Edit Design Intent form from the Navigator or the canvas.

   A submenu is displayed if there are multiple design intents available for the selected objects.

2. Choose the design intent to be edited.

   The form is divided into three parts: only the lower section is completed in Layout XL.



3. Review any *Notes* in the middle section of the form. The information provided can be used as the basis for the layout designer to implement the design intent.

4. In the lower section, provide any additional *Notes* to query or update the schematic designer regarding the implementation of the design intent
5. Update the Implementation Status as required on the lower section of the form.
6. Select *OK*.
7. In Schematics XL, the *Sync* button on the Design Intent toolbar is enabled and the affected objects are immediately displayed with a sync icon as a prompt to Sync Design Intent.

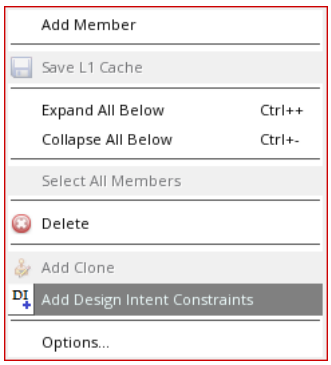
Adding a Physical Constraint to an Existing Design Intent

Note: This action is not necessary in either application when using the *Design Intent – Sync* option as this synchronizes only the design intent information.

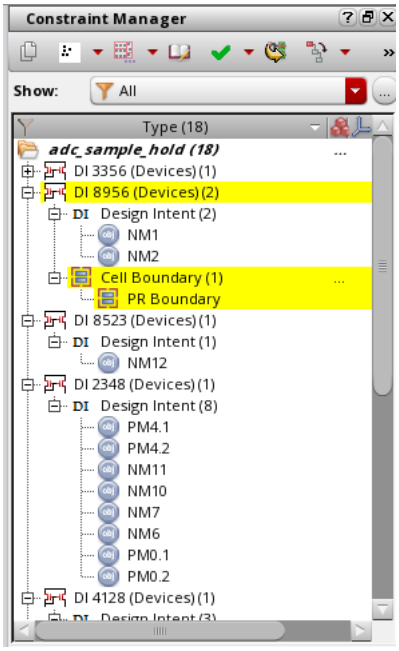
To avoid creating a constraint view when transferring from Layout XL to Schematics XL using the *Update Layout/Schematic Constraints* button, all the constraints on a design must be physical constraints and be included in the design intent template. Constraints are added to a design intent using the Constraint Manager assistant's *Add Design Intent Constraints* option.

Note: When adding a constraint to a design intent, there is no restriction on the constraint type or design intent type being associated.

1. Select the object and the constraint to be added in the *Constraint Manager* assistant in Layout XL.
2. Right-click and from the menu displayed, choose *Add Design Intent Constraints*.



The constraint is added to the design intent and its associated objects.



For more information on how to use constraints and the *Constraint Manager*, see the Constraint Manager Assistant in the Virtuoso Unified Custom Constraints User Guide.

Reviewing and Editing a Design Intent Created in Layout XL

If a design intent was created in Layout XL, the process differs as all design intents must be reviewed, synced, and validated within Schematics XL before they are fully integrated into the normal implementation process.

In the Schematics XL Navigator, the new design intent has a sync icon and the `review` status icon displayed beside it.

1. In the Navigator, right-click the design intent, or on the canvas right-click an object associated with the new design intent and choose *Design Intent – Edit Design Intent*.
2. If there are multiple design intents available for the selected objects, a submenu is displayed. Choose the design intent to be edited.
3. Review the *Notes* in the lower section of the form:
   - If you identify the need for further changes, or any issues or queries that prevent you from validating the new design intent, record these within the middle section's *Notes*.

- Alternatively, if the new design intent has been satisfactorily implemented and requires no other action, it can be considered validated and signed off. In the *Notes* within the middle section, confirm that the design intent is acceptable and add any additional comments if required . Select the *Signed Off* option and click *OK*. The status icon for the object changes to *complete* ✓.

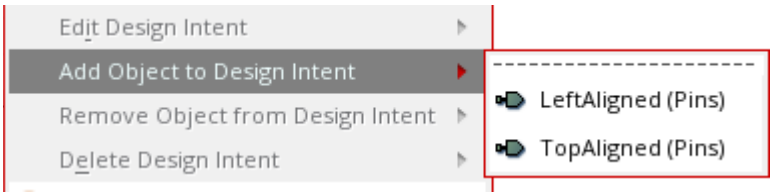
4. Edit the *Name*, *Style* or *Profile* on the upper section of the form as required.
5. Click *OK*.
6. In the Navigator, individually sync the new design intent.

   Only the selected design intent is synced between the schematic and layout cellviews. The sync icon is removed indicating that this design intent is now up to date.

   In Layout XL, the *Sync* button on the Design Intent toolbar is enabled and the affected objects are immediately displayed with a sync icon as a prompt to Sync Design Intent. Each object has an implementation status of *none* as the layout designer is required to respond to the notes or queries and can be processed as normal.

Adding Objects to a Design Intent

You can add single or multiple objects to a design intent by selecting the required objects in the Navigator or on the canvas and choose *Design Intent —Add Object to Design Intent* option.



When there are multiple design intents available, a submenu is displayed from the *Add Object to Design Intent* option, listing the different design intents available for the selected objects.

Adding an object to a design intent in Layout XL, requires the design intent to be reviewed and validated within Schematics XL before it is fully integrated into the normal implementation process. For more details, see Reviewing and Editing a Design Intent Created in Layout XL.

To add an object to a design intent:

1. Right-click the selected objects to be added to a design intent in the Navigator or on the canvas.
2. Choose *Design Intent – Add Object to Design Intent*.

   A submenu is displayed if there are multiple design intents available for the selected objects.
3. Choose the design intent to which the objects are to be added.

   Design intent Annotations are created on the canvas for the selected objects.
4. Record the addition and any relevant information in the Edit Design Intent form. This helps the designer of the other application understand the reasons behind the change. For more details, see Editing a Design Intent.

   If the change is made in Schematics XL, the Sync button on the Design Intent toolbar is enabled in Layout XL and the affected objects are immediately displayed with a sync icon as a prompt to Sync Design Intent.

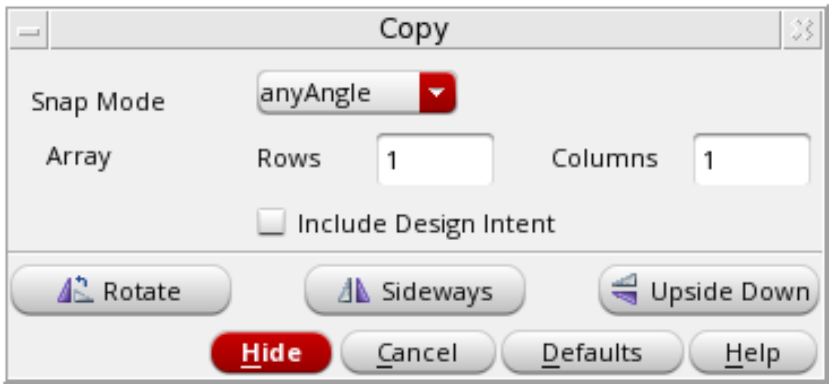
   If the change is made in Layout XL, the Sync button on the Design Intent toolbar is enabled in Schematics XL and the affected objects are immediately displayed with a sync icon as a prompt to Sync Design Intent. The implementation status of the design intent

changes to `review`  allowing the schematic designer to review and validate the design intent.

Copying Objects with Design Intents

To copy an object including any associated design intent:

1. From the view, choose *Edit – Copy*.
2. Press `F3` to display the Copy form.



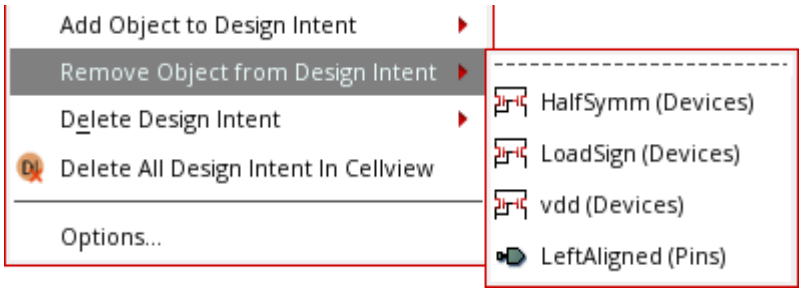
3. Select the *Include Design Intent* option.
4. Select the object you want to copy.
5. Click a destination for the copy.

Renaming Objects with Design Intents Associated

Objects with design intents associated can be renamed as normal. The new name is reflected in any design intent associated with the object when the *Check and Save* command is run. See Editing Object Properties (Text Edit) Directly on the Design Canvas in the *Virtuoso Schematic Editor User Guide* for more details.

Removing Objects from Design Intents

You can remove single or multiple objects from a design intent by selecting an associated object in the Navigator or on the canvas, and choose *Design Intent – Remove Object from Design Intent*.



When the selected objects are associated with multiple design intents, a submenu is displayed listing the different design intents available to be removed.

Removing an object from a design intent in Layout XL, requires the design intent to be reviewed and validated within Schematics XL before it is fully integrated into the normal implementation process. For more details, see Reviewing and Editing a Design Intent Created in Layout XL.

Important

If the selected object is the only object associated with a design intent, choose *Design Intent —Remove Object from Design Intent* to delete the design intent.

To remove an object from a design intent:

1. Select the objects to be removed from the design intent either in the Navigator or on the canvas.
2. Right-click and from the menu displayed, choose *Design Intent – Remove Object from Design Intent*.

   A submenu is displayed if there are multiple design intents available for the selected objects.
3. Choose the design intent from which the objects are to be removed.

   The selected objects are removed from the design intent.

4. Record the removal and any relevant information in the Edit Design Intent form. This helps the designer of the other application understand the reasons behind the change. For more details, see Editing a Design Intent.

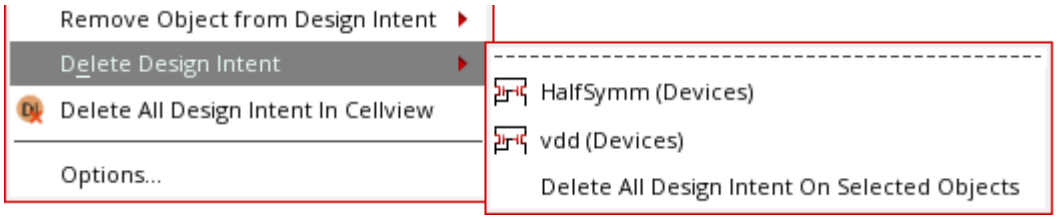
   If the change is made in Schematics XL, the *Sync* button on the Design Intent toolbar is enabled in Layout XL and the affected objects are immediately displayed with a sync icon as a prompt to Sync Design Intent.

   If the change is made in Layout XL, the *Sync* button on the Design Intent toolbar is enabled in Schematics XL and the affected objects are immediately displayed with a sync icon as a prompt to Sync Design Intent. The implementation status of the design intent changes to `review` allowing the schematic designer to review and validate the design intent.

Deleting a Design Intent

Note: This menu option is available only in Schematics XL.

You can delete a design intent by selecting the design intent in the Navigator and choosing *Design Intent – Delete Design Intent* or by selecting an associated object and right-clicking in the Navigator or on the canvas and choosing *Design Intent – Delete Design Intent*.



The *Delete Design Intent* option deletes the design intent from the selected object, all associated objects and from the cellview. If you only want to remove selected objects from a design intent, use the *Remove Object from Design Intent* option.

A submenu is displayed if there are multiple design intents available for the selected objects. An additional option to *Delete All Design Intent On Selected Objects* is also available which deletes the associated design intents from the selected object, all associated objects, and from the cellview.

To delete all the design intent in the cellview use the *Delete All Design Intent in Cellview* option which is accessed by right-clicking on the navigator or on the canvas, or using on the toolbar.

The following sections provide information about deleting design intents:

- Deleting Design Intent on Objects
- Deleting All Design Intent in Cellview

Deleting Design Intent on Objects

Note: To remove an object from a design intent and continue to associate other objects with that design intent, use the Remove Object from Design Intent option.

To delete a design intent:

1. In the Navigator select the design intent to be deleted, or on the canvas right-click an object associated with the design intent to be deleted.

2. From the menu displayed, choose *Design Intent – Delete Design Intent*.
3. If there are multiple design intents available for the selected objects, a submenu is displayed. Choose the design intent to be deleted.

   The design intent is deleted from all associated objects and from the cellview.

   In Layout XL, the *Sync* button on the Design Intent toolbar is enabled and the affected objects are immediately displayed with a sync icon as a prompt to Sync Design Intent.

To delete all design intent associated with an object:

1. Select the object in the Navigator or on the canvas.
2. Right-click and choose *Design Intent – Delete Design Intent – Delete All Design Intent On Selected Objects*.

   The associated design intents are deleted from the selected objects, all associated objects, and from the cellview.

   In Layout XL, the *Sync* button on the Design Intent toolbar is enabled and the affected objects are immediately displayed with a sync icon as a prompt to Sync Design Intent.

Deleting All Design Intent in Cellview

Note: This menu option is enabled only in Schematics XL.

This function deletes all the design intent on the cellview. If you want to delete only the design intent associated with specific objects, use the Delete All Design Intent On Selected Objects ***option.***

The *Delete All Design Intent in Cellview* option removes all the design intent from all objects in the cellview. It is available by selecting the button on the toolbar, or right-clicking on any object in the Navigator or on the canvas.

To delete all design intent using the design intent toolbar:

1. Save the cellview.

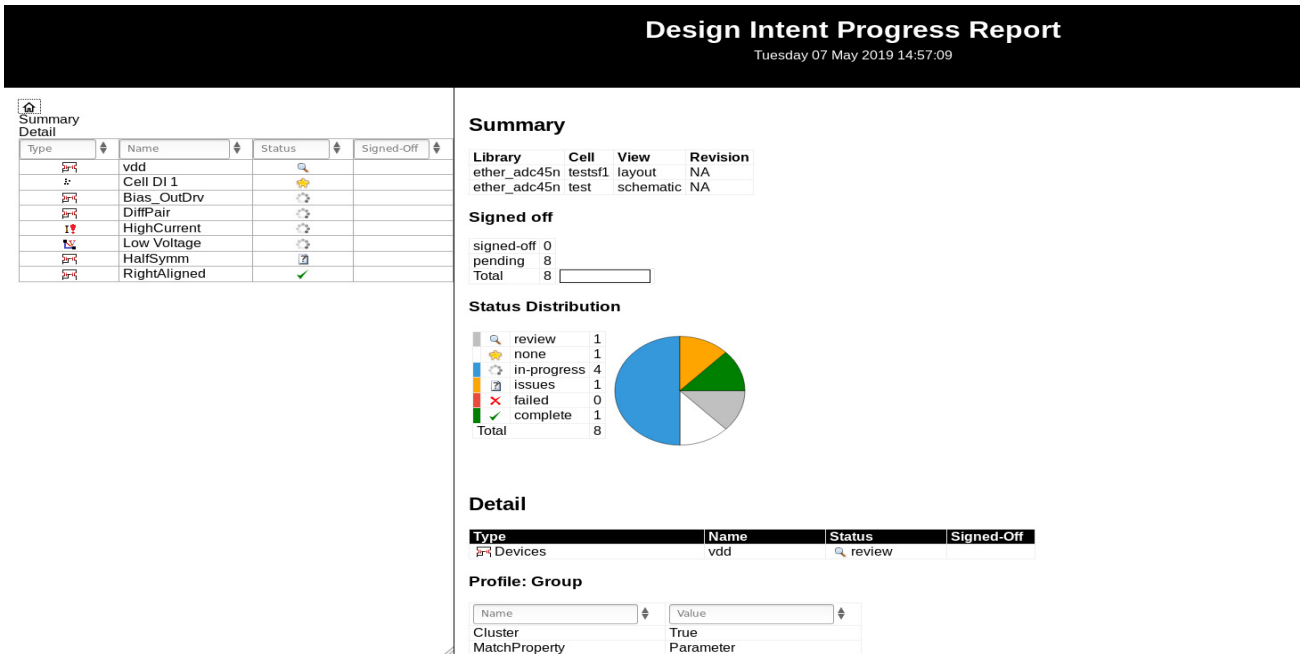
   Deleting all the design intent in a cellview cannot be undone. By saving a copy of the cellview you can revert back to the saved cellview if required.

2. Click the button on the *Design Intent* toolbar or right-click any object in the Navigator or on the canvas. You are prompted to confirm the deletion before the design intent is removed.

In Layout XL, the *Sync* button on the Design Intent toolbar is enabled and the affected objects are immediately displayed with a sync icon as a prompt to Sync Design Intent.

Generating Reports on Design Intent

The progress of all the design intent implementation on a design can be checked at any point using a high level report generated from Schematics XL or Layout XL. The report provides a summary of the design intent within a design, reporting the implementation status of each design intent with the associated notes capturing the communication between the layout and schematic designers. It is useful to keep abreast of the latest design changes and to review the progress of the project as a whole.



The report displays the date and time the report was generated and is divided with a navigation pane on the left and a pane containing the report content on the right.

On the navigation pane:

Summary selecting this option displays the design intent summary in the report pane.

Detail selecting this displays the Detail section at the top of the report pane. The Detail table summarizes the status of each design intent. All the columns are filterable and selecting any item on a row, displays that design intent at the top of the report pane. Each row includes the following information:

Type A pictorial symbol for the design intent type.

Name The name of each design intent.

Status The current implementation status of each design intent.

Signed Off The current stage of validation for each design intent.

On the report pane:

Summary

Library / **Cell** / **View** / **Revision** details the library, cell, view, and revision from which the design intent information is sourced.

Signed Off provides an overview of the number of design intents currently pending validation or validated as having been achieved in Schematic XL. For more details, see Verifying and Signing Off a Design Intent.

Status Distribution provides an overview of the implementation progress of all design intent into the design.

Detail lists the following information for each design intent, if available:

Type displays the type of design intent.

Name displays the name of the design intent.

Status displays the current implementation status of the design intent.

Signed Off displays the Sign-off status of the design intent. A tick denotes that the design intent is completed in Layout XL and validated in Schematic XL as having been achieved in the design. For more details, see Verifying and Signing Off a Design Intent.

Profile displays the property configurations assigned to the design intent.

Members Information (available for categories HighCurrent and MaxVoltageDrop only) displays the objects on which the design intent applies and the voltage or current that has been specified for each.

Design Notes displays the schematic designer's design goals and any additional comments. This is the information entered in the *Notes* field of the middle section of the Edit Design Intent form.

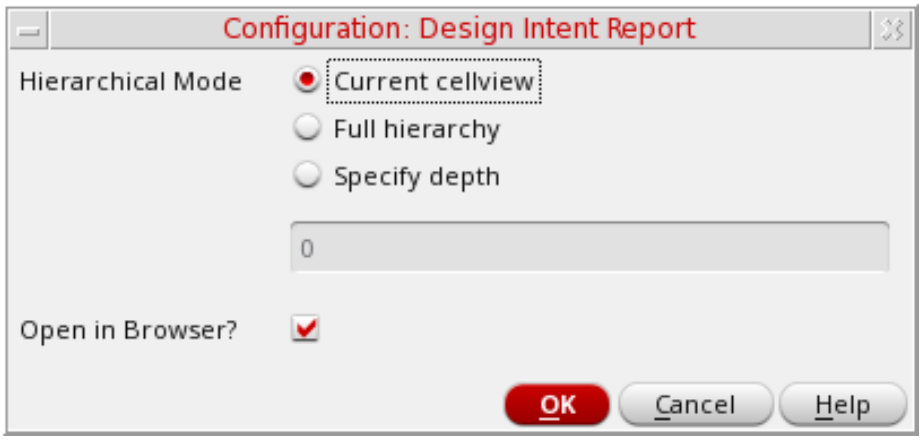
Implementation Notes displays the progress or strategy used by the layout designer to implement the design intent. This is the information entered in the *Notes* field of the lower section of the Edit Design Intent form.

Implementation Constraints displays the constraints defined for the design intent.

Running Design Intent Reports

To run a design intent report from either Schematics XL or Layout XL:

1. Click Report on the Design Intent toolbar. The Configuration: Design Intent Report dialog box is displayed.



2. Select the *Hierarchical Mode* required:
   - **Current cellview** displays the design intent for the current cellview only.
   - **Full hierarchy** displays the design intent for the full hierarchy.
   - **Specify depth** allows you to specify the hierarchical level that is to be displayed.
3. If *Open in Browser?* is selected, when you click *OK*, a new web browser is opened displaying the full report for the design. Deselecting this option generates the report without opening it.
4. Click *OK*. The report is generated and displayed for review.

Verifying and Signing Off a Design Intent

The *Signed Off* option is located on the Edit Design Intent form and is enabled only in Schematics XL for design intents with a status of *complete* ✓ or *review* 🔍. Selecting this option validates that the design intent has been implemented in Layout XL and confirms that the design goal has been satisfactorily achieved within the design.

Note: The `review` 🔍 status icon is automatically given to design intents that are created or objects that are added or removed in Layout XL. Each requires to be reviewed and validated in Schematics XL before they are signed off. For more details, see Reviewing and Editing a Design Intent Created in Layout XL.

To sign off a design intent:

When the layout designer considers the design intent to have fulfilled the original goal including any adjustments (captured within the design intent) they set the implementation status to *complete* ✓. In order to be considered fully implemented, the design intent must be reviewed and signed off by the schematic designer.

1. Identify the design intent with a status of *complete* ✓ in the Schematics XL Navigator.
2. Right-click a design intent or an object associated with the design intent and choose *Design Intent – Edit Design Intent*.
3. Review the original design intent that was specified against any recorded notes and the actual outcome.
4. If the design intent is considered to have been satisfactorily implemented into the design, select the *Signed Off* option and click *OK*.

   When design intent Generating Reports on Design Intent are run, the *Signed-off* column displays a ✓ icon against the design intent.

A

Design Intent Forms

Create Design Intent

Use the Create Design Intent form to create design intents. Access the Create Design Intent form by right-clicking an object on the canvas or in the Navigator and choosing *Design Intent – Create Design Intent.*

Note: The options that are available vary between Schematics XL and Layout XL.

Name

Specify a unique name to identify a single or group of objects associated with the design intent. This is used as the label for the annotation on the canvas.

Style

Define a style to identify the design intent annotation on the canvas and distinguish it from others by adjusting the Shape, Color, Font Size and Font.

Shape
Specify the shape used to outline the design intent on the canvas. The options available depend on the category of object selected:

Device/Pin:

- Circle Around Each Object
- Bounding Box Around Each Object
- Single Bounding Box Around All Objects

MaxVoltageDrop:

 Lightning Flash

HighCurrent:

 Arrow

Net:

 Bounding Box Around Each Net Segment

 Single Bounding Box Around All Net Segments

 Box Around All Net Segments

Color
Specify the color used for the design intent icon in the Navigator and for the annotation label, shape and glyph on the Canvas. For more details, see Interacting with Design Intent in the Navigator Assistant.

Font Size
Specify the text size used in the annotation label on the canvas. You can also use the environment variable diNoteFontHeight to change the font size.

Note: In Schematics XL, the selected font size also changes the size of the glyph for the design intent.

Font
Specify the font used for the annotation label on the canvas.

Note: The following section is only enabled in Schematics XL.

Profile (optional) Select a *Profile* from the drop-down list.

Contains a list of predefined custom profiles to avoid duplication of effort when creating design intents. Each profile comprises a group of frequently used property configurations. The property options available are determined by the *Profile* you select and how it has been defined. For more details see Defining Property Profiles.

Note: The categories *HighCurrent* and *MaxVoltageDrop* automatically have profiles preselected, *Current* and *Voltage* respectively. These cannot be overridden. For more details, see Creating a HighCurrent Design Intent and Creating a MaxVoltageDrop Design Intent.

You can also change the property profile selected for a design intent using the `ciTemplateChangeDIProfile` function. For more details, see Design Intent Functions in the *Virtuoso Unified Custom Constraints SKILL Reference*.

(ICADVM18.1 Only) If scopes are defined for the current profile, a combo box listing the available scopes is displayed alongside the *Profile* field. A scope filters the properties that are visible for the current profile.The default scope displayed is defined using the environment variable diPropDefaultScopes. For more details, see Defining Scopes for Profile Properties (ICADVM18.1 Only).

Show/Hide Disabled Toggles the display of a list of objects available to which the MaxVoltageDrop or HighCurrent design intents can be applied, either displaying all available objects or only those that are currently selected. This button is only available for design intent types MaxVoltageDrop and HighCurrent.

Notes Document the schematic designer's design goals and any additional comments. The information entered is used as the basis for the layout designer to implement the design intent.

Note: The information entered populates the *Design Notes* in design intent info balloons and reports. When responding to a query or comment, Cadence recommends that you include the date and time to help track the conversation between the schematic and layout designers.

Signed Off This option is disabled and is visible only in Layout XL.

Note: The following section is only enabled in Layout XL.

Implementation Status	Indicate the layout designer's progress in implementing the design intent into the design. The options are *none*, *in progress*, *issues*, *failed*, or *complete*. **Note:** This option is available in the Edit Design Intent form. Each status has a corresponding icon which is displayed in the Navigator following synchronization. See Design Intent Icons in Navigator for more details.
Notes	Document the progress or strategy used by the layout designer to implement the design intent recorded by the schematic designer in the Notes. Use this text box to record any queries, comments, or responses relating to the implementation of the design intent. If the design intent is being created in Layout XL, use this text box to record the reason for creating it. For more details see Creating Design Intents in Layout XL. **Note:** The information entered populates the Implementation Notes column in reports. When responding to a query or comment, Cadence recommends that you include the date and time to help track the conversation between the schematic and layout designers.

Related Topics

Creating a Design Intent

Creating Design Intents in Layout XL

Edit Design Intent

Use the Edit Design Intent form to amend the design intents in a design. Access the form by right-clicking an object on the canvas or in the Navigator and choosing *Design Intent – Edit Design Intent.* If an object is associated to more than one design intent, a submenu is displayed listing the design intents available.

The form can also be accessed by right-clicking on a glyph and clicking *Edit* on the Design Intent Info Balloon that is displayed. For more details on editing a design intent, see Editing a Design Intent.

Note: Only the lower section of the form is enabled in Layout XL.

Note: The following section is only enabled in Schematics XL.

Name Specifies the unique name to identify a single or group of objects associated with the design intent. This is used as the label for the annotation on the canvas.

Style Defines a style to identify the design intent annotation on the canvas and distinguish it from others by adjusting the Shape, Color, Font Size and Font.

Shape
Specify the shape used to outline the design intent on the canvas. The options available depend on the category of object selected:

Device/Pin:

 Circle Around Each Object

 Bounding Box Around Each Object

 Single Bounding Box Around All Objects

MaxVoltageDrop:

 Lightning Flash

HighCurrent:

 Arrow

Net:

 Bounding Box Around Each Net Segment

 Single Bounding Box Around All Net Segments

 Box Around All Net Segments

Color
Specifies the color used for the design intent icon in the Navigator and for the annotation label, shape and glyph on the Canvas. For more details, see Interacting with Design Intent in the Navigator Assistant.

Font Size
Specifies the text size used in the annotation label on the canvas. You can also use the environment variable diNoteFontHeight to change the font size.

Note: In Schematics XL, the selected font size also changes the size of the glyph for the design intent.

Font
Specifies the font used for the annotation label on the canvas.

Note: The following section is only enabled in Schematics XL.

Profile (optional) The drop-down list contains a list of predefined custom profiles to avoid duplication of effort when creating design intents. Each profile comprises a group of frequently used property configurations. For more details see Defining Property Profiles.

The property options available are determined by the *Profile* you select and how it has been defined.

You can also change the property profile selected for a design intent using the `ciTemplateChangeDIProfile` function. For more details, see Design Intent Functions in the *Virtuoso Unified Custom Constraints SKILL Reference*.

(ICADVM18.1 Only) If scopes are defined for the current profile, a combo box listing the available scopes is displayed alongside the *Profile* field. A scope filters the properties that are visible for the current profile.The default scope displayed is defined using the environment variable diPropDefaultScopes. For more details, see Defining Scopes for Profile Properties (ICADVM18.1 Only).

Show/Hide Disabled Toggles the display of a list of objects available to which the MaxVoltageDrop or HighCurrent design intents can be applied, either displaying all available objects or only those that are currently selected. This button is only available for the design intent types MaxVoltageDrop and HighCurrent.

Note: The following section is only enabled in Schematics XL.

Notes

Documents the schematic designer's design goals and any additional comments. The information entered is used as the basis for the layout designer to implement the design intent.

Use this text box to respond to any queries or comments raised by the layout designer in the Implementation Status notes.

Note: The information entered populates the *Design Notes* in design intent info balloons and reports. When responding to a query or comment, Cadence recommends that you include the date and time to help track the conversation between the schematic and layout designers.

Signed Off

Validates that the design intent has been implemented in Layout XL and confirmed that the design goal has been achieved within the design. For more details, see Verifying and Signing Off a Design Intent.

Note: This option is enabled only for design intents with a status of Complete  or `review`  .

Note: The following section is only enabled in Layout XL.

Implementation Status

Indicates the layout designer's progress in implementing the design intent into the design. The options are *none*, *in progress*, *issues*, *failed*, or *complete*.

Note: This option is available in the Edit Design Intent form.

Each status has a corresponding icon which is displayed in the Navigator following synchronization. See Design Intent Icons in Navigator for more details.

Notes

Document the progress or strategy used by the layout designer to implement the design intent recorded by the schematic designer in the Notes.

Use this text box to record any queries, comments, or responses relating to the implementation of the design intent.

If the design intent is being created in Layout XL, use this text box to record the reason for creating it. For more details see Creating Design Intents in Layout XL.

Note: The information entered populates the Implementation Notes column in reports. When responding to a query or comment, Cadence recommends that you include the date and time to help track the conversation between the schematic and layout designers.

Related Topics

Editing a Design Intent

Editing a Design Intent in Schematics XL

Editing a Design Intent in Layout XL

B

Design Intent Environment Variables

This appendix covers the environment variables that can be used to control and customize the characteristics and behavior of design intent. Refer to the List of Environment Variables section for a complete list of environment variables available for use.

For information about the environment variables specific to:

- Schematics XL, refer to the Environment Variables chapter in the *Virtuoso Schematic Editor User Guide*.
- Layout XL, refer to the Environment Variables chapter in the *Virtuoso Layout Suite XL User Guide*.

diNoteFontHeight

In the .cdsenv file:

```
constraint diNoteFontHeight float font_height
```

In the .cdsinit file or the CIW:

```
envSetVal("constraint" "diNoteFontHeight" 'float f_height)
```

Description

Specifies the font height for the text used in design intent annotations. The default is `0.1` (user units).

GUI Equivalent

Command	Create Design Intent or Edit Design Intent forms.
Form Field	Font Size

Example

```
constraint diNoteFontHeight float 0.3
```

diPropDefaultScopes

In the .cdsenv file:

```
constraint diPropDefaultScopes string ""
```

In the .cdsinit file or the CIW:

```
envSetVal("constraint" "diPropDefaultScopes" 'string "")
```

Description

(ICADVM18.1 Only) Specifies the default scope for each profile. The value is a comma-separated list of scopes. The first scope value listed is the default scope.

When a profile with scopes defined is selected for a design intent on the Create Design Intent and Edit Design Intent forms, a combo box situated beside the *Profile* field lists all the defined scopes available. The default scope specified is automatically selected in the combo box.

You can filter which of the available defined scopes are visible on the design intent forms using the environment variable diPropScopesFilter.

GUI Equivalent

None

Example

A design intent has three profiles:

Profile1 has scopes `Scope1`, `Scope2`, `Scope3`

Profile2 has scopes `ScopeA`, `ScopeB`, `ScopeC`

Profile3 has scopes `Scope1`, `ScopeB`

To define that scopes `Scope1` and `ScopeB` are the default scopes for these profiles:

```
envSetVal("constraint" "diPropDefaultScopes" 'string "Scope1,ScopeB")
```

The defaults for the profiles are:

Profile1: `Scope1`

Profile2: `ScopeB`

Profile3: `Scope1`

Although Profile3 has scopes `Scope1` and `ScopeB` which are both defined as defaults, listing `Scope1` first, defines it as the default.

Related topics

Defining Scopes for Profile Properties (ICADVM18.1 Only)

diPropScopesFilter

diPropScopesFilter

In the .cdsenv file:

```
constraint diPropScopesFilter string ""
```

In the .cdsinit file or the CIW:

```
envSetVal("constraint" "diPropScopesFilter" 'string "")
```

Description

(ICADVM18.1 Only) Creates a subset of the available scopes for the current profile. The value is a comma-separated list of scopes. Only the scopes listed are visible on the Create Design Intent and Edit Design Intent forms.

Used in situations where a group of users only want some of the available scopes to be visible for a profile.

Example

To specify that only scopes `Scope1` and `Scope3` should be visible for selection from the current profile on the Create Design Intent and Edit Design Intent forms:

```
envSetVal("constraint" "diPropScopesFilter" 'string "Scope1,Scope3")
```

Related topics

Defining Scopes for Profile Properties (ICADVM18.1 Only)

`diPropDefaultScopes`

C

Defining Property Profiles

To avoid manually entering a large amount of textual information each time you create a design intent, frequently-used properties can be grouped to create user defined profiles. A number of sample profiles defined for design intents are supplied in the `diPropGroupDefs.il` and are visible from the Create Design Intent form. If required, this file can be edited by your Virtuoso system administrator to add new profiles or change existing ones to meet the specific requirements of a particular design or team.

(ICADVM18.1 Only) You can associate a property with multiple scopes to control which properties are visible for the current profile. For more details, see Defining Scopes for Profile Properties (ICADVM18.1 Only).

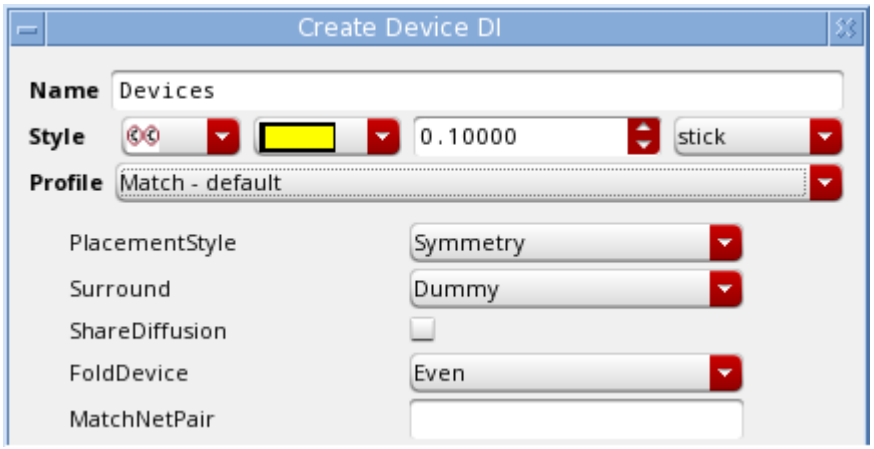
Property Profiles

A property profile is a group of property definitions stored in the *Design Intent Property Group Definitions* file, `diPropGroupDefs.il`. The master file is located at `$CDS/Share/cdssetup/dfII/ci`. Each profile comprises a group of frequently used properties, for example, add shield, add guard ring, and so on. Your Virtuoso system administrator sets up the profiles as per your requirements.

Example of a Property Profile

The *Match – default* profile comprises the properties *PlacementStyle*, *Surround*, *ShareDiffusion*, *FoldDevice*, and *MatchNetPair*.

On the Create Design Intent form this profile is displayed as follows:



In the property group definitions file, this profile is listed as follows:

```
/*************************************************************************************************
**
**   list(
**       ;; Property Group DPL
**       list(nil 'name      "GroupName"
**                'category [ "Cell" | "Devices" | "Nets" | "Pins" | "MaxVoltageDrop" | "HighCurrent" ]
**                ['toolTip "toolTip"]
**                ['properties
**                     list(
**                       list(nil 'name "PropertyName"
**                                'type ['bool|'string|'int|'float|'enum|'current|'voltage]
**                                ['defValue value]
**                                ['range    '(low high)]             ;; int/float range
**                                ['items    '("item1" ... "itemN") ] ;; enum choices
**                       )
**                     )
**
**
**   )
**
*************************************************************************************************/

list(

    ;;; ============ ================ Devices

    list(nil 'name "None" 'category  "Devices" 'toolTip "Simple Note") ;;; No properties just a simple note

    list(nil 'name "Match - default"  category "Devices" 'toolTip "How devices are to match"
            'properties list(
                        list(nil 'name "PlacementStyle" 'type         'enum   'defValue "Symmetry" 'items '("Symmetry" "CommonCentroid" "Interdigitate"))
                        list(nil 'name "Surround"        'type        'enum   'defValue "Dummy"    'items '("Dummy" "FGR" "Both" "asNeeded") 'toolTip "GuardRing style")
                        list(nil 'name "ShareDiffusion" 'type         'bool   'defValue nil )
                        list(nil 'name "FoldDevice"       'type       'enum   'defValue "Even"     'items '("Even" "Odd" "Avoid"))
                        list(nil 'name "MatchNetPair"     'type   'string 'defValue "")
                )
```

Sample syntax

A sample profile

The property definitions of the profile

Each property in the profile is defined. The property *PlacementStyle* is a list (`enum`) and a default value (`Symmetry`) has been specified:

```
list(nil 'name "PlacementStyle" 'type 'enum 'defValue "Symmetry"  'items
'("Symmetry" "CommonCentroid" "Interdigitate"))
```

Understanding the Profile Properties Syntax

```
list(
    ;; Property Group DPL
    list(nil 'name     "GroupName"
             'category [ "Cell" | "Devices" | "Nets" | "Pins" | "MaxVoltageDrop" |
                         "HighCurrent" ]
             ['toolTip "toolTip"]
             ['properties
                  list(
                    list(nil 'name "PropertyName"
                             'type ['bool|'string|'int|'float|'enum|'current|
                                    'voltage]
                             ['defValue value]
                             ['range    '(low high)] ;; int/float range
                             ['items    '("item1" ... "itemN") ] ;; enum choices
                    )
                  )
             ]
         )
)
```

Description

Sample syntax required to create and define custom profiles to capture frequently used property configurations for design intents. Each profile comprises a group of frequently used disembodied property list (DPL) properties, for example, add shield, add guard ring, and so on. The profiles can be created by your system administrator.

Arguments

`'name "GroupName"`	A unique name for the profile.
`'category`	The object type that the profile applies to. Choose one of `'Cell`, `'Devices`, `'Nets`, `'Pins`, `'MaxVoltageDrop`, or `'HighCurrent`.
`'toolTip`	A short description is displayed on the GUI as a tooltip when you hover the mouse over the profile name on the Create or Edit Design Intent forms.

There is a separate row for each property to be listed in the profile. Each property must have a `name`, `type`, and `defValue`.

`'name`	Specify the property name. This is displayed on the Create or Edit Design Intent forms. In ICADVM18.1 only, this is replaced with the value for `'guiName`, if specified.
`'scopes`	(ICADVM18.1 Only) A list of scopes associated with the property. This attribute is required only if the property is to be associated to a scope. The value can be a string, a string list, or `nil`. If the value is `nil` or if the `'scopes` attribute is not listed for the property, the property is visible to all scopes.
`'type`	Specify the default type for the property. The options are `'bool`, `'string`, `'int`, `'float`, `'enum`, `'current`, or `'voltage`.
`'guiName`	(ICADVM18.1 Only) Specify the property name as it is to be displayed in the Create Design Intent and Edit Design Intent forms. This is required only if the definition of a property differs between scopes. In such cases, different property definitions are created for each scope with a different `'name` value, but the same `'guiName` value.
`'defValue`	Specify the default value for the property. For example, if creating a property type of `'enum`, enter one of the items listed in `items` to display as the default.
`'items`	List the values for the property. This is required for properties of type `'enum` only.
`'range`	Specify minimum and maximum values for properties with a variable type of `integer` or `float` (where required). The values must be enclosed in parentheses, `'range '(1 5)`.

Modifying Sample Profiles and Property Definitions

The sample profiles and their properties are supplied as a base for your Virtuoso system administrator to create new profiles and adapt the definitions of their properties to meet the specific requirements of a design or team.

Note: Before creating or amending design intent profiles, the schematic and layout designers must agree on the type of information to be shared within a design intent profile.

Important

When creating or defining property profiles based on the supplied master file `diPropGroupDefs.il`, be aware that the property definitions supplied with the profiles *Current* and *Voltage*, are specifically designed for the design intent categories, *HighCurrent* and *MaxVoltageDrop*, respectively. They are supplied with defined properties specific to the expected function of that profile and must not be edited or overridden.

To amend the property group definitions file:

1. In an editor, open the *Design Intent Property Group Definitions* file, `diPropGroupDefs.il`.
2. Copy and paste one of the supplied default profiles including its DPL properties.
3. Referring to Understanding the Profile Properties Syntax, in `'name`, give the new profile a unique name. For example, replace `'name "GroupName"` with `'name "New Profile 1"`.
4. In `'category` specify a object type for the new profile.
5. In `'toolTip` enter a description for the new profile. This displays when you hover over the new profile's name on the Create Design Intent form.
6. For each property required for the new profile, either create a new property or replace one of the existing properties that was copied, by carrying out the following:
   a. In `'name`, enter a `"PropertyName"`. This must be a unique name for the property.
   b. Specify a `'type`. The options are `bool`, `string`, `int`, `float`, `enum`, `current`, or `voltage`.
      - If `enum` is selected, list the type values in `'items`.
      - If `integer` or `float` is selected, specify minimum and maximum values for the property in `'range`.
   c. In `defValue`, specify a default value for the property.

7. Repeat Step 6 for each property to be added to the profile.
8. Remove any properties not required by the profile.
9. Comment out any sample text provided so it is not displayed on the Create Design Intent form.
10. Repeat Steps 2 to 9 for any additional profiles required.
11. Do not delete or change any of the supplied default profiles or properties. Comment out any of the supplied profiles or their properties that are not required.
12. Save the file. A copy of the master file including the profile changes is automatically created, `diPropGroupDefs.il`. and saved to the `.cadence` directory or the directory defined by the Cadence Setup Search File mechanism (CSF) (`setup.loc`). For more details, refer to the Cadence Application Infrastructure User Guide. Subsequent changes to the profiles override this copy, not the master file.

The new or amended profiles are available from the Profile drop-down list on the Create Design Intent and Edit Design Intent forms.

For information on how to control the visibility of property settings for a profile using scopes, see Defining Scopes for Profile Properties (ICADVM18.1 Only).

For information on the SKILL functions available for modifying profiles and property definitions, see Design Intent Functions.

Defining Scopes for Profile Properties (ICADVM18.1 Only)

Using scopes in your design lets you control the visibility of the property settings for the current profile on the Create Design Intent and Edit Design Intent forms. This means schematic and layout designers working together on a design can use scopes to filter the profile properties to focus on only those properties in which they are interested.

If no scope is defined for a profile, all properties are displayed. When a scope is created, a combo box appears alongside the *Profile* field on the Create Design Intent and Edit Design Intent forms. The combo box lists the new scopes created for the profile. The selected scope determines the properties that are visible for that profile.

A profile may have multiple scopes, each of which can be associated with a selection of the available profile properties. Similarly, a property can be associated with multiple scopes. Although a design intent might have multiple profiles to select from and each profile have multiple scopes defined, only a single profile and a single scope can be selected for the current design intent. Those properties without a scope are displayed regardless of the scope selected because these properties are considered to belong to all the scopes on the profile.

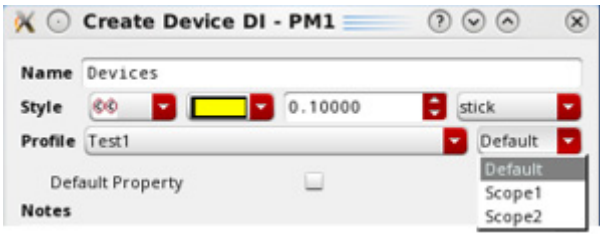
To define a scope:

1. In an editor, open the Design Intent Property Group Definitions file, `diPropGroupDefs.il`.
2. Identify the profile to have scopes defined.
3. For each property to be associated with a scope, use the `ciSetDIPropertyGroupDefs` environment variable to add a new `scopes` attribute to the property definition. For its value, add the name of the scope to be associated with the property. For example: `"Scope1,Scope2"`.

```
(ciSetDIPropertyGroupDefs '(
    (nil name "Profile1"
        category "Devices"
        properties ((nil name "DefaultProp" type bool guiName "Default
                              Property" defValue nil)
                    (nil name "Group1Prop1" scopes list("Devices" "Macro)
                              type bool defValue nil)
                    (nil name "Group2Prop1" scopes "Scope2" guiName
                              "Group2 Prop1" type bool defValue nil))
                    )
    )
)
```

4. Use the diPropDefaultScopes environment variable to specify the default scopes for the profile.

On the Create Design Intent and Edit Design Intent forms, a combo box now appears alongside the *Profile* field from which you can select a scope to control which properties are visible for `Profile1`.



Note: The property `Default Property` is displayed regardless of the scope selected because a scope was not defined for this property.