Netlist Viewer User Interface User Guide

Libero SoC v2025.1

Introduction (Ask a Question)

As Field Programmable Gate Array (FPGA) designs grow in size and complexity, it has become essential for FPGA designers to traverse the netlist to analyze their designs. The Microchip Netlist Viewer is a graphical representation of the design netlist that displays different views for the different stages of the design process.

Table of Contents

1. Supported Families and Platforms (Ask a Question)

The Netlist Viewer supports SmartFusion® 2, IGLOO® 2, RTG4™, PolarFire®, and PolarFire SoC family devices and runs on Windows® and Linux® systems.

Important: Depending on the device selected, some user interface elements such as icons, options, tabs, and dialog boxes may vary slightly in appearance and/or content. Basic Netlist Viewer functionality remains the same, regardless of the device chosen. In this user guide, a PolarFire device is used in the example figures.

2. Views (Ask a Question)

The Netlist Viewer is a Graphical User Interface (GUI) that displays different views for the different stages of the design process:

- Register Transfer Level (RTL) Netlist view—shows how the Verilog code appears in design format. Using this view, you can confirm whether software implemented the correct logic. Cross probing between this view and the HDL code aids in troubleshooting when the design does not work as desired.
- Hierarchical Post-Synthesis view—hierarchical view of the netlist after synthesis and after technology mapping to the Microchip FPGA technology.
- Flat Post-Compile Netlist view—a flattened netlist after synthesis, technology mapping and further optimization based on the Design Rules Check (DRC) rules of the device family and/or die.
- Flat Post-Compile Cone 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 2-1. Netlist Viewer—RTL View

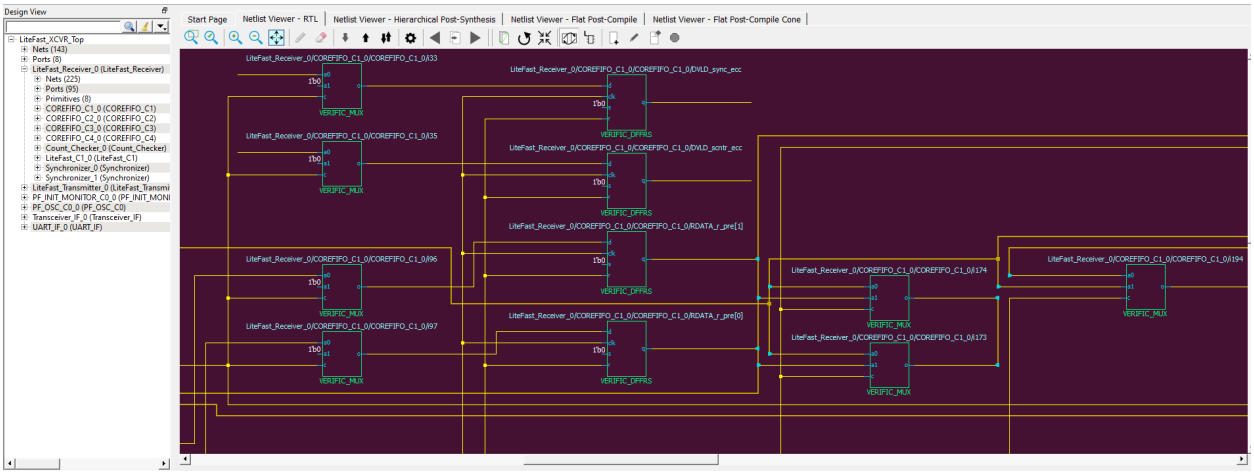


Figure 2-2. Netlist Viewer—Hierarchical Post-Synthesis View

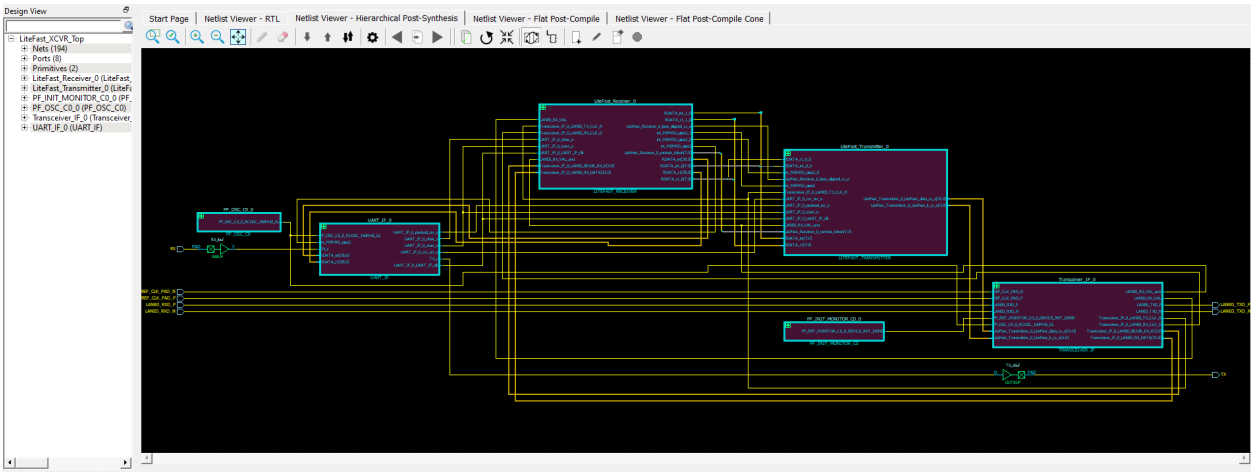
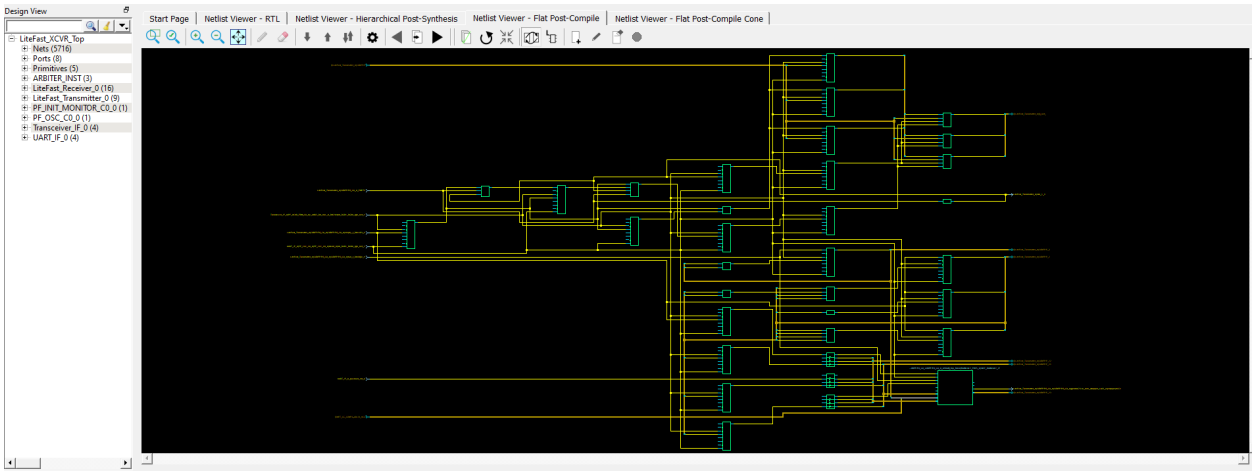
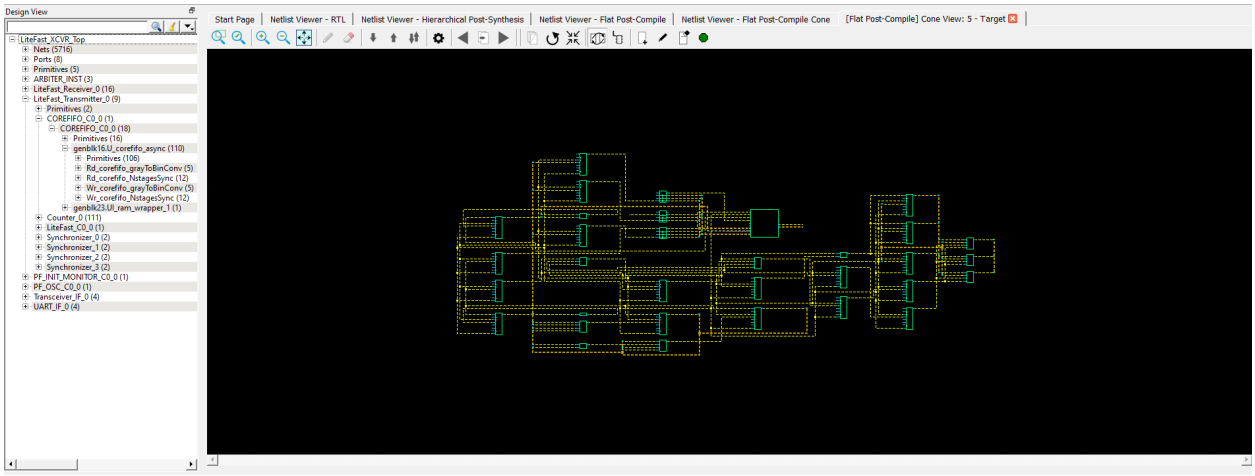


Figure 2-3. Netlist Viewer—Flat Post-Compile View



Important: A progress bar indicates that 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 2-4. Netlist Viewer—Flat Post-Compile Cone View



3. The Netlist Viewer User Interface (Ask a Question)

Netlist Viewer is a GUI designed for you 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).

3.1. Netlist Viewer Icons (Ask a Question)

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

Figure 3-1. Netlist Viewer Toolbar and Icons

The following table lists the icons used to navigate the netlist.

Table 3-1. Nelist Viewer Icons

Icon	Name	Descriptions	Notes
	Zoom Tool	When activated (ON), cursor operation is in the zoom to selection mode. 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 to select all design objects in the delineated area.	—
	Zoom In	Zoom in to 2x of original (magnitude of +1).	Alternatively, click 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, click 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.
	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 pushes into it and single clicking on black space pops 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, click this icon (ON) to highlight the selected item with a color you set in the Properties setting.	Highlights the object with a color according to your preference in the Properties setting.

Table 3-1. Nelist Viewer Icons (continued)

Icon	Name	Descriptions	Notes
	Remove Highlight	Removes all highlights from the current view.	—
	Properties	Allows you to set preferences for the highlight color, selection color, design object colors, and enable/disable display of design object names.	See Customizing Display Preferences.
	Previous Page	Goes to the previous page when the netlist view span multiple pages.	Disabled if currently on the first page.
	To Page	Displays 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 cross-probing between Design and netlist views, Chip Canvas view. and 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.
	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. For more information, see Creating a Logical Cone. 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. Use this setting for multiple large pages with very large designs. When disabled (OFF), the display splits into multiple pages. Default setting is OFF.
	Regenerate View	Places the netlist view optimally with minimal empty spaces.	—
	Fold All Instances	Collapses all expanded modules into the parent module.	—
	Rename Cone	Allows you to change the cone name.	Only the cones view has 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 becomes 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.

3.2. Right Click Mouse Menu Items (Ask a Question)

Right clicking on a design object such as an Instance, net, module, or pin opens a menu. This menu is context-sensitive and varies with the kind of design objects and the view (RTL, Flattened, or Hierarchical). Some of the menu items may contain sub-menu items.

The following table lists the menu items for all views and all design objects.

Tip: Some menu items are specific to a view or a design object.

Table 3-2. 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 the current module in place inside the current view or folds the current instance inside the current view.	Pushable Module is selected.	For more information, see Fold Instance and Expand InPlace.
Get Inst Param	Collects all instance parameters and shows them in a table on a separate window.	An Instance with parameters is selected. An example is a RAM HDL module or entity that has parameterized address bus width and data bus width in the HDL code.	—
Open File Location	Displays the netlist file in a new tab. 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 RTL and Hierarchical views of stand-alone 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. In Local Selection mode, the driver on the current level is selected. Otherwise, the lowest level driver is 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.
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 magnified view of the pin.	A pin on an instance is selected.	For more information, see Magnifying Pins.
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 3-2. 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.	—

3.3. Customizing Display Preferences (Ask a Question)

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.

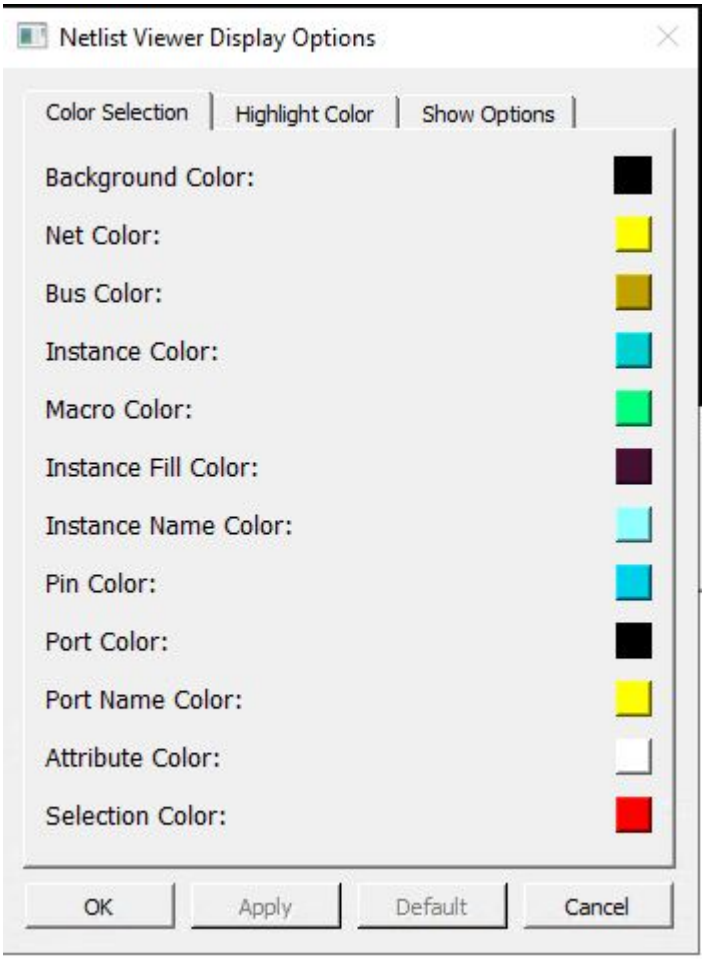
The Netlist Viewer Display Options dialog box opens when the Properties Icon is clicked. Three tabs are available to configure your personal preferences.

3.3.1. Color Selection (Ask a Question)

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—Color of the I/O attributes. For example, 1'b1 for a port tied to V_{CC} or 1'b0 for a port tied to ground. The attribute color applies to the attribute label "1'b1" or "1b0".
- Selection Color—Color that shows when design objects such as instances, modules, nets, and pins are clicked and selected.

Figure 3-2. Color Selection Tab

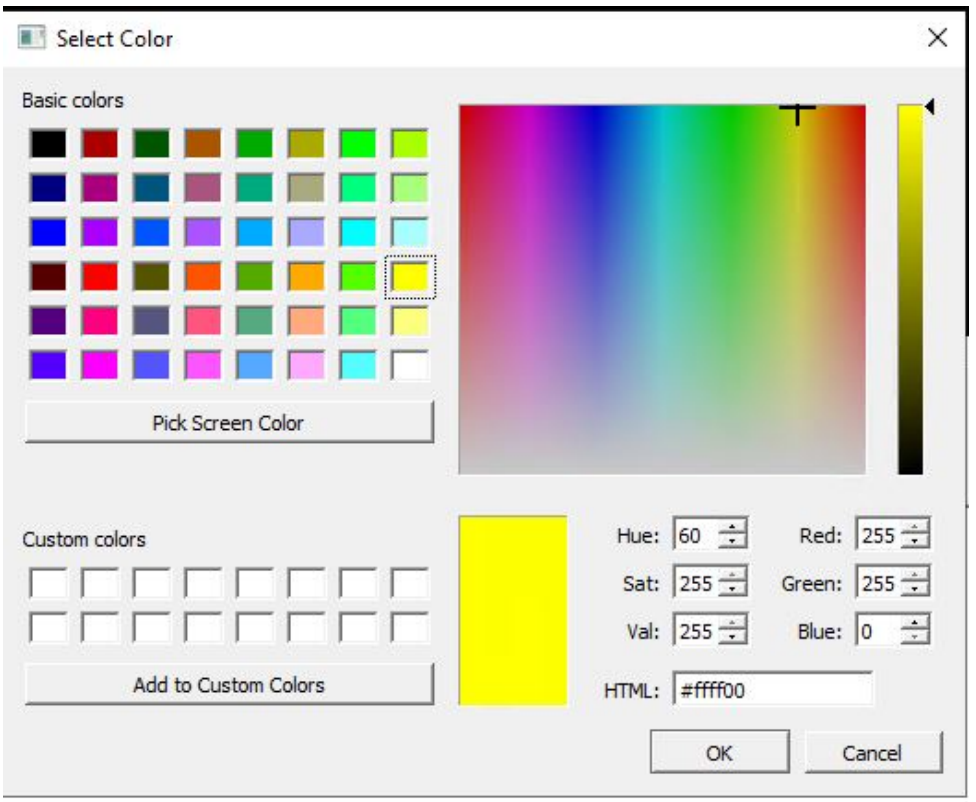


3.3.1.1. Changing the Color Setting (Ask a Question)

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.
2. In the color palette dialog box, drag the cross-bar across the color spectrum and select the desired color.
3. Click **OK**.

Figure 3-3. Color Palette

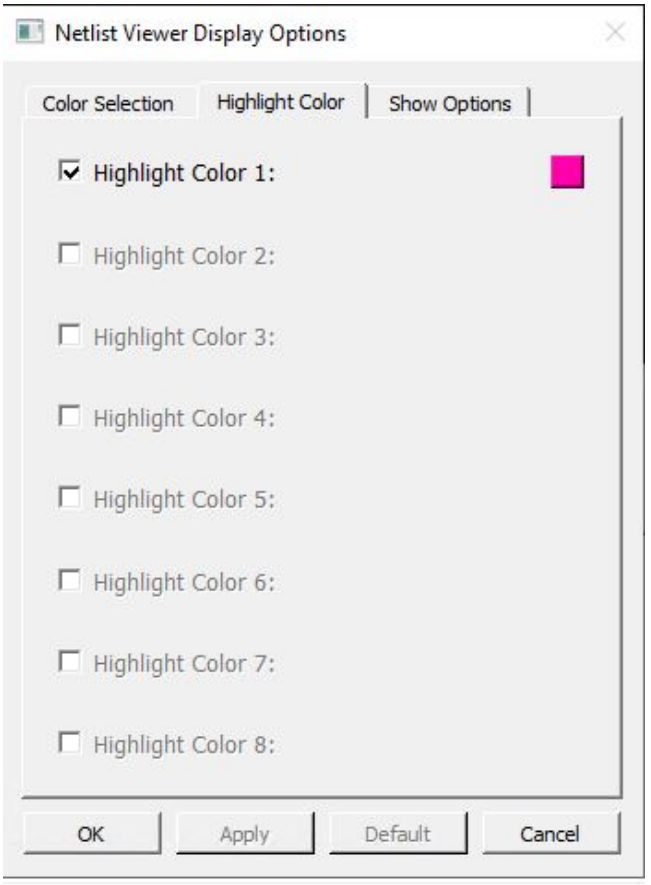


3.3.2. Highlight Color (Ask a Question)

The Highlight Color is the color of the design objects (nets, instances, and 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, is cycled through the selected design objects when the Highlight Mode is enabled. To unhighlight an object, click the **Unhighlight All** button or re-highlight the object with the same color.

Figure 3-4. Highlight Color Tab



3.3.2.1. Changing the Color Setting (Ask a Question)

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

1. Select the check box to enable the color setting.
2. Click the item you want to change to open the color palette dialog.
3. Drag the cross-bar across the color spectrum and drop in onto a color of your choice.
4. Click **OK**.

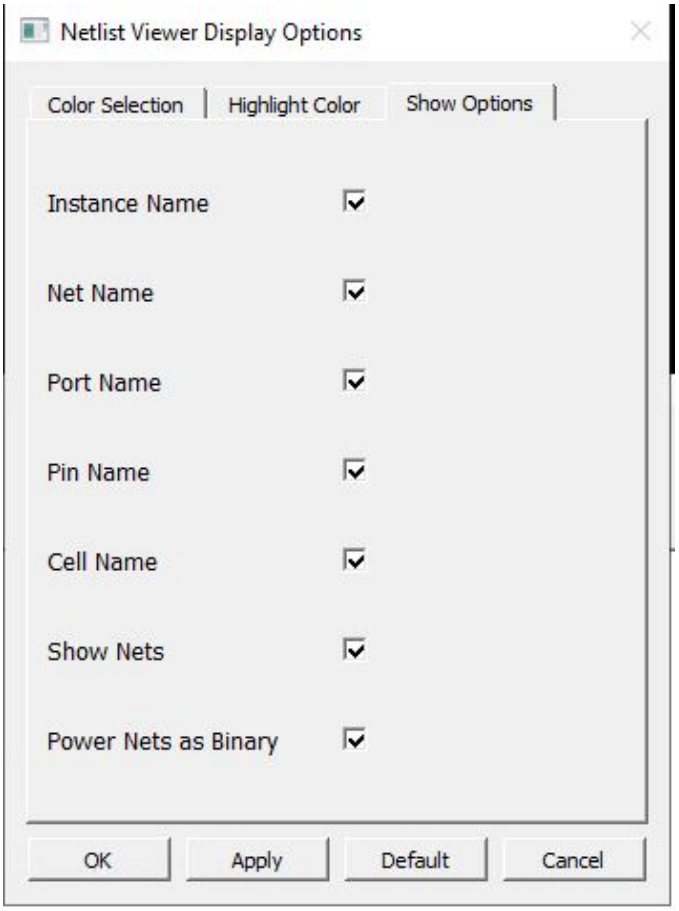
3.3.3. Show Options Tab (Ask a Question)

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—displays power net labels in binary instead of hex (the default)

Select one or more items you want to display and clear the items you want to hide. Hiding object names and/or nets makes the display less cluttered.

Figure 3-5. Show Options Tab



3.4. Basic Operations (Ask a Question)

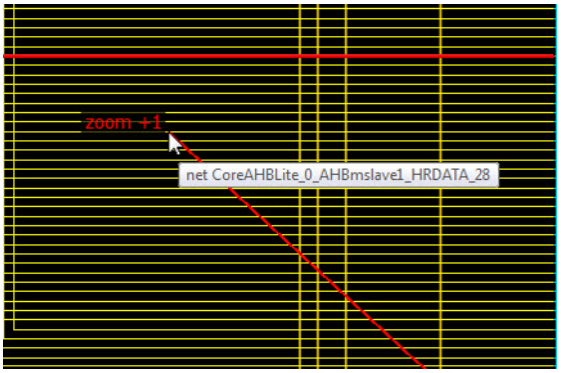
Basic operations to control the display are available with Shift/Control + Mouse clicks.

3.4.1. Zoom In/Out (Ask a Question)

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

- Use Ctrl+ 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 3-6. 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.

3.4.2. Zoom To Fit (Ask a Question)

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

3.4.3. Zoom In to Selection Area (Ask a Question)

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

3.4.4. Select Design Objects (Ask a Question)

Left click to select single item on nets, instances, modules, and ports. Use SHIFT+Left click to select multiple items.

Important: Selection is limited to 1,000 items. Selecting a large component may exceed the limit, in which case the selection will not be accepted. A warning message appears at the first occurrence of the limit being exceeded.

3.4.5. Copying Design Objects (Ask a Question)

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.

3.4.6. Move Design Objects (Ask a Question)

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.

3.4.7. Moving the View in the Canvas (Ask a Question)

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.

4. Navigating in the Netlist Viewer (Ask a Question)

You may move vertically (up and down) the design hierarchy in a hierarchical view of the Netlist Viewer or move horizontally to a different page for Hierarchical and Flattened Netlist Views. Push and Pop create a new view for displaying the new hierarchy level.

4.1. Vertical Navigation (Ask a Question)

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

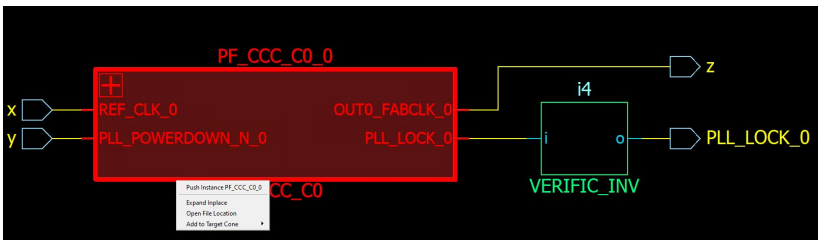
4.1.1. Push (Ask a Question)

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

1. Click to select the instance.
2. Click the **Push** icon on 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.

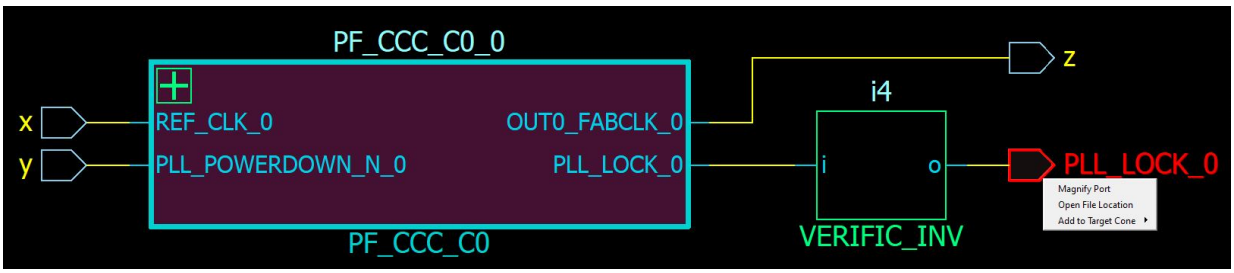
Important: The right-mouse click push instance menu item is not available at the bottom of the design hierarchy.

Figure 4-1. 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 in the following figure, Netlist Viewer centers on the port corresponding to the pin you selected as shown.

Figure 4-2. Push Instance by the Pin to Center the Pin on Display View



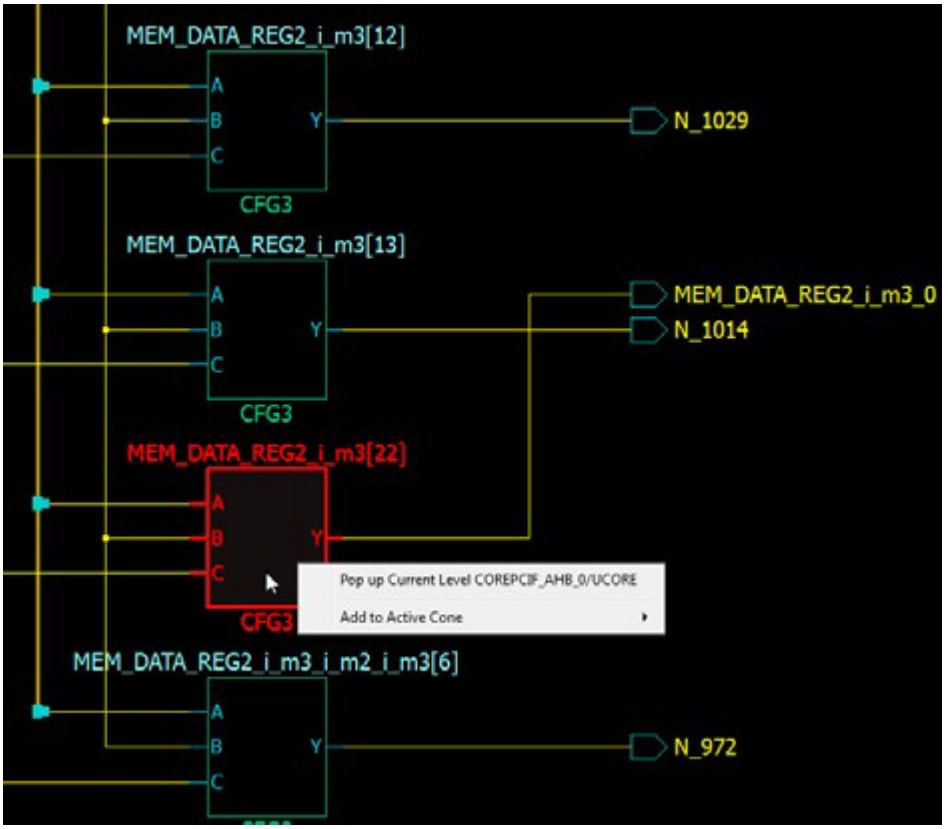
4.1.2. Pop Instance (Ask a Question)

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

1. Click to select an instance and right click **Pop Up current level <instance_name>**.
2. Click the Pop icon on the Toolbar or right click and select **Pop Instance <instance_name>** or double click an empty area in the canvas to move up one level of hierarchy.

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

Figure 4-3. Pop Up Instance



4.1.3. Dive In/Out of Design Hierarchy Without Creating New Views (Ask a Question)

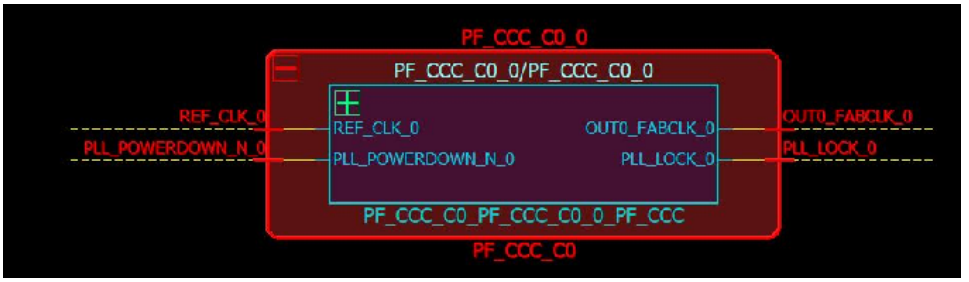
Design hierarchy can be traversed without creating a new view as do push and pop. Instances that can be pushed into or popped out of can be expanded or folded. Their contents can be displayed on the current level without having to push into the module.

4.1.3.1. Expand InPlace (Ask a Question)

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

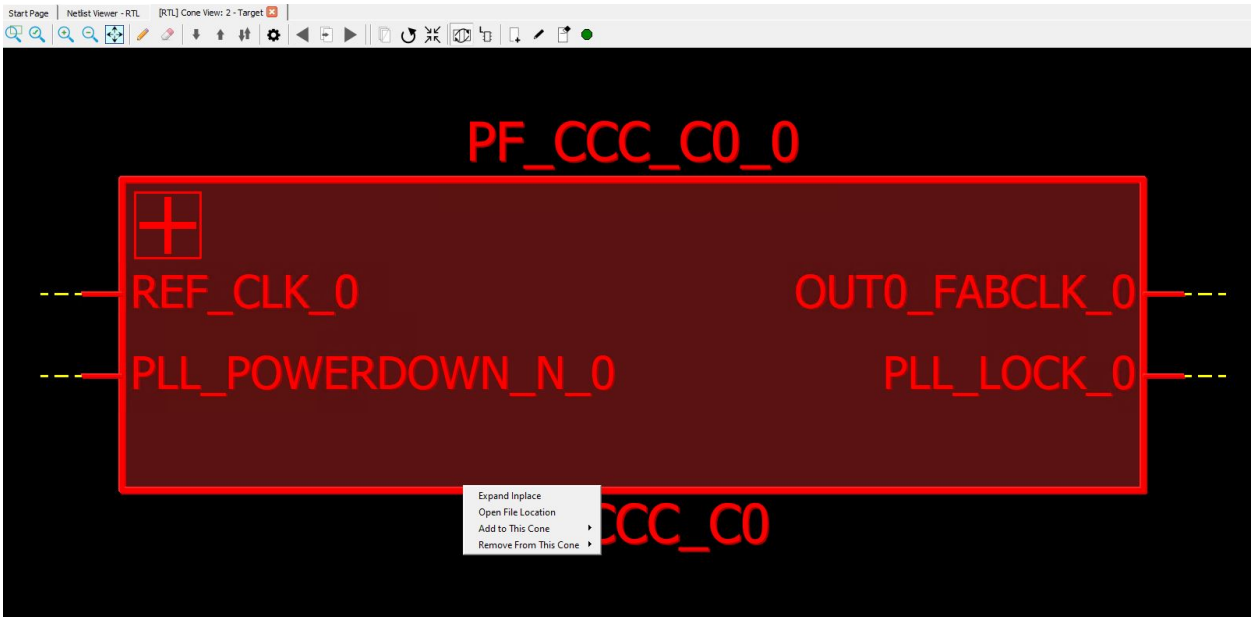
- Click the **+** button at the top left of the instance, which indicates that it is pushable.

Figure 4-4. Expand in Place icon



- Left click to select the instance and then right click to select **Expand Inplace**.

Figure 4-5. Expand Inplace Menu Selection



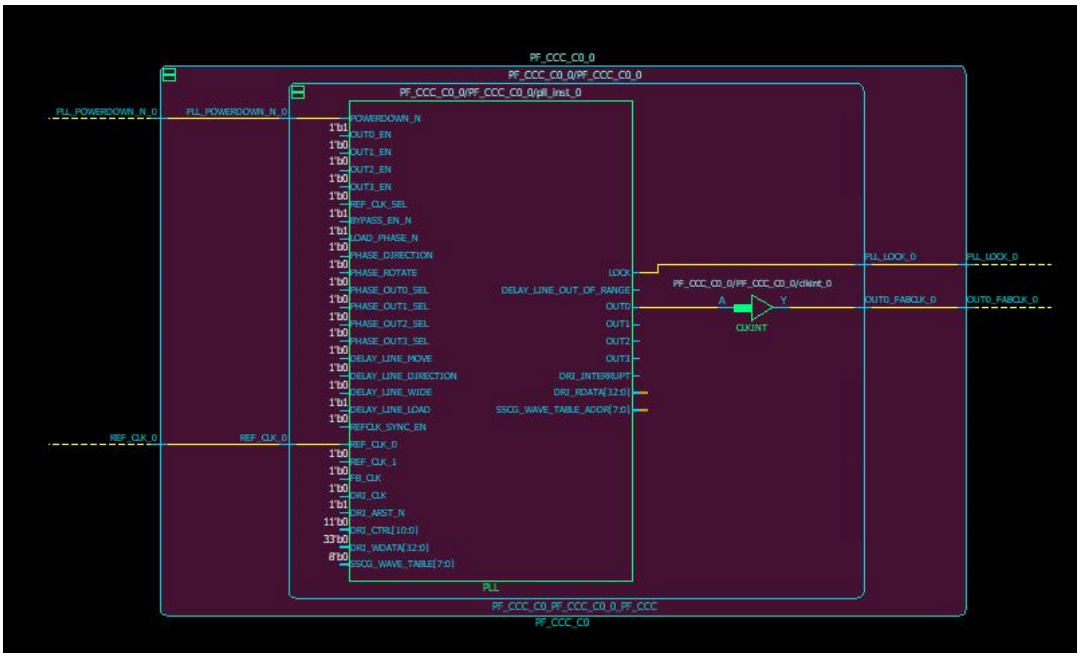
4.1.3.2. Fold Instance (Ask a Question)

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:

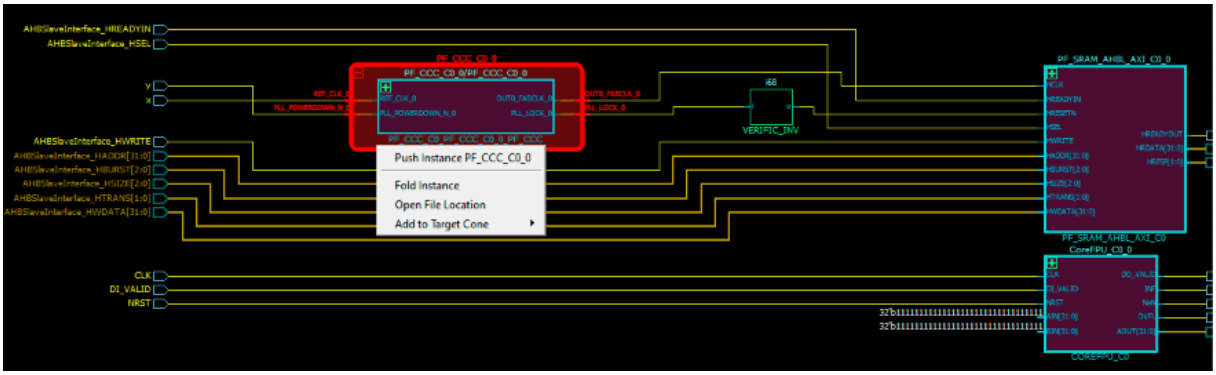
- Click the **-** button at the top left of the instance.

  Figure 4-6. Fold Instance Icon



- Click to select the expanded instance, then right click and select **Fold Instance** in the menu.

Figure 4-7. Fold Instance Menu Item



- Click the **Fold All Instance** icon.

4.1.4. Tracing Nets (Ask a Question)

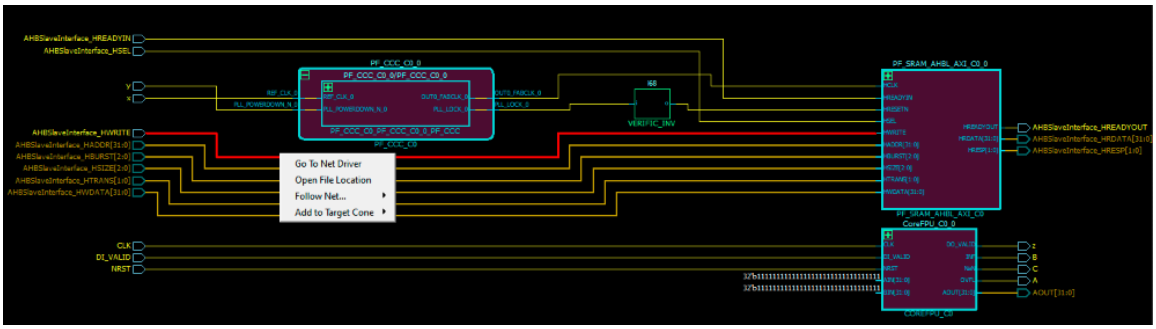
Netlist Viewer allows you to trace the nets to the net driver and selects it. You can also follow the net to all the instances and components connected to the net.

4.1.4.1. Go to Net Driver (Ask a Question)

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 stops 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 4-8. Go to Net Driver

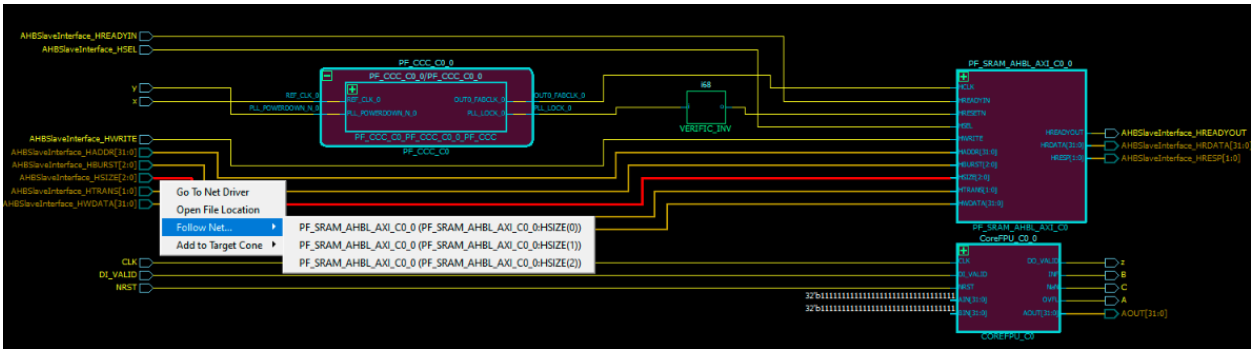


4.1.4.2. Follow Net (Ask a Question)

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 where the net is ON. Click the page number to follow the net. If the number of pages on which the net appears exceeds 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.

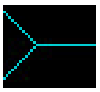
Figure 4-9. Follow Net to Page List



Double clicking a net jumps to the next page where 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.

Table 4-1. Net Terminators

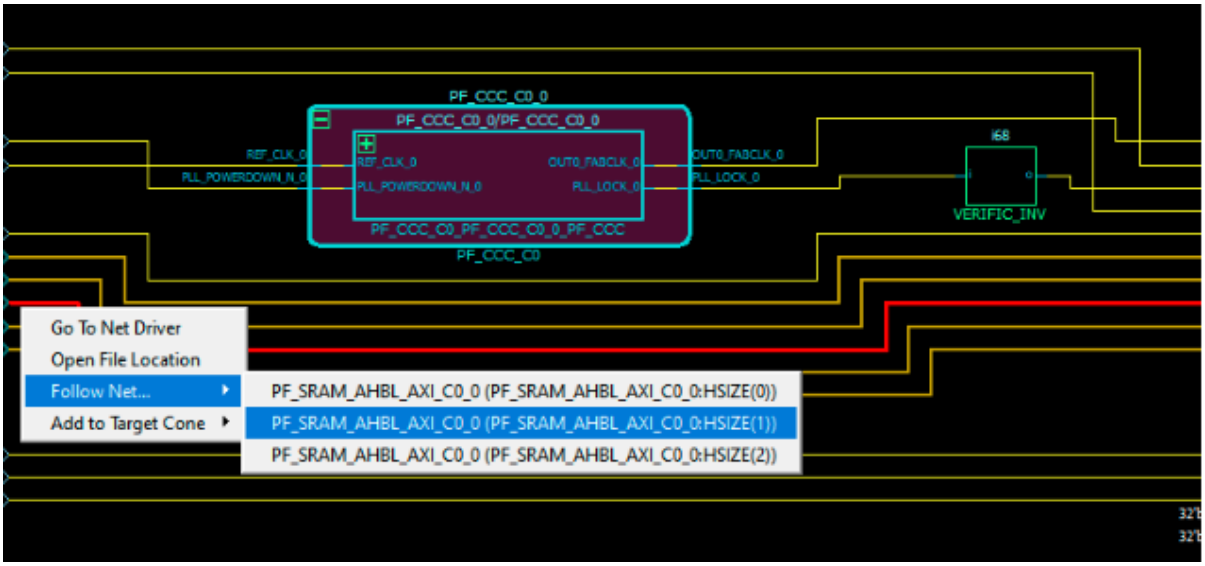
Symbol	Descriptions
	Indicates the net ends on another page.
	Indicates the net begins on another page.

If following a net involves crossing design hierarchical boundaries, Netlist Viewer 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 and 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 4-10. Follow Net Selections



4.2. Horizontal Navigation (Ask a Question)

When large designs do not fit into one single page for display, especially in the case of flattened netlist, the Netlist Viewer 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 on the toolbar to go to the next page.
- Go to Previous Page—Click the Previous Page icon on the toolbar to go to the previous page.
- Go to <page number>—Enter the specific page number to go to.

> **Tip: Go to <page number>** always displays the current page number by default.

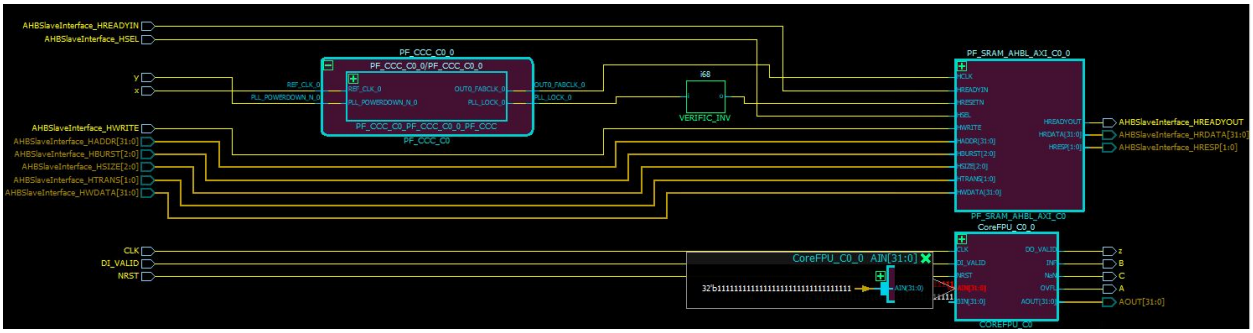
- Double clicking a net that spans across multiple pages jumps to the next page where the net is on.

4.3. Magnifying Pins (Ask a Question)

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

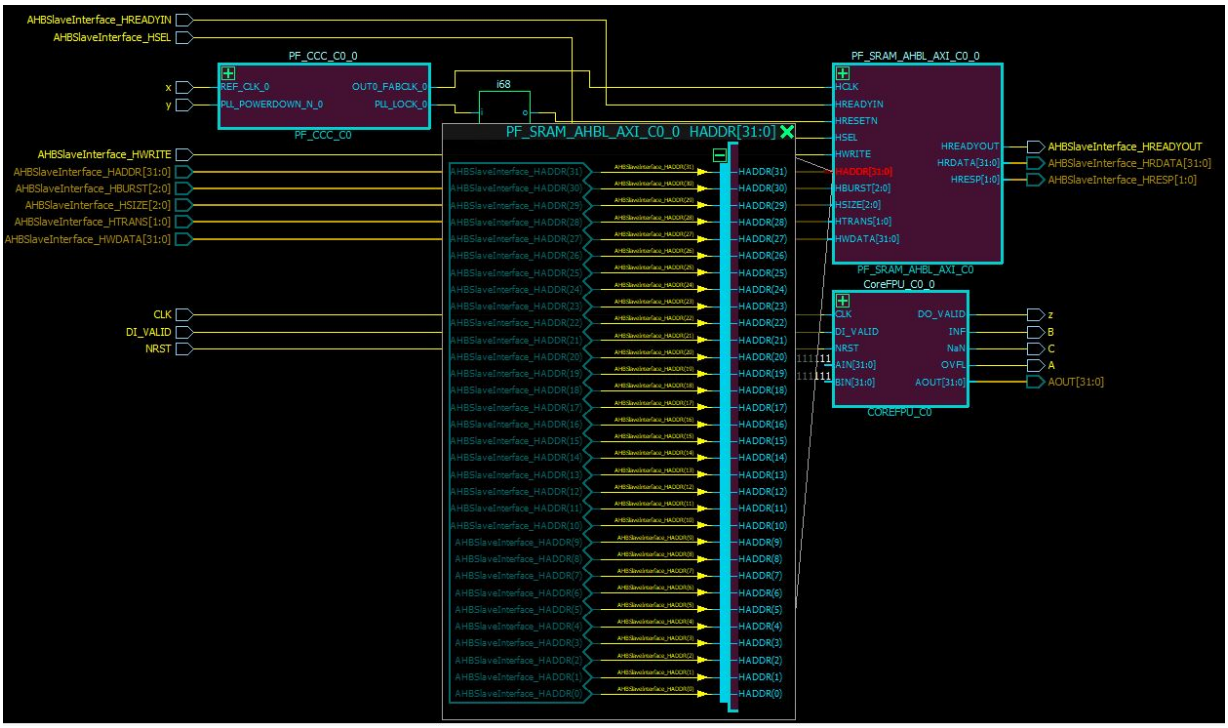
Double click a pin to magnify a pin.

Figure 4-11. Magnifying Pins



Click the **+** sign to expand the bus. When expanded, all the elements of the pin bus is displayed.

Figure 4-12. Expanded Pin Bus



Click the **-** sign to fold or collapse the expanded pins.

5. Logical Cones (Ask a Question)

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, and 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 in which you are interested. 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.

5.1. Creating a Logical Cone (Ask a Question)

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 select a design object such as a net or an instance.
3. Right click and select **Add to Target Cone**, **Driver/Load**, **All Connected Logic**.

5.2. Naming of Logical Cones (Ask a Question)

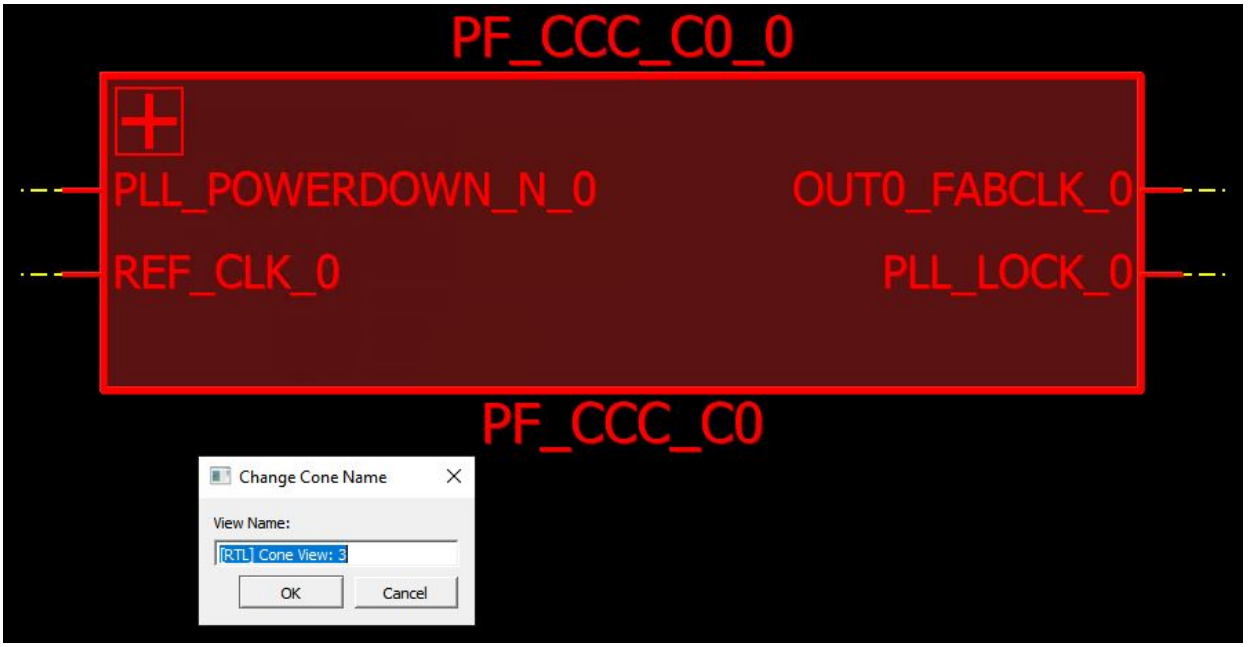
By default, a logical cone created from the Hierarchical Netlist is named [Hier] Cone View:1, 2, 3. and so on. A logical cone created from the Flattened Netlist view is named [Flat] Cone View: 1, 2, 3... and so on. A logical cone created from the RTL view is named [RTL] Cone View:1, 2, 3... and so on.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

5.3. Renaming a Logical Cone (Ask a Question)

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.

Tip: Only the cone name changes. The property "Target Cone" of the cone remains intact if you rename a Target Cone.

Figure 5-1. Change Cone Name Dialog Box



5.4. Setting the Target Cone (Ask a Question)

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

5.5. Adding Selected Objects to a Cone (Ask a Question)

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

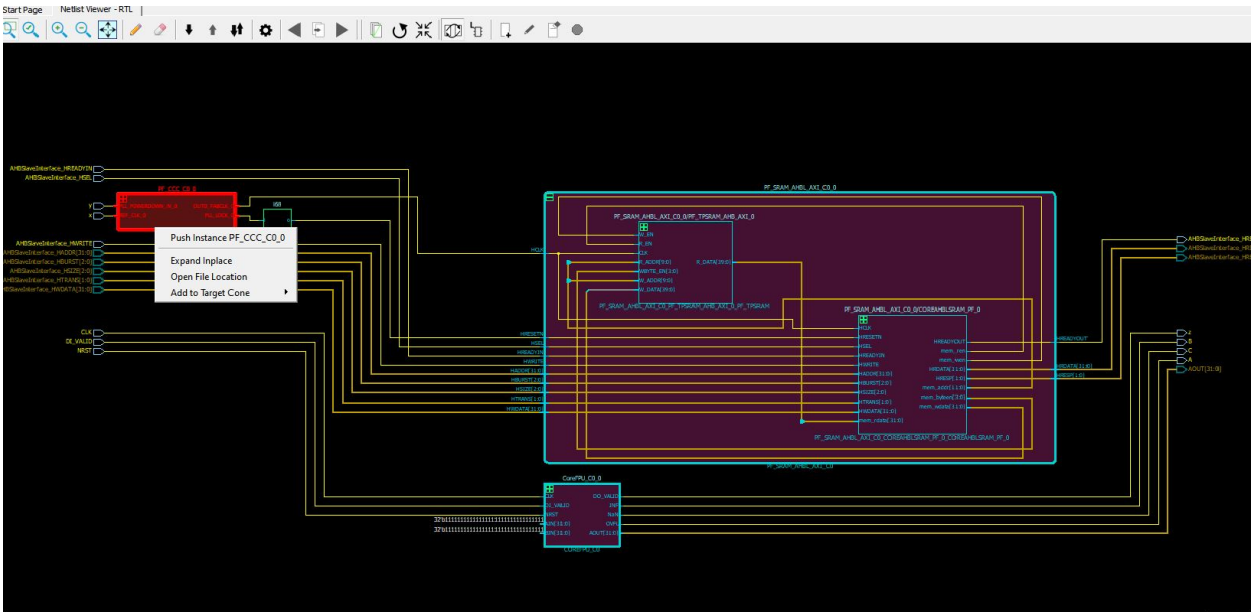
- When a net is selected and added to the cone, the instance connected to the net is added, and the net is a solid line in the cone.
- Nets that are automatically added to the view (because of added instances) is always shown as dashed (regardless of whether or not they have additional connections not 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 a 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**.

Important: 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 5-2. Adding Design Objects to Active Logical Cone



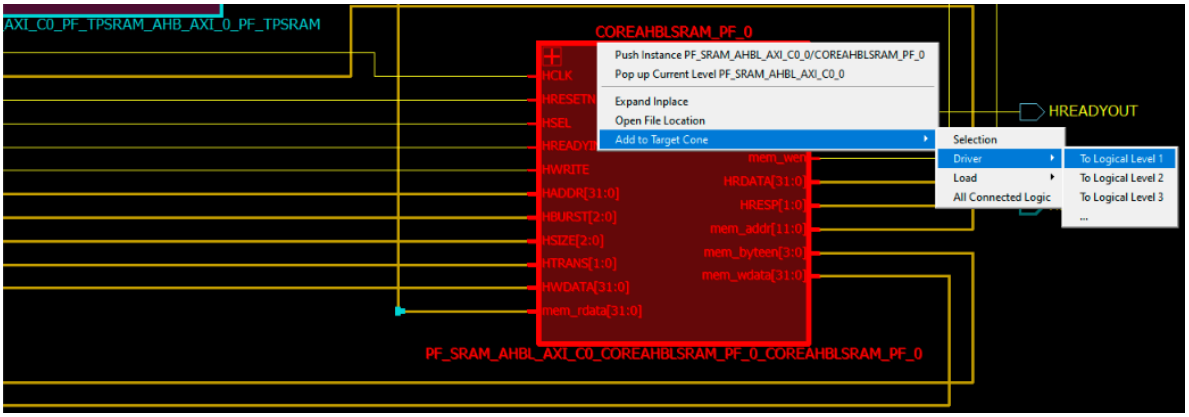
5.5.1. Adding Drivers and Loads into the Logical Cone (Ask a Question)

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.

Important: 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 5-3. Add to Target Cone Drivers/Loads up to Logical Levels



5.5.2. Adding All Connected Logic (Ask a Question)

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

5.6. Deleting a Logical Cone (Ask a Question)

Click the **X** button to delete a Logical Cone view.

5.7. Removing Objects from a Logical Cone (Ask a Question)

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.

5.8. Logical Cone Persistency (Ask a Question)

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.

6. Revision History (Ask a Question)

The revision history describes the changes that were implemented in the document. The changes are listed by revision, starting with the most current publication.

Revision	Date	Description
P	05/2025	This document is released with Libero SoC Design Suite v2025.1 without changes from v2024.2.
N	08/2024	This document is released with Libero SoC Design Suite v2024.2 without changes from v2024.1.
M	02/2024	This document is released with Libero SoC Design Suite v2024.1 without changes from v2023.2.
L	08/2023	Editorial updates only. No technical content updates.
K	08/2023	Editorial updates only. No technical content updates.
J	05/2023	Updated the document with the latest and better quality graphics.
H	04/2023	This document is released with Libero SoC Design Suite v2023.1 without changes from v2022.3.
G	12/2022	This document is released with Libero SoC Design Suite v2022.3 without changes from v2022.2.
F	08/2022	This document is released with Libero SoC Design Suite v2022.2 without changes from v2022.1.
E	04/2022	This document is released with Libero SoC Design Suite v2022.1 without changes from v2021.3.
D	12/2021	• In section Supported Families and Platforms, added PolarFire SoC to the list of supported devices. • Updated the document with better quality graphics.
C	08/2021	This document is released with Libero SoC Design Suite v2021.2 without changes from v2021.1.
B	04/2021	Editorial updates only. No technical content updates.
A	11/2020	Document converted to Microchip template.
4.0	12/2018	Document template updates and minor text edits
3.0	10/2017	Added Flat Post-Compile Cone View
2.0	05/2017	Minor updates
1.0	12/2016	Initial Revision

Microchip FPGA Support

Microchip FPGA products group backs its products with various support services, including Customer Service, Customer Technical Support Center, a website, and worldwide sales offices. Customers are suggested to visit Microchip online resources prior to contacting support as it is very likely that their queries have been already answered.

Contact Technical Support Center through the website at www.microchip.com/support. Mention the FPGA Device Part number, select appropriate case category, and upload design files while creating a technical support case.

Contact Customer Service for non-technical product support, such as product pricing, product upgrades, update information, order status, and authorization.

- From North America, call **800.262.1060**
- From the rest of the world, call **650.318.4460**
- Fax, from anywhere in the world, **650.318.8044**

Microchip Information

Trademarks

The “Microchip” name and logo, the “M” logo, and other names, logos, and brands are registered and unregistered trademarks of Microchip Technology Incorporated or its affiliates and/or subsidiaries in the United States and/or other countries (“Microchip Trademarks”). Information regarding Microchip Trademarks can be found at https://www.microchip.com/en-us/about/legal-information/microchip-trademarks.

ISBN: 979-8-3371-1097-4

Legal Notice

This publication and the information herein may be used only with Microchip products, including to design, test, and integrate Microchip products with your application. Use of this information in any other manner violates these terms. Information regarding device applications is provided only for your convenience and may be superseded by updates. It is your responsibility to ensure that your application meets with your specifications. Contact your local Microchip sales office for additional support or, obtain additional support at www.microchip.com/en-us/support/design-help/client-support-services.

THIS INFORMATION IS PROVIDED BY MICROCHIP “AS IS”. MICROCHIP MAKES NO REPRESENTATIONS OR WARRANTIES OF ANY KIND WHETHER EXPRESS OR IMPLIED, WRITTEN OR ORAL, STATUTORY OR OTHERWISE, RELATED TO THE INFORMATION INCLUDING BUT NOT LIMITED TO ANY IMPLIED WARRANTIES OF NON-INFRINGEMENT, MERCHANTABILITY, AND FITNESS FOR A PARTICULAR PURPOSE, OR WARRANTIES RELATED TO ITS CONDITION, QUALITY, OR PERFORMANCE.

IN NO EVENT WILL MICROCHIP BE LIABLE FOR ANY INDIRECT, SPECIAL, PUNITIVE, INCIDENTAL, OR CONSEQUENTIAL LOSS, DAMAGE, COST, OR EXPENSE OF ANY KIND WHATSOEVER RELATED TO THE INFORMATION OR ITS USE, HOWEVER CAUSED, EVEN IF MICROCHIP HAS BEEN ADVISED OF THE POSSIBILITY OR THE DAMAGES ARE FORESEEABLE. TO THE FULLEST EXTENT ALLOWED BY LAW, MICROCHIP’S TOTAL LIABILITY ON ALL CLAIMS IN ANY WAY RELATED TO THE INFORMATION OR ITS USE WILL NOT EXCEED THE AMOUNT OF FEES, IF ANY, THAT YOU HAVE PAID DIRECTLY TO MICROCHIP FOR THE INFORMATION.

Use of Microchip devices in life support and/or safety applications is entirely at the buyer’s risk, and the buyer agrees to defend, indemnify and hold harmless Microchip from any and all damages, claims, suits, or expenses resulting from such use. No licenses are conveyed, implicitly or otherwise, under any Microchip intellectual property rights unless otherwise stated.

Microchip Devices Code Protection Feature

Note the following details of the code protection feature on Microchip products:

- Microchip products meet the specifications contained in their particular Microchip Data Sheet.
- Microchip believes that its family of products is secure when used in the intended manner, within operating specifications, and under normal conditions.
- Microchip values and aggressively protects its intellectual property rights. Attempts to breach the code protection features of Microchip products are strictly prohibited and may violate the Digital Millennium Copyright Act.
- Neither Microchip nor any other semiconductor manufacturer can guarantee the security of its code. Code protection does not mean that we are guaranteeing the product is “unbreakable”. Code protection is constantly evolving. Microchip is committed to continuously improving the code protection features of our products.