UG0738
User Guide
Netlist Viewer Interface
Libero SoC v12.0

Microsemi

a **MICROCHIP** company

Microsemi

a **MICROCHIP** company

About Microsemi

Microsemi, a wholly owned subsidiary of Microchip Technology Inc. (Nasdaq: MCHP), offers a comprehensive portfolio of semiconductor and system solutions for aerospace & defense, communications, data center and industrial markets. Products include high-performance and radiation-hardened analog mixed-signal integrated circuits, FPGAs, SoCs and ASICs; power management products; timing and synchronization devices and precise time solutions, setting the world's standard for time; voice processing devices; RF solutions; discrete components; enterprise storage and communication solutions, security technologies and scalable anti-tamper products; Ethernet solutions; Power-over-Ethernet ICs and midspans; as well as custom design capabilities and services. Learn more at www.microsemi.com.

Revision History

The following table shows changes made in this document for each revision.

Revision	Changes
Revision 4 (December 2018)	Document template updates and minor text edits
Revision 3 (October 2017)	Add Flat Post-Compile Cone View
Revision 2 (May 2017)	Minor updates
Revision 1 (December 2016)	Initial release

Contents

List of Tables

List of Figures

1 Introduction

The Netlist Viewer is a graphical representation of the design netlist. As FPGA designs grow in size and complexity, it has become essential for the FPGA designer to traverse the netlist to analyze the design.

1.1 Supported Families and Platforms

The Netlist Viewer supports SmartFusion2, IGLOO2, RTG4, and PolarFire devices and runs on Windows and Linux systems.

Note: Depending on the device selected, some UI elements such as icons, options, and dialog boxes may vary slightly in appearance and/or content. Basic Netlist Viewer functionality remains the same, regardless of the device chosen.

1.2 Views

The Microsemi Netlist Viewer is a graphical user interface that displays different netlist views:

- RTL Netlist view - Technology-independent netlist view of the design before mapping of the design elements to the Microsemi-specific technology. Using the RTL view is a fast and easy way to determine whether the correct logic has been implemented by the software. Cross-probing from this view to the HDL code facilities troubleshooting when the design is not working as desired.
- Hierarchical Post-Synthesis view - Hierarchical view of the netlist after synthesis and after technology mapping to the Microsemi FPGA technology.
- Flat Post-Compile view - A flattened netlist after synthesis, technology mapping and further optimization based on the DRC rules of the device family and/or die.
- Flat Post-Compile Cone view - This view loads the same netlist as the Flat Post-Compile view, but does not initially draw anything on the canvas. Important parts of the design can be added to the canvas from the tree or from the existing items in the view. This view opens much more quickly than the Flat Post-Compile view. It allows you to load only the parts of the design you are interested in. This view is well-suited for use with large designs. This view is not available for all families.

Figure 1 • **RTL View**
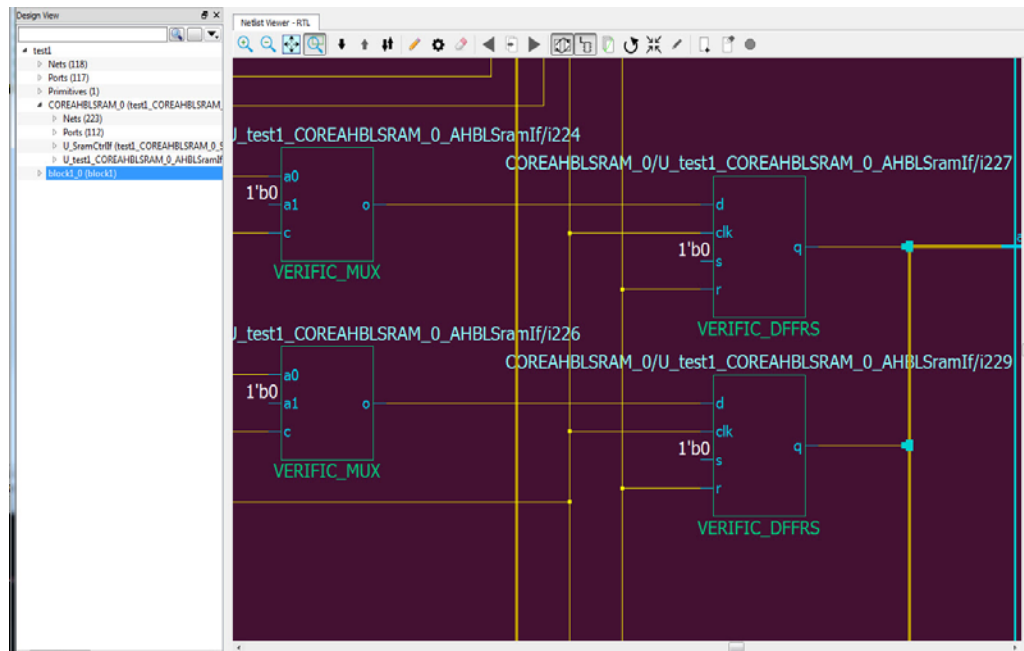
Figure 2 • **Hierarchical Post-Synthesis View**
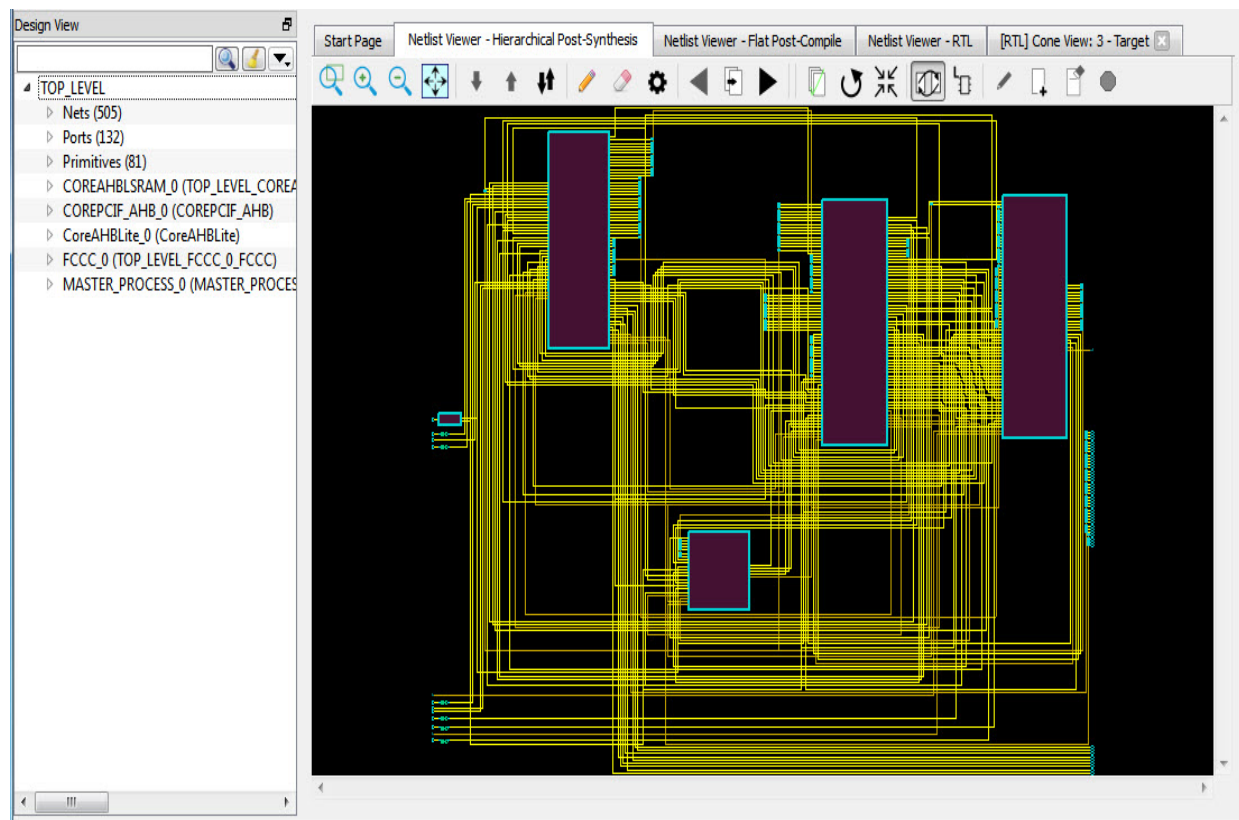
Figure 3 • **Flat Post-Compile View**
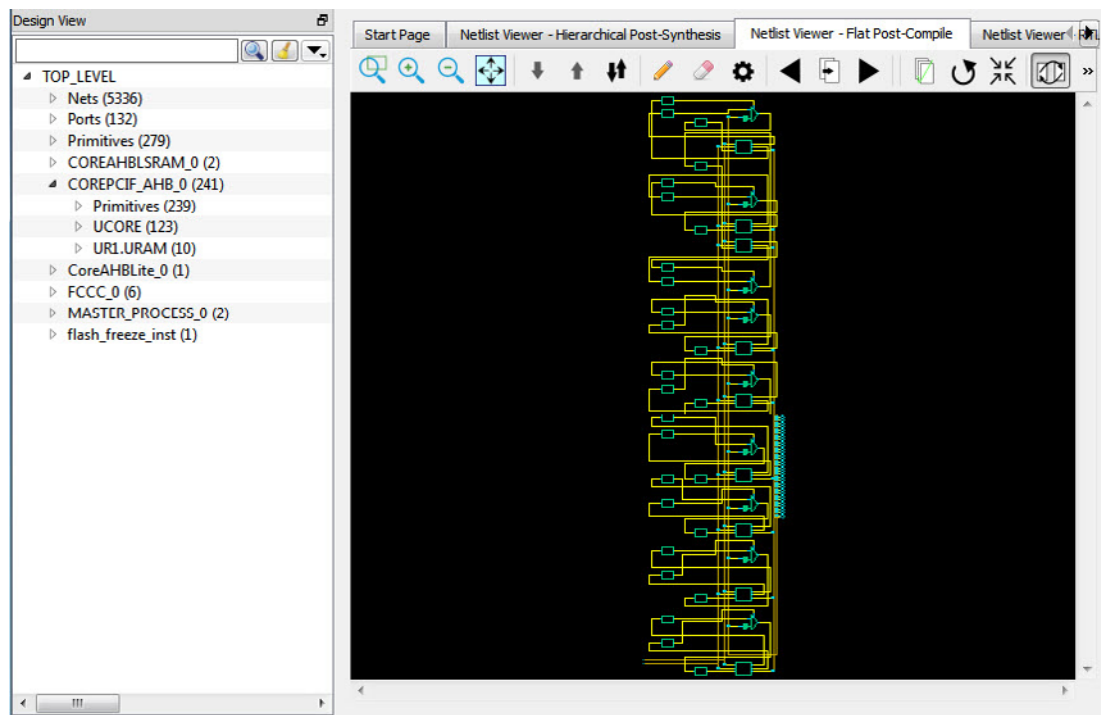
Note: A progress bar pops up to indicate the flattened netlist is being loaded. For a large netlist, the loading may incur some runtime penalty. A **Cancel** button is available to cancel the loading.

Figure 4 • **Flat Post-Compile Cone View**
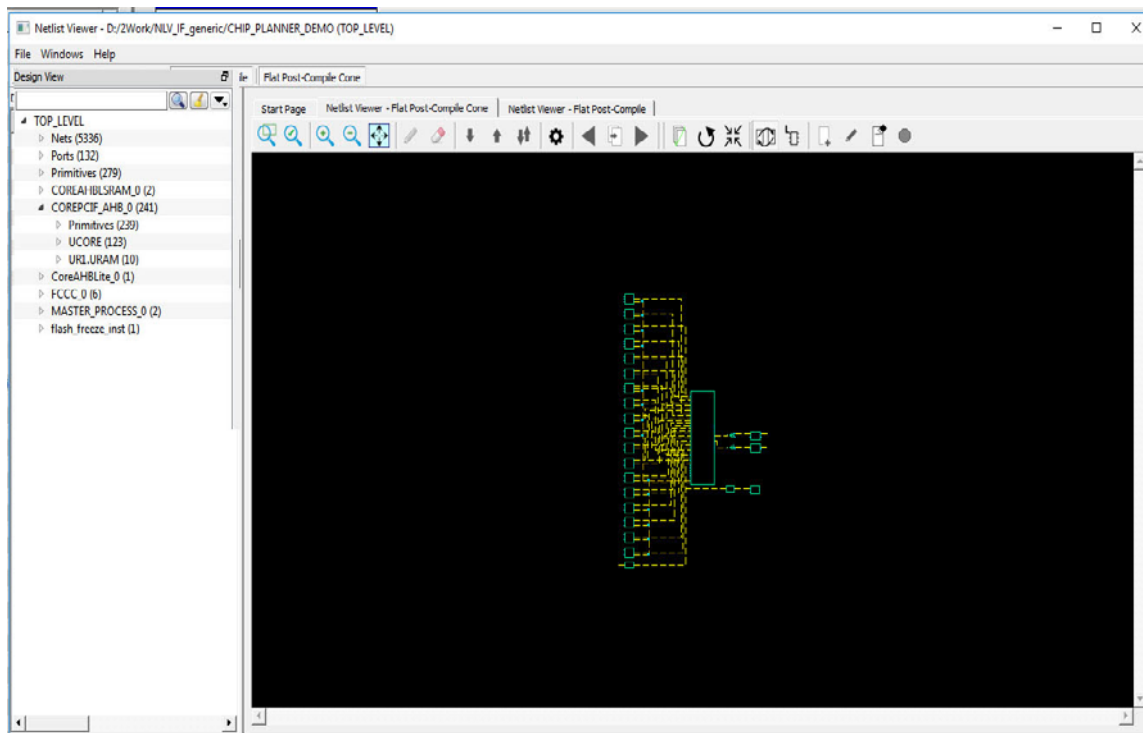
2 The Netlist Viewer User Interface

The Netlist Viewer is a Graphical User Interface (GUI) designed for user to traverse the design hierarchy and to examine the nets for debugging purposes. It has a rich set of icons for navigating the netlist, both vertically (up and down the design hierarchy) and horizontally (to different pages of the netlist view).

2.1 Netlist Viewer Icons

The Netlist Viewer Toolbar has a list of icons. A tooltip appears when the mouse is hovered over the icon.

Figure 1 • **Netlist Viewer Toolbar and Icons**

Table 1 · NelistViewer Icons lists the icons used to navigate the netlist.

Table 1 • **NelistViewer Icons**

Icon	Name	Descriptions	Notes
	Zoom Tool	When activated (ON), cursor operation is in the zoom to selection mode. Left-click and drag to lower right to zoom to selection.	Default setting is OFF.
	Rubber Band Select	When activated (ON), selects a rectangular area in the canvas. Click in the canvas view and drag the mouse to the lower right to delineate a rectangular area. Release the mouse and all design objects inside the delineated area are selected.	
	Zoom In	Zoom in to 2x of original (magnitude of +1)	Alternatively, left-click mouse and drag to upper left to zoom in.
	Zoom Out	Zoom out to one-half of original (magnitude of -1)	Alternatively, left-click mouse and drag to upper right to zoom out.
	Zoom To Fit	Zoom to fit the size of canvas	Alternatively, left-click mouse and drag to lower left to zoom to fit.
	Push Into Module	With a module selected, click this icon to go down the design hierarchy one level at a time	Disabled if the current selected module is the lowest level in the hierarchy

Table 1 • **NelistViewer Icons** *(continued)*

Icon	Name	Descriptions	Notes
	Pop Out of Module	With a module selected, click this icon to go up the design hierarchy one level at a time.	Disabled if current module is the top level in the hierarchy
	Push/Pop Mode	Click icon to activate (ON). Single clicking on a selected module will push into it and single clicking on black space will pop out of the current module. Default setting is OFF.	Cursor operation in one of four modes: • Zoom mode • highlight mode (ON/OFF) • push/pop mode (ON/OFF) • selection mode (default)
	Highlight Tool	With a design object (net, instance, port) selected, single click this icon (ON) to highlight the selected item with a color the user sets in the Properties setting.	Highlights the object with a color per the user's preference in the Properties setting.
	Remove Highlight	Removes all highlights from current view	
	Properties	Pops up a properties window to set preferences for the highlight color, selection color, design object colors, enable/disable display of design object names.	See the Properties Window Section for details.
	Previous Page	Goes to the previous page (when the netlist view span multiple pages).	Disabled if currently on the first page.
	To Page	Pops up a window to display the current page number. Change this number to go to a different page.	Disabled if design has only one page.
	Next Page	Goes to the next page (when the netlist view span multiple pages)	Disabled if currently on the last page.
	Cross Probing	Toggles the cross probing between the design view and the net list views, the Chip Canvas view and the World View (Chip Planner only)	When enabled (ON), modules selected in the netlist view are automatically selected in the design view and vice versa. Default setting is ON.

Table 1 • **NelistViewer Icons** *(continued)*

Icon	Name	Descriptions	Notes
	Local Selection	Toggles local selection option. Local means the same level of hierarchy in the design.	When enabled (ON), the Go to Net Driver traces the selected net to the driver up to the hierarchy boundary (stay local). The same restriction/non-restriction applies when adding to the active cone the drivers/loads of a selected design object. When enabled/local, the design hierarchy boundary is not crossed. Refer to Creating Cones for more information. Default setting is OFF.
	Disable Page Splitting	Toggles the page splitting feature	When enabled (ON), page splitting is disabled and the netlist stays on one page (Note: Multiple large pages for very large designs). When disabled (OFF), the display is split into multiple pages. Default setting is OFF.
	Regenerate View	Reconstructs the netlist view, placing it optimally with minimal empty spaces	-
	Fold All Instances	Folds all expanded modules and collapse them into the parent module.	
	Rename Cone	Opens a dialog box to change the name of the cone	Only the cones view have this feature enabled. Names of cones must be unique across all netlist views.
	Create Cone	Creates a new active cone associated with the view	When a logical cone is first created, it is automatically set as the active cone.
	Remove All	Removes all items in a cone	Disabled if not a cone view.
	Set As Active Cone	Makes the current cone the active cone	Disabled if not a cone view

2.2   Right-Click Mouse Menu Items

A right-mouse click on a design object (Instance, net, module, pin) opens a menu. This menu is context-sensitive and varies with the kind of design objects and the view (RTL view, Flattened View, Hierarchical View). Some of the menu items may contain sub-menu items.

Table x is a list of the menu items for all views and all design objects. Note that Some menu items are specific to a view or specific to a design object

Table 2 • **Right-Click Mouse Menu Items**

Name	Description	Shown When	Notes
Push Instance	Pushes into the module selected	Pushable Module (Not the lowest level) is selected	-
Expand Inplace/ Fold Instance	Expands current module in place inside current view or folds current instance inside current view.	Pushable Module is selected	More information in the Folding/Expanding Module Section
Get Inst Param	Gets all the instance parameters and shows it in a table on a separate window	An Instance with parameters is selected. An example is a RAM HDL module/entity which has parameterized address bus width and Data bus width in the HDL code.	
Open File Location	Opens a new tab and displays the netlist file. The cursor goes to the definition of the item selected. The file is opened for read only and is not editable.	When a Verilog/VHDL/EDIF netlist is opened, and the location of the module in the netlist file is available.	Available only in the RTL view and Hierarchical view of standalone Netlist Viewer.
Go To Net Driver	Goes to the driver of the net selected	Only one net is selected.	Behavior is affected by the Local Selection Mode.When in Local Selection mode, the driver on the current level will be selected. Otherwise, the lowest level driver will be selected (may be on a different level) across hierarchical boundaries.
Follow Net ..	Follows a net to the components it is connected to.	Shown when a net/net bundle that spans multiple pages or crosses hierarchical boundaries is selected.	Does not appear if net/net bundle is not selected. Disabled if net is contained in the current page. More info in the Follow Net Section
Follow Pin	Follows pin to the driver/load of the pin.	A pin on an instance is selected	Highlights the net attached to the pin as well as the driver/load of the net
Magnify Pin	Creates a magnify view of the pin	A pin on an instance is selected	More information in the Magnify Pin Section
Add to Active Cone > Selection	Adds current selected design objects to the Active Cone		Available only when at least one net/instance is selected
Add to Active Cone > Driver	Adds the Driver of all nets and instances selected to the Active Cone	Contains sub-menu to select the number of logic level to traverse to collect all the drivers. Max logic level is 10.	Available only when at least one net/instance is selected
Add to Active Cone > Load	Adds the Load of all nets and instances selected to the Active Cone	Contains sub-menu to select the number of logic level to traverse to collect all the loads. Max logic level is 10.	Available only when at least one net/instance is selected

Table 2 • **Right-Click Mouse Menu Items** *(continued)*

Name	Description	Shown When	Notes
Add to Active Cone > All Connected Logic	Adds all instances connected to adjacent nets to the active cone		Available only when at least one instance is selected
Pop Instance	Pops out of the current module into the next higher level of hierarchy	Currently not on the top level of the design hierarchy.	

2.3 Customizing Display Preferences

The Properties Icon allows you to set personal preference on the look-and-feel of the Netlist Viewer You may set the color preferences for design objects such as nets, instances, ports and the color preference for highlighted design objects.

The Netlist Viewer Display Options dialog box opens when the Properties Icon is clicked. Three tabs are available to configure your personal preferences.

2.3.1 Color Selection

The Color Selection tab allows you to set your color preferences for the different design objects:

- Background Color
- Net Color
- Bus Color
- Instance Color
- Macro Color
- Instance Fill Color
- Instance Name Color
- Pin Color
- Port Color
- Port Name Color
- Attribute Color - The color of the I/O attributes. Examples are 1'b1 for a port tied to VCC or 1'b0 for a port tied to ground. The attribute color applies to the attribute label "1'b1" or "1b0".
- Selection Color - The color that shows when design objects such as instances, modules, nets, pins are clicked and selected.

Figure 2 • **Color Selection Tab**
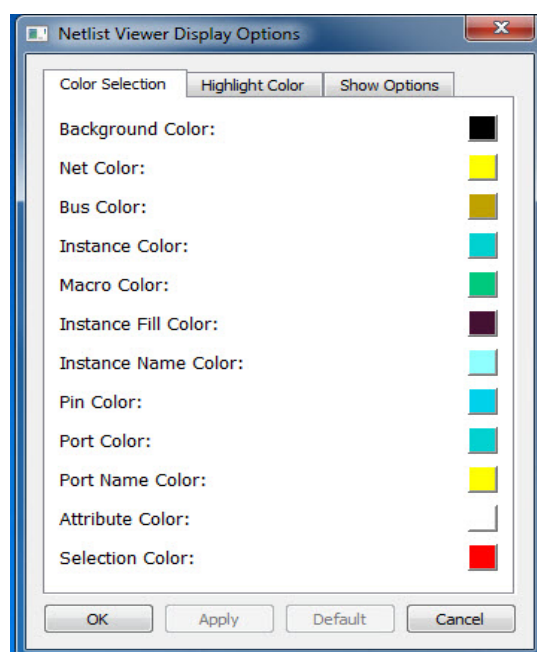
2.3.1.1 Changing the Color Setting

To change the color from the default settings and set your own preference:

1. click the color icon of the item you want to change to open the color palette dialog.
2. Drag the cross-bar across the color spectrum and drop in onto a color of your choice.
3. Click **OK.**

Figure 3 • **Color Palette**
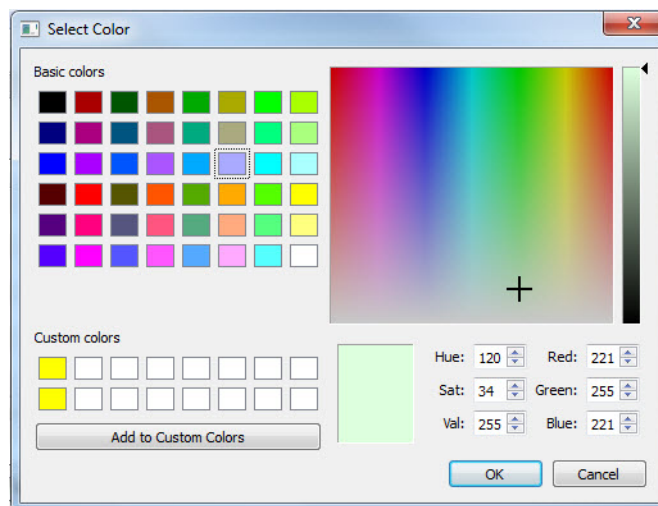
2.3.2 Highlight Color

The Highlight color is the color of the design objects (nets, instances, pins) when you highlight them with the Highlight icon.The Highlight Color tab allows you to change the color from the default settings to your own personal preferences. This tabs allows up to eight highlight colors of your own preferences. The selections and settings of the highlight colors, up to a maximum of eight, will be cycled through the

selected design objects when the Highlight Mode is enabled. To unhighlight an object, click the **Unhighlight All** button or rehighlight the object with the same color.

Figure 4 • **Highlight Color Tab**
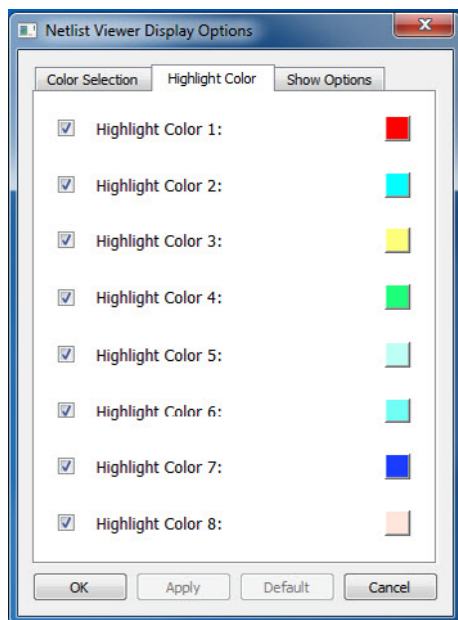
2.3.2.1 Changing the Color Setting

To change the Highlight Color from the default settings and set your own preference:

1. Check the check box to enable the color setting.
2. click the item you want to change to open the color palette dialog (Figure 7 · Color Palette)
3. Drag the cross-bar across the color spectrum and drop in onto a color of your choice.
4. Click **OK.**

2.3.3 Show Options Tab

The Show Options tab allows you to display or hide design object names:

* Instance Name
* Net Name
* Port Name
* Pin Name
* Cell Name
* Show Nets - allows you to display or hide nets
* Power Nets as Binary - display power net labels in binary instead of hex (the default).

Check the item(s) you want to display and uncheck the item(s) you want to hide.

Hiding object names and/or nets makes the display less cluttered.

Figure 5 • **Show Options Tab**
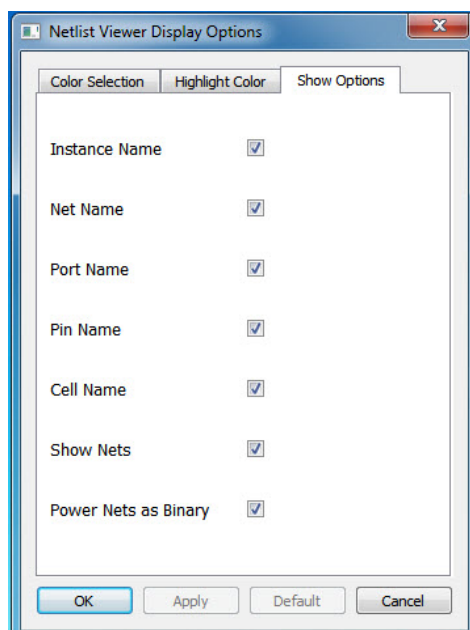
2.4 Basic Operations

Basic operations to control the display are available with Shift/Control + Mouse clicks.

2.4.1 Zoom In/Out

There are multiple ways To zoom in/out in the Netlist Viewer:

- use CTL + mouse scroll wheel.
- Left-click and drag towards upper left corner to zoom in. Left-click and drag towards upper right corner to zoom out. The drag distance determines the zoom in and zoom out magnitude. The bigger the distance of the drag, the bigger is the zoom magnitude, which is displayed as a positive integer for zoom in and a negative integer for zoom out.

Figure 6 • **Zoom in/Zoom Out Magnitude**
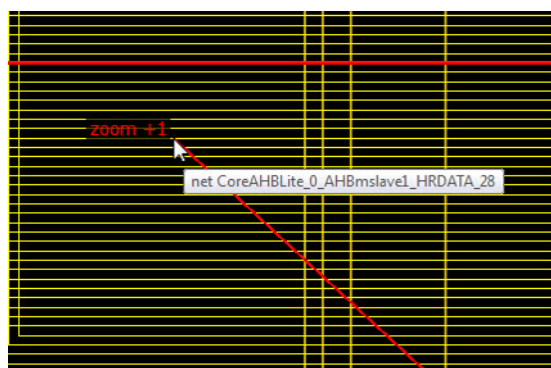
- Use the Zoom In/Zoom Out icons in the toolbar.The Zoom In/Zoom Out icons are equivalent to a left-click mouse drag magnitude of +1/-1.

2.4.2 Zoom To Fit

There are two ways to zoom the display to fit the canvas:

- Left-click and drag towards lower left to zoom in/out to fit the canvas.
- Use the Zoom To Fit icon in the toolbar

2.4.3 Zoom in to Selection Area

- Click the Zoom Icon.
- Left-click and drag towards lower right to draw a selected area to zoom into.

2.4.4 Select Design Objects

Left-click to select single item on nets/instances/modules/ports.

Use SHIFT+Left-click to select multiple items.

Note: Selection is limited to 1,000 items. Selecting a large component may go over the limit and the selection will not be honored. A warning message appears at the first occurrence of the limit being exceeded.

2.4.5 Copying Design Objects

Select the design object in the view. Use CTRL+C to copy the name of design objects (nets/instances/ports/pins/macros). The object name is copied to your clipboard. Use CTRL+V to paste the design object name from the clipboard to the Filter window of the Design Tree. This is helpful when searching for components in the design tree or when the design object name is very long with hierarchical separators in the name.

2.4.6 Move Design Objects

Use Shift + Left-click to make a selection on a design object and drag it to move it around in the display. The net connections remain intact.This is helpful when examining nets and instances in cluttered areas of the display.

2.4.7 Moving the View in the Canvas

Use CTRL + Left-click to pan the Netlist Viewer and move the view around in the Canvas. Alternatively, you can use the middle mouse button to pan the view in the canvas.

3 Navigating in the NetlistViewer

You may traverse up and down the design hierarchy (Vertical Navigation) in a hierarchical view of the NetlistViewer or navigate to a different page (Horizontal Navigation) for both Hierarchical and Flattened Netlist View. Push and Pop create a new view to display the new hierarchy level.

3.1 Vertical Navigation

Use the Push/Pop command to push into or pop out of a module/instance in the Hierarchical ADL Netlist View. The up and down traverse of the hierarchy is one hierarchy level at a time.

3.1.1 Push

To go down one level of hierarchy in the NetlistViewer:

1. Click to select the Instance.
2. Click the Push icon in the Toolbar or right-click and select Push Instance <Instance_name> or double-click a pushable instance (not at the bottom of the design hierarchy) to push into the instance.

Note: The right-mouse click push instance menu item is not available at the bottom of the design hierarchy.

Figure 1 • **Push Instance**
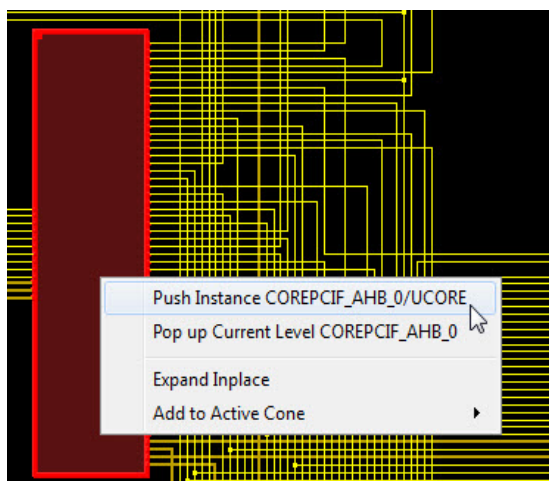
You can also select one pin of an instance, and then choose Push from the right-click menu to move the focus to that pin. For example, if you select pin N_972 of UCORE and choose Push from the right-click menu (as shown below), NetlistViewer centers on the port corresponding to the pin you selected as shown.

Figure 2 • **Push Instance by the Pin to center the Pin on Display View**
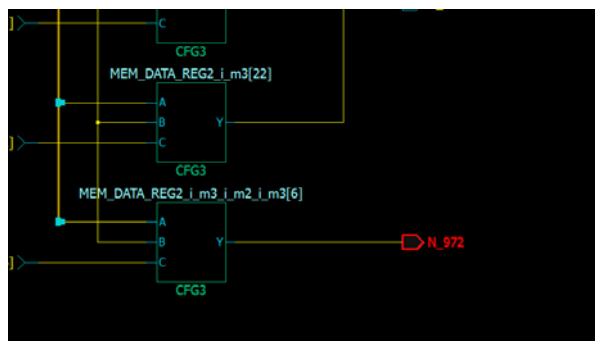
3.1.2 Pop Instance

To go up one level of hierarchy in the Netlist Viewer:

1. Click to select an instance
2. and right-click Pop Up current level <instance_name>
3. Click the Pop icon in the Toolbar or right-click and select Pop Instance <Instance_name> or Double-click an empty area in the canvas to pop up one level of hierarchy.

Note: The right-mouse click pop instance menu item is not available at the top of the design hierarchy.

Figure 3 • **Pop Up Instance**
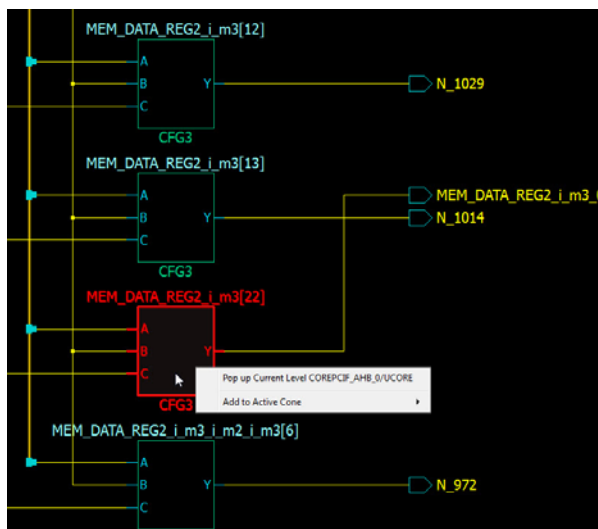
3.1.3 Dive In/Out of Design Hierarchy Without Creating New Views

Design hierarchy can be traversed without creating a new view as do push and pop. Instances that can be pushed into or popped out of can be expanded or folded. Their contents can be displayed on the current level without having to push into the module.

3.1.3.1 Expand InPlace

Use one of the following two ways to execute the **Expand Inplace** operation on an instance

- Click the "+" button on the top left of the instance, which indicates that it is pushable.

Figure 4 • **Expand in Place icon**
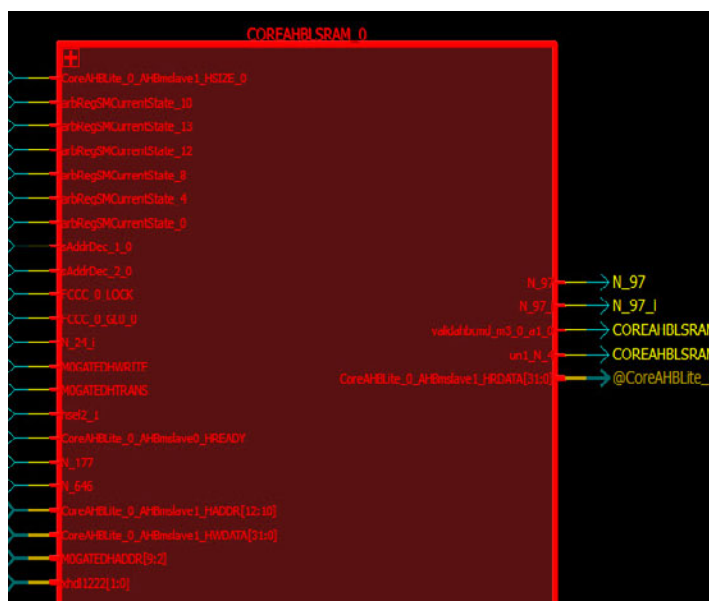
- Left-click to select the instance and then right click to select **Expand Inplace**.

Figure 5 • **Expand Inplace Menu Selection**
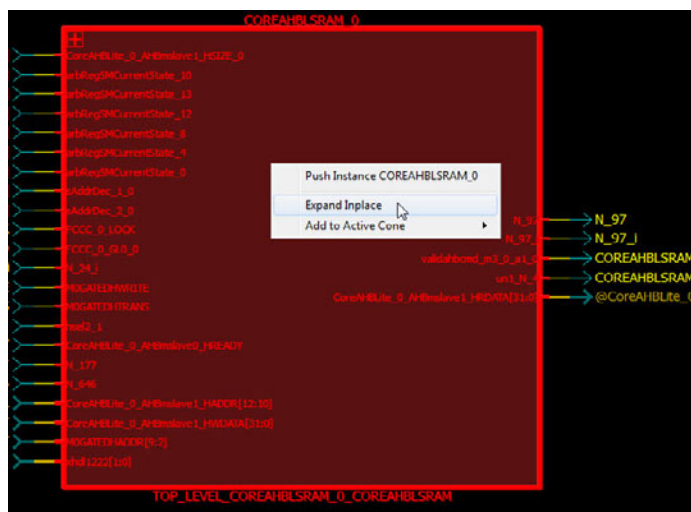
3.1.3.2 Fold Instance

The expanded hierarchy of an instance can be folded or collapsed back into the parent instance.

Use one of the following three ways to fold/collapse the expanded instance back into the parent instance:

- Click the "-" button on the top left of the instance.

Figure 6 • **Fold Instance icon**
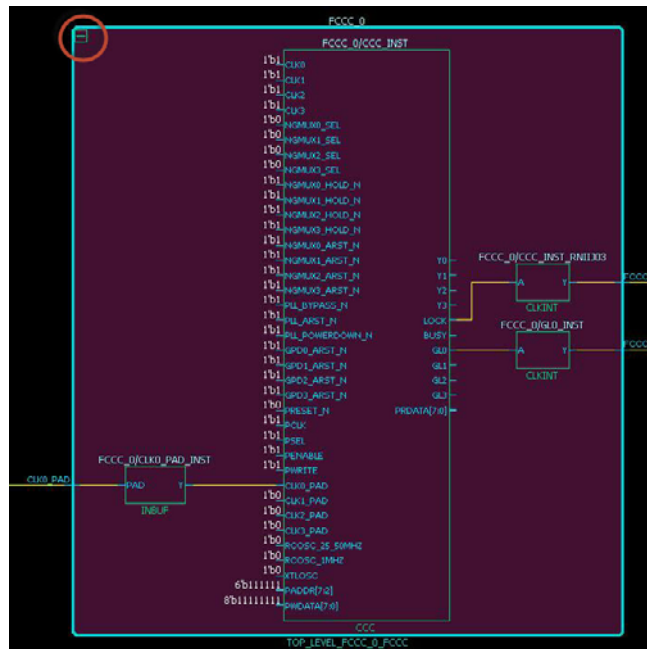
• Click to select the expanded instance, then right-click and select **Fold Instance** in the menu

Figure 7 • **Fold Instance Menu Item**
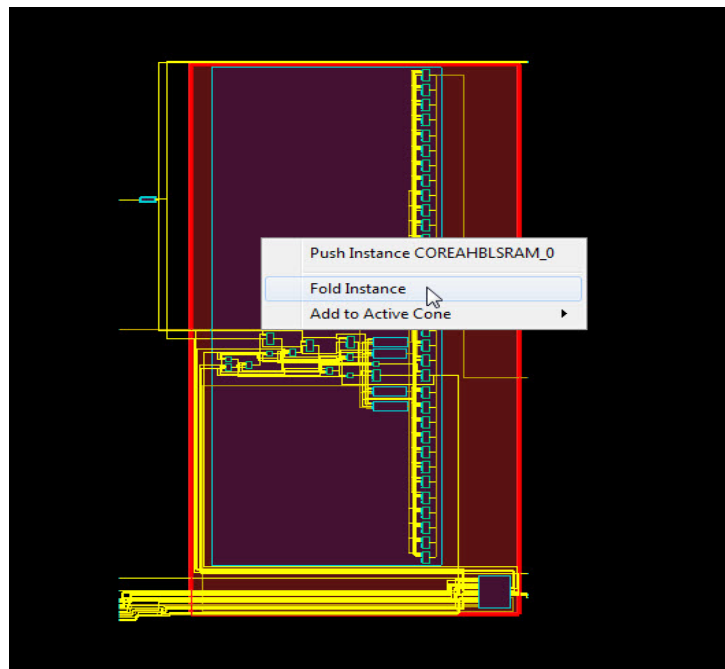
• Click the **Fold All Instance** Icon

3.1.4 Tracing Nets

NetlistViewer allows you to trace the nets to the net driver and selects it You may also follow the net to all the instances/components connected to the net.

3.1.4.1 Go to Net Driver

Right-click the net and select **Go to Net Driver** to trace the net to the driver. This feature allows you to easily retrieve the driver of a net. When the net driver is retrieved, the display is centered on the net driver.

If the **Do Selection Locally** option is turned on, the trace will stop on the current level of hierarchy. If the **Do Selection Locally** option is turned off, the trace crosses hierarchy boundaries until the instance driving the net is reached. The instance (net driver) is selected in the view.

Figure 8 • **Go to Net Driver**
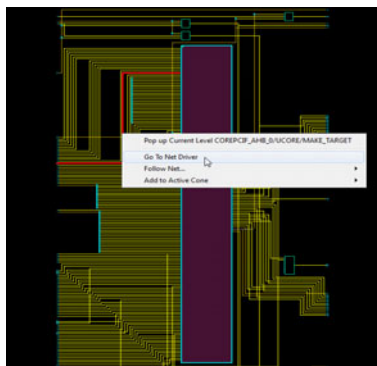
3.1.4.2 Follow Net

Following a net might take you to another page or another level of design hierarchy in your design. Following nets is useful when your design is split into multiple pages or if it includes some hierarchical logic levels.

For a net that spans across multiple pages or hierarchical boundaries, the Follow Net right-click menu option lists all the pages that the net is on. Click the page number to follow the net. If the number of pages the net appears on is over 24, the page list shows ellipses (~~~) to shorten the list. Pages that are close to the current page are not shown as ellipses. Only pages farthest away (except the first and the last page) from the current page are shown as ellipses.

Figure 9 • **Follow Net to Page List**
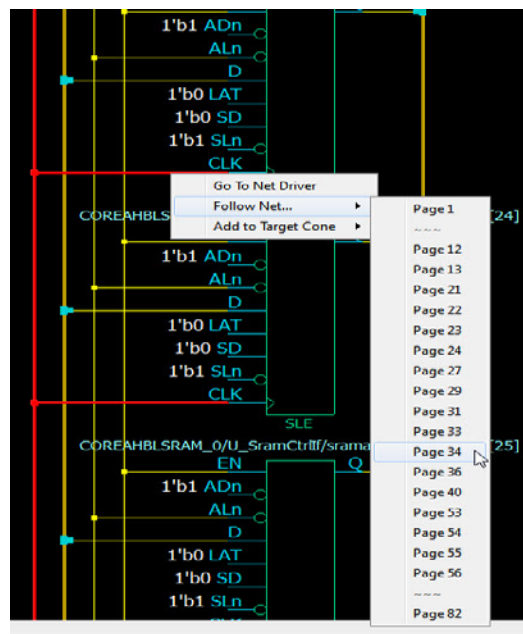
Double-clicking a net jumps to the next page the net is on.

Nets that continue on other pages are terminated by a page connector symbol (>). Note that a net can continue on many pages..

Figure 10 • **Net Terminators**

Symbol	Descriptions
	Indicates the net ends on another page
	Indicates the net begins on another page

If following a net involves crossing design hierarchical boundaries, NetlistViewer always executes push and pop operations to reach the instance the net is connected to, regardless of whether the **Do Selection Locally** option is enabled or not.

To follow the net to all the components connected to the net:

1. Right-click the net to select it.
2. Choose **Follow Net** to follow the net to all components/instances connected to the net.
3. From the sub-menu, choose one of the following:
    - Page x: The component is located on a different page. No hierarchical boundary is crossed.Only this option is available in the Flattened Netlist View.
    - Relative path Instance Name (Upper Level : Pin_name) . Need to go up one level of hierarchy
    - Relative Instance Name (Instance Name: Pin_name_) .Need to go down one level of hierarchy.

Figure 11 • **Follow Net Selections**
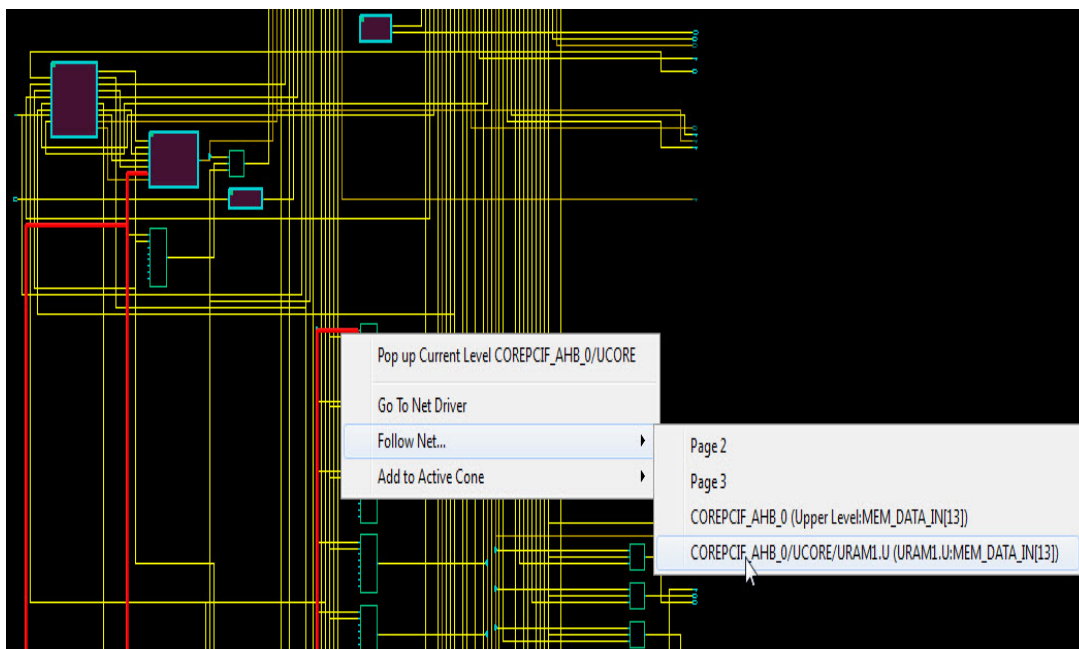
3.2 Horizontal Navigation

When large designs do not fit into one single page for display, especially in the case of flattened netlist, the NetlistViewer splits the design netlist into multiple pages. Page splitting enables you to quickly compute and display the schematic netlist view. You can turn on and turn off page splitting to view your netlist.

When the **Split Page** option is enabled, navigate the netlist horizontally in one of three ways:

- Go to Next Page - Click the Next Page icon in the toolbar to go to the next page.
- Go to Previous Page - Click the Next Page icon in the toolbar to go to the next page.
- Go to <page number> -Enter the specific page number to go to.
  Note: Go to <page number> always displays the current page number by default.
- Double-clicking a net (which spans across multiple pages) jumps to the next page the net is on.

3.3 Magnifying Pins

NetlistViewer allows you to magnify a pin and display it in a window. It the pin is connected to a bus, the bus can be expanded to display all its elements.

Double-click a pin to magnify a pin.

Figure 12 • **Magnifying Pins**
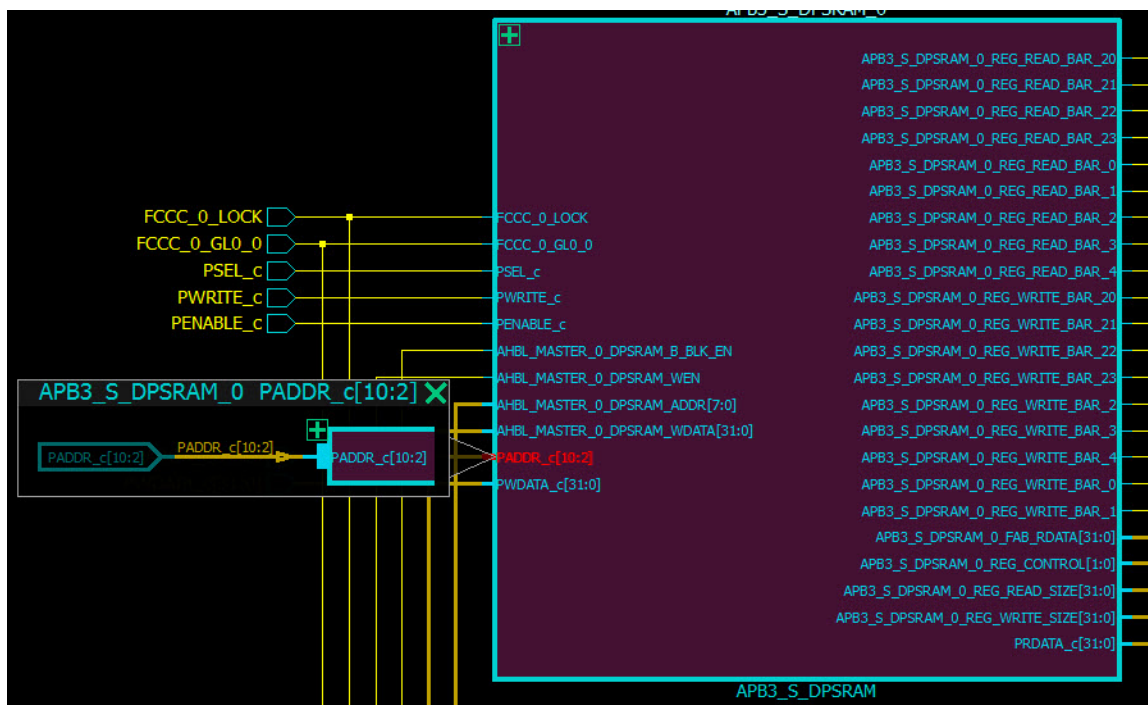
Click the "+" sign to expand the bus. When expanded, all the elements of the pin bus is displayed.

Figure 13 • **Expanded Pin Bus**
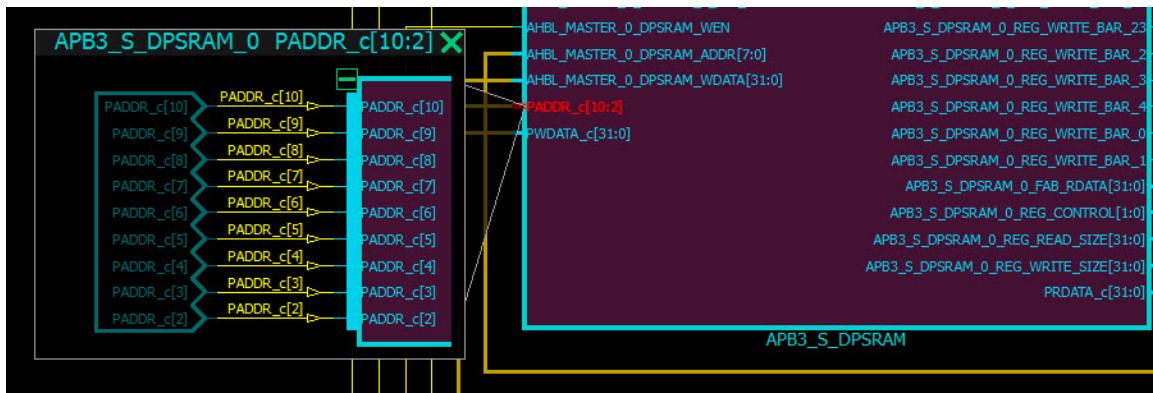
Click the "-" sign to fold/collapse the expanded pins

4 Logical Cones

A logical cone is a window that displays only a portion of the netlist. You create this window in a Netlist Viewer and simply select the objects that you want to appear in the separate view. You can add individual instances, blocks, nets, ports to a logical cone. You can also remove design objects from this cone. Logical cones are useful for debugging and path analysis, especially for timing-critical nets.

Logical cones help you navigate and analyze a specific part of the design you are interested in. A logical cone view is very similar to the Netlist Viewer view. The main differences between the Logical cone and Netlist Viewer views are:

- In a logical cone, you see only the pieces of the design you want to focus on (for example, analysis of timing-critical paths) whereas in Netlist Viewer, you see the entire netlist.
- In a logical cone, a net appears as a dashed line rather than a solid line unless all instances that are connected to that net are also present in the Logical Cone view. These dash-line nets are designated as partially connected, as opposed to fully connected nets (solid lines).
- In a logical cone, all objects of the netlist appear on a single sheet, with hierarchical boundaries still visible. This is a trade-off between the classical hierarchical view, where you must use Push and Pop commands to navigate in the netlist, and the flattened view, where hierarchy is simply ignored.

4.1 Creating a Logical Cone

Use logical cones to view, highlight, and cross-probe a selected subset of the netlist. Logical cones can be created from all three views: RTL, Hierarchical Post-Synthesis, and Flat Post-Compile.

To create a logical cone:

1. Click to select the netlist view (RTL/Hierarchical/Flat) from which to create a logical cone.
2. Click the Create Cone icon on the toolbar and add design objects to the cone or
3. Select a design object such as a net or an instance.
4. Right-mouse click and select Add to Target Cone, Driver/Load, All Connected Logic.

4.2 Naming of Logical Cones

By default, a logical cone created from the Hierarchical Netlist is named [Hier] Cone View:1, 2, 3. and so on. A logical cone created from the Flattened Netlist view is named [Flat] Cone View: 1, 2, 3... and so on. A logical cone created from the RTL view is named [RTL] Cone View:1, 2, 3... and so on. The default names can be changed.

4.3 Renaming a Logical Cone

Click the **Rename Cone** icon to rename a cone. In the Change Cone Name dialog box, enter a new name for the logical cone. Cone names must be unique. Note that only the cone name changes. The property "Target Cone" of the cone remains intact if you rename a Target Cone.

Figure 1 • **Change Cone Name Dialog**
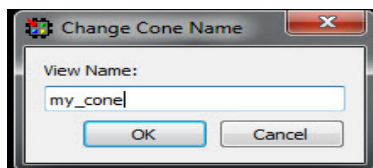
4.4 Setting the Target Cone

While multiple cone views may co-exist, only one of them is active (the Target cone) at one time. There is only one Target Cone each for the ADL Hierarchical netlist view, the AFL Flattened netlist view, and the

RTL view. The Target Cone is identified by the suffix "- Target" in the cone name. Design objects, when added to a Logical Cone, are always added to the Target Cone only. When no logical cones exist and design objects are added, Netlist Viewer creates a new logical cone and sets it as the Target Cone to display the design objects. Whenever a new logical cone is created (Create Cone icon clicked), the new cone is set as the Target Cone.

To set as Target Cone:

1. Select the cone view
2. Click the Set as Active icon in the toolbar.

4.5 Adding Selected Objects to a Cone

Design objects such as port, nets, and instances can be added to a logical cone.

- When a net is selected and added to the cone, the instance connected to the net are added and the net is a solid line in the cone.
- Nets that are automatically added to the view (because of added instances) will always be shown as dashed (regardless of whether or not they have additional connections not yet added to the view).
- Nets are solid lines when they are added through the tree, or double clicking a dashed net, or when adding a port to the view (which adds its attached net).
- Ports are not added to the view if an attached dashed net is already in the view.
- When an instance is selected and added to the cone, all the partial nets of the instance are added to the instance pins. Nets which are represented by dash lines are partial nets, which are automatically added to the cone view when an instances is added. Double-clicking a partial net in a cone view adds to the cone all the ports and instances the partial net is connected to. The dashed net changes to a solid line.

To add design objects to a logical cone:

1. Make sure the cone to which you want to add design objects is the Target Cone. Only one cone can be active at one time.
2. In Netlist Viewer or a Logical Cone view, select the instance, pin, or net to add to the logical cone.
3. From the Netlist Viewer/Logical Cone view, choose **Add to Target Cone > Selection**

Note: When an instance is added to a logical cones, all the lower levels of hierarchy, if any, of the instance are included in the logical cone. The instance in the cone can be expanded in place (by clicking the "+" sign) or folded/collapsed (by clicking the "-" sign).

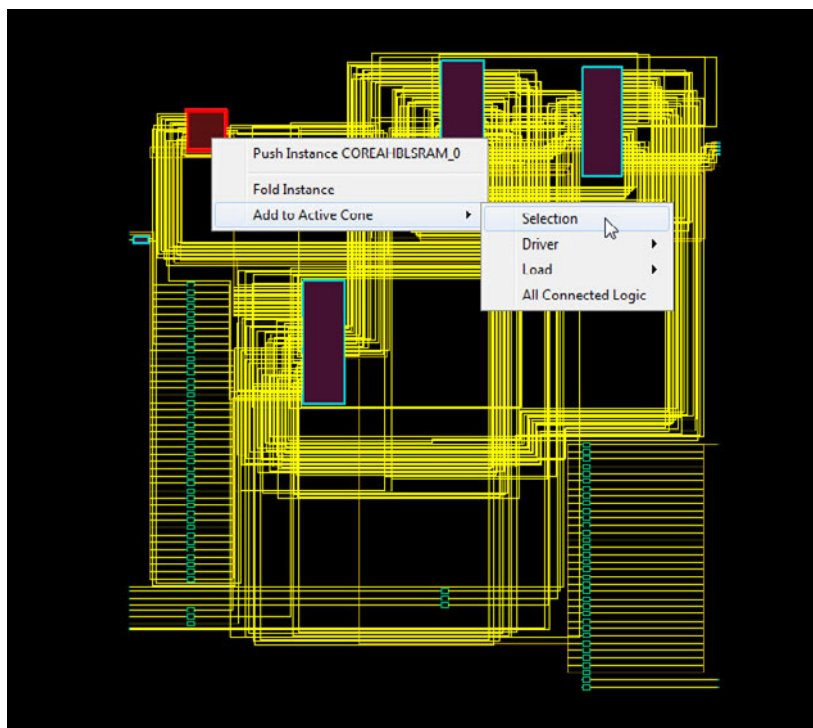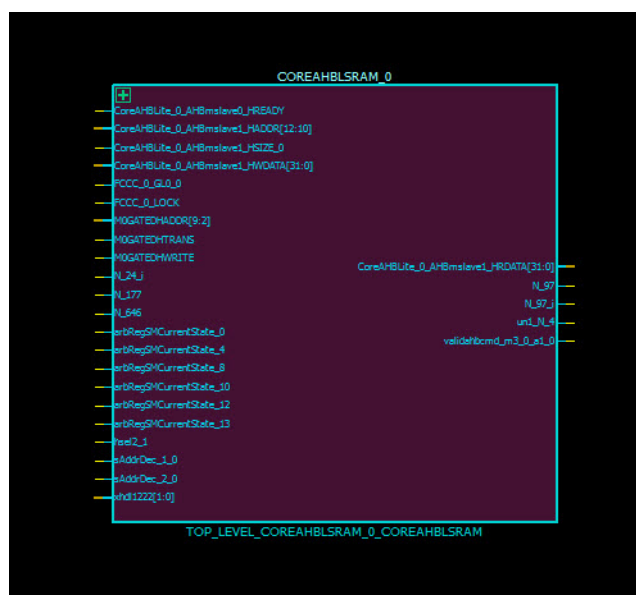*Figure 2 •* **Adding Design Objects to Active Logical Cone**

Figure 3 • **Logical Cone View**

## 4.5.1 Adding Drivers/Load into the Logical Cone

For objects such as Ports, Nets, and Instances, the drivers or loads of these design objects, when selected, can be added to the logical cone. The driver or load of design objects can be added across multiple logical levels. That means the driver of a driver (two logical level) or the load of a load (two logical levels) can be added to the logical cone. At the expense of some runtime penalty for large designs, up to a maximum of ten logical levels of a selected design object can be added to the logical cone.

- Click **Add to Target Cone > Selection** to add a selected design object to the Target Cone,'
- Click **Add to Target Cone > Driver/Load > Logical Level 1/2/3** to include in the cone the driver/load of the selected object.
  If the **Do Selection Locally** feature is enabled, Netlist Viewer stops at the hierarchical boundary when adding drivers/loads to the cone. If this feature is disabled, Netlist Viewer crosses design hierarchy boundaries to reach the specified number of logic levels of drivers/loads.
  Note: To go beyond three Logic Levels, click the ellipse (...) and specify the number of logic levels, up to a maximum of 10, in the dialog box.

Figure 4 • **Add to Target Cone Drivers/Loads up to Logical Levels**
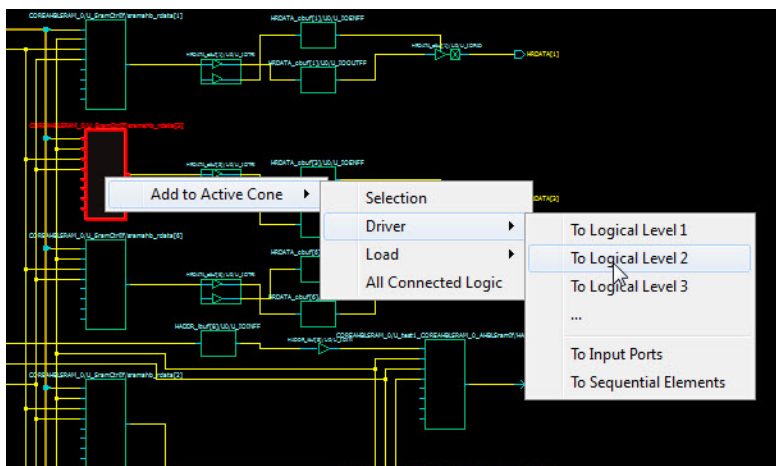
4.5.2 Adding All Connected Logic

Add to Target Cone > All connected Logic selects and adds all instances connected to the adjacent nets to the Target Cone.

4.6 Deleting a Logical Cone

Click the X button to delete a Logical Cone view.

4.7 Removing Objects from a Logical Cone

Design objects may be removed from a Logical Cone:

- Select Remove from Current Cone > All Logic to remove all logic from the Logical Cone.
- Select Remove from Current Cone > Selected Instance to remove the Selected Instance from the Cone View.

4.8 Logical Cone Persistency

A logical cone is created from any one of the three views and is associated with the view it is created from. A logical cone stay active as long as the netlist view it is created from is still open and active. Closing a netlist view also closes the logical cone view it is associated with. Opening a netlist view also brings back the cone view it is associated with. However, closing a cone view deletes the cone and it cannot be brought back into view. The cone has to be re-created.