cadence®

Virtuoso Design Intent User Guide

Product Version IC23.1
November 2023

Printed in the United States of America.

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

Contents

1

Introduction to 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 requirements of the schematic designer 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.

Related Topics

Virtuoso Design Intent Flow

Benefits of Using Design Intent

Design Intent Interaction Between Schematics XL and Layout XL

Virtuoso 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.

It is recommended that before implementing design intent into a design, the schematic and layout designers agree on the type of information to be shared within a design intent and that custom profiles are set up.

1. The 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, for example, add shield, add guard ring, and so on.)
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. The layout designer can clearly identify the objects with design intents and begin 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.

Related Topics

Property Profiles

Create Design Intent Form

Design Intent Glyphs

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. This saves the schematic designer from having to design and set up each object individually. Using annotations allows the schematic designer to focus on capturing the design intent without having to create physical constraints on the design.
- The Navigator assistant displays design intent information within its own specific category. It also displays design intent information on individual objects associated with a design intent using 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 the accompanying glyph of an annotation 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 can 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 design intents 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 to Layout XL using syncing, 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. Many of the options are disabled 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.

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.

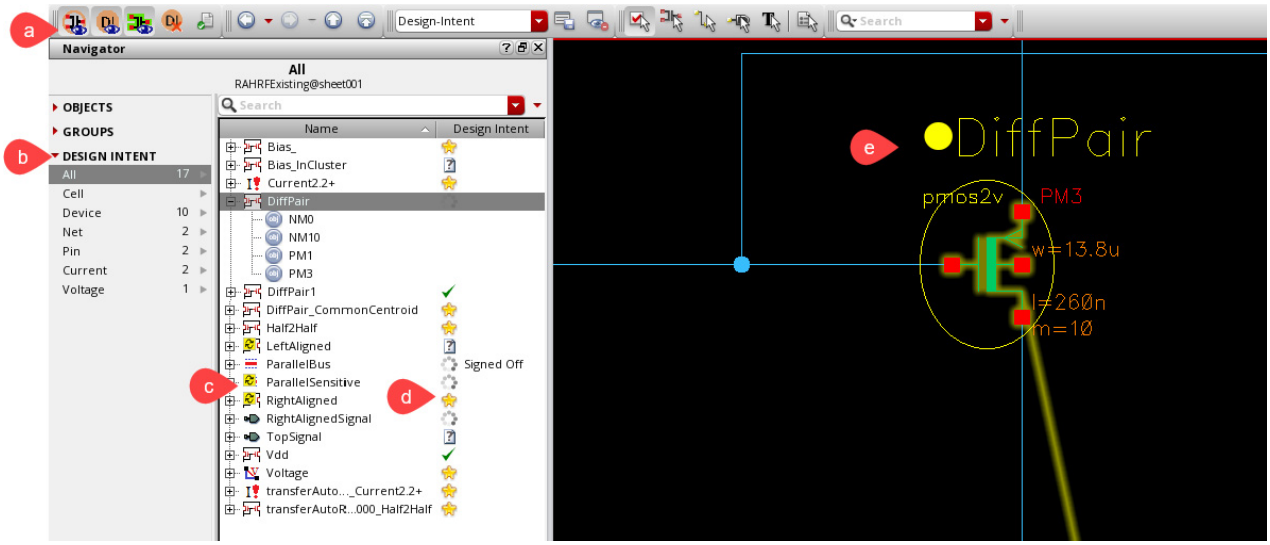
Design Intent complements the constraints flow in your organization by allowing schematic designers to capture their design goals in Schematics XL without actually creating constraints so a separate constraint view is not created. The design intent is synchronized to Layout XL, where the layout designer 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.

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 sync icons and implementation status icons, and 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.
c Sync icons	Alerts designer to any design specification or implementation status changes.
d Implementation Status icons	Indicates the current implementation status for each design intent.
e Design Intent Annotations	Holds key design intent requirements, stores detailed instructions for individual objects, and can easily be identified on the canvas.

Related Topics

Navigator Interaction with Design Intent

Design Intent Toolbar

Design Intent Icons in Navigator

Design Intent Annotations

Design Intent Toolbar

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

Icon	Command	Description
	*Annotation Show/Hide*	Toggles the display of the annotation shape on the canvas. This option is not available in Layout XL.
	*Glyph Show/ Hide*	Toggles the display of the annotation name and glyphs on the canvas.
	*Halo Show/ Hide*	Toggles the halo functionality in the Navigator and on the canvas.
	*Delete all DI in Cellview*	Removes all design intents from the cellview. This option is not available in Layout XL.
	*Report*	Generates a high-level report from Layout XL or Schematics XL to report the design intent within a design.
	*Show/Hide Info Balloons*	Toggles the display of info balloons on the canvas. If the info balloons are hidden then they are shown and vice-versa. In addition, when there is a selection of a design intent in Navigator Assistant or on the layout canvas, then this button works for the selected design intent.
	*Sync All Design Intent*	Synchronizes all design intents between the schematic and layout cellview. The tooltip lists the individual design intents that shall be synchronized. This icon is only enabled if there are design intent changes to be synchronized.

Hierarchy Propagation

Runs auto propagation across multiple hierarchies for selected nets or HighCurrent design intents.

This feature provides the following options:

- *DI Push-Down*  : Pushes the selected nets or HighCurrent design intents down the hierarchy.
- *DI Push-Up*  : Pushes the selected net or HighCurrent design intents to the top of the hierarchy.
- *DI Consistency Check*  : Runs a consistency check to compare the reference design intent specification with the specification defined at different design hierarchies for a given net and reports any discrepancies between them. The reference design intent is the one from which the consistency check was run. This works only for Net design intents.

This option is not available in Layout XL.

Related Topics

Design Intent Annotations

Design Intent Glyphs

Design Intent Haloing

Deleting Design Intent on Objects

Running Design Intent Reports

Design Intent Synchronization

Navigator Interaction with Design Intent

The Navigator assistant 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 Navigator:

- *Objects* lists each object in the design. Each object associated with a design intent has an overlay over its normal icon to indicate presence of a design intent. Also, the *Design Intent* column is displayed for the selected object.
  - *All*: displays all types of design intent and hierarchical folders, which can be further expanded to view the design intents at lower-level.
  - *Instances*: displays devices and instances.
  - *Nets*: displays nets that are available in the design.
  - *Pins*: displays pins that are available in the design.
  - *Nets and Pins*: displays both nets and pins in the design.

  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 members of the same design intent. Also, the *Design Intent* column is available to see what design intent is associated with it. If no design intent is available with an object, the column is left blank.
- *Groups* lists each design intent within the cellview, grouped by set.
- *Design Intent* lists the design intent for each object category. When an object category is expanded, each design intent within the cellview for that category is listed. An icon is associated with each category of design intent. For more information, see Design Intent Icons in Navigator.

  Selecting a set opens the *Details* pane, where each design intent within the cellview for that category is listed. 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.

  The *DI Status* and *DI Profile* columns show the design intent information. The *DI Status* column displays the implementation status of each design intent and the *DI Profile* column displays the profile of each design intent. These columns are only displayed when you are in a design intent category. You can display or hide the column based on your need.

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

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.

Icon	Tooltip
	*Cell DI*
	*Device DI*
	*Net DI*
	*Pin DI*
	*HighCurrent DI*
	*MaxVoltageDrop 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 .

Icon	Tooltip	Description
	*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*.
	*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 *DI Status* column indicate the current implementation status for the design intent. The implementation status is updated by the layout designer in the Edit Design Intent Form.

Icon	Tooltip	Description
	*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.
	*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.

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

Context-Sensitive Menu Options and Tooltips 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:

Menu	Description
Design Intent...	
Create Design Intent	Create a new design intent on the selected objects.
Select All Members	Select all members associated with a design intent.

Menu	Description
Edit Design Intent	View/amend existing design intents associated with the selected objects using the Edit Design Intent form. If the selected object has multiple design intent options available, a slider option is displayed providing access to each design intent.
Add to Design Intent	Add the selected objects to an existing design intent. If the selected object has multiple design intent options available, a slider option is displayed providing access to each design intent.
Remove Object from Design Intent	Remove the selected objects from a design intent. If the selected object has multiple design intent options available, a slider option is displayed providing access to each design intent.
Delete Design Intent	Delete the selected design intent by removing it from all associated objects. **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. If the selected object has multiple design intent options available, a slider option is displayed providing access to each design intent.
Delete All Design Intent in Cellview	Delete all design intents from the cellview. This option is also available by right-clicking anywhere on the canvas in Schematics XL only.

Menu	Description
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. *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. 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.

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.

Related Topics

Navigator Assistant

Create Design Intent Form

Editing a Design Intent

Adding Objects to a Design Intent

Removing Objects from Design Intents

Deleting a Design Intent

Design Intent Synchronization

Interacting with Design Intent on the Canvas

You can interact with design intents on the canvas the following ways:

- Using the context-sensitive menu 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.
- Utilizing design intent annotations which are created the same time as design intents as an easily identifiable name, glyph, and shape.
- Reviewing design intent info balloons by clicking on a design intent glyph. 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.

Related Topics

Context-Sensitive Menu Options on Canvas

Design Intent Annotations

Design Intent 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 the glyph of an annotation displays a menu with options available specific to that design intent and the selections that are currently made.

The options are as follows:

Menu	Description
Design Intent...	
Create Design Intent	Create a new design intent on the selected objects.

Menu	Description
*Edit Design Intent**	View/amend existing design intents associated with the selected objects using the Edit Design Intent form. ■ 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. ■ 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. ■ 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. ■ 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.
Delete All Design Intent in Cellview	Delete all design intents from the cellview. ■ This option is also available by right-clicking anywhere on the canvas in Schematics XL only.

Menu	Description
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. **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.

* 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.

Related Topics

Create Design Intent Form

Editing a Design Intent

Adding Objects to a Design Intent

Removing Objects from Design Intents

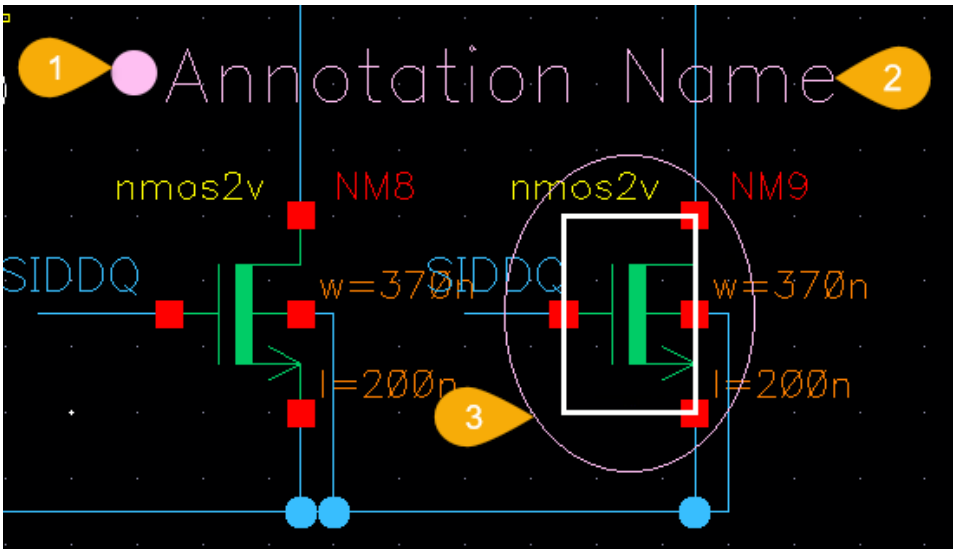
Deleting a Design Intent

Design Intent Synchronization

Design Intent Annotations

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 statuses of associated objects.

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. Glpyh — 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 *HighCurrent* or *MaxVoltageDrop* 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.

Related Topics

diShowAnnotations

Design Intent 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 font size of the design intent, 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.
- Double-clicking a glyph displays the Edit Design Intent form. You can use this form to edit the design intent.

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.

Related Topics

Create Design Intent Form

diShowLayoutGlyphs

diShowSchematicGlyphs

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.

Related Topics

diPinInfoBalloonCellDI

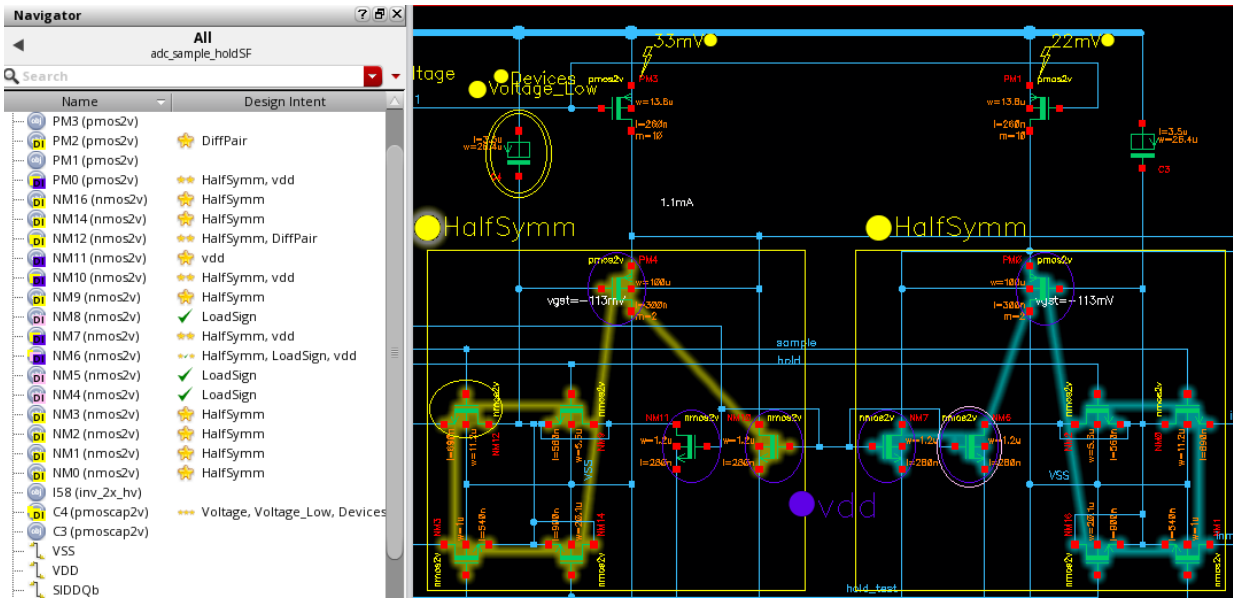
Design Intent Haloing

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.

Related Topics

Design Intent Toolbar

Edit Design Intent Form

Create Design Intent Form

Interacting with Design Intent on the Canvas

diShowLayoutHalos

diShowSchematicHalos

Design Intent 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 intents to update any design intent changes in the schematic and layout views to ensure that both systems are synced.

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

Related Topics

Design Intent Synchronization

Sync

Profile

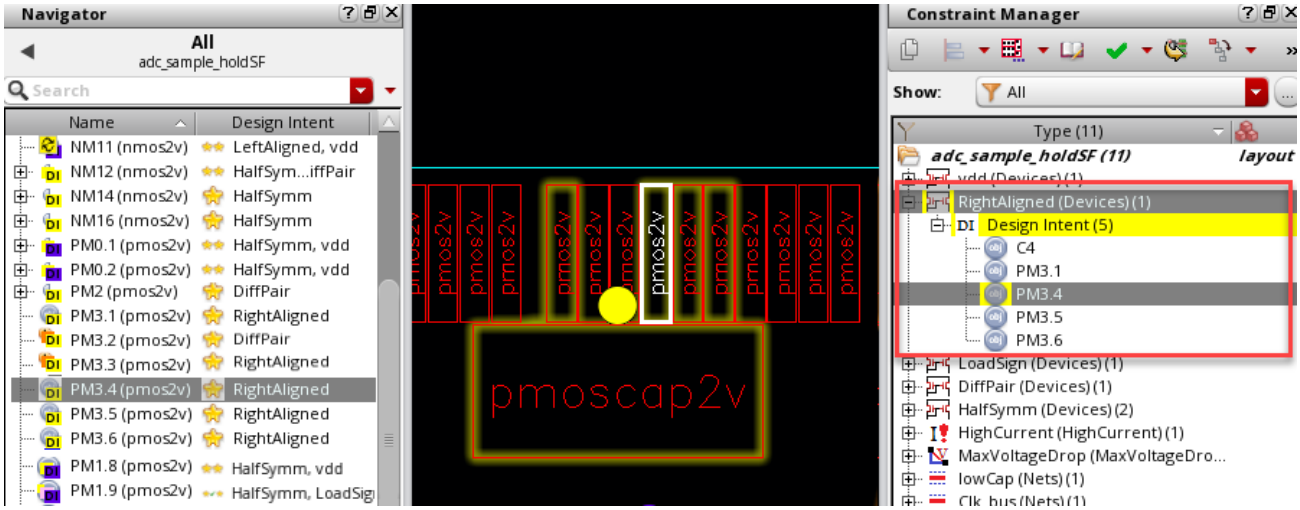
Schematic and Layout View Synchronization

Design Intents in the 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.

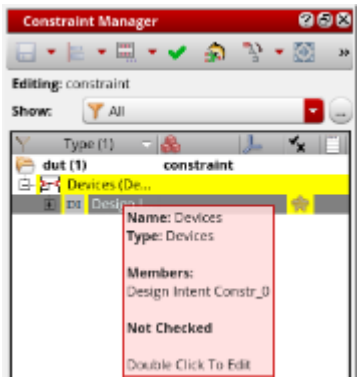


Design intents are grouped together in the Constraints Manager. You can create constraints in the layout view using the *Constraint Manager* assistant. The constraints created are supported by the various physical design applications.

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

Double-clicking a design intent in the *Constraint Manager* assistant opens the Edit Design Intent form.

Hovering the pointer over a design intent in the *Constraint Manager* displays a tooltip as shown in the following figure.



Related Topics

Adding a Physical Constraint to an Existing Design Intent

2

Design Intent Creation

You can create a design intent using the Create Design Intent Form. 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*, *HighCurrent DI*, and *MaxVoltageDrop DI*.

The following table shows the type of design intent that can be created based on the type of object selected:

Object Type	Cell	Device	Net	Pin	HighCurrent	MaxVoltage Drop
Cell	X					
Instance		X				
Net			X		X	X
Pin				X	X	X
InstTerm					X	X

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.

Related Topics

Create Design Intent Form

Navigator Interaction with Design Intent

Creating a Design Intent in Schematics XL

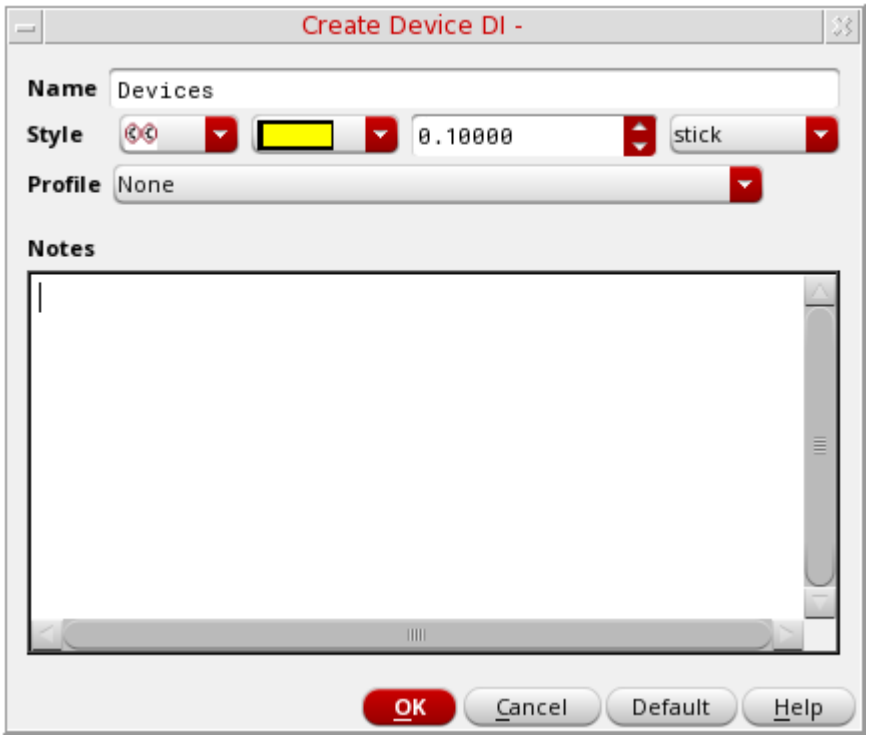
Note: The options available on the Create Design Intent form differ between Schematics XL and Layout XL.

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.

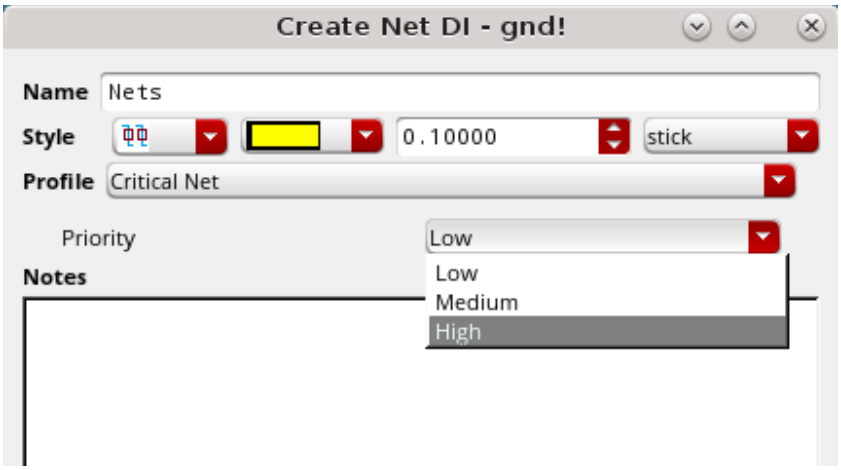
   The Create Design Intent Form is displayed.



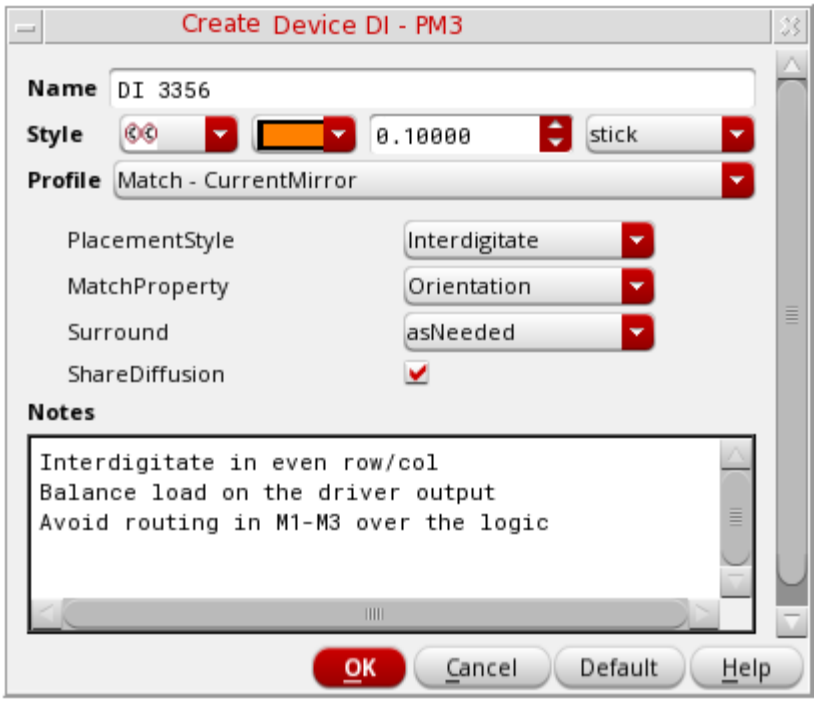
   Note: The options displayed on the form vary to the type of object, category, and profile selected.

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 assistant.
4. (Optional) In Schematics XL, define property requirements by selecting a pre-configured *Profile*. Additional selection options are displayed specific to the profile selected.

   In the *Profile* drop-down list, select *Critical Net* to filter for critical net design intents. You can customize the priority of the design intent by choosing from *Low*, *Medium*, or *High*.



5. In Schematics XL, define property requirements in the *Notes* section. 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.

Differences Between Global and Normal Nets

Global nets in schematics differ from 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.

Related Topics

Pins and Pin Names

Creating a Design Intent in Layout XL

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

Syncing Design Intents

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 asterisk to signify that it is the reference terminal, for example `5A*`.

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.

   For currents with positive polarity, the arrow points outward from the pin. For currents with negative polarity, the arrow points inward to the pin.

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

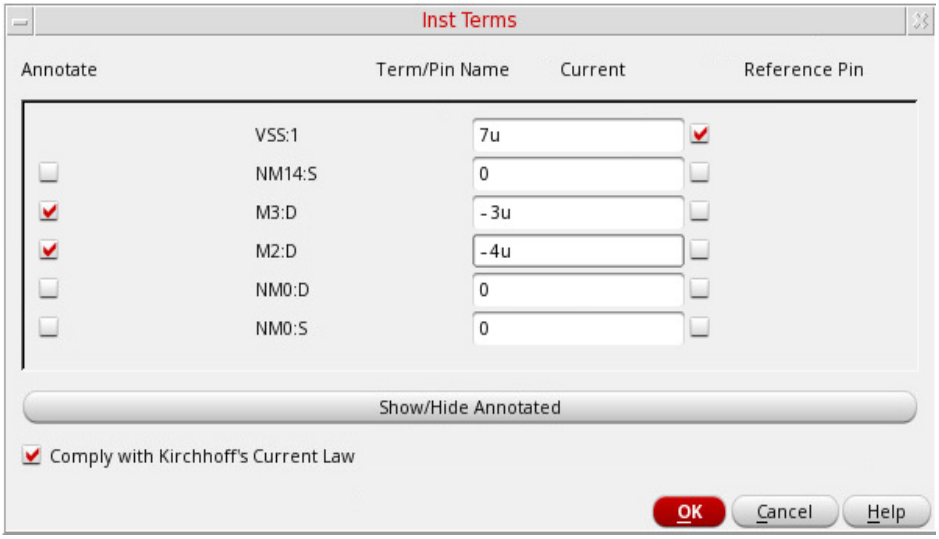
4. Define the requirements for the following pre-defined profile properties:

   - *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 Currents*.

   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.

   Note: Newly added members are auto-annotated.

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

Creating a HighCurrent Design Intent for Multiple-bit Instances

Modifying Sample Profiles and Property Definitions

Adding Objects to a Design Intent

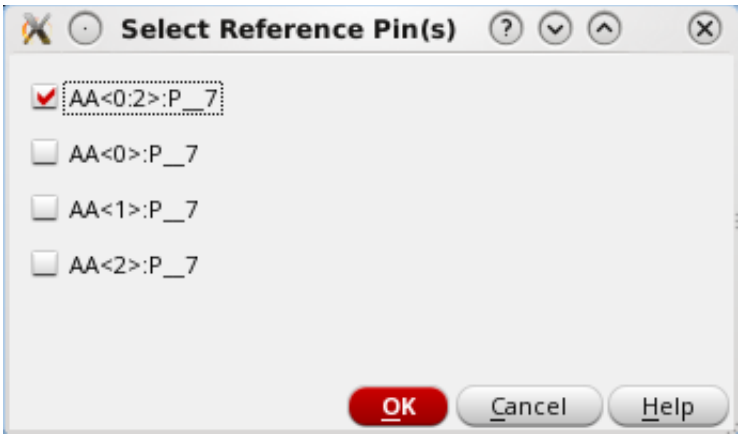
Creating a HighCurrent Design Intent for Multiple-bit Instances

Note: An instance terminal can be associated with only 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.

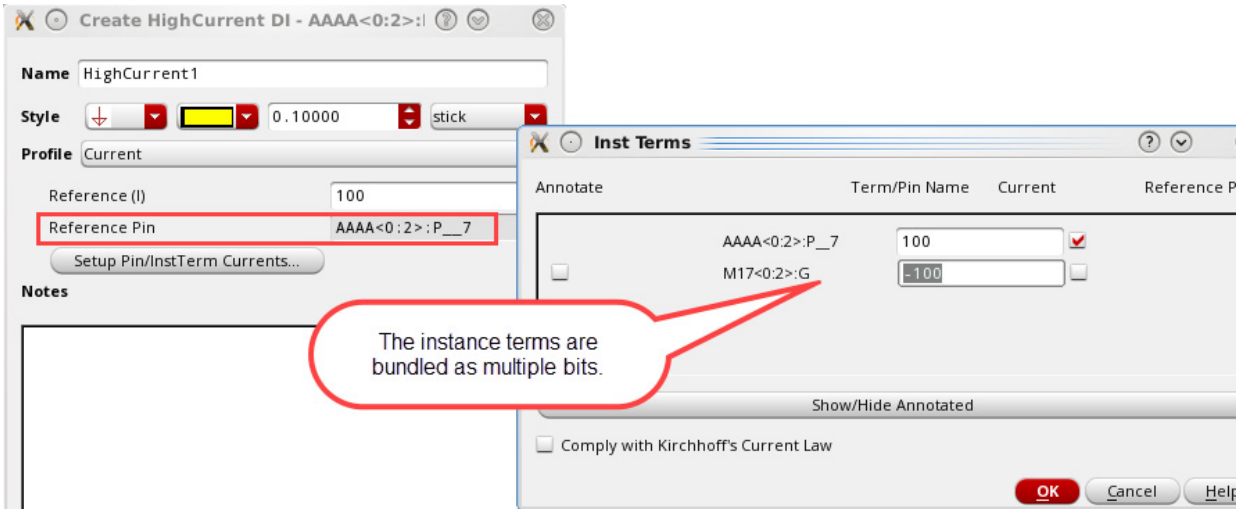


2. Select whether the reference pin for the design intent is to be a bundled multiple-bit pin or a single-bit pin and click *OK*.
3. Specify a name and style for the design intent.
4. In the *Profile* field, *Current* is the only profile available for the HighCurrent design intent category and must not 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 Currents*.

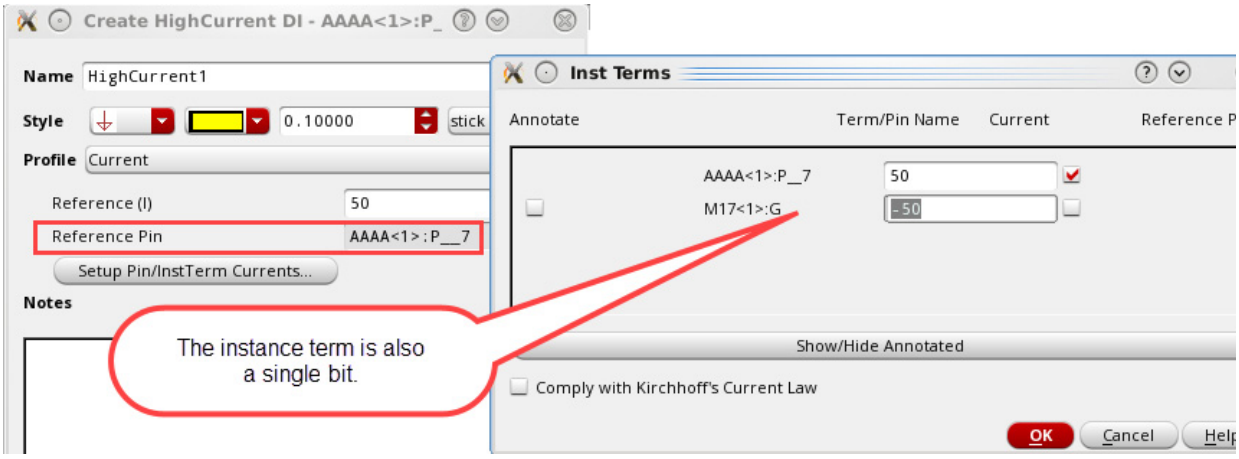
   The Inst Terms dialog box opens. It lists all instance terminals available on the net and lets you define the current for each instance terminal to be sourced.
6. Enter the current value for the required instance terminals or pins.

   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`.

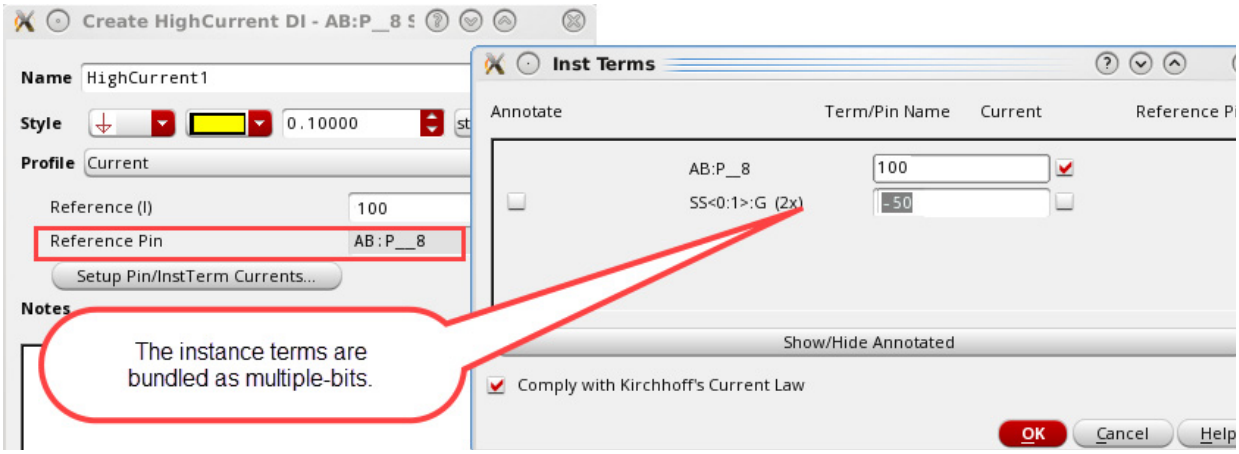


If you specified that the reference pin is a single-bit instance, the instance term is also single bit. The *Current* value applies to the single-instance term.



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. 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.

8. 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

Creating a HighCurrent Design Intent

Modifying Sample Profiles and Property Definitions

Creating a MaxVoltageDrop Design Intent

An instance terminal can only be associated with a single MaxVoltageDrop design intent. To apply a design intent to a net, pin, or instance terminal with voltages:

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.

   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 asterisk symbol to signify that it is the reference pin/terminal, for example `12V*`.
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. 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.

   Note: Newly added members are auto-annotated.

Important

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.

Related Topics

Create Design Intent Form

Property Profiles

Modifying Sample Profiles and Property Definitions

Adding Objects to a Design Intent

Creating a Cell Design Intent for a Cellview

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

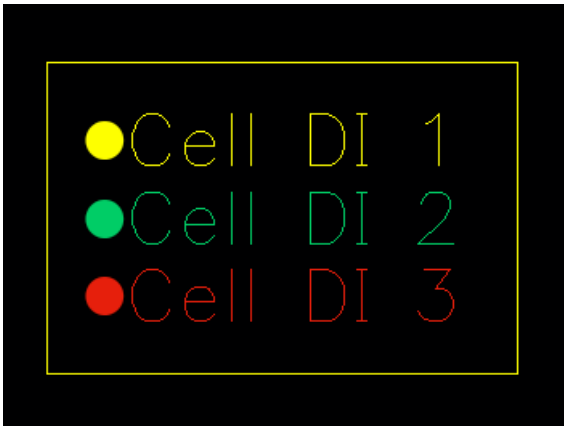
1. Ensuring that no objects are selected, right-click the canvas, (or in the Navigator, right-click the cell) and choose *Design Intent – Create Design Intent - Cell*.

   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*.

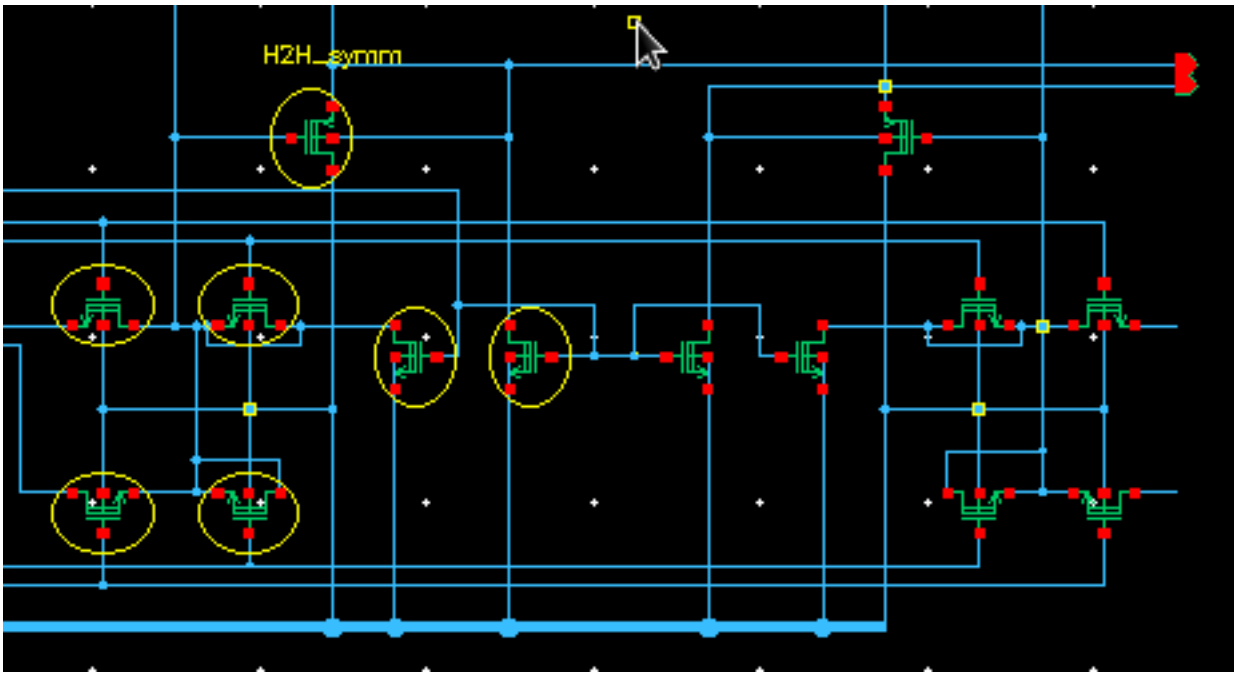
Creating a Device Design Intent for Half Cells

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*.

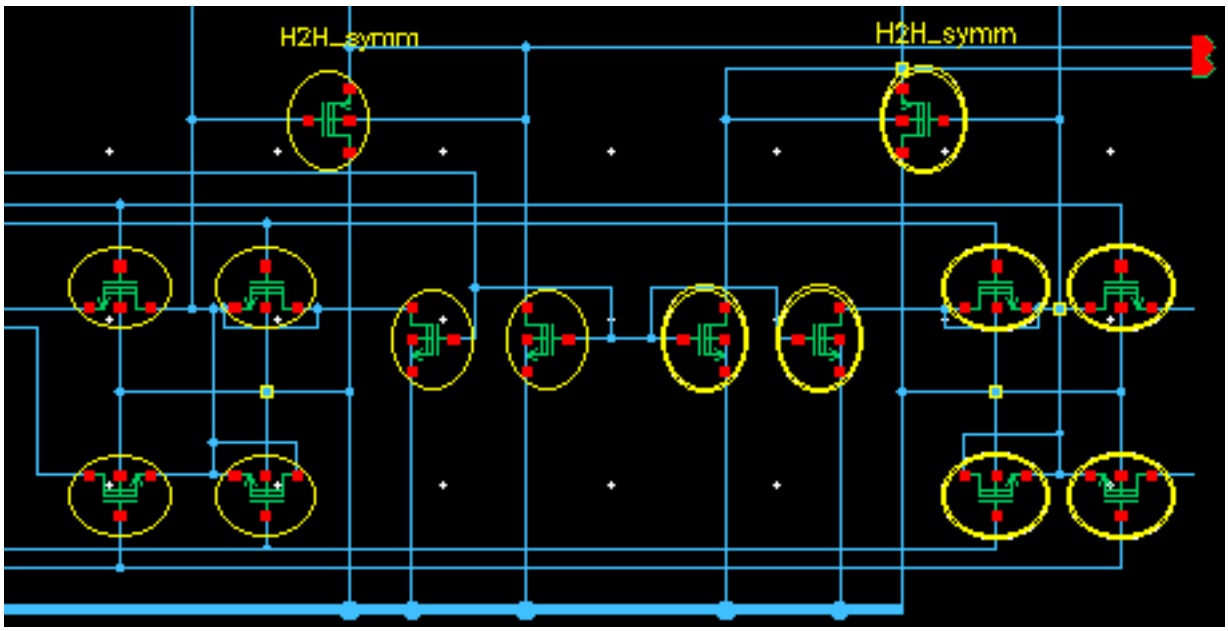
Additional selection options are displayed specific to the Half to Half profile.

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.

Related Topics

Create Design Intent Form

Property Profiles

Syncing Design Intents

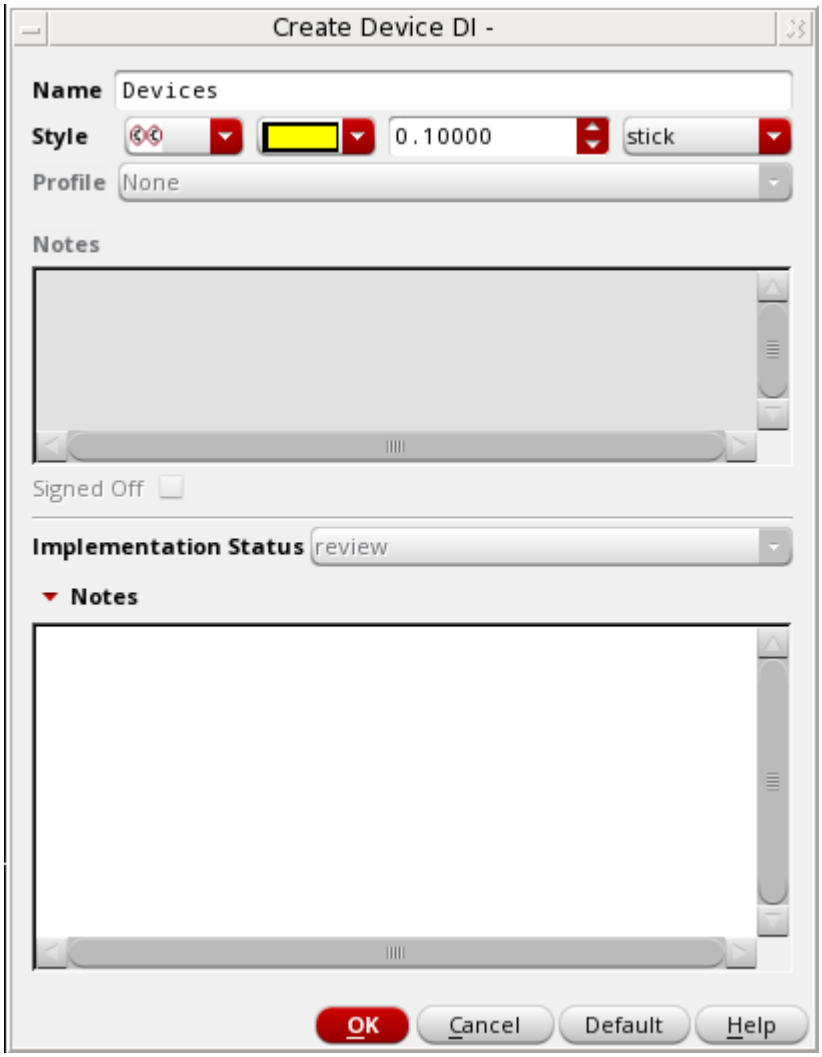
Creating a Design Intent in Layout XL

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

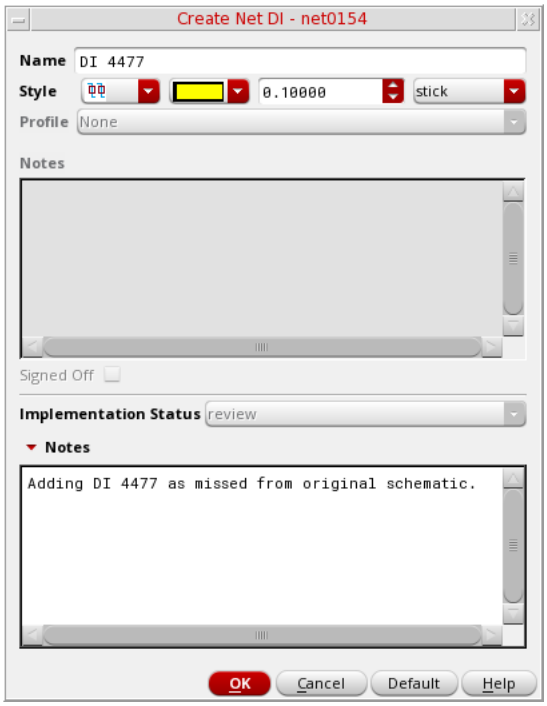
Note: The options available on the Create Design Intent form differ between Schematics XL and Layout XL. To create a new design intent in Schematics XL, see Creating a Design Intent in Schematics XL.

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.

Related Topics

Create Design Intent Form

Schematic and Layout View Synchronization

Reviewing and Editing a Design Intent Created in Layout XL

Creating a Design Intent in the Constraint Manager

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.

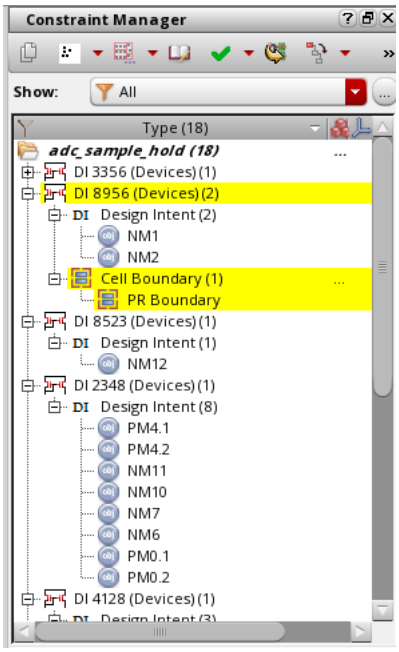
To create a design intent from the Constraint Manager:

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.
4. Select the object and the constraint to be added in the *Constraint Manager* assistant.
5. Right-click and choose *Design Intent – Add Design Intent Constraints* from the shortcut menu.

   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.

Related Topics

Category

Creating a Design Intent in Schematics XL

Creating a Design Intent in Layout XL

Constraint Manager Assistant

Design Intent Synchronization

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.

Update Verification 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.

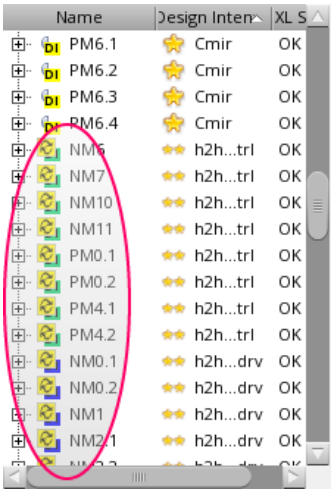
You can review individual design intent updates before syncing by visiting the Edit Design Intent Form for each design intent with a Sync icon and reviewing the Notes of the relevant section. If you are reviewing changes made in Layout XL, any updates are made to the lower section of the form. Changes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

Schematic and Layout View Synchronization

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.

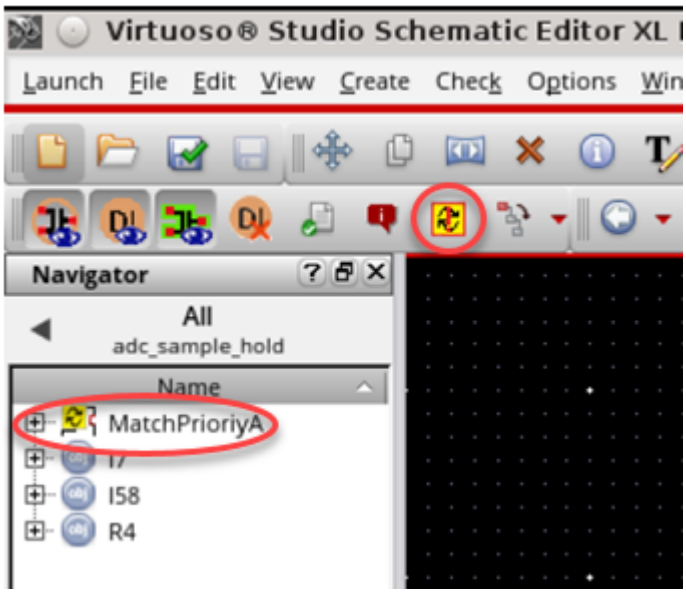
Important

The *Update Schematic Constraints* option in 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).

Alert for Synchronization Required in Schematic

It is possible to preview an alert for sync required on design intents without having the need of schematic and layout views to be opened together. This means that you can preview sync required on design intents with only schematic view opened. To preview the design intents that have a sync required alert on them, use the *Check and Save* button on the toolbar. With

the use of *Check and Save*, the design intent that has a sync required gets an icon for pending sync in the navigator and on toolbar, as shown in the following figure.



Related Topics

Update Verification Before Syncing a Design Intent

Schematic and Layout View Synchronization

Syncing in Layout XL

Syncing in Schematics XL

Syncing Design Intent When Schematic and Layout are Open

Syncing Design Intent When Only Schematic is Open

Navigator Interaction with Design Intent

Create Design Intent Form

Edit Design Intent Form

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 or 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.

Related Topics

Navigator Interaction with Design Intent

Design Intent Interaction Between Schematics XL and Layout XL

Implementation Status of Design Intent

Syncing Design Intents

You can either sync an individual design intent or all design intents to update any design intent changes in the schematic and layout views to ensure that both systems are synced.

Syncing Design Intent When Schematic and Layout are Open

To review and update design intents in a design:

1. Ensure that the corresponding view is open in Schematics XL and Layout XL. TheLayout XL can be read-only.
2. Review the updates for each design intent or object with a sync notification displayed.
3. Depending on whether you are syncing only a selected design intent, or multiple design intents one at a time,
   - To sync only a selected design intent, in the Navigator or on the canvas, right-click the design intent 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 intents in the design is up to date.
   - When syncing multiple design intents simultaneously, right-click the object associated with the design intent in Navigator or in the canvas (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. When syncing multiple design intents simultaneously, repeat for any remaining objects with sync notifications displayed in the Navigator.

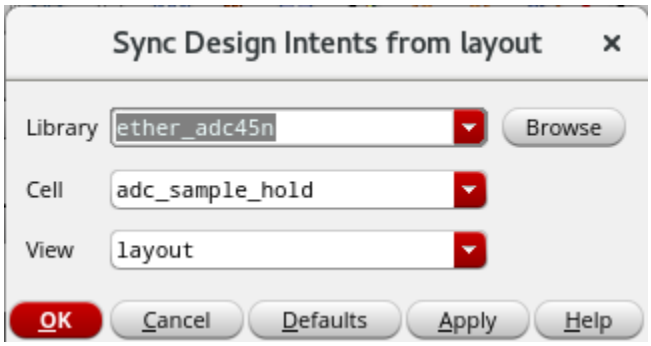
Design intents implemented using constraints do not generate a sync indication.

Syncing Design Intent When Only Schematic is Open

To synchronize a design intent in a schematic view with the changes in the layout view, when no corresponding layout view is opened, perform the following steps:

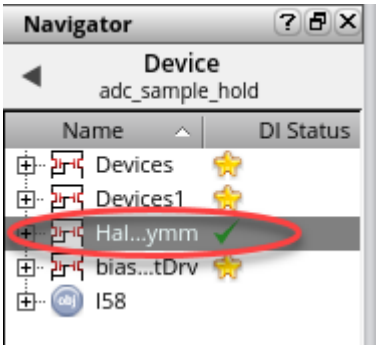
1. Ensure that the design is open in Schematics XL.
2. Review the updates for each design intent or object with a sync notification displayed. For more information, see Alert for Synchronization Required in Schematic.
3. In the Navigator or on the canvas in Schematics XL, right-click and choose *Design Intent – Sync... – Sync All Design Intent.* Alternatively, select the required DI and click the *Sync* button on the toolbar.

   The Sync Design Intents from Layout form displays. The form is displayed only if the associated layout view for the schematic is not open.



4. Select the library, cell, and view name that contains the DI in the layout for which the selected DI is to be synchronized in Schematic XL.
5. Click *OK*.

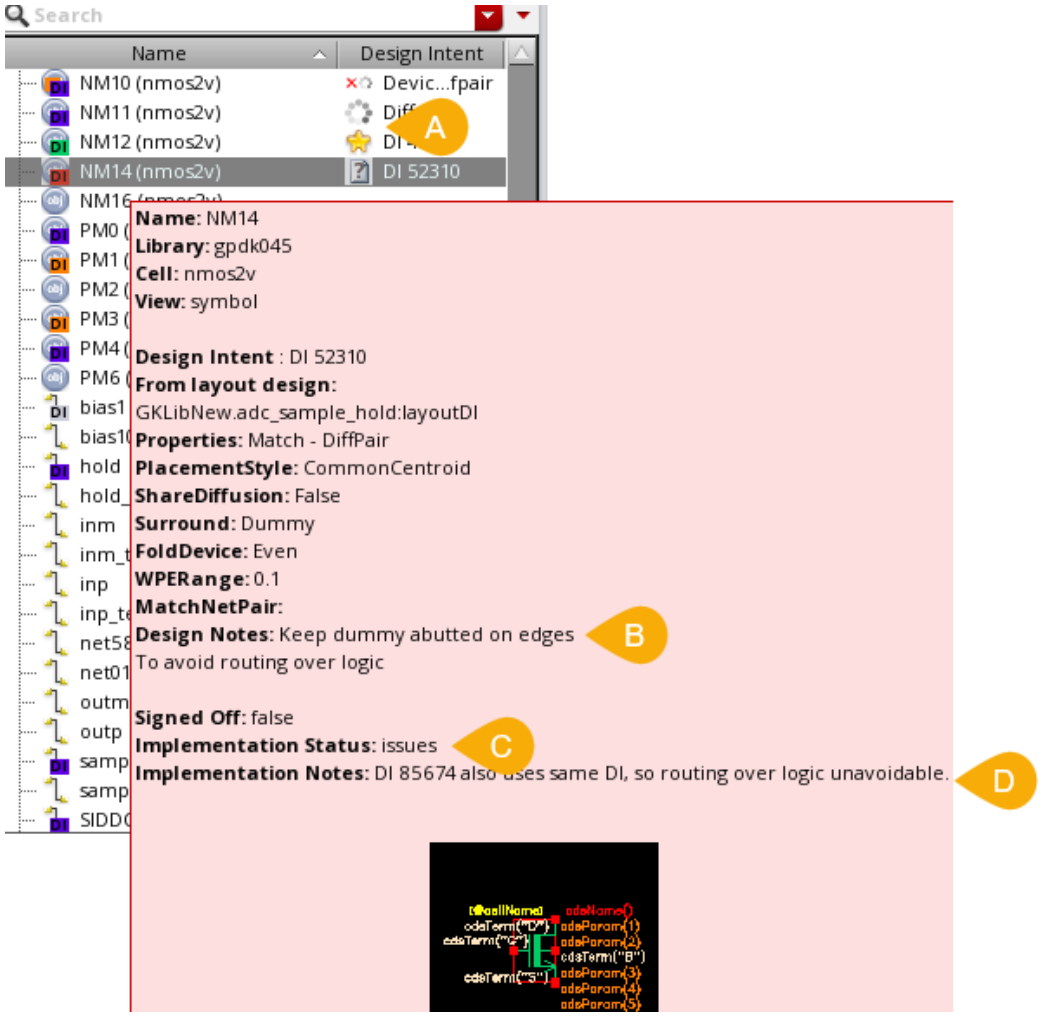
   The design intent in the design is synchronized between the schematic and layout cellviews.



Design intents that are present only in the layout cellview are not deleted if their implementation status is `review`. The sync icons are removed and the *Sync* button on the toolbar is disabled, indicating that the design is now up-to-date.

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

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.

Related Topics

Edit Design Intent Form

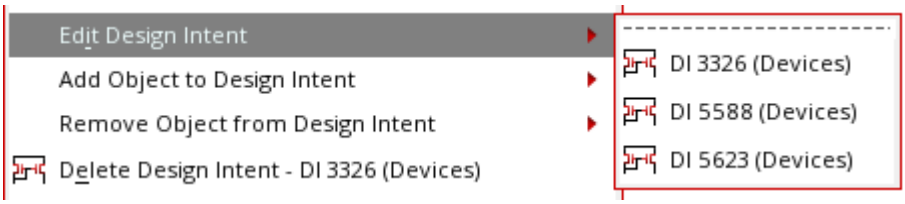
Implementation Status of Design Intent

Editing a Design Intent

You can edit a design intent using the Editing 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 glyph of the object and click *Edit* on the Design Intent Info Balloon that is displayed.
- Click the glyph of the object 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.



Related Topics

Edit Design Intent Form

Editing a Design Intent in Schematics XL

Editing a Design Intent in Layout XL

Design Intent Annotations

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

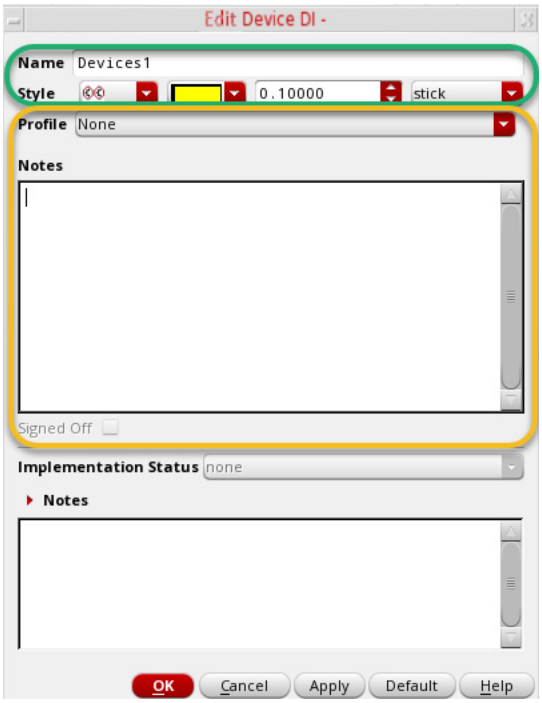
To edit a design intent in Schematics XL,

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 Edit Design Intent form is displayed. 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.

   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.

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 *HighCurrent* or *MaxVoltageDrop*, which display the object name rather than the design intent name, and therefore cannot be amended in this way.

Related Topics

Create Design Intent Form

Schematic and Layout View Synchronization

Reviewing and Editing a Design Intent Created in Layout XL

Design Intent Functions

Editing a Design Intent in Layout XL

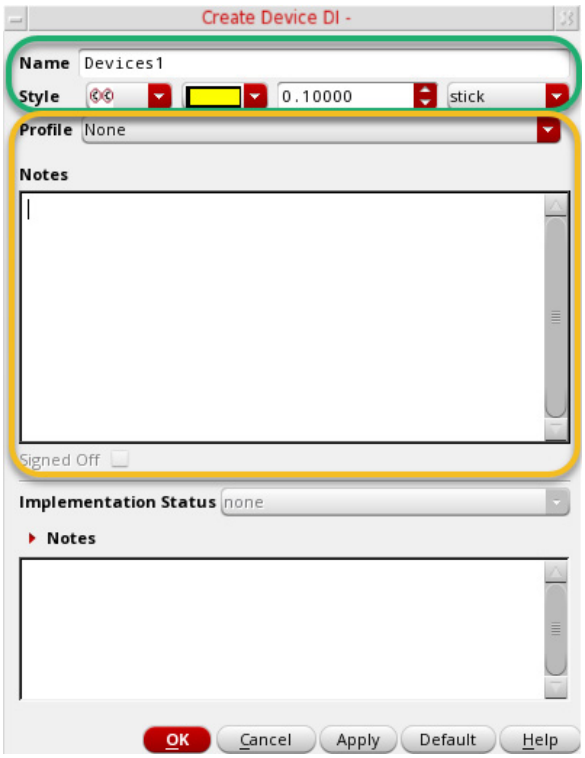
To edit a design intent in Layout XL,

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.

Related Topics

Create Design Intent Form

Schematic and Layout View Synchronization

Adding a Physical Constraint to an Existing Design Intent

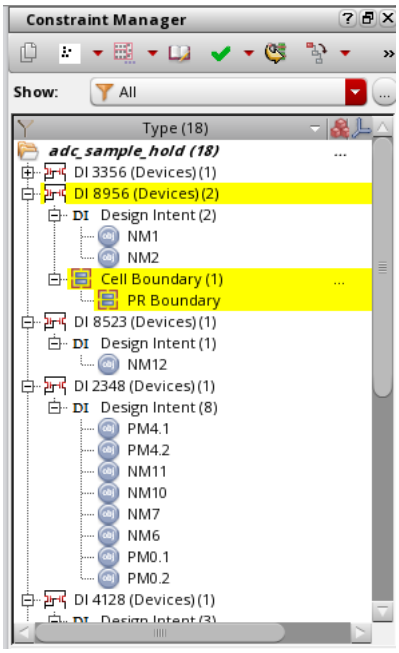
Note: This action is not necessary in either Schematics XL or Layout XL when using the *Design Intent – Sync* option as this synchronizes only the design intent information.

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.

Related Topics

Constraint Manager Assistant

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.

Related Topics

Create Design Intent Form

Schematic and Layout View Synchronization

Adding Objects to a Design Intent

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. Choose *Design Intent – Add Object to Design Intent*.

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

2. Choose the design intent to which the objects are to be added.

   Design intent annotations are created on the canvas for the selected objects.

   When adding a HighCurrent or MaxVoltageDrop design intent, the Inst Terms form is opened. On the form, members that are new or selected to be added; are marked with the text **selected**. This way the new or selected instTerms can be easily distinguished.

   Note: You cannot add unrelated nodes to HighCurrent and MaxVoltageDrop design intents.

3. 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.

   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.

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.

Related Topics

Design Intent Annotations

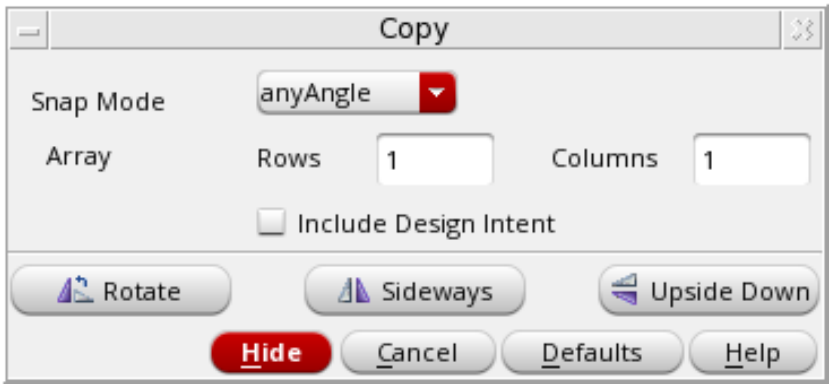
Edit Design Intent Form

Editing a Design Intent

Copying and Renaming 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.

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.

Related Topics

Copy Form

Editing Object Properties (Text Edit) Directly on the Design Canvas

Removing Objects from Design Intents

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.

   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.

   Note: You cannot remove a reference node using this method.
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.

   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.

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.

Related Topics

Editing a Design Intent

Schematic and Layout View Synchronization

Reviewing and Editing a Design Intent Created in Layout XL

Deleting a Design Intent

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

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.

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.

Related Topics

Deleting Design Intent on Objects

Removing Objects from Design Intents

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*.
3.

   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.

Related Topics

Removing Objects from Design Intents

Schematic and Layout View Synchronization

Deleting All Design Intent in Cellview

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

Caution

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.

Related Topics

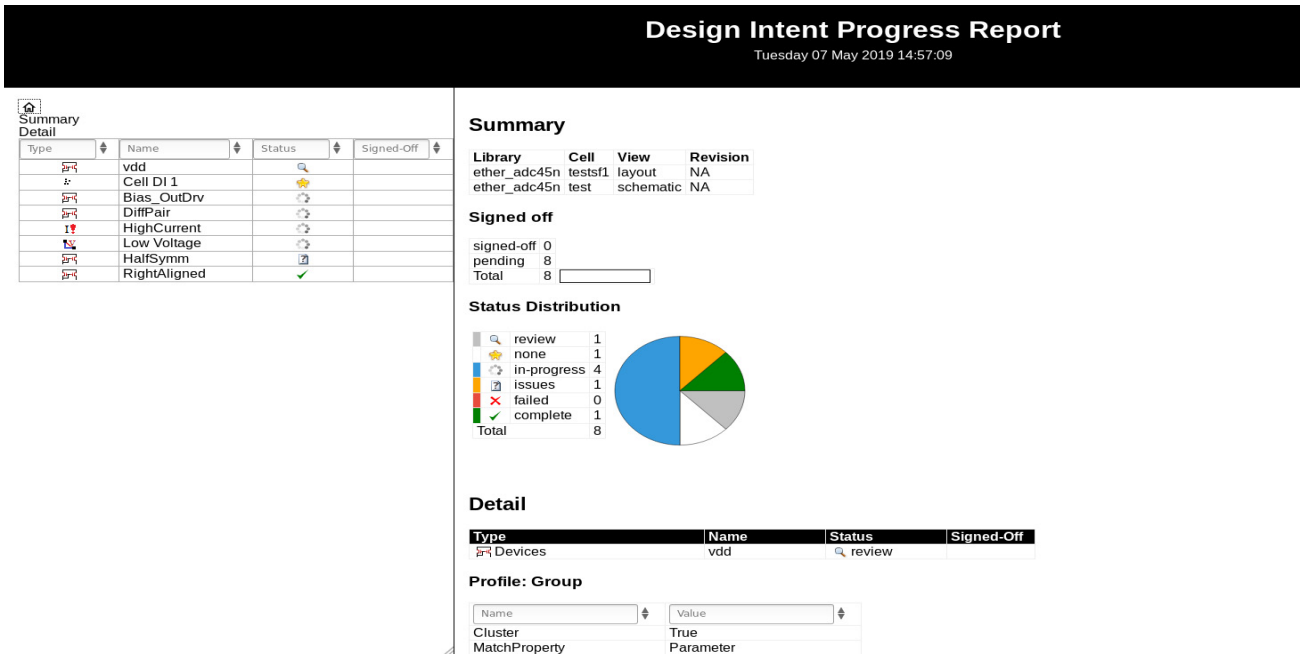
Deleting All Design Intent in Cellview

Deleting Design Intent on Objects

Schematic and Layout View Synchronization

Design Intent Report

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.

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.

On the navigation pane:

Field	Description
Summary	Displays the design intent summary in the report pane.
Detail	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:

Field	Description
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. ■ **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. ■ **Profile** displays the property configurations assigned to the design intent. ■ **Members Information:** Displays the objects on which the design intent applies and the voltage or current that has been specified for each category. ■ **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.

Related Topics

Signed Off

Reviewing and Editing a Design Intent Created in Layout XL

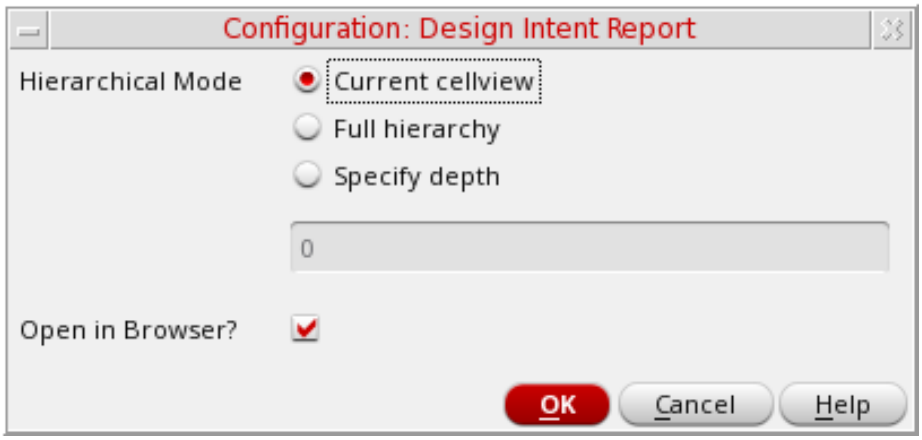
Signing off Design Intent Reports

Notes

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.

Signing off Design Intent Reports

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.

Note: If you change the design intent in either Schematics XL or Layout XL after setting the implementation status to *complete* ✓, the status is automatically reset to *none* ☆.

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 Design Intent Report is run, the *Signed-off* column displays a ✓ icon against the design intent.

Related Topics

Create Design Intent Form

Design Intent Report

A

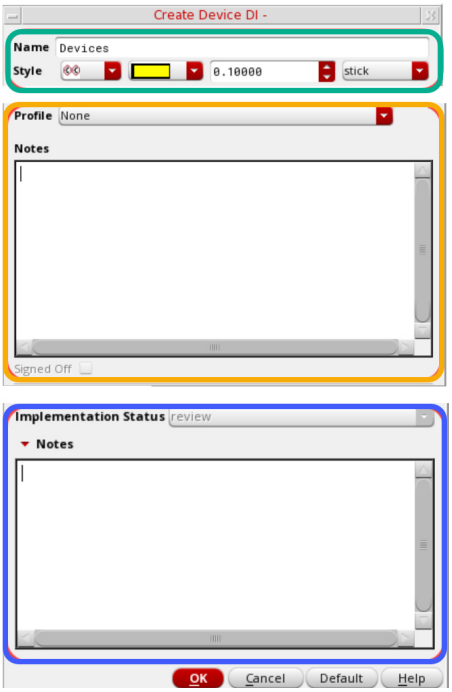
Design Intent Forms

The design intent feature provides you with the following forms that enable you to create and edit design intents.

- Sections in the Create/Edit Design Intent Form
- Create Design Intent Form
- Edit Design Intent Form

Sections in 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 Design Intent Report.

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.

Create Design Intent Form

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.

Field	Description
Name	Specifies 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	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* Specifies 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 ❑  Elliptical Group 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. By default, the colors used when creating design intents now cycles through the available color palette. To create all design intents in the default color instead, set the `CI_ENABLE_DI_AUTO_COLORS` shell environment variable to `0` before you start Virtuoso. *Font Size* Specifies the text size used in the annotation label on the canvas. Environment variable: diNoteFontHeight 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.
The following options are available only in Schematics XL.	
Profile (optional)	Selects 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. **Note:** The categories *HighCurrent* and *MaxVoltageDrop* automatically have profiles preselected, *Current* and *Voltage* respectively. These cannot be overridden. You can also change the property profile selected for a design intent using the ciTemplateChangeDIProfile function. For more details, see Design Intent Functions. 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.
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	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. All non-Latin characters are supported. **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.
The following options are available only 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.
Notes	Documents 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. **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

Navigator Interaction with Design Intent

Property Profiles

Creating a HighCurrent Design Intent

Edit Design Intent Form

Use the Edit Design Intent form to update 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.

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

Field	Description
The following options are available only 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 ❑ Elliptical Group 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.

Font Size
Specifies the text size used in the annotation label on the canvas.

Environment variable: diNoteFontHeight

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.

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.

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*.

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.

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.
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. All non-Latin characters are supported. 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. This option is enabled only for design intents with a status of Complete  or `review` .
The following options are available only 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.

Notes	Documents 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. **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

Design Intent Functions

Editing a Design Intent

Editing a Design Intent in Schematics XL

Editing a Design Intent in Layout XL

Navigator Interaction with Design Intent

Property Profiles

Signing off Design Intent Reports

B

Design Intent Environment Variables

The design intent feature uses the following environment variables:

- diComplyKirchhoffLaw
- diNoteFontHeight
- diPinInfoBalloonCellDI
- diPropDefaultScopes
- diPropScopesFilter
- diShowAnnotations
- diShowLayoutGlyphs
- diShowLayoutHalos
- diShowSchematicGlyphs
- diShowSchematicHalos

diComplyKirchhoffLaw

```
constraint diComplyKirchhoffLaw boolean { t | nil }
```

Description

Ensures compliance with Kirchhoff's Current Law, meaning that the sum of all currents entering and exiting a node is zero. This variable corresponds to the *Comply with Kirchhoff's Current Law* check box in the Inst Terms form. The default is `nil`.

GUI Equivalent

None

Example

```
envGetVal("constraint" "diComplyKirchhoffLaw")
envSetVal("constraint" "diComplyKirchhoffLaw" 'boolean t)
```

Related topics

Creating a HighCurrent Design Intent

diNoteFontHeight

`constraint diNoteFontHeight float` *`font_height`*

Description

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

GUI Equivalent

Command	*Design Intent – Create Design Intent* or *Design Intent – Edit Design Intent*
Form Field	*Font Size*

Example

```
envGetVal("constraint" "diNoteFontHeight")
envSetVal("constraint" "diNoteFontHeight" 'float 0.2)
```

Related topics

Create Design Intent Form

Edit Design Intent Form

diPinInfoBalloonCellDI

```
constraint diPinInfoBalloonCellDI boolean { t | nil }
```

Description

Automatically pops up the cell design intent info balloon when the schematic is opened. You can pop up info balloons of any design intent type selectively or using *Show/Hide Info Balloons for design intents in the cellview* button in the *Design Intent Toolbar*. The default is `nil`.

GUI Equivalent

None

Example

```
envGetVal("constraint" "diPinInfoBalloonCellDI")
envSetVal("constraint" "diPinInfoBalloonCellDI" 'boolean t)
```

Related Topics

Design Intent Info Balloon

diPropDefaultScopes

```
constraint diPropDefaultScopes string "scopeName"
```

Description

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 Form and Edit Design Intent Form, 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

```
envGetVal("constraint" "diPropDefaultScopes")
```

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

Create Design Intent Form

Edit Design Intent Form

diPropScopesFilter

```
constraint diPropScopesFilter string "scopeName"
```

Description

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 Form and Edit Design Intent Form

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

GUI Equivalent

None

Example

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

Related topics

Defining Scopes for Profile Properties

Create Design Intent Form

Edit Design Intent Form

diShowAnnotations

```
constraint diShowAnnotations boolean { t | nil }
```

Description

Controls whether design intent annotations are visible on the canvas. The default is `t`.

GUI Equivalent

Design Intent Toolbar – Toggle Design Intent Annotation Visibility

Example

```
envGetVal("constraint" "diShowAnnotations")
envSetVal("constraint" "diShowAnnotations" 'boolean nil)
```

Related Topics

Design Intent Annotations

diShowLayoutGlyphs

```
constraint diShowLayoutGlyphs boolean { t | nil }
```

Description

Controls whether design intent glyphs are visible on the layout canvas. The default is `t`.

GUI Equivalent

Design Intent Toolbar (Layout) – Toggle Design Intent Glyph Visibility

Example

```
envGetVal("constraint" "diShowLayoutGlyphs")
envSetVal("constraint" "diShowLayoutGlyphs" 'boolean nil)
```

Related Topics

Design Intent Glyphs

diShowLayoutHalos

```
constraint diShowLayoutHalos boolean { t | nil }
```

Description

Controls whether the selected design intent is haloed on the layout canvas. The default is `t`.

GUI Equivalent

Design Intent Toolbar (Layout) – Toggle Design Intent Halo Visibility

Example

```
envGetVal("constraint" "diShowLayoutHalos")
envSetVal("constraint" "diShowLayoutHalos" 'boolean nil)
```

Related Topics

Design Intent Haloing

diShowSchematicGlyphs

```
constraint diShowSchematicGlyphs boolean { t | nil }
```

Description

Controls whether design intent glyphs are visible on the schematic canvas. The default is `t`.

GUI Equivalent

Design Intent Toolbar (Schematic) – Toggle Design Intent Glyph Visibility

Example

```
envGetVal("constraint" "diShowSchematicGlyphs")
envSetVal("constraint" "diShowSchematicGlyphs" 'boolean nil)
```

Related Topics

Design Intent Glyphs

diShowSchematicHalos

```
constraint diShowSchematicHalos boolean { t | nil }
```

Description

Controls whether the selected design intent is haloed on the schematic canvas. The default is `t`.

GUI Equivalent

Design Intent Toolbar (Schematic) – Toggle Design Intent Halo Visibility

Example

```
envGetVal("constraint" "diShowSchematicHalos")
envSetVal("constraint" "diShowSchematicHalos" 'boolean nil)
```

Related Topics

Design Intent Haloing

C

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` file 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.

You can associate a property with multiple scopes to control which properties are visible for the current profile.

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.

Related Topics

Create Design Intent Form

Defining Scopes for Profile Properties

Profile Properties Syntax

```
list(
  ;; Property Group DPL
  list(nil 'name     "GroupName"
           'category [ "Cell" | "Devices" | "Nets" | "Pins" | "MaxVoltageDrop" |
                       "HighCurrent" ]
           'aprGroup [ 't | 'nil ]
           ['toolTip "toolTip"]
           ['properties
                list(
                  list(nil 'name "PropertyName"
                           'type ['bool|'string|'int|'float|'enum|'current|
                                  'voltage|'aprPattern]
```

```
                                ['widget    'multiLineTextEdit]
                                ['defValue value]
                                ['range     '(low high)] ;; int/float range
                                ['items     '("item1" ... "itemN") ] ;; enum choices
                                ['callback S_callback]
                    )
                )
            ]
        )
)
```

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`.
`'aprGroup`	Specify that the device group should be treated as an APR group by the Auto Device P&R assistant. The options are `'t` and `'nil`.
`'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`	The profile name to be displayed on the Create or Edit Design Intent forms. It is replaced with the value for `'guiName`, if specified.

`'scopes`	A list of scopes associated with the property. This attribute is required only if the property is to be associated with 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`, `'voltage` or `'aprPattern`.
`'widget`	Use multi-line text fields for properties instead of the default single-line text field. The only option is `'multiLineTextEdit`.
`'guiName`	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. The dpl format for specifying default values for the `aprGroup` is: `;(list nil Pattern "None"|"Preset"|"Template" PatternPreset "Clustered"|"Interdigitate"|"Compact"|" Custom"... PatternString t_string_patter DeviceGroup "Generic Group"|... TemplateFile fileName)`
`'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)`.
`'callback`	Set a callback function symbol on the callback attribute of a property. The callback is triggered when the specified value in the dialog changes.

Example of a Property Profile

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

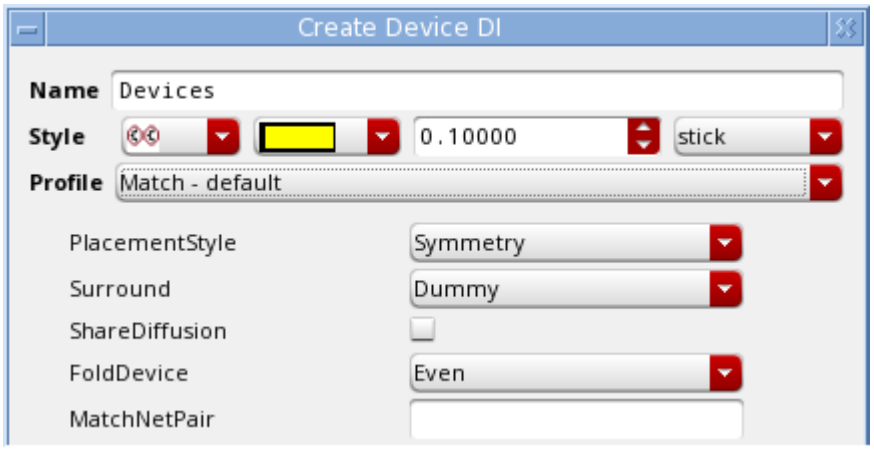
In the property group definitions file, this profile is listed as follows:

```
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 "")
```

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"))
```

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



The following example illustrates how to add callback function on a profile property:

```
(defvar myProfilesWithCB
  '((nil name "Profile With CB"
          category "Devices"
          tooltip "DI with callback available in the dialog"
          properties ((nil name "Prop bool"
                          type bool
                          defValue t
                          callback S_callback)
                      (nil name "Prop int"
                          type int
                          defValue 0
                          callback S_callback)
                      (nil name "Prop float"
                          type float
                          defValue 0.0
                          callback S_callback)
                      (nil name "Prop enum"
                          type enum
                          defValue "value1"
                          items ("value1" "value2" "value3")
                          callback S_callback)
                      (nil name "Prop string"
                          type string
                          defValue ""
                          callback S_callback)
                      (nil name "Prop ML string"
                          type string
                          defValue ""
                          widget multiLineTextEdit
                          callback S_callback))))))
```

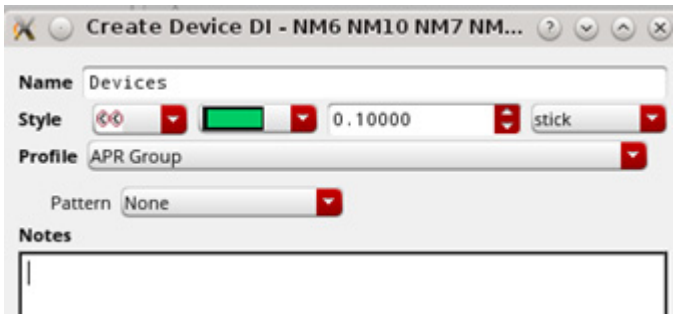
You can evaluate a custom procedure when a profile property is used.

Example of an APR Group Profile

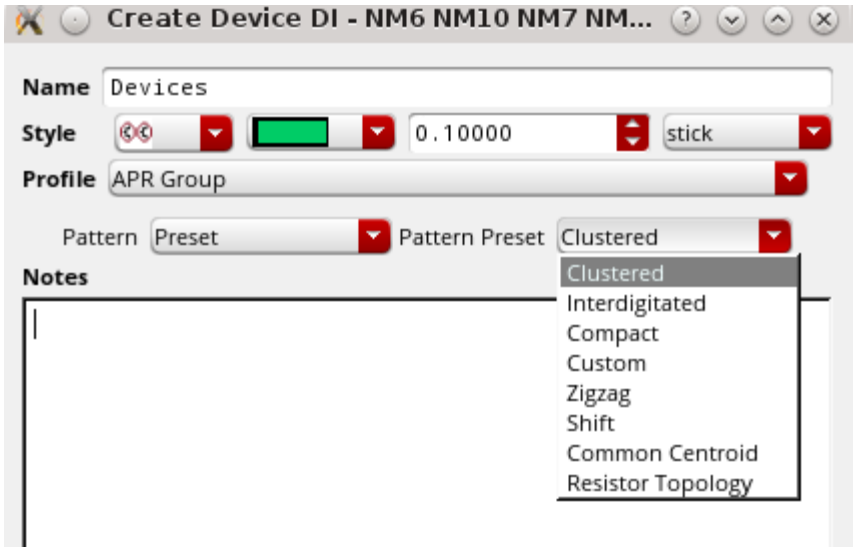
The *APR Group* profile comprises the properties *Pattern*.

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

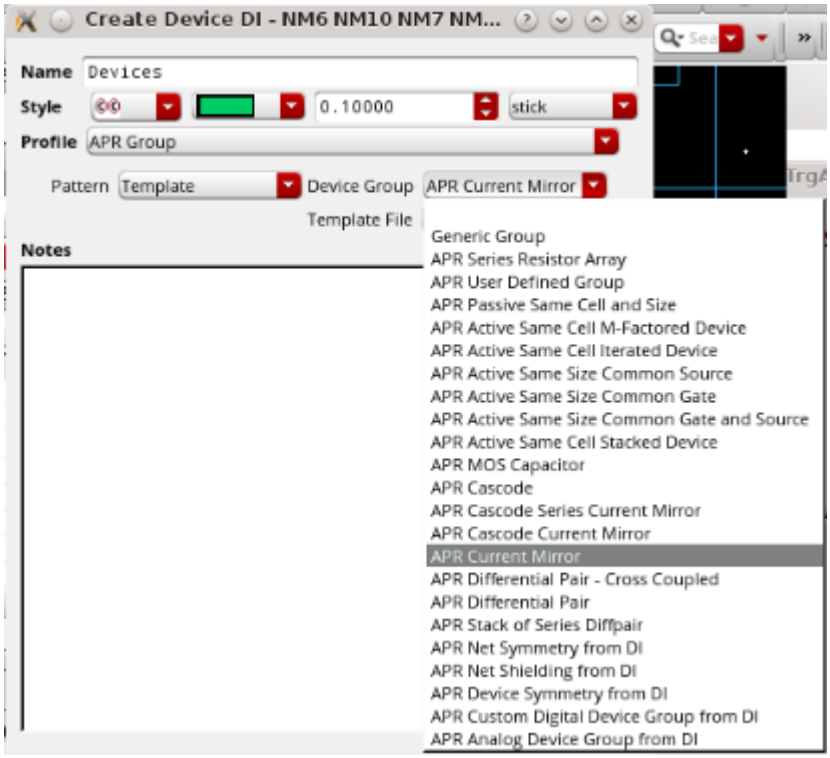
```
list(

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

    list(nil 'name "APR Group" 'category "Devices" 'toolTip "How devices are to be
grouped" 'aprGroup t 'properties list(
                        list(nil 'name "Apr" 'type 'aprPattern
                         )

        )
```



You can specify an pattern set as shown in the following figure.



You can also specify the APR finder for device group. This can be applied when used in layout with APR flow. In addition, there is a provision to include an existing template file to reuse the configuration.

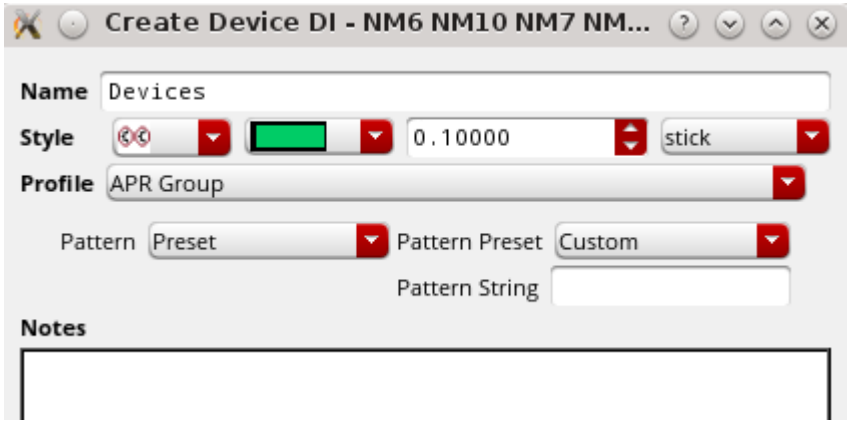


Setting Default Value for APR Group Design Intent

You can customize to specify the state and values of a field when opened as a default setting in the Profile - APR Group.

```
list(nil 'name "APR Group" 'category "Devices" 'toolTip "How devices are to be
grouped" 'aprGroup t
            'properties list(
              list(nil 'name "Apr" 'type 'aprPattern 'defValue '(nil Pattern
"Preset" PatternPreset "Custom"))
             )
```

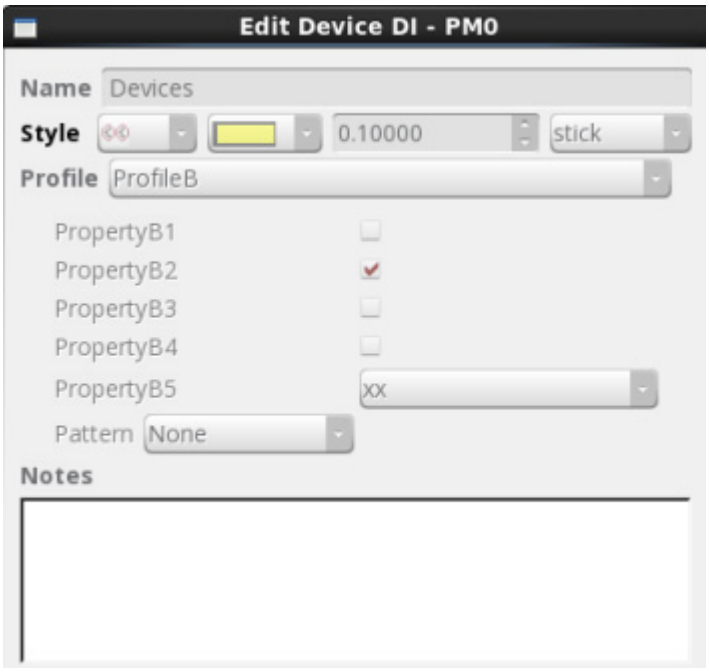
```
)
```



To extend the APR group with additional custom properties, you can either update the `diPropGroupDefs.il` or update on the fly in the current Virtuoso session using the following.

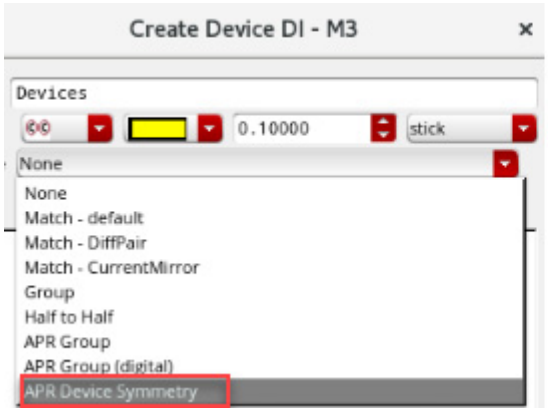
```
ciDiReplaceOrAddPropertyGroupDef(
list(nil 'name "ProfileB" 'category "Devices" 'toolTip "***" 'aprGroup t
'properties list(
                list(nil 'name "PropertyB1" 'type 'bool 'defValue nil)
                list(nil 'name "PropertyB2" 'type 'bool 'defValue t)
                list(nil 'name "PropertyB3" 'type 'bool 'defValue nil)
                list(nil 'name "PropertyB4" 'type 'bool 'defValue nil)
                list(nil 'name "PropertyB5" 'type 'enum 'defValue "xx" 'items
'("xx" "yy" "zz"))
                list(nil 'name "Apr" 'type 'aprPattern 'defValue '(nil Pattern
"Preset" PatternPreset "Custom"))
)))
```

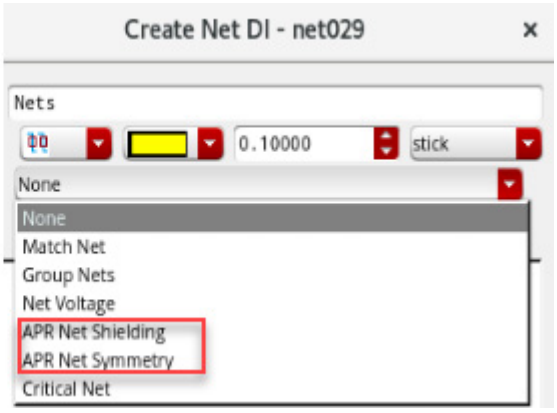
On the Create Design Intent form this profile is displayed as follows:



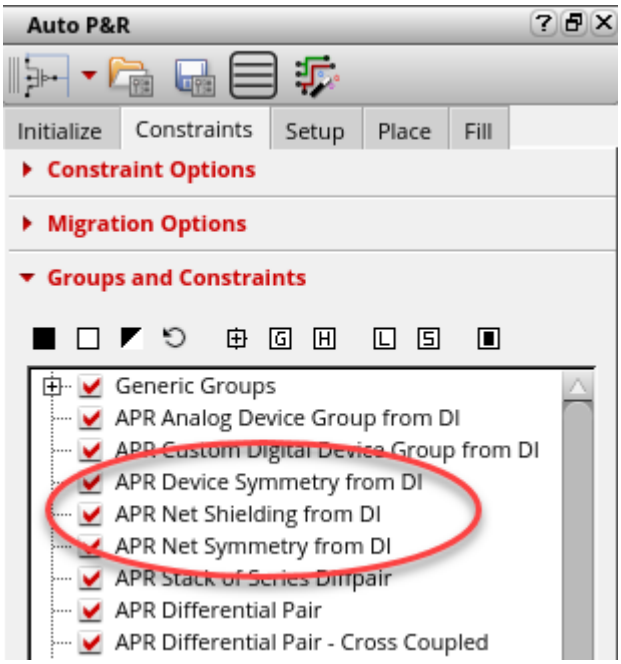
Additionaly, there are more DI profiles available by default. For example:

- **For devices**: *APR Device Symmetry*
- **For Nets**: *APR Net Symmetry* and *APR Device Symmetry*





The automatic placement and routing flow in layout can search for the design intents with APR profile defined on them to create right set of constraints.



Related topics

Property Profiles

Modifying Sample Profiles and Property Definitions

Profile Property Scopes

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 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`). 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.

Related topics

Property Profiles

Profile Properties Syntax

Profile Property Scopes

Cadence Application Infrastructure User Guide

Create Design Intent Form

Edit Design Intent Form

Profile Property Scopes

Using scopes in your design lets you control the visibility of the property settings for the current profile on the Create Design Intent Form and Edit Design Intent Form. 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.

Related Topics

Create Design Intent Form

Edit Design Intent Form

Defining Scopes for Profile Properties

Defining Scopes for Profile Properties

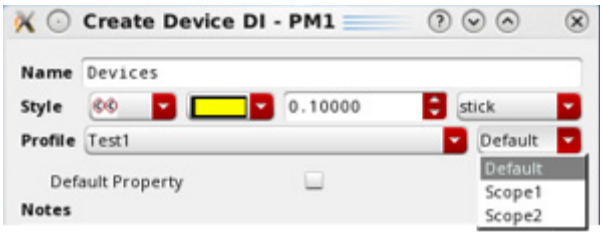
To define a scope:

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.