Gowin HDL Schematic Viewer

User Guide

SUG755-1.2.2E, 11/30/2023

Copyright © 2023 Guangdong Gowin Semiconductor Corporation. All Rights Reserved.

GOWIN is a trademark of Guangdong Gowin Semiconductor Corporation and is registered in China, the U.S. Patent and Trademark Office, and other countries. All other words and logos identified as trademarks or service marks are the property of their respective holders. No part of this document may be reproduced or transmitted in any form or by any denotes, electronic, mechanical, photocopying, recording or otherwise, without the prior written consent of GOWINSEMI.

Disclaimer

GOWINSEMI assumes no liability and provides no warranty (either expressed or implied) and is not responsible for any damage incurred to your hardware, software, data, or property resulting from usage of the materials or intellectual property except as outlined in the GOWINSEMI Terms and Conditions of Sale. GOWINSEMI may make changes to this document at any time without prior notice. Anyone relying on this documentation should contact GOWINSEMI for the current documentation and errata.

.

Revision History

Date	Version	Description
11/17/2020	1.0E	Initial version published.
06/17/2021	1.1E	Primitive INIT supported.
11/05/2021	1.2E	The descriptions of Chapter 2 Introduction modified.
12/13/2022	1.2.1E	Schematic Viewer supports RTL Design Viewer and Post-Synthesis Netlist Viewer.
11/30/2023	1.2.2E	The right-click menu of the schematic view supports "Save as PDF File" option.

Contents

Contents i
List of Figures ii
List of Tables iv
1 About This Guide 1
1.1 Purpose 1
1.2 Related Documents 1
1.3 Terminology and Abbreviations 1
1.4 Support and Feedback 1
2 Introduction 2
3 Start Schematic Viewer 3
4 GUI 4
4.1 Tool Bar 4
4.2 Hierarchy Pane 5
4.2.1 Hierarchy Structure 5
4.2.2 Introduction to Objects 5
4.3 Schematic Pane 15
4.3.1 Schematic Display 15
4.3.2 Introduction to Drawings and Functions 16

List of Figures

Figure 3-1 Start Schematic Viewer 3
Figure 3-2 Error 3
Figure 4-1 Schematic Viewer GUI 4
Figure 4-2 Hierarchy Structure 5
Figure 4-3 Expand Nets 6
Figure 4-4 Select Any One Net 6
Figure 4-5 Net Finder Dialog Box 7
Figure 4-6 Bus Net Drop-list 7
Figure 4-7 Right-click to Select Expand Net 8
Figure 4-8 Expand Primitives 8
Figure 4-9 Select One Primitive 9
Figure 4-10 Primitive Property Dialog Box 9
Figure 4-11 Expand Ports 10
Figure 4-12 Select One Port 10
Figure 4-13 Right-click to Select Expand Port 11
Figure 4-14 Port Property Dialog Box 11
Figure 4-15 Expand Modules 12
Figure 4-16 Module Hierarchy 12
Figure 4-17 Select One Module 12
Figure 4-18 Module Property Dialog Box 13
Figure 4-19 Expand Black Boxes 13
Figure 4-20 Select One Black Box 14
Figure 4-21 Black Box Property Dialog Box 14
Figure 4-22 Filter Multiple Objects with Connection 15
Figure 4-23 Top Layer Schematic 15
Figure 4-24 Nets Drawing 16
Figure 4-25 Select a Net 16
Figure 4-26 Hover Over a Net 17
Figure 4-27 Dissolve 17
Figure 4-28 Save as PDF File Dialog Box 18

Figure 4-29 Search Dialog Box 19
Figure 4-30 Search Dialog Box and Schematic Pane 19
Figure 4-31 Select All 20
Figure 4-32 Normal Mode 21
Figure 4-33 Wildcard Mode 21
Figure 4-34 Objects to Search 22
Figure 4-35 Select a Primitive in Schematic 23
Figure 4-36 Hover Over a Primitive 23
Figure 4-37 Hover Over a Primitive Name 24
Figure 4-38 Input Port 25
Figure 4-39 Inout/Out Port 25
Figure 4-40 Select a Port 25
Figure 4-41 Hover Over a Port Module 26
Figure 4-42 Hover Over a Port Name 26
Figure 4-43 Select a Module 27
Figure 4-44 Hover Over a Module 27
Figure 4-45 Hover Over an Instance Name 28
Figure 4-46 Hover Over a Module Definition 28
Figure 4-47 Logic Implementation of uut2 29
Figure 4-48 Select a Black Box 30
Figure 4-49 Hover Over a Black Box 30
Figure 4-50 Hover Over an Instance Name 31
Figure 4-51 Hover Over a Black Box Definition 31

List of Tables

Table 1-1 Abbreviations and Terminology 1

1 About This Guide

1.1 Purpose

This manual describes Gowin Schematic Viewer and introduces the GUI and usage of Schematic Viewer, which aims to help you quickly be familiar with the use of Schematic Viewer. As the software is subject to change without notice, some information may not remain relevant and may need to be adjusted according to the software that is in use.

1.2 Related Documents

The latest user guides are available on GOWINSEMI Website. You can find the related documents at www.gowinsemi.com: SUG100, Gowin Software User Guide.

1.3 Terminology and Abbreviations

Table 1-1 shows the abbreviations and terminology that are employed in this guide.

Table 1-1 Abbreviations and Terminology

Terminology and Abbreviations	Meaning
BSRAM	Block Static Random Access Memory
DSP	Digital Signal Processing
HDL	Hardware Description Language
PLL	Phase-locked Loop
RTL	Register Transfer Level

1.4 Support and Feedback

Gowin Semiconductor provides customers with comprehensive technical support. If you have any questions, comments, or suggestions, please feel free to contact us directly by the following ways.

Website: www.gowinsemi.com

E-mail: support@gowinsemi.com

2 Introduction

Schematic Viewer is designed in-house by Gowin. It aims to help you understand the algorithms and logic in the design more conveniently and intuitively.

The features of Schematic Viewer are as follows.

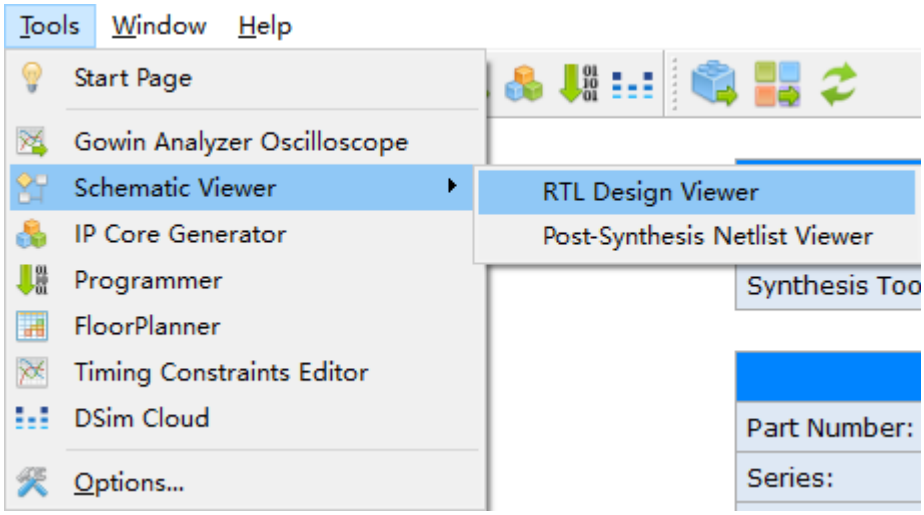
- Displays the hierarchical logic of user RTL design file.
- Displays the hierarchical logic of post-synthesis netlist file.
- Supports the search and retrieval of the modules
- Supports view filter and mouseover of the modules
- Supports the display of RTL logic circuits, such as adder, multiplier, comparator, AND gate, and OR gate.
- Supports the display of Gowin primitive devices, such as look-up table, register, BSRAM, PLL, DSP.

3 Start Schematic Viewer

You can start Schematic View via Gowin Software.

Click "IDE > Tools" to start "Schematic View", as shown in Figure 3-1. The drop-down menu options include "RTL Design Viewer" and "Post-Synthesis Netlist Viewer". The "RTL Design Viewer" displays the hierarchical logic of the user RTL design and the "Post-Synthesis Netlist Viewer Viewer" displays the hierarchical logic of the post-synthesis netlist. The "Post-Synthesis Netlist Viewer" option can only be selected after running "Synthesize" successfully in Gowin Software Process window.

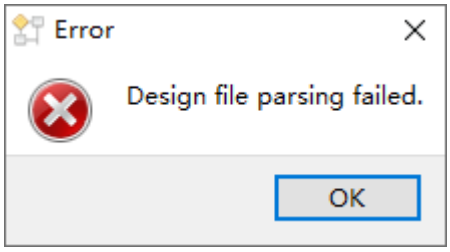
Figure 3-1 Start Schematic Viewer



Note!

You should first load the source file in the IDE project to start Schematic View. If the source file has a syntax error, Schematic Viewer cannot be opened and the Error prompt will pop up, as shown in Figure 3-2.

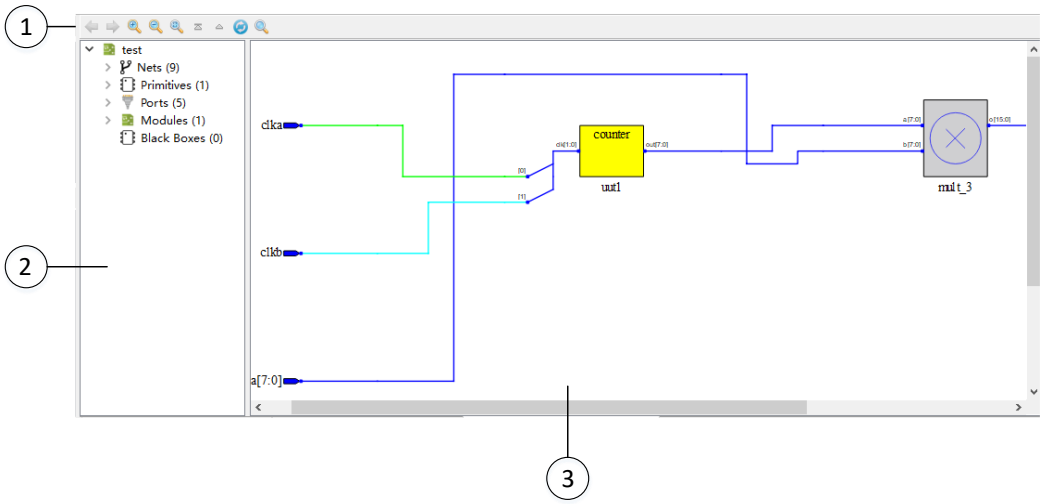
Figure 3-2 Error



4 GUI

The GUI includes three parts: toolbar, hierarchy pane and schematic pane, as shown in Figure 4-1.

Figure 4-1 Schematic Viewer GUI



① Tool Bar
② Hierarchy Pane
③ Schematic Pane

4.1 Tool Bar

It provides quick access to some commonly used functions, and the buttons from left to right are:

- "": Previous
- "": Next
- "": Zoom In
- "": Zoom Out
- "": Zoom Fit
- "": Show Top View
- "": Pop Hierarchy

- " ": Reload
- " ": Search

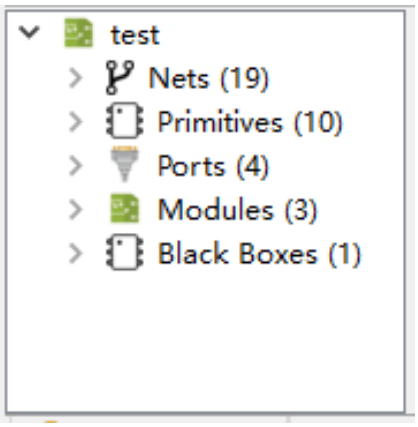
4.2 Hierarchy Pane

The hierarchy pane displays the hierarchy structure of all the objects in the design. You can clearly learn the hierarchy structure, all the modules at each hierarchy and the number.

4.2.1 Hierarchy Structure

The objects in the design are divided into five types: "Nets", "Primitives", "Ports", "Modules", and "Black Boxes". As shown in Figure 4-2, the top module of each object and its number are displayed by default.

Figure 4-2 Hierarchy Structure



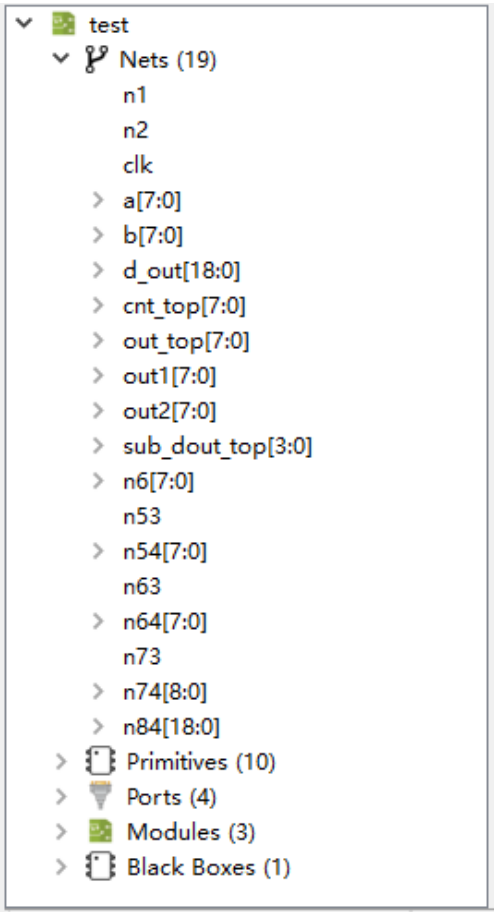
4.2.2 Introduction to Objects

You can right-click and click on each type of object, which can interact with the schematic pane.

Nets

The Nets displays the net parsed in the design. Click the button before Nets to display all the nets at the current level. The number of nets parsed in the design is marked in parenthesis, as shown in Figure 4-3.

Figure 4-3 Expand Nets

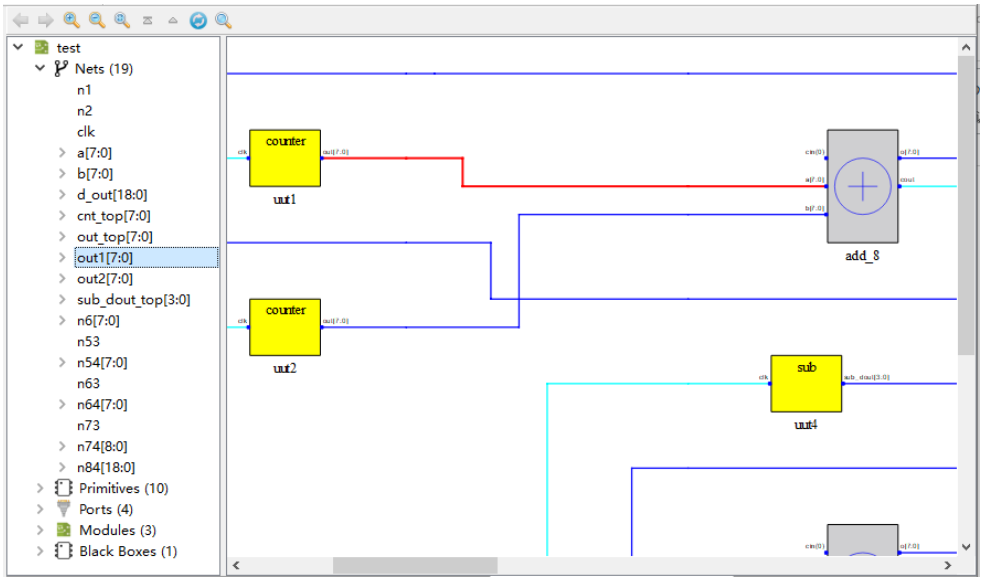


Note!

In the statistics of the number of nets, Bus Net is as one net.

Click any one net, and the corresponding net in the schematic pane will be selected in red. As shown in Figure 4-4, select "out1[7:0]" and the corresponding out1[7:0] in the schematic pane is selected.

Figure 4-4 Select Any One Net



The Nets supports the right-click menu and the options are as follows.

- Property: View the properties of the selected net. As shown in Figure 4-5, the properties include "type", "source pin", "sink pin", and "fanout", where "source pin" is the pin corresponding to the source, and "sink pin" is the pin corresponding to the sink. The drop-down list of bus Net Property allows you to select different bit widths, as shown in Figure 4-6.

Figure 4-5 Net Finder Dialog Box

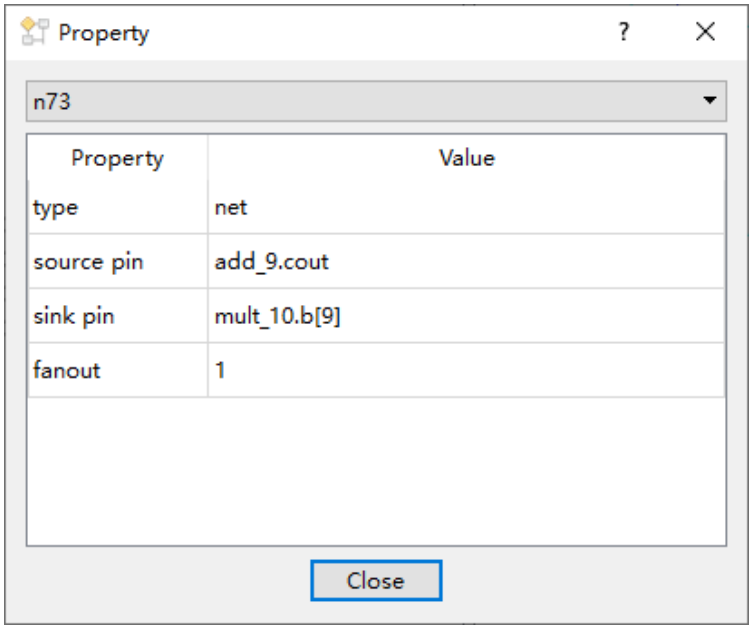
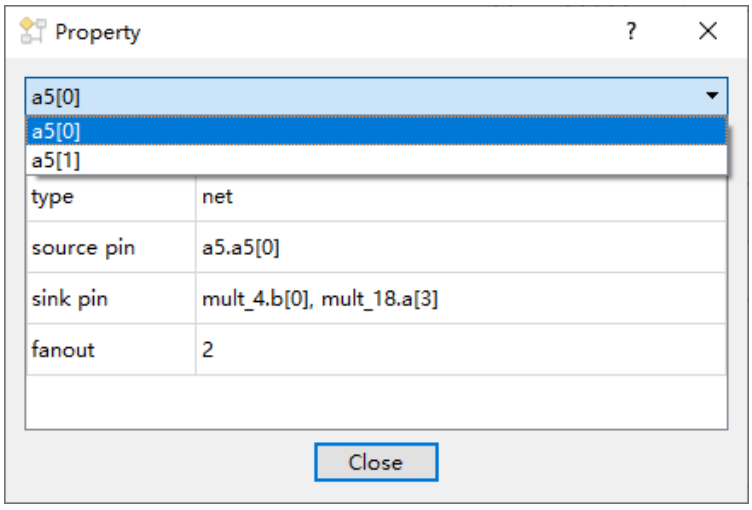
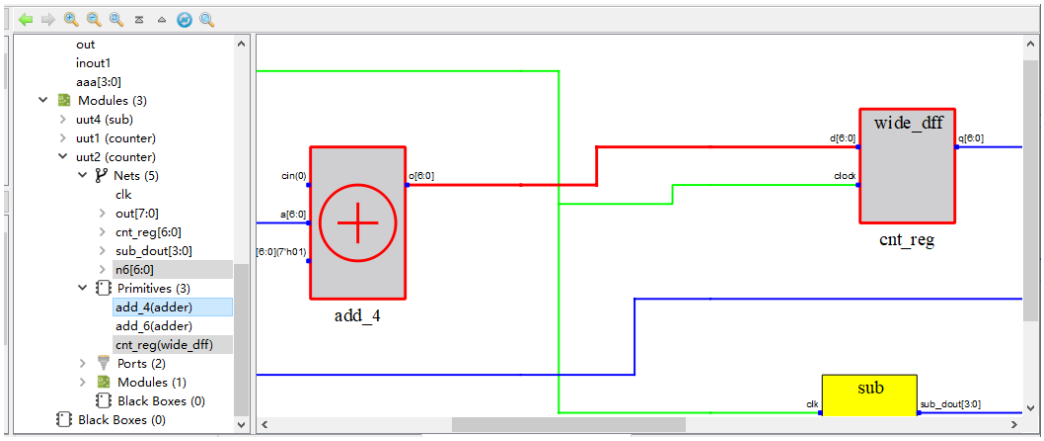


Figure 4-6 Bus Net Drop-list



- Expand Net: The net in hierarchy and schematic view and the source and sink of this net are all selected. As shown in Figure 4-7, when n6[7:0] is selected, right-click to select "Expand Net", the source "add_4" and sink "cnt_top" of n6[7:0] are also selected.

Figure 4-7 Right-click to Select Expand Net

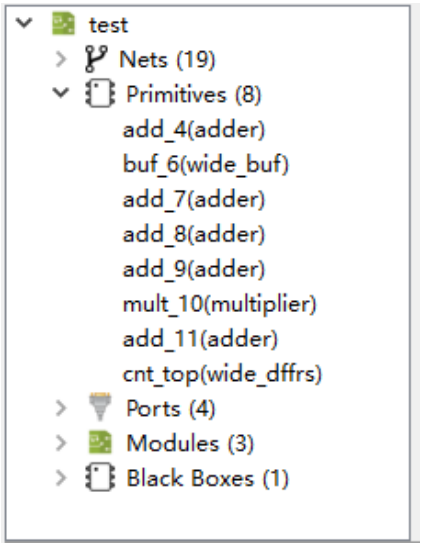


- Zoom Fit Selected: Select a Net to zoom in/out the schematic to fit.

Primitives

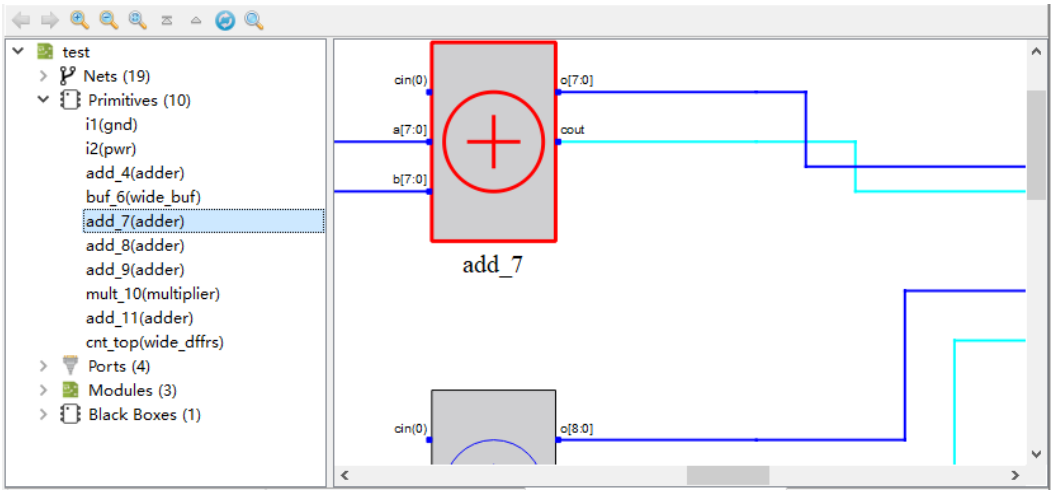
The Primitives can be logic circuits, such as, an adder, a multiplier, a comparator, an AND gate, an OR gate or Gowin primitive devices, such as a lookup table, a register, a BSRAM, a PLL, or a DSP. As shown in Figure 4-8, you can click the button before Primitives to display all the primitives at the current level, and the corresponding component is marked in parentheses.

Figure 4-8 Expand Primitives



Click any one primitive, and the corresponding primitive in the schematic pane will be selected in red. As shown in Figure 4-9, select "add_7" and the corresponding"add_7" in the schematic pane is selected.

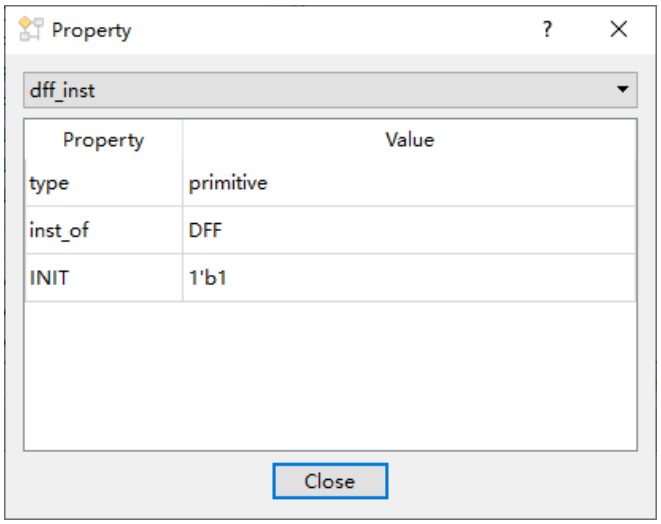
Figure 4-9 Select One Primitive



The Primitives support the right-click menu and the options are as follows.

- View Instance In Source: Jumps to the implementation of the selected primitive in the source file.
- Property: View the properties of the selected primitive. The properties of primitive include "type", "inst_of" and INIT, where "inst_of" is the component corresponding to a primitive. When a primitive has an INIT, the Property window will have the INIT value, as shown in Figure 4-10.

Figure 4-10 Primitive Property Dialog Box



- Filter: Displays the selected primitive separately in the schematic pane.
- UnFilter: Skips back to the old schematic.
- Zoom Fit Selected: Selects a Primitive to zoom in/out the schematic to fit.

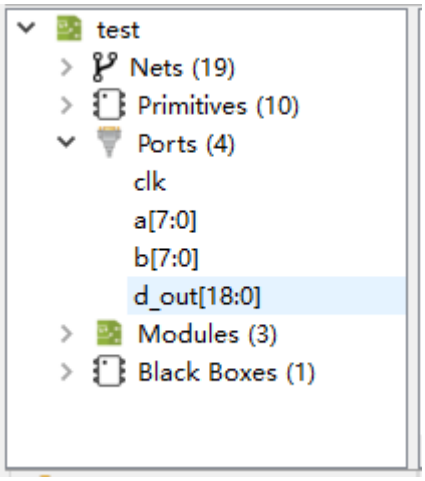
Note!

The Unfilter appears only if you right-click the menu after the Filter operation.

Ports

The Ports display the defined ports in the design. Click the button before Ports to display all the ports at the current level, as shown in Figure 4-11.

Figure 4-11 Expand Ports

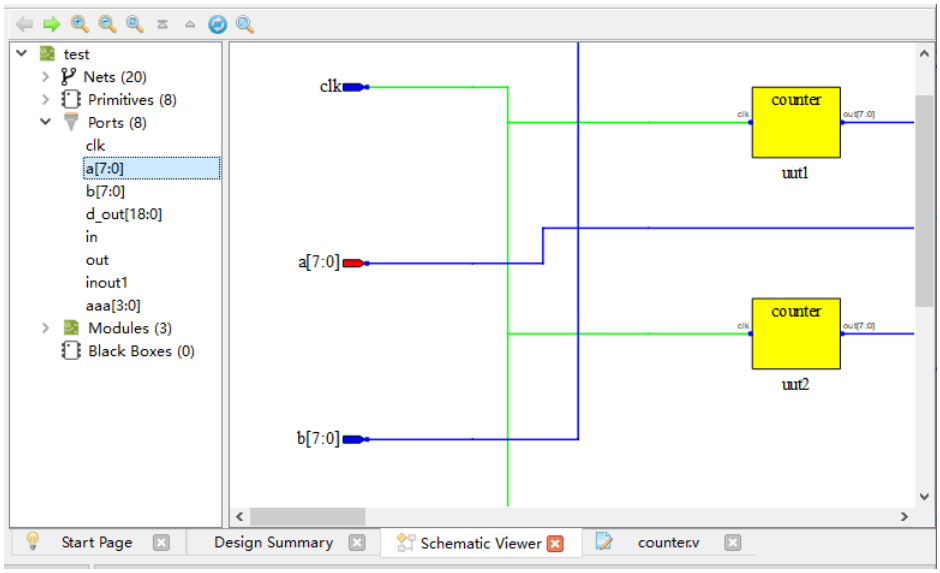


Note!

In the statistics of the number of ports, Bus Port is as one port.

Click any one port, and the corresponding port in the schematic pane will be selected in red. As shown in Figure 4-12, select "a[7:0]" and the corresponding "a[7:0]" in the schematic pane is selected.

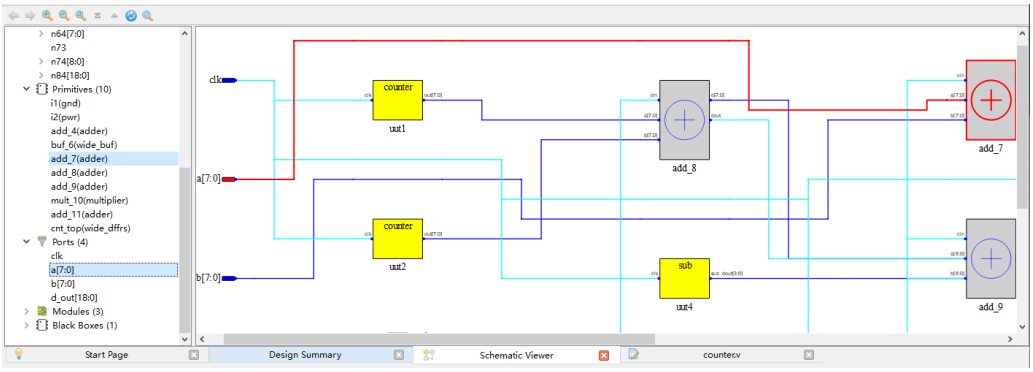
Figure 4-12 Select One Port



The Ports support the right-click menu and the options are as follows.

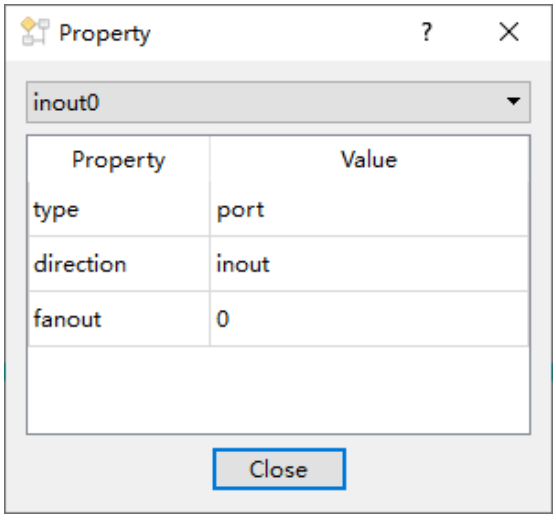
- View Instance In Source: Jumps to the port in the source file.
- Expand Port: The port in hierarchy and schematic, the object driving or driven by this port and the net are selected. As shown in Figure 4-13, when a[7:0] is selected, right-click to select "Expand Net", the "add_7" and net are also selected.

Figure 4-13 Right-click to Select Expand Port



Property: View the properties of the selected Port. The properties include "type", "direction" and "fanout", as shown in Figure 4-14.

Figure 4-14 Port Property Dialog Box



- Filter: Displays the selected Port separately in the schematic pane.
- UnFilter: Skips back to the old schematic.
- Zoom Fit Selected: Selects a Port to zoom in/out the schematic to fit.

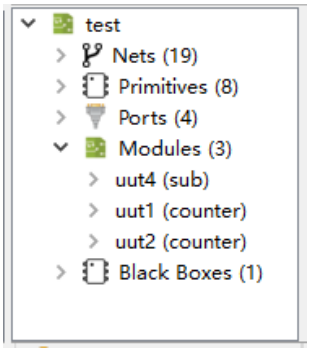
Note!

The Unfilter appears only if you right-click the menu after the Filter operation.

Modules

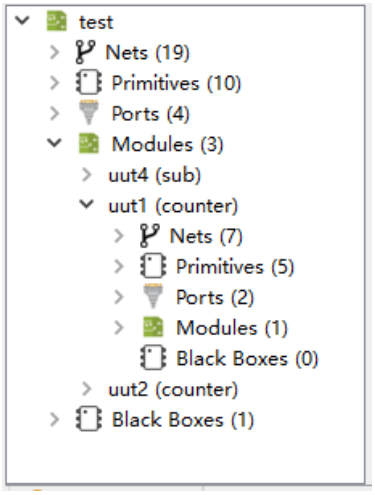
The modules in the hierarchy display the bottom module of all the logic of current instantiated hierarchy. As shown in Figure 4-15, you can click the button before Module to display all the instantiated modules at the current level, and the corresponding name is marked in parentheses.

Figure 4-15 Expand Modules



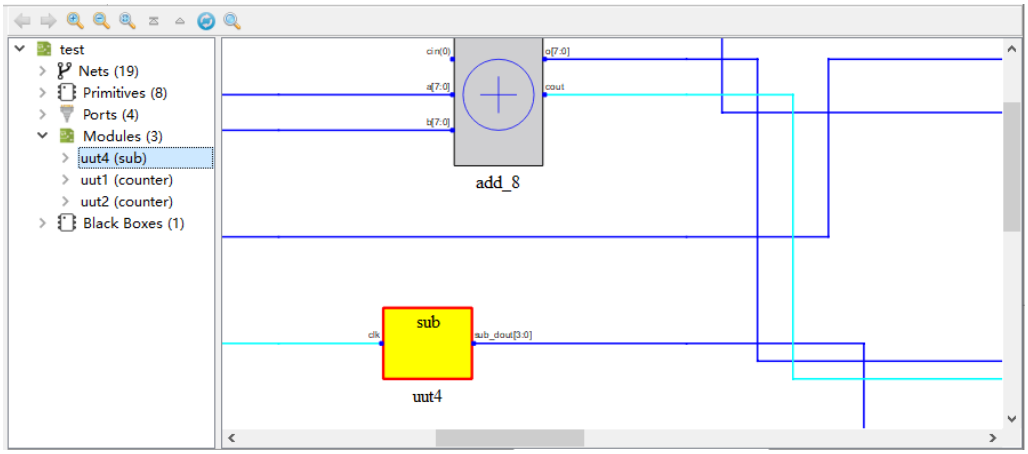
Each Module is also displayed by Nets, Primitives, Ports, Modules, and Black Boxes. Figure 4-16 shows the hierarchy of bottom module "uut1".

Figure 4-16 Module Hierarchy



Click any one module, and the corresponding module in the schematic pane will be selected in red. As shown in Figure 4-17, select "uut4" and the corresponding "uut4" in the schematic pane is selected.

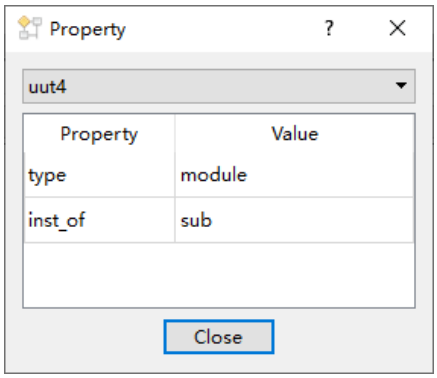
Figure 4-17 Select One Module



The Modules support the right-click menu and the options are as follows.

- View Instance In Source: Jumps to the instance of the selected Module in the source file.
- View Module In Source: Jumps to the module definition in the source file.
- Push: Displays the logic circuit of the selected Module.
- Property: Views the properties of the selected Module. The properties of Module include "type" and "inst_of", where "inst_of" is the defined name of Module, as shown in Figure 4-18.

Figure 4-18 Module Property Dialog Box



- Filter: Displays the selected Module separately in the schematic pane.
- UnFilter: Skips back to the old schematic.
- Zoom Fit Selected: Selects a module to zoom in/out the schematic to fit.

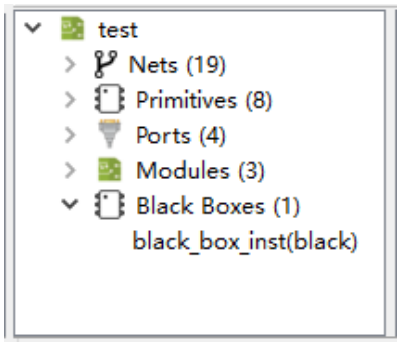
Note!

The Unfilter appears only if you right-click the menu after the Filter operation.

Black Boxes

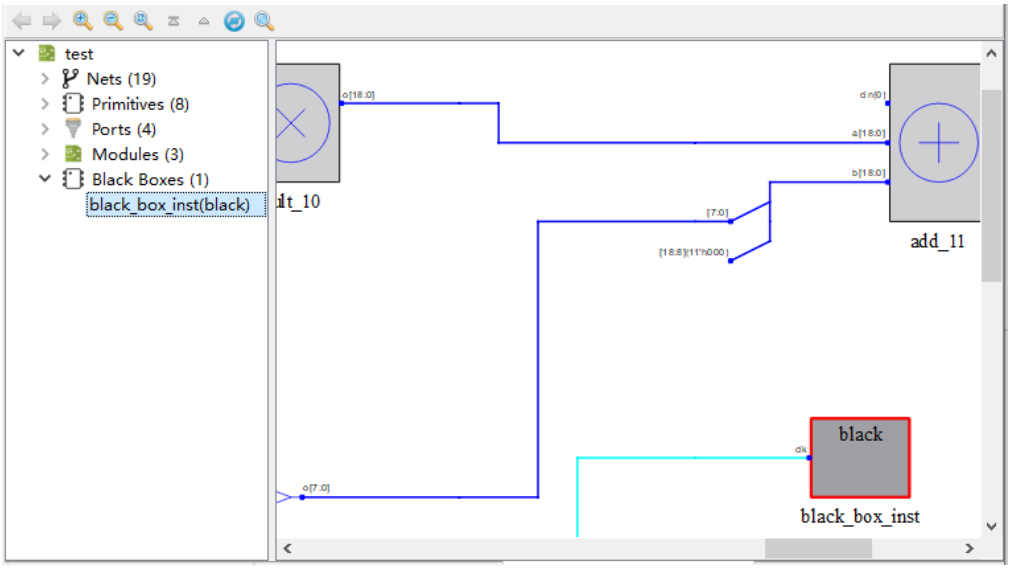
The Black Boxes in the hierarchy displays that only the bottom module instantiated at the current level is defined without a logical implementation or is encrypted. As shown in Figure 4-19, you can click the button before Black Box to display all the Black Boxes at the current level, and the corresponding name is marked in parentheses.

Figure 4-19 Expand Black Boxes



Click any one Black Box, and the corresponding Black Box in the schematic pane will be selected in red. As shown in Figure 4-20, select "black_box_inst" and the corresponding "black_box_inst" in the schematic pane is selected.

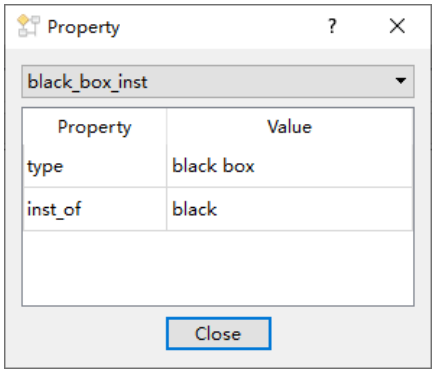
Figure 4-20 Select One Black Box



The Black Boxes support the right-click menu and the options are as follows.

- View Instance In Source: Jumps to the instance of the selected Black Box in the source file.
- View Module In Source: Jumps to the definition of Black Box in the source file, or jumps to the header of the source file where Black Box is defined if Black Box is an encryption.
- Property: View the properties of the selected Black Box. The properties of Black Box, including "type" and "inst_of", where "inst_of" is the defined name of Black Box, as shown in Figure 4-21.

Figure 4-21 Black Box Property Dialog Box



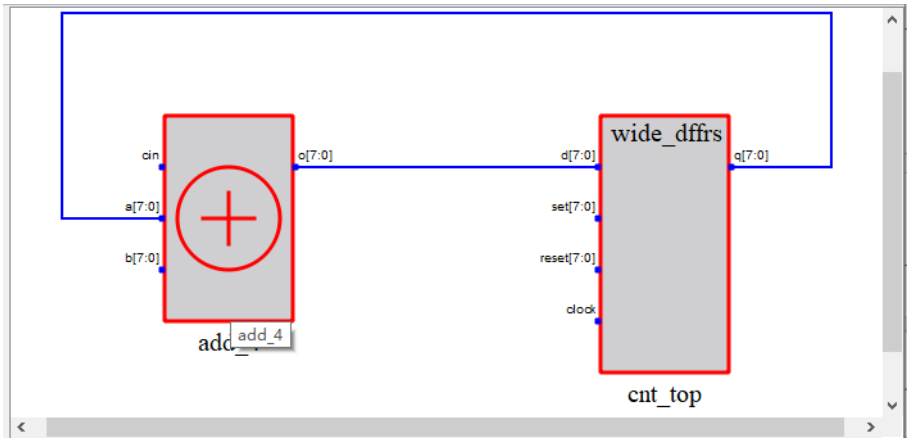
- Filter: Displays the selected Black Box separately in the schematic pane.
- UnFilter: Skips back to the old schematic.

- Zoom Fit Selected: Selects a Black Box to zoom in/out the schematic to fit.

Note!

When multiple objects such as Primitives, Ports, Modules, and Black Boxes are selected for right-click to perform Filter operation, if each object has a connection, the Net will also be displayed together. As shown in Figure 4-22, when you select "add_4" and "cnt_top", right-click and select Filter, the net between them is also filtered out for display.

Figure 4-22 Filter Multiple Objects with Connection



Note!

The Unfilter appears only if you right-click the menu after the Filter operation.

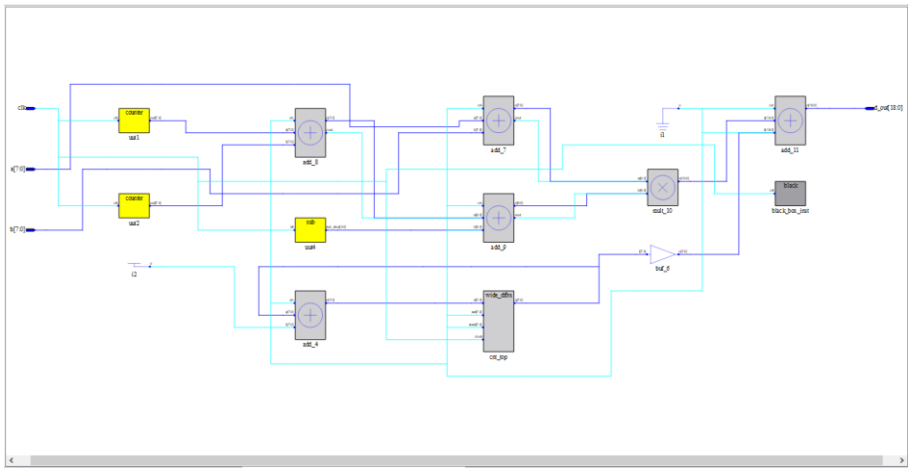
4.3 Schematic Pane

On the right side of the GUI is the schematic pane, which displays the logical connections of the modules in the design with the hierarchy structure. You can understand the design logic more clearly, which is helpful to trace the signal flow and the connection of each module.

4.3.1 Schematic Display

In the schematic, all Primitives, Ports, Modules, Black Boxes are connected by nets according to the logic of design, and the design is drawn in circuit diagrams. The schematic is displayed with hierarchy structure. Figure 4-23 shows a top logic shematic. If you want to view the bottom logic, select the bottom module and right-click to select Push.

Figure 4-23 Top Layer Schematic



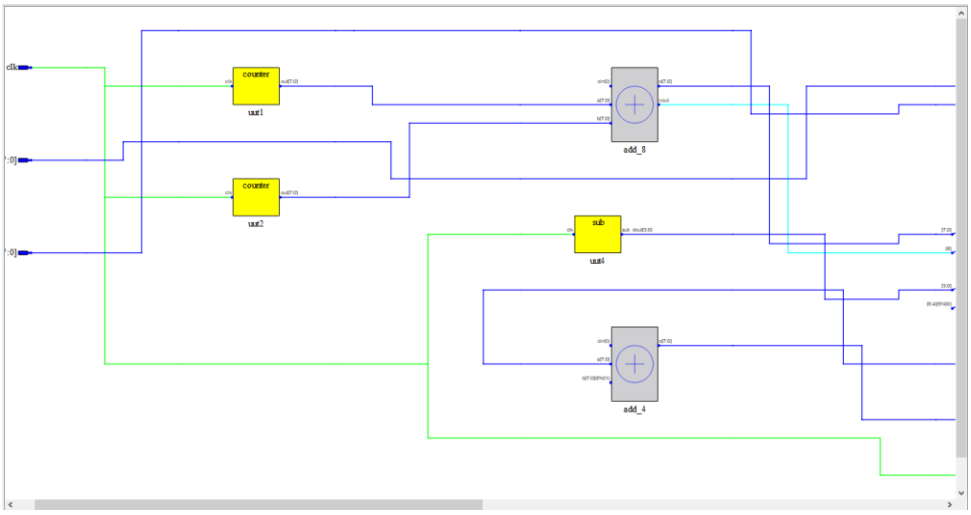
4.3.2 Introduction to Drawings and Functions

You can right-click, click and double-click each object in the schematic, which can interact with the hierarchy pane.

Net

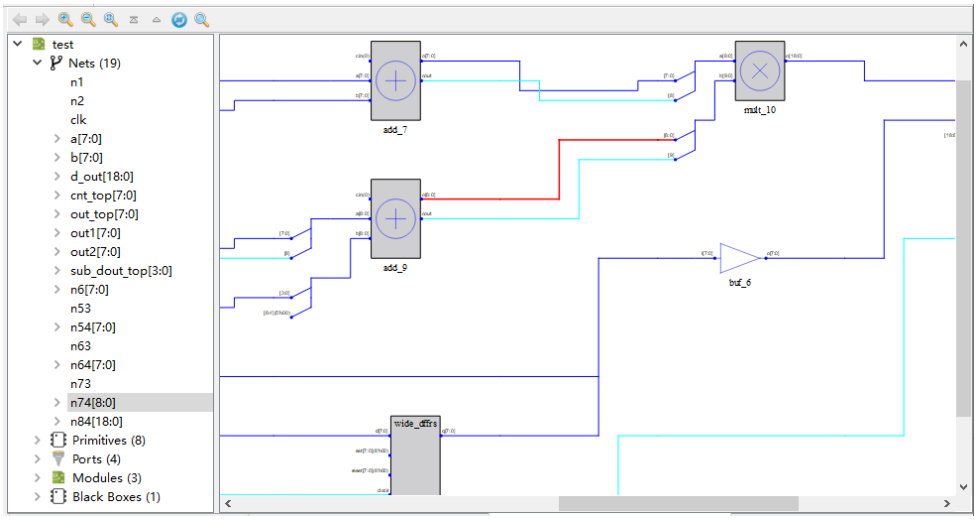
The nets in the schematic are connected according to the Primitives, Ports, Modules, Black, Boxes modules in the design. Single Net is drawn in light blue, Bus Net is drawn in dark blue, and Clk Net is drawn in light green. As shown Figure 4-24, the light blue one is Single Net, the dark blue one is Bus Net, and the light green one is Clk Net.

Figure 4-24 Nets Drawing



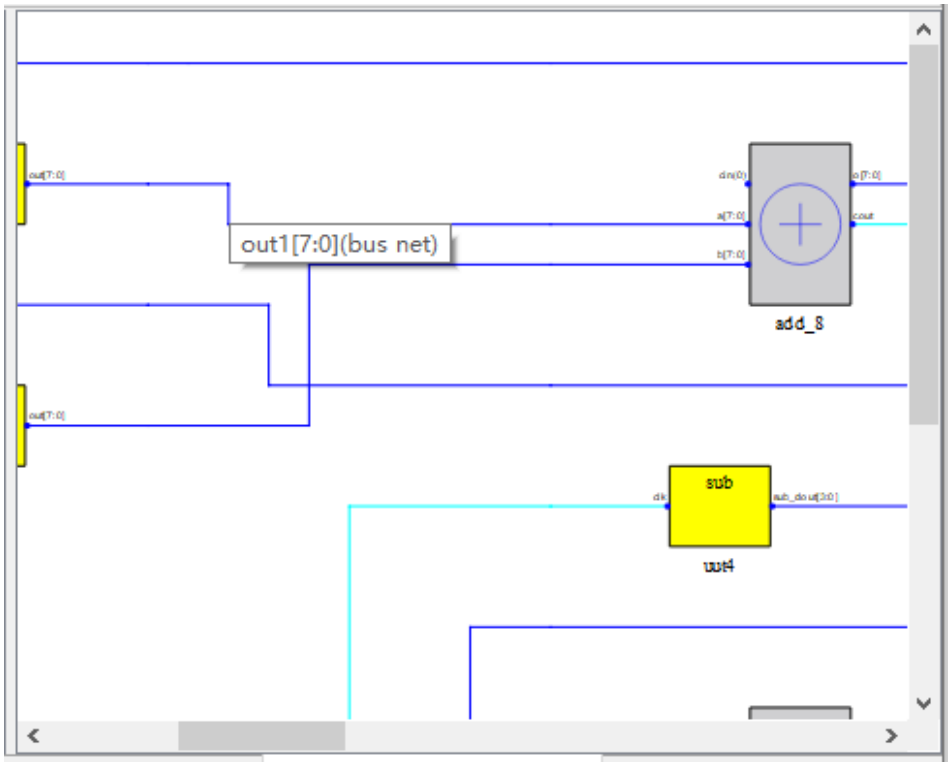
After you select a net in the schematic, the corresponding net in the hierarchy pane is selected, as shown in Figure 4-25.

Figure 4-25 Select a Net



When your mouse hovers over the net, it can display the name and type of the net, as shown in Figure 4-26.

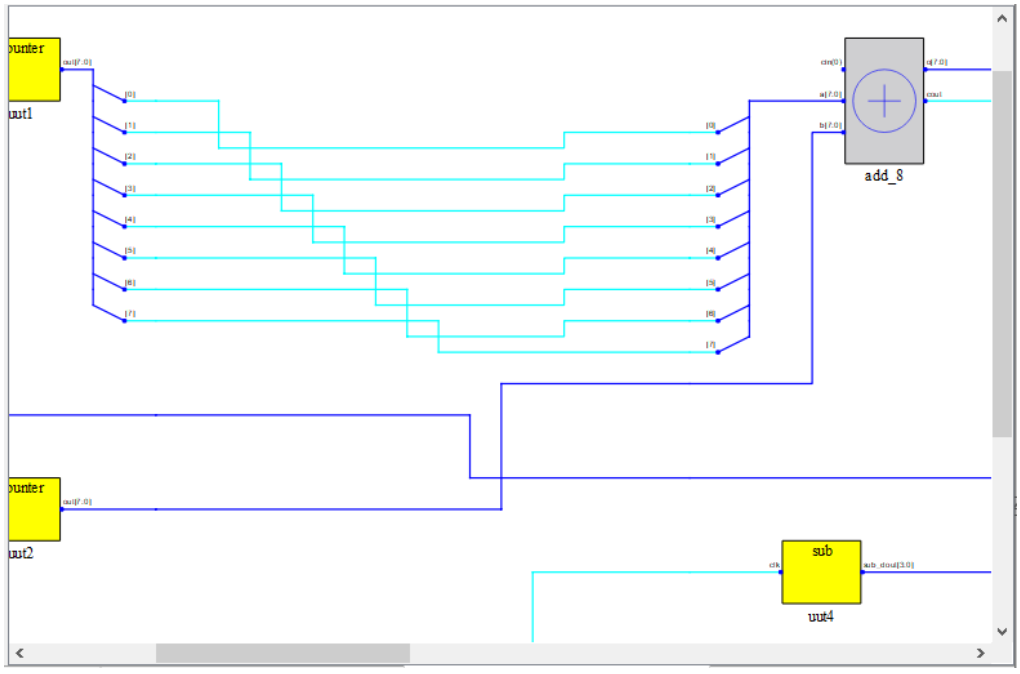
Figure 4-26 Hover Over a Net



The Net in the schematic supports right-click menu and the options are as follows.

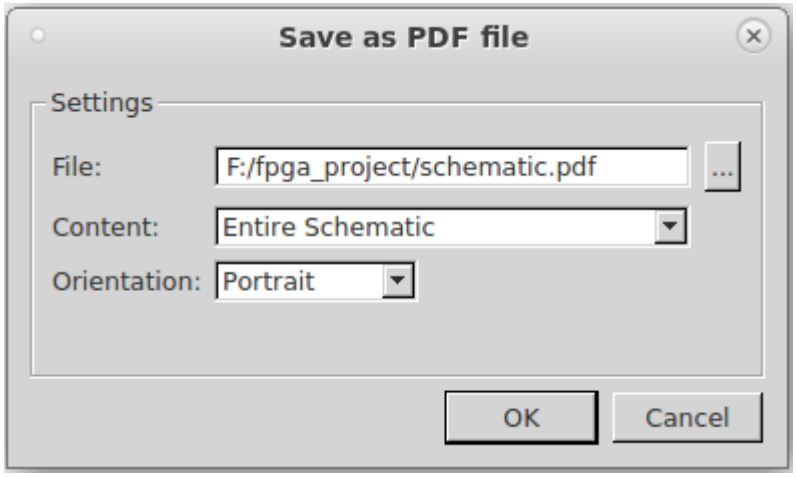
- Property: View the properties of net.
- Dissolve: Dissolve Bus Net to Single Net, only Bus Net has this function; Right-click to dissolve "out1[7:0]" in the Figure 4-26, as shown in Figure 4-27.

Figure 4-27 Dissolve



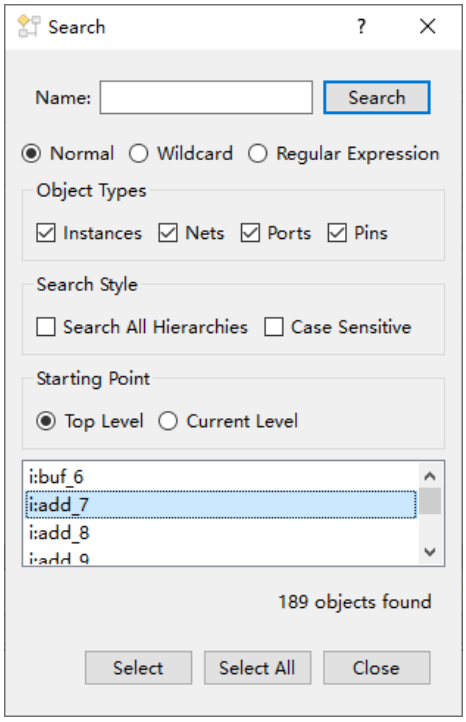
- Combine: Combine the dissolved Single Nets to Bus Net, which is only available for the dissolved nets.
- Expand Net: The net in hierarchy and schematic and the source and sink of this net are all selected.
- Zoom Fit Selected: Select a net to zoom in/out the schematic to fit.
- Select All Ports: Select all Ports modules in the displayed schematic.
- Select All Primitives: Select all Primitives modules in the displayed schematic.
- Select All Modules: Select all Modules in the displayed schematic.
- Zoom In: Zoom in schematic.
- Zoom Out: Zoom out schematic.
- Zoom Fit: Zoom in/out the schematic to fit.
- Save as PDF File: Save the schematic view as a PDF file, as shown in Figure 4-28.

Figure 4-28 Save as PDF File Dialog Box



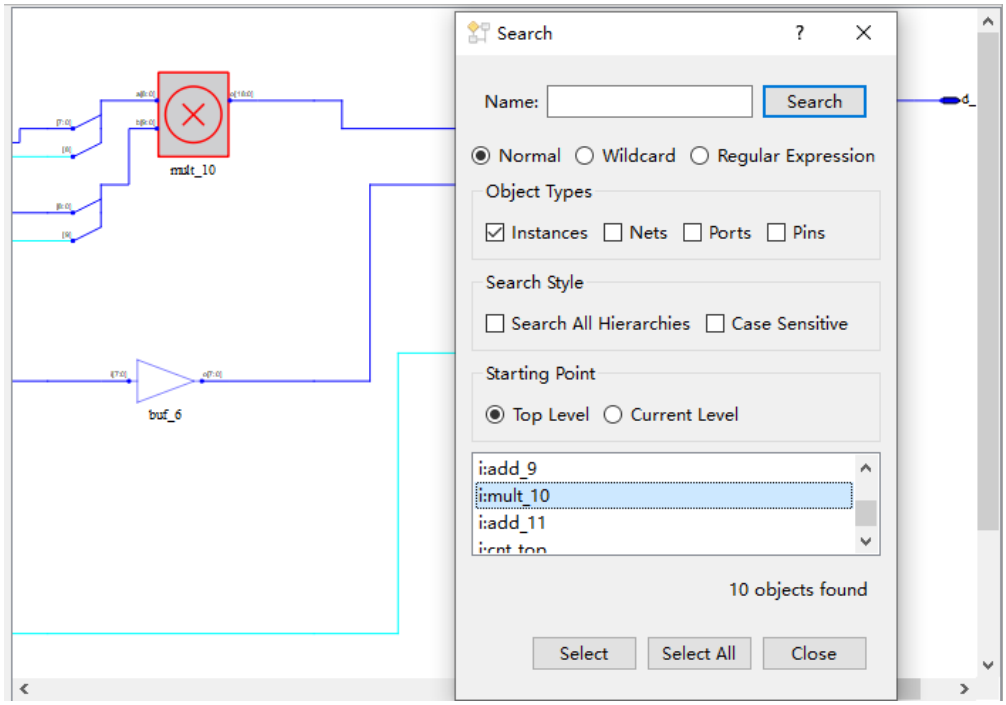
  - File: Set the save path of the PDF file.
  - Content: Set the content of the schematic view, including Entire Schematic and Visible part of Schematic; Entire Schematic is the default option, and it means saving the entire current schematic view; Visible part of Schematic means saving the current visible schematic view.
  - Orientation: Set the direction in which the schematic view is saved, including Portrait and Landscape; Portrait is the default option, and it meaning saving the schematic view vertically, and Landscape means saving it horizontally.
- Search: Search objects in the design. You can search all objects in the design according to the search criteria, as shown in Figure 4-29.

Figure 4-29 Search Dialog Box



Select an object and click "Search", this object is also selected in the schematic. As shown in Figure 4-30, select "mult_10" in the Search dialog box, and click the "Select" below, then "mult_10" in the schematic is selected.

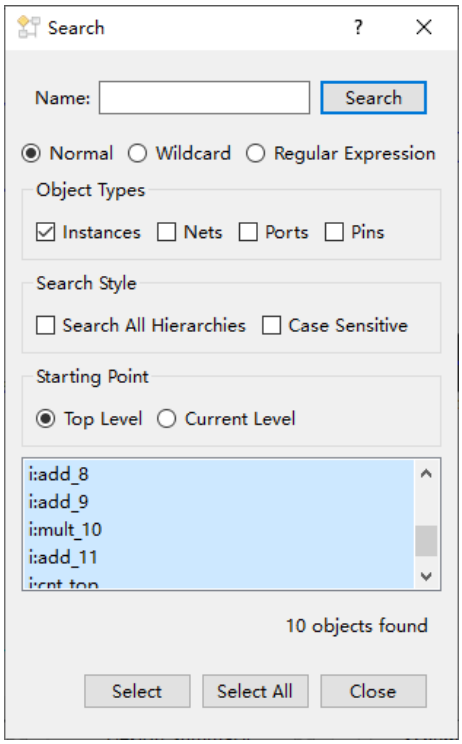
Figure 4-30 Search Dialog Box and Schematic Pane



As shown in Figure 4-31, click the "Select All" to select all the

searched objects displayed in the Search Dialog Box. Click the "Close" to close the Search Dialog Box.

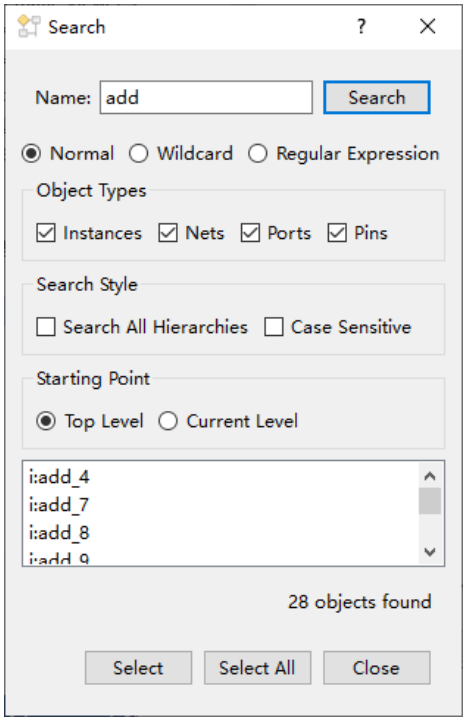
Figure 4-31 Select All



Normal, Wildcard, and Regular Expression are mutually exclusive, and the default is Normal.

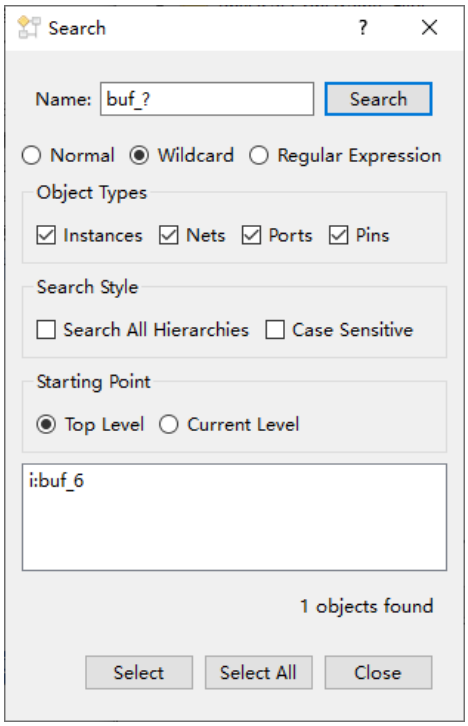
Normal means searching in a normal way. Click "Search" above to match the string in "Name" text box, as shown in Figure 4-32.

Figure 4-32 Normal Mode



Wildcard means searching with wildcard. Click "Search" above to match the string in "Name" text box. The string contains wildcards (*,?) , as shown in Figure 4-33.

Figure 4-33 Wildcard Mode



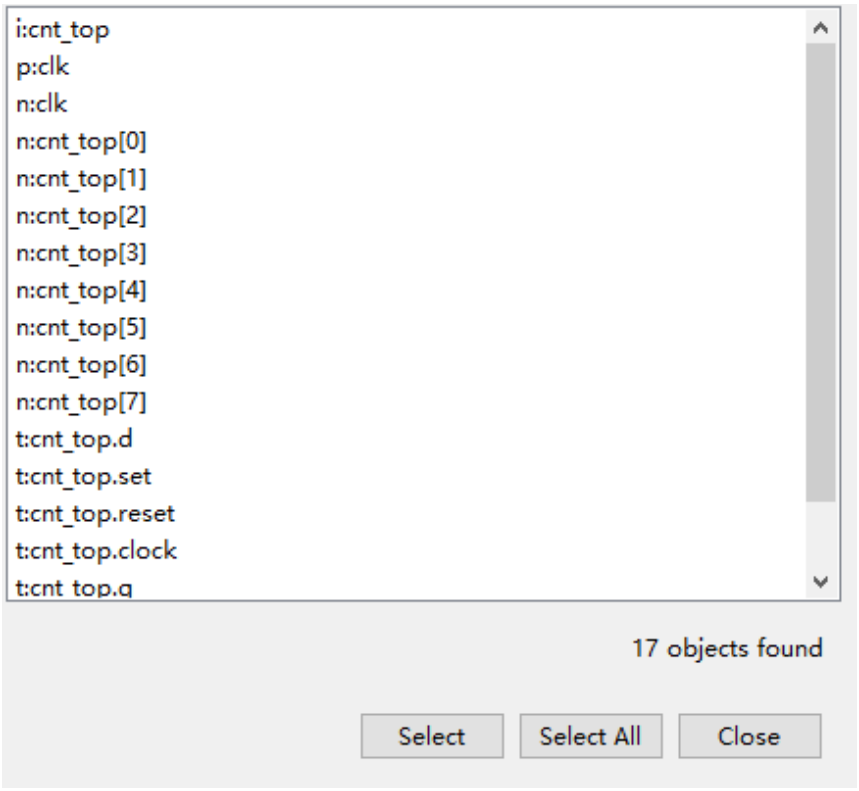
Regular expression means searching with a regular expression. Click "Search" to match the string in "Name" text box. The string can be regular

expression.

- The Instances, Nets, Ports, and Pins options in Object Types can be checked, all checked by default. You can set the type of object in Object Types.
- The Search All Hierarchies and Case Sensitive options in Search Style can be checked. Neither option is checked by default. You can set search mode in Search Style. Check Search All Hierarchies, it will search for objects that meet the criteria. Check Case Sensitive, it will search objects with case sensitive.
- Top Level and Current Level are mutually exclusive in Starting Point. Top Level is checked by default. You can set search start point in Starting Point. Top Level starts from the top module of the design, and Current Level starts from the current displayed schematic.

There is a letter before all the search objects in the Name text, which represents the type of objects. As shown in Figure 4-34, "i" represents instances, "n" represents nets, "p" represents ports, and "t" represents the pins.

Figure 4-34 Objects to Search

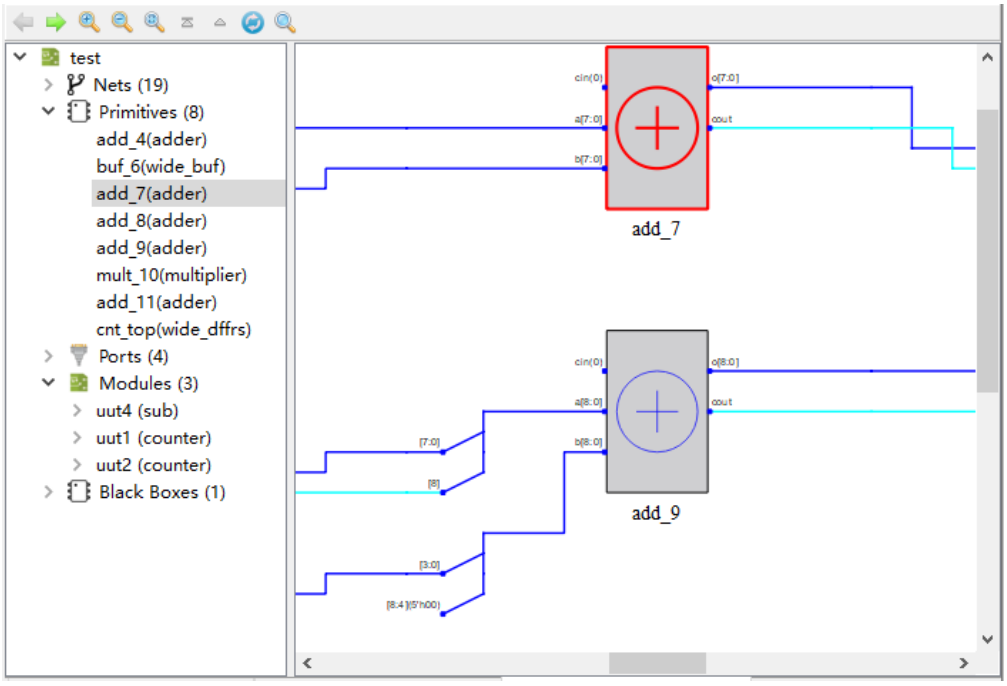


Primitive

In primitive drawing, the background color of AND gate, OR gate, inverters, and buffer is transparent, the rest background color is light gray.

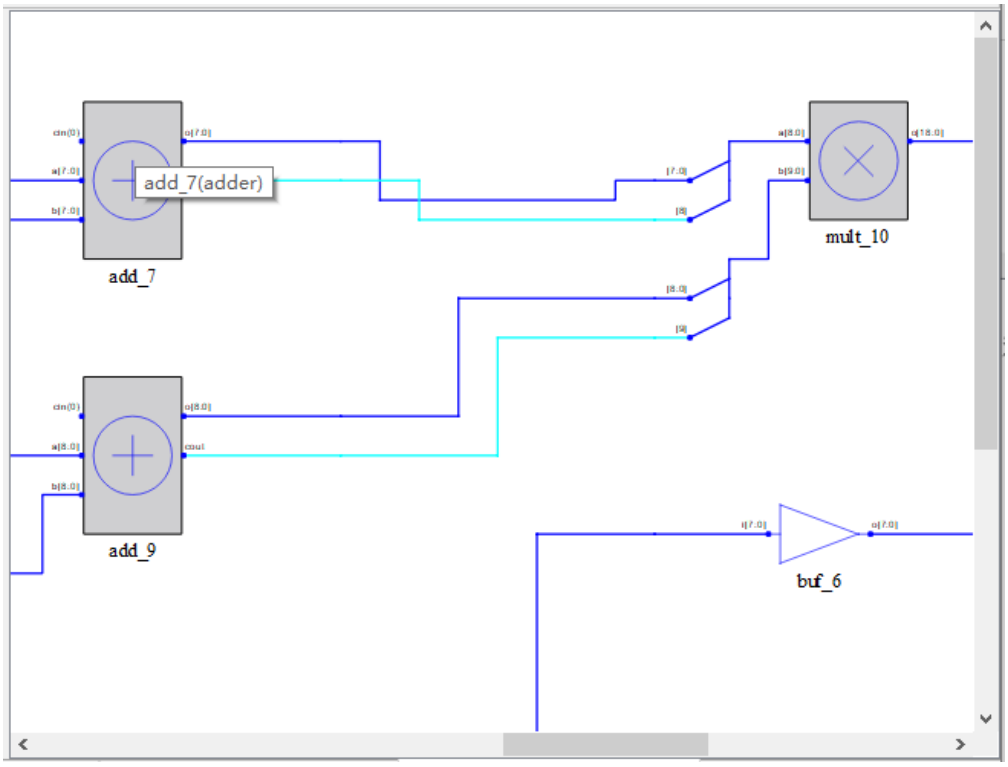
After you select a Primitive schematic, the corresponding net in the hierarchy pane is selected, as shown in Figure 4-35.

Figure 4-35 Select a Primitive in Schematic



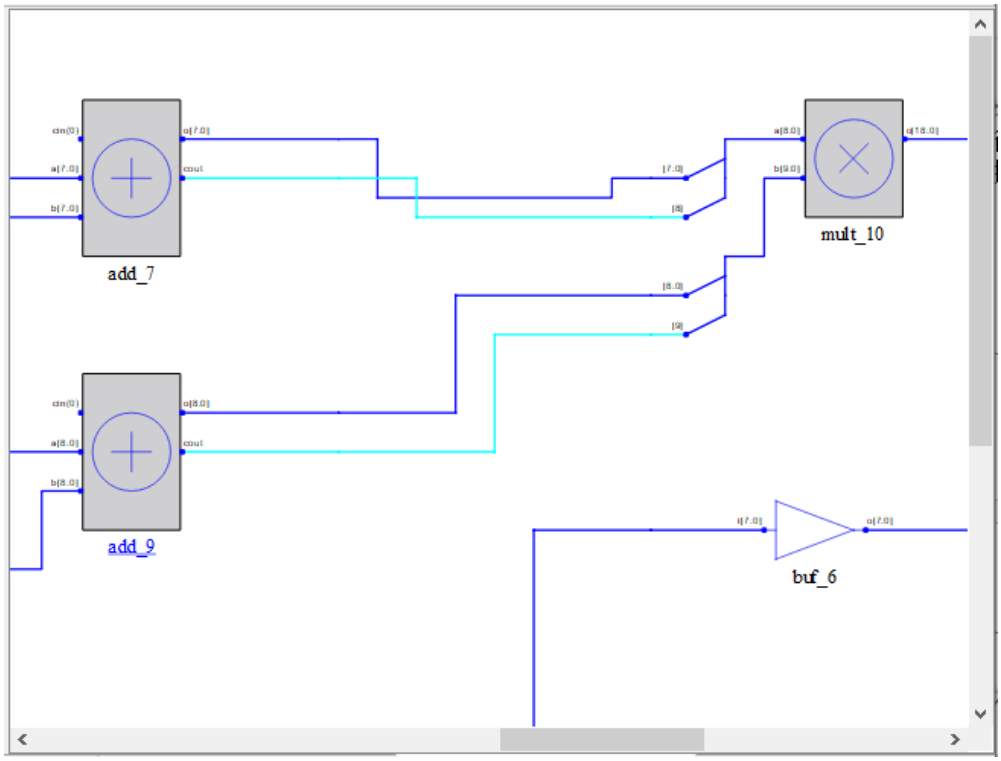
When your mouse hovers over the primitive, it will display the name and component type of the primitive, as shown in Figure 4-36.

Figure 4-36 Hover Over a Primitive



The mouse hovers over the primitive, which is displayed as a link. As shown in Figure 4-37, "add_9" name appears as a link, and click the link, it can go to the implementation of the selected primitive in the source file.

Figure 4-37 Hover Over a Primitive Name



The Primitive in the schematic supports right-click menu and the options are as follows.

- View Instance In Source: Jumps to the implementation of the selected primitive in the source file.
- Property: Views the properties of the selected primitive.
- Filter: Displays the selected primitive separately in the schematic pane.
- UnFilter: Skips back to the old schematic.
- Zoom Fit Selected: Selects a Primitive to zoom in/out the schematic to fit.
- The functions of Select All Ports, Select All Primitives, Select All Modules, Zoom In, Zoom Out, Zoom Fit, Save as PDF File, and Search are same as the those of Net.

Port

The ports in the schematic is dark blue and vary depending on its direction. Figure 4-38 shows the input port drawing and Figure 4-39 shows the inout/out port drawing.

Figure 4-38 Input Port

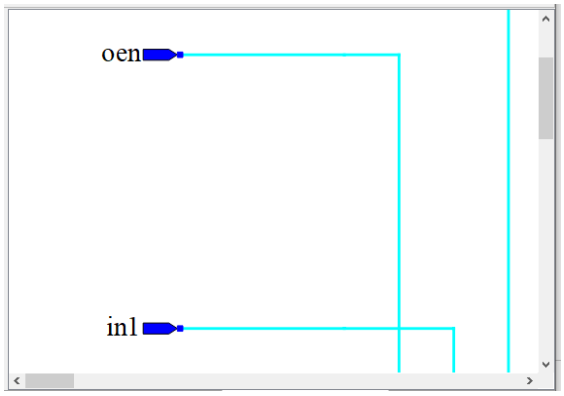
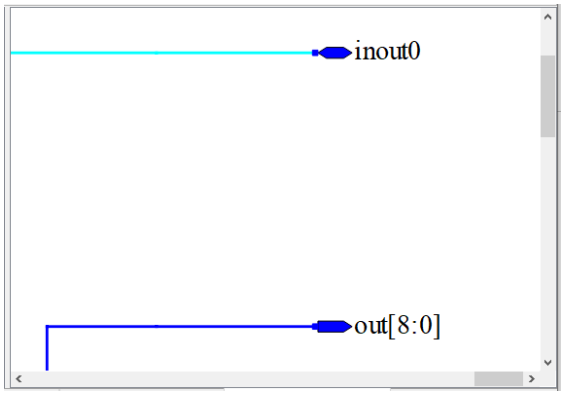
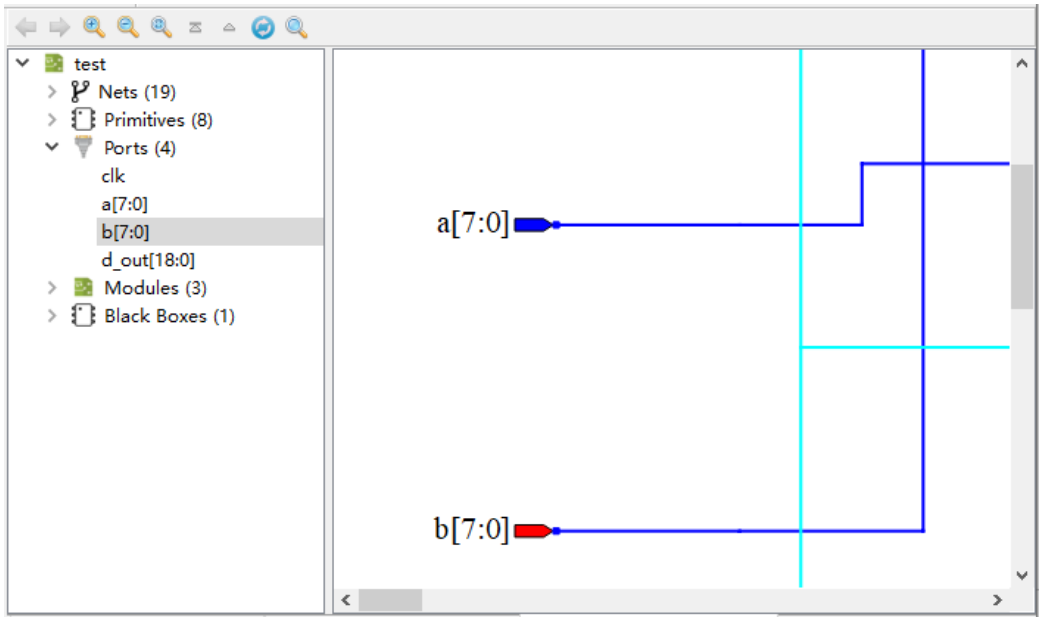


Figure 4-39 Inout/Out Port



After you select a Port in the schematic, the corresponding port in the hierarchy pane is selected, as shown in Figure 4-40.

Figure 4-40 Select a Port



The port module also supports mouse hover and link to the source file.

As shown in Figure 4-41, when your mouse hovers over the port module, it can display the name, type, and defined direction of the port. As shown in Figure 4-42, when your mouse hovers over the port name, the port name is displayed as a link, and click the port name to go to the corresponding port definition in the source file.

Figure 4-41 Hover Over a Port Module

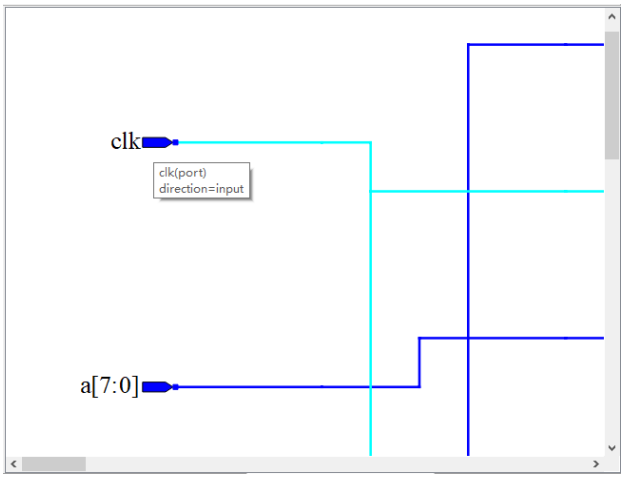
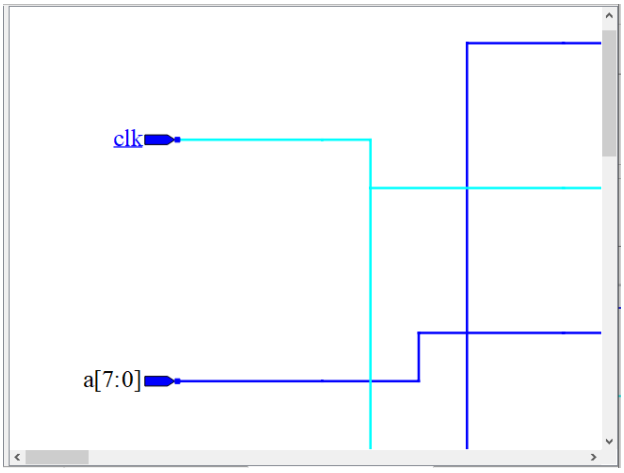


Figure 4-42 Hover Over a Port Name



The Port in the schematic supports right-click menu and the options are as follows.

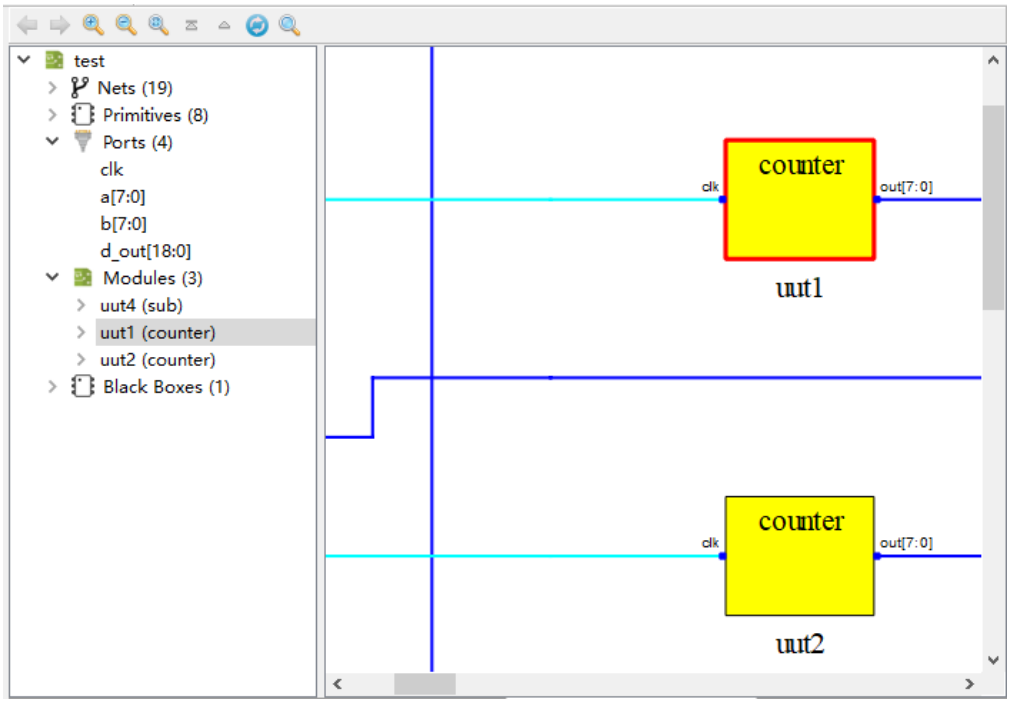
- View Instance In Source: Jumps to the instance of the selected Port in the source file.
- Expand Port: The port in hierarchy and schematic, the port to drive or driven and the net are selected.
- Property: Views the properties of the selected Port.
- Filter: Displays the selected Port separately in the schematic pane.
- UnFilter: Skips back to the old schematic.
- Zoom Fit Selected: Select a Port to zoom in/out the schematic to fit.
- The functions of Select All Ports, Select All Primitives, Select All

Modules, Zoom In, Zoom Out, Zoom Fit, Save as PDF File, and Search are same as the those of Net.

Module

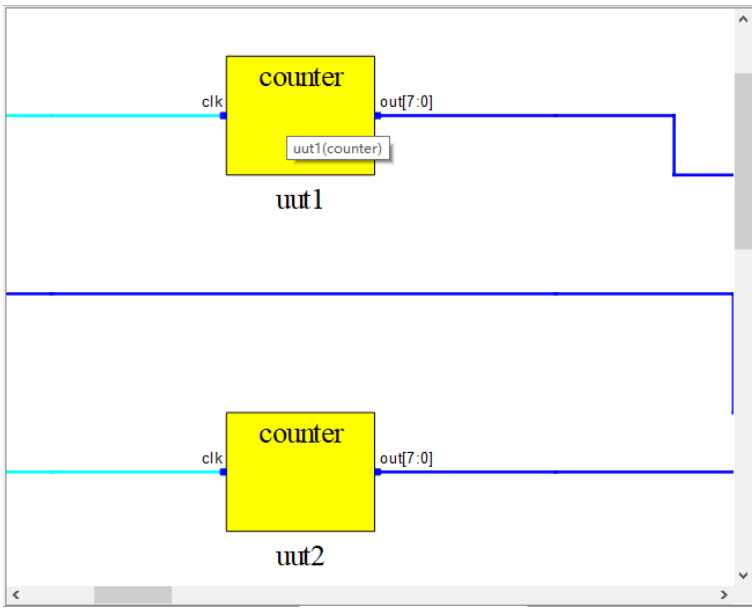
The module drawing in the schematic is with a light yellow background. After you select a module in the schematic, the corresponding module in the hierarchy pane is selected, as shown in Figure 4-43.

Figure 4-43 Select a Module



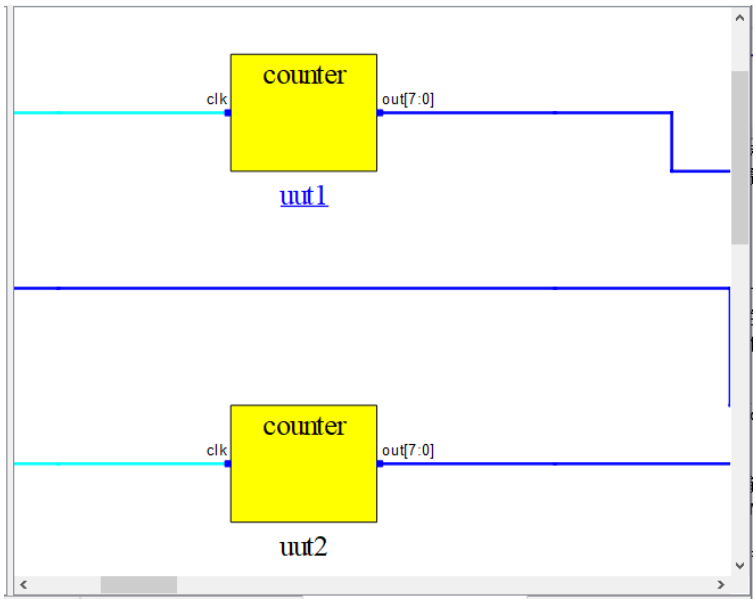
The Module module also supports mouse hover and link to the source file. As shown in Figure 4-44, when your mouse hovers over the module, it can display the instance name and definition.

Figure 4-44 Hover Over a Module



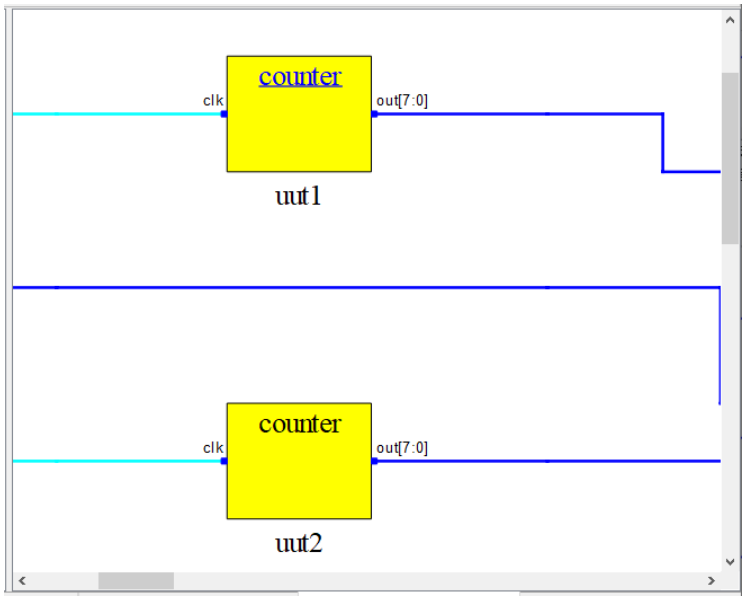
As shown in Figure 4-45, when your mouse hovers over the module instance name, the name is displayed as a link, and you can click it to go to the corresponding instance location in the source file.

Figure 4-45 Hover Over an Instance Name



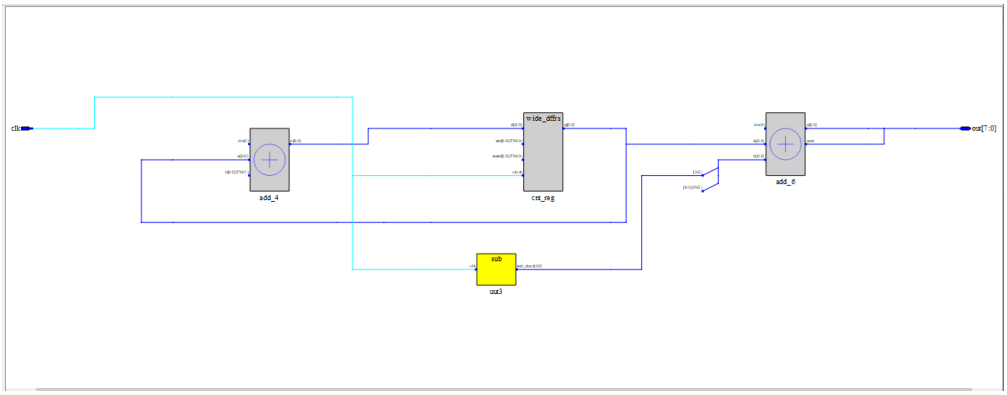
As shown in Figure 4-46, when your mouse hovers over the name, the name is displayed as a link, and you can click it to go to the corresponding module location in the source file.

Figure 4-46 Hover Over a Module Definition



Double-click module schematic to go to the specific logic implementation circuit of this module. As shown in Figure 4-47, double-click module uut2 to go to the specific logic implementation of uut2.

Figure 4-47 Logic Implementation of uut2



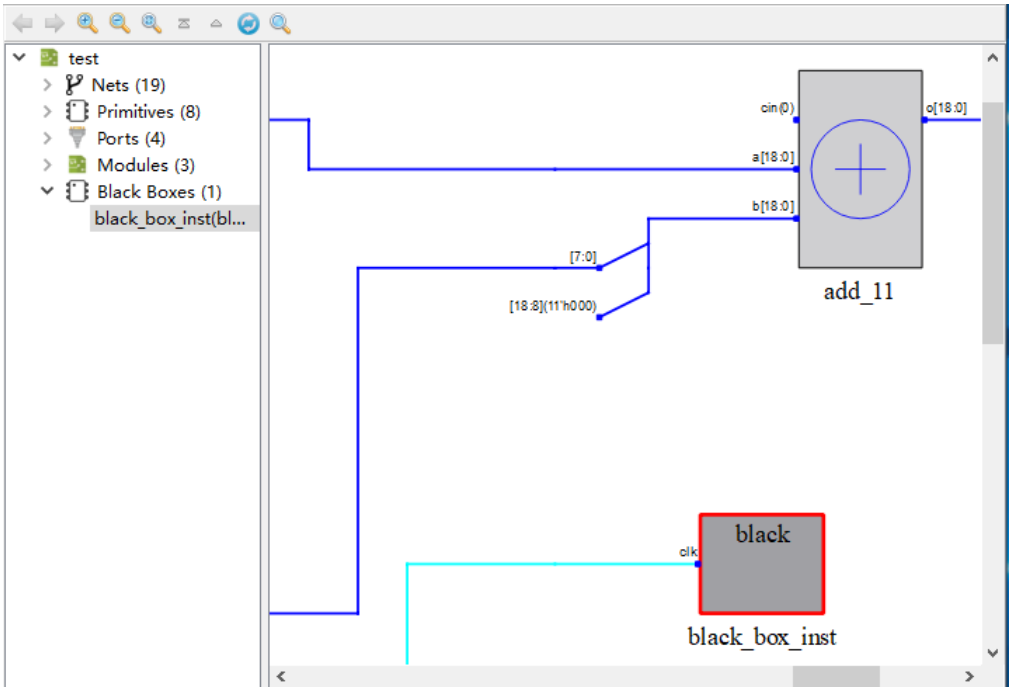
The Module in the schematic supports right-click menu and the options are as follows.

- View Instance In Source: Jumps to the instance of the selected Module in the source file.
- View Module In Source: Jumps to the module definition in the source file.
- Push: Displays the logic circuit of the selected module.
- Pop: Displays the upper level schematic of the current one, which will work only when the current schematic is bottom logic.
- Property: Views the properties of the selected Module.
- Filter: Displays the selected Module separately in the schematic pane.
- UnFilter: Skips back to the old schematic.
- Zoom Fit Selected: Selects a module to zoom in/out the schematic to fit；
- The functions of Select All Ports, Select All Primitives, Select All Modules, Zoom In, Zoom Out, Zoom Fit, Save as PDF File, and Search are same as the those of Net.

Black Box

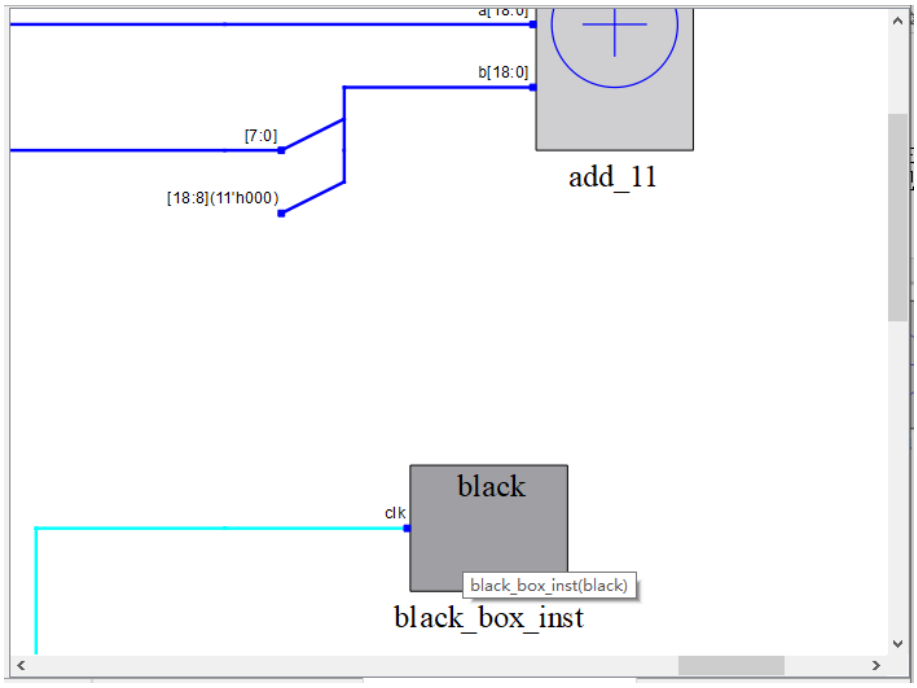
The Black Box drawing in the schematic is with dark gray background. After you select a Black Box schematic, the corresponding Black Box in the hierarchy pane is selected, as shown in Figure 4-48.

Figure 4-48 Select a Black Box



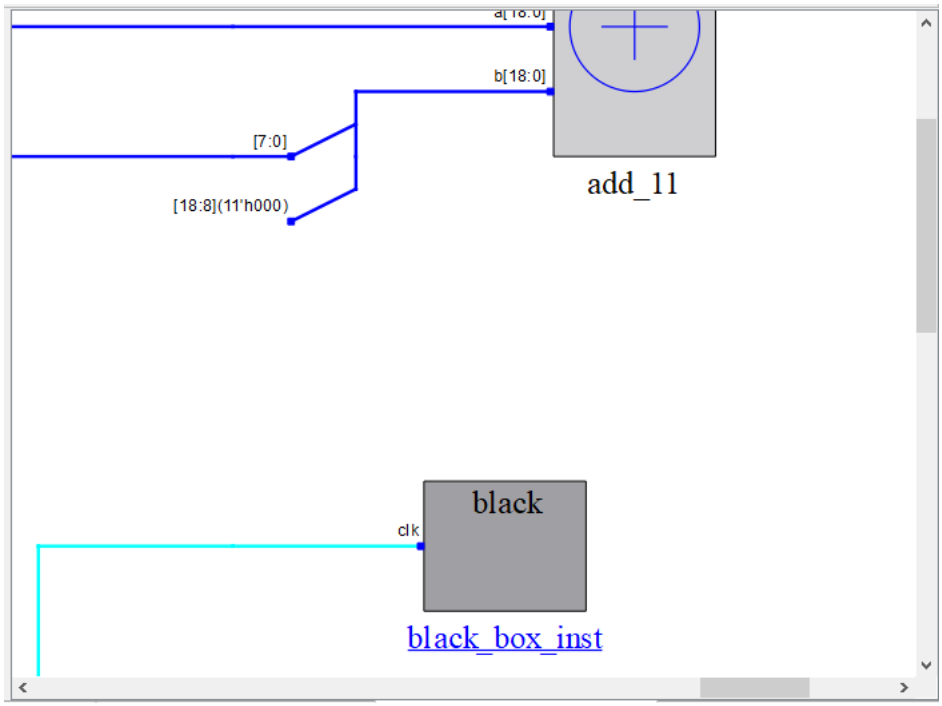
The Black Box also supports mouse hover and link to the source file. As shown in Figure 4-49, when your mouse hovers over the module, it can display the instance name and definition.

Figure 4-49 Hover Over a Black Box



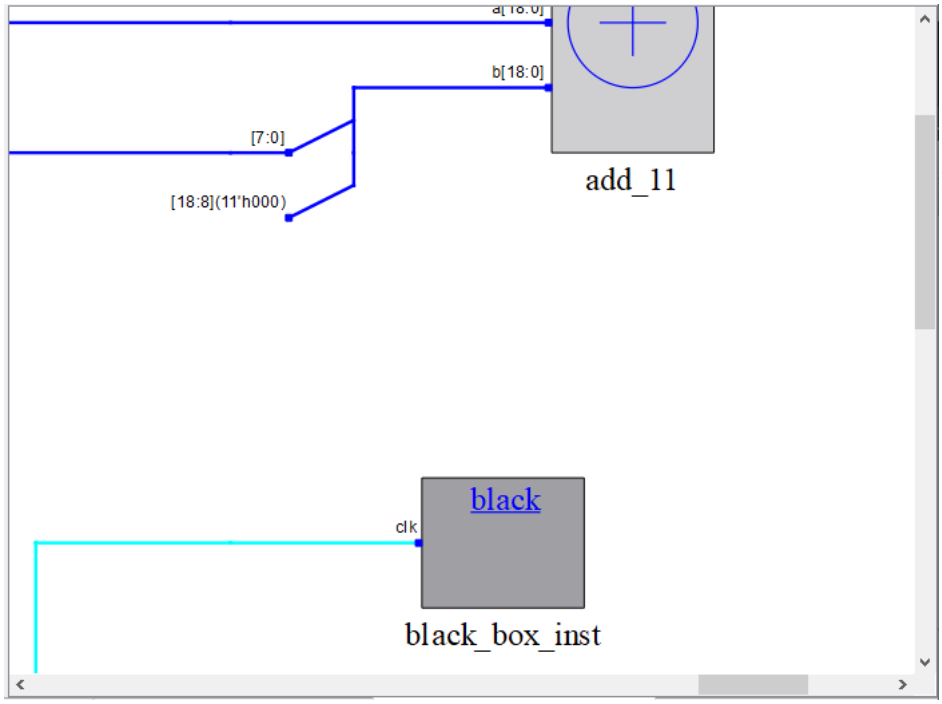
As shown in Figure 4-50, when your mouse hovers over the Black Box instance name, the name is displayed as a link, and you can click it to go to the corresponding instance location in the source file.

Figure 4-50 Hover Over an Instance Name



As shown in Figure 4-51, when your mouse hovers over the name, the name is displayed as a link, and you can click it to go to the corresponding definition in the source file.

Figure 4-51 Hover Over a Black Box Definition



Note!

When the definition corresponding to Black Box is encrypted in the source file, click its definition to go to the file header of the source file.

The Black Box in the schematic supports right-click menu and the

options are as follows.

- View Instance In Source: Jumps to the instance of the selected Black Box in the source file.
- View Module In Source: Jumps to the definition of the Black Box in the source file.
- Property: View the properties of the selected Black Box.
- Filter: Displays the selected Black Box separately in the schematic pane.
- UnFilter: Skips back to the old schematic.
- Zoom Fit Selected: Selects a Black Box to zoom in/out the schematic to fit.
- The functions of Select All Ports, Select All Primitives, Select All Modules, Zoom In, Zoom Out, Zoom Fit, Save as PDF File, and Search are same as the those of Net.

GOWIN
PROGRAMMING FOR THE FUTURE