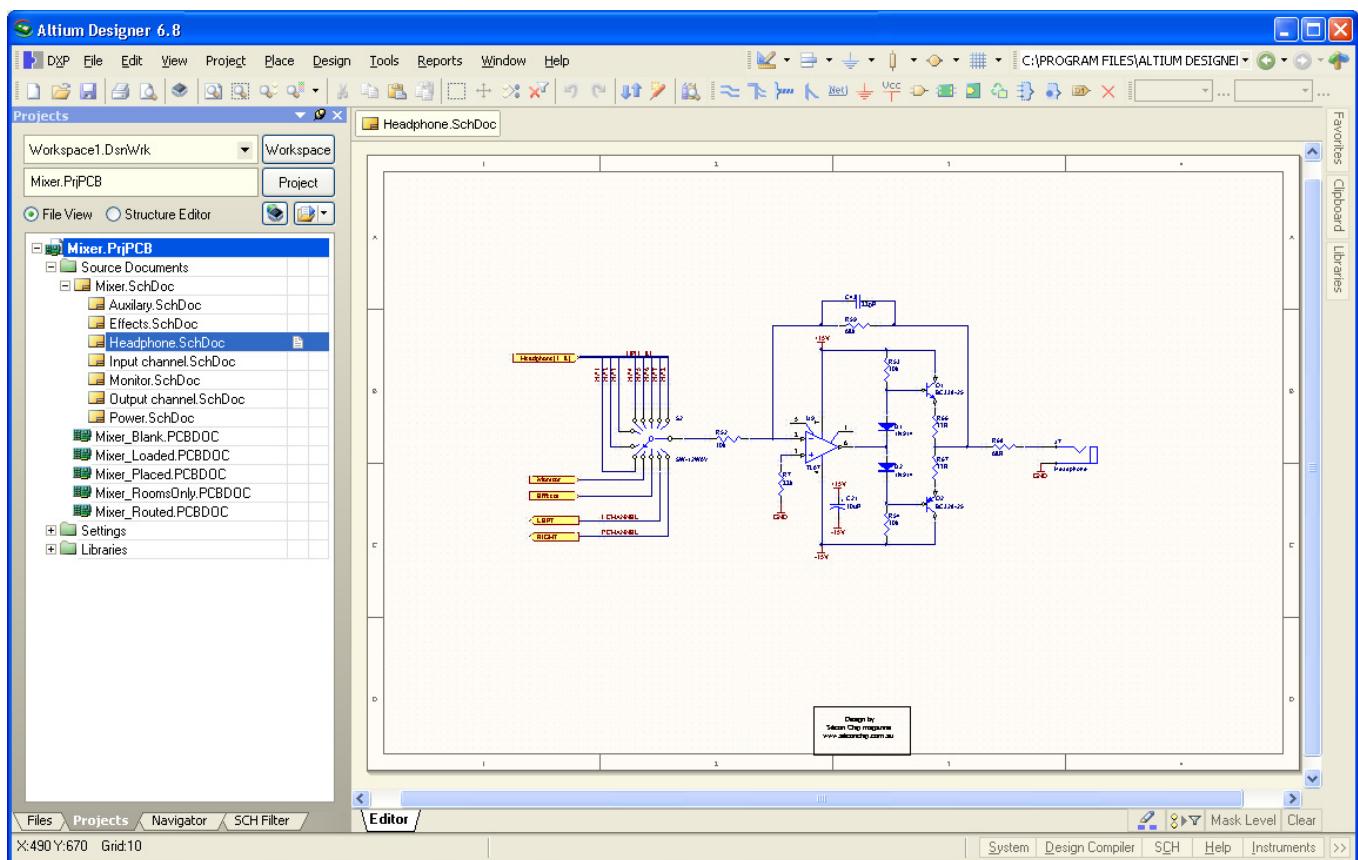


## Summary

This comprehensive reference provides information on the Schematic Editor and the various objects that can be used in order to capture your design.

## Schematic Editor



## Function

The Schematic Editor allows you to create, edit, check and print the schematic sheets that make up a design project. All the tools and utilities needed to perform checks for electrical and drafting violations, generate reports and create presentation quality schematic drawings are available in the editor.

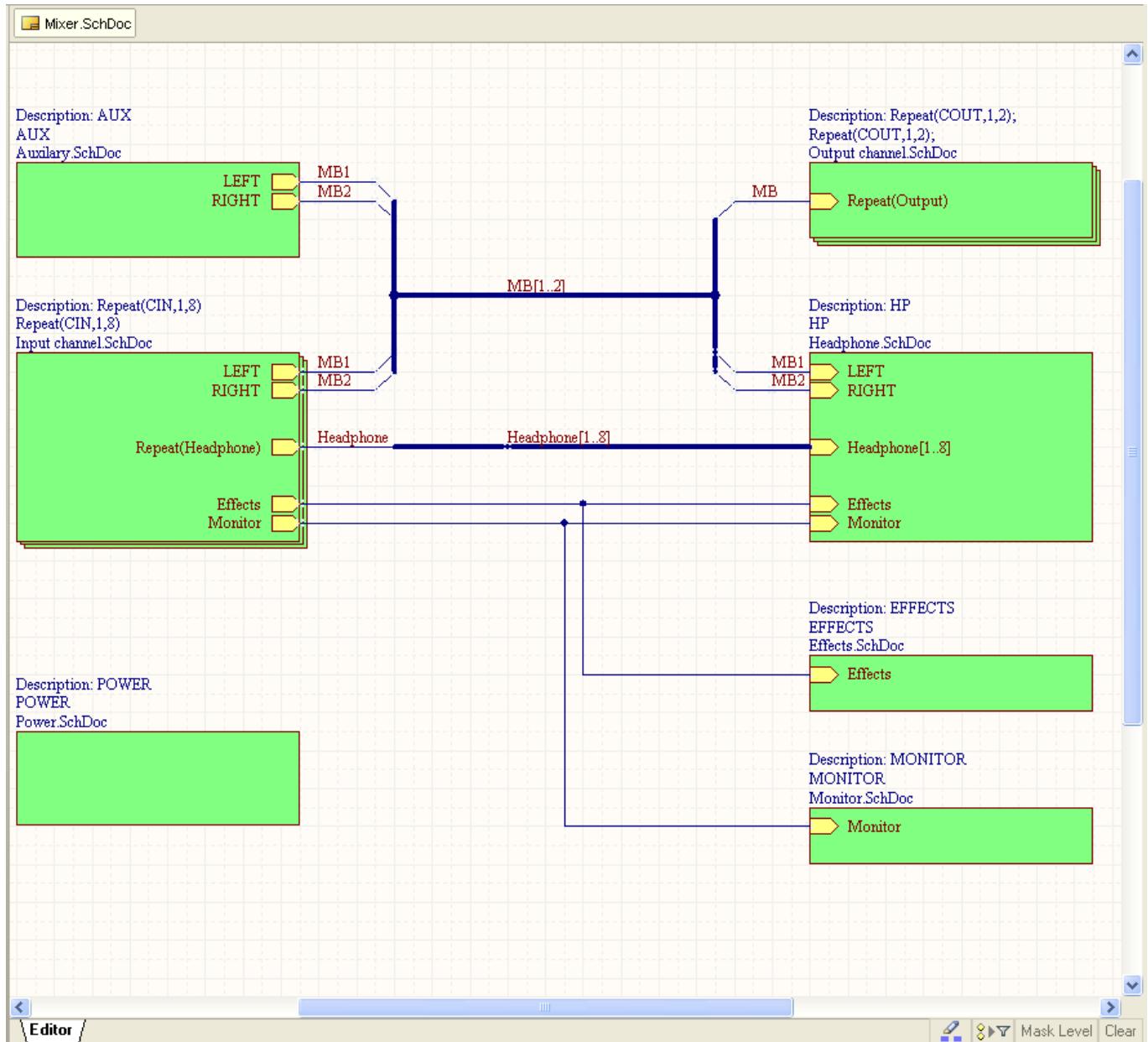
## Editor Environment

When the Schematic Editor is active i.e. a schematic document (\*.SchDoc is open and active) the main application window will contain:

- a main design window in which to capture the design

- editor-specific menus and toolbars
- workspace panels - both global and editor-specific.

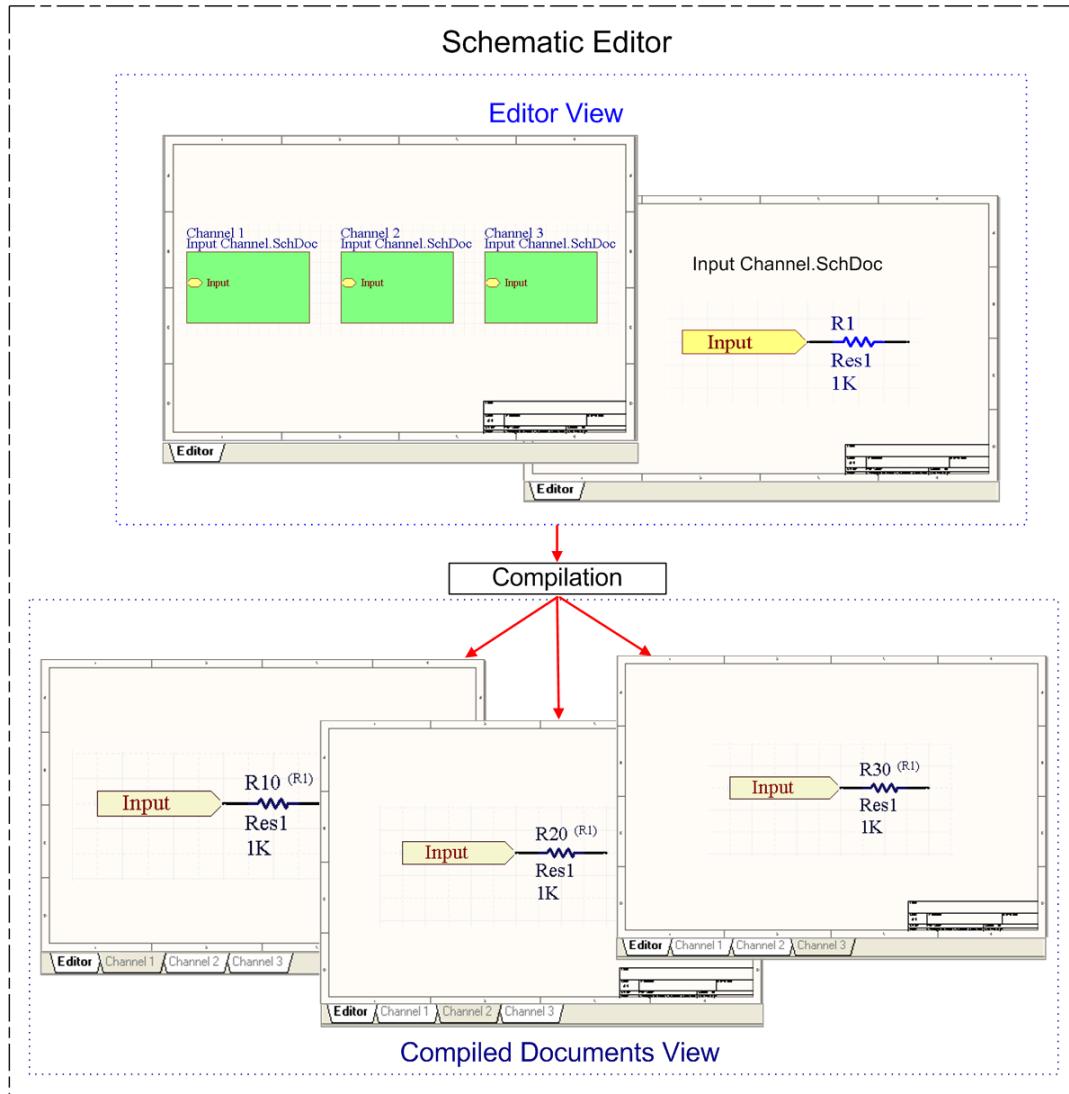
Object placement, wiring and graphical editing are carried out on one or more schematic sheets, each of which appears, when opened, as a tabbed document view in the editor's main design window.



The use of the main design window in terms of actual design (placement, editing, compilation, etc) is outside the scope of this topic and information for such should be sought in the relevant documentation. The following sections, however, offer useful hints and tips with respect to working on schematic documents in the main design window in general.

### The Editor View and Compiled Documents View

The Schematic Editor has a main design window where all of your designs are drawn and viewed. You will notice each Schematic Document in your project has an **Editor** tab. The design constructed in the **Editor** tab is the source for compilation which produces **Compiled Documents** that can be viewed using Compiled Document tabs. Editor and Compiled Document tabs are located along the bottom of the Schematic Document in the design window.



For a multi-channel design, there is one Compiled Document per Channel. If there are no channels in your design, there is only one Compiled Document per Schematic Document.

Compiled Document tabs are named using the Room Name which is based on the Sheet Symbol Designator. The top sheet in a hierarchical design does not have a Sheet Symbol Designator so the Compiled Document tab Name is the same as the filename. For example, *Mixer.SchDoc*, has the Compiled Document tab, *Mixer*. You can customize your Compiled tab Names further in your *Project Options* dialog (**Project** » **Options** » **Multi-Channel**) by changing your **Room Naming Style**.

### Compiled Documents View

The Editor tab is the only tab visible in the Schematic Editor before the project is compiled. When Schematic Documents are edited, the Editor tab becomes the active tab. After you have compiled your project, Compiled Documents can be viewed by clicking on the Compiled Document tabs. If you attempt to make any changes to your design in a Compiled Document, editing mode is activated and the Editor tab becomes active.

Compiled Document tabs are distinguished from the Editor tab by both their naming convention and the font properties of the tab names. The Editor tab name is black and bold whereas the Compiled Document tab names are not bold and are gray.



Note: The Editor tab is the only tab you will see when you first open a project. You have to compile your project to view Compiled Documents.

### Compiled Document Preferences – Gray Scale

Compiled Documents are displayed with a different color scheme from the source document in the Editor tab. They are grayed out by default which helps to identify when you are viewing Compiled Documents which are read only.



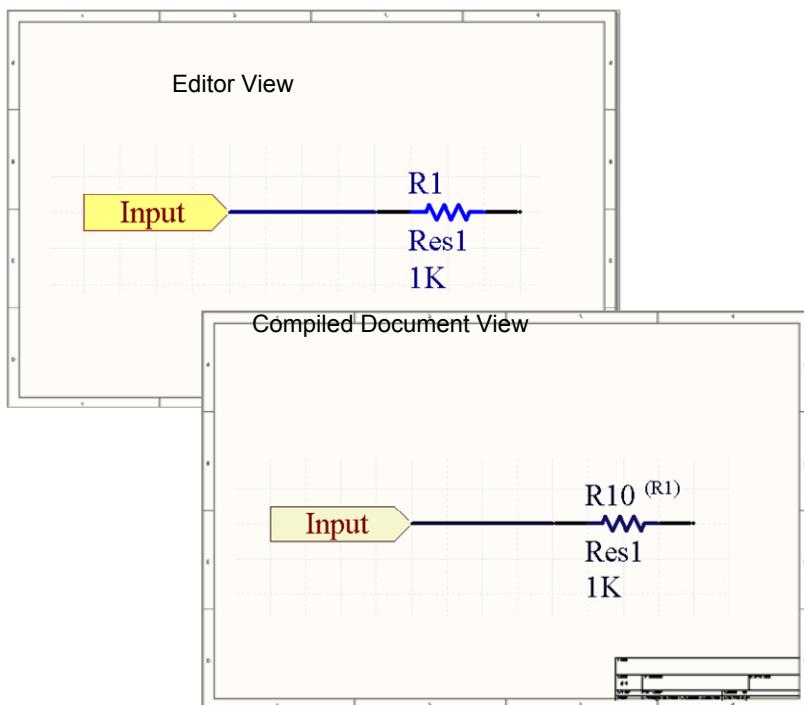
By default, the Compiled Documents View is set at 70% Gray Scale.

You can customize the Gray Scale of your design in the Compiled Documents by:

- Selecting the **DXP » Preferences** command which brings up the *Preferences* dialog
- Navigating to the **Compiler** tab under the **Schematic** folder
- In the **Compiled Names Expansion** section of the dialog, use the **Full Color to Gray Scale** slider bar to specify the extent of color in your Compiled Documents.

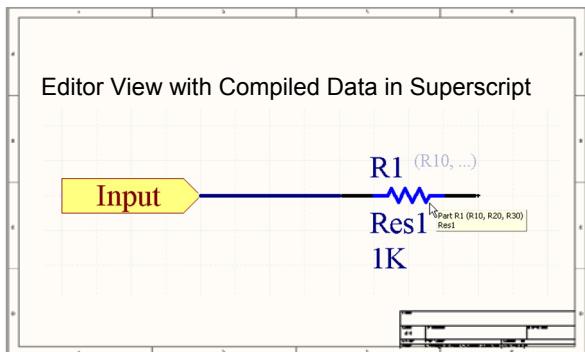
#### Compiled Document Preferences – Compiled Names Expansion

Compiled Documents display the compiled data with the corresponding source data displayed in brackets using superscript.



By default, expanded names for Designators have the option, **Display superscript if necessary** selected. This means that expanded names are only displayed for Designators in the Compiled Documents if they are different from the source in the Editor view and vice versa.

Upon compilation, the corresponding compiled data of your objects is also displayed in brackets using superscript in the Editor tab. If there are multiple compiled names for an object, the first name is displayed followed by 3 ellipses (...). You can view the list of Compiled Names by hovering your mouse over the object.



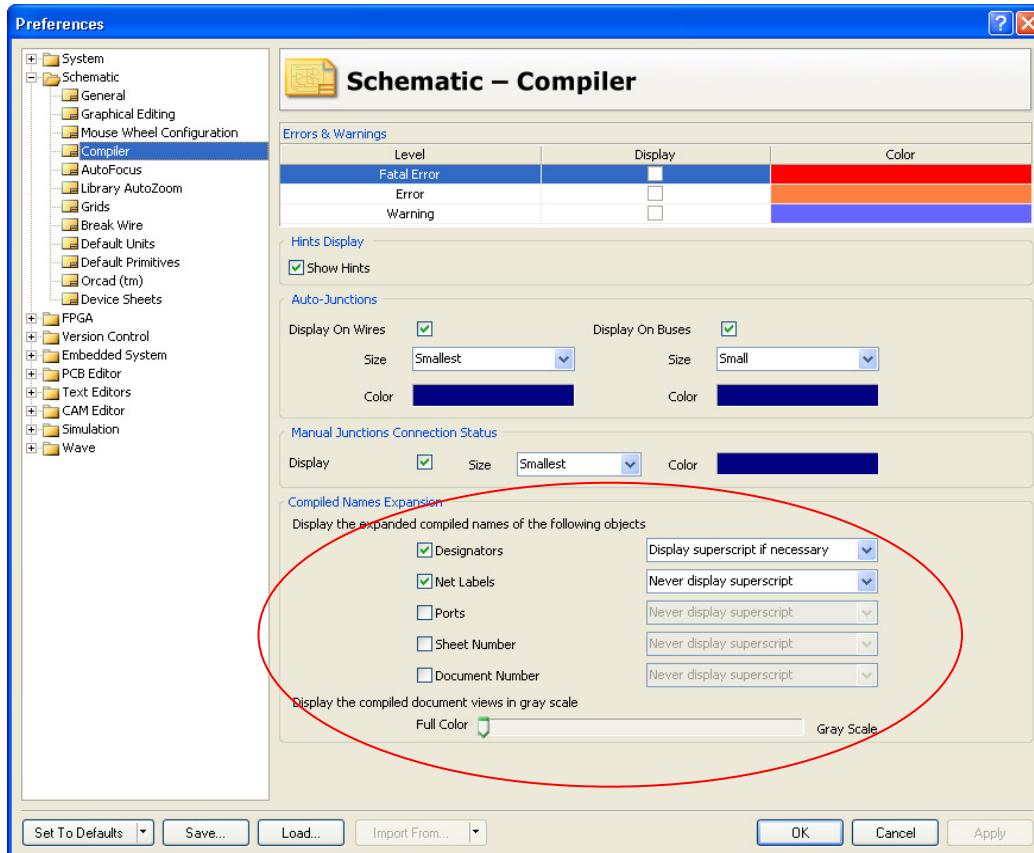
You can customize how the source data is displayed in your Compiled and Source Documents by:

- Selecting the **DXP » Preferences** command which brings up the *Preferences* dialog
- Navigating to the **Compiler** tab under the **Schematic** folder

In the **Compiled Names Expansion** section of the dialog, choose which names you would like to be displayed in superscript. These options are applicable to both the Editor tab and your Compiled Document tabs.

There are three options to choose from for each object:

- **Never display superscript** – expanded names are never displayed
- **Always display superscript** – expanded names are always displayed
- **Display superscript if necessary** – expanded names are only displayed if they are different from the source



### Searching for Compiled Names

You can use the **Edit » Find Text** command to search for Compiled Names in your project.

For example, using the circuitry above, R1 is the designator in the Editor View and R10, R20 and R30 are the compiled names. You can search for R10, R20 and R30 providing you have compiled your project. You do not have to have the compiled document active to search for compiled names.

## Printing Compiled Documents

Compiled Documents cannot be printed from the Schematic Editor. If you attempt to print Compiled Documents, the source document in the Editor tab will be printed.

To print Compiled Documents, create or use an existing Output Job and either:

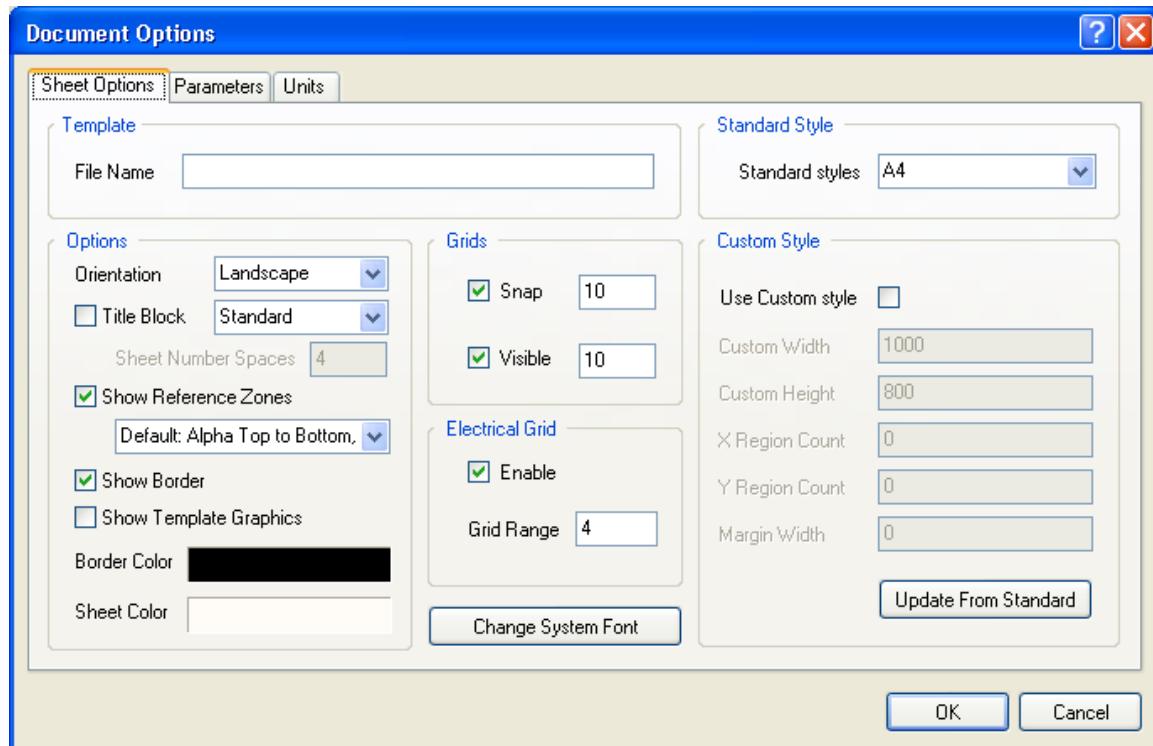
- Select the data source: [Project Physical Documents] for your Schematic Prints outputs, indicating that you want to print Compiled Documents, then right-click and select **Print** from the pop-up menu that appears or use the **File » Print** command
- Use the [Publish to PDF](#) feature to create custom PDF documents which can include both the logical design in the Editor tab and the physical design in your Compiled Documents for all or some of your Schematic Documents.

 For more detailed information on the OutputJob Editor, refer to the [OutputJob Editor Reference](#) document.

## Specifying Document Options

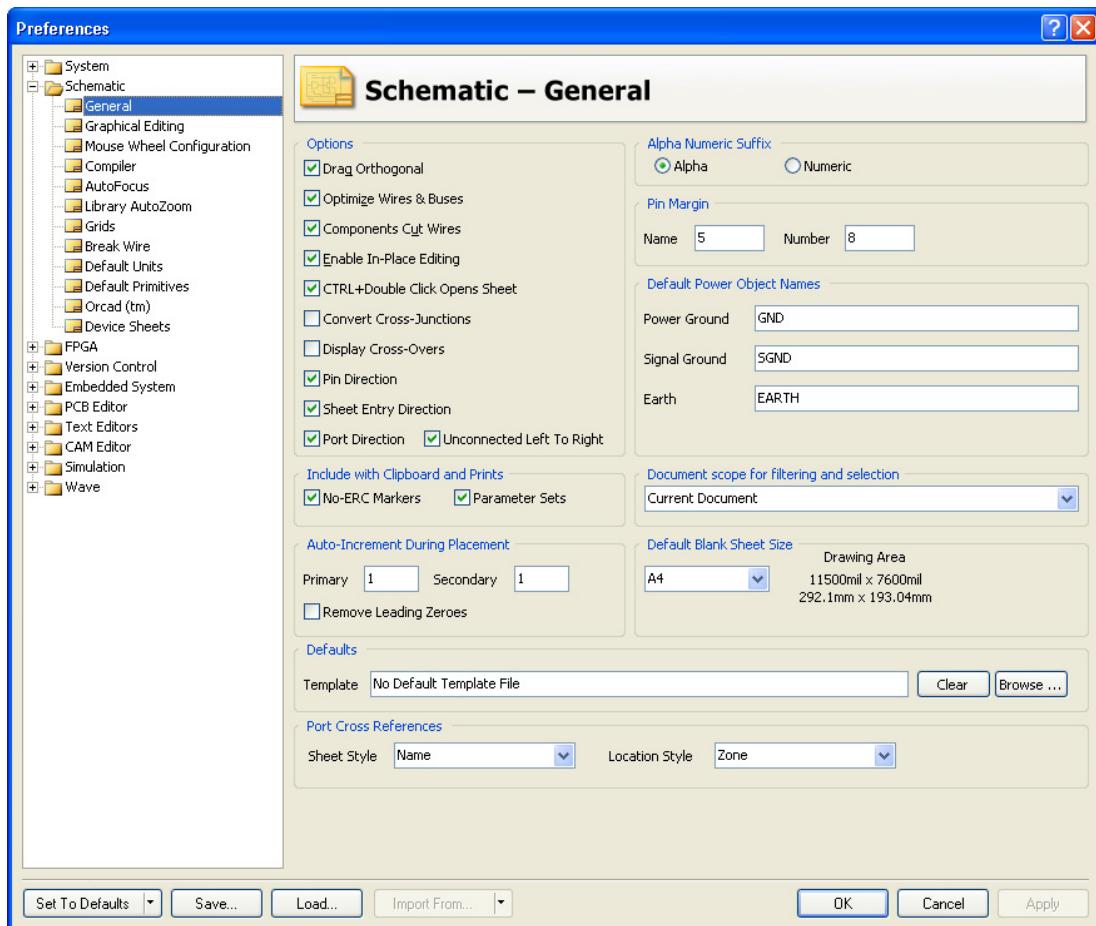
Options specific to the active schematic document are defined in the *Document Options* dialog, which can be accessed by choosing **Design » Document Options** from the main menus.

This dialog provides controls for defining the look and feel of the schematic sheet, enabling and sizing grids, specifying the units of measurement to be used and any relevant document parameters. Use the dialog's 'What's This Help' feature to obtain detailed information about each of the options available. Click on the question mark button at the top right of the dialog and then click over a field or option to pop-up information specific to that field or option.



## Specifying Workspace Preferences

General workspace preferences - applicable to all schematic documents - are defined on the relevant pages contained within the **Schematic** section of the *Preferences* dialog. Choosing **Tools » Schematic Preferences** from the main menus will take you to the **Schematic - General** page of this dialog. Again, use the dialog's 'What's This Help' feature to obtain detailed information about each of the options available across the various pages.



## Right-Click Menus

Right-clicking in the main design window will pop-up a menu providing commands to access commonly used features such as document options and workspace preferences, as well as commands that are in context with the object currently under the cursor, (such as **Properties** and **Find Similar Objects**).

## Panning

Panning in the workspace can be carried out in the following ways:

- using the horizontal and vertical toolbars
- using the keyboard arrow keys (holding **Shift** key for faster movement)
- using the mouse wheel (**Roll Up** - pan up; **Roll Down** - pan down; **Shift+Roll Up** - pan left; **Shift+Roll Down** - pan right)
- right-click and hold to access the panning hand.

## Zooming

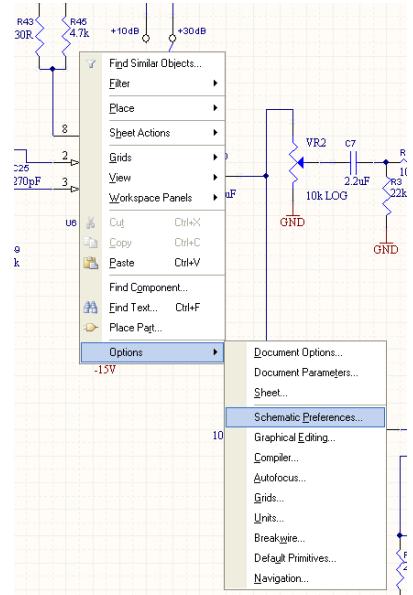
Zooming in the workspace can be achieved in the following ways:

- using the **Page Up** (zoom in) and **Page Down** (zoom out) keyboard shortcuts.
- using the mouse wheel (**Ctrl+Roll Up** - zoom in; **Ctrl+Roll Down** - zoom out).
- using the mouse wheel (push the wheel button down and move mouse up to zoom in, move mouse down to zoom out).

## Moving a group of selected objects

You can move selected objects using a combination of the **ctrl** key and arrow keys (vertically or horizontally) or the **ctrl** and **shift** keys and arrow keys on the schematic document.

The movement of selected objects are set according to the current **Snap Grid** setting in the *Document Options* dialog (**Document** » **Options** or short cut **D,O**). Use this dialog to change the Snap Grid Value. This Grid value also appears on the



Status bar of Altium Designer. The **Schematic – Grids** page of the *Preferences* dialog (**Tools » Schematic Preferences** or shortcut **T, P**) can also be used to set Imperial and Metric grid presets. Use the **G** shortcut to cycle through different snap grid setting values. You can also use the **View » Grids** submenu or the **Grids** right-click menu.

- Selected objects can be 'nudged' by small amounts (according to the current snap grid value) by pressing the arrow keys while holding down the ctrl key.
- Selected objects can also be 'nudged' by large amounts (snap grid value by a factor of 10) by pressing the arrow keys while holding down the ctrl and shift keys together.

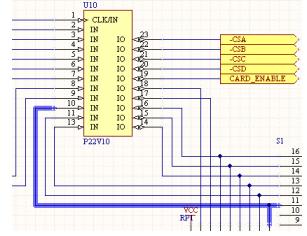
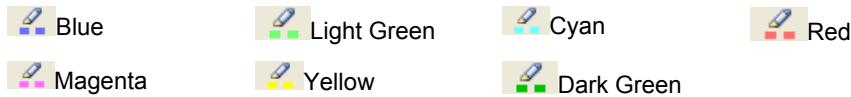
## Jumping to Harness Definition Files

Click the  button at the bottom right of the main design window to jump to the Harness Definition File for the active Schematic Document. Notice that this icon flashes red when you make any changes to your Signal Harness, indicating that processing is occurring and your Harness Definition Files are being updated.

 For information on the harness objects and their harness definition files, please refer to the article [Using Signal Harnesses](#).

## Highlighting Pens

Click on the  button at the bottom right of the main design window to access the highlighting pen feature. This feature allows you to highlight connections and/or entire nets within the design. Pressing the **Spacebar** while the feature is active will change the color of the pen. The following colors are available:

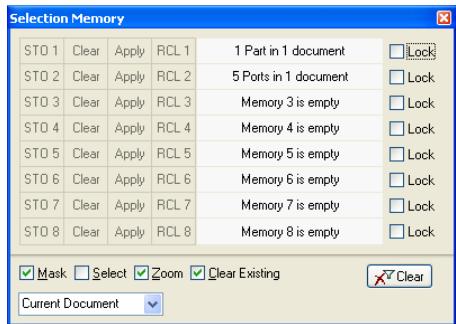


Pressing **Ctrl** and clicking on a port or sheet entry while the feature is active will highlight the connection/net on the target schematic sheet.

To clear all highlighting on the active sheet, click on the **Clear** button at the bottom right of the main design window.

## Selection Memory

Click on the  button at the bottom right of the main design window to access the *Selection Memory* dialog, from where you can control all aspects of the selection memory feature.



 For information on the dialog's use, press F1 when the cursor is over the dialog.

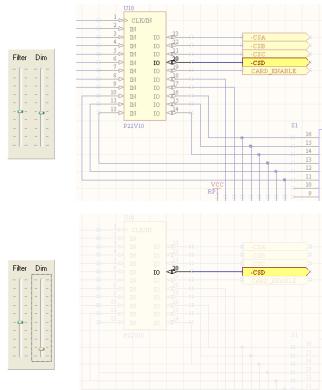
## Mask Level Controls

Click on the **Mask Level** button at the bottom right of the design window to access a pop-up containing controls for adjusting the masking level when the mask highlight method is employed as part of temporary or permanent filtering.

The **Filter** slider bar controls the 'dimming' when masking is applied using a permanent filter - e.g. when applying a query from the **SCH Filter** panel.

The **Dim** slider bar controls the 'dimming' when masking is applied using a temporary filter - e.g. when browsing design objects on a schematic sheet using the **Navigator** panel or Interactive Navigation feature.

In both cases, moving a slider downwards will result in a greater level of masking - with all design objects not falling under the scope of the applied filter becoming more dimmed in the workspace.



## Clear Filtering

Click on the **Clear** button at the bottom right of the main design window in order to clear any existing filtering applied to the current schematic document. If the filtering is temporary in nature, you can click anywhere inside the main design window in order to clear the filtering. If the applied filtering is permanent in nature, you must use this button, or one of its counterparts which can be found in the respective dialog(s) from which the original filtering was initiated. Using this button will also remove any highlighting applied using the Highlighting Pen feature.

## Associated Panels

The following workspace panels are specific to the Schematic Editor:

- **SCH Filter** panel
- **SCH Inspector** panel
- **SCH List** panel
- **Sheet** panel

Certain workspace panels, although not specific to the Schematic Editor, will be used frequently when capturing your design. These include the **Projects** panel, **Navigator** panel and **Messages** panel.

 For more information on a specific panel, press **F1** when the cursor is over that panel. For a complete listing of all workspace panels, refer to the [Altium Designer Panels Reference](#).

## Associated Design Objects

The following is a list of the various objects available for capturing your design. Pressing **F1** over a design object in the main design window will access information for that object directly.

<i>Arc</i>	<i>Bezier</i>	<i>Bus</i>	<i>Bus Entry</i>
<i>C Code Entry</i>	<i>C Code Symbol</i>	<i>Comment</i>	<i>Compile Mask</i>
<i>Compiler Generated Junction</i>	<i>Designator</i>	<i>Device Sheet Symbol</i>	<i>Ellipse</i>
<i>Elliptical Arc</i>	<i>Graphic</i>	<i>Harness Connector</i>	<i>Harness Entry</i>
<i>IEEE Symbols</i>	<i>Instrument Probe</i>	<i>Line</i>	<i>Manual Junction</i>
<i>Net Label</i>	<i>No ERC</i>	<i>Note</i>	<i>Off Sheet Connector</i>
<i>Parameter</i>	<i>Parameter Set</i>	<i>Part</i>	<i>Pie Chart</i>
<i>Pin</i>	<i>Polygon</i>	<i>Port</i>	<i>Power Port</i>
<i>Probe</i>	<i>Rectangle</i>	<i>Round Rectangle</i>	<i>Sheet Entry</i>
<i>Sheet Symbol</i>	<i>Sheet Symbol Designator</i>	<i>Sheet Symbol Filename</i>	<i>Signal Harness</i>
<i>Text Frame</i>	<i>Text String</i>	<i>Wire</i>	

## Re-entrant Editing

The Schematic Editor includes a powerful feature which allows you to perform a second operation without having to quit from the operation you are currently carrying out. This facility is known as re-entrant editing.

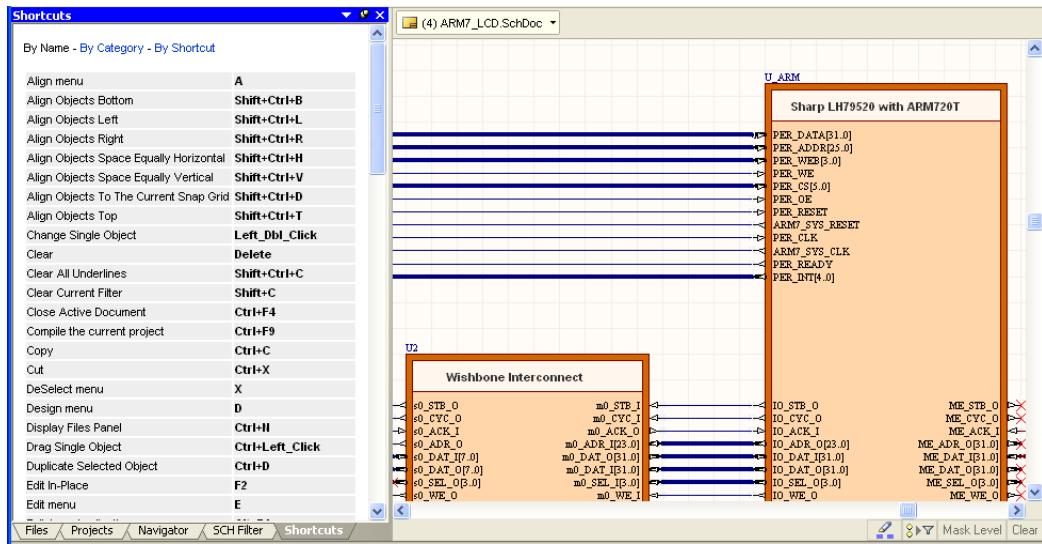
Re-entrant editing allows you to work more flexibly and intuitively. For example, you start placing a wire then remember that it needs to be connected to a port. There is no need to drop out of Place Wire mode, press the Place Port shortcut keys (**P, R**), place the port, press **Esc** to drop out of the Place Port process and then connect the wire to the port.

The second operation can only be accessed by using its shortcut keys.

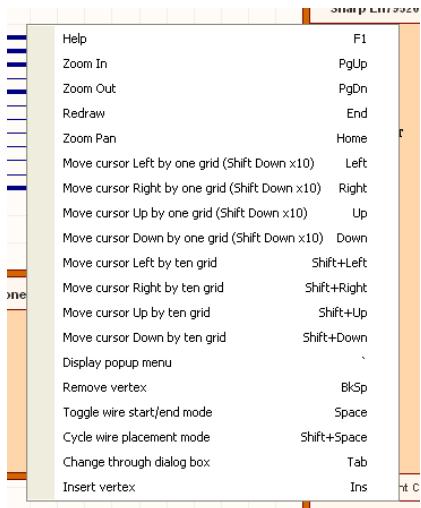
A large number of processes can be completed within another process. The number of times another process can be launched before the current process is complete depends on the demands each of these incomplete processes is placing on the software.

## Access to Shortcuts

While capturing a design, knowledge of the various shortcuts to commonly used commands can be priceless from a productivity perspective. By keeping the **Shortcuts** panel visible, you can quickly peruse the shortcuts available to you, depending on what you are working on. For example, having a schematic document open will show the various Schematic Editor shortcuts.



When an interactive command is running, for example placing a wire, you can access an additional pop-up menu listing all valid shortcuts for that stage of the interactive command. To do this, press the tilde key (~).



Use the menu to refresh your memory about the shortcuts available, or use it in the traditional menu sense, to select the required option with the mouse.

Arrangement of panels and toolbars is totally configurable and, once you have set up the working environment to your liking, can be saved using the **View » Desktop Layouts » Save Layout** command.

## Notes

The Schematic Editor can generate single sheet, multiple sheet and fully hierarchical designs of virtually any size, limited only by the available memory and storage capacity of your PC. The editor also supports true multi-channel design - often a key feature in designs destined to live in an FPGA device.

Sheet sizes include standard A-E, Orcad A-E, metric sizes A4-A0, Letter, Legal and Tabloid. You can also create your own custom sheet sizes, sheet borders and title blocks, which can be saved as templates for re-use.

A major feature of the Schematic Editor is its use of connectivity. This is the ability of the Software to recognize the physical links between objects inside the sheet and the ability to associate the logical connections that exist between various sheets in a multi-sheet design. Upon compilation of the source documents in a design project, the connective model of the design is used as the foundation for navigation using the **Navigator** panel.

---

The naming of your Compiled Document tabs is dependent on the **Room Naming Style** specified *Project Options* dialog (**Project » Options » Multi-Channel**). Note that if you perform a Board Level Annotation on your project, any Room Naming Styles specified here will take precedence over your Project Options and the Compiled Document tabs will be renamed accordingly. For more information about Board Level Annotation, refer to the Application Note, [Understanding Design Annotation](#).

## Arc



### Description

An arc is a non-electrical drawing primitive. It is essentially a curved line segment which can be used when, for example, creating graphical symbols, custom sheet borders and title blocks.

### Availability

Arcs are available for placement in both Schematic and Schematic Library Editors:

#### Schematic Editor

Choose **Place » Drawing Tools » Arc [P, D, A]** from the Schematic Editor main menus.

#### Schematic Library Editor

Choose **Place » Arc [P, A]** from the Schematic Library Editor main menus.

### Placement

After launching the command, the cursor will change to a cross-hair and you will enter arc placement mode. Placement is made by performing the following sequence of actions:

- click or press **Enter** to anchor the center point of the arc
- move the cursor to adjust the radius of the arc, then click or press **Enter** to set it
- move the cursor to adjust the start point for the arc, then click or press **Enter** to anchor it.
- move the cursor to change the position of the arc's end point, then click or press **Enter** to anchor it and complete placement of the arc.

Continue placing further arcs, or right-click or press **Esc** to exit placement mode.

The arc object can be rotated or flipped while in placement mode and before the center point of the arc is anchored:

- Press the **Spacebar** to rotate the arc. Rotation is anti-clockwise and in steps of 90°.
- Press the **X** or **Y** keys to flip the arc along the X-axis or Y-axis respectively.

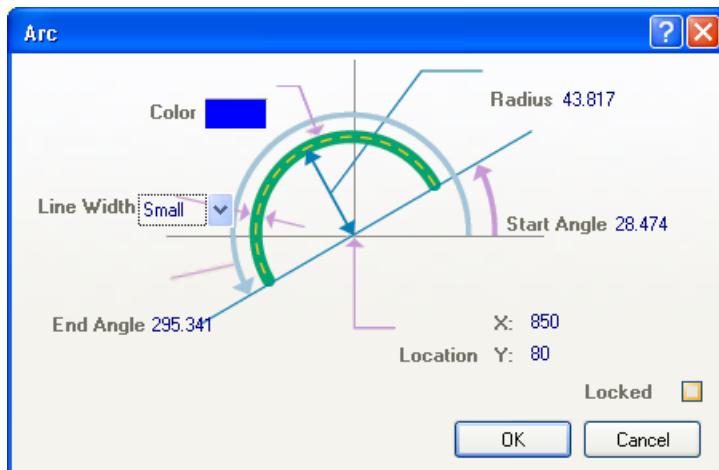
### Editing

The properties of an arc object can be modified before, during and after placement. Editing itself falls into two categories - graphical and non-graphical.

The following three methods of non-graphical editing are available:

#### Editing via an associated properties dialog

This method of editing uses the following dialog to modify the properties of an arc object.



Use the dialog's 'What's This Help' feature to obtain detailed information about each of the options available. Click on the question mark button at the top right of the dialog and then click over a field or option to pop-up information specific to that field or option.

The **Arc** dialog can be accessed prior to entering placement mode, from the **Schematic - Default Primitives** page of the **Preferences** dialog (**Tools » Schematic Preferences**). This allows you to change the default properties for the arc object, which will be applied when placing subsequent arcs.

During placement, the **Arc** dialog can be accessed by pressing the **Tab** key.

After placement, the **Arc** dialog can be accessed in one of the following ways:

- double-clicking on the placed arc object
- selecting the arc object and choosing **Properties** from the right-click pop-up menu (Schematic Editor only)
- choosing the **Change** command from the **Edit** menu and then clicking once over the placed arc object.

#### Editing via the SCH Inspector panel

The **SCH Inspector** panel enables you to interrogate and edit the properties of one or more design objects in the active document. Used in conjunction with appropriate filtering, the panel can be used to make changes to multiple objects of the same kind, from one convenient location.

For more information on the **SCH Inspector** panel, press **F1** when the cursor is over the panel.

#### Editing via the SCH List panel

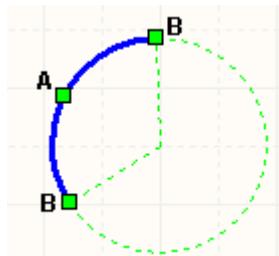
The **SCH List** panel allows you to display design objects from one or more documents in tabular format, enabling you to quickly inspect and modify object attributes. When used in conjunction with the **SCH Filter** panel, it enables you to display just those objects falling under the scope of the active filter - allowing you to target and edit multiple design objects with greater accuracy and efficiency.

For more information on the **SCH List** and **SCH Inspector** panels, press **F1** when the cursor is over a panel.

#### Graphical editing

This method of editing allows you to select a placed arc object directly in the workspace and change its size, shape or location, graphically.

When an arc object is selected, the following editing handles are available:



Click and drag **A** to adjust the radius.

Click and drag **B** to adjust the end points.

Click anywhere on the arc - away from editing handles - and drag to reposition it. The arc can be rotated or flipped while dragging.

If you attempt to graphically modify an arc object that has its **Locked** property enabled, a dialog will appear asking for confirmation to proceed with the edit.

If the **Protect Locked Objects** option is enabled in the **Schematic – Graphical Editing** page of the *Preferences* dialog (**Tools** » **Schematic Preferences**), and the **Locked** option for this design object is enabled as well then this object cannot be selected or graphically edited.

You will have to double click on this locked object directly and disable the **Locked** property or disable the **Protect Locked Objects** option in the **Schematic – Graphical Editing** Preferences dialog to graphically edit this object.

If you attempt to select locked objects along with other objects, only those objects that are unlocked can be selected and moved as a group if the **Protect Locked Objects** option is enabled. Otherwise if the **Protect Locked Objects** option is disabled you will be prompted for confirmation to proceed with the edit of the objects including locked objects.

## Notes

Any changes made to object properties during placement will cause the default properties for the object to be updated, unless the **Permanent** option - on the **Schematic - Default Primitives** page of the *Preferences* dialog - is enabled. When this option is enabled, changes made will affect only the object being placed and subsequent objects placed during the same placement session.

## Bezier



### Description

A Bezier curve is a non-electrical drawing primitive. It is a free-form curved line that can be placed on a schematic sheet. The curve is defined by a series of vertex points that 'pull' the line into a curved shape.

### Availability

Beziers are available for placement in both Schematic and Schematic Library Editors:

#### Schematic Editor

- Choose **Place » Drawing Tools » Bezier [P, D, B]** from the Schematic Editor main menus
- Click the button on the **Utility Tools** drop-down of the **Utilities** toolbar

#### Schematic Library Editor

- Choose **Place » Bezier [P, B]** from the Schematic Library Editor main menus
- Click the button on the **Utility Tools** drop-down of the **Utilities** toolbar

### Placement

After launching the command, the cursor will change to a cross-hair and you will enter Bezier placement mode. Placement is made by performing the following sequence of actions:

- click or press **Enter** to anchor the starting point for the curve
- move the cursor and click or press **Enter** to place a series of vertex points to define the curve. As you move the cursor the curve will be continually redrawn to indicate how it would look if you placed a vertex at the cursor position
- after placing the final vertex point, right-click or press **Esc** to complete placement of the curve.

Continue placing further Bezier curves, or right-click or press **Esc** to exit placement mode.

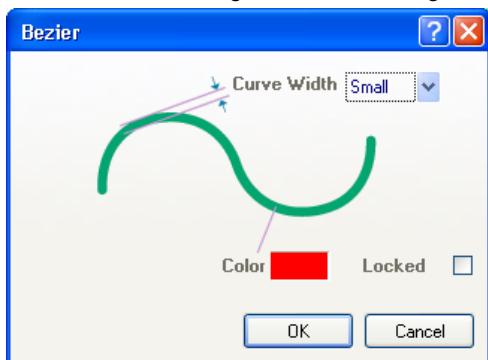
### Editing

The properties of a Bezier object can be modified before, during and after placement. Editing itself falls into two categories - graphical and non-graphical.

The following three methods of non-graphical editing are available:

#### Editing via an associated properties dialog

This method of editing uses the following dialog to modify the properties of a Bezier object.



Use the dialog's 'What's This Help' feature to obtain detailed information about each of the options available. Click on the question mark button at the top right of the dialog and then click over a field or option to pop-up information specific to that field or option.

The **Bezier** dialog can be accessed prior to entering placement mode, from the **Schematic - Default Primitives** page of the **Preferences** dialog (**Tools » Schematic Preferences**). This allows you to change the default properties for the Bezier object, which will be applied when placing subsequent Bezier curves.

During placement, the **Bezier** dialog can be accessed by pressing the **Tab** key.

After placement, the **Bezier** dialog can be accessed in one of the following ways:

- double-clicking on the placed Bezier curve
- selecting the Bezier curve and choosing **Properties** from the right-click pop-up menu (Schematic Editor only)
- choosing the **Change** command from the **Edit** menu and then clicking once over the placed Bezier curve.

#### Editing via the SCH Inspector panel

The **SCH Inspector** panel enables you to interrogate and edit the properties of one or more design objects in the active document. Used in conjunction with appropriate filtering, the panel can be used to make changes to multiple objects of the same kind, from one convenient location.



For more information on the **SCH Inspector** panel, press **F1** when the cursor is over the panel.

#### Editing via the SCH List panel

The **SCH List** panel allows you to display design objects from one or more documents in tabular format, enabling you to quickly inspect and modify object attributes. When used in conjunction with the **SCH Filter** panel, it enables you to display just those objects falling under the scope of the active filter - allowing you to target and edit multiple design objects with greater accuracy and efficiency.

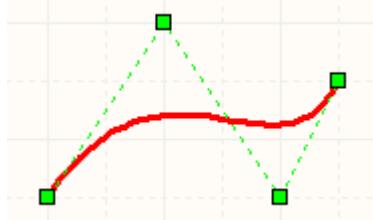


For more information on the **SCH List** and **SCH Inspector** panels, press **F1** when the cursor is over a panel.

#### Graphical editing

This method of editing allows you to select a placed Bezier curve object directly in the workspace and change its size and/or shape, graphically.

When a Bezier curve object is selected, the following editing handles are available:



Click and drag an editing handle to "bend" the curve.

If you attempt to graphically modify a bezier object that has its **Locked** property enabled, a dialog will appear asking for confirmation to proceed with the edit.

If the **Protect Locked Objects** option is enabled in the **Schematic – Graphical Editing** page of the **Preferences** dialog (**Tools » Schematic Preferences**), and the **Locked** option for this design object is enabled as well then this object cannot be selected or graphically edited.

You will have to double click on this locked object directly and disable the **Locked** property or disable the **Protect Locked Objects** option in the **Schematic – Graphical Editing** Preferences dialog to graphically edit this object.

If you attempt to select locked objects along with other objects, only those objects that are unlocked can be selected and moved as a group if the **Protect Locked Objects** option is enabled. Otherwise if the **Protect Locked Objects** option is disabled you will be prompted for confirmation to proceed with the edit of the objects including locked objects.

#### Notes

A Bezier must have at least four vertices to form a curve.

The maximum number of vertices supported by a single Bezier object is 50.

---

Any changes made to object properties during placement will cause the default properties for the object to be updated, unless the **Permanent** option - on the **Schematic - Default Primitives** page of the **Preferences** dialog - is enabled. When this option is

enabled, changes made will affect only the object being placed and subsequent objects placed during the same placement session.

## Bus



### Description

A bus is an electrical design primitive. It is a polyline object that represents a multi-wire connection.

### Availability

Buses are available for placement in the Schematic Editor only. Use one of the following methods to access the placement command:

- choose **Place » Bus [P, B]** from the Schematic Editor main menus
- click the  button on the **Wiring** toolbar.

### Placement

After launching the command, the cursor will change to a cross-hair and you will enter bus placement mode. Placement is made by performing the following sequence of actions:

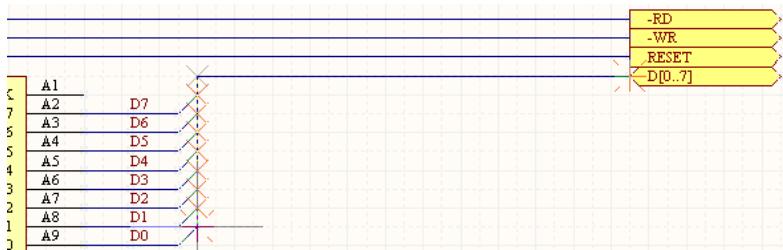
- click or press **Enter** to anchor the starting point for the bus
- position the cursor and click or press **Enter** to anchor a series of vertex points that define the shape of the bus
- after placing the final vertex point, right-click or press **Esc** to complete placement of the bus.

Continue placing further bus objects, or right-click or press **Esc** to exit placement mode.

Use the **Backspace** or **Delete** keys to remove the last bus segment placed. If you do remove segments in this way, you must click to place a final segment, otherwise right-clicking will place the bus as it was, with all deleted segments reinstated.

### Guided wiring

Schematics have a definable electrical grid that makes it easy to define electrical connections between objects. As you are placing a bus, when the bus falls within the electrical grid range of another electrical object the cursor will snap to the fixed object and a Hot Spot (red cross) will appear.



The Hot Spot guides you to where a valid connection can be made and automatically snaps the cursor to electrical connection points.

The electrical grid can be defined on the **Sheet Options** tab of the *Document Options* dialog (**Design » Document Options**). It is recommended that you set the electrical grid to be slightly smaller than the current snap grid, or it becomes difficult to position electrical objects one snap grid apart.

### Auto-junctioning

The schematic auto-junctioning feature places an electrical junction (compiler generated junction) when two buses are connected in a T-type fashion, or when a bus connects orthogonally to a pin or bus power port.



This feature allows you to easily create electrical connections at junction points without the need to manually define the connection (through placement of a manual junction). Buses that cross away from their end points do not have a junction automatically inserted.

Display of auto-junctions on the schematic sheet, with respect to buses, can be controlled from the **Schematic - Compiler** page of the *Preferences* dialog (**Tools » Schematic Preferences**). Additional options provide control over junction size and color.

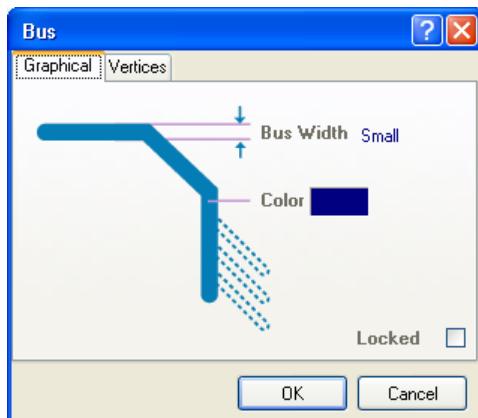
## Editing

The properties of a bus object can be modified before, during and after placement. Editing itself falls into two categories - graphical and non-graphical.

The following three methods of non-graphical editing are available:

### Editing via an associated properties dialog

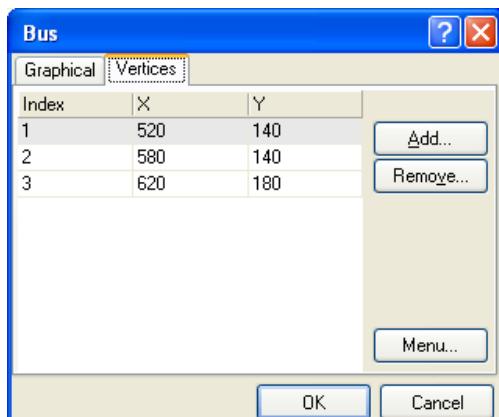
This method of editing uses the following dialog to modify the properties of a bus object.



Use the dialog's 'What's This Help' feature to obtain detailed information about each of the individual options available. Click on the question mark button at the top right of the dialog and then click over a field or option to pop-up information specific to that field or option.

### Editing Vertices

The **Bus** dialog provides a **Vertices** tab, from where you can edit the individual vertices of the currently selected bus object as required.



The main region of the tab lists all of the vertex points currently defined for the bus. You can add new vertices to the bus, edit the coordinates of existing vertices, or remove selected vertices altogether.

Click the **Menu** button or right-click within the main list region to access a pop-up menu containing the following commands:

- **Edit** - right click on a coordinate cell (X or Y) for a vertex and use this command to edit the value in that cell. Alternatively, click directly on the cell

- **Add** - use this command to add a new vertex point. The new vertex will be added below the currently focused vertex entry (as distinguished by a dotted outline around a cell in its row) and will initially have the same coordinates as the focused entry
- **Remove** - use this command to remove the currently selected vertex entries in the list. This command will be unavailable if there are only two vertices present for the bus
- **Copy** - use this command to copy the content of the selected cells in the list to the clipboard (alternatively use **Ctrl+C**)
- **Paste** - use this command to paste the content of the clipboard into the list, starting at the selected cell (alternatively use **Ctrl+V**)
- **Select All** - use this command to quickly select the entire grid contents of the list
- **Select Column** - use this command to quickly select the entire column in which the currently focused cell resides
- **Move Up** - use this command to move the selected vertex upward in the list
- **Move Down** - use this command to move the selected vertex downward in the list
- **Move Bus By XY** - use this command to move the entire bus object. The Move Bus By dialog will appear, from where you can enter the increment value to be applied to each vertex point's X and Y coordinates.

### Dialog access

The **Bus** dialog can be accessed prior to entering placement mode, from the **Schematic - Default Primitives** page of the **Preferences** dialog (**Tools » Schematic Preferences**). This allows you to change the default properties for the bus object, which will be applied when placing subsequent buses.

During placement, the **Bus** dialog can be accessed by pressing the **Tab** key.

After placement, the **Bus** dialog can be accessed in one of the following ways:

- double-clicking on the placed bus object
- selecting the bus object and choosing **Properties** from the right-click pop-up menu
- choosing the **Change** command from the **Edit** menu and then clicking once over the placed bus object.

### Editing via the SCH Inspector panel

The **SCH Inspector** panel enables you to interrogate and edit the properties of one or more design objects in the active document. Used in conjunction with appropriate filtering, the panel can be used to make changes to multiple objects of the same kind, from one convenient location.

 For more information on the **SCH Inspector** panel, press **F1** when the cursor is over the panel.

### Editing via the SCH List panel

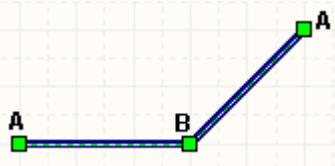
The **SCH List** panel allows you to display design objects from one or more documents in tabular format, enabling you to quickly inspect and modify object attributes. When used in conjunction with the **SCH Filter** panel, it enables you to display just those objects falling under the scope of the active filter - allowing you to target and edit multiple design objects with greater accuracy and efficiency.

 For more information on the **SCH List** and **SCH Inspector** panels, press **F1** when the cursor is over a panel.

### Graphical editing

This method of editing allows you to select a placed bus object directly in the workspace and change its size and/or shape, graphically.

When a bus object is selected, the following editing handles are available:



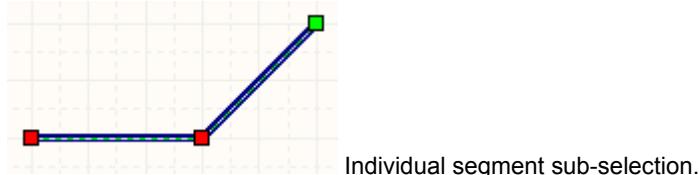
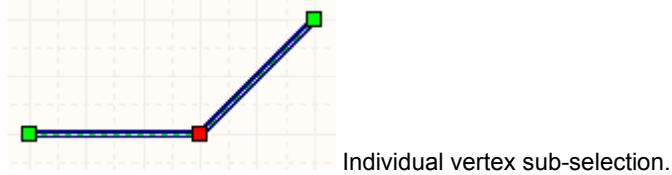
Click and drag **A** to reposition the end points of the bus.

Click and drag **B** to move a bus vertex. The end points will remain anchored.

Click and drag near the center of a bus segment to "grab" that segment and reposition it. The end points and other vertices will remain anchored.

Right-click on a vertex point and choose the **Edit Bus Vertex n** command to access the **Vertices** tab of the *Bus* dialog, with the entry for the **n**th vertex selected ready for editing.

With the bus selected, click on a vertex or segment to individually select that vertex or segment. This bus 'sub-selection' is distinguished by the associated editing handles becoming red in color.



The associated vertex (or vertices for a segment) can then be edited directly using the **SCH Inspector** or **SCH List** panels, with any changes appearing immediately on the schematic.

If you attempt to graphically modify a bus object that has its **Locked** property enabled, a dialog will appear asking for confirmation to proceed with the edit.

If the **Protect Locked Objects** option is enabled in the **Schematic – Graphical Editing** page of the *Preferences* dialog (**Tools** » **Schematic Preferences**), and the **Locked** option for this design object is enabled as well then this object cannot be selected or graphically edited.

You will have to double click on this locked object directly and disable the **Locked** property or disable the **Protect Locked Objects** option in the **Schematic – Graphical Editing** Preferences dialog to graphically edit this object.

If you attempt to select locked objects along with other objects, only those objects that are unlocked can be selected and moved as a group if the **Protect Locked Objects** option is enabled. Otherwise if the **Protect Locked Objects** option is disabled you will be prompted for confirmation to proceed with the edit of the objects including locked objects.

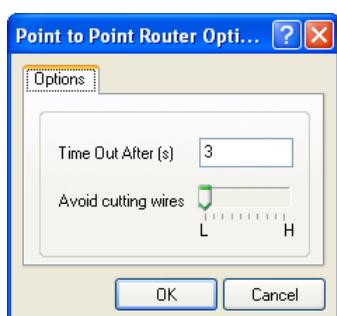
### Notes

When placing a bus, various placement modes are available: 90 Degree, 45 Degree, Any Angle and Auto Wire. The mode specifies how corners are created when placing buses and the angles at which buses can be placed. The 90 Degree and 45 Degree modes (true orthogonal modes) both have Start and End sub-modes.

If you attempt to graphically modify a bus object that has its **Locked** property enabled, a dialog will appear asking for confirmation to proceed with the edit.

---

The Auto Wire mode is a special mode that allows you to automatically connect between two points on the schematic. The Autowirer will automatically route the bus around obstacles. When in this mode, press the **Tab** key to set the Autowirer options in the subsequent *Point to Point Router Options* dialog:




---

When placing a bus, press **Shift+Spacebar** to cycle through the bus placement modes. Press **Spacebar** to toggle between the Start and End sub-modes (when in 90 Degree or 45 Degree modes), or between Any Angle and Auto Wire modes (when either of these modes is active).

The current placement mode is displayed in the status bar. You can change modes at any time during bus placement.

In all modes other than Any Angle, the line segment attached to the cursor is a "look ahead" segment. The segment you are actually placing precedes this look ahead segment.

Use a Bus Entry object to connect to, or branch from, a bus.

Buses can be attached to ports or sheet symbols for connection to other schematic sheets.

Assign a bus to multiple nets using a Net Label of the form **D[0..7]** or **D[7..0]**, indicating that the individual (and distinct) nets D0 to D7 are carried by the bus. Whichever form you use, the ordering must be consistent between connected bus objects.

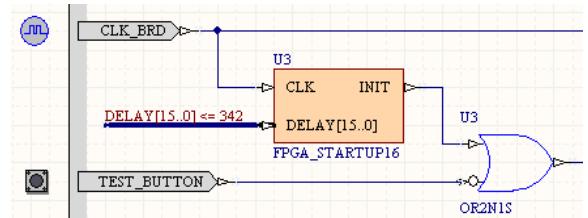
Assigning a net label with this syntax (bus syntax) to a bus changes the bus object from being purely graphical to logical.

Remember that net identifiers of different types do not automatically connect to one another, even if they share the same name. This holds true for net identifiers with bus syntax; a net label D[0..7] will not automatically connect to a port with the same name. The bus is required to connect them together.

It is recommended that you define the net label in bus syntax to contain alpha characters only. For example, if you named the bus D2[0..7], it would be expanded to D20, D21..D27 which can cause net name conflicts.

In FPGA designs, define a constant value for a bus by specifying the required value in the net label for the bus. The constant can be declared in either decimal, binary or hexadecimal format. For example, consider an **FPGA\_STARTUP16** device, with a required **DELAY** input of 342 (decimal). The following entries for the bus net label could be used to define the constant:

- **Decimal:** `DELAY[15..0] <= 342`
- **Binary:** `DELAY[15..0] <= b101010110`
- **Hexadecimal:** `DELAY[15..0] <= $156`



In a binary definition, the leading zeroes need not be declared. You can of course declare all bits and, for added readability, separate nibbles by a space or a hyphen. Therefore the decimal value 342 could be defined in binary as:

0000 0001 0101 0110 or

0000-0001-0101-0110

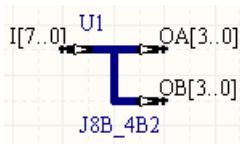
Leading zeroes need not be declared for a hexadecimal value either. Also, you can use \$ or 0x prefixes to denote the value as being hexadecimal.

### Using Bus Joiners

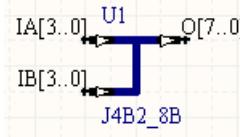
In FPGA designs, to manage the mapping of nets in buses there is a special class of component, known as a bus joiner. Bus joiners can be placed from the **FPGA Generic** integrated library (**FPGA\_Generic.IntLib** file from the **\Library\Fpga\** folder of the Altium Designer installation.). By using bus joiner components, you can split, merge and reorder buses in a design with relative ease and efficiency.

### Splitting/Merging Buses

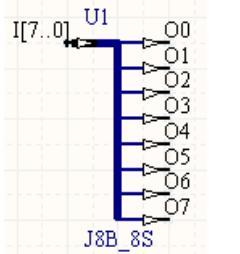
Bus joiners separate/merge the bits (or bus slice) from least significant bit (or slice) down to most significant bit (or slice). Bus joiners are available in a range of sizes, catering for bus to pin, pin to bus and bus to bus splitting and merging. The following examples demonstrate this splitting and merging behavior, respectively.



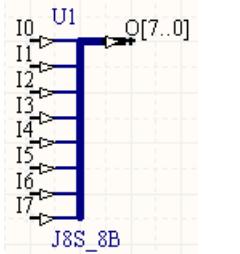
- this particular bus joiner component splits the incoming 8-bit bus on pin  $I[7..0]$  into two 4-bit bus slices,  $OA[3..0]$  and  $OB[3..0]$ . Obeying the least to most mapping at the slice level, the lower four bits of the input bus map to  $OA[3..0]$  and the upper four bits map to  $OB[3..0]$ . Following this through to the bit level,  $I0$  will connect to  $OA0$ , and  $I7$  will connect to  $OB3$



- this particular bus joiner component merges the two incoming 4-bit bus slices on pins  $IA[3..0]$  and  $IB[3..0]$  into a single 8-bit bus  $O[7..0]$ . Obeying the least to most mapping at the slice level,  $IA[3..0]$  maps to the lower four bits of the output and  $IB[3..0]$  maps to the upper four bits of the output. Following this through to the bit level,  $IA0$  will connect to  $O0$ , and  $IB3$  will connect to  $O7$

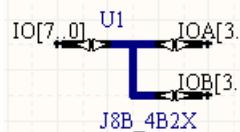


- this particular bus joiner component splits the incoming 8-bit bus on pin  $I[7..0]$  into eight single pin outputs,  $O0$  to  $O7$ . Obeying the least to most mapping at the bit level,  $I0$  will connect to  $O0$ , and  $I7$  will connect to  $O7$



- this particular bus joiner component merges the eight single pin inputs on pins  $I0$  to  $I7$  into a single 8-bit bus  $O[7..0]$ . Obeying the least to most mapping at the bit level,  $I0$  will connect to  $O0$ , and  $I7$  will connect to  $O7$

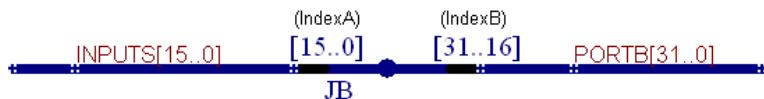
In addition, bus joiners are available that allow you to split/merge bidirectional signals. The image below shows a bus joiner that joins an 8-bit IO bus on the left to two 4-bit IO buses on the right.



Note that apart from the JB-type joiner, all bus joiner pins have an IO direction – use the correct joiner to maintain the IO flow. Pin IO can be displayed on sheet by enabling the **Pin Direction** option on the **Schematic – General** page of the *Preferences* dialog.

### Matching Buses of Different Widths

The JB-type bus joiner allows you to match nets in buses of different widths. It does this via 2 component parameters, IndexA and IndexB that map from one bus through to the other bus. These indices must be defined when you use a JB-type bus joiner.



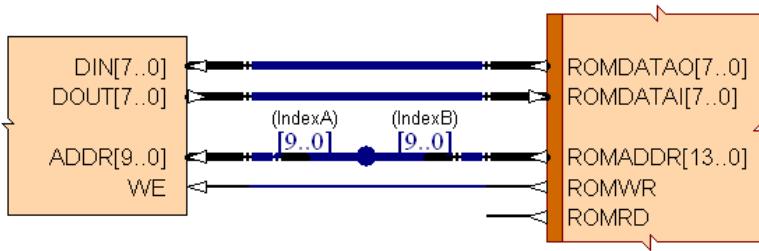
Note that there is no IO direction for a JB-type bus joiner.

Read the flow of nets through a JB-type bus joiner by matching from the nets in the first bus net label (on the left), to the first index on the bus joiner, to the second index on the bus joiner, to the nets defined in the second bus net label (on the right). Using the image above, the mapping can be summarized as:

$Left\ Bus \leftrightarrow IndexA \leftrightarrow IndexB \leftrightarrow Right\ Bus$

The following sections provide examples of using a JB-type bus joiner in a design and the rules used for matching nets.

### Sub-set Mapping



If both bus ranges are descending, match by same bus index (one range must lie within the other for valid connections).

Mapping is as follows:

ADDR9 ↔ IndexA9 ↔ IndexB9 ↔ ROMADDR9 thru to

ADDR0 ↔ IndexA0 ↔ IndexB0 ↔ ROMADDR0

(In this example ROMADDR10 thru ROMADDR13 will be unconnected).

### Offset Mapping



In this example, mapping is as follows:

INPUTS15 ↔ IndexA15 ↔ IndexB31 ↔ PORTB31 thru to

INPUTS0 ↔ IndexA0 ↔ IndexB0 ↔ PORTB16

### Range Inversion

If one bus range is descending and another is ascending, the indices are matched from left to right.



In the image above the range of the left-hand bus is ascending, while that of the right-hand bus is descending. The mapping is as follows:

INPUTS0 ↔ IndexA15 ↔ IndexB31 ↔ PORTB31 thru to

INPUTS15 ↔ IndexA0 ↔ IndexB16 ↔ PORTB16



In the image above the range of the left-hand bus is descending, while that of the right-hand bus is ascending. The mapping is as follows:

INPUTS15 ↔ IndexA15 ↔ IndexB31 ↔ PORTB0 thru to

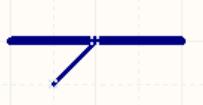
INPUTS0 ↔ IndexA0 ↔ IndexB16 ↔ PORTB15

For an example of using bus joiners, refer to the example \Examples\FPGA Design Tips\Bus Interconnect\Interconnect.PrjFpg in the Altium Designer installation.

---

Any changes made to object properties during placement will cause the default properties for the object to be updated, unless the **Permanent** option - on the **Schematic - Default Primitives** page of the *Preferences* dialog - is enabled. When this option is enabled, changes made will affect only the object being placed and subsequent objects placed during the same placement session.

## Bus Entry



### Description

A bus entry is an electrical design primitive. It is a special wire at an angle of 45 degrees that is used to connect a wire to a bus line. A bus entry allows you to connect two different nets to the same point on a bus. If this was done using wires the two nets would short.

### Availability

Bus entries are available for placement in the Schematic Editor only. Use one of the following methods to access the placement command:

- choose **Place » Bus Entry [P, U]** from the Schematic Editor main menus
- click the  button on the **Wiring** toolbar.

### Placement

After launching the command, the cursor will change to a cross-hair and you will enter bus entry placement mode. Click or press **Enter** to place a bus entry at the cursor position.

Continue placing bus entries or right-click or press **Esc** to exit placement mode.

Press the **Spacebar** while in placement mode to rotate the bus entry. Rotation is anti-clockwise and in steps of 90°.

Press the **X** or **Y** keys while in placement mode to flip the bus entry along the X-axis or Y-axis respectively.

### Editing

The properties of a bus entry object can be modified before, during and after placement. Editing itself falls into two categories - graphical and non-graphical.

The following three methods of non-graphical editing are available:

#### Editing via an associated properties dialog

This method of editing uses the following dialog to modify the properties of a bus entry object.

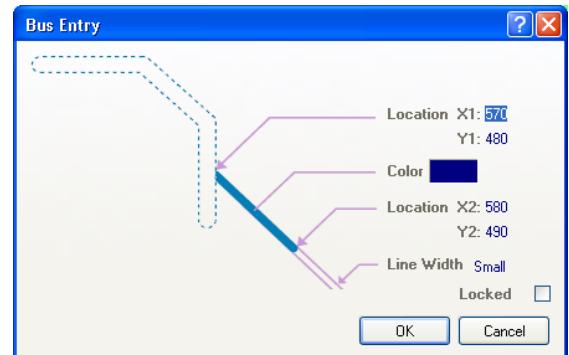
Use the dialog's 'What's This Help' feature to obtain detailed information about each of the options available. Click on the question mark button at the top right of the dialog and then click over a field or option to pop-up information specific to that field or option.

The **Bus Entry** dialog can be accessed prior to entering placement mode, from the **Schematic - Default Primitives** page of the **Preferences** dialog (**Tools » Schematic Preferences**). This allows you to change the default properties for the bus entry object, which will be applied when placing subsequent bus entries.

During placement, the **Bus Entry** dialog can be accessed by pressing the **Tab** key.

After placement, the **Bus Entry** dialog can be accessed in one of the following ways:

- double-clicking on the placed bus entry object
- selecting the bus entry object and choosing **Properties** from the right-click pop-up menu
- choosing the **Change** command from the **Edit** menu and then clicking once over the placed bus entry object.



#### Editing via the SCH Inspector panel

The **SCH Inspector** panel enables you to interrogate and edit the properties of one or more design objects in the active document. Used in conjunction with appropriate filtering, the panel can be used to make changes to multiple objects of the same kind, from one convenient location.

 For more information on the **SCH Inspector** panel, press **F1** when the cursor is over the panel.

## Editing via the SCH List panel

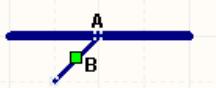
The **SCH List** panel allows you to display design objects from one or more documents in tabular format, enabling you to quickly inspect and modify object attributes. When used in conjunction with the **SCH Filter** panel, it enables you to display just those objects falling under the scope of the active filter - allowing you to target and edit multiple design objects with greater accuracy and efficiency.

 For more information on the **SCH List** and **SCH Inspector** panels, press **F1** when the cursor is over a panel.

## Graphical editing

This method of editing allows you to select a placed bus entry object directly in the workspace and change its location graphically.

When a bus entry object is selected, the following editing handles are available:



Click and drag **A** to move the bus connection point.

Click and drag **B** to move the entire bus entry object.

In either case, the bus entry can be rotated or flipped while dragging.

If you attempt to graphically modify a bus entry object that has its **Locked** property enabled, a dialog will appear asking for confirmation to proceed with the edit.

If the **Protect Locked Objects** option is enabled in the **Schematic – Graphical Editing** page of the *Preferences* dialog (**Tools** » **Schematic Preferences**), and the **Locked** option for this design object is enabled as well then this object cannot be selected or graphically edited.

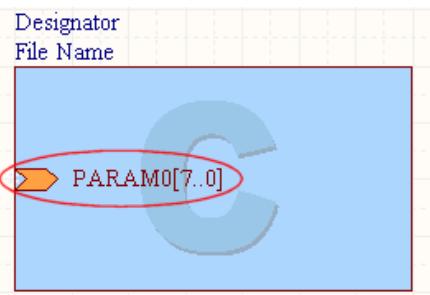
You will have to double click on this locked object directly and disable the **Locked** property or disable the **Protect Locked Objects** option in the **Schematic – Graphical Editing** Preferences dialog to graphically edit this object.

If you attempt to select locked objects along with other objects, only those objects that are unlocked can be selected and moved as a group if the **Protect Locked Objects** option is enabled. Otherwise if the **Protect Locked Objects** option is disabled you will be prompted for confirmation to proceed with the edit of the objects including locked objects.

## Notes

Any changes made to object properties during placement will cause the default properties for the object to be updated, unless the **Permanent** option - on the **Schematic - Default Primitives** page of the *Preferences* dialog - is enabled. When this option is enabled, changes made will affect only the object being placed and subsequent objects placed during the same placement session.

## C Code Entry



### Description

A C Code Entry is an electrical design primitive that belongs within a C Code Symbol. A C Code Symbol represents one top-level exported C function, resident in a referenced C source file. The C Code Entries provide the means by which to access the parameters of the exported function. The entries themselves can be wired to other areas of the circuit design, allowing for transfer of data.

### Availability

There are two types of C Code Entry – Parameter and Control. Parameter-type C Code Entries correspond to the parameters of the exported C function. This type of entry can be placed by the user. Control-type C Code Entries (START, DONE, CLOCK, RESET, RESET\_DONE) are not part of the exported C function, but are automatically added to the C Code Symbol when the Multi-cycle interface mode for that symbol is chosen.

Parameter-type C Code Entries are available for placement in the Schematic Editor only. They can be placed within C Code Symbols that in turn reside on schematic sheets that are part of either an FPGA project (\*.PrjFpg), or a Core project (\*.PrjCor). Use one of the following methods to access the placement command:

- Choose Place » Add C Code Entry [P, Y] from the main menus
- Click the button on the Wiring toolbar.

### Placement

After launching the command, the cursor will change to a cross-hair and you will enter C Code Entry placement mode.

Placement is made by performing the following sequence of actions:

- The C Code Symbol is implicitly chosen by the position of the new C Code Entry floating on the cursor. The entry will change from being grayed-out to colored when it passes within the borders of a C Code Symbol.
- Move the cursor to adjust the position of the C Code Entry in relation to any edge of the C Code Symbol, then click or press Enter to anchor the entry and complete placement.

Continue placing further C Code Entries, or right-click or press Esc to exit placement mode.

### Editing

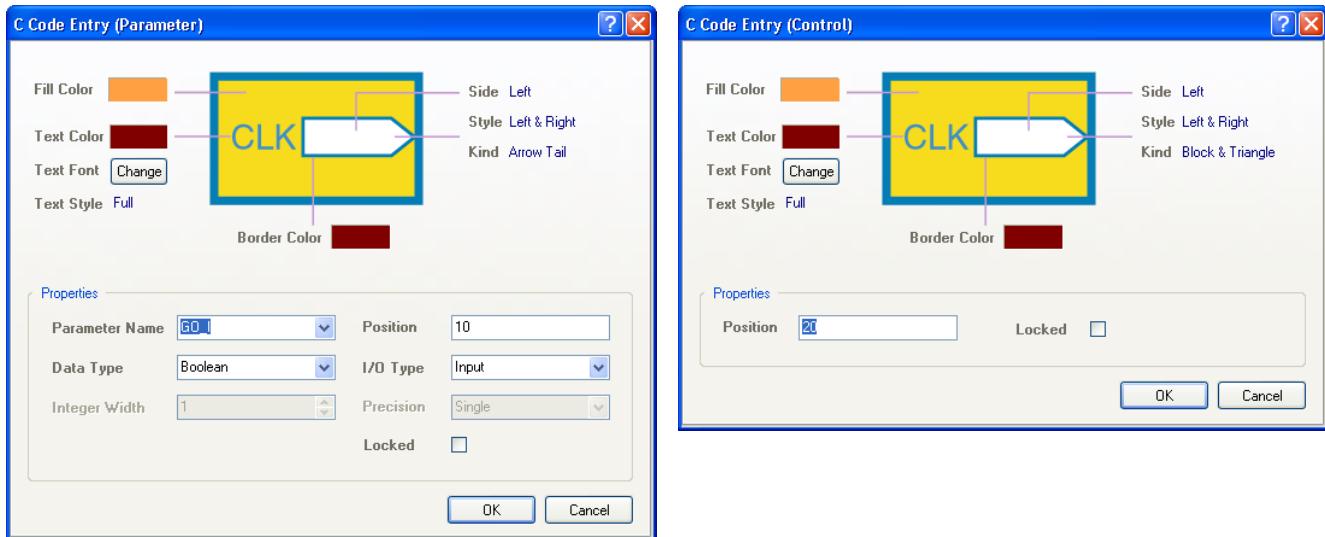
The properties of a Parameter-type C Code Entry object can be modified before, during and after placement. Editing itself falls into two categories - graphical and non-graphical.

**Note:** As a Control-type C Code Entry can not be placed manually, it can only be edited after its automatic placement within the parent C Code Symbol. Only the graphical properties of a Control-type C Code Entry can be viewed and modified.

The following three methods of non-graphical editing are available:

#### Editing via an associated properties dialog

This method of editing uses the following dialogs to modify the properties of a Parameter-type C Code Entry object (left), or a Control-type C Code Entry object (right), respectively.



Use the dialog's 'What's This Help' feature to obtain detailed information about each of the options available. Click on the question mark button at the top right of the dialog and then click over a field or option to pop-up information specific to that field or option.

### Data Type

It is worth taking a closer look at the data types that can be specified for a Parameter-type C Code Entry, and how these map to data types used in the C source code. A C Code Entry can be classed as one of the following data types:

- Boolean
- Floating Point
- Unsigned Integer
- Signed Integer.

The Boolean data type corresponds to the `bool` data type in C, as defined in `stdbool.h`.

For the Floating Point data type, the associated **Precision** property can be set to `Single` or `Double`. These correspond to 32-bit float and 64-bit double C data types respectively.

For integer data types, the associated **Integer Width** property can be defined in the range 1 to 64. You are not limited to the widths of C data types. The following table shows the mapping of variable integer widths (defined for the C Code Entry) to C data types.

Integer Width range specified for a C Code Entry with an integer data type	is mapped to the following C unsigned integer types	is mapped to the following C signed integer types
1-8	<code>unsigned char</code> , <code>uint8_t</code>	<code>char</code> , <code>int8_t</code>
9-16	<code>unsigned short</code> , <code>uint16_t</code>	<code>short</code> , <code>int16_t</code>
17-32	<code>unsigned int</code> , <code>unsigned long</code> , <code>uint32_t</code>	<code>int</code> , <code>long</code> , <code>int32_t</code>
33-64	<code>unsigned long long</code> , <code>uint64_t</code>	<code>long long</code> , <code>int64_t</code>

This mapping becomes especially relevant during synchronization of C Code Entries with corresponding parameters in the exported C function.

### Dialog Access

The *C Code Entry (Parameter)* dialog can be accessed prior to entering placement mode, from the **Schematic - Default Primitives** page of the *Preferences* dialog (**Tools** » **Schematic Preferences**). This allows you to change the default properties for the C Code Entry object, which will be applied when placing subsequent C Code Entries.

During placement, the *C Code Entry (Parameter)* dialog can be accessed by pressing the **Tab** key.

After placement, the *C Code Entry (Parameter)* dialog (or *C Code Entry (Control)* dialog) can be accessed in one of the following ways:

- Double-clicking on the placed C Code Entry object
- Selecting the C Code Entry object and choosing **Properties** from the right-click pop-up menu
- Choosing the **Change** command from the **Edit** menu and then clicking once over the placed C Code Entry object.

### Editing via the SCH Inspector panel

The **SCH Inspector** panel enables you to interrogate and edit the properties of one or more design objects in the active document. Used in conjunction with appropriate filtering, the panel can be used to make changes to multiple objects of the same kind, from one convenient location.

 For more information on the **SCH Inspector** panel, press **F1** when the cursor is over the panel.

### Editing via the SCH List panel

The **SCH List** panel allows you to display design objects from one or more documents in tabular format, enabling you to quickly inspect and modify object attributes. When used in conjunction with the **SCH Filter** panel, it enables you to display just those objects falling under the scope of the active filter - allowing you to target and edit multiple design objects with greater accuracy and efficiency.

 For more information on the **SCH List** and **SCH Inspector** panels, press **F1** when the cursor is over a panel.

### Graphical editing

This method of editing allows you to select a placed C Code Entry object directly in the workspace and change its location graphically.

C Code Entries can only be adjusted with respect to their size by changing the size of the font used (accessed through the respective properties dialog). As such, editing handles are not available when the C Code Entry object is selected:



Click anywhere inside the dashed box and drag to reposition the C Code Entry within the C Code Symbol as required. The C Code Symbol will be resized automatically if you attempt to move the C Code Entry beyond the current extents of the symbol.

If the **Enable In-Place Editing** option is enabled on the **Schematic – General** page of the *Preferences* dialog (**Tools** » **Schematic Preferences**), you will be able to edit the **Parameter Name** for the C Code Entry (Parameter-type only) directly in the workspace. Select the C Code Entry object and then click once to invoke the feature. Type the new name as required and then click away from the C Code Entry object or press **Enter** to effect the change.

### Sheet Entry Actions

Right-clicking over a placed C Code Entry will pop-up a context-sensitive menu, from which a variety of commands are available that act on that C Code Entry (or on all selected C Code Entries where applicable). The following sections detail each of these commands.

**Note:** Many of the following commands are also available from the Schematic Editor's main menus. Commands on the main menus apply to the selected C Code Entry(ies) or allow you to choose the C Code Entry on which the command will act, rather than just the C Code Entry under the cursor. Where such commands exist, reference to their access is made.

#### Jumping to the Corresponding Parameter (Parameter-type only)

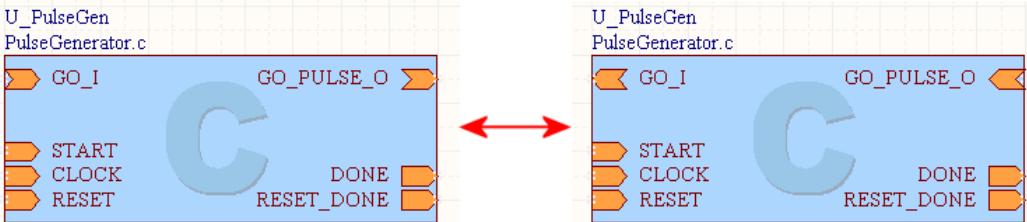
This command is used to jump to the chosen C Code Entry's corresponding parameter in the exported function, in the C source file referenced by that entry's parent C Code Symbol. The command is accessed by right-clicking over the required C Code Entry and choosing **Code Entry Actions** » **Jump to Parameter [ParameterName]** from the menu that appears.

#### Toggling C Code Entry I/O Type (Parameter-type only)

This command is used to toggle the I/O Type for a C Code Entry.

The command can be accessed by right -clicking over the required C Code Entry (or one C Code Entry in a selection of C Code Entries) and choosing **Code Entry Actions** » **Toggle Parameter IO Type** from the menu that appears.

After launching the command, the I/O Type defined for each C Code Entry will be toggled.

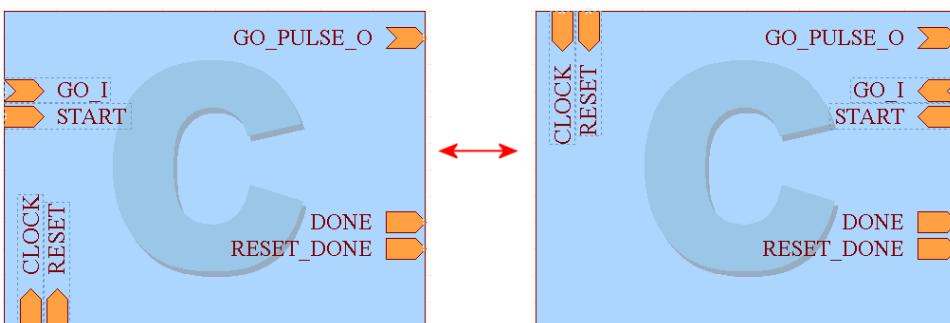


### Swapping C Code Entry Side

This command is used to relocate a C Code Entry to the side of its parent C Code Symbol that is directly opposite to its current position. The C Code Entry's I/O Type is not changed by the swap.

The command can be accessed by right-clicking over the required C Code Entry (or one C Code Entry in a selection of C Code Entries) and choosing **Code Entry Actions » Swap Code Entry Side** from the menu that appears.

After launching the command, each C Code Entry will be swapped to the opposite side of its C Code Symbol.



### Reversing C Code Entry Order

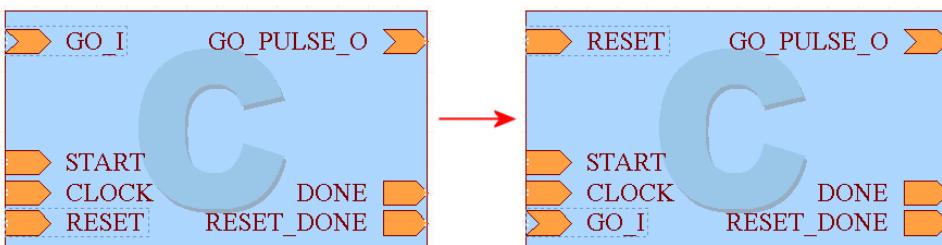
This command allows you to reverse the order that selected C Code Entries appear along a side of a parent C Code Symbol. The I/O Type of a C Code Entry is not changed by the reordering.

The command can be accessed by choosing **Edit » Move » Reverse Selected Sheet Entries Order** from the Schematic Editor's main menus.

Ensure that all C Code Entries that you wish to reorder are selected prior to launching the command. Two or more C Code Entries must be selected for a particular side of a C Code Symbol in order for the command to have effect. You can simultaneously reorder C Code Entries along different sides of the same parent C Code Symbol and across different C Code Symbols on the active schematic sheet.

After launching the command, the reordering will take place. The reordering is achieved by mirroring the positions of the selected C Code Entries - along a particular C Code Symbol side - about an imaginary line at the mid point of the distance between the extents of the two most-outer selected C Code Entries.

The following image shows reordering for two selected C Code Entries, whereby they exchange positions.



## Notes

The C language allows parameters of a function to have the same name, if they differ in case. However, it is illegal for C Code Entries in the the same C Code Symbol to have the same name, even if those names differ in case.

---

The **Parameter Name** for a C Code Entry and the corresponding name for the parameter in the exported C function must be the same and be the same case.

---

The parameters for the exported C function can also be defined directly on the **Signature** tab of the C Code Symbol's properties dialog. Corresponding C Code Entries for any parameters defined here will be added to the C Code Symbol accordingly, and automatically.

---

If the exported C function provides a return value, such a value would appear as a special C Code Entry, with an **IO Type** of **Output** and a **Parameter Name** of `retval`.

---

C functions allow only one return value, but it is possible to have more outputs by using pointer parameters. Output C Code Entries correspond to pointer parameters in the C function. Note that values must be assigned to output pointer parameters (and don't forget to dereference the pointer when assigning a value to it).

---

Control-type C Code Entries `START`, `DONE` and `CLOCK` are added to the C Code Symbol when the interface mode for the symbol is set to `Multi-cycle`. If the **Enable reset logic** option is enabled, two additional Control-type C Code Entries will be added to the symbol – `RESET` and `RESET_DONE`.

---

Should you need to negate (include a bar over the top of) a C Code Entry name, this can be done in one of two ways:

- Include a backslash character after each character in the name (e.g. `E\N\A\B\L\E\`)
- Enable the **Single '\'' Negation** option on the **Schematic - Graphical Editing** page of the *Preferences* dialog, then include one backslash character at the start of the name (e.g. `\ENABLE`).

---

Changes made to object properties during placement will cause the default properties for the object to be updated, unless the **Permanent** option - on the **Schematic - Default Primitives** page of the *Preferences* dialog - is enabled. With this option enabled, changes will affect only the object being placed and subsequent objects placed during the same placement session.

---

If you attempt to graphically modify a C Code Entry object that has its **Locked** property enabled, a dialog will appear asking for confirmation to proceed with the edit. Additionally, if the **Protect Locked Objects** option is enabled on the **Schematic - Graphical Editing** page of the *Preferences* dialog, then the object cannot be selected or graphically edited in any way. In this case, either disable the object's **Locked** property, or disable the **Protect Locked Objects** option.

## C Code Symbol



### Description

A C Code Symbol is an electrical design primitive. Similar in nature to a standard Sheet Symbol, it is used to reference an underlying C source file. More specifically, it represents a single, top-level exported function within that file. Access to parameters in the exported function is made using C Code Entry primitives – placed on the C Code Symbol and functionally similar to Sheet Entries placed on a standard Sheet Symbol.

The exported C function is subsequently translated into a hardware function using Altium Designer's C-to-Hardware Compilation (CHC) technology. The end result is an independent module of FPGA logic which connects, through defined IO ports, to other areas of the circuit design.

### Availability

C Code Symbols are available for placement in the Schematic Editor only. They can be placed on schematic sheets that are part of either an FPGA project (\*.PrjFpg), or a Core project (\*.PrjCor). Use one of the following methods to access the placement command:

- Choose **Place » C Code Symbol [P, M]** from the main menus
- Click the  button on the **Wiring** toolbar.

### Placement

After launching the command, the cursor will change to a cross-hair and you will enter C Code Symbol placement mode. Placement is made by performing the following sequence of actions:

- Click or press **Enter** to anchor the top-left corner of the C Code Symbol.
- Move the cursor to adjust the size of the C Code Symbol, then click or press **Enter** to anchor the diagonally-opposite corner and thereby complete placement.
- Continue to place other C Code Symbols, or right-click or press **Esc** to exit placement mode.

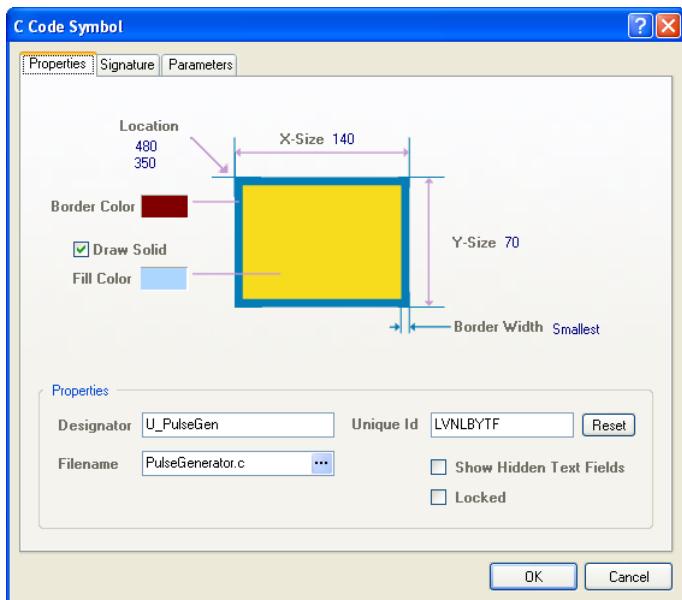
### Editing

The properties of a C Code Symbol object can be modified before, during and after placement. Editing itself falls into two categories - graphical and non-graphical.

The following three methods of non-graphical editing are available:

#### Editing via an associated properties dialog

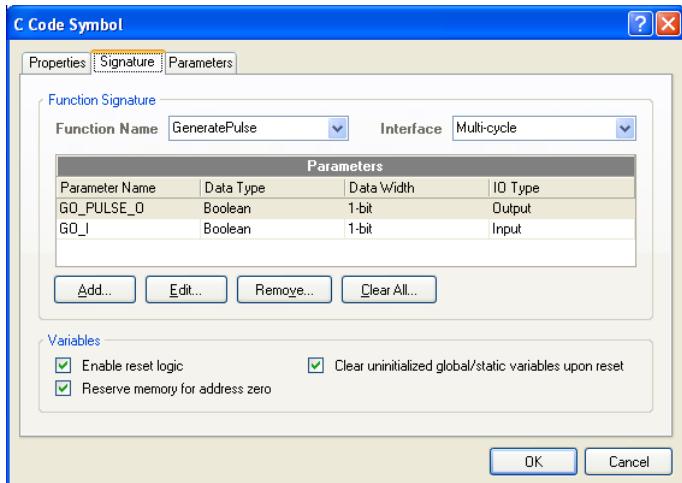
This method of editing uses the following dialog to modify the properties of a C Code Symbol object:



Use the dialog's 'What's This Help' feature to obtain detailed information about each of the options available. Click on the question mark button at the top right of the dialog and then click over a field or option to pop-up information specific to that field or option.

### Signature Tab

It is worth taking a closer look at the **Signature** tab of the **C Code Symbol** dialog, as it is here that the interface between the C Code Symbol and the exported function in the referenced C source file is defined.



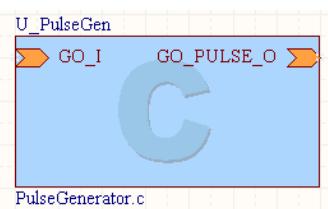
The required C function is 'hooked-up' to the C Code Symbol using the **Function Name** field.

The **Parameters** region lists the parameters (and their properties) of the exported function. The entries here are made available on the C Code Symbol as C Code Entries – allowing for data transfer to/from other logic in the design.

The C-to-Hardware Compiler can generate two types of circuit from the C source code for an exported function – **Combinatorial** and **Multi-cycle**. This is specified using the **Interface** field, which is set to **Combinatorial** by default. It is worth looking more closely at the difference between the two:

- **Combinatorial** – this type of circuit consists of logic gates whose outputs at any time are determined only by the values of the inputs. In this interface mode, only the parameters of the exported function appear as C Code Entries on the C Code Symbol, commonly referred to as 'Parameter' entries.

Combinatorial circuits can be generated for simple C functions which do not depend on stored state (memory, previous executions, etc). The C-to-Hardware Compiler will attempt to create a combinatorial circuit if requested but if this is not possible, an error will be issued during compilation and the interface mode must be changed to Multi-cycle. Note

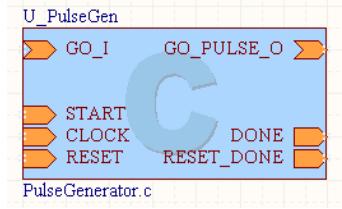


C Code Symbol with Combinatorial interface.

that if a function accesses global variables, this implies stored state and therefore a combinatorial circuit cannot be generated.

- **Multi-cycle** – this type of circuit requires more than one clock cycle to complete. When this interface mode is selected, additional START, DONE and CLOCK C Code Entries will automatically be added to the C Code Symbol, commonly referred to as 'Control' entries. These entries are *not* part of the exported C function.

The circuit begins operating when the START signal is High and is clocked at the frequency arriving at the CLOCK line. The inputs are assumed to be valid. When the circuit finishes operating, the DONE signal will be driven High and the outputs will be valid for one clock cycle. Due to the short availability of the outputs, external latching of the outputs is common practice.



C Code Symbol with Multi-cycle interface.

### Additional Variables

Three additional options are available in the lower region of the **Signature** tab – entitled **Variables**:

- **Enable reset logic** – If this option is enabled, reset logic will be built into the C Code Symbol. Two additional Control-type C Code Entries will be added to the symbol – RESET and RESET\_DONE. The RESET signal will typically be the same system reset used to reset other processor and peripheral devices within the design. When the RESET signal is asserted (active High), all initialized global and static variables (initialized when the FPGA device is programmed) will be reinitialized – set back to their initial values.
- When all variables have finished being reset, the RESET\_DONE signal will be taken High.
- **Clear uninitialized global/static variables upon reset** – This option is only available provided the Enable reset logic option is enabled. When enabled, uninitialized global and static variables will be set to zero during a reset. This option essentially lets you skip the resetting of variables which do not need to be initialized to a predefined value (e.g. an array which is used as a buffer).
- **Reserve memory for address zero** – In C, address zero is reserved for null pointers. With this option disabled, address zero in memories will not be reserved, allowing optimization of memory usage.

### Dialog Access

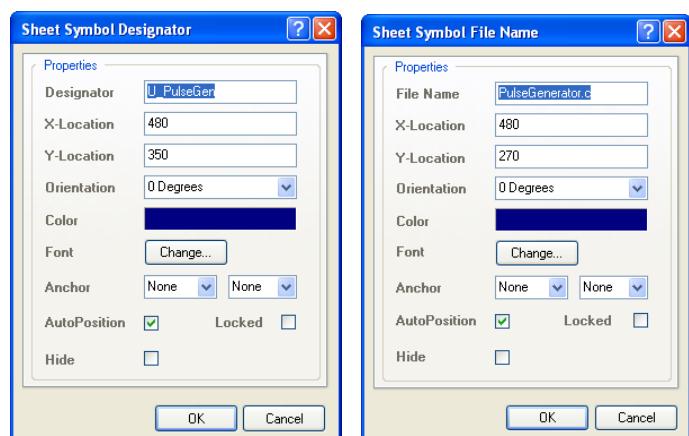
The C Code Symbol dialog can be accessed prior to entering placement mode, from the **Schematic - Default Primitives** page of the **Preferences** dialog (**Tools** » **Schematic Preferences**). This allows you to change the default properties for the C Code Symbol object, which will be applied when placing subsequent C Code Symbols.

During placement, the C Code Symbol dialog can be accessed by pressing the **Tab** key.

After placement, the C Code Symbol dialog can be accessed in one of the following ways:

- Double-clicking on the placed C Code Symbol object.
- Selecting the C Code Symbol object and choosing **Properties** from the right-click pop-up menu.
- Choosing the **Change** command from the **Edit** menu and then clicking once over the placed C Code Symbol object.

The **Designator** and **Filename** text fields associated with a C Code Symbol can be formatted independently of the C Code Symbol itself. The corresponding properties dialogs for each - the *Sheet Symbol Designator* and *Sheet Symbol File Name* dialogs respectively - can be accessed using the three methods described above (replacing C Code Symbol with the relevant object whose properties you wish to view/modify).



### Editing via the SCH Inspector panel

The **SCH Inspector** panel enables you to interrogate and edit the properties of one or more design objects in the active document. Used in conjunction with appropriate filtering, the panel can be used to make changes to multiple objects of the same kind, from one convenient location.



For more information on the **SCH Inspector** panel, press **F1** when the cursor is over the panel.

## Editing via the SCH List panel

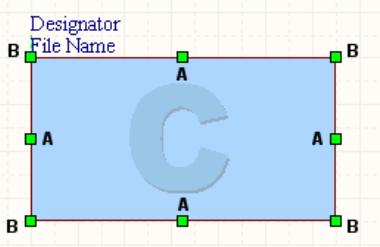
The **SCH List** panel allows you to display design objects from one or more documents in tabular format, enabling you to quickly inspect and modify object attributes. When used in conjunction with the **SCH Filter** panel, it enables you to display just those objects falling under the scope of the active filter - allowing you to target and edit multiple design objects with greater accuracy and efficiency.

 For more information on the **SCH List** and **SCH Inspector** panels, press **F1** when the cursor is over a panel.

## Graphical editing

This method of editing allows you to select a placed C Code Symbol object directly in the workspace and change its size, shape or location, graphically.

When a C Code Symbol object is selected, the following editing handles are available:



Click and drag **A** to resize the C Code Symbol in the vertical and horizontal directions separately.

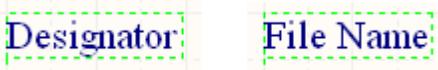
Click and drag **B** to resize the C Code Symbol in the vertical and horizontal directions simultaneously.

Resizing the C Code Symbol will not affect the absolute positions of any defined C Code Entries within.

Click anywhere on the C Code Symbol - away from editing handles - and drag to reposition it.

The C Code Symbol's Designator and Filename text fields can only be adjusted with respect to their size by changing the size of the font used (accessed through the *Sheet Symbol Designator* and *Sheet Symbol File Name* dialogs, respectively). The same is true for any parameters that have been defined for the symbol, and which are visibly displayed in the workspace. In this case, the font size can be changed in the associated *Parameter Properties* dialog.

Editing handles are not available when any of these text objects are selected, as illustrated for Designator and Filename objects below:



Click anywhere inside the dashed box and drag to reposition the text object as required.

The C Code Symbol and any of its associated text objects can be rotated or flipped while dragging:

- Press the **Spacebar** to rotate. Rotation is anti-clockwise and in steps of 90°.
- Press the **X** or **Y** keys to flip along the X-axis or Y-axis respectively.

If the **Enable In-Place Editing** option is enabled on the **Schematic - General** page of the *Preferences* dialog (**Tools » Schematic Preferences**), you will be able to edit the name for an associated text object directly in the workspace. Select the text object and then click once to invoke the feature. Type the new name as required and then click away from the text object or press **Enter** to effect the change.

## C Code Symbol Actions

Right-clicking over a placed C Code Symbol will pop-up a context-sensitive menu, from which a variety of commands are available that act on that C Code Symbol (or on all selected C Code Symbols where applicable). The following sections detail each of these commands.

**Note:** Many of the following commands are also available from the Schematic Editor's main menus. Commands on the main menus apply to the selected C Code Symbol(s) or allow you to choose the C Code Symbol on which the command will act, rather than just the C Code Symbol under the cursor. Where such commands exist, reference to their access is made.

### Opening a Selected C Code Symbol's Source File

This command applies only to the C Code Symbol under the cursor and is accessed by right-clicking and choosing **Code Symbol Actions » Open Source File "FileName.c"** from the menu that appears.

The child C source file for the symbol will be opened (if not already) and made the active document in the main design window.

### Creating a C Source File directly from a C Code Symbol

This command is used to create a new C source code document from a C Code Symbol. A shell function is created using the information specified on the **Signature** tab of the **C Code Symbol** dialog. In this way, you can define how you want an independent module of FPGA logic to 'plug-in' to the rest of your design, prior to writing the C-based functionality of that logic.

The command can be accessed either by:

- Choosing **Design » Create C File From Code Symbol** from the main menus. You will be prompted to choose a C Code Symbol.
- Right-clicking over the required C Code Symbol and choosing **Code Symbol Actions » Create C File From Code Symbol** from the menu that appears.

After launching the command (and choosing the C Code Symbol if applicable), the C source document will be created and opened as the active document. The file will contain a shell function, named using the entry in the **Function Name** field, on the **Signature** tab of the **C Code Symbol** dialog. Parameters for the function are taken from the **Parameters** region of the **Signature** tab, which are synchronized to the corresponding C Code Entries on the symbol.

The C source document that is created is named using the entry in the C Code Symbol's **Filename** field. You can either enter the intended name for the document in this field before launching the command, complete with extension (i.e.

`DocumentName.c`), or leave the name blank and enter the name when saving the generated document at a later stage.

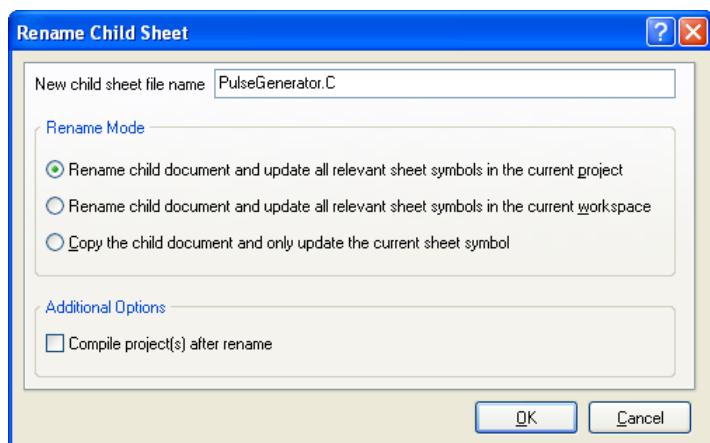
Care should be taken when creating a C source file from a C Code Symbol and a file with that filename already exists. A new C source file with the same filename will be created. The duplication can be resolved when saving, by either saving the new file with a different name, or overwriting the existing file if required.

### Renaming a C Code Symbol's Source File

This command provides a quick and efficient facility for renaming the underlying C source file referenced by a selected C Code Symbol.

The command can be accessed by right-clicking over the required C Code Symbol and choosing **Code Symbol Actions » Rename Source File** from the menu that appears.

After launching the command, the *Rename Child Sheet* dialog will appear.



Initially, the **New child sheet file name** field will contain the current name for the document. Type the new name for the document as required - ensuring that the `.c` extension remains.

The **Rename Mode** region of the dialog allows you to determine how the renaming should proceed. The first two options, as their names suggest, rename the source file. The options differ in the scope of the update with respect to the C Code Symbols that point to this sheet - either in the active project or across all projects in the active design workspace. In each case, the **Filename** field for the C Code Symbol will be updated to reflect the newly named source file.

The third option takes a copy of the source file before renaming the original. Only the current C Code Symbol is updated using this option. This is useful when the current source file is referenced by multiple C Code Symbols, and one C Code Symbol needs to reference a modified version of a function contained in that file. You still want to keep the original source file - you are creating a renamed copy of this file with which to point to from a single C Code Symbol. You can then modify the content of the file as required.

Enabling the **Compile project(s) after rename** option will ensure that the newly named source file is correctly inserted into the design hierarchy, which will be reflected on the **Projects** panel.

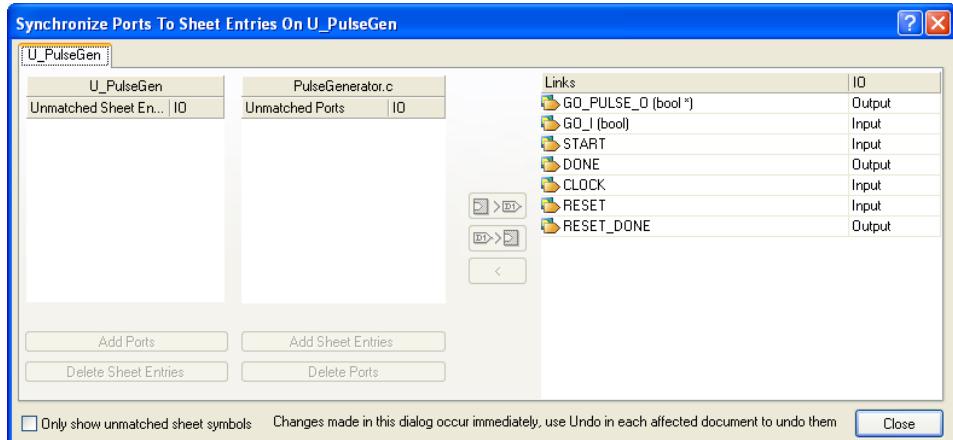
### Synchronizing C Code Entries and Parameters

The Schematic Editor provides a facility for maintaining synchronization between the C Code Entries on a C Code Symbol and the corresponding parameters of the exported function, in the referenced C source file below. Two commands are used to implement this facility, differing only in their scope.

### Synchronization for Current C Code Symbol

Allows you to synchronize the C Code Entries and sub-file function parameters for the C Code Symbol currently under the cursor. The command can be accessed by right-clicking over the required C Code Symbol and choosing **Code Symbol Actions** » **Synchronize Code Entries and Parameters** from the menu that appears.

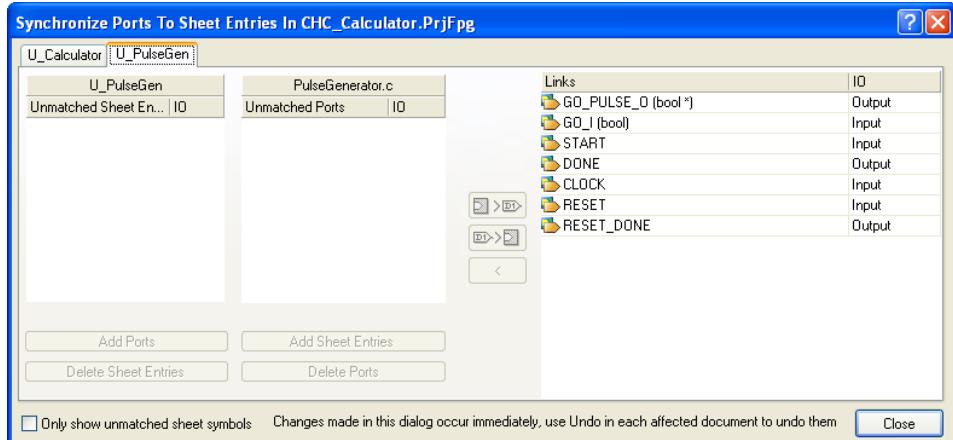
After launching the command, the *Synchronize Ports To Sheet Entries On [CCodeSymbolDesignator]* dialog will appear, as shown in the image on the following page. This incarnation of the Synchronize dialog includes a single tab for the C Code Symbol under the cursor.



### Synchronization for all C Code Symbols

Allows you to synchronize the C Code Entries and sub-file parameters for each C Code Symbol in the active design project. The command can be accessed by choosing **Design** » **Synchronize Sheet Entries And Ports** from the main menus.

After launching the command, the *Synchronize Ports To Sheet Entries In [ActiveProjectName]* dialog will appear:



This incarnation of the Synchronize dialog includes a separate tab for each C Code Symbol found in the project.

### Using the Dialog

The aim, when using the Synchronize dialog, is to ensure that all C Code Entries on a C Code Symbol are matched to parameters of the referenced function in the underlying C source file, both in terms of name and I/O Type.

**Note:** Control-type C Code Entries (START, DONE, CLOCK, RESET, RESET\_DONE) are always synchronized by default, when used. You cannot manipulate the entries for these within the Synchronize dialog.

The left-hand side of the dialog provides two regions:

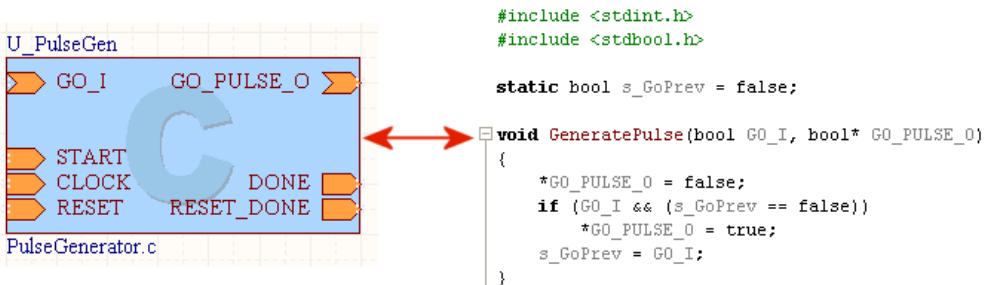
- One listing all currently unmatched C Code Entries (Parameter-type) associated to the chosen C Code Symbol. The Designator of the C Code Symbol appears as the header for the region. Each C Code Entry in this region is listed in terms of its Parameter Name and I/O Type. The Parameter Name will have additional information appended (in brackets), reflecting the data type and width of the C Code Entry.
- One listing all currently unmatched parameters of the exported function in the C source file referenced by the C Code Symbol. The document name appears as the header for the region. Each parameter entry in this region is listed in terms of

its Name and I/O Type. The Name will have additional information appended (in brackets), reflecting the data type of the parameter. If the parameter has I/O Type of **Output**, it will be a pointer parameter (denoted by an asterisk (\*) after its data type).

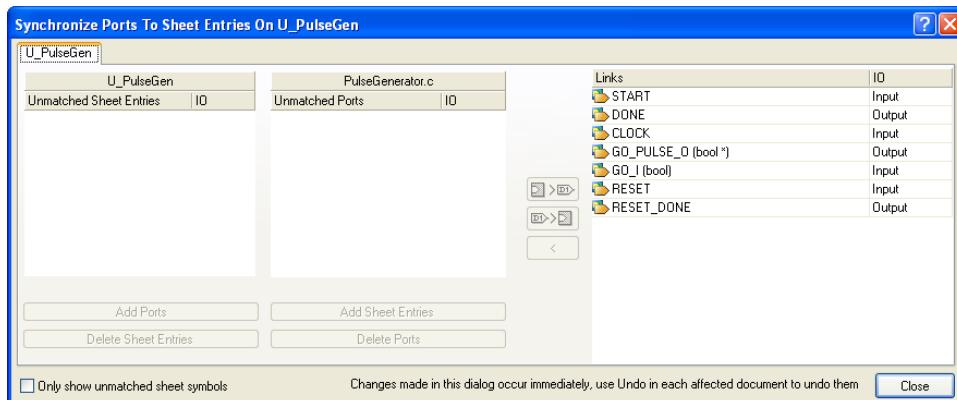
The right-hand side of the dialog lists all currently linked (or matched) C Code Entry-parameter pairings. Each entry shows the common name used by both the C Code Entry as its Parameter Name and the parameter as its Name (again, information on the source function parameter's data type, width and direction is included in brackets). The **IO** column shows the current I/O Type set for each pairing.

**Note:** The Parameter Name of a C Code Entry and the the name of the corresponding parameter in the C source function, must have the same case.

The simplest way to explain the workings of the dialog, in terms of matching C Code Entries with parameters, is to use an example. Consider the C Code Symbol and corresponding exported function parameters in the following image.

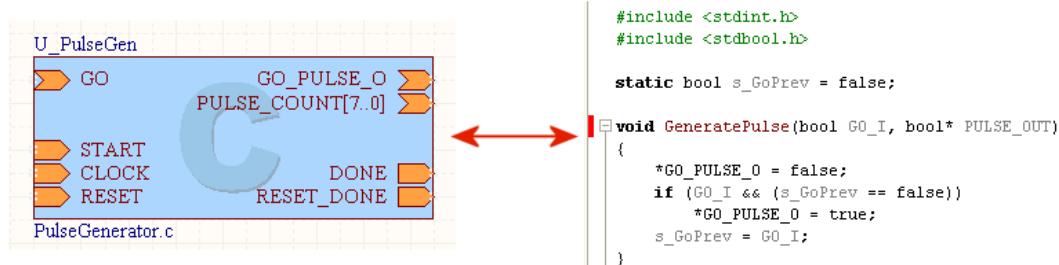


The two Parameter-type C Code Entries (**GO\_I** and **GO\_PULSE\_O**) in the symbol correspond, both in name and I/O Type, to a parameters in the exported function in the referenced C source file. Accessing the Synchronize dialog for this C Code Symbol, there are no unmatched entries (Remember that Control-type C Code Entries are automatically synchronized by default).

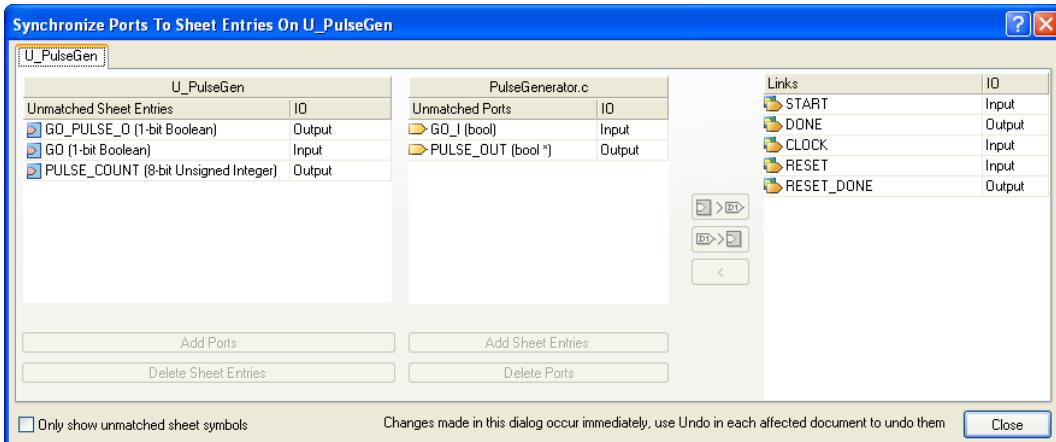


Now consider the same C Code Symbol, but with a few intentional changes made within the design, as listed below and summarized in the following image:

- **Parameter Name** for the C Code Entry **GO\_I** changed to **GO**
- Name of parameter in C source function changed from **GO\_PULSE\_O** to **PULSE\_OUT**
- Extra C Code Entry added, with **Parameter Name** **PULSE\_COUNT**, **Data Type** set to **Unsigned Integer**, **Integer Width** set to 8 and **I/O Type** set to **Output**



This time when the Synchronize dialog is accessed, the changes introduced cause links between C Code Entries and parameters to be broken and the unmatched lists become populated.



The two unmatched regions must now be used to resolve the current unsynchronized state that exists between the C Code Entries of the symbol and the parameters of the function in the underlying C source file. Below each region are two buttons, the use of which will involve making one or more of the following matching decisions:

- If there are C Code Entries that you no longer require, select each (use **Ctrl+click**, **Shift+click** or click-and-drag for multi-select) and click the **Delete Sheet Entries** button. Each C Code Entry in the selection will be removed from the C Code Symbol.
- If there are parameters in the exported function that you no longer require, select each and click the **Delete Ports** button. Each parameter in the selection will be removed from the C source function.
- If there are existing C Code Entries that you need to keep but no corresponding parameters exist for these entries, you can automatically create parameters with the same names and I/O Types (and applicable data types/widths) by selecting each C Code Entry and clicking the **Add Ports** button. The required parameter(s) will be added to the C functions parameter list. The Synchronize dialog will automatically update, with an entry for each C Code Entry-parameter pairing automatically entered into the **Links** region of the dialog.
- If there are existing parameters that you need to keep but no corresponding C Code Entries for these parameters, you can automatically create C Code Entries with the same names and I/O Types (and applicable data types/widths) by selecting each parameter and clicking the **Add Sheet Entries** button. You will be taken to the C Code Symbol, with the C Code Entry(ies) floating on the cursor ready for initial placement. Click or press **Enter** to place the C Code Entry(ies). The Synchronize dialog will reappear, with an entry for each C Code Entry-parameter pairing automatically entered into the **Links** region of the dialog.

All other residual unmatched entries will either be due to name or I/O Type mismatching. For these, manual matching can be carried out. This is done by selecting a C Code Entry in one region and a parameter in the other and then using one of the following two buttons to determine which name and I/O Type attributes are used when linking:

- Use this button to link the selected C Code Entry with the selected parameter, using the name and I/O Type defined for the C Code Entry. The parameter will be renamed and/or its I/O type changed, and the applicable data type/width set also.
- Use this button to link the selected C Code Entry with the selected parameter, using the name and I/O Type defined for the parameter. The C Code Entry will be renamed and/or its I/O type changed, and the applicable data type/width set also.

If you want to break up a linked entry, select it in the **Links** region of the dialog and click on the button.

The actual process of matching will vary from C Code Symbol to C Code Symbol, depending on the discrepancies involved and your preferred matching techniques. Considering the point reached with the underlying example:

U_PulseGen		PulseGenerator.c	
Unmatched Sheet Entries	IO	Unmatched Ports	IO
<input checked="" type="checkbox"/> GO_PULSE_0 (1-bit Boolean)	Output	<input checked="" type="checkbox"/> GO_I (bool)	Input
<input checked="" type="checkbox"/> GO (1-bit Boolean)	Input	<input checked="" type="checkbox"/> PULSE_OUT (bool *)	Output
<input checked="" type="checkbox"/> PULSE_COUNT (8-bit Unsigned Integer)	Output		

the C Code Entries and parameters could be quite easily matched and re-synchronized back to their original states by performing the following actions:

- Selecting the entry for C Code Entry `PULSE_COUNT` and clicking the **Delete Sheet Entries** button
- Selecting the entries for C Code Entry `GO` and parameter `GO_I` and clicking the  button
- Selecting the entries for C Code Entry `GO_PULSE_O` and parameter `PULSE_OUT` and clicking the  button.

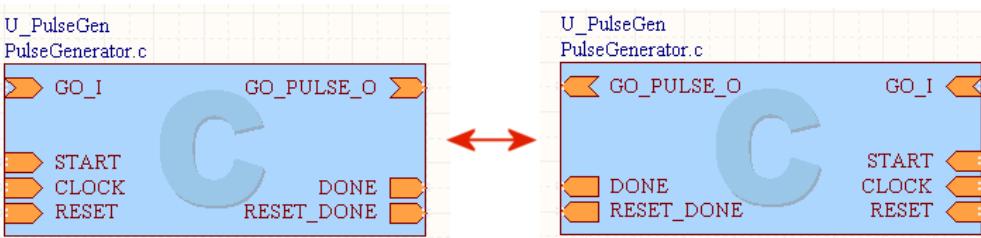
The approach to matching is not set in stone - the tools are provided to allow you to match as quickly and as efficiently as possible.

### Flipping C Code Symbols along the X-Axis

This command allows you to flip C Code Symbols along the X-axis.

The command can be accessed by right-clicking over the required C Code Symbol (or a symbol in a selected group of symbols) and choosing **Code Symbol Actions » Flip Code Symbol Along X** from the menu that appears.

After launching the command the C Code Symbol(s) will be flipped. The C Code Entries associated with a symbol will essentially be swapped to the opposite side of the symbol (in the horizontal plane) - those on the left will be repositioned on the right and vice-versa.



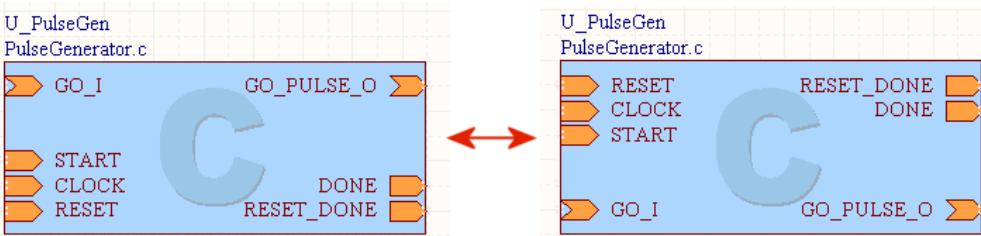
**Note:** When flipping multiple selected C Code Symbols, the symbols will be flipped about an imaginary vertical line which is located mid-way between the bounding extents of the symbols in the selection.

### Flipping C Code Symbols along the Y-Axis

This command allows you to flip C Code Symbols along the Y-axis.

The command can be accessed by right-clicking over the required C Code Symbol (or a symbol in a selected group of symbols) and choosing **Code Symbol Actions » Flip Code Symbol Along Y** from the menu that appears.

After launching the command the C Code Symbol(s) will be flipped. The C Code Entries associated with a symbol will essentially be swapped to the opposite side of the symbol (in the vertical plane) - those at the top will be repositioned at the bottom and vice-versa.



**Note:** When flipping multiple selected C Code Symbols, the symbols will be flipped about an imaginary horizontal line which is located mid-way between the bounding extents of the symbols in the selection.

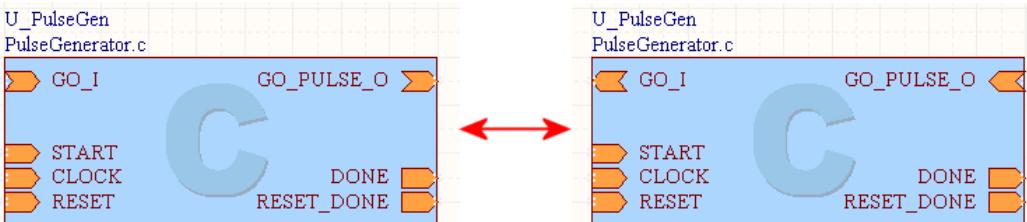
### Toggling C Code Entry IO Type in a C Code Symbol

This command allows you to toggle the I/O Type for C Code Entries in one or more C Code Symbols, simultaneously.

**Note:** This command only applies to Parameter-type C Code Entries, corresponding to actual parameters in the exported C function. The I/O type of Control-type C Code Entries will remain unchanged.

The command can be accessed by right-clicking over the required C Code Symbol (or a symbol in a selected group of symbols) and choosing **Code Symbol Actions » Toggle All Parameters IO Type In Code Symbol** from the menu that appears.

After launching the command, the I/O Type defined for each C Code Entry will be toggled.



## Notes

The **Filename** property of the C Code Symbol, set on the **Properties** tab of the *C Code Symbol* dialog, must be set to the file name of the C source code file that the symbol represents.

Each C Code Symbol is associated with *one* top-level exported function in a C source file. However, this exported function may call other functions, which themselves may be distributed among several source files.

All required C source files and header files need to be added to the FPGA project. This allows Altium Designer's C-to-Hardware Compiler to find all source files needed to generate the FPGA logic.

The C language allows parameters of a function to have the same name, if they differ in case. However, it is illegal for C Code Entries in the the same C Code Symbol to have the same name, even if those names differ in case.

The exported C function can *not* be called `main`.

The C Code Symbol must specify an exported function – using the **Function Name** field, on the **Signature** tab of the *C Code Symbol* dialog. This property cannot be left blank. The name entered must be identical (in case also) to the name of the function in the referenced C source file. If the specified function does not exist in the source file, a violation will occur.

The C-to-Hardware Compiler can generate VHDL or Verilog from the C code. By default, VHDL will be generated. Change this as required using the corresponding options available in the **Compiler Netlist** region, on the **FPGA – General** page of the *Preferences* dialog (**DXP » Preferences**).

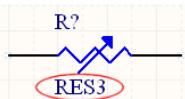
Unlike code in an embedded software project, it is not possible to debug C code that is added to an FPGA or Core project through use of a C Code Symbol and C-to-Hardware Compilation technology. This is because the C code in the exported function is transformed into a hardware circuit, not a series of instructions to be carried out by a host processor. The only available avenue for test is use of a LAX configurable logic analyzer virtual instrument.

If a group of C Code Entries is pasted into a selected C Code Symbol and those entries fall outside the current bounds of the symbol, it will automatically be resized to accommodate them.

Changes made to object properties during placement will cause the default properties for the object to be updated, unless the **Permanent** option - on the **Schematic – Default Primitives** page of the *Preferences* dialog - is enabled. With this option enabled, changes will affect only the object being placed and subsequent objects placed during the same placement session.

If you attempt to graphically modify a C Code Symbol object that has its **Locked** property enabled, a dialog will appear asking for confirmation to proceed with the edit. Additionally, if the **Protect Locked Objects** option is enabled on the **Schematic - Graphical Editing** page of the *Preferences* dialog, then the object cannot be selected or graphically edited in any way. In this case, either disable the object's **Locked** property, or disable the **Protect Locked Objects** option.

## Comment



### Description

The comment text field is a non-electrical child object of an electrical design primitive. It is used to provide a description of a placed part, such as the value (220nF) or device type (74HC32).

### Availability and Placement

The comment is automatically placed when the parent part object is placed. As such, it is not a design object that can be accessed and placed by the user.

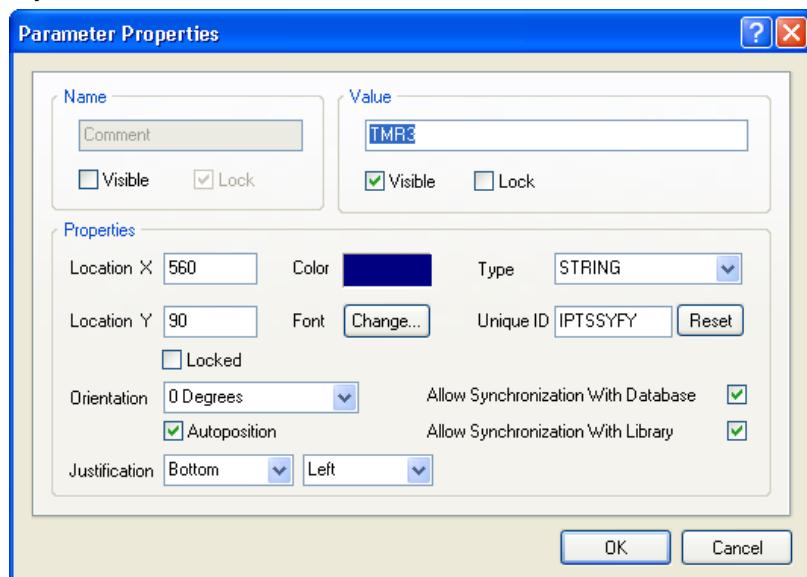
### Editing

The properties of a comment object can be modified before and after placement of the parent part. Editing itself falls into two categories - graphical and non-graphical.

The following three methods of non-graphical editing are available:

#### Editing via an associated properties dialog

This method of editing uses the following dialog to modify the properties of a comment object, independently of the parent part object.



Use the dialog's 'What's This Help' feature to obtain detailed information about each of the options available. Click on the question mark button at the top right of the dialog and then click over a field or option to pop-up information specific to that field or option.

The *Parameter Properties* dialog can be accessed prior to entering part placement mode, from the **Schematic - Default Primitives** page of the *Preferences* dialog (**Tools » Schematic Preferences**). This allows you to change the default properties for the comment object, which will be applied when placing subsequent parts.

After placement, the *Parameter Properties* dialog can be accessed in one of the following ways:

- double-clicking on the comment field of the placed part object
- selecting the comment field of the part object and choosing **Properties** from the right-click pop-up menu
- choosing the **Change** command from the **Edit** menu and then clicking once over the comment field of the placed part object.

## Editing via the SCH Inspector panel

The **SCH Inspector** panel enables you to interrogate and edit the properties of one or more design objects in the active document. Used in conjunction with appropriate filtering, the panel can be used to make changes to multiple objects of the same kind, from one convenient location.

 For more information on the **SCH Inspector** panel, press **F1** when the cursor is over the panel.

## Editing via the SCH List panel

The **SCH List** panel allows you to display design objects from one or more documents in tabular format, enabling you to quickly inspect and modify object attributes. When used in conjunction with the **SCH Filter** panel, it enables you to display just those objects falling under the scope of the active filter - allowing you to target and edit multiple design objects with greater accuracy and efficiency.

 For more information on the **SCH List** and **SCH Inspector** panels, press **F1** when the cursor is over a panel.

## Graphical editing

This method of editing allows you to select a comment object directly in the workspace and change its location graphically. Comments can only be adjusted with respect to their size by changing the size of the font used (accessed through the *Parameter Properties* dialog). As such, editing handles are not available when the comment object is selected:

 Comment: RES3

Click anywhere inside the dashed box and drag to reposition the comment object as required. The object can be rotated or flipped while dragging:

- Press the **Spacebar** to rotate the comment. Rotation is anti-clockwise and in steps of 90°.
- Press the **X** or **Y** keys to flip the comment along the X-axis or Y-axis respectively.

If the **Enable In-Place Editing** option is enabled on the **Schematic - General** page of the *Preferences* dialog (**Tools » Schematic Preferences**), you will be able to edit the value for the comment directly in the workspace. Select the comment and then click once to invoke the feature. Type the new value as required and then click away from the comment field or press **Enter** to effect the change.

## Notes

Any changes made to the Comment field during part placement (either in the *Place Part* dialog or the *Component Properties* dialog) will cause the default properties for the comment object to be updated, unless the **Permanent** option - on the **Schematic - Default Primitives** page of the *Preferences* dialog - is enabled. When this option is enabled, changes made will affect only the comment of the part being placed and subsequent parts placed during the same placement session.

## Compile Mask



### Description

A compile mask is a design directive. It is used to effectively hide the area of the design it contains from the Compiler, allowing you to manually prevent error checking for circuitry that may not yet be complete and you know will generate compile errors. This can prove very useful if you need to compile the active document or project to check the integrity of the design in other specific areas, but do not want the clutter of compiler-generated messages associated with unfinished portions of the design.

### Availability

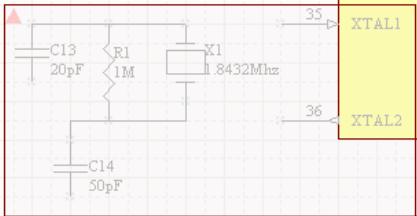
Compile masks are available for placement in the Schematic Editor only, by choosing **Place » Directives » Compile Mask [P, V, K]** from the Schematic Editor main menus.

### Placement

After launching the command, the cursor will change to a cross-hair and you will enter compile mask placement mode.

Placement is made by performing the following sequence of actions:

- click or press **Enter** to anchor the first corner of the mask
- move the cursor to adjust the size of the mask, such that it completely encapsulates the area of the design that you want hidden from the Compiler, then click or press **Enter** to anchor the diagonally-opposite corner and thereby complete placement of the mask. Design objects falling completely within the bounds of the mask will become grayed-out, as illustrated in the image below:



Continue placing further compile masks, or right-click or press **Esc** to exit placement mode.

The compile mask object can be rotated or flipped while in placement mode and before the first corner of the mask is anchored:

- Press the **Spacebar** to rotate the compile mask. Rotation is anti-clockwise and in steps of 90°.
- Press the **X** or **Y** keys to flip the compile mask along the X-axis or Y-axis respectively.

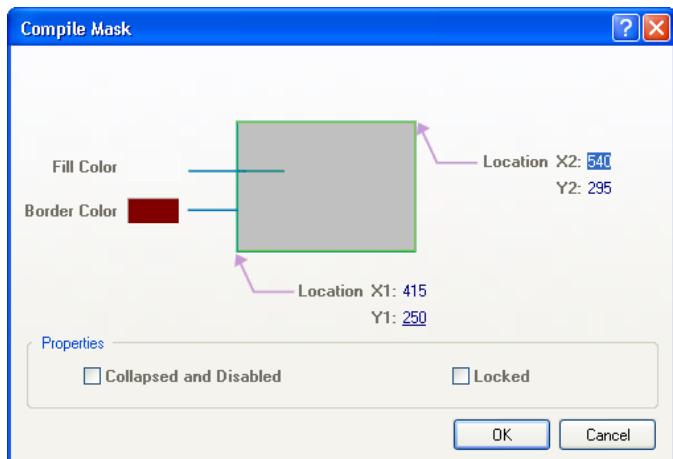
### Editing

The properties of a compile mask object can be modified before, during and after placement. Editing itself falls into two categories - graphical and non-graphical.

The following three methods of non-graphical editing are available:

#### Editing via an associated properties dialog

This method of editing uses the following dialog to modify the properties of a compile mask object.



Use the dialog's 'What's This Help' feature to obtain detailed information about each of the options available. Click on the question mark button at the top right of the dialog and then click over a field or option to pop-up information specific to that field or option.

The **Compile Mask** dialog can be accessed prior to entering placement mode, from the **Schematic - Default Primitives** page of the **Preferences** dialog (**Tools » Schematic Preferences**). This allows you to change the default properties for the compile mask object, which will be applied when placing subsequent compile masks.

During placement, the **Compile Mask** dialog can be accessed by pressing the **Tab** key.

After placement, the **Compile Mask** dialog can be accessed in one of the following ways:

- double-clicking on the placed compile mask object
- selecting the compile mask object and choosing **Properties** from the right-click pop-up menu
- choosing the **Change** command from the **Edit** menu and then clicking once over the placed compile mask object.

#### Editing via the SCH Inspector panel

The **SCH Inspector** panel enables you to interrogate and edit the properties of one or more design objects in the active document. Used in conjunction with appropriate filtering, the panel can be used to make changes to multiple objects of the same kind, from one convenient location.

For more information on the **SCH Inspector** panel, press **F1** when the cursor is over the panel.

#### Editing via the SCH List panel

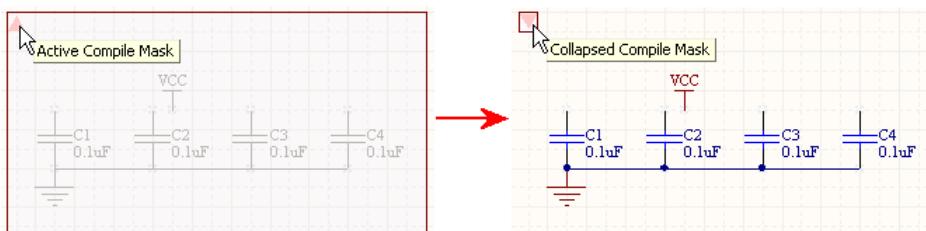
The **SCH List** panel allows you to display design objects from one or more documents in tabular format, enabling you to quickly inspect and modify object attributes. When used in conjunction with the **SCH Filter** panel, it enables you to display just those objects falling under the scope of the active filter - allowing you to target and edit multiple design objects with greater accuracy and efficiency.

For more information on the **SCH List** and **SCH Inspector** panels, press **F1** when the cursor is over a panel.

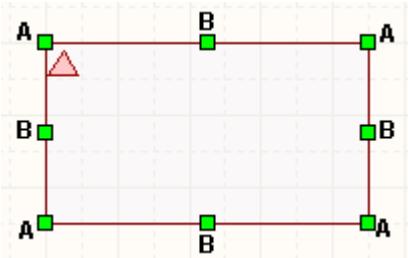
#### Graphical editing

This method of editing allows you to select a placed compile mask object directly in the workspace and change its size, shape or location, graphically.

A compile mask can be displayed in either expanded (full frame) or collapsed (small triangle) modes. These modes correspond to the mask being enabled and disabled respectively. Toggle the display mode either by using the **Collapsed and Disabled** option in the **Compile Mask** dialog, or by clicking on the top left corner of a placed compile mask.



When a fully expanded compile mask object is selected, the following editing handles are available:



Click and drag **A** to resize the compile mask in the vertical and horizontal directions simultaneously.

Click and drag **B** to resize the compile mask in the vertical and horizontal directions separately.

Click anywhere on the compile mask - away from editing handles - and drag to reposition it. The compile mask can be rotated or flipped while dragging.

The size and shape of a compile mask cannot be changed graphically when the mask is in collapsed mode, only its location/orientation. As such, editing handles are not available when a collapsed compile mask object is selected:



Click anywhere inside the dashed box and drag to reposition the collapsed compile mask as required. The collapsed compile mask can be rotated or flipped while dragging.

If you attempt to graphically modify a compiler mask object that has its **Locked** property enabled, a dialog will appear asking for confirmation to proceed with the edit.

If the **Protect Locked Objects** option is enabled in the **Schematic – Graphical Editing** page of the *Preferences* dialog (**Tools** » **Schematic Preferences**), and the **Locked** option for this design object is enabled as well then this object cannot be selected or graphically edited.

You will have to double click on this locked object directly and disable the **Locked** property or disable the **Protect Locked Objects** option in the **Schematic – Graphical Editing** Preferences dialog to graphically edit this object.

If you attempt to select locked objects along with other objects, only those objects that are unlocked can be selected and moved as a group if the **Protect Locked Objects** option is enabled. Otherwise if the **Protect Locked Objects** option is disabled you will be prompted for confirmation to proceed with the edit of the objects including locked objects.

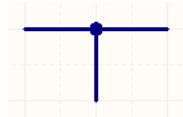
## Notes

While compile masks can be rotated or flipped along the X or Y axis, this has no effect on the orientation of the design circuitry within.

---

Any changes made to object properties during placement will cause the default properties for the object to be updated, unless the **Permanent** option - on the **Schematic - Default Primitives** page of the *Preferences* dialog - is enabled. When this option is enabled, changes made will affect only the object being placed and subsequent objects placed during the same placement session.

## Compiler Generated Junction



### Description

A junction is an electrical design primitive. It is a small circular object used to logically join intersecting wires or buses on a schematic sheet. A Compiler generated junction is a junction that is automatically placed by the Auto-junctioning feature when two wires/buses are connected in a T-type fashion, or when a wire/bus connects orthogonally to a pin or power port/bus power port. This allows you to create electrical connections at junction points without the need to manually define the connections (through placement of manual junctions).

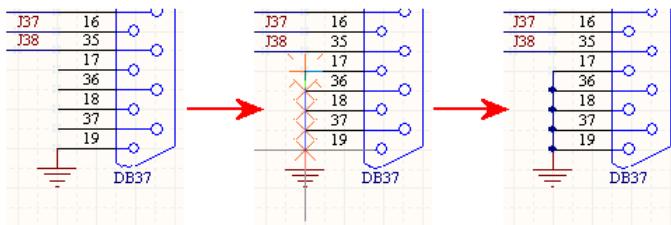
### Availability

This type of junction is placed automatically by the Auto-junctioning feature. As such, it is not a design object that can be accessed and placed by the user.

### Placement

Compiler generated junctions are placed automatically when:

- placing a wire that starts or terminates anywhere along the length of another wire (T-junction), or when the wire connects orthogonally to a pin or power port.



- placing a bus that starts or terminates anywhere along the length of another bus (T-junction), or when the bus connects orthogonally to a pin or bus power port.



Wires or buses that cross away from their end points do not have a junction automatically inserted.

### Editing

A Compiler generated junction object cannot be edited with respect to properties in the usual manner - it cannot be selected in the workspace, has no corresponding properties dialog and cannot be edited graphically.

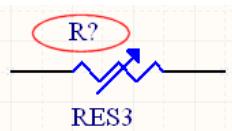
Display of auto-junctions on the schematic sheet, with respect to wires and buses can be controlled from the **Schematic - Compiler** page of the **Preferences** dialog (**Tools » Schematic Preferences**). Additional options provide control over junction size and color.



### Notes

When a wire crosses other wires (or a bus crosses other buses), no junction is inserted. If you want to create a connection between crossed wires or buses, place a manual junction at the crossing point.

## Designator



### Description

The designator text field is a non-electrical child object of an electrical design primitive. It is used to uniquely identify a placed part, thereby distinguishing it from other parts placed not only on the same schematic sheet, but placed across all source schematic sheets in the active project.

### Availability and Placement

The designator is automatically placed when the parent part object is placed. As such, it is not a design object that can be accessed and placed by the user.

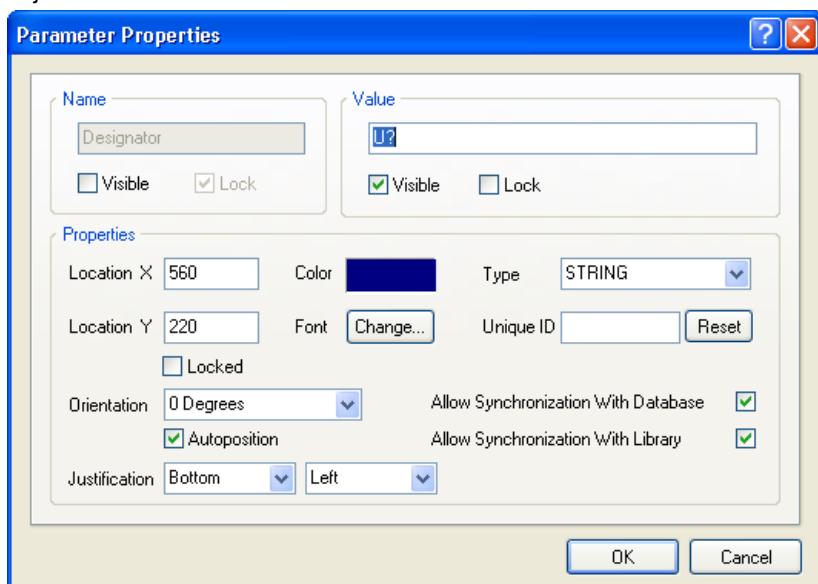
### Editing

The properties of a designator object can be modified before and after placement of the parent part. Editing itself falls into two categories - graphical and non-graphical.

The following three methods of non-graphical editing are available:

#### Editing via an associated properties dialog

This method of editing uses the following dialog to modify the properties of a designator object, independently of the parent part object.



Use the dialog's 'What's This Help' feature to obtain detailed information about each of the options available. Click on the question mark button at the top right of the dialog and then click over a field or option to pop-up information specific to that field or option.

The *Parameter Properties* dialog can be accessed prior to entering part placement mode, from the **Schematic - Default Primitives** page of the *Preferences* dialog (**Tools** » **Schematic Preferences**). This allows you to change the default properties for the designator object, which will be applied when placing subsequent parts.

After placement, the *Parameter Properties* dialog can be accessed in one of the following ways:

- double-clicking on the designator field of the placed part object
- selecting the designator field of the part object and choosing **Properties** from the right-click pop-up menu
- choosing the **Change** command from the **Edit** menu and then clicking once over the designator field of the placed part object.

### Editing via the SCH Inspector panel

The **SCH Inspector** panel enables you to interrogate and edit the properties of one or more design objects in the active document. Used in conjunction with appropriate filtering, the panel can be used to make changes to multiple objects of the same kind, from one convenient location.

 For more information on the **SCH Inspector** panel, press **F1** when the cursor is over the panel.

### Editing via the SCH List panel

The **SCH List** panel allows you to display design objects from one or more documents in tabular format, enabling you to quickly inspect and modify object attributes. When used in conjunction with the **SCH Filter** panel, it enables you to display just those objects falling under the scope of the active filter - allowing you to target and edit multiple design objects with greater accuracy and efficiency.

 For more information on the **SCH List** and **SCH Inspector** panels, press **F1** when the cursor is over a panel.

### Graphical editing

This method of editing allows you to select a designator object directly in the workspace and change its location graphically. Designators can only be adjusted with respect to their size by changing the size of the font used (accessed through the *Parameter Properties* dialog). As such, editing handles are not available when the designator object is selected:



Click anywhere inside the dashed box and drag to reposition the designator object as required. The object can be rotated or flipped while dragging:

- Press the **Spacebar** to rotate the designator. Rotation is anti-clockwise and in steps of 90°.
- Press the **X** or **Y** keys to flip the designator along the X-axis or Y-axis respectively.

If the **Enable In-Place Editing** option is enabled on the **Schematic - General** page of the *Preferences* dialog (**Tools » Schematic Preferences**), you will be able to edit the value for the designator directly in the workspace. Select the designator and then click once to invoke the feature. Type the new value as required and then click away from the designator field or press **Enter** to effect the change.

### Notes

When placing multiple instances of the same part, it is advisable to set the designator of the first part before placing it (press the **Tab** key during placement to edit the part's properties). By doing this, the designator will automatically increment as each subsequent part is placed. If you do not enter a designator before you place a part its designator will be the pre-assigned default, such as R?, C? or U?.

---

When placing multi-part components, a part suffix will automatically be assigned, for example U3A, U3B, and so on. The suffix will automatically increment if the designator is assigned before the first part is placed.

---

Any changes made to the Designator field during part placement (either in the *Place Part* dialog or the *Component Properties* dialog) will cause the default properties for the designator object to be updated, unless the **Permanent** option - on the **Schematic - Default Primitives** page of the *Preferences* dialog - is enabled. When this option is enabled, changes made will affect only the designator of the part being placed and subsequent parts placed during the same placement session.

## Device Sheet Symbol



### Description

A Device Sheet Symbol is an electrical design primitive. It is used to represent a Device Sheet and usually contain predefined circuits. A Device Sheet Symbol references a Device Sheet and can be re-used within and across design projects. Device Sheet Symbols include Sheet Entry Symbols, which provide a connection point for signals between the parent and child sheets, similarly to sheet symbols.

### Availability

Device Sheet symbols are available for placement in the Schematic Editor only. Use one of the following methods to access the placement command:

- choose **Place » Device Sheet Symbol [P, I]** from the main menus
- right click on your Schematic Document and choose **Place » Device Sheet Symbol** from the pop-up menu that appears
- click the button on the **Wiring** toolbar.

### Placement

After launching the command, the *Select Device Sheet* dialog appears with all of your declared Device Sheet Folders. Choose a Device Sheet to place and then click OK. The cursor will change to a cross-hair and you will enter Device Sheet Symbol placement mode. Placement is made by performing the following sequence of actions:

- click or press **Enter** to anchor the top-left corner of the Device Sheet Symbol
- move the cursor to adjust the size of the Device Sheet Symbol, then click or press **Enter** to anchor the diagonally-opposite corner and thereby complete placement of the Device Sheet Symbol.

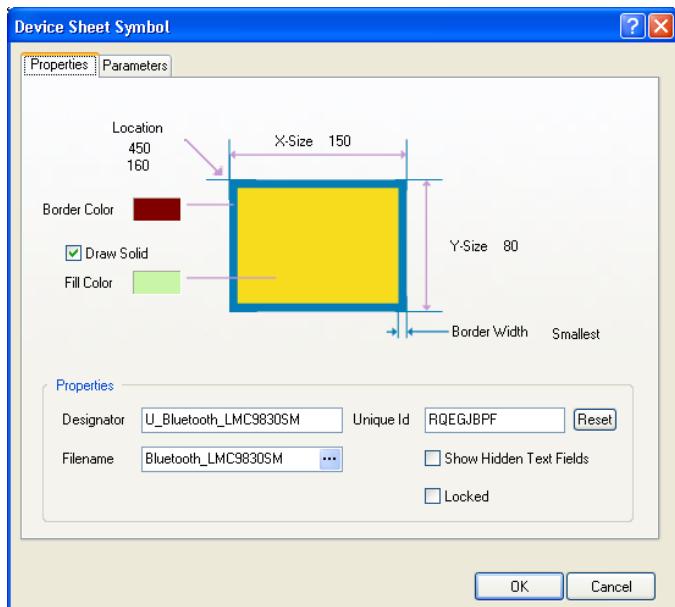
### Editing

The properties of a Device Sheet Symbol object can be modified before, during and after placement. Editing itself falls into two categories - graphical and non-graphical.

The following three methods of non-graphical editing are available:

#### Editing via an associated properties dialog

This method of editing uses the following dialog to modify the properties of a Device Sheet Symbol object.



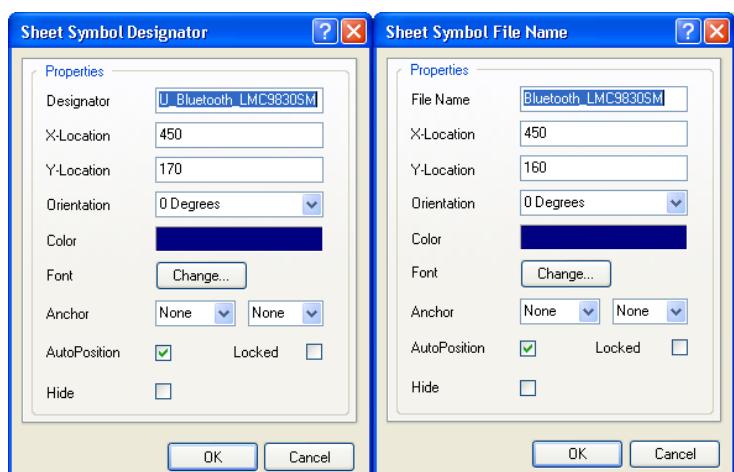
Use the dialog's 'What's This Help' feature to obtain detailed information about each of the options available. Click on the question mark button at the top right of the dialog and then click over a field or option to pop-up information specific to that field or option.

During placement, the *Device Sheet Symbol* dialog can be accessed by pressing the **Tab** key.

After placement, the *Device Sheet Symbol* dialog can be accessed in one of the following ways:

- double-clicking on the placed Device Sheet Symbol object
- selecting the Device Sheet Symbol object and choosing **Properties** from the right-click pop-up menu
- choosing the **Change** command from the **Edit** menu and then clicking once over the placed Device Sheet Symbol object.

The Device Sheet Symbol **Designator** and **Filename** text fields can be formatted independently of the Device Sheet Symbol itself. The corresponding properties dialogs for each - the *Sheet Symbol Designator* and *Sheet Symbol File Name* dialogs respectively - can be accessed using the three methods described above (replacing Device Sheet Symbol with the relevant object whose properties you wish to view/modify).



#### Editing via the SCH Inspector panel

The **SCH Inspector** panel enables you to interrogate and edit the properties of one or more design objects in the active document. Used in conjunction with appropriate filtering, the panel can be used to make changes to multiple objects of the same kind, from one convenient location.

For more information on the **SCH Inspector** panel, press **F1** when the cursor is over the panel.

#### Editing via the SCH List panel

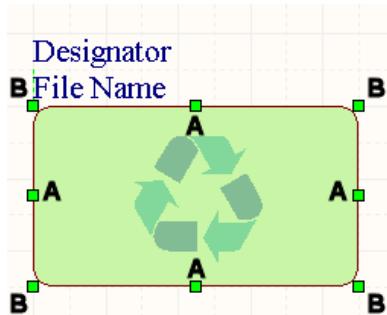
The **SCH List** panel allows you to display design objects from one or more documents in tabular format, enabling you to quickly inspect and modify object attributes. When used in conjunction with the **SCH Filter** panel, it enables you to display just those objects falling under the scope of the active filter - allowing you to target and edit multiple design objects with greater accuracy and efficiency.

For more information on the **SCH List** and **SCH Inspector** panels, press **F1** when the cursor is over a panel.

## Graphical editing

This method of editing allows you to select a placed Device Sheet Symbol object directly in the workspace and change its size, shape or location, graphically.

When a Device Sheet Symbol object is selected, the following editing handles are available:



Click and drag **A** to resize the Device Sheet Symbol in vertical and horizontal directions separately.

Click and drag **B** to resize the Device Sheet Symbol in vertical and horizontal directions simultaneously.

Resizing the Device Sheet Symbol will not affect the absolute positions of any Sheet Entries within it.

Click anywhere on the Device Sheet Symbol - away from editing handles - and drag to reposition it.

The Device Sheet Symbol's Designator and Filename text fields can only be adjusted with respect to their size by changing the size of the font used (accessed through the *Sheet Symbol Designator* and *Sheet Symbol File Name* dialogs, respectively).

Editing handles are not available when either of these objects are selected:



Click anywhere inside the dashed box and drag to reposition the text object as required. The object can be rotated or flipped while dragging:

- Press the **Spacebar** to rotate the text. Rotation is anti-clockwise and in steps of 90°
- Press the **X** or **Y** keys to flip the text along the X-axis or Y-axis respectively.

If the **Enable In-Place Editing** option is enabled on the **Schematic - General** page of the *Preferences* dialog (**Tools** » **Schematic Preferences**), you will be able to edit the name for the Designator or File Name directly in the workspace. Select the text object and then click once to invoke the feature. Type the new name as required and then click away from the text object or press **Enter** to effect the change.

 For more detailed information on using the Device Sheets, refer to the [Using Device Sheets](#) document.

## Notes

The **Filename** property of the Device Sheet Symbol, set on the **Properties** tab of the *Sheet Symbol* dialog, must be set to the file name of the Schematic Sheet that the symbol represents. The `*.SchDoc` extension is not used in the **Filename** for Device Sheets.

---

If you attempt to graphically modify a Device Sheet Symbol object that has its **Locked** property enabled, a dialog will appear asking for confirmation to proceed with the edit.

If the **Protect Locked Objects** option is enabled in the **Schematic – Graphical Editing** page of the *Preferences* dialog (**Tools** » **Schematic Preferences**), and the **Locked** option for this design object is enabled as well then this object cannot be selected or graphically edited.

You will have to double click on this locked object directly and disable the **Locked** property or disable the **Protect Locked Objects** option in the **Schematic – Graphical Editing** Preferences dialog to graphically edit this object.

If you attempt to select locked objects along with other objects, only those objects that are unlocked can be selected and moved as a group if the **Protect Locked Objects** option is enabled. Otherwise if the **Protect Locked Objects** option is disabled you will be prompted for confirmation to proceed with the edit of the objects including locked objects.

## Ellipse



### Description

An ellipse is a non-electrical drawing primitive that can be placed on a schematic sheet. It can be filled or unfilled.

### Availability

Ellipses are available for placement in both Schematic and Schematic Library Editors:

#### Schematic Editor

- Choose **Place » Drawing Tools » Ellipse [P, D, E]** from the main menus
- Click the button on the **Utility Tools** drop-down of the **Utilities** toolbar

#### Schematic Library Editor

- Choose **Place » Ellipse [P, E]** from the main menus
- Click the button on the **Utility Tools** drop-down of the **Utilities** toolbar

### Placement

After launching the command, the cursor will change to a cross-hair and you will enter ellipse placement mode. Placement is made by performing the following sequence of actions:

- click or press **Enter** to anchor the center of the ellipse
- move the cursor to adjust the horizontal radius of the ellipse, then click or press **Enter** to set it
- move the cursor to adjust the vertical radius of the ellipse, then click or press **Enter** to set it and complete placement.

Continue placing further ellipses or right-click or press **Esc** to exit placement mode.

The ellipse object can be rotated or flipped while in placement mode and before the center of the ellipse is anchored:

- Press the **Spacebar** to rotate the ellipse. Rotation is anti-clockwise and in steps of 90°.
- Press the **X** or **Y** keys to flip the ellipse along the X-axis or Y-axis respectively.

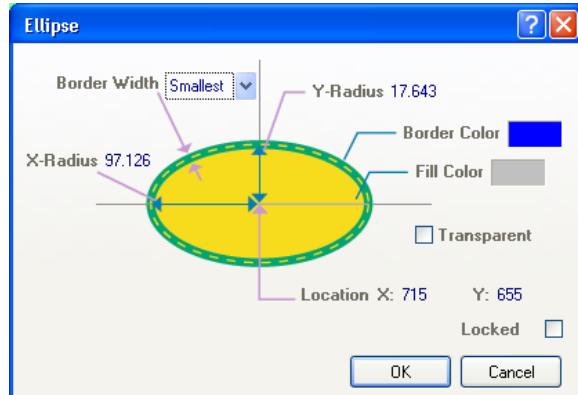
### Editing

The properties of an ellipse object can be modified before, during and after placement. Editing itself falls into two categories - graphical and non-graphical.

The following three methods of non-graphical editing are available:

#### Editing via an associated properties dialog

This method of editing uses the following dialog to modify the properties of an ellipse object.



Use the dialog's 'What's This Help' feature to obtain detailed information about each of the options available. Click on the question mark button at the top right of the dialog and then click over a field or option to pop-up information specific to that field or option.

The *Ellipse* dialog can be accessed prior to entering placement mode, from the **Schematic - Default Primitives** page of the **Preferences** dialog (**Tools » Schematic Preferences**). This allows you to change the default properties for the ellipse object, which will be applied when placing subsequent ellipses.

During placement, the *Ellipse* dialog can be accessed by pressing the **Tab** key.

After placement, the *Ellipse* dialog can be accessed in one of the following ways:

- double-clicking on the placed ellipse object
- selecting the ellipse object and choosing **Properties** from the right-click pop-up menu (Schematic Editor only)
- choosing the **Change** command from the **Edit** menu and then clicking once over the placed ellipse object.

#### Editing via the SCH Inspector panel

The **SCH Inspector** panel enables you to interrogate and edit the properties of one or more design objects in the active document. Used in conjunction with appropriate filtering, the panel can be used to make changes to multiple objects of the same kind, from one convenient location.

 For more information on the **SCH Inspector** panel, press **F1** when the cursor is over the panel.

#### Editing via the SCH List panel

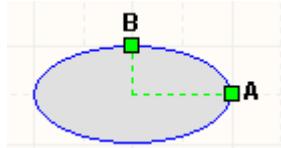
The **SCH List** panel allows you to display design objects from one or more documents in tabular format, enabling you to quickly inspect and modify object attributes. When used in conjunction with the **SCH Filter** panel, it enables you to display just those objects falling under the scope of the active filter - allowing you to target and edit multiple design objects with greater accuracy and efficiency.

 For more information on the **SCH List** and **SCH Inspector** panels, press **F1** when the cursor is over a panel.

#### Graphical editing

This method of editing allows you to select a placed ellipse object directly in the workspace and change its size, shape or location, graphically.

When an ellipse object is selected, the following editing handles are available:



Click and drag **A** to change the horizontal radius.

Click and drag **B** to change the vertical radius.

Click anywhere on the ellipse - away from editing handles - and drag to reposition it. The ellipse can be rotated or flipped while dragging.

If you attempt to graphically modify an ellipse object that has its **Locked** property enabled, a dialog will appear asking for confirmation to proceed with the edit.

If the **Protect Locked Objects** option is enabled in the **Schematic – Graphical Editing** page of the **Preferences** dialog (**Tools » Schematic Preferences**), and the **Locked** option for this design object is enabled as well then this object cannot be selected or graphically edited.

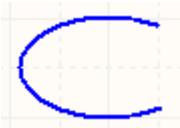
You will have to double click on this locked object directly and disable the **Locked** property or disable the **Protect Locked Objects** option in the **Schematic – Graphical Editing** Preferences dialog to graphically edit this object.

If you attempt to select locked objects along with other objects, only those objects that are unlocked can be selected and moved as a group if the **Protect Locked Objects** option is enabled. Otherwise if the **Protect Locked Objects** option is disabled you will be prompted for confirmation to proceed with the edit of the objects including locked objects.

#### Notes

Any changes made to object properties during placement will cause the default properties for the object to be updated, unless the **Permanent** option - on the **Schematic - Default Primitives** page of the **Preferences** dialog - is enabled. When this option is enabled, changes made will affect only the object being placed and subsequent objects placed during the same placement session.

## Elliptical Arc



### Description

An elliptical arc is a non-electrical drawing primitive. It is essentially an open circular or elliptical curve that can be placed on a schematic sheet.

### Availability

Elliptical arcs are available for placement in both Schematic and Schematic Library Editors:

#### Schematic Editor

- Choose **Place** » **Drawing Tools** » **Elliptical Arc [P, D, I]** from the main menus
- Click the button on the **Utility Tools** drop-down of the **Utilities** toolbar

#### Schematic Library Editor

- Choose **Place** » **Elliptical Arc [P, I]** from the main menus
- Click the button on the **Utility Tools** drop-down of the **Utilities** toolbar

### Placement

After launching the command, the cursor will change to a cross-hair and you will enter elliptical arc placement mode. Placement is made by performing the following sequence of actions:

- click or press **Enter** to anchor the center point of the arc
- move the cursor to adjust the X radius of the arc, then click or press **Enter** to set it
- move the cursor to adjust the Y radius of the arc, then click or press **Enter** to set it
- move the cursor to adjust the first end point of the arc, then click or press **Enter** to anchor it.
- move the cursor to change the position of the arc's other end point, then click or press **Enter** to anchor it and complete placement of the arc.

Continue placing further elliptical arcs, or right-click or press **Esc** to exit placement mode.

The elliptical arc object can be rotated or flipped while in placement mode and before the center point of the arc is anchored:

- Press the **Spacebar** to rotate the elliptical arc. Rotation is anti-clockwise and in steps of 90°.
- Press the **X** or **Y** keys to flip the elliptical along the X-axis or Y-axis respectively.

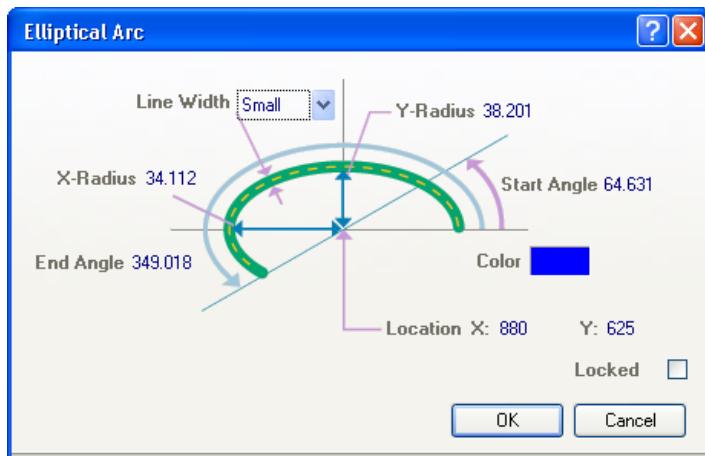
### Editing

The properties of an elliptical arc object can be modified before, during and after placement. Editing itself falls into two categories - graphical and non-graphical.

The following three methods of non-graphical editing are available:

#### Editing via an associated properties dialog

This method of editing uses the following dialog to modify the properties of an elliptical arc object.



Use the dialog's 'What's This Help' feature to obtain detailed information about each of the options available. Click on the question mark button at the top right of the dialog and then click over a field or option to pop-up information specific to that field or option.

The *Elliptical Arc* dialog can be accessed prior to entering placement mode, from the **Schematic - Default Primitives** page of the *Preferences* dialog (**Tools** » **Schematic Preferences**). This allows you to change the default properties for the elliptical arc object, which will be applied when placing subsequent elliptical arcs.

During placement, the *Elliptical Arc* dialog can be accessed by pressing the **Tab** key.

After placement, the *Elliptical Arc* dialog can be accessed in one of the following ways:

- double-clicking on the placed elliptical arc object
- selecting the elliptical arc object and choosing **Properties** from the right-click pop-up menu (Schematic Editor only)
- choosing the **Change** command from the **Edit** menu and then clicking once over the placed elliptical arc object.

#### Editing via the SCH Inspector panel

The **SCH Inspector** panel enables you to interrogate and edit the properties of one or more design objects in the active document. Used in conjunction with appropriate filtering, the panel can be used to make changes to multiple objects of the same kind, from one convenient location.

For more information on the **SCH Inspector** panel, press **F1** when the cursor is over the panel.

#### Editing via the SCH List panel

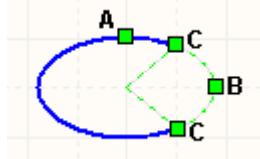
The **SCH List** panel allows you to display design objects from one or more documents in tabular format, enabling you to quickly inspect and modify object attributes. When used in conjunction with the **SCH Filter** panel, it enables you to display just those objects falling under the scope of the active filter - allowing you to target and edit multiple design objects with greater accuracy and efficiency.

For more information on the **SCH List** and **SCH Inspector** panels, press **F1** when the cursor is over a panel.

#### Graphical editing

This method of editing allows you to select a placed elliptical arc object directly in the workspace and change its size, shape or location, graphically.

When an elliptical arc object is selected, the following editing handles are available:



Click and drag **A** to change the vertical radius.

Click and drag **B** to change the horizontal radius.

Click and drag **C** to adjust the end points.

Click anywhere on the elliptical arc - away from editing handles - and drag to reposition it. The elliptical arc can be rotated or flipped while dragging.

If you attempt to graphically modify an elliptical arc object that has its **Locked** property enabled, a dialog will appear asking for confirmation to proceed with the edit.

If the **Protect Locked Objects** option is enabled in the **Schematic – Graphical Editing** page of the *Preferences* dialog (**Tools** » **Schematic Preferences**), and the **Locked** option for this design object is enabled as well then this object cannot be selected or graphically edited.

You will have to double click on this locked object directly and disable the **Locked** property or disable the **Protect Locked Objects** option in the **Schematic – Graphical Editing** Preferences dialog to graphically edit this object.

If you attempt to select locked objects along with other objects, only those objects that are unlocked can be selected and moved as a group if the **Protect Locked Objects** option is enabled. Otherwise if the **Protect Locked Objects** option is disabled you will be prompted for confirmation to proceed with the edit of the objects including locked objects.

## Notes

Any changes made to object properties during placement will cause the default properties for the object to be updated, unless the **Permanent** option - on the **Schematic - Default Primitives** page of the *Preferences* dialog - is enabled. When this option is enabled, changes made will affect only the object being placed and subsequent objects placed during the same placement session.

## Graphic



### Description

A graphic object is a non-electrical drawing primitive. It is essentially a container for an image file that can be imported and placed onto a schematic sheet. The image associated with a graphic object can either be linked or embedded.

### Availability

Graphic objects are available for placement in both Schematic and Schematic Library Editors:

#### Schematic Editor

- Choose **Place** » **Drawing Tools** » **Graphic [P, D, G]** from the Schematic Editor main menus
- Click the button on the **Utility Tools** drop-down of the **Utilities** toolbar

#### Schematic Library Editor

- Choose **Place** » **Graphic [P, G]** from the Schematic Library Editor main menus
- Click the button on the **Utility Tools** drop-down of the **Utilities** toolbar

### Placement

After launching the command, the cursor will change to a cross-hair and you will enter graphic placement mode. Placement is made by performing the following sequence of actions:

- position the cursor and click or press **Enter** to anchor the first corner of the graphic frame, in which the image itself will reside
- move the cursor to adjust the size of the frame and click or press **Enter** to complete frame placement
- the *Open* dialog will appear, from where you can browse to and select the required image. Select the file you wish to insert and press the **Open** button to complete graphic placement.

Continue placing further graphic objects, or right-click or press **Esc** to exit placement mode.

The graphic object can be rotated or flipped while in placement mode and before the first corner of the graphic frame is anchored:

- Press the **Spacebar** to rotate the graphic. Rotation is anti-clockwise and in steps of 90°.
- Press the **X** or **Y** keys to flip the graphic along the X-axis or Y-axis respectively.

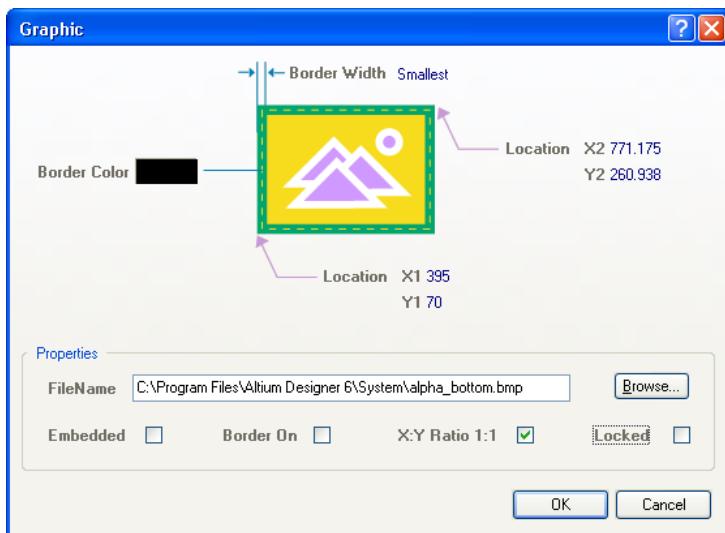
### Editing

The properties of a graphic object can be modified before, during and after placement. Editing itself falls into two categories - graphical and non-graphical.

The following three methods of non-graphical editing are available:

#### Editing via an associated properties dialog

This method of editing uses the following dialog to modify the properties of a graphic object.



Use the dialog's 'What's This Help' feature to obtain detailed information about each of the options available. Click on the question mark button at the top right of the dialog and then click over a field or option to pop-up information specific to that field or option.

The **Graphic** dialog can be accessed prior to entering placement mode, from the **Schematic - Default Primitives** page of the Preferences dialog (**Tools** » **Schematic Preferences**). This allows you to change the default properties for the graphic object, which will be applied when placing subsequent graphic objects.

During placement, the *Graphic* dialog can be accessed by pressing the **Tab** key.

After placement, the *Graphic* dialog can be accessed in one of the following ways:

- double-clicking on the placed graphic object
- selecting the graphic object and choosing **Properties** from the right-click pop-up menu (Schematic Editor only)
- choosing the **Change** command from the **Edit** menu and then clicking once over the placed graphic object.

#### **Editing via the SCH Inspector panel**

The **SCH Inspector** panel enables you to interrogate and edit the properties of one or more design objects in the active document. Used in conjunction with appropriate filtering, the panel can be used to make changes to multiple objects of the same kind, from one convenient location.

For more information on the **SCH Inspector** panel, press **F1** when the cursor is over the panel.

#### **Editing via the SCH List panel**

The **SCH List** panel allows you to display design objects from one or more documents in tabular format, enabling you to quickly inspect and modify object attributes. When used in conjunction with the **SCH Filter** panel, it enables you to display just those objects falling under the scope of the active filter - allowing you to target and edit multiple design objects with greater accuracy and efficiency.

For more information on the **SCH List** and **SCH Inspector** panels, press **F1** when the cursor is over a panel.

#### **Graphical editing**

This method of editing allows you to select a placed graphic object directly in the workspace and change its size, shape or location, graphically.

When a graphic object is selected, the following editing handles are available:



Click and drag **A** to resize the graphic frame in the vertical and horizontal directions simultaneously.

Click and drag **B** to resize the graphic frame in the vertical and horizontal directions separately (provided the **X:Y Ratio 1:1** option in the *Graphic* dialog is disabled).

Click anywhere on the graphic - away from editing handles - and drag to reposition it. The graphic can be rotated or flipped while dragging.

If you attempt to graphically modify an image object that has its **Locked** property enabled, a dialog will appear asking for confirmation to proceed with the edit.

If the **Protect Locked Objects** option is enabled in the **Schematic – Graphical Editing** page of the *Preferences* dialog (**Tools** » **Schematic Preferences**), and the **Locked** option for this design object is enabled as well then this object cannot be selected or graphically edited.

You will have to double click on this locked object directly and disable the **Locked** property or disable the **Protect Locked Objects** option in the **Schematic – Graphical Editing** Preferences dialog to graphically edit this object.

If you attempt to select locked objects along with other objects, only those objects that are unlocked can be selected and moved as a group if the **Protect Locked Objects** option is enabled. Otherwise if the **Protect Locked Objects** option is disabled you will be prompted for confirmation to proceed with the edit of the objects including locked objects.

## Notes

The following image formats are supported: .bmp, .dcx, .dib, .jpg, .pcx, .rle, .tif, .tga, .wmf.

---

To retain the graphic image's original aspect ratio, ensure that the **X:Y Ratio 1:1** option is enabled in the *Graphic* dialog. When this option is enabled, the graphic will be scaled to fit optimally into the frame size specified while maintaining the original aspect ratio of the image. If the option is disabled, the image is stretched to fit exactly into the drawn frame size.

---

A copy of a placed image will only be stored inside the schematic sheet if the corresponding **Embedded** option is enabled in the *Graphic* dialog. If this option is disabled, only a link to the image file will be stored. Care should be taken when using linked images - if the location of the image changes, you will need to update the link accordingly, using the **FileName** field in the *Graphic* dialog.

---

Any changes made to object properties during placement will cause the default properties for the object to be updated, unless the **Permanent** option - on the **Schematic - Default Primitives** page of the *Preferences* dialog - is enabled. When this option is enabled, changes made will affect only the object being placed and subsequent objects placed during the same placement session.

## Harness Connector



### Description

A Harness Connector object is an electrical drawing primitive. It is essentially a container to group various signals together to form a Signal Harness including buses and wires. A Harness Connector is defined by the Harness Type.

### Availability

Harness Connector objects are available for placement in the Schematic editor only:

#### Schematic Editor

- Choose **Place » Harness » Harness Connector [P, H, C]** from the Schematic Editor main menus
- Click the button on the **Wiring** toolbar

### Placement

After launching the command, the cursor will change to a cross-hair and you will enter Harness Connector placement mode. Placement is made by performing the following sequence of actions:

- click or press **Enter** to anchor the tip of the Harness Connector
- move the cursor to adjust the size of the Harness Connector, then click or press **Enter** to anchor the corner and thereby complete placement of the Harness Connector

Continue to place other Harness Connectors or right-click or press **Esc** to exit placement mode.

- Press the **Spacebar** to rotate the Harness Connector. Rotation is anti-clockwise and in steps of 90°
- Press the **X** or **Y** keys to flip the Harness Connector along the X-axis or Y-axis respectively.

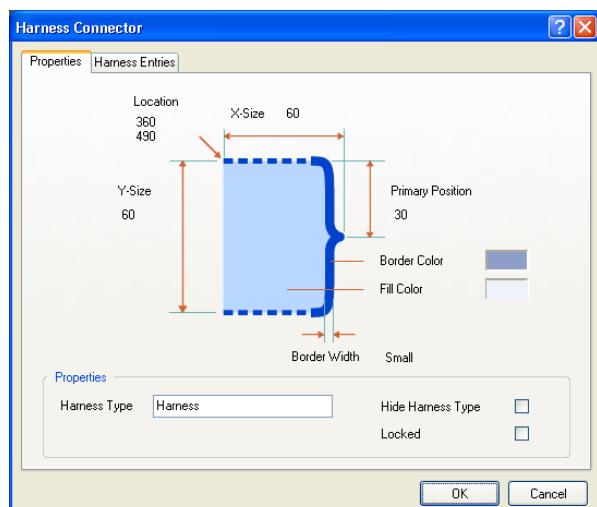
### Editing

The properties of a Harness Connector object can be modified before, during and after placement. Editing itself falls into two categories - graphical and non-graphical.

The following three methods of non-graphical editing are available:

#### Editing via an associated properties dialog

This method of editing uses the following dialog to modify the properties of a Harness Connector object.



Use the dialog's 'What's This Help' feature to obtain detailed information about each of the options available. Click on the question mark button at the top right of the dialog and then click over a field or option to pop-up information specific to that field or option.

The **Harness Connector** dialog can be accessed prior to entering placement mode, from the **Schematic - Default Primitives** page of the **Preferences** dialog (**Tools » Schematic Preferences**). This allows you to change the default properties for the Harness Connector object, which will be applied when placing subsequent Harness Connector objects.

During placement, the **Harness Connector** dialog can be accessed by pressing the **Tab** key.

After placement, the **Harness Connector** dialog can be accessed in one of the following ways:

- double-clicking on the placed Harness Connector object
- selecting the graphic object and choosing **Properties** from the right-click pop-up menu (Schematic Editor only)
- choosing the **Change** command from the **Edit** menu and then clicking once over the placed harness connector object.

#### **Editing via the SCH Inspector panel**

The **SCH Inspector** panel enables you to interrogate and edit the properties of one or more design objects in the active document. Used in conjunction with appropriate filtering, the panel can be used to make changes to multiple objects of the same kind, from one convenient location.

 For more information on the **SCH Inspector** panel, press **F1** when the cursor is over the panel.

#### **Editing via the SCH List panel**

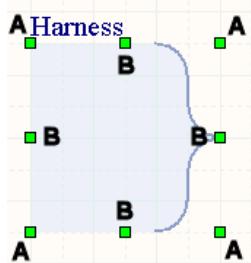
The **SCH List** panel allows you to display design objects from one or more documents in tabular format, enabling you to quickly inspect and modify object attributes. When used in conjunction with the **SCH Filter** panel, it enables you to display just those objects falling under the scope of the active filter - allowing you to target and edit multiple design objects with greater accuracy and efficiency.

 For more information on the **SCH List** and **SCH Inspector** panels, press **F1** when the cursor is over a panel.

#### **Graphical editing**

This method of editing allows you to select a placed Harness Connector object directly in the workspace and change its size, shape or location, graphically.

When a Harness Connector object is selected, the following editing handles are available:



Click and drag **A** to resize the Harness Connector in the vertical and horizontal directions simultaneously.

Click and drag **B** to resize the Harness Connector in the vertical and horizontal directions separately

Click anywhere on the Harness Connector - away from editing handles - and drag to reposition it. The Harness Connector can be rotated or flipped while dragging.

If you attempt to graphically modify a Harness Connector object that has its **Locked** property enabled, a dialog will appear asking for confirmation to proceed with the edit.

If the **Protect Locked Objects** option is enabled in the **Schematic – Graphical Editing** page of the **Preferences** dialog (**Tools » Schematic Preferences**), and the **Locked** option for this design object is enabled as well then this object cannot be selected or graphically edited.

You will have to double click on this locked object directly and disable the **Locked** property or disable the **Protect Locked Objects** option in the **Schematic – Graphical Editing** Preferences dialog to graphically edit this object.

If you attempt to select locked objects along with other objects, only those objects that are unlocked can be selected and moved as a group if the **Protect Locked Objects** option is enabled. Otherwise if the **Protect Locked Objects** option is disabled you will be prompted for confirmation to proceed with the edit of the objects including locked objects.

 For more detailed information on using the Harness objects, refer to the [Using Signal Harnesses](#) document.

## Notes

The **Harness Type** option in the *Harness Connector* dialog defines the type of the Harness Connector. When a Harness Type is used within your project, the corresponding Harness Definition for this Harness Type is referenced. When you place a Harness Connector, you can specify the Harness Type by double clicking on the Harness Connector object and populating the **Harness Type** field in the dialog or clicking directly on the Harness Connector Type field. By default, newly placed Harness Connectors are given the Harness Type **Harness**.

The Harness Type can be hidden from view or moved to save space in your design. Enable the **Hide Harness Type** option in the *Harness Connector* dialog or select the Harness Connector, open the **SCH List** and enable the **Hide Harness Type** option or double click on the Harness Type object and enable the **Hide** option.

Any changes made to object properties during placement will cause the default properties for the object to be updated, unless the **Permanent** option - on the **Schematic - Default Primitives** page of the *Preferences* dialog - is enabled. When this option is enabled, changes made will affect only the object being placed and subsequent objects placed during the same placement session.

## Harness Entry



### Description

A Harness Entry is an electrical design primitive that is placed within a Harness Connector. A Harness Entry is the connection point through which actual nets, buses and Signal Harnesses are combined to form a higher level Signal Harness.

### Availability

Harness Entries are available for placement in the Schematic Editor only. Use one of the following methods to access the placement command:

- choose **Place » Harness » Harness Entry [P, H, E]** from the main menus
- click the button on the **Wiring** toolbar.

### Placement

After launching the command, the cursor will change to a cross-hair and you will enter Harness Entry placement mode. Placement is made by performing the following sequence of actions:

- the Harness Connector is implicitly chosen by the position for the new Harness Entry by the mouse on the schematic sheet (this sheet needs to have at least one Harness Connector).
- move the cursor to adjust the position of the Harness Entry in relation to any edge of the Harness Connector, then click or press **Enter** to anchor the Harness Entry and complete placement.

Continue placing further Harness Entries, or right-click or press **Esc** to exit placement mode.

### Editing

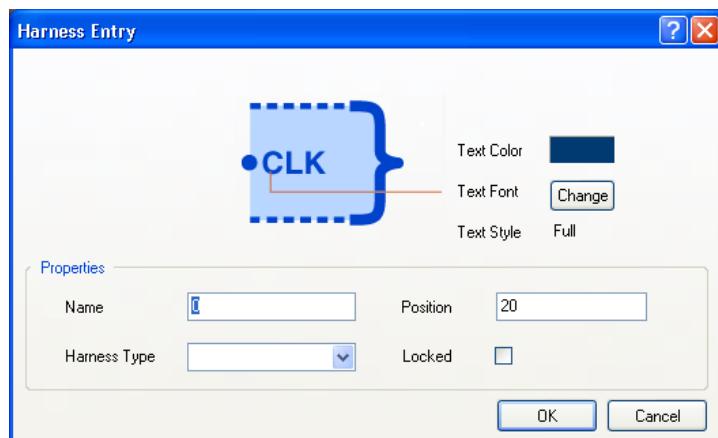
The properties of a Harness Entry object can be modified before, during and after placement. Editing itself falls into two categories - graphical and non-graphical.

The following three methods of non-graphical editing are available:

#### Editing via an associated properties dialog

This method of editing uses the following dialog to modify the properties of a Harness Entry object.

Use the dialog's 'What's This Help' feature to obtain detailed information about each of the options available. Click on the question mark button at the top right of the dialog and then click over a field or option to pop-up information specific to that field or option.



The *Harness Entry* dialog can be accessed prior to entering placement mode, from the **Schematic - Default Primitives** page of the *Preferences* dialog (**Tools** » **Schematic Preferences**). This allows you to change the default properties for the Harness Entry object, which will be applied when placing subsequent Harness Entries.

During placement, the *Harness Entry* dialog can be accessed by pressing the **Tab** key.

After placement, the *Harness Entry* dialog can be accessed in one of the following ways:

- double-clicking on the placed Harness Entry object
- selecting the Harness Entry object and choosing **Properties** from the right-click pop-up menu
- choosing the **Change** command from the **Edit** menu and then clicking once over the placed Harness Entry object.

#### Editing via the SCH Inspector panel

The **SCH Inspector** panel enables you to interrogate and edit the properties of one or more design objects in the active document. Used in conjunction with appropriate filtering, the panel can be used to make changes to multiple objects of the same kind, from one convenient location.

 For more information on the **SCH Inspector** panel, press **F1** when the cursor is over the panel.

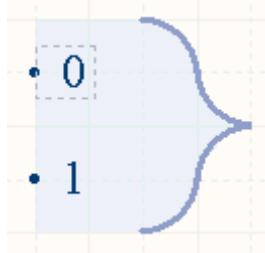
#### Editing via the SCH List panel

The **SCH List** panel allows you to display design objects from one or more documents in tabular format, enabling you to quickly inspect and modify object attributes. When used in conjunction with the **SCH Filter** panel, it enables you to display just those objects falling under the scope of the active filter - allowing you to target and edit multiple design objects with greater accuracy and efficiency.

 For more information on the **SCH List** and **SCH Inspector** panels, press **F1** when the cursor is over a panel.

#### Graphical editing

This method of editing allows you to select a placed Harness Entry object directly in the workspace and change its location graphically. The Harness Connector itself gets resized automatically when you attempt to move the existing Harness Entry beyond the current extends of the Harness Connector.



Click anywhere inside the dashed box and drag to reposition the Harness Entry within the Harness Connector as required. You can move a Harness Entry object outside of the Harness Connector and into another Harness Connector by holding down the **Ctrl** key while dragging the object.

If the **Enable In-Place Editing** option is enabled on the **Schematic - General** page of the *Preferences* dialog (**Tools** » **Schematic Preferences**), you will be able to edit the name for the Harness Entry directly in the workspace. Select the Harness Entry object and then click once to invoke the feature. Type the new name as required and then click away from the Harness Entry object or press **Enter** to effect the change.

 For more detailed information on using the Harness objects, refer to the [Using Signal Harnesses](#) document.

#### Notes

A Harness Entry can be connected directly to a wire, a bus or a Signal Harness. The **Harness Type** field in the *Harness Entry* dialog is used in special cases.

Should you need to negate (include a bar over the top of) a Harness Entry name, this can be done in one of two ways:

- Include a backslash character after each character in the name (e.g. `\N\A\B\L\E\`)
- Enable the **Single '\ Negation** option on the **Schematic - Graphical Editing** page of the *Preferences* dialog, then include one backslash character at the start of the name (e.g. `\ENABLE`).

Any changes made to object properties during placement will cause the default properties for the object to be updated, unless the **Permanent** option - on the **Schematic - Default Primitives** page of the *Preferences* dialog - is enabled. When this option is enabled, changes made will affect only the object being placed and subsequent objects placed during the same placement session.

If you attempt to graphically modify a Harness Entry object that has its **Locked** property enabled, a dialog will appear asking for confirmation to proceed with the edit.

If the **Protect Locked Objects** option is enabled in the **Schematic – Graphical Editing** page of the *Preferences* dialog (**Tools** » **Schematic Preferences**), and the **Locked** option for this design object is enabled as well then this object cannot be selected or graphically edited.

You will have to double click on this locked object directly and disable the **Locked** property or disable the **Protect Locked Objects** option in the **Schematic – Graphical Editing** Preferences dialog to graphically edit this object.

If you attempt to select locked objects along with other objects, only those objects that are unlocked can be selected and moved as a group if the **Protect Locked Objects** option is enabled. Otherwise if the **Protect Locked Objects** option is disabled you will be prompted for confirmation to proceed with the edit of the objects including locked objects.

## IEEE Symbols



### Description

IEEE symbols are non-electrical drawing primitives. They are used for representing logic functions or devices. These symbols enable users to understand the logic characteristics of these functions or devices without requiring specific knowledge of their internal characteristics.

### Availability

IEEE Symbol objects are available for placement in the Schematic Library Editor only. A total of 33 symbols are available, the associated placement commands for which are located:

- on the **Place » IEEE Symbols [P, S]** sub-menu (accessed from the Schematic Library Editor main menus)
- on the **IEEE Symbols** drop-down of the **Utilities** toolbar.

### IEEE Symbol listing

The following table lists each of the 33 IEEE symbol objects available for placement within a Schematic Library document.

Symbol	Description
	Active Low Input
	Active Low Output
	Analog Signal In
	AND Gate
	Bidirectional Signal Flow
	Clock
	Delay
	Digital Signal In
	Dot
	Group Binary
	Open Collector Pull-up
	Open Emitter
	Open Emitter Pull-up
	Open Output
	OR Gate

Symbol	Description
	Group Line
	High Current
	HiZ
	Invertor
	Left To Right Signal Flow
	Less Than Or Equal To
	Not Logic Connection
	Open Collector
	Greater Than Or Equal To
	Postponed Output
	Pulse
	Right To Left Signal Flow
	Schmitt
	Shift Left
	Shift Right

	Pi Symbol
	XOR Gate

	Sigma
---	-------

## Placement

After launching a command, the cursor will change to a cross-hair and you will enter IEEE symbol placement mode. The chosen IEEE symbol will appear floating on the cursor. Position the cursor as required and click or press **Enter** to place the symbol.

Continue placing further symbols or right-click or press **Esc** to exit placement mode.

Press the **Spacebar** while in placement mode to rotate the symbol. Rotation is anti-clockwise and in steps of 90°.

Press the **X** or **Y** keys while in placement mode to flip the symbol along the X-axis or Y-axis respectively.

Press the **+** and **-** keys (on the numeric keypad) to enlarge or shrink the symbol as you place it.

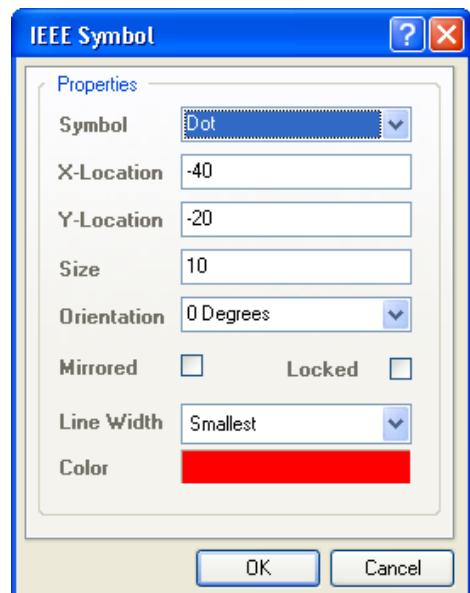
## Editing

The properties of an IEEE symbol object can be modified before, during and after placement. Editing itself falls into two categories - graphical and non-graphical.

The following three methods of non-graphical editing are available:

### Editing via an associated properties dialog

This method of editing uses the following dialog to modify the properties of an IEEE symbol object.



Use the dialog's 'What's This Help' feature to obtain detailed information about each of the options available. Click on the question mark button at the top right of the dialog and then click over a field or option to pop-up information specific to that field or option.

The *IEEE Symbol* dialog can be accessed prior to entering placement mode, from the **Schematic - Default Primitives** page of the *Preferences* dialog (**Tools** » **Schematic Preferences**). This allows you to change the default properties for the IEEE symbol object, which will be applied when placing subsequent IEEE symbols.

During placement, the *IEEE Symbol* dialog can be accessed by pressing the **Tab** key.

After placement, the *IEEE Symbol* dialog can be accessed in one of the following ways:

- double-clicking on the placed IEEE symbol object
- choosing the **Change** command from the **Edit** menu and then clicking once over the placed IEEE symbol object.

### Editing via the SCH Inspector panel

The **SCH Inspector** panel enables you to interrogate and edit the properties of one or more design objects in the active document. Used in conjunction with appropriate filtering, the panel can be used to make changes to multiple objects of the same kind, from one convenient location.

 For more information on the **SCH Inspector** panel, press **F1** when the cursor is over the panel.

### Editing via the SCH List panel

The **SCH List** panel allows you to display design objects from one or more documents in tabular format, enabling you to quickly inspect and modify object attributes. When used in conjunction with the **SCH Filter** panel, it enables you to display just those objects falling under the scope of the active filter - allowing you to target and edit multiple design objects with greater accuracy and efficiency.

 For more information on the **SCH List** and **SCH Inspector** panels, press **F1** when the cursor is over a panel.

## Graphical editing

This method of editing allows you to select a placed IEEE symbol object directly in the workspace and change its location or orientation, graphically. IEEE symbols are fixed with respect to their shape and can be resized only through use of the relevant keyboard shortcuts. As such, editing handles are not available when the IEEE symbol object is selected:



Click anywhere inside the dashed box and drag to reposition the IEEE symbol as required. The IEEE symbol can be rotated, flipped or resized while dragging.

If you attempt to graphically modify an IEEE symbol object that has its **Locked** property enabled, a dialog will appear asking for confirmation to proceed with the edit.

If the **Protect Locked Objects option** is enabled in the **Schematic – Graphical Editing** page of the *Preferences* dialog (**Tools** » **Schematic Preferences**), and the **Locked** option for this design object is enabled as well then this object cannot be selected or graphically edited.

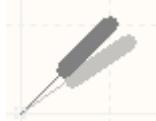
You will have to double click on this locked object directly and disable the **Locked** property or disable the **Protect Locked Objects** option in the **Schematic – Graphical Editing** Preferences dialog to graphically edit this object.

If you attempt to select locked objects along with other objects, only those objects that are unlocked can be selected and moved as a group if the **Protect Locked Objects** option is enabled. Otherwise if the **Protect Locked Objects** option is disabled you will be prompted for confirmation to proceed with the edit of the objects including locked objects.

### Notes

Any changes made to object properties during placement will cause the default properties for the object to be updated, unless the **Permanent** option - on the **Schematic - Default Primitives** page of the *Preferences* dialog - is enabled. When this option is enabled, changes made will affect only the object being placed and subsequent objects placed during the same placement session.

## Instrument Probe



### Description

An instrument probe is a design directive. It instructs the system to connect the net to which it is attached directly to the monitoring instrument (e.g. a logic analyzer) without having to explicitly wire that net up through the design hierarchy to the sheet with the instrument on it.

### Availability

Probes are available for placement in the Schematic Editor only, by choosing **Place » Directives » Instrument Probe [P, V, I]** from the main menus.

### Placement

After launching the command, the cursor will change to a cross-hair and you will enter instrument probe placement mode.

Position the cursor over a wire or other net object and click or press **Enter** to effect placement.

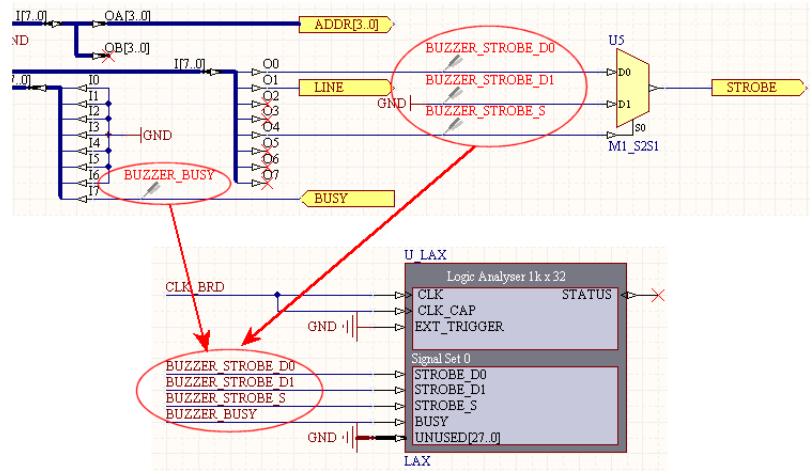
Continue placing further instrument probe directives or right-click or press **Esc** to exit placement mode.

Press the **Spacebar** while in placement mode to rotate the instrument probe directive. Rotation is anti-clockwise and in steps of 90°.

Press the **X** or **Y** keys while in placement mode to flip the instrument probe directive along the X-axis or Y-axis respectively.

### Connecting to the Instrument

After placing an instrument probe directive at the point of interest, you need to define a value for its **InstrumentProbe** parameter. Enter a meaningful name for the probe point, for example the name of the associated net or the particular signal being monitored. Then, connect a wire to the required input of the monitoring instrument and attach a net label to the wire, the name of which is the same name you have defined for the **InstrumentProbe** parameter.



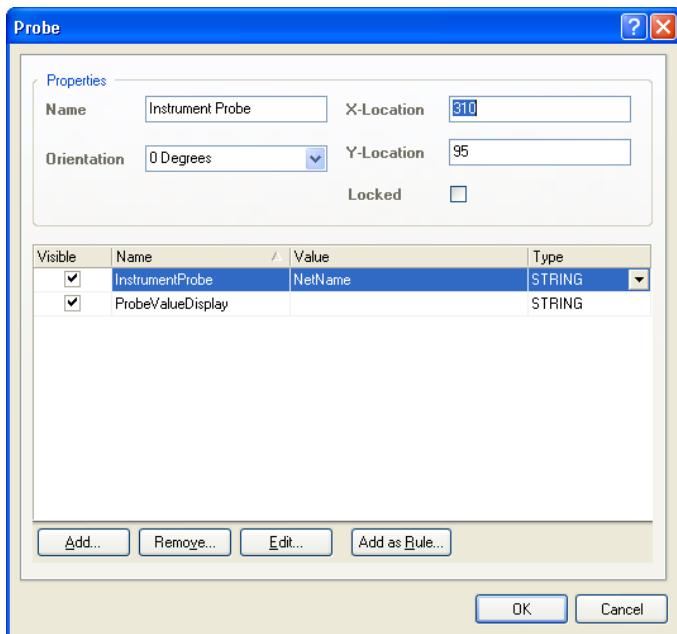
### Editing

The properties of an instrument probe directive can be modified before, during and after placement. Editing itself falls into two categories - graphical and non-graphical.

The following three methods of non-graphical editing are available:

#### Editing via an associated properties dialog

This method of editing uses the following dialog to modify the properties of an instrument probe directive.



Use the dialog's 'What's This Help' feature to obtain detailed information about each of the options available. Click on the question mark button at the top right of the dialog and then click over a field or option to pop-up information specific to that field or option.

The *Probe* dialog can be accessed prior to entering placement mode, from the **Schematic - Default Primitives** page of the **Preferences** dialog (**Tools » Schematic Preferences**). This allows you to change the default properties for the instrument probe directive, which will be applied when placing subsequent instrument probe directives.

During placement, the *Probe* dialog can be accessed by pressing the **Tab** key.

After placement, the *Probe* dialog can be accessed in one of the following ways:

- double-clicking on the placed instrument probe directive
- selecting the instrument probe directive and choosing **Properties** from the right-click pop-up menu
- choosing the **Change** command from the **Edit** menu and then clicking once over the placed instrument probe directive.

#### Editing via the SCH Inspector panel

The **SCH Inspector** panel enables you to interrogate and edit the properties of one or more design objects in the active document. Used in conjunction with appropriate filtering, the panel can be used to make changes to multiple objects of the same kind, from one convenient location.

For more information on the **SCH Inspector** panel, press **F1** when the cursor is over the panel.

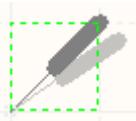
#### Editing via the SCH List panel

The **SCH List** panel allows you to display design objects from one or more documents in tabular format, enabling you to quickly inspect and modify object attributes. When used in conjunction with the **SCH Filter** panel, it enables you to display just those objects falling under the scope of the active filter - allowing you to target and edit multiple design objects with greater accuracy and efficiency.

For more information on the **SCH List** and **SCH Inspector** panels, press **F1** when the cursor is over a panel.

#### Graphical editing

This method of editing allows you to select a placed instrument probe directive directly in the workspace and change its location graphically. Instrument probe directives are fixed with respect to their size and shape. As such, editing handles are not available when the instrument probe directive is selected:



Click anywhere inside the dashed box and drag to reposition the instrument probe directive as required. The instrument probe directive can be rotated or flipped while dragging.

If you attempt to graphically modify an instrument object that has its **Locked** property enabled, a dialog will appear asking for confirmation to proceed with the edit.

If the **Protect Locked Objects** option is enabled in the **Schematic – Graphical Editing** page of the *Preferences* dialog (**Tools** » **Schematic Preferences**), and the **Locked** option for this design object is enabled as well then this object cannot be selected or graphically edited.

You will have to double click on this locked object directly and disable the **Locked** property or disable the **Protect Locked Objects** option in the **Schematic – Graphical Editing** Preferences dialog to graphically edit this object.

If you attempt to select locked objects along with other objects, only those objects that are unlocked can be selected and moved as a group if the **Protect Locked Objects** option is enabled. Otherwise if the **Protect Locked Objects** option is disabled you will be prompted for confirmation to proceed with the edit of the objects including locked objects.

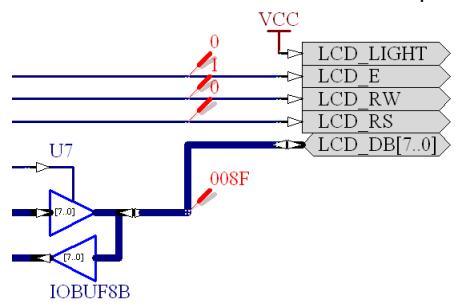
## Notes

The instrument probe directive is essentially the same as the probe directive, but with the additional `InstrumentProbe` parameter. It is this parameter that turns it from being a simple probe into a means of monitoring any point in a design – connecting that point directly to the input line of a monitoring instrument.

If the instrument probe directive is placed on a net that connects to an FPGA pin, it can be used to monitor the status of that pin directly on the schematic sheet, as well as being used as an input source to a monitoring instrument. The former is achieved using the directive's `ProbeValueDisplay` parameter.

**Note:** For this feature to function, the source FPGA design must be downloaded to the physical FPGA device and the associated **JTAG Viewer** panel for that device must be open and remain open.

The **JTAG Viewer** panel for a physical device is accessed by clicking the **JTAG Viewer Panel** button on the associated instrument panel for that device. The latter is loaded into the **Instrument Rack – Hard Devices** panel upon double-clicking the entry for the device, in the Hard Devices chain of the **Devices** view.



When an instrument probe is attached to a bus, the entire bus is taken up to the top-level sheet, irrespective of the name you assign to the `InstrumentProbe` parameter. When you add a net label to the input for the monitoring instrument, you must define the bus width required. For example, you may have attached an instrument probe to a bus with identifier `Port1_Out[7..0]` on a lower level sheet. The value for the `InstrumentProbe` parameter could be set to `Port1_Out`. The entire bus will be connected up to the sheet with the monitoring device (e.g. a LAX). Should you wish to wire up the entire bus as an input signal to the device, you would place a bus to the required input and add a net label of `Port1_Out[7..0]`. If you only wanted a particular signal or range of signals from the bus, you can define the width required in the attached net label.

Any changes made to object properties during placement will cause the default properties for the object to be updated, unless the **Permanent** option - on the **Schematic - Default Primitives** page of the *Preferences* dialog - is enabled. When this option is enabled, changes made will affect only the object being placed and subsequent objects placed during the same placement session.

## Line



### Description

A line is a non-electrical drawing primitive. Lines are used for adding reference information to a document, such as building graphical symbols, custom sheet borders and title blocks.

### Availability

Line objects are available for placement in both Schematic and Schematic Library Editors:

#### Schematic Editor

- Choose **Place » Drawing Tools » Line [P, D, L]** from the main menus
- Click the button on the **Utility Tools** drop-down of the **Utilities** toolbar

#### Schematic Library Editor

- Choose **Place » Line [P, L]** from the main menus
- Click the button on the **Utility Tools** drop-down of the **Utilities** toolbar

### Placement

After launching the command, the cursor will change to a cross-hair and you will enter line placement mode. Placement is made by performing the following sequence of actions:

- click or press **Enter** to anchor the starting point for the line
- position the cursor and click or press **Enter** to anchor a series of vertex points that define the shape of the line
- after placing the final vertex point, right-click or press **Esc** to complete placement of the line.

Continue placing further line objects, or right-click or press **Esc** to exit placement mode.

Use the **Backspace** or **Delete** keys to remove the last line segment placed. If you do remove segments in this way, you must click to place a final segment, otherwise right-clicking will place the line as it was, with all deleted segments reinstated.

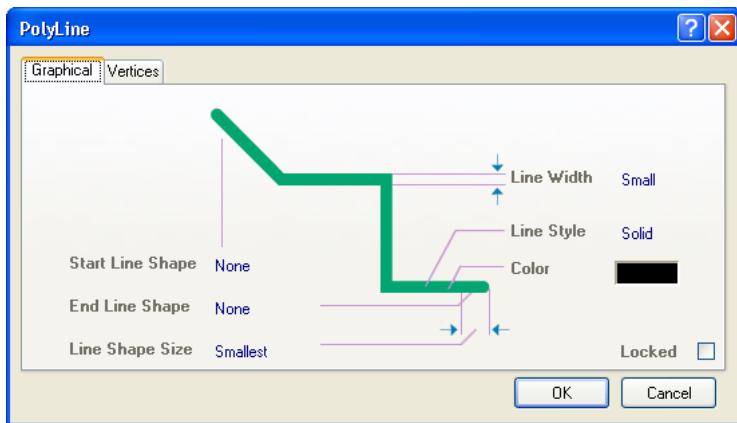
### Editing

The properties of a line object can be modified before, during and after placement. Editing itself falls into two categories - graphical and non-graphical.

The following three methods of non-graphical editing are available:

#### Editing via an associated properties dialog

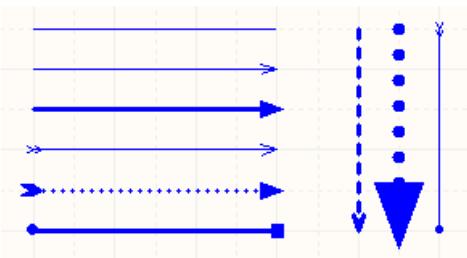
This method of editing uses the following dialog to modify the properties of a line object.



Use the dialog's 'What's This Help' feature to obtain detailed information about each of the options available. click on the question mark button at the top right of the dialog and then click over a field or option to pop-up information specific to that field or option.

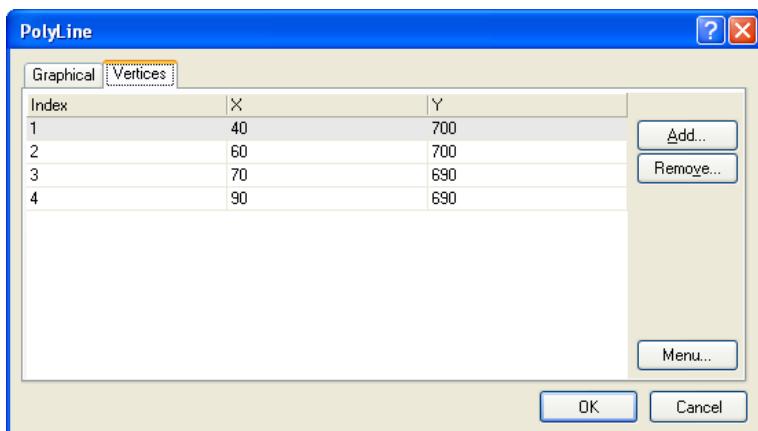
### Adding a Touch of Style

The **Polyline** dialog provides several options with which to change the style of a line. For example use the **Line Style** option to change between a solid, dotted or dashed line. By using the **Start Line Shape** and **Stop Line Shape** options, you can further style the line by adding various arrow head and tail shapes.



### Editing Vertices

The **PolyLine** dialog provides a **Vertices** tab, from where you can edit the individual vertices of the currently selected line object as required.



The main region of the tab lists all of the vertex points currently defined for the line. You can add new vertices to the line, edit the coordinates of existing vertices, or remove selected vertices altogether.

Click the **Menu** button or right-click within the main list region to access a pop-up menu containing the following commands:

- **Edit** - right click on a coordinate cell (X or Y) for a vertex and use this command to edit the value in that cell. Alternatively, click directly on the cell
- **Add** - use this command to add a new vertex point. The new vertex will be added below the currently focused vertex entry (as distinguished by a dotted outline around a cell in its row) and will initially have the same coordinates as the focused entry
- **Remove** - use this command to remove the currently selected vertex entries in the list. This command will be unavailable if there are only two vertices present for the line
- **Copy** - use this command to copy the content of the selected cells in the list to the clipboard (alternatively use **Ctrl+C**)
- **Paste** - use this command to paste the content of the clipboard into the list, starting at the selected cell (alternatively use **Ctrl+V**)
- **Select All** - use this command to quickly select the entire grid contents of the list
- **Select Column** - use this command to quickly select the entire column in which the currently focused cell resides

- **Move Up** - use this command to move the selected vertex upward in the list
- **Move Down** - use this command to move the selected vertex downward in the list
- **Move Line By XY** - use this command to move the entire line object. The Move Line By dialog will appear, from where you can enter the increment value to be applied to each vertex point's X and Y coordinates.

### Dialog access

The **PolyLine** dialog can be accessed prior to entering placement mode, from the **Schematic - Default Primitives** page of the **Preferences** dialog (**Tools » Schematic Preferences**). This allows you to change the default properties for the line object, which will be applied when placing subsequent lines.

During placement, the **PolyLine** dialog can be accessed by pressing the **Tab** key.

After placement, the **PolyLine** dialog can be accessed in one of the following ways:

- double-clicking on the placed line object
- selecting the line object and choosing **Properties** from the right-click pop-up menu (Schematic Editor only)
- choosing the **Change** command from the **Edit** menu and then clicking once over the placed line object.

### Editing via the SCH Inspector panel

The **SCH Inspector** panel enables you to interrogate and edit the properties of one or more design objects in the active document. Used in conjunction with appropriate filtering, the panel can be used to make changes to multiple objects of the same kind, from one convenient location.

 For more information on the **SCH Inspector** panel, press **F1** when the cursor is over the panel.

### Editing via the SCH List panel

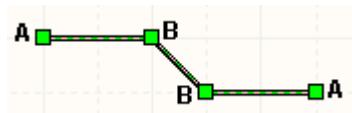
The **SCH List** panel allows you to display design objects from one or more documents in tabular format, enabling you to quickly inspect and modify object attributes. When used in conjunction with the **SCH Filter** panel, it enables you to display just those objects falling under the scope of the active filter - allowing you to target and edit multiple design objects with greater accuracy and efficiency.

 For more information on the **SCH List** and **SCH Inspector** panels, press **F1** when the cursor is over a panel.

### Graphical editing

This method of editing allows you to select a placed line object directly in the workspace and change its size and/or shape, graphically.

When a line object is selected, the following editing handles are available:



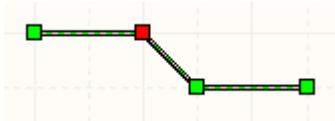
Click and drag **A** to adjust the end points.

Click and drag **B** to move the vertex point of a line.

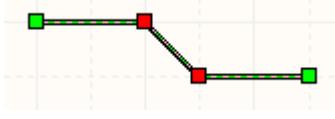
Click and drag near the center of a line segment to "grab" that segment and reposition it. The end points and other vertices will remain anchored.

Right-click on a vertex point and choose the **Edit Line Vertex n** command to access the **Vertices** tab of the **PolyLine** dialog, with the entry for the **n**th vertex selected ready for editing.

With the line selected, click on a vertex or segment to individually select that vertex or segment. This line 'sub-selection' is distinguished by the associated editing handles becoming red in color.



Individual vertex sub-selection.



Individual segment sub-selection.

The associated vertex (or vertices for a segment) can then be edited directly using the **SCH Inspector** or **SCH List** panels, with any changes appearing immediately on the schematic.

If you attempt to graphically modify a line object that has its **Locked** property enabled, a dialog will appear asking for confirmation to proceed with the edit.

If the **Protect Locked Objects** option is enabled in the **Schematic – Graphical Editing** page of the *Preferences* dialog (**Tools** » **Schematic Preferences**), and the **Locked** option for this design object is enabled as well then this object cannot be selected or graphically edited.

You will have to double click on this locked object directly and disable the **Locked** property or disable the **Protect Locked Objects** option in the **Schematic – Graphical Editing** Preferences dialog to graphically edit this object.

If you attempt to select locked objects along with other objects, only those objects that are unlocked can be selected and moved as a group if the **Protect Locked Objects** option is enabled. Otherwise if the **Protect Locked Objects** option is disabled you will be prompted for confirmation to proceed with the edit of the objects including locked objects.

## Notes

A line is purely a drawing object. To make an electrical connection between points in your schematic, use the wire object.

---

When placing a line, various line placement modes are available: Any Angle, 90 Degree Start, 90 Degree End, 45 Degree Start and 45 Degree End. The mode specifies how corners are created when placing lines and the angles at which lines can be placed.

---

When placing a line, press the **Spacebar** to cycle through the various placement modes. You can change modes at any time during line placement.

---

In all modes other than Any Angle, the line segment attached to the cursor is a "look ahead" segment. The segment you are actually placing precedes this look ahead segment.

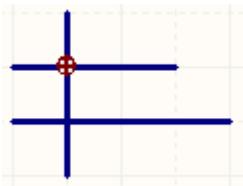
---

When launching the command to place a line, the initial placement mode is dependant on the placement mode last used when previously placing lines or wires. Therefore, if the last wire you placed used 90 Degree Start, then the next wire or line you place will have 90 Degree Start as the initial placement mode. If you last placed a wire using Auto Wire mode, placement of a subsequent line will be confined to Any Angle mode only. You will need to place a wire in any mode other than Auto Wire in order for line placement to have all placement modes fully restored and functional.

---

Any changes made to object properties during placement will cause the default properties for the object to be updated, unless the **Permanent** option - on the **Schematic - Default Primitives** page of the *Preferences* dialog - is enabled. When this option is enabled, changes made will affect only the object being placed and subsequent objects placed during the same placement session.

## Manual Junction



### Description

A junction is an electrical design primitive. It is a small circular object used to logically join intersecting wires or buses on a schematic sheet. A manual junction allows you to create a connection between crossed wires or buses, by placing the junction at the crossing point. This is different from a Compiler generated junction, which is automatically inserted when two wires/buses are connected in a T-type fashion, or when a wire/bus connects orthogonally to a pin or power port/bus power port.

### Availability

Manual junctions are available for placement in the Schematic Editor only, by choosing **Place » Manual Junction [P, J]** from the Schematic Editor main menus.

### Placement

After launching the command, the cursor will change to a cross-hair and you will enter junction placement mode. Click or press **Enter** to place a junction at the cursor position.

Continue placing junctions or right-click or press **Esc** to exit placement mode.

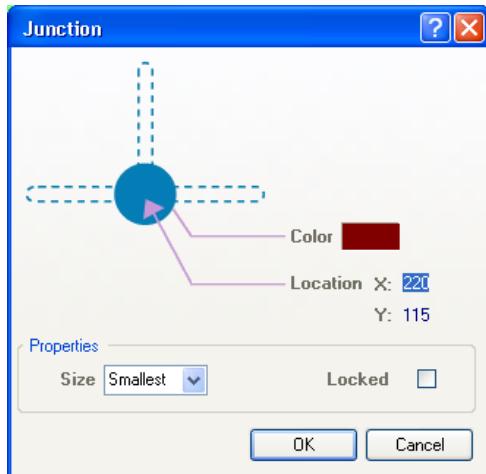
### Editing

The properties of a manual junction object can be modified before, during and after placement. Editing itself falls into two categories - graphical and non-graphical.

The following three methods of non-graphical editing are available:

#### Editing via an associated properties dialog

This method of editing uses the following dialog to modify the properties of a manual junction object.



Use the dialog's 'What's This Help' feature to obtain detailed information about each of the options available. Click on the question mark button at the top right of the dialog and then click over a field or option to pop-up information specific to that field or option.

The *Junction* dialog can be accessed prior to entering placement mode, from the **Schematic - Default Primitives** page of the **Preferences** dialog (**Tools » Schematic Preferences**). This allows you to change the default properties for the manual junction object, which will be applied when placing subsequent manual junctions.

During placement, the *Junction* dialog can be accessed by pressing the **Tab** key.

After placement, the *Junction* dialog can be accessed in one of the following ways:

- double-clicking on the placed manual junction object

- selecting the manual junction object and choosing **Properties** from the right-click pop-up menu
- choosing the **Change** command from the **Edit** menu and then clicking once over the placed manual junction object.

### Editing via the SCH Inspector panel

The **SCH Inspector** panel enables you to interrogate and edit the properties of one or more design objects in the active document. Used in conjunction with appropriate filtering, the panel can be used to make changes to multiple objects of the same kind, from one convenient location.

 For more information on the **SCH Inspector** panel, press **F1** when the cursor is over the panel.

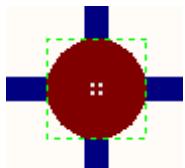
### Editing via the SCH List panel

The **SCH List** panel allows you to display design objects from one or more documents in tabular format, enabling you to quickly inspect and modify object attributes. When used in conjunction with the **SCH Filter** panel, it enables you to display just those objects falling under the scope of the active filter - allowing you to target and edit multiple design objects with greater accuracy and efficiency.

 For more information on the **SCH List** and **SCH Inspector** panels, press **F1** when the cursor is over a panel.

### Graphical editing

This method of editing allows you to select a placed manual junction object directly in the workspace and change its location graphically. Manual junctions can only be adjusted with respect to their size through the *Junction* dialog. As such, editing handles are not available when the manual junction object is selected:



Click anywhere inside the dashed box and drag to reposition the manual junction as required.

If you attempt to graphically modify a junction object that has its **Locked** property enabled, a dialog will appear asking for confirmation to proceed with the edit.

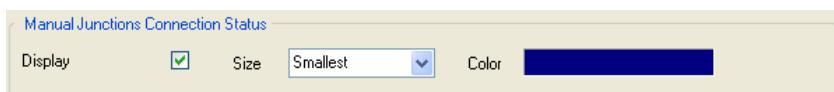
If the **Protect Locked Objects** option is enabled in the **Schematic – Graphical Editing** page of the *Preferences* dialog (**Tools** » **Schematic Preferences**), and the **Locked** option for this design object is enabled as well then this object cannot be selected or graphically edited.

You will have to double click on this locked object directly and disable the **Locked** property or disable the **Protect Locked Objects** option in the **Schematic – Graphical Editing** Preferences dialog to graphically edit this object.

If you attempt to select locked objects along with other objects, only those objects that are unlocked can be selected and moved as a group if the **Protect Locked Objects** option is enabled. Otherwise if the **Protect Locked Objects** option is disabled you will be prompted for confirmation to proceed with the edit of the objects including locked objects.

### Notes

Display of manual junctions on the schematic sheet can be controlled from the **Schematic - Compiler** page of the *Preferences* dialog (**Tools** » **Schematic Preferences**). Additional options provide control over junction size and color:



Any changes made to object properties during placement will cause the default properties for the object to be updated, unless the **Permanent** option - on the **Schematic - Default Primitives** page of the *Preferences* dialog - is enabled. When this option is enabled, changes made will affect only the object being placed and subsequent objects placed during the same placement session.

## Net Label

### NetLabel1

#### Description

A wire is an electrical design primitive. Wires that join electrical objects (such as component pins and ports) create what is called a Net. These nets will be assigned system-allocated name unless there is a net label placed on the wire, overriding the system-allocated name with the user-specified Net name. As well as being used to name a net, Net labels allow you to connect points on a schematic without actually physically wiring them together.

#### Availability

Net labels are available for placement in the Schematic Editor only. Use one of the following methods to access the placement command:

- choose **Place » Net Label [P, N]** from the Schematic Editor main menus
- click the **Net** button on the **Wiring** toolbar.

#### Placement

After launching the command, the cursor will change to a cross-hair and you will enter net label placement mode. The label will appear floating on the cursor. Position the label so that its bottom-left corner touches the object to which you want to assign it and click or press **Enter** to place the label.

Continue placing further net labels, or right-click or press **Esc** to exit placement mode.

Press the **Spacebar** while in placement mode to rotate the net label. Rotation is anti-clockwise and in steps of 90°.

Press the **X** or **Y** keys while in placement mode to flip the net label along the X-axis or Y-axis respectively.

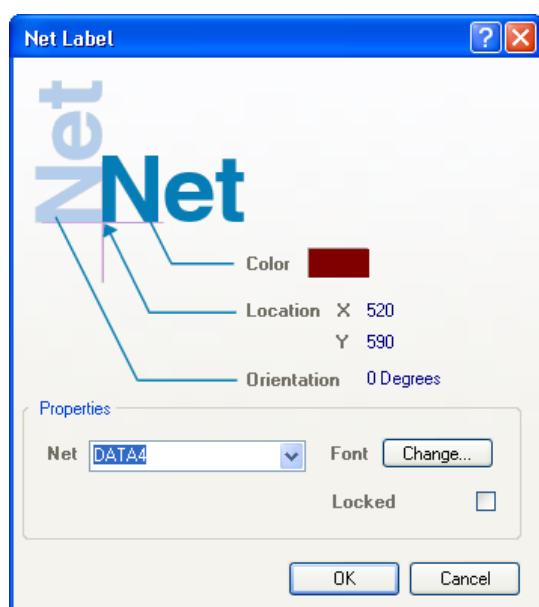
#### Editing

The properties of a net label object can be modified before, during and after placement. Editing itself falls into two categories - graphical and non-graphical.

The following three methods of non-graphical editing are available:

#### Editing via an associated properties dialog

This method of editing uses the following dialog to modify the properties of a net label object.



Use the dialog's 'What's This Help' feature to obtain detailed information about each of the options available. Click on the question mark button at the top right of the dialog and then click over a field or option to pop-up information specific to that field or option.

The *Net Label* dialog can be accessed prior to entering placement mode, from the **Schematic - Default Primitives** page of the *Preferences* dialog (**Tools » Schematic Preferences**). This allows you to change the default properties for the net label object, which will be applied when placing subsequent net labels.

During placement, the *Net Label* dialog can be accessed by pressing the **Tab** key.

After placement, the *Net Label* dialog can be accessed in one of the following ways:

- double-clicking on the placed net label object
- selecting the net label object and choosing **Properties** from the right-click pop-up menu
- choosing the **Change** command from the **Edit** menu and then clicking once over the placed net label object.

#### Editing via the SCH Inspector panel

The **SCH Inspector** panel enables you to interrogate and edit the properties of one or more design objects in the active document. Used in conjunction with appropriate filtering, the panel can be used to make changes to multiple objects of the same kind, from one convenient location.

 For more information on the **SCH Inspector** panel, press **F1** when the cursor is over the panel.

#### Editing via the SCH List panel

The **SCH List** panel allows you to display design objects from one or more documents in tabular format, enabling you to quickly inspect and modify object attributes. When used in conjunction with the **SCH Filter** panel, it enables you to display just those objects falling under the scope of the active filter - allowing you to target and edit multiple design objects with greater accuracy and efficiency.

 For more information on the **SCH List** and **SCH Inspector** panels, press **F1** when the cursor is over a panel.

#### Graphical editing

This method of editing allows you to select a placed net label object directly in the workspace and change its location graphically. Net labels can only be adjusted with respect to their size by changing the size of the font used (accessed through the **Annotation** dialog). As such, editing handles are not available when the net label object is selected:



Click anywhere inside the dashed box and drag to reposition the net label as required. The net label can be rotated or flipped while dragging.

If the **Enable In-Place Editing** option is enabled on the **Schematic - General** page of the *Preferences* dialog (**Tools » Schematic Preferences**), you will be able to edit the name for the net label directly in the workspace. Select the net label and then click once to invoke the feature. Type the new name as required and then click away from the net label or press **Enter** to effect the change.

If you attempt to graphically modify a netlabel object that has its **Locked** property enabled, a dialog will appear asking for confirmation to proceed with the edit.

If the **Protect Locked Objects** option is enabled in the **Schematic - General** page of the *Preferences* dialog (**Tools » Schematic Preferences**), and the **Locked** option for this design object is enabled as well then this object cannot be moved. This locked object is not part of the selected group of objects (only those objects that are unlocked can be selected and moved as a group). You will have to click on this locked object directly and accept the confirmation before moving this object.

#### Notes

A newly placed net label will initially have a default name of the form `NetLabel1n`. Edit the properties of the net label to change the name to that required.

---

Should you need to negate (include a bar over the top of) a net label, this can be done in one of two ways:

- Include a backslash character after each character in the net name (e.g. `E\N\A\B\L\E\`)
- Enable the **Single '\'' Negation** option on the **Schematic - Graphical Editing** page of the *Preferences* dialog, then include one backslash character at the start of the net name (e.g. `\ENABLE`).

If you are creating a design using multiple schematic sheets, the type of hierarchy model you choose to employ will affect the way net names on different sheets are interpreted. If you set the **Net Identifier Scope** on the **Options** tab of the *Options for Project dialog (Project » Project Options)* to **Global**, any net labels you have assigned will be interpreted globally and all net labels with the same name will be treated as if they are connected, regardless of which sheets they are on. With other types of hierarchy, net labels are local to the current sheet and do not connect to nets of the same name on other sheets.

Assign a bus to multiple nets using a Net Label of the form **D[0..7]** or **D[7..0]**, indicating that the individual (and distinct) nets D0 to D7 are carried by the bus. Whichever form you use, the ordering must be consistent between connected bus objects.

Assigning a net label with this syntax (bus syntax) to a bus changes the bus object from being purely graphical to logical.

Remember that net identifiers of different types do not automatically connect to one another, even if they share the same name. This holds true for net identifiers with bus syntax; a net label **D[0..7]** will not automatically connect to a port with the same name. The bus is required to connect them together.

It is recommended that you define the net label in bus syntax to contain alpha characters only. For example, if you named the bus **D2[0..7]**, it would be expanded to **D20, D21..D27** which can cause net name conflicts.

In FPGA designs, define a constant value for a bus by specifying the required value in the net label for the bus. The constant can be declared in either decimal, binary or hexadecimal format. For example, consider an **FPGA\_STARTUP16** device, with a required **DELAY** input of 342 (decimal). The following entries for the bus net label could be used to define the constant:

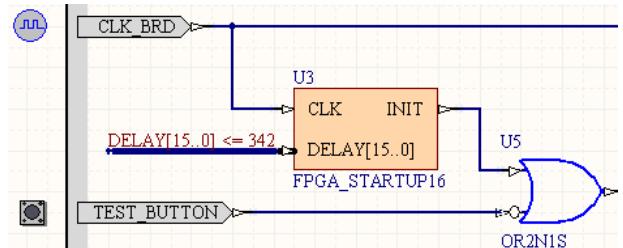
- **Decimal:** `DELAY[15..0] <= 342`
- **Binary:** `DELAY[15..0] <= b101010110`
- **Hexadecimal:** `DELAY[15..0] <= $156`

In a binary definition, the leading zeroes need not be declared. You can of course declare all bits and, for added readability, separate nibbles by a space or a hyphen. Therefore the decimal value 342 could be defined in binary as:

`0000 0001 0101 0110` or

`0000-0001-0101-0110`

Leading zeroes need not be declared for a hexadecimal value either. Also, you can use `$` or `0x` prefixes to denote the value as being hexadecimal.



Any changes made to object properties during placement will cause the default properties for the object to be updated, unless the **Permanent** option - on the **Schematic - Default Primitives** page of the *Preferences* dialog - is enabled. When this option is enabled, changes made will affect only the object being placed and subsequent objects placed during the same placement session.

## No ERC



### Description

The No ERC object is a design directive. Placing this directive on a node in the circuit suppresses any report warnings and errors that may be generated when compiling the schematic.

Use this directive to deliberately prevent error checking of certain parts of a circuit that you know will generate a warning (such as unfinished connections) while checking the rest of the circuit.

### Availability

No ERC design directives are available for placement in the Schematic Editor only. Use one of the following methods to access the placement command:

- choose **Place » Directives » No ERC [P, V, N]** from the main menus
- click the button on the **Wiring** toolbar.

### Placement

After launching the command, the cursor will change to a cross-hair and you will enter No ERC placement mode. Position the cursor over a wire or other net object and click or press **Enter** to place the directive.

Continue placing further No ERC directives or right-click or press **Esc** to exit placement mode.

Press the **Spacebar** while in placement mode to rotate the No ERC directive. Rotation is anti-clockwise and in steps of 90°.

Press the **X** or **Y** keys while in placement mode to flip the No ERC directive along the X-axis or Y-axis respectively.

### Editing

The properties of a No ERC directive can be modified before, during and after placement. Editing itself falls into two categories - graphical and non-graphical.

The following three methods of non-graphical editing are available:

#### Editing via an associated properties dialog

This method of editing uses the following dialog to modify the properties of a No ERC directive.

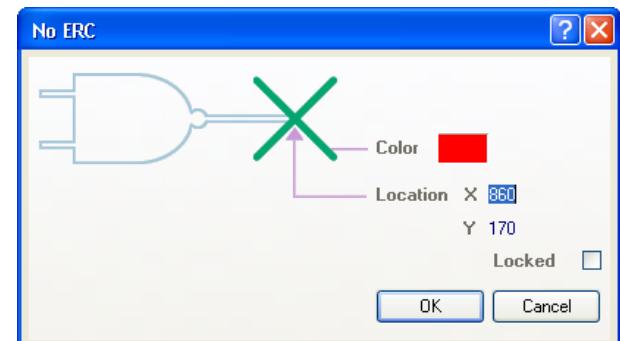
Use the dialog's 'What's This Help' feature to obtain detailed information about each of the options available. Click on the question mark button at the top right of the dialog and then click over a field or option to pop-up information specific to that field or option.

The *No ERC* dialog can be accessed prior to entering placement mode, from the **Schematic - Default Primitives** page of the **Preferences** dialog (**Tools » Schematic Preferences**). This allows you to change the default properties for the No ERC directive, which will be applied when placing subsequent No ERC directives.

During placement, the *No ERC* dialog can be accessed by pressing the **Tab** key.

After placement, the *No ERC* dialog can be accessed in one of the following ways:

- double-clicking on the placed No ERC directive
- selecting the No ERC directive and choosing **Properties** from the right-click pop-up menu
- choosing the **Change** command from the **Edit** menu and then clicking once over the placed No ERC directive.



#### Editing via the SCH Inspector panel

The **SCH Inspector** panel enables you to interrogate and edit the properties of one or more design objects in the active document. Used in conjunction with appropriate filtering, the panel can be used to make changes to multiple objects of the same kind, from one convenient location.



For more information on the **SCH Inspector** panel, press **F1** when the cursor is over the panel.

### Editing via the SCH List panel

The **SCH List** panel allows you to display design objects from one or more documents in tabular format, enabling you to quickly inspect and modify object attributes. When used in conjunction with the **SCH Filter** panel, it enables you to display just those objects falling under the scope of the active filter - allowing you to target and edit multiple design objects with greater accuracy and efficiency.



For more information on the **SCH List** and **SCH Inspector** panels, press **F1** when the cursor is over a panel.

### Graphical editing

This method of editing allows you to select a placed No ERC directive directly in the workspace and change its location graphically. No ERC directives are fixed with respect to their size and shape. As such, editing handles are not available when the No ERC directive is selected:



Click anywhere inside the dashed box and drag to reposition the No ERC directive as required. The No ERC directive can be rotated or flipped while dragging.

If you attempt to graphically modify a NoERC object that has its **Locked** property enabled, a dialog will appear asking for confirmation to proceed with the edit.

If the **Protect Locked Objects** option is enabled in the **Schematic – Graphical Editing** page of the *Preferences* dialog (**Tools** » **Schematic Preferences**), and the **Locked** option for this design object is enabled as well then this object cannot be selected or graphically edited.

You will have to double click on this locked object directly and disable the **Locked** property or disable the **Protect Locked Objects** option in the **Schematic – Graphical Editing** Preferences dialog to graphically edit this object.

If you attempt to select locked objects along with other objects, only those objects that are unlocked can be selected and moved as a group if the **Protect Locked Objects** option is enabled. Otherwise if the **Protect Locked Objects** option is disabled you will be prompted for confirmation to proceed with the edit of the objects including locked objects.

### Notes

Any changes made to object properties during placement will cause the default properties for the object to be updated, unless the **Permanent** option - on the **Schematic - Default Primitives** page of the *Preferences* dialog - is enabled. When this option is enabled, changes made will affect only the object being placed and subsequent objects placed during the same placement session.

## Note



### Description

A note is a non-electrical drawing primitive. It is used to add informational or instructional text to a specific area within a schematic, in a similar vain to that of commenting a program's source code. The note is a resizable rectangular area that can contain multiple lines of text and can automatically wrap and clip text to keep it within the bounds of the note.

### Availability

Notes are available for placement in the Schematic Editor only, by choosing **Place** » **Notes** » **Note [P, E, O]** from the Schematic Editor main menus.

### Placement

After launching the command, the cursor will change to a cross-hair and you will enter note placement mode. Placement is made by performing the following sequence of actions:

- click or press **Enter** to anchor the first corner of the note
- move the cursor to adjust the size of the note, then click or press **Enter** to anchor the diagonally-opposite corner and thereby complete placement of the note.

Continue placing further notes, or right-click or press **Esc** to exit placement mode.

The note object can be rotated or flipped while in placement mode and before the first corner of the note is anchored:

- Press the **Spacebar** to rotate the note. Rotation is anti-clockwise and in steps of 90°.
- Press the **X** or **Y** keys to flip the note along the X-axis or Y-axis respectively.

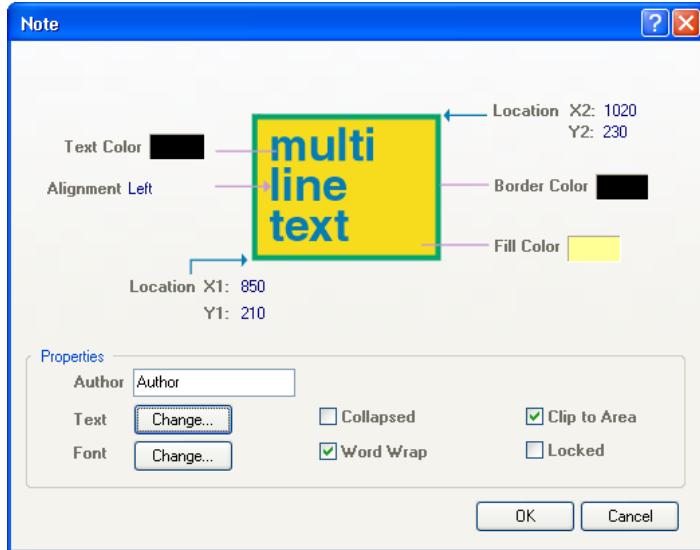
### Editing

The properties of a note object can be modified before, during and after placement. Editing itself falls into two categories - graphical and non-graphical.

The following three methods of non-graphical editing are available:

#### Editing via an associated properties dialog

This method of editing uses the following dialog to modify the properties of a note object.



Use the dialog's 'What's This Help' feature to obtain detailed information about each of the options available. Click on the question mark button at the top right of the dialog and then click over a field or option to pop-up information specific to that field or option.

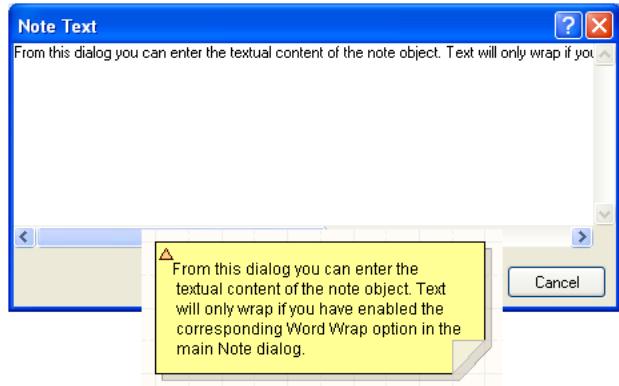
The **Note** dialog can be accessed prior to entering placement mode, from the **Schematic - Default Primitives** page of the **Preferences** dialog (**Tools » Schematic Preferences**). This allows you to change the default properties for the note object, which will be applied when placing subsequent notes.

During placement, the **Note** dialog can be accessed by pressing the **Tab** key.

After placement, the **Note** dialog can be accessed in one of the following ways:

- double-clicking on the placed note object
- selecting the note object and choosing **Properties** from the right-click pop-up menu
- choosing the **Change** command from the **Edit** menu and then clicking once over the placed note object.

The actual text of the note can be entered/edited via the **Note** dialog. Press the **Change** button associated with the note's **Text** property to access the **Note Text** dialog.

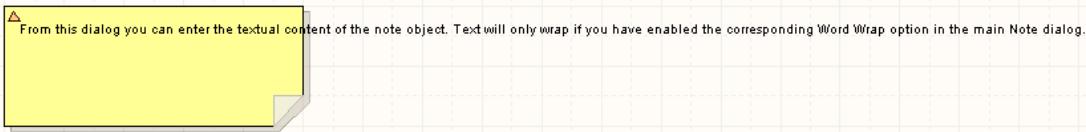


In addition to providing a **Word Wrap** option, the main **Note** dialog provides a **Clip to Area** option. With this option enabled, text will be kept within the bounds of the note's frame. When disabled, text will spill out of the frame onto the schematic sheet.

For example, consider the note in the previous image. Both **Word Wrap** and **Clip to Area** options were enabled. If word wrapping is disabled, leaving only the clipping option enabled, only text that fits within the frame of the note will remain displayed:

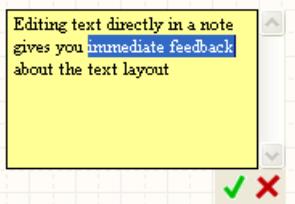


However, if the option to clip the text is disabled, the text is free to be displayed beyond the constraints of the frame's boundary:



### In-Place Editing

If the **Enable In-Place Editing** option is enabled on the **Schematic - General** page of the *Preferences* dialog (**Tools » Schematic Preferences**), you will be able to edit the textual content of the note directly in the workspace. Select the note (in its expanded state) and then click once to invoke the feature.



Make changes to the text as required. The available right-click menu provides standard editing commands such as cut, copy, paste and delete.

To effect a change either click away from the note or press the green tick button. If you decide the change made is not needed, press the red cross button to discard the change.

If the **Word Wrap** option is disabled in the *Note* dialog, a horizontal scroll bar will also be available when editing the text in-situ.



### Editing via the SCH Inspector panel

The **SCH Inspector** panel enables you to interrogate and edit the properties of one or more design objects in the active document. Used in conjunction with appropriate filtering, the panel can be used to make changes to multiple objects of the same kind, from one convenient location.

For more information on the **SCH Inspector** panel, press **F1** when the cursor is over the panel.

### Editing via the SCH List panel

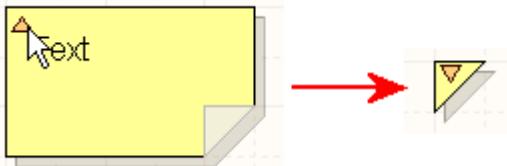
The **SCH List** panel allows you to display design objects from one or more documents in tabular format, enabling you to quickly inspect and modify object attributes. When used in conjunction with the **SCH Filter** panel, it enables you to display just those objects falling under the scope of the active filter - allowing you to target and edit multiple design objects with greater accuracy and efficiency.

For more information on the **SCH List** and **SCH Inspector** panels, press **F1** when the cursor is over a panel.

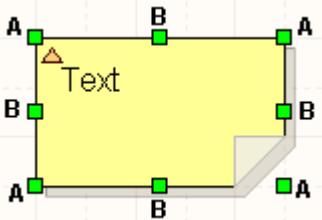
### Graphical editing

This method of editing allows you to select a placed note object directly in the workspace and change its size, shape or location, graphically.

A note can be displayed in either expanded (full frame) or collapsed (small triangle) modes. Toggle the display mode either by using the **Collapsed** option in the *Note* dialog, or by clicking on the top left corner of a placed note.



When a fully expanded note object is selected, the following editing handles are available:



Click and drag **A** to resize the note in the vertical and horizontal directions simultaneously.

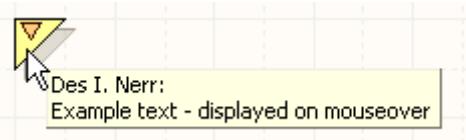
Click and drag **B** to resize the note in the vertical and horizontal directions separately.

Click anywhere on the note - away from editing handles - and drag to reposition it. The note can be rotated or flipped while dragging.

The size and shape of a note cannot be changed graphically when the note is in collapsed mode, only its location/orientation. As such, editing handles are not available when a collapsed note object is selected:



Click anywhere inside the dashed box and drag to reposition the note as required. The note can be rotated or flipped while dragging.



When a note is in collapsed mode, hovering the cursor over it will display a pop-up containing the name of the note's author and the actual text content of the note.

If you attempt to graphically modify a note object that has its **Locked** property enabled, a dialog will appear asking for confirmation to proceed with the edit.

If the **Protect Locked Objects** option is enabled in the **Schematic – Graphical Editing** page of the *Preferences* dialog (**Tools** » **Schematic Preferences**), and the **Locked** option for this design object is enabled as well then this object cannot be selected or graphically edited.

You will have to double click on this locked object directly and disable the **Locked** property or disable the **Protect Locked Objects** option in the **Schematic – Graphical Editing** Preferences dialog to graphically edit this object.

If you attempt to select locked objects along with other objects, only those objects that are unlocked can be selected and moved as a group if the **Protect Locked Objects** option is enabled. Otherwise if the **Protect Locked Objects** option is disabled you will be prompted for confirmation to proceed with the edit of the objects including locked objects.

## Notes

While notes can be rotated or flipped along the X or Y axis, this has no effect on the orientation of the text within.

---

Any changes made to object properties during placement will cause the default properties for the object to be updated, unless the **Permanent** option - on the **Schematic - Default Primitives** page of the *Preferences* dialog - is enabled. When this option is enabled, changes made will affect only the object being placed and subsequent objects placed during the same placement session.

## Off Sheet Connector



### Description

An off sheet connector is an electrical design primitive. Off sheet connectors are used to connect nets across multiple schematic sheets that are descended from the same parent sheet symbol.

### Availability

Off sheet connectors are available for placement in the Schematic Editor only, by choosing **Place » Off Sheet Connector [P, C]** from the Schematic Editor main menus.

### Placement

After launching the command, the cursor will change to a cross-hair and you will enter off sheet connector placement mode. Position the cursor on the sheet and click or press **Enter** to effect placement of the off sheet connector.

Continue placing further off sheet connectors, or right-click or press **Esc** to exit placement mode.

Press the **Spacebar** while in placement mode to rotate the off sheet connector. Rotation is anti-clockwise and in steps of 90°.

Press the **X** or **Y** keys while in placement mode to flip the off sheet connector along the X-axis or Y-axis respectively.

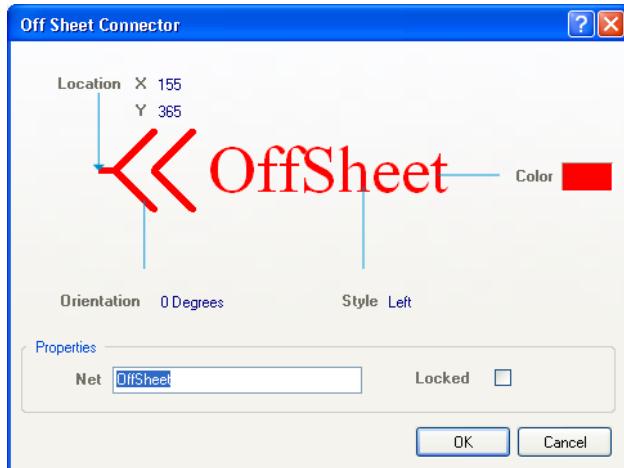
### Editing

The properties of an off sheet connector object can be modified before, during and after placement. Editing itself falls into two categories - graphical and non-graphical.

The following three methods of non-graphical editing are available:

#### Editing via an associated properties dialog

This method of editing uses the following dialog to modify the properties of an off sheet connector object.



Use the dialog's 'What's This Help' feature to obtain detailed information about each of the options available. Click on the question mark button at the top right of the dialog and then click over a field or option to pop-up information specific to that field or option.

The *Off Sheet Connector* dialog can be accessed prior to entering placement mode, from the **Schematic - Default Primitives** page of the *Preferences* dialog (**Tools » Schematic Preferences**). This allows you to change the default properties for the off sheet connector object, which will be applied when placing subsequent off sheet connectors.

During placement, the *Off Sheet Connector* dialog can be accessed by pressing the **Tab** key.

After placement, the *Off Sheet Connector* dialog can be accessed in one of the following ways:

- double-clicking on the placed off sheet connector object
- selecting the off sheet connector object and choosing **Properties** from the right-click pop-up menu
- choosing the **Change** command from the **Edit** menu and then clicking once over the placed off sheet connector object.

### Editing via the SCH Inspector panel

The **SCH Inspector** panel enables you to interrogate and edit the properties of one or more design objects in the active document. Used in conjunction with appropriate filtering, the panel can be used to make changes to multiple objects of the same kind, from one convenient location.

 For more information on the **SCH Inspector** panel, press **F1** when the cursor is over the panel.

### Editing via the SCH List panel

The **SCH List** panel allows you to display design objects from one or more documents in tabular format, enabling you to quickly inspect and modify object attributes. When used in conjunction with the **SCH Filter** panel, it enables you to display just those objects falling under the scope of the active filter - allowing you to target and edit multiple design objects with greater accuracy and efficiency.

 For more information on the **SCH List** and **SCH Inspector** panels, press **F1** when the cursor is over a panel.

### Graphical editing

This method of editing allows you to select a placed off sheet connector object directly in the workspace and change its location graphically. Off sheet connectors are fixed with respect to their size and shape. As such, editing handles are not available when the off sheet connector object is selected:



Click anywhere inside the dashed box and drag to reposition the off sheet connector as required. The off sheet connector can be rotated or flipped while dragging.

If the **Enable In-Place Editing** option is enabled on the **Schematic - General** page of the *Preferences* dialog (**Tools** » **Schematic Preferences**), you will be able to edit the assigned net for the off sheet connector directly in the workspace. Select the off sheet connector and then click once to invoke the feature. Type the name of the assigned net as required and then click away from the off sheet connector or press **Enter** to effect the change.

If you attempt to graphically modify a off sheet connector object that has its **Locked** property enabled, a dialog will appear asking for confirmation to proceed with the edit.

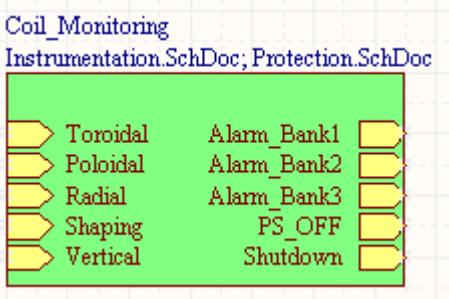
If the **Protect Locked Objects** option is enabled in the **Schematic – Graphical Editing** page of the *Preferences* dialog (**Tools** » **Schematic Preferences**), and the **Locked** option for this design object is enabled as well then this object cannot be selected or graphically edited.

You will have to double click on this locked object directly and disable the **Locked** property or disable the **Protect Locked Objects** option in the **Schematic – Graphical Editing** Preferences dialog to graphically edit this object.

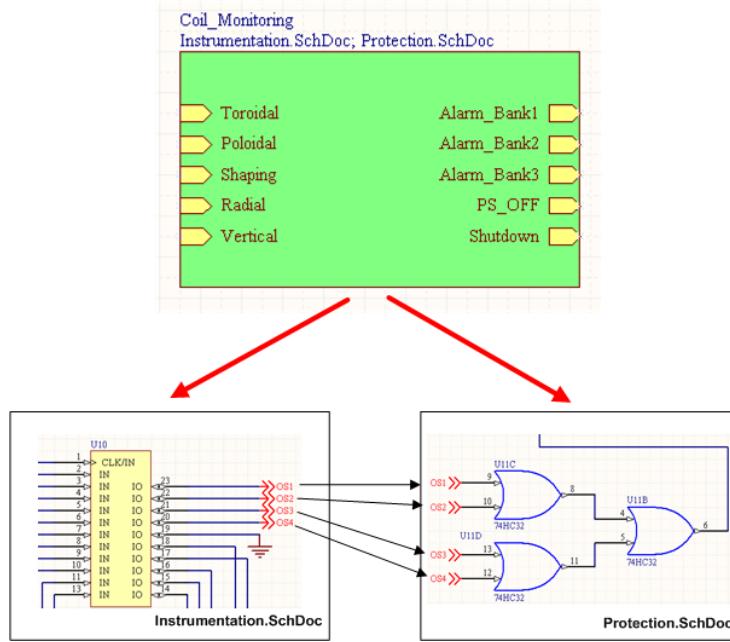
If you attempt to select locked objects along with other objects, only those objects that are unlocked can be selected and moved as a group if the **Protect Locked Objects** option is enabled. Otherwise if the **Protect Locked Objects** option is disabled you will be prompted for confirmation to proceed with the edit of the objects including locked objects.

### Notes

Multiple schematic sub-sheets can be referenced by a single sheet symbol by separating each filename by a semi-colon, in the **Filename** field.

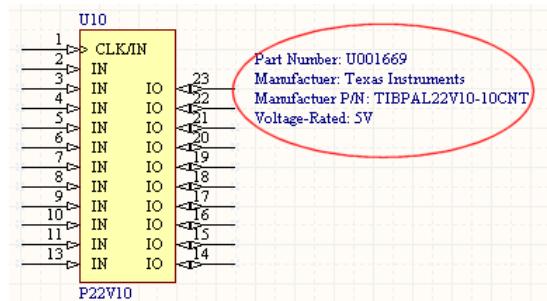


Off sheet connectors can only be used to connect nets across the descendant child schematic sheets of the same parent sheet symbol. To successfully connect a particular net across two or more sheets, the off sheet connectors on each sheet must be assigned to the same net.



Any changes made to object properties during placement will cause the default properties for the object to be updated, unless the **Permanent** option - on the **Schematic - Default Primitives** page of the *Preferences* dialog - is enabled. When this option is enabled, changes made will affect only the object being placed and subsequent objects placed during the same placement session.

## Parameter



### Description

A parameter is a non-electrical child object of an electrical design primitive or design directive. It is a user-definable object that allows you to add additional information to a design object supporting the use of parameters (e.g. a part). Such an object can have multiple parameters defined, each of which can be displayed with respect to its name and/or value.

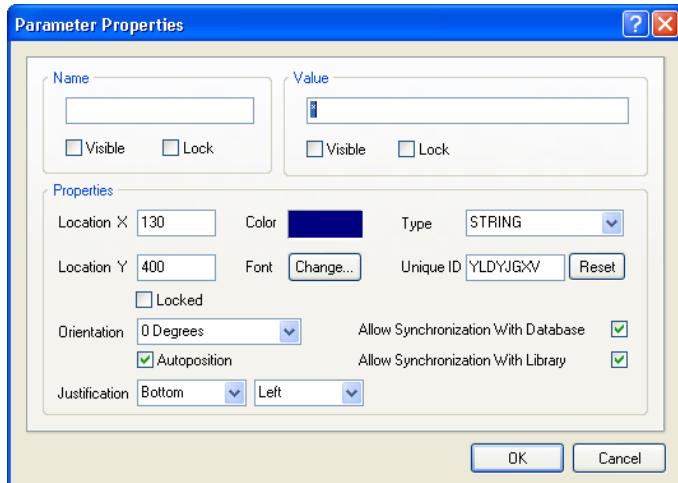
### Availability and Placement

Parameters may be added to any of the following design objects:

- **Part** - from the Parameters region of the *Component Properties* dialog (or *Library Component Properties* dialog if added when defining a component in the Schematic Library Editor)
- **Pin** - from the Parameters tab of the *Pin Properties* dialog
- **Port** - from the Parameters tab of the *Port Properties* dialog
- **Sheet Symbol** - from the Parameters tab of the *Sheet Symbol* dialog
- **Parameter Set** - from the associated *Parameters* dialog
- **PCB Layout directive** - from the associated *Parameters* dialog
- **Probe directive** - from the *Probe* dialog
- **Stimulus directive** - from the associated *Parameters* dialog
- **Test Vector Index directive** - from the associated *Parameters* dialog
- **Net Class directive** - from the associated *Parameters* dialog
- **Differential Pair directive** - from the associated *Parameters* dialog.

In addition, you can add parameters at the document level (from the **Parameters** tab of the *Document Options* dialog (**Design » Document Options**)) and the project level (from the **Parameters** tab of the *Options For Project* dialog (**Project » Project Options**)).

In each case the controls for defining new parameters, or editing/removing existing ones, are the same. Click the **Add** button to create a new parameter - the *Parameter Properties* dialog will appear.

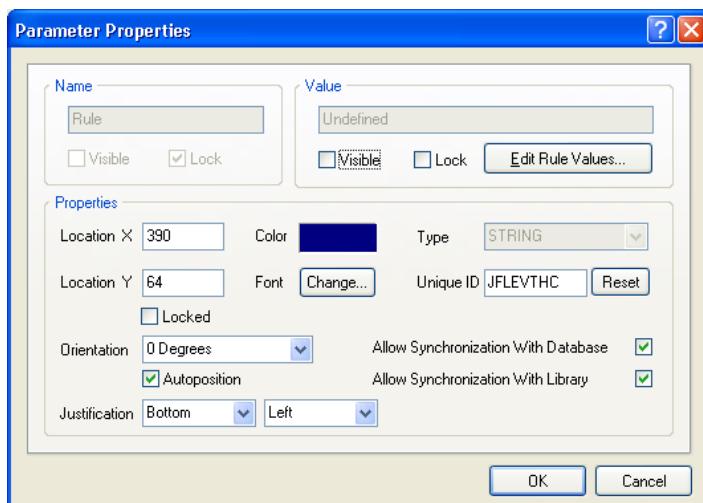


Use this dialog to define a name and value for the parameter and to setup graphical properties that will determine how the parameter information appears in the workspace.

Each defined parameter will appear in the corresponding parameters list, showing its name, value and type. You can control the visibility of each parameter as required, using the corresponding option in the **Visible** column.

### Adding a Parameter as a Rule

Use the **Add as Rule** button to specifically add a design rule directive into the schematic. This feature allows you to define constraints for the design prior to PCB layout. The *Parameter Properties* dialog will appear, with the **Name** and **Type** fields set to **Rule** and **STRING** respectively and inactive.



Clicking the **Edit Rule Values** button will open the *Choose Design Rule Type* dialog. This dialog lists each of the rule categories and rule types that are available in the PCB document. Double-click on a rule type to open its corresponding *Edit Rule Type* dialog, from where you can define the constraints for the rule.

### Editing

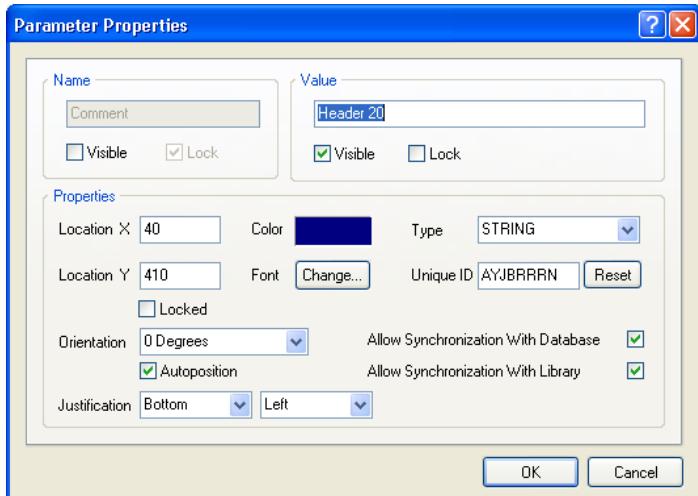
The properties of a parameter object can be modified before and after placement of the parent design object. Editing itself falls into two categories - graphical and non-graphical.

The following three methods of non-graphical editing are available:

#### Editing via an associated properties dialog

This method of editing uses the following dialog to modify the properties of a parameter object, independently of the parent design object.

Use the dialog's 'What's This Help' feature to obtain detailed information about each of the options available. Click on the question mark button at the top right of the dialog and then click over a field or option to pop-up information specific to that field or option.



The **Parameter Properties** dialog can be accessed prior to adding a new parameter to a design object, from the **Schematic - Default Primitives** page of the **Preferences** dialog (**Tools » Schematic Preferences**). This allows you to change the default properties for the parameter object, which will be applied when adding subsequent new parameters to design objects.

After placement of the parent design object, the **Parameter Properties** dialog can be accessed in one of the following ways:

- double-clicking on the required parameter field of the placed design object
- selecting the required parameter field of the design object and choosing **Properties** from the right-click pop-up menu
- choosing the **Change** command from the **Edit** menu and then clicking once over the required parameter field of the placed design object.

#### Editing via the SCH Inspector panel

The **SCH Inspector** panel enables you to interrogate and edit the properties of one or more design objects in the active document. Used in conjunction with appropriate filtering, the panel can be used to make changes to multiple objects of the same kind, from one convenient location.

For more information on the **SCH Inspector** panel, press **F1** when the cursor is over the panel.

#### Editing via the SCH List panel

The **SCH List** panel allows you to display design objects from one or more documents in tabular format, enabling you to quickly inspect and modify object attributes. When used in conjunction with the **SCH Filter** panel, it enables you to display just those objects falling under the scope of the active filter - allowing you to target and edit multiple design objects with greater accuracy and efficiency.

For more information on the **SCH List** and **SCH Inspector** panels, press **F1** when the cursor is over a panel.

#### Graphical editing

This method of editing allows you to select a parameter object directly in the workspace and change its location graphically. Parameters can only be adjusted with respect to their size by changing the size of the font used (accessed through the **Parameter Properties** dialog). As such, editing handles are not available when a parameter object is selected:

Manufacturer: Texas Instruments

Click anywhere inside the dashed box and drag to reposition the parameter object as required. The object can be rotated or flipped while dragging:

- Press the **Spacebar** to rotate the parameter. Rotation is anti-clockwise and in steps of 90°.
- Press the **X** or **Y** keys to flip the parameter along the X-axis or Y-axis respectively.

If the **Enable In-Place Editing** option is enabled on the **Schematic - General** page of the **Preferences** dialog (**Tools » Schematic Preferences**), you will be able to edit the value for a parameter directly in the workspace. Select the parameter and then click once to invoke the feature. Type the new value as required and then click away from the parameter field or press **Enter** to effect the change.

## Notes

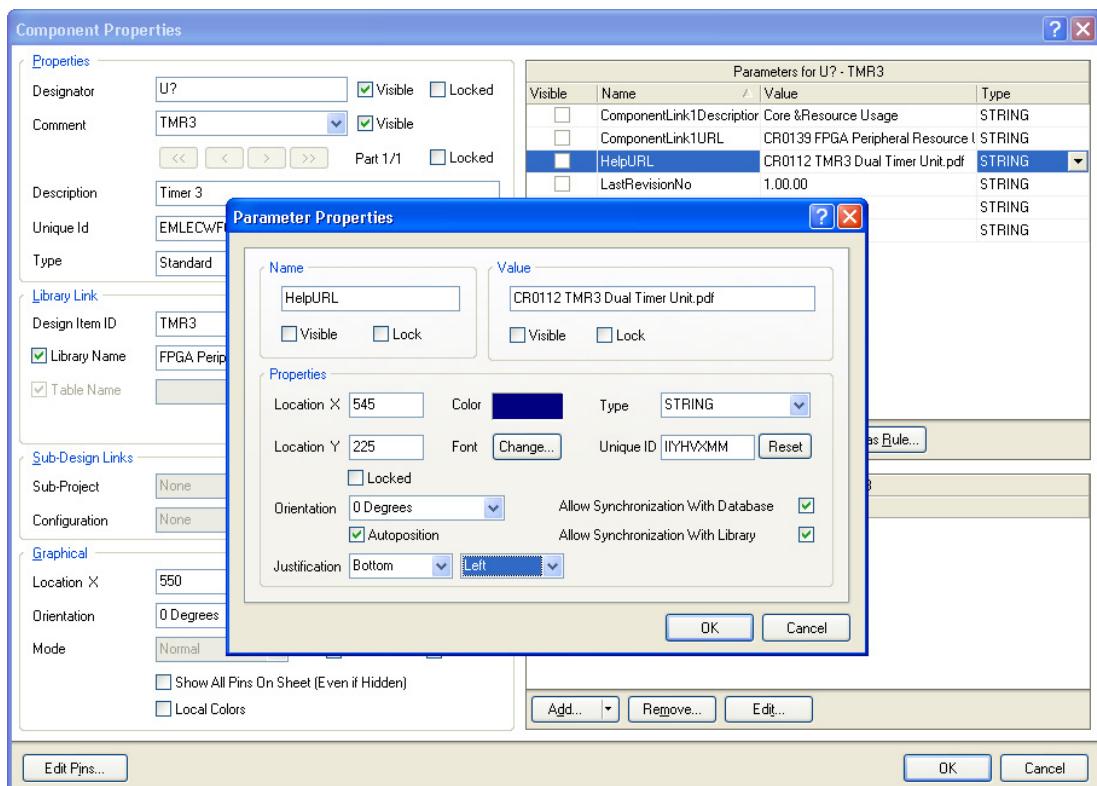
Parameters can be added, edited and removed from applicable design objects using the **Parameters** region of the **SCH Inspector** panel (with the relevant object(s) selected).

There may be times where you want (need, or prefer) to access your own reference material from within a design project. To facilitate this, the application provides two methods for linking from a component on the schematic sheet to reference information. Linkage is established through the addition of specific component parameters. One approach gives **F1** access to a referenced document. The other, which caters for multiple references, uses the right-click context menu.

### The HelpURL Parameter

This parameter allows you to explicitly define which document will be accessed when you press **F1** - either over the associated component on the schematic sheet, or for the selected entry for that component in the **Libraries** panel.

Using this parameter, you can reference any document that can be opened by the Altium Designer environment, including PDF, HTML and Text documents. To use this feature, add the **HelpURL** parameter to the required component and set the value of the parameter to the document you wish to open.



When specifying the value for the parameter, you can either include an absolute path or just enter the document name. The following examples are valid entries for the value of the parameter:

```
C:\Design_Projects\Schematics\Modifications.txt
AP0102 Linking an FPGA Project to a PCB Project.pdf
www.opencores.com
```

For a PDF document, you can also specify at which page you wish the document to be opened. Do this by adding the #page=xx option at the end of the document name, as illustrated below:

```
CR0118 FPGA Generic Library Guide.pdf#page=364
```

When **F1** is pressed with the cursor hovering over the placed object, a search for the document is conducted as follows:

- If a path to a particular location is specified, this location will be searched first
- If the document cannot be found at this specific location, or if no path is specified, the \Help folder of the installation will be searched
- If the document still cannot be found, the default help topic for the object will be used and displayed.

## The ComponentLink Parameter Pair

This feature enables you to define and present named links to one or more target reference documents in a part's right-click context menu. Multiple `ComponentLink` parameter pairings can be defined. To use this feature, add and configure the two parameters for each pairing as follows:

First parameter - used to define the target document:

`Name = ComponentLinknURL`

`Value = target document name`

Again, the parameter's value must include the full path if the document does not reside in the `\Help` folder of the installation.

Specify the page number for a PDF document as required.

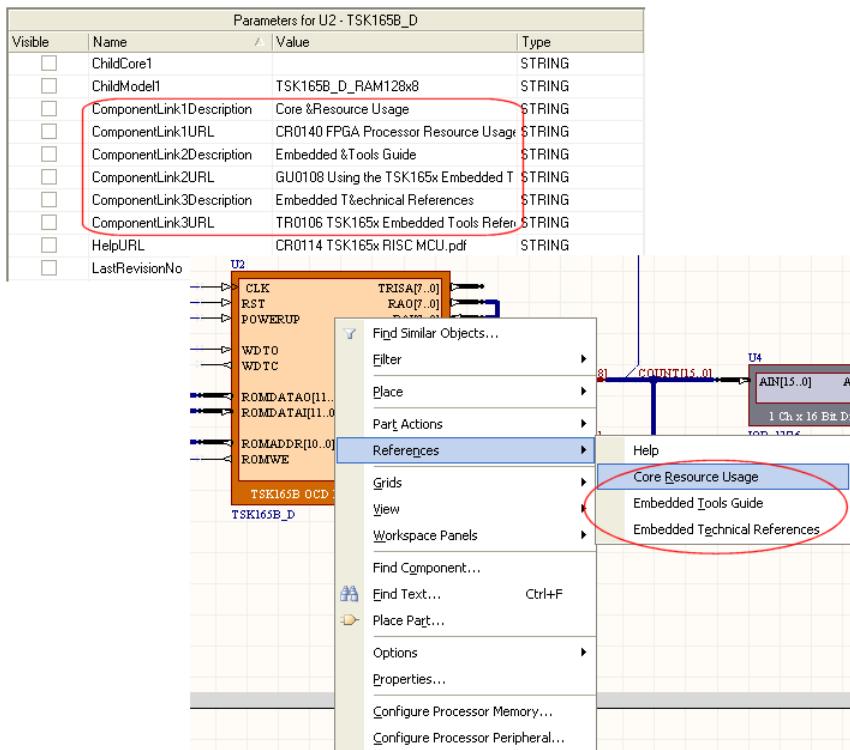
Second parameter - used to define the link caption that appears in the menu:

`Name = ComponentLinknDescription`

`Value = any meaningful description`

**Note:** For both parameters in the pairing, `n` is an integer value, allowing you to define multiple parameter pairings. For each `ComponentLink` pairing, ensure that `n` is the same for both constituent parameters.

To access the defined link, right-click on the placed part in the workspace. The entry for the link will appear in the **References** sub-menu.



The following points relate directly to the use of the `HelpURL` and the `ComponentLink` parameters:

- The `HelpURL`, `ComponentLinknURL` and `ComponentLinknDescription` parameter names must each be one word, but are case insensitive.
- When using either of the component-to-datasheet linking features to reference a PDF document, the document will open in an Acrobat Viewer window, separate to the Altium Designer Documentation Library. Once opened, calls to further PDFs will populate the open window and not start a new instance.

For other document types that are referenced using these features, the resulting document will open as a tabbed document view in the main design window.

- The document referenced by the `HelpURL` parameter for an object will take display precedence over any default topic written for that object.
- Where both `HelpURL` and `ComponentLink` parameters have been defined for a component, the right-click menu will contain entries for both. In the case of the former, the entry will be listed first and will be given the default description `Help`.

## Parameter Set



### Description

A Parameter Set is a design directive. It is essentially a container for one or more parameters, which can be associated to a net object within a schematic design.

### Availability

Parameter sets are available for placement in the Schematic Editor only. Both default (empty) and specific (Test Vector Index, Stimulus, PCB Layout, Net Class, Differential Pair) parameter set directives are available. Access the corresponding commands from the main **Place** menu as follows:

- choose **Place** » **Directives** » **Parameter Set [P, V, M]**
- choose **Place** » **Directives** » **Test Vector Index [P, V, T]**
- choose **Place** » **Directives** » **Stimulus [P, V, S]**
- choose **Place** » **Directives** » **PCB Layout [P, V, P]**
- choose **Place** » **Directives** » **Net Class [P, V, C]**
- choose **Place** » **Directives** » **Differential Pair [P, V, F]**.

### Placement

After launching the command, the cursor will change to a cross-hair and you will enter design directive placement mode. Position the cursor over a wire or other net object and click or press **Enter** to effect placement.

Continue placing further directives or right-click or press **Esc** to exit placement mode.

Press the **Spacebar** while in placement mode to rotate the directive. Rotation is anti-clockwise and in steps of 90°.

Press the **X** or **Y** keys while in placement mode to flip the directive along the X-axis or Y-axis respectively.

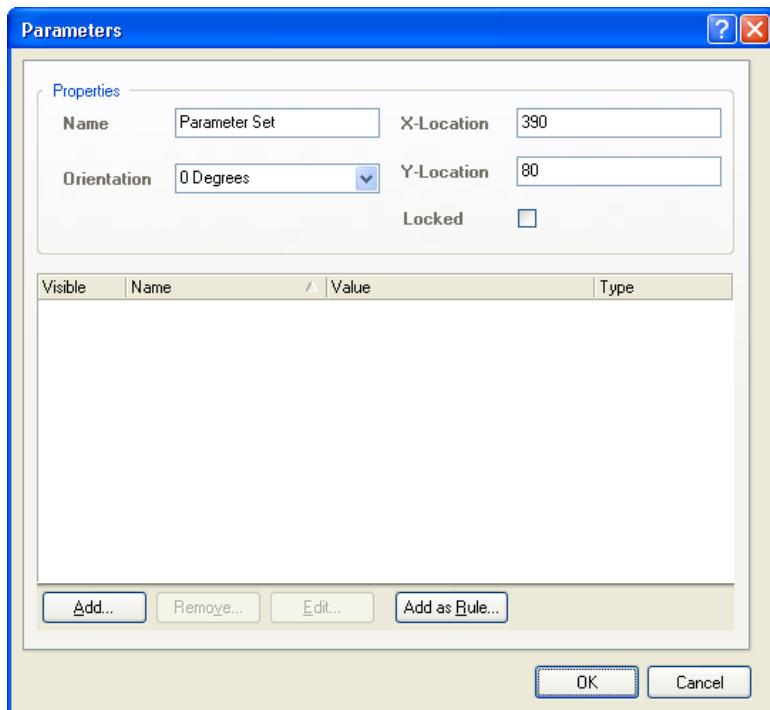
### Editing

The properties of a parameter set directive can be modified before, during and after placement. Editing itself falls into two categories - graphical and non-graphical.

The following three methods of non-graphical editing are available:

#### Editing via an associated properties dialog

This method of editing uses the following dialog to modify the properties of a parameter set directive.



Use the dialog's 'What's This Help' feature to obtain detailed information about each of the individual options available. Click on the question mark button at the top right of the dialog and then click over a field or option to pop-up information specific to that field or option.

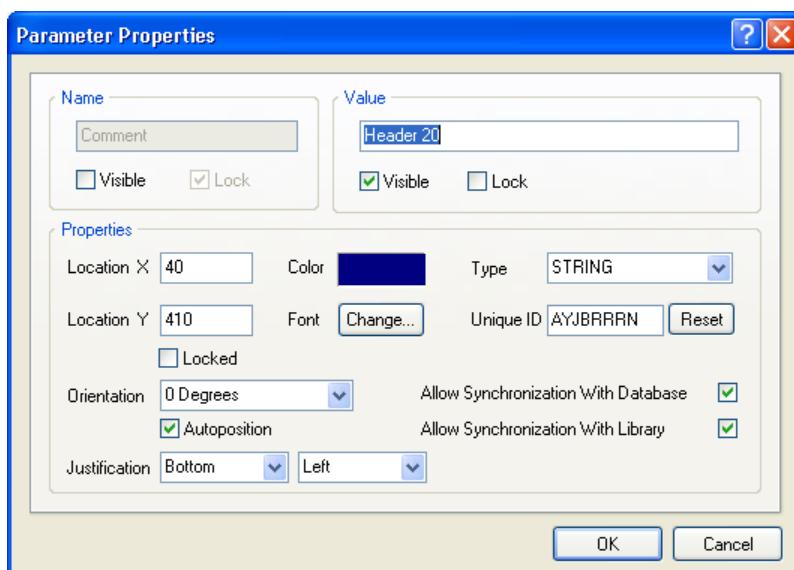
The **Parameters** dialog can be accessed prior to entering placement mode, from the **Schematic - Default Primitives** page of the **Preferences** dialog (**Tools » Schematic Preferences**). This allows you to change the default properties for the parameter set directive, which will be applied when placing subsequent parameter set directives.

During placement, the **Parameters** dialog can be accessed by pressing the **Tab** key.

After placement, the **Parameters** dialog can be accessed in one of the following ways:

- double-clicking on the placed parameter set directive
- selecting the parameter set directive and choosing **Properties** from the right-click pop-up menu
- choosing the **Change** command from the **Edit** menu and then clicking once over the placed parameter set directive.

The parameter set directive's member parameters can be added, edited or removed from within the *Parameters* dialog. The properties of a parameter are available to view/modify in the *Parameter Properties* dialog.



When a parameter is added as a rule, the parameter name (Rule) is locked and cannot be changed.

The parameters of a parameter set directive can be edited independently of the parent set directive. As such, the *Parameter Properties* dialog can be accessed using the three methods described previously (replacing parameter set directive with the relevant parameter object whose properties you wish to view/modify).

### Editing via the SCH Inspector panel

The **SCH Inspector** panel enables you to interrogate and edit the properties of one or more design objects in the active document. Used in conjunction with appropriate filtering, the panel can be used to make changes to multiple objects of the same kind, from one convenient location.

 For more information on the **SCH Inspector** panel, press **F1** when the cursor is over the panel.

### Editing via the SCH List panel

The **SCH List** panel allows you to display design objects from one or more documents in tabular format, enabling you to quickly inspect and modify object attributes. When used in conjunction with the **SCH Filter** panel, it enables you to display just those objects falling under the scope of the active filter - allowing you to target and edit multiple design objects with greater accuracy and efficiency.

 For more information on the **SCH List** and **SCH Inspector** panels, press **F1** when the cursor is over a panel.

### Graphical editing

This method of editing allows you to select a placed parameter set directive directly in the workspace and change its location or orientation graphically.

When a parameter set directive is selected in the workspace, a dashed box will appear around the directive. The box encloses the area occupied by the directive only. For each visibility-enabled member parameter of the set a dashed line will be visible, connecting the text field of the parameter to the body of the directive, thereby affirming association:



Click anywhere inside the dashed box and drag to reposition the parameter set directive as required. The directive can be rotated or flipped while dragging.

The parameter set directive's parameter text fields, which can be graphically edited independently of the parent directive, can only be adjusted with respect to their size by changing the size of the font used (accessed through the relevant *Parameter Properties* dialog). As such, editing handles are not available when any of these objects are selected:



Click anywhere inside the dashed box and drag to reposition the text object as required. The object can be rotated or flipped while dragging:

If the **Enable In-Place Editing** option is enabled on the **Schematic - General** page of the *Preferences* dialog (**Tools » Schematic Preferences**), you will be able to edit the value for a parameter directly in the workspace (with the exception of parameters that have been added as rules). Select the text object and then click once to invoke the feature. Type the new value as required and then click away from the text object or press **Enter** to effect the change.

If you attempt to graphically modify a parameter set object that has its **Locked** property enabled, a dialog will appear asking for confirmation to proceed with the edit.

If the **Protect Locked Objects** option is enabled in the **Schematic – Graphical Editing** page of the *Preferences* dialog (**Tools » Schematic Preferences**), and the **Locked** option for this design object is enabled as well then this object cannot be selected or graphically edited.

You will have to double click on this locked object directly and disable the **Locked** property or disable the **Protect Locked Objects** option in the **Schematic – Graphical Editing** Preferences dialog to graphically edit this object.

If you attempt to select locked objects along with other objects, only those objects that are unlocked can be selected and moved as a group if the **Protect Locked Objects** option is enabled. Otherwise if the **Protect Locked Objects** option is disabled you will be prompted for confirmation to proceed with the edit of the objects including locked objects.

## Notes

When placing a default parameter set directive there will be no existing parameters.

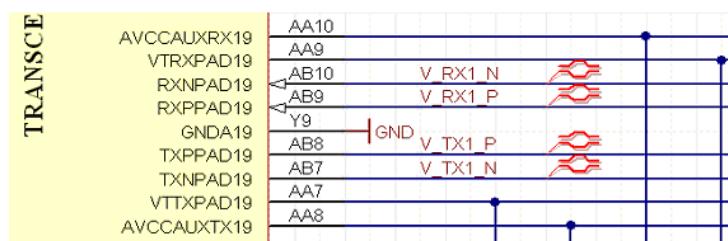
A PCB Layout directive allows you to assign PCB layout information to a net in the schematic. When a PCB is created from the schematic, the information in the PCB layout directive is used to create relevant PCB design rules.

The information specified by a PCB Layout directive is applied only to the net to which it is connected.

Net Class directives enable you to create user-defined net classes on the schematic. When a PCB is created from the schematic, the information in a Net Class directive is used to create the corresponding Net Class on the PCB. To make a net a member of a net class, attach a Net Class directive to the relevant wire or bus and set the directive's `ClassName` parameter to the name of the desired class.

The **Generate Net Classes** option (for User-Defined Classes) must be enabled, on the **Class Generation** tab of the *Options for Project* dialog, to make use of this feature.

A Differential Pair directive allows you to define a differential pair object on the schematic. Attach a directive of this type to both the positive and negative nets of the intended pair. The nets themselves must be named with the suffixes of `_P` and `_N` respectively. Both parameter set objects will contain a single parameter entry, with Name: `DifferentialPair` and Value: True.



Each pair of directives (one for the positive net, one for the negative) of this type will yield a differential pair object when transferred to the PCB during the synchronization process. Each of these differential pair objects will be added to the default Differential Pair class: All Differential Pairs.

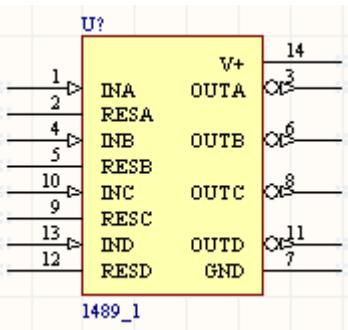
The name of a generated differential pair object will be the root name for the net pair on the schematic. For example directives added to `RX0_N` and `RX0_P` on the schematic will generate a differential pair object on the PCB with the name `RX0`. You can rename differential pair objects on the PCB side only.

A Stimulus directive is used to identify a node or net to be stimulated when a digital simulation is run. This directive is only used during netlist generation. It holds no significance for any schematic processes.

Test vector directives are used to identify a node with a simulation test vector. The test vectors are referred to by a column number, which indicates the column of the test vector file to use when the simulation is run. This directive is only used during netlist generation. It holds no significance for any schematic processes.

Any changes made to object properties during placement will cause the default properties for the object to be updated, unless the **Permanent** option - on the **Schematic - Default Primitives** page of the *Preferences* dialog - is enabled. When this option is enabled, changes made will affect only the object being placed and subsequent objects placed during the same placement session.

## Part



## Description

A part is an electrical design primitive. It is a schematic symbol that represents an electronic device, such as a resistor, switch, operational amplifier, IC, etc. Parts are stored within components in schematic component libraries (\*.SchLib) for example. A component within a library represents the physical device that is placed on the actual printed circuit board. Each component can contain one or more parts.

A physical component and a logical symbol are the same if they come from a standard library. But for database libraries, a physical component represents a record in a table of a database library. For example a physical component, 20AED15 and its logical symbol is a Capacitor - non polarized and comes from the Capacitor - Ceramic table in a database library. Note that a table within a database library is a collection of similar physical components.

## Availability

Parts are available for placement in the Schematic Editor only. Use one of the following methods to place a part:

- choose **Place » Part [P, P]** from the main menus
- click the button on the **Wiring** toolbar
- place a specific part directly from the **Libraries** panel
- place a specific part from within the Schematic Library Editor.

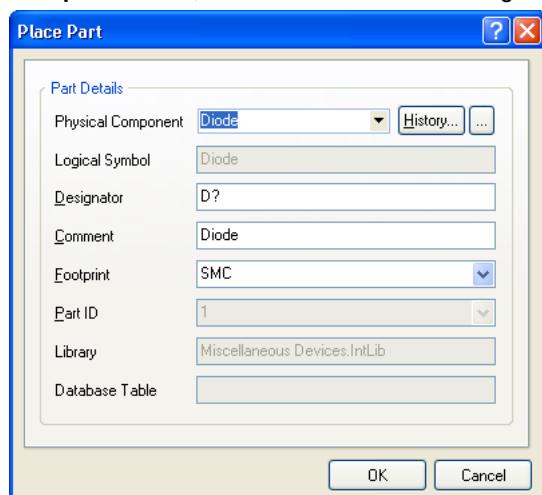
## Placement

The way in which a part is placed on a schematic sheet depends on how, and from where, placement mode is invoked.

### Placement using menu or toolbar command

After launching the command, the *Place Part* dialog will appear.

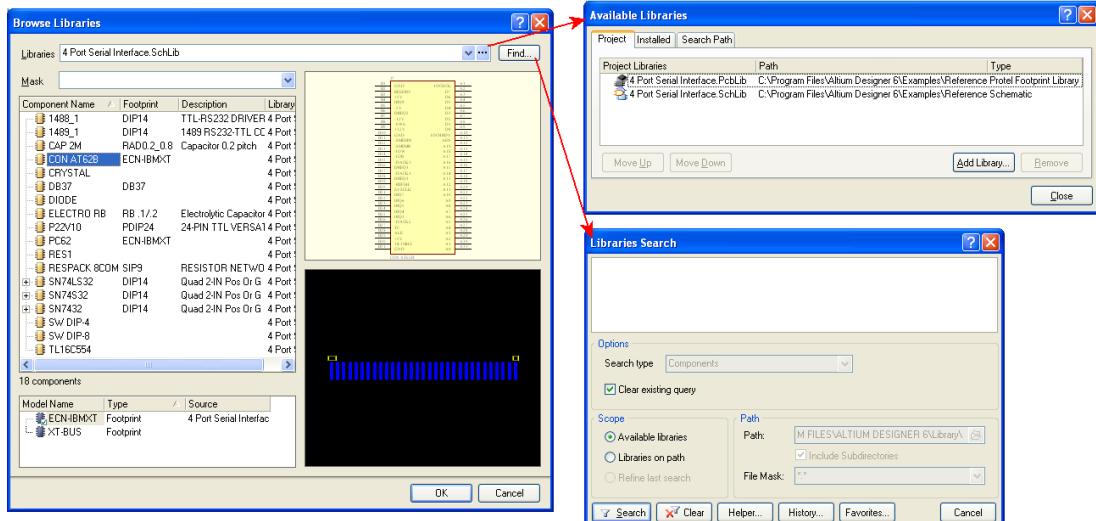
Use the dialog to choose the part you wish to place. You can either type the name of the required part directly into the **Physical Component** field, or use the ... button to the right of this field to open the *Browse Libraries* dialog.



From here, you can browse through the currently Available Libraries for the active project. The Available Libraries consist of project libraries, installed libraries and libraries found along search paths defined in the **Search Paths** tab of the *Options for*

**Project dialog (Project » Project Options).** Clicking the ... button in the *Browse Libraries* dialog will give access to the *Available Libraries* dialog, from where you can add/remove additional libraries to/from the overall list of those available to the project.

The dialog also provides a search facility - accessed by clicking the **Find** button - allowing you to search for a specific component across the Available Libraries or in any library along an external search path.



Click the **History** button in the *Place Part* dialog to access the *Placed Parts History* dialog.

The dialog, as its name suggests, contains all parts that you have previously placed on schematic sheets. The history list is persistent across design sessions unless purposefully cleared using the available **Clear History** button.

After choosing the required part, the fields of the *Place Part* dialog will be filled with information associated to the chosen part. The Designator will initially be of the default form C?, D?, J?, U?, etc. You can enter the specific designator you require here, or at a later stage.

After clicking **OK**, you will return to the schematic document and an outline of the part will appear floating on the cursor. Position the part at the location required and click or press **Enter** to effect placement.

Continue placing further instances of the same part, or right-click or press **Esc** to exit. The *Place Part* dialog will reappear. Either browse for a different part to place or click **Cancel** to exit placement mode.

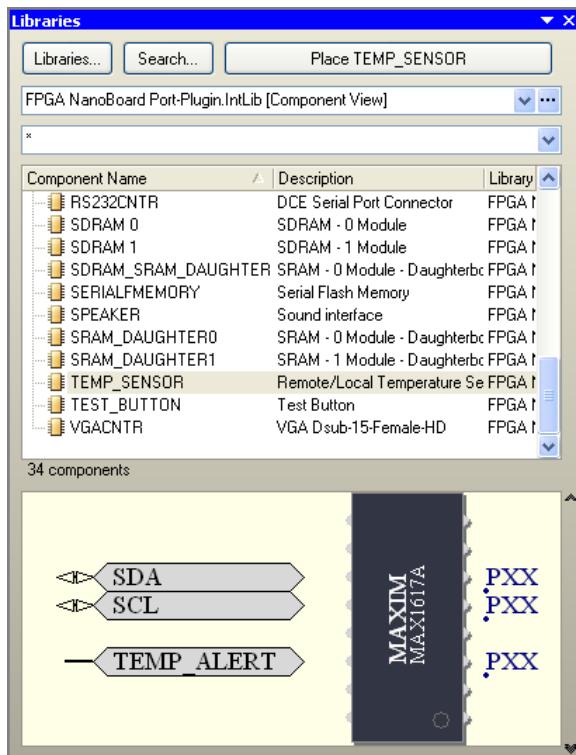
Press the **Spacebar** while in placement mode to rotate the part. Rotation is anti-clockwise and in steps of 90°.

Press the **X** or **Y** keys while in placement mode to flip the part along the X-axis or Y-axis respectively.

#### Placement from Libraries panel

Parts can be placed onto the active schematic directly from the **Libraries** panel. Remember that parts are stored within components in schematic component libraries and, as such, entries in the panel are referred to as Components rather than Parts.

Placed Parts History				
Lib. Reference	Designator	Comment	Footprint	Part ID
SN7432	U?	SN7432	DIP14	A
CRYSTAL		CRYSTAL	None Available	1
1489_1	U?	1489_1	DIP14	1



Parts can only be placed from the panel when the library being browsed is either a schematic component library (\*.SchLib) or an integrated library (\*.IntLib) and a schematic document is active in the main design window. If the browse mode for the panel is set to display additional model types (Footprint, PCB3D), then use library entries with the suffix [Component View].

When a selected component can be validly placed onto the active document, the **Place** button at the top-right of the panel will become available and its text will change to incorporate the name of that component.

To place a selected component, either:

- click on the **Place** button,
- double-click on the component entry,
- right-click and choose the **Place ComponentName** command, or
- click on the component entry and drag it onto the schematic sheet.

When placing a part on a schematic sheet using any of the first three methods of placement, the selected part will appear floating on the cursor. Position the part at the required location and click to effect placement.

Continue placing further instances of the same part or right-click or press **Esc** to exit placement mode.

Press the **Spacebar** while in placement mode to rotate the part. Rotation is anti-clockwise and in steps of 90°.

Press the **X** or **Y** keys while in placement mode to flip the part along the X-axis or Y-axis respectively.

When using the click-and-drag placement method, only a single instance of the part is placed. You do not remain in placement mode and the part cannot be rotated or flipped.

### Placement from within Schematic Library Editor

Parts can be placed onto a Schematic document directly from the active Schematic Library document.

Placement is carried out from the **SCH Library** panel. Remember that parts are stored within components in schematic component libraries and, as such, entries in the panel are referred to as Components rather than Parts.



Select the entry for the component you wish to place and click the **Place** button. The schematic document upon which the component is placed will depend on whether or not any schematic documents are currently open:

- If there are no schematic documents open, clicking the **Place** button will cause a new schematic document (`Sheet1.SchDoc`) to be created and made the active document in the main design window. The active library component will appear floating on the cursor, ready for placement
- If one or more schematic documents are currently open, the last document to have been active (irrespective of the project it belongs to) will be made the active document in the main design window and the active library component will appear floating on the cursor, ready for placement.

When in placement mode, position the part at the location required and click or press **Enter** to effect placement.

Continue placing further instances of the same part, or right-click or press **Esc** to exit placement mode.

Press the **Spacebar** while in placement mode to rotate the part. Rotation is anti-clockwise and in steps of 90°.

Press the **X** or **Y** keys while in placement mode to flip the part along the X-axis or Y-axis respectively.

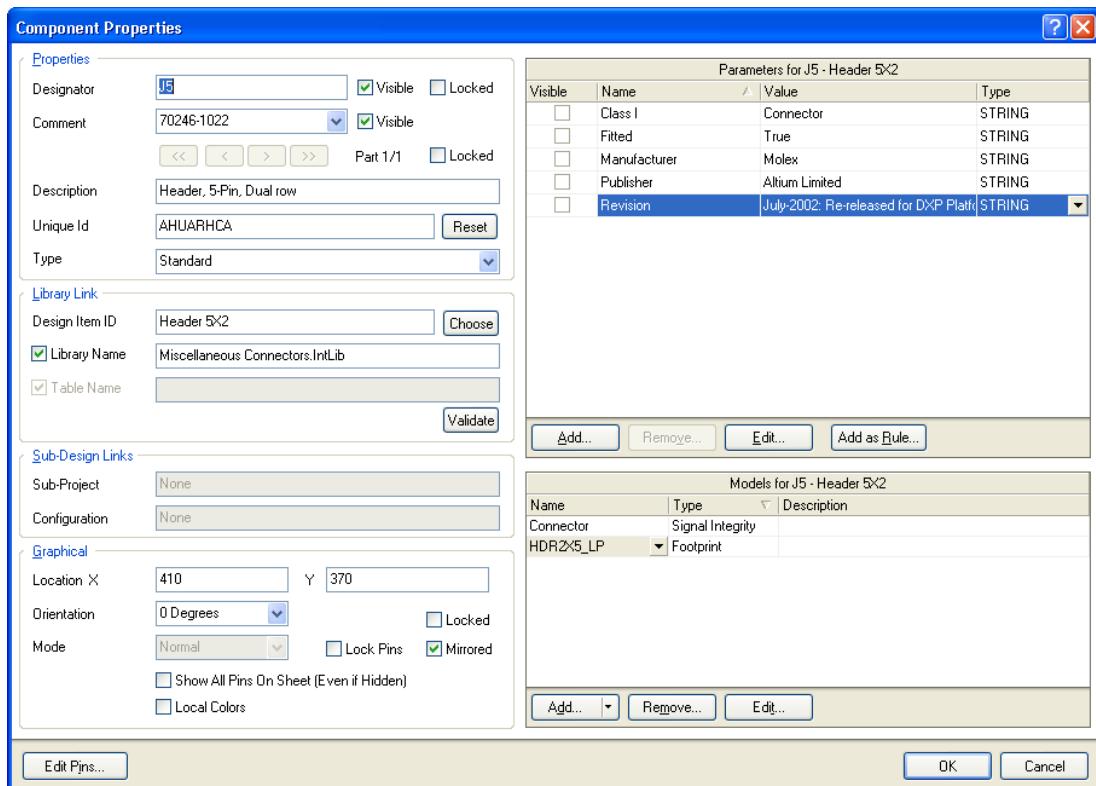
## Editing

The properties of a part object can be modified before, during and after placement. Editing itself falls into two categories - graphical and non-graphical.

The following three methods of non-graphical editing are available:

### Editing via an associated properties dialog

This method of editing uses the following dialog to modify the properties of a part object.



Use the dialog's 'What's This Help' feature to obtain detailed information about each of the individual options available. Click on the question mark button at the top right of the dialog and then click over a field or option to pop-up information specific to that field or option.

The following sections provide an overview of each of the specific regions of the dialog:

### Properties

This section of the dialog contains editable fields for the component comment and description, where available. The designator of the component can be defined here, as well as a **Locked** option to exclude the component from any reannotation of schematic designators.

For multi-part components, an additional **Locked** option can be enabled to prevent reannotation of the sub-parts.

The Unique Id for the component can either be changed by directly typing in the **Unique Id** field, or by pressing the **Reset** button. The latter will result in another system-assigned ID being generated.

You can also change the part number for the selected part, when considering a multi-part component. The designator suffix and pin numbering will be changed to reflect the part number chosen.

The **Type** field allows you to specify what type of component you are using. The following six types are supported:

- **Standard** - standard electrical component loaded onto board. Always synchronized, always in BOM.
- **Mechanical** - non-electrical component, e.g. heat sink or mounting bracket. Synchronized if exists on both schematic and PCB documents, always in BOM.
- **Graphical** - non-electrical component used for company logo, title block, etc. Never synchronized and not included in BOM.
- **Net Tie (In BOM)** - for shorting two (or more) nets together in the routing. Typically used if a jumper type component needs to be fitted and also provide shorting in the same location. Always synchronized and included in BOM.
- **Net Tie** - as above but designed so you couldn't tell a component existed at the location where the shorting is to occur. Always synchronized but not included in BOM. When placing components of this type, use the Verify Shorting Copper option in the Design Rule Checker dialog (when performing a DRC in the PCB), to verify the short (i.e. that no unconnected copper exists in the component).
- **Standard (No BOM)** - standard electrical component loaded onto board. Always synchronized, not included in BOM.

### Library Link

This region of the dialog provides information regarding the library from which the part was placed. It also includes the **Design Item ID** for the part – as it appears in the library. Use the **Choose** button to access the *Browse Libraries* dialog, from where you

can choose (and therefore change) the current part to any other part in the same library, or any library available as part of the Available Libraries list. The component placed from a schematic or integrated library will have its **Design Item ID** set to the symbol reference (library reference).

If the component is placed from a database library then the **Library Link** region will provide information on;

- the component's **Design Item Id** which represents the unique part number and represents a record within the table of a database.
- the parent database library filename in the **Library Name** field
- the specific database table in which the component resides in the **Table Name** field

Again, the **Choose** button will be available to access the *Browse Libraries* dialog. Change the part to another one in the same table of the linked database, or browse to one in a different table of the same or different linked database.

### Sub-Design Links

This region of the dialog provides non-editable information with respect to any sub-design linking for the component. There are two possible scenarios:

- an FPGA sub-project linked to an FPGA component in a PCB project
- an embedded sub-project linked to an MCU core in an FPGA project

The **Configuration** field can only be populated for the first scenario, where a specific configuration has been chosen containing a constraint file in the FPGA project targeting the physical FPGA device on the PCB.

### Graphical

This section of the dialog provides options that allow you to control the orientation and location of the component in the workspace and to set up and implement local colors used to define its fill, lines and pins. Options are also available to allow mirroring of the component and also to show any hidden pins.

The **Mode** drop-down field displays the current graphical representation of the component. Every component has a Normal mode or representation. In addition, a further 255 Alternate graphical representations (modes) of the component can be created. If any Alternate modes have been defined for the current component, the drop-down field will become available and you may select which mode to use for the graphical representation of the component on the schematic document.

### Parameters List

This section of the dialog enables you to define any parameter information for the component. Parameters are a way of defining and associating additional information and could include strings that identify component manufacturer, date added to the document and also a string for the component's value, where applicable (e.g. 100K for a resistor or 10PF for a capacitor).

New parameters can be defined, or existing ones edited or removed. Click the **Add** button to create a new parameter - the *Parameter Properties* dialog will appear. Use this dialog to define a name and value for the parameter and to setup graphical properties that will determine how the parameter information appears in the workspace.

Each defined parameter will appear in the Parameters list, showing its name, value and type. You can control the visibility of each parameter as required, using the corresponding option in the **Visible** column.

Use the **Add as Rule** button to specifically add a design rule directive into the schematic. This feature allows you to define constraints for the design prior to PCB layout. The *Parameter Properties* dialog will appear, with the **Name** and **Type** fields set to **Rule** and **STRING** respectively and inactive.

Clicking the **Edit Rule Values** button will open a dialog listing each of the rule categories and rule types that are available in the PCB document. Double-click on a rule type to open its corresponding *Edit Rule Type* dialog, from where you can define the constraints for the rule.

### Models List

This section of the dialog is used to define links to the following model types:

- Footprint
- Simulation
- PCB3D
- Signal Integrity

You can add any number of new model links or edit/remove existing ones.

Click the **Add** button to open the *Add New Model* dialog, from where you can select which particular model type to add.

For each model link that is created, the name of the model, its associated type and any description is listed.

Use the **Name** column to define which model of each available type is the currently linked model.

To edit an underlying model definition, select the entry for the link and click the **Edit** button (or double-click on the entry). The dialog that appears will depend on the type of model you are editing.

### Dialog access

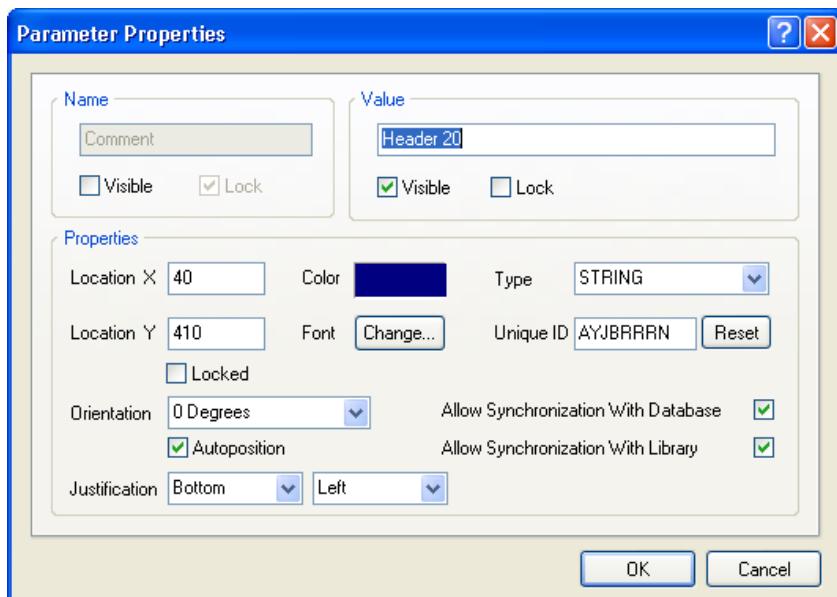
A variant of this dialog - the *Library Component Properties* dialog - can be accessed prior to entering placement mode, from the **Schematic - Default Primitives** page of the *Preferences* dialog (**Tools » Schematic Preferences**). This allows you to change the default properties for the part object, which will be applied when placing subsequent parts.

During placement, the *Component Properties* dialog can be accessed by pressing the **Tab** key.

After placement, the *Component Properties* dialog can be accessed in one of the following ways:

- double-clicking on the placed part object
- selecting the part object and choosing **Properties** from the right-click pop-up menu
- choosing the **Change** command from the **Edit** menu and then clicking once over the placed part object.

The part's **Designator** and **Comment** fields, as well as any user-defined parameters for the part, can be formatted independently of the part itself. In each case the properties of a parameter are available to view/modify in the *Parameter Properties* dialog.



For the Designator and Comment, the parameter name is locked and cannot be changed.

The *Parameter Properties* dialog can be accessed using the three methods described previously (replacing part with the relevant parameter object whose properties you wish to view/modify).

### Editing via the SCH Inspector panel

The **SCH Inspector** panel enables you to interrogate and edit the properties of one or more design objects in the active document. Used in conjunction with appropriate filtering, the panel can be used to make changes to multiple objects of the same kind, from one convenient location.

 For more information on the **SCH Inspector** panel, press **F1** when the cursor is over the panel.

### Editing via the SCH List panel

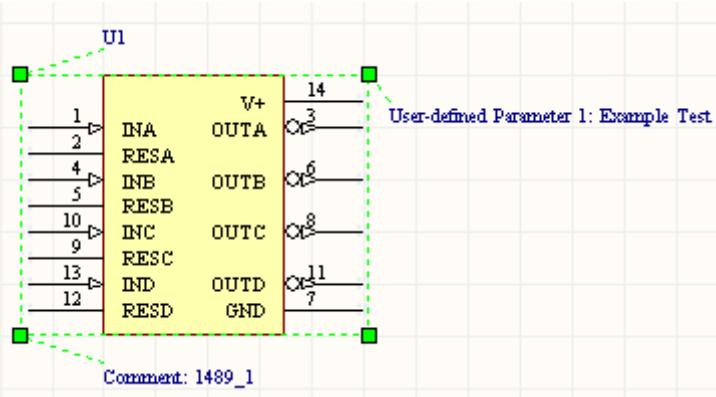
The **SCH List** panel allows you to display design objects from one or more documents in tabular format, enabling you to quickly inspect and modify object attributes. When used in conjunction with the **SCH Filter** panel, it enables you to display just those objects falling under the scope of the active filter - allowing you to target and edit multiple design objects with greater accuracy and efficiency.

 For more information on the **SCH List** and **SCH Inspector** panels, press **F1** when the cursor is over a panel.

## Graphical editing

This method of editing allows you to select a placed part object directly in the workspace and change its location or orientation, graphically. Parts are fixed with respect to their size and shape - changes affecting these attributes can only be carried out on the source library component.

When a part object is selected in the workspace, a dashed box will appear around the part. The box encloses the area occupied by the part. For each text field associated to the part (Designator, Comment, user-defined parameters) a dashed line will be visible, connecting the text field to the body of the part, thereby affirming association:



Click anywhere inside the dashed box and drag to reposition the part as required. The part can be rotated or flipped while dragging.

The part's Designator, Comment and user-defined parameter text fields, which can be graphically edited independently of the parent part, can only be adjusted with respect to their size by changing the size of the font used (accessed through the relevant *Parameter Properties* dialog). As such, editing handles are not available when any of these objects are selected:

**Comment: 1489\_1**

Click anywhere inside the dashed box and drag to reposition the text object as required. The object can be rotated or flipped while dragging:

- Press the **Spacebar** to rotate the text. Rotation is anti-clockwise and in steps of 90°.
- Press the **X** or **Y** keys to flip the text along the X-axis or Y-axis respectively.

If the **Enable In-Place Editing** option is enabled on the **Schematic - General** page of the *Preferences* dialog (**Tools » Schematic Preferences**), you will be able to edit the value for the Designator, Comment or user-defined parameter directly in the workspace. Select the text object and then click once to invoke the feature. Type the new value as required and then click away from the text object or press **Enter** to effect the change.

## Part Actions

Right-clicking over a placed part will pop-up a context-sensitive menu, from which a variety of commands are available that act on that part (or on all selected parts where applicable). The following sections detail each of these commands.

**Note:** Many of the following commands are also available from the Schematic Editor's main menus. Commands on the main menus apply to the selected part(s) or allow you to choose the part on which the command will act, rather than just the part under the cursor. Where such commands exist, reference to their access is made.

### Incrementing the Part Number

This command is used to visually toggle through the available part numbers for the chosen multi-part component on the current document.

The command can be accessed either by:

- choosing **Edit » Increment Part Number** from the main menus. You will be prompted to choose a part. After incrementing a part, you will remain in increment mode, enabling you to increment further parts
- right-clicking over the required part and choosing **Part Actions » Increment Part Number** from the menu that appears.

After launching the command (and choosing the part if applicable), the designator of the part will change to display the next part number for the device and the associated pins for the part will change accordingly.

Using this command on a non-multi-part component will have no effect.

## Pushing a Part onto a New Sub-Sheet

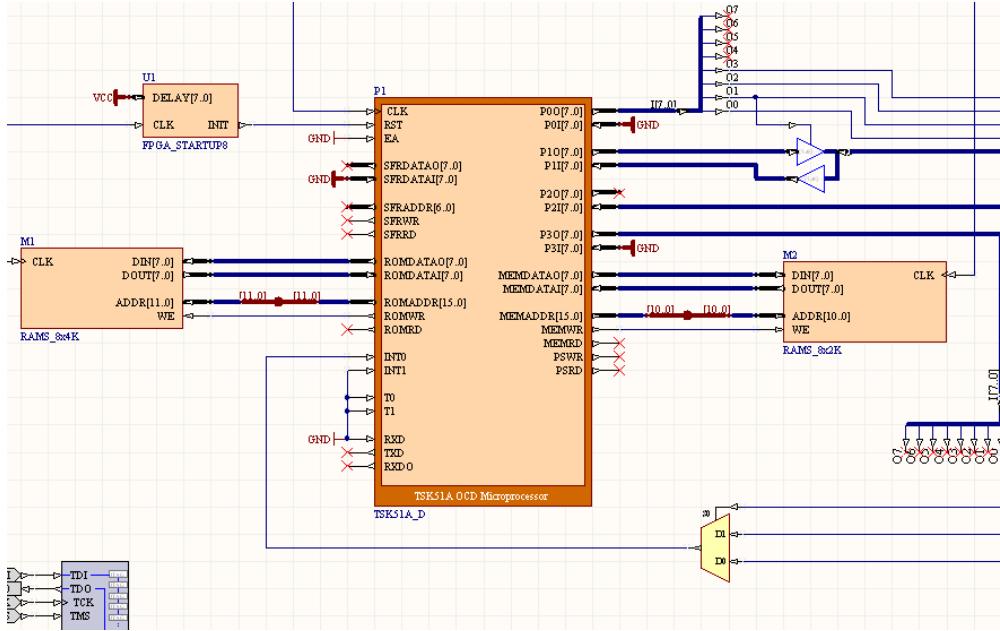
This command allows you to "push" a placed part onto a new schematic sub-sheet. Access this command by:

- choosing **Tools » Convert » Push Part to Sheet** from the main menus. You will be prompted to choose a part
- right-clicking over the required part and choosing **Part Actions » Push Part to Sheet** from the menu that appears.

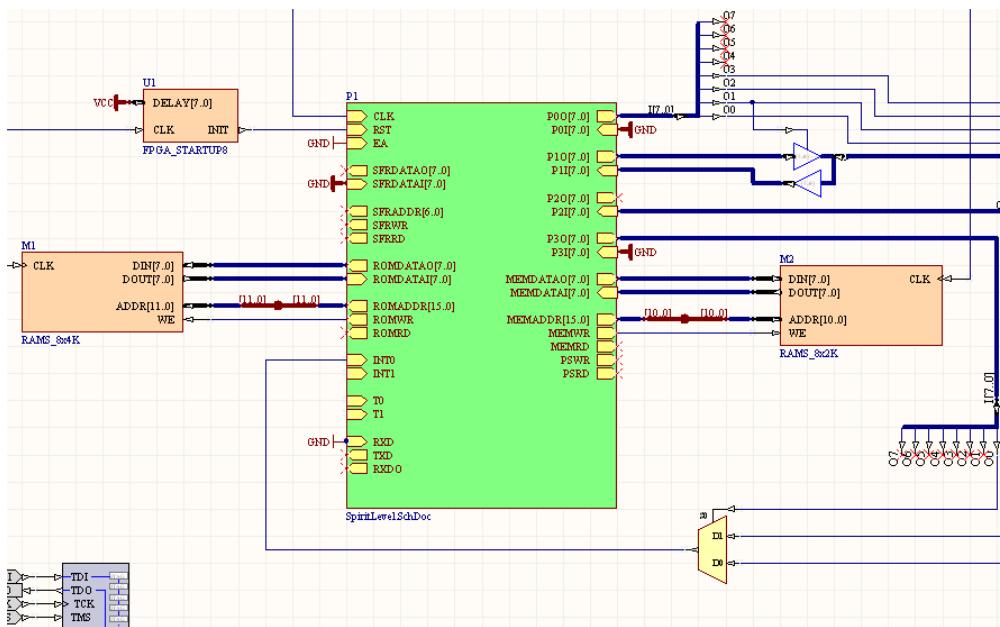
After launching the command (and choosing the part if applicable), the part will be pushed to a newly-created sub-sheet.

Breaking the process down, the following sequence of steps are essentially performed:

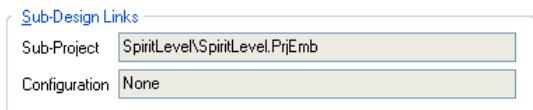
- the chosen part is copied



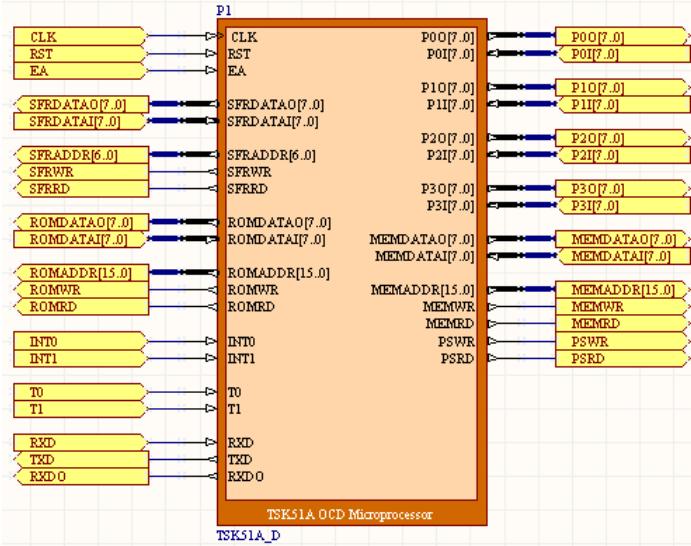
- the original part is converted to a sheet symbol, whose **Designator** field is set to the designator of the original part and whose **Filename** field will have an entry of the format `PartComment.SchDoc`.



**Note:** If the part has a linked sub-project, such as a processor with a linked embedded software project, then the sheet symbol's **Filename** field will have an entry of the format `LinkedProjectName.SchDoc`. In the previous example image, the TSK51A\_D processor has a linked sub-project called `SpiritLevel.PrvEmb`, which can be seen in the **Sub-Design Links** region of the corresponding properties dialog for the part.



- a new schematic sub-sheet is then created from the sheet symbol, the name of which is taken from the symbol's **Filename** field
- the copy of the original part is then pasted onto the center of this sheet and the corresponding ports wired to the part's pins accordingly



**Note:** The new sheet is initially unsaved. You will need to save it and recompile before the modification to the project structure is reflected in the **Projects** panel.

### Converting a Part to a Sheet Symbol

This command is used to convert a chosen part to a sheet symbol.

The command can be accessed either by:

- choosing **Tools** » **Convert** » **Convert Part To Sheet Symbol** from the main menus. You will be prompted to choose a part. After converting a part, you will remain in conversion mode, enabling you to convert further parts
- right-clicking over the required part and choosing **Part Actions** » **Convert Part To Sheet Symbol** from the menu that appears.

After launching the command (and choosing the part if applicable), the part will be converted to a sheet symbol, with **Designator** initially set to the part's designator and **Filename** initially set to the part's comment text. Sheet entries will be created based on the part's pins.

Change the **Filename** for the sheet symbol as required, in order to point to the required schematic sub-sheet.

Change the sheet entries as required, in accordance with the ports defined on the sub-sheet that you wish to reference from the sheet symbol.

### Converting a Part to Ports

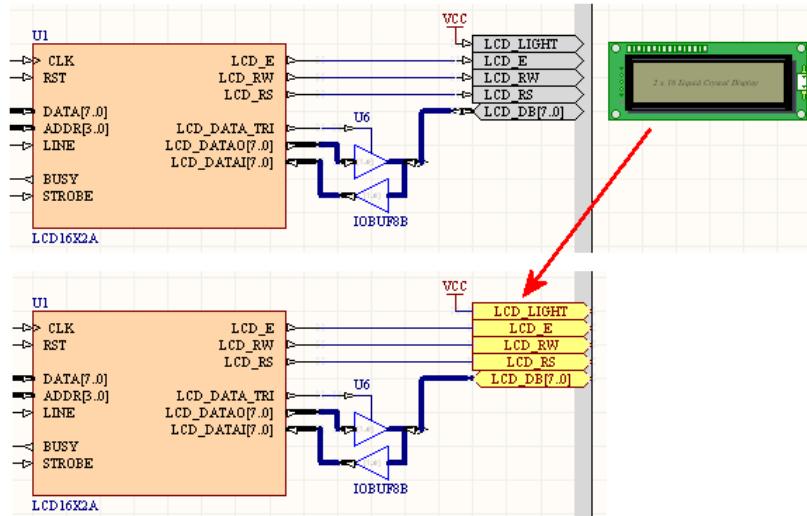
This command is used to convert a chosen port component into standard port primitives.

Although this feature will convert any schematic component into equivalent ports, it is specifically intended for conversion of port components (placed from the FPGA NanoBoard Port-Plugin integrated library). These components illustrate the hardware on the NanoBoard that the signals will eventually be taken to. However, the signals do not connect directly to the illustrated board hardware. Rather they are routed to the physical pins of the FPGA device. It is therefore often clearer to represent the signal lines to these physical pins using standard ports, rather than the port components.

The command can be accessed either by:

- choosing **Tools** » **Convert** » **Convert Part To Ports** from the main menus. You will be prompted to choose a part. After converting a part, you will remain in conversion mode, enabling you to convert further parts
- right-clicking over the required part and choosing **Part Actions** » **Convert Part To Ports** from the menu that appears.

After launching the command (and choosing the part if applicable), the port component will be converted into electrically equivalent ports.



### Creating a VHDL File from a Part

This command is used to create a new VHDL document (\*.Vhd) from a placed part on the current schematic document. Although the command will create a VHDL file from any schematic part, it is intended for use when creating core components.

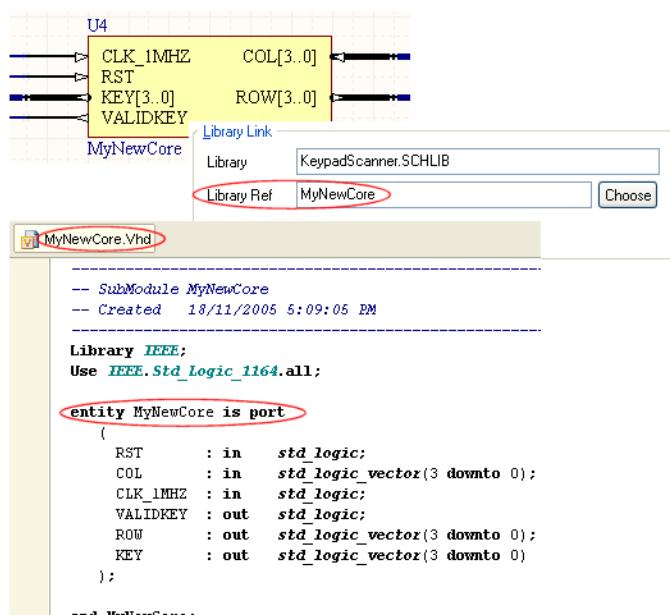
The command can be accessed either by:

- choosing **Tools » Convert » Create VHDL from FPGA-Part** from the main menus. You will be prompted to choose an FPGA part
- right-clicking over the required part and choosing **Part Actions » Create VHDL from FPGA-Part** from the menu that appears.

After launching the command (and choosing the part if applicable), the VHDL document will be created and opened as the active document. The pins on the part will be included as declared ports in the document's entity definition.

The template for the architecture will be defined, but the component and signal declarations are left ready for you to code in.

The VHDL document is named using the entry in the part's **Library Ref** field. This entry is also used as the name for the document's entity.



## Creating a Verilog File from a Part

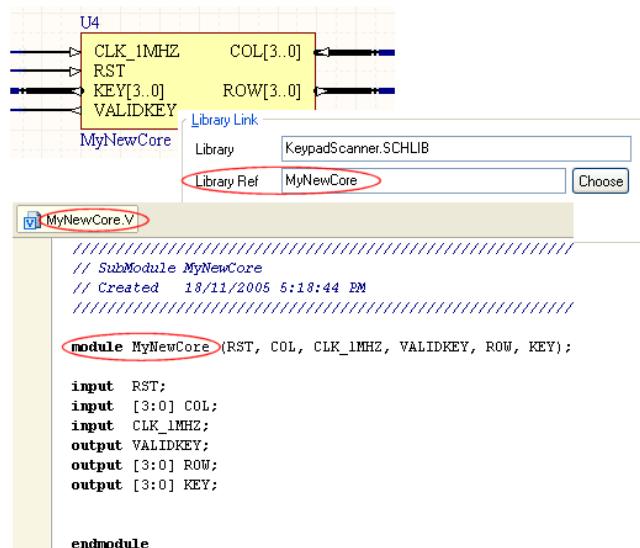
This command is used to create a new Verilog document (\*.v) from a placed part on the current schematic document. Although the command will create a VHDL file from any schematic part, it is intended for use when creating core components.

The command can be accessed either by:

- choosing **Tools » Convert » Create Verilog from FPGA-Part** from the main menus. You will be prompted to choose an FPGA part
- right-clicking over the required part and choosing **Part Actions » Create Verilog from FPGA-Part** from the menu that appears.

After launching the command (and choosing the part if applicable), the Verilog document will be created and opened as the active document. The pins on the part will be included in the document's module definition.

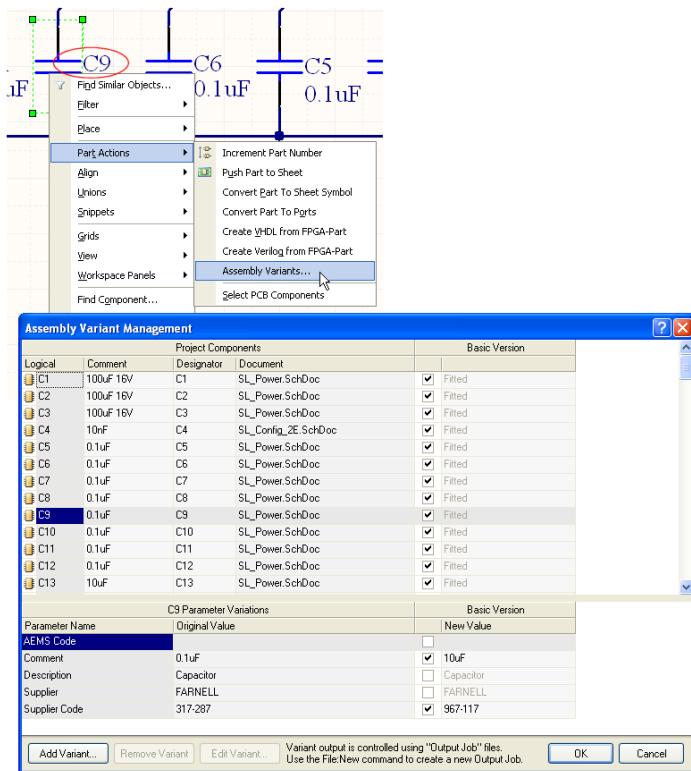
The Verilog document is named using the entry in the part's **Library Ref** field. This entry is also used as the name for the document's module.



## Accessing Variant Information

This command enables you to access the variant-related information for the chosen part directly from the schematic. The command is accessed by right-clicking on the part of interest and choosing the **Part Actions » Assembly Variants** command from the menu that appears.

The *Assembly Variant Management* dialog will open with the target part already selected – enabling you to quickly assess both its inclusion state in any defined board variants, and view its component-level parameters across all variations of the design.



### Cross-Selecting on the PCB

This command is used to cross-select between parts selected on one or more schematic source documents and the corresponding component footprints on the PCB document, for the active project. This can be very useful should you wish to select a set of parts in the source documents and create a new component class quickly in the PCB document.

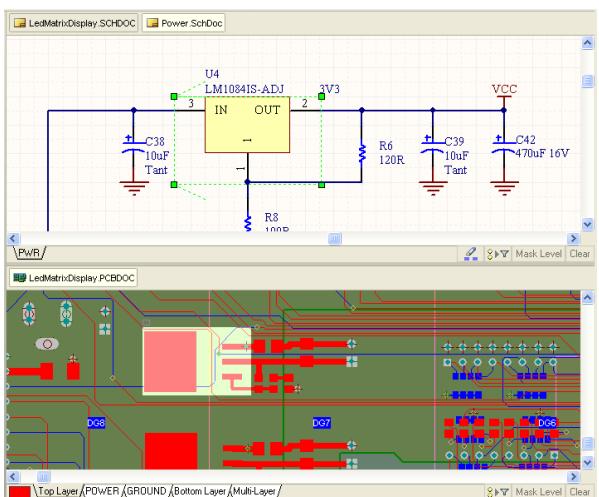
The command can be accessed either by:

- choosing **Tools » Select PCB Components** from the main menus
- right-clicking over the required part (or one part in a selection of parts) and choosing **Part Actions » Select PCB Components** from the menu that appears.

If you wish to cross select for the part under the cursor, use the second method of access, without prior selection. If you wish to cross select multiple parts, ensure they are first selected (on their respective sheets) before launching the command. Also ensure that the target PCB document is open.

After launching the command, the PCB document for the project will become the active document in the main design window and all corresponding component footprints for the selection will become selected. The view of the main design window will change to fit all component footprints in the selection.

For ease of reference, you may find it easier to have the source document(s) and target PCB document open side-by-side in the workspace, for example tiled or as separate windows across dual monitors.



All schematic source documents will be automatically compiled before the PCB document is made active. Those schematic documents that are not currently open as tabbed documents in the main design window will be opened and automatically hidden, to avoid cluttering the design window.

If the active project contains multiple PCB documents, you should open only the document upon which you wish to work/have the component footprints selected. If more than one PCB document is opened, the command will interrogate all documents for a corresponding match to the parts selected on the schematic document(s).

### Notes

When placing multiple instances of the same part, it is advisable to set the designator of the first part before placing it (press the **Tab** key during placement to edit the part's properties). By doing this, the designator will automatically increment as each subsequent part is placed.

---

If you change the Unique ID for the component, the unique ID entry in the corresponding component on the PCB design document will no longer match. You will need to match components again using the *Edit Component Links between Flattened Project and PCB* dialog (from the PCB document).

---

Colors defined for Fill, Lines and Pins will only be used if the **Local Colors** option is enabled. These colors are used to override those that are defined for the component in the source schematic library.

---

For a multi-part component, the relevant pins for the selected part will be highlighted with a white background in the *Component Pin Editor* dialog. All pins of other parts will be lowlighted with a grey background. You are, however, still able to edit the pins of these non-selected parts.

---

Each parameter has a Unique ID assigned to it. This is used for those parameters that have been added as design rule directives. When transferring the design to the PCB document, any defined rule parameters will be used to generate the relevant design rules in the PCB. These generated rules will be given the same Unique IDs, allowing you to change rule constraints in either schematic or PCB and push the change across when performing a synchronization.

---

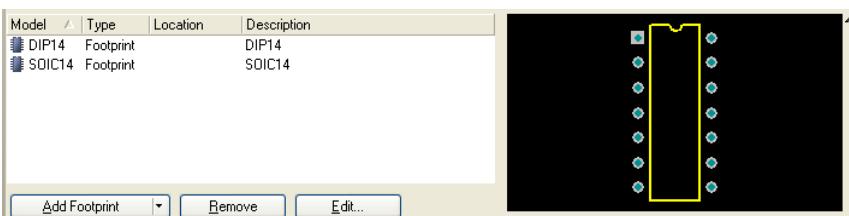
In the Schematic Library Editor, model linking for a component can also be carried out:

- in the **Models** section of the **SCH Library** panel

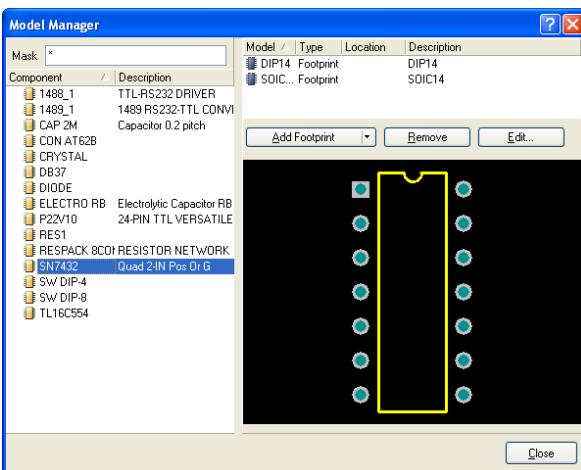
Model	Type	Description
DIP14	Footprint	DIP14
SOIC14	Footprint	SOIC14

**Add**    **Delete**    **Edit**

- in the **Models** section of the main design window



- in the dedicated *Model Manager* dialog (**Tools » Model Manager**)



Only one model of a particular model type can be enabled as the currently linked model, at any one time.

---

The name and description for a model can be edited directly in the **Models List** section of the *Component Properties/Library Component Properties* dialog.

---

Any parameters defined in the **Parameters List** section of the *Component Properties/Library Component Properties* dialog will be made available in the **Matching Options** region of the *Annotate* dialog (**Tools » Annotate Schematics**). This is particularly useful if you wish to group specific parts of a multi-part component, using a unique parameter that you have defined and included for those parts.

---

Any changes made to object properties during placement will cause the default properties for the object to be updated, unless the **Permanent** option - on the **Schematic - Default Primitives** page of the *Preferences* dialog - is enabled. When this option is enabled, changes made will affect only the object being placed and subsequent objects placed during the same placement session.

---

If you attempt to graphically modify a component object that has its **Locked** property enabled, a dialog will appear asking for confirmation to proceed with the edit.

If the **Protect Locked Objects** option is enabled in the **Schematic – Graphical Editing** page of the *Preferences* dialog (**Tools » Schematic Preferences**), and the **Locked** option for this design object is enabled as well then this object cannot be selected or graphically edited.

You will have to double click on this locked object directly and disable the **Locked** property or disable the **Protect Locked Objects** option in the **Schematic – Graphical Editing** Preferences dialog to graphically edit this object.

If you attempt to select locked objects along with other objects, only those objects that are unlocked can be selected and moved as a group if the **Protect Locked Objects** option is enabled. Otherwise if the **Protect Locked Objects** option is disabled you will be prompted for confirmation to proceed with the edit of the objects including locked objects.

## Pie Chart



### Description

A pie chart is a non-electrical drawing primitive. It is a circular sector graphic element that can be placed on a schematic sheet. It can be filled or unfilled.

### Availability

Pie charts are available for placement in both Schematic and Schematic Library Editors:

#### Schematic Editor

- Choose **Place » Drawing Tools » Pie Chart [P, D, C]** from the main menus
- Click the  button on the **Utility Tools** drop-down of the **Utilities** toolbar

#### Schematic Library Editor

- Choose **Place » Pie Chart [P, C]** from the main menus

### Placement

After launching the command, the cursor will change to a cross-hair and you will enter pie chart placement mode. Placement is made by performing the following sequence of actions:

- click or press **Enter** to anchor the center point of the pie chart
- move the cursor to adjust the radius of the pie chart, then click or press **Enter** to set it
- move the cursor to adjust the start point of the segment, then click or press **Enter** to anchor it.
- move the cursor to change the position of the segment's end point, then click or press **Enter** to anchor it and complete placement of the pie chart.

Continue placing further pie charts, or right-click or press **Esc** to exit placement mode.

The pie chart object can be rotated or flipped while in placement mode and before the center of the pie chart is anchored:

- Press the **Spacebar** to rotate the pie chart. Rotation is anti-clockwise and in steps of 90°.
- Press the **X** or **Y** keys to flip the pie chart along the X-axis or Y-axis respectively.

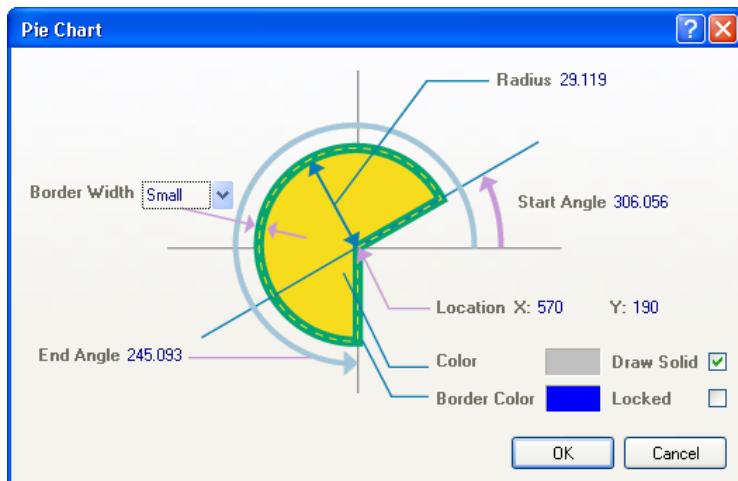
### Editing

The properties of a pie chart object can be modified before, during and after placement. Editing itself falls into two categories - graphical and non-graphical.

The following three methods of non-graphical editing are available:

#### Editing via an associated properties dialog

This method of editing uses the following dialog to modify the properties of a pie chart object.



Use the dialog's 'What's This Help' feature to obtain detailed information about each of the options available. Click on the question mark button at the top right of the dialog and then click over a field or option to pop-up information specific to that field or option.

The *Pie Chart* dialog can be accessed prior to entering placement mode, from the **Schematic - Default Primitives** page of the **Preferences** dialog (**Tools » Schematic Preferences**). This allows you to change the default properties for the pie chart object, which will be applied when placing subsequent pie charts.

During placement, the *Pie Chart* dialog can be accessed by pressing the **Tab** key.

After placement, the *Pie Chart* dialog can be accessed in one of the following ways:

- double-clicking on the placed pie chart object
- selecting the pie chart object and choosing **Properties** from the right-click pop-up menu (Schematic Editor only)
- choosing the **Change** command from the **Edit** menu and then clicking once over the placed pie chart object.

#### Editing via the SCH Inspector panel

The **SCH Inspector** panel enables you to interrogate and edit the properties of one or more design objects in the active document. Used in conjunction with appropriate filtering, the panel can be used to make changes to multiple objects of the same kind, from one convenient location.

For more information on the **SCH Inspector** panel, press **F1** when the cursor is over the panel.

#### Editing via the SCH List panel

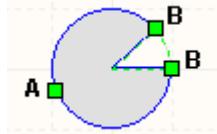
The **SCH List** panel allows you to display design objects from one or more documents in tabular format, enabling you to quickly inspect and modify object attributes. When used in conjunction with the **SCH Filter** panel, it enables you to display just those objects falling under the scope of the active filter - allowing you to target and edit multiple design objects with greater accuracy and efficiency.

For more information on the **SCH List** and **SCH Inspector** panels, press **F1** when the cursor is over a panel.

#### Graphical editing

This method of editing allows you to select a placed pie chart object directly in the workspace and change its size, shape or location, graphically.

When a pie chart object is selected, the following editing handles are available:



Click and drag **A** to adjust the radius.

Click and drag **B** to adjust the end points.

Click anywhere on the pie chart - away from editing handles - and drag to reposition it. The pie chart can be rotated or flipped while dragging.

If you attempt to graphically modify a pie chart object that has its **Locked** property enabled, a dialog will appear asking for confirmation to proceed with the edit.

If the **Protect Locked Objects** option is enabled in the **Schematic – Graphical Editing** page of the *Preferences* dialog (**Tools** » **Schematic Preferences**), and the **Locked** option for this design object is enabled as well then this object cannot be selected or graphically edited.

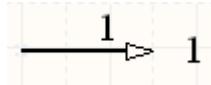
You will have to double click on this locked object directly and disable the **Locked** property or disable the **Protect Locked Objects** option in the **Schematic – Graphical Editing** Preferences dialog to graphically edit this object.

If you attempt to select locked objects along with other objects, only those objects that are unlocked can be selected and moved as a group if the **Protect Locked Objects** option is enabled. Otherwise if the **Protect Locked Objects** option is disabled you will be prompted for confirmation to proceed with the edit of the objects including locked objects.

### Notes

Any changes made to object properties during placement will cause the default properties for the object to be updated, unless the **Permanent** option - on the **Schematic - Default Primitives** page in the **Schematic** section of the *Preferences* dialog - is enabled. When this option is enabled, changes made will affect only the object being placed and subsequent objects placed during the same placement session.

## Pin



### Description

A pin is an electrical design primitive. Pins give a part its electrical properties and define connection points on the part for directing signals in and out.

### Availability

Pins are available for placement in the Schematic Library Editor only. Use one of the following methods to place a pin:

- choose **Place » Pin [P, P]** from the main menus
- click the button on the **Utility Tools** drop-down of the **Utilities** toolbar
- click the **Add** button in the **Pin List** section of the **SCH Library** panel
- click the **Add** button in the *Component Pin Editor* dialog

### Placement

The way in which a pin is placed on a sheet depends on how, and from where, placement mode is invoked.

#### Placement using menu or toolbar command

After launching the command, the cursor will change to a cross-hair and you will enter pin placement mode. The pin will appear floating on the cursor held by its electrical end, which goes against the component body. Position the cursor and click or press **Enter** to effect placement.

Continue placing further pins or right-click or press **Esc** to exit placement mode.

Press the **Spacebar** while in placement mode to rotate the pin. Rotation is anti-clockwise and in steps of 90°.

Press the **X** or **Y** keys while in placement mode to flip the pin along the X-axis or Y-axis respectively.

#### Placement from SCH Library panel

Pins can be added directly from the **SCH Library** panel.

First ensure that the component to which you wish to add one or more pins to is active in the main design window. To do this, select its entry in the main **Components** list section of the panel.

To add a new pin, click on the **Add** button beneath the **Pin List** section of the panel (which lists all pins currently defined for the active component). The pin will appear floating on the cursor in the **Symbol** section of the main design window, held by its electrical end, which goes against the component body. Position the cursor and click or press **Enter** to effect placement. Upon placement, an entry for the new pin will be added to the **Pin List** section of the panel.

Continue placing further pins or right-click or press **Esc** to exit placement mode.

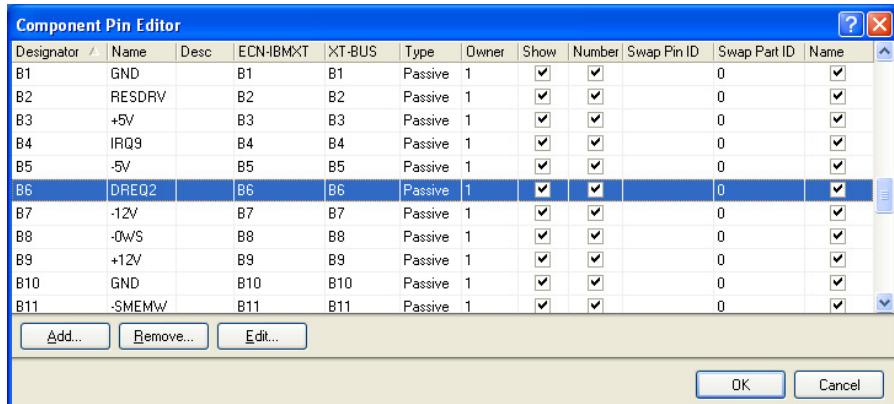
Press the **Spacebar** while in placement mode to rotate the pin. Rotation is anti-clockwise and in steps of 90°.

Press the **X** or **Y** keys while in placement mode to flip the pin along the X-axis or Y-axis respectively.



#### Placement from Component Pin Editor dialog

Pins can be placed directly from the *Component Pin Editor* dialog.



To place an additional pin on the sheet for the active component in the library, click the **Add** button at the bottom left of the dialog.

Only a single pin is placed directly on the sheet. You do not remain in placement mode and the pin cannot be rotated or flipped.

The Designator of the new pin will be the next available unused designator. If one or more pins have previously been removed, such that the designators of the remaining pins are not sequentially continuous, a new pin will be given the first available unused designator in the sequence.

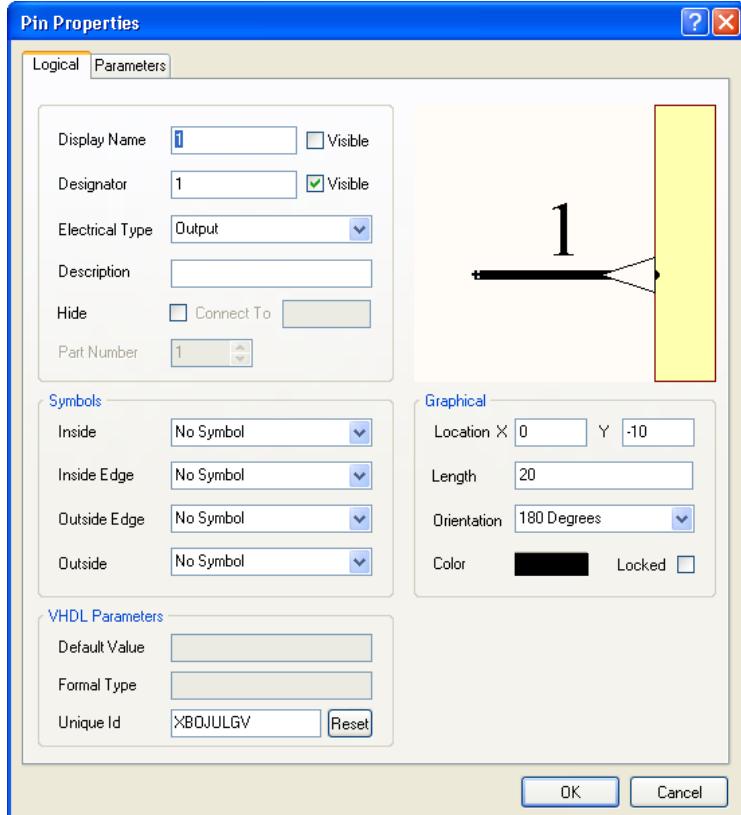
## Editing

The properties of a pin object can be modified before, during and after placement. Editing itself falls into two categories - graphical and non-graphical.

The following five methods of non-graphical editing are available:

### Editing via an associated properties dialog

This method of editing uses the following dialog to modify the properties of a pin object.



Use the dialog's 'What's This Help' feature to obtain detailed information about each of the options available. Click on the question mark button at the top right of the dialog and then click over a field or option to pop-up information specific to that field or option.

The *Pin Properties* dialog can be accessed prior to entering placement mode, from the **Schematic - Default Primitives** page of the *Preferences* dialog (**Tools** » **Schematic Preferences**). This allows you to change the default properties for the pin object, which will be applied when placing subsequent pins.

During placement, the *Pin Properties* dialog can be accessed by pressing the **Tab** key.

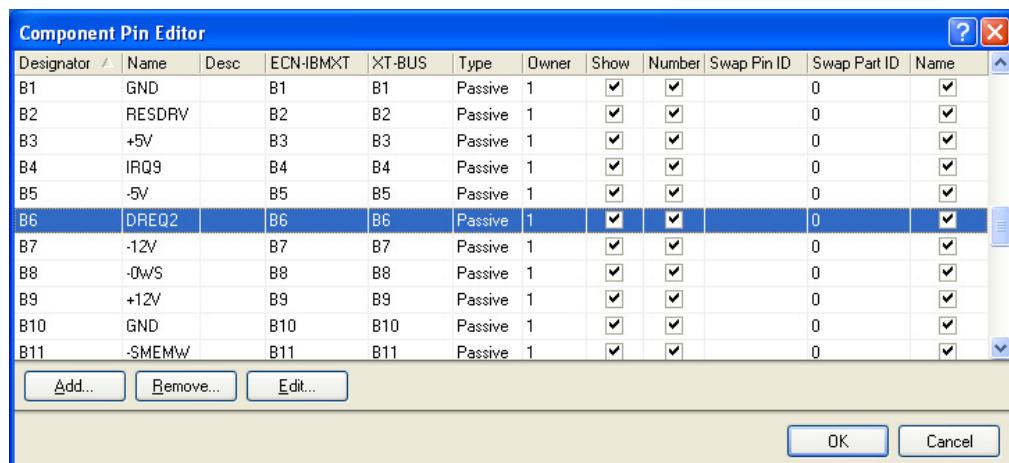
After placement, the *Pin Properties* dialog can be accessed in one of the following ways:

- double-clicking on the placed pin object in the **Symbol** section of the Schematic Library Editor's main design window
- choosing the **Change** command from the **Edit** menu and then clicking once over the placed pin object
- double-clicking on the pin's entry in the **Pin List** section of the **SCH Library** panel
- selecting the pin's entry in the **Pin List** section of the **SCH Library** panel and then clicking on the associated **Edit** button for the section
- double-clicking on the required pin entry (or selecting it and pressing the **Edit** button) in the *Component Pin Editor* dialog. This dialog is accessed by clicking the **Edit Pins** button at the bottom left of the *Library Component Properties* dialog, which itself is accessed by double-clicking on the pin's parent component entry in the **Components** list of the **SCH Library** panel (or, alternatively, selecting the component entry and clicking the **Edit** button beneath the list).

The *Pin Properties* dialog can also be accessed from the Schematic Editor, once a part has been placed. Double-click on the placed part to access the *Component Properties* dialog. When the dialog appears, click on the **Edit Pins** button at the bottom left, to access the *Component Pin Editor* dialog. Then either double-click on the required pin entry or select the entry and press the **Edit** button, to bring up the *Pin Properties* dialog.

#### Editing via the Component Pin Editor

Certain pin properties can be edited from the *Component Pin Editor* dialog, which is accessed by pressing the **Edit Pins** button in either the *Component Properties* or *Library Component Properties* dialogs.



The dialog contains all pins associated with either:

- the selected, placed component in the Schematic Editor or
- the active component in the current schematic library document.

With the exception of fields displaying mapping information for any models linked to the parent part, displayed fields for a pin are directly editable within the dialog. Click once on a field to focus it for editing and then type the value or select the option as required. Click away from the field or press **Enter** to effect the change.

The following table shows the corresponding entries for each of these fields in the *Pin Properties* dialog.

Field in Component Pin Editor dialog	Corresponding field in Pin Properties dialog
Designator	Designator
Name	Display Name
Desc	Description
Type	Electrical Type
Owner	Part Number

Show	Hide
Number	Visible option for Designator
Name	Visible option for Display Name

In addition to providing a means of editing pin properties, the *Component Pin Editor* dialog also allows you to add new pins or delete existing ones.

#### Editing via the SCH Inspector panel

The **SCH Inspector** panel enables you to interrogate and edit the properties of one or more design objects in the active document. Used in conjunction with appropriate filtering, the panel can be used to make changes to multiple objects of the same kind, from one convenient location.

 For more information on the **SCH Inspector** panel, press **F1** when the cursor is over the panel.

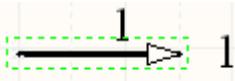
#### Editing via the SCH List panel

The **SCH List** panel allows you to display design objects from one or more documents in tabular format, enabling you to quickly inspect and modify object attributes. When used in conjunction with the **SCH Filter** panel, it enables you to display just those objects falling under the scope of the active filter - allowing you to target and edit multiple design objects with greater accuracy and efficiency.

 For more information on the **SCH List** and **SCH Inspector** panels, press **F1** when the cursor is over a panel.

#### Graphical editing

This method of editing allows you to select a placed pin object directly in the workspace and change its location or orientation, graphically. Pins can only be adjusted with respect to their size by changing their **Length** property (accessed through the *Pin Properties* dialog). As such, editing handles are not available when the pin object is selected:



Click anywhere inside the dashed box and drag to reposition the pin object as required. The object can be rotated or flipped while dragging.

If you attempt to graphically modify a pin object that has its **Locked** property enabled, a dialog will appear asking for confirmation to proceed with the edit.

If the **Protect Locked Objects** option is enabled in the **Schematic – Graphical Editing** page of the *Preferences* dialog (**Tools** » **Schematic Preferences**), and the **Locked** option for this design object is enabled as well then this object cannot be selected or graphically edited.

You will have to double click on this locked object directly and disable the **Locked** property or disable the **Protect Locked Objects** option in the **Schematic – Graphical Editing** Preferences dialog to graphically edit this object.

If you attempt to select locked objects along with other objects, only those objects that are unlocked can be selected and moved as a group if the **Protect Locked Objects** option is enabled. Otherwise if the **Protect Locked Objects** option is disabled you will be prompted for confirmation to proceed with the edit of the objects including locked objects.

#### Notes

While placing pins on the workspace, the pin name and pin number will automatically increment if the first or last characters are numbers.

Should you need to negate (include a bar over the top of) a pin name, this can be done in one of two ways:

- Include a backslash character after each character in the name (e.g. E\N\A\B\L\E\)
- Enable the **Single '\ Negation** option on the **Schematic - Graphical Editing** page of the *Preferences* dialog, then include one backslash character at the start of the name (e.g. \ENABLE).

When you are creating a component that represents a physical multi-pin, multi-part device, such as an IC, you should number the pins carefully to correspond to the physical pinout of the device.

As well as graphical properties, pins also have electrical properties. On the **Logical** tab of the *Pin Properties* dialog, set the **Electrical Type** field to represent the type of electrical connection the pin makes. This can be used by the Compiler to detect electrical wiring errors in your schematic.

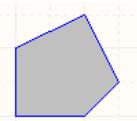
---

Enable the **Hide** option on the **Logical** tab of the *Pin Properties* dialog (or disable the **Show** option if editing directly using the *Component Pin Editor* dialog) to have the pin hidden when it is placed in a schematic. As well as being hidden in the schematic, hidden pins also have the property that they are automatically assumed to be connected to any other hidden pins with the same name and to any net on the schematic with the same net name. This is normally used to define power and ground pins on a component, which will then automatically be connected to power and ground nets of the same name. Use the associated **Connect To** field in the *Pin Properties* dialog to specifically define the net to which the hidden pin should be attached.

---

Any changes made to object properties during placement will cause the default properties for the object to be updated, unless the **Permanent** option - on the **Schematic - Default Primitives** page of the *Preferences* dialog - is enabled. When this option is enabled, changes made will affect only the object being placed and subsequent objects placed during the same placement session.

## Polygon



### Description

A polygon is a non-electrical drawing primitive. It is a multi-sided graphical object that can be placed on a schematic sheet. A polygon must have at least three sides and can be filled or unfilled.

### Availability

Polygons are available for placement in both Schematic and Schematic Library Editors:

#### Schematic Editor

- Choose **Place » Drawing Tools » Polygon [P, D, Y]** from the main menus
- Click the button on the **Utility Tools** drop-down of the **Utilities** toolbar

#### Schematic Library Editor

- Choose **Place » Polygon [P, Y]** from the main menus
- Click the button on the **Utility Tools** drop-down of the **Utilities** toolbar

### Placement

After launching the command, the cursor will change to a cross-hair and you will enter polygon placement mode. Placement is made by performing the following sequence of actions:

- click or press **Enter** to anchor the starting point for the polygon
- position the cursor and click or press **Enter** to anchor a series of vertex points that define the shape of the polygon
- after placing the final vertex point, right-click or press **Esc** to complete placement of the polygon.

Continue placing further polygons, or right-click or press **Esc** to exit placement mode.

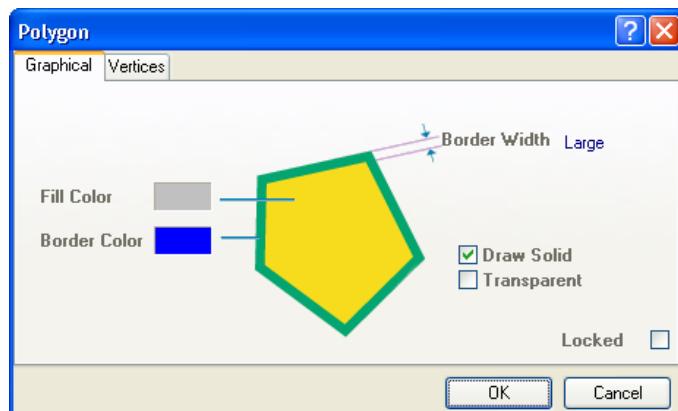
### Editing

The properties of a polygon object can be modified before, during and after placement. Editing itself falls into two categories - graphical and non-graphical.

The following three methods of non-graphical editing are available:

#### Editing via an associated properties dialog

This method of editing uses the following dialog to modify the properties of a polygon object.



Use the dialog's 'What's This Help' feature to obtain detailed information about each of the options available. Click on the question mark button at the top right of the dialog and then click over a field or option to pop-up information specific to that field or option.

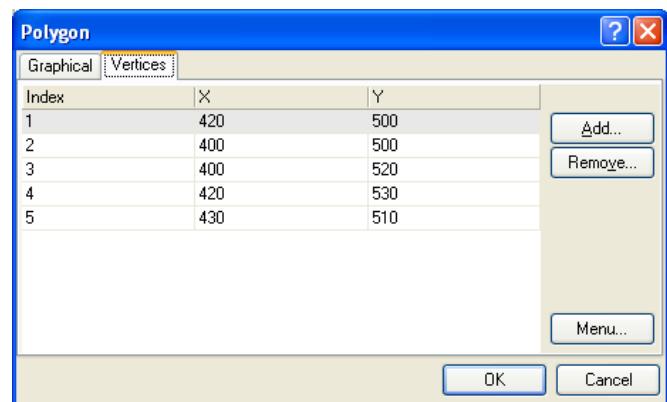
## Editing Vertices

The **Polygon** dialog provides a **Vertices** tab, from where you can edit the individual vertices of the currently selected polygon object as required.

The main region of the tab lists all of the vertex points currently defined for the polygon. You can add new vertices to the polygon, edit the coordinates of existing vertices, or remove selected vertices altogether.

Click the **Menu** button or right-click within the main list region to access a pop-up menu containing the following commands:

- **Edit** - right click on a coordinate cell (X or Y) for a vertex and use this command to edit the value in that cell.  
Alternatively, click directly on the cell
- **Add** - use this command to add a new vertex point. The new vertex will be added below the currently focused vertex entry (as distinguished by a dotted outline around a cell in its row) and will initially have the same coordinates as the focused entry
- **Remove** - use this command to remove the currently selected vertex entries in the list. This command will be unavailable if there are only two vertices present for the polygon
- **Copy** - use this command to copy the content of the selected cells in the list to the clipboard (alternatively use **Ctrl+C**)
- **Paste** - use this command to paste the content of the clipboard into the list, starting at the selected cell (alternatively use **Ctrl+V**)
- **Select All** - use this command to quickly select the entire grid contents of the list
- **Select Column** - use this command to quickly select the entire column in which the currently focused cell resides
- **Move Up** - use this command to move the selected vertex upward in the list
- **Move Down** - use this command to move the selected vertex downward in the list
- **Move Polygon By XY** - use this command to move the entire polygon object. The Move Polygon By dialog will appear, from where you can enter the increment value to be applied to each vertex point's X and Y coordinates.



## Dialog access

The **Polygon** dialog can be accessed prior to entering placement mode, from the **Schematic - Default Primitives** page of the **Preferences** dialog (**Tools » Schematic Preferences**). This allows you to change the default properties for the polygon object, which will be applied when placing subsequent polygons.

During placement, the **Polygon** dialog can be accessed by pressing the **Tab** key.

After placement, the **Polygon** dialog can be accessed in one of the following ways:

- double-clicking on the placed polygon object
- selecting the polygon object and choosing **Properties** from the right-click pop-up menu (Schematic Editor only)
- choosing the **Change** command from the **Edit** menu and then clicking once over the placed polygon object.

## Editing via the SCH Inspector panel

The **SCH Inspector** panel enables you to interrogate and edit the properties of one or more design objects in the active document. Used in conjunction with appropriate filtering, the panel can be used to make changes to multiple objects of the same kind, from one convenient location.

For more information on the **SCH Inspector** panel, press **F1** when the cursor is over the panel.

## Editing via the SCH List panel

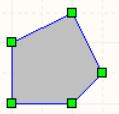
The **SCH List** panel allows you to display design objects from one or more documents in tabular format, enabling you to quickly inspect and modify object attributes. When used in conjunction with the **SCH Filter** panel, it enables you to display just those objects falling under the scope of the active filter - allowing you to target and edit multiple design objects with greater accuracy and efficiency.

For more information on the **SCH List** and **SCH Inspector** panels, press **F1** when the cursor is over a panel.

## Graphical editing

This method of editing allows you to select a placed polygon object directly in the workspace and change its size, shape or location, graphically.

When a polygon object is selected, the following editing handles are available:



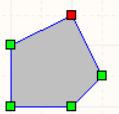
Click and drag an editing handle to move a vertex point of the polygon.

Click anywhere on the polygon - away from editing handles - and drag to reposition it. The polygon can be rotated or flipped while dragging:

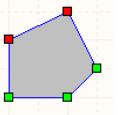
- Press the **Spacebar** to rotate the polygon. Rotation is anti-clockwise and in steps of 90°.
- Press the **X** or **Y** keys to flip the polygon along the X-axis or Y-axis respectively.

Right-click on a vertex point and choose the **Edit Polygon Vertex n** command to access the **Vertices** tab of the *Polygon* dialog, with the entry for the **n**th vertex selected ready for editing.

With the polygon selected, click on a vertex or segment to individually select that vertex or segment. This polygon 'sub-selection' is distinguished by the associated editing handles becoming red in color.



Individual vertex sub-selection.



Individual segment sub-selection.

The associated vertex (or vertices for a segment) can then be edited directly using the **SCH Inspector** or **SCH List** panels, with any changes appearing immediately on the schematic.

If you attempt to graphically modify a polygon object that has its **Locked** property enabled, a dialog will appear asking for confirmation to proceed with the edit.

If the **Protect Locked Objects** option is enabled in the **Schematic – Graphical Editing** page of the *Preferences* dialog (**Tools** » **Schematic Preferences**), and the **Locked** option for this design object is enabled as well then this object cannot be selected or graphically edited.

You will have to double click on this locked object directly and disable the **Locked** property or disable the **Protect Locked Objects** option in the **Schematic – Graphical Editing** Preferences dialog to graphically edit this object.

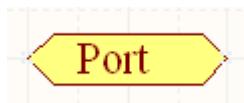
If you attempt to select locked objects along with other objects, only those objects that are unlocked can be selected and moved as a group if the **Protect Locked Objects** option is enabled. Otherwise if the **Protect Locked Objects** option is disabled you will be prompted for confirmation to proceed with the edit of the objects including locked objects.

## Notes

Complex polygonal shapes are supported. A complex polygon is a self-intersecting or self-crossing polygon.

Any changes made to object properties during placement will cause the default properties for the object to be updated, unless the **Permanent** option - on the **Schematic - Default Primitives** page of the *Preferences* dialog - is enabled. When this option is enabled, changes made will affect only the object being placed and subsequent objects placed during the same placement session.

## Port



### Description

A port is an electrical design primitive. It is used to make an electrical connection between one schematic sheet and another sheet or sheet symbol in a design using multiple sheets (both flat and hierarchical designs). The name of the port defines the connection (i.e. a port on a schematic sheet connects to ports or sheet entries with the same name on other sheets in the project).

### Availability

Ports are available for placement in the Schematic Editor only. Use one of the following methods to access the placement command:

- choose **Place » Port [P, R]** from the main menus
- click the  button on the **Wiring** toolbar.

### Placement

After launching the command, the cursor will change to a cross-hair and you will enter port placement mode. Placement is made by performing the following sequence of actions:

- click or press **Enter** to anchor the left hand edge of the port
- move the cursor to adjust the radius of the pie chart, then click or press **Enter** to set it
- move the cursor to adjust the start point of the segment, then click or press **Enter** to anchor it.
- move the cursor to adjust the length of the port as required, then click or press **Enter** to complete placement of the port.

Continue placing further ports or right-click or press **Esc** to exit placement mode.

The port object can be rotated or flipped while in placement mode and before the left hand edge of the port is anchored:

- Press the **Spacebar** to rotate the port. Rotation is anti-clockwise and in steps of 90°.
- Press the **X** or **Y** keys to flip the port along the X-axis or Y-axis respectively.

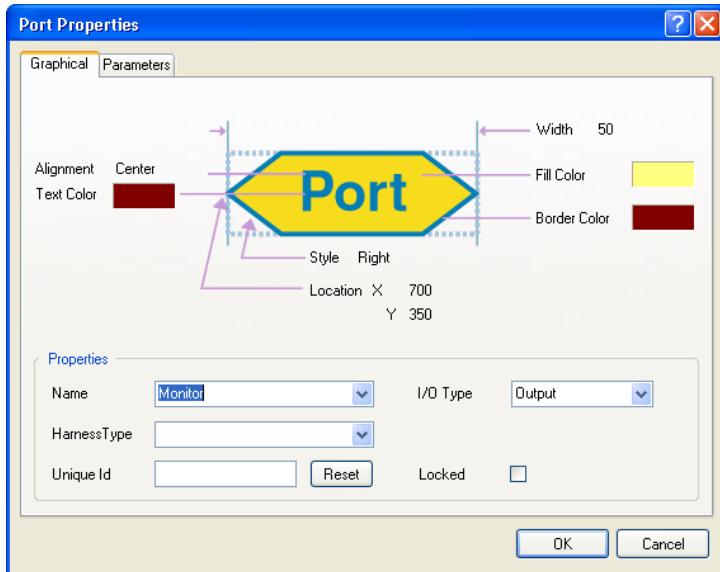
### Editing

The properties of a port object can be modified before, during and after placement. Editing itself falls into two categories - graphical and non-graphical.

The following three methods of non-graphical editing are available:

#### Editing via an associated properties dialog

This method of editing uses the following dialog to modify the properties of a port object.



Use the dialog's 'What's This Help' feature to obtain detailed information about each of the options available. Click on the question mark button at the top right of the dialog and then click over a field or option to pop-up information specific to that field or option.

The **Port Properties** dialog can be accessed prior to entering placement mode, from the **Schematic - Default Primitives** page of the **Preferences** dialog (**Tools » Schematic Preferences**). This allows you to change the default properties for the port object, which will be applied when placing subsequent ports.

During placement, the **Port Properties** dialog can be accessed by pressing the **Tab** key.

After placement, the **Port Properties** dialog can be accessed in one of the following ways:

- double-clicking on the placed port object
- selecting the port object and choosing **Properties** from the right-click pop-up menu
- choosing the **Change** command from the **Edit** menu and then clicking once over the placed port object.

#### Editing via the SCH Inspector panel

The **SCH Inspector** panel enables you to interrogate and edit the properties of one or more design objects in the active document. Used in conjunction with appropriate filtering, the panel can be used to make changes to multiple objects of the same kind, from one convenient location.

For more information on the **SCH Inspector** panel, press **F1** when the cursor is over the panel.

#### Editing via the SCH List panel

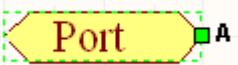
The **SCH List** panel allows you to display design objects from one or more documents in tabular format, enabling you to quickly inspect and modify object attributes. When used in conjunction with the **SCH Filter** panel, it enables you to display just those objects falling under the scope of the active filter - allowing you to target and edit multiple design objects with greater accuracy and efficiency.

For more information on the **SCH List** and **SCH Inspector** panels, press **F1** when the cursor is over a panel.

#### Graphical editing

This method of editing allows you to select a placed port object directly in the workspace and change its length or location graphically.

When a port object is selected, the following editing handles are available:



Click and drag **A** to change the length of the port.

Click anywhere on the port - away from editing handles - and drag to reposition it. The port can be rotated or flipped while dragging.

If the **Enable In-Place Editing** option is enabled on the **Schematic - General** page of the *Preferences* dialog (**Tools » Schematic Preferences**), you will be able to edit the name for the port directly in the workspace. Select the port and then click once to invoke the feature. Type the new name as required and then click away from the port or press **Enter** to effect the change.

If you attempt to graphically modify a port object that has its **Locked** property enabled, a dialog will appear asking for confirmation to proceed with the edit.

If the **Protect Locked Objects** option is enabled in the **Schematic – Graphical Editing** page of the *Preferences* dialog (**Tools » Schematic Preferences**), and the **Locked** option for this design object is enabled as well then this object cannot be selected or graphically edited.

You will have to double click on this locked object directly and disable the **Locked** property or disable the **Protect Locked Objects** option in the **Schematic – Graphical Editing** Preferences dialog to graphically edit this object.

If you attempt to select locked objects along with other objects, only those objects that are unlocked can be selected and moved as a group if the **Protect Locked Objects** option is enabled. Otherwise if the **Protect Locked Objects** option is disabled you will be prompted for confirmation to proceed with the edit of the objects including locked objects.

## Notes

All Ports with the same name, within one schematic document, are considered to be electrically connected.

---

When a Port is connected to a Signal Harness, the Port becomes a Harness object. By default, the Port will change color to match the color of the Signal Harness. Disable the option, **Sheet Entries and Ports use Harness Color** in **Schematic – Graphical Editing** page of the *Preferences* dialog to specify your own color for Ports or to use the default color.

---

When a Port is connected to a Harness Connector by a Signal Harness, the **Harness Type** in the *Port Properties* dialog is automatically populated with the **Harness Type** of the Harness Connector. When a Port is connected to a Sheet Entry by a Signal Harness and the Sheet Entry has a Harness Type declared, the Port will become a Harness object and change to the color of the Signal Harness. If you move the Port away from the Harness Connector or the Sheet Entry, the Port will revert back to the default color.

---

The **I/O Type** option in the *Port Properties* dialog allows you to define the port's electrical type. Choose from either **Input**, **Output**, **Bidirectional** or **Unspecified**.

---

Should you need to negate (include a bar over the top of) a port name, this can be done in one of two ways:

- Include a backslash character after each character in the name (e.g. E\N\A\B\L\E\)
- Enable the **Single '\'' Negation** option on the **Schematic - Graphical Editing** page of the *Preferences* dialog, then include one backslash character at the start of the name (e.g. \ENABLE).

---

When compiling a schematic or generating a netlist, the relationship between ports and sheet symbols is determined by the Net Identifier Scope chosen for the project. This scope is defined by setting the **Net Identifier Scope** option in the **Options** tab of the *Options for Project* dialog (**Project » Project Options**).

---

Any changes made to object properties during placement will cause the default properties for the object to be updated, unless the **Permanent** option - on the **Schematic - Default Primitives** page of the *Preferences* dialog - is enabled. When this option is enabled, changes made will affect only the object being placed and subsequent objects placed during the same placement session.

## Power Port



### Description

A power port is an electrical design primitive. It is a special schematic object that lets you easily define a power or ground net. Power ports allow you to conveniently indicate a power net at any location in the design, which can then be connected to pins or wires.

### Availability

Power ports are available for placement in the Schematic Editor only. Use one of the following methods to access a placement command:

- choose **Place » Power Port [P, O]** from the main menus
- click the button on the **Wiring** toolbar to place a bar style power port, pre-assigned to the VCC net
- click the button on the **Wiring** toolbar to place a bar style bus power port, pre-assigned to the VCC net. This button is only available when the source schematic is part of an FPGA project
- click the button on the **Wiring** toolbar to place a power ground style power port, pre-assigned to the GND net
- click the button on the **Wiring** toolbar to place a power ground style bus power port, pre-assigned to the GND net. This button is only available when the source schematic is part of an FPGA project
- click the button on the **Utilities** toolbar to access a drop-down providing an array of power port commands, including the various styles and several net-pre-assigned power ports.

### Power port styles

The following seven graphical styles of power port are available and can be set by editing the object's properties:

	Circle
	Arrow
	Bar
	Wave
	Power Ground
	Signal Ground
	Earth

### Placement

After launching the command, the cursor will change to a cross-hair and you will enter power port placement mode. A power port symbol will appear "floating" on the cursor. Position the object and click or press **Enter** to effect placement.

Continue placing further power ports, or right-click or press **Esc** to exit placement mode.

Press the **Spacebar** while in placement mode to rotate the power port. Rotation is anti-clockwise and in steps of 90°.

Press the **X** or **Y** keys while in placement mode to flip the power port along the X-axis or Y-axis respectively.

## Editing

The properties of a power port object can be modified before, during and after placement. Editing itself falls into two categories - graphical and non-graphical.

The following three methods of non-graphical editing are available:

### Editing via an associated properties dialog

This method of editing uses the following dialog to modify the properties of a power port object.

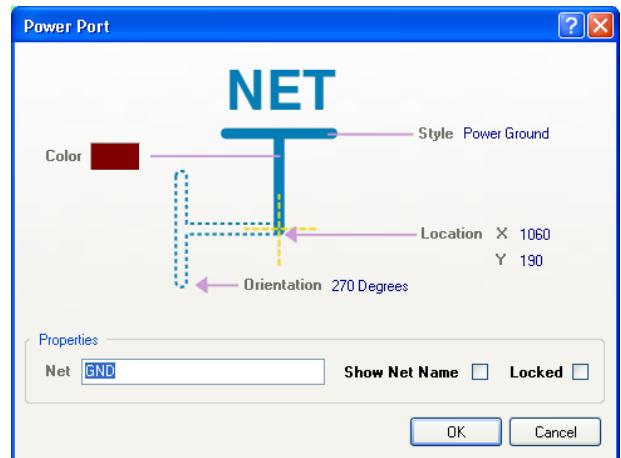
Use the dialog's 'What's This Help' feature to obtain detailed information about each of the options available. Click on the question mark button at the top right of the dialog and then click over a field or option to pop-up information specific to that field or option.

The *Power Port* dialog can be accessed prior to entering placement mode, from the **Schematic - Default Primitives** page of the *Preferences* dialog (**Tools » Schematic Preferences**). This allows you to change the default properties for the power port object, which will be applied when placing subsequent power ports.

During placement, the *Power Port* dialog can be accessed by pressing the **Tab** key.

After placement, the *Power Port* dialog can be accessed in one of the following ways:

- double-clicking on the placed power port object
- selecting the power port object and choosing **Properties** from the right-click pop-up menu
- choosing the **Change** command from the **Edit** menu and then clicking once over the placed power port object.



### Editing via the SCH Inspector panel

The **SCH Inspector** panel enables you to interrogate and edit the properties of one or more design objects in the active document. Used in conjunction with appropriate filtering, the panel can be used to make changes to multiple objects of the same kind, from one convenient location.

For more information on the **SCH Inspector** panel, press **F1** when the cursor is over the panel.

### Editing via the SCH List panel

The **SCH List** panel allows you to display design objects from one or more documents in tabular format, enabling you to quickly inspect and modify object attributes. When used in conjunction with the **SCH Filter** panel, it enables you to display just those objects falling under the scope of the active filter - allowing you to target and edit multiple design objects with greater accuracy and efficiency.

For more information on the **SCH List** and **SCH Inspector** panels, press **F1** when the cursor is over a panel.

## Graphical editing

This method of editing allows you to select a placed power port object directly in the workspace and change its location graphically. Power ports are fixed with respect to their size and shape. As such, editing handles are not available when the power port object is selected:



Click anywhere inside the dashed box and drag to reposition the power port as required. The power port object can be rotated or flipped while dragging.

If the **Enable In-Place Editing** option is enabled on the **Schematic - General** page of the *Preferences* dialog (**Tools » Schematic Preferences**), you will be able to edit the assigned net for the power port directly in the workspace. Select the power port and then click once to invoke the feature. Type the new net name as required and then click away from the power port or press **Enter** to effect the change.

If you attempt to graphically modify a power port object that has its **Locked** property enabled, a dialog will appear asking for confirmation to proceed with the edit.

If the **Protect Locked Objects** option is enabled in the **Schematic – Graphical Editing** page of the *Preferences* dialog (**Tools** » **Schematic Preferences**), and the **Locked** option for this design object is enabled as well then this object cannot be selected or graphically edited.

You will have to double click on this locked object directly and disable the **Locked** property or disable the **Protect Locked Objects** option in the **Schematic – Graphical Editing** Preferences dialog to graphically edit this object.

If you attempt to select locked objects along with other objects, only those objects that are unlocked can be selected and moved as a group if the **Protect Locked Objects** option is enabled. Otherwise if the **Protect Locked Objects** option is disabled you will be prompted for confirmation to proceed with the edit of the objects including locked objects.

### Notes

The graphical symbol selected for a power port does not determine which net it is assigned to. You must explicitly set the net name in the *Power Port* dialog.

---

Any changes made to object properties during placement will cause the default properties for the object to be updated, unless the **Permanent** option - on the **Schematic - Default Primitives** page of the *Preferences* dialog - is enabled. When this option is enabled, changes made will affect only the object being placed and subsequent objects placed during the same placement session.

## Probe



### Description

A probe is a design directive. It is a special marker which is placed on a schematic sheet to identify nodes for digital simulation during netlist generation. It can also be used to interrogate the status, in real time, of a net connecting to a pin on a programmed FPGA device.

### Availability

Probes are available for placement in the Schematic Editor only, by choosing **Place » Directives » Probe [P, V, R]** from the main menus.

### Placement

After launching the command, the cursor will change to a cross-hair and you will enter probe placement mode. Position the cursor over a wire or other net object and click or press **Enter** to effect placement.

Continue placing further probe directives or right-click or press **Esc** to exit placement mode.

Press the **Spacebar** while in placement mode to rotate the probe directive. Rotation is anti-clockwise and in steps of 90°.

Press the **X** or **Y** keys while in placement mode to flip the probe directive along the X-axis or Y-axis respectively.

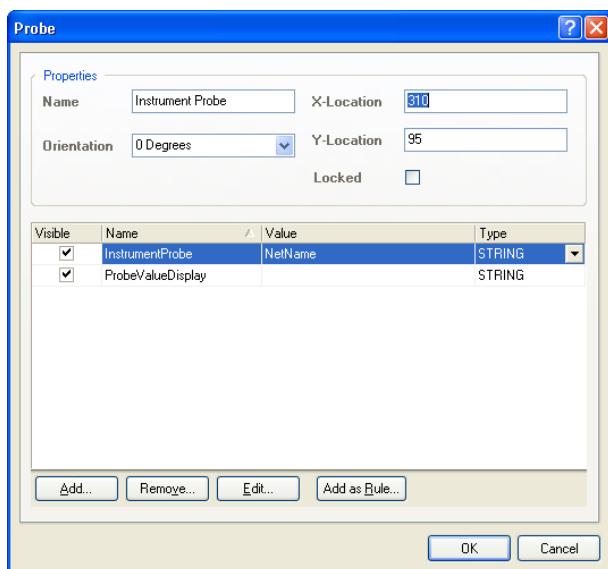
### Editing

The properties of a probe directive can be modified before, during and after placement. Editing itself falls into two categories - graphical and non-graphical.

The following three methods of non-graphical editing are available:

#### Editing via an associated properties dialog

This method of editing uses the following dialog to modify the properties of a probe directive.



Use the dialog's 'What's This Help' feature to obtain detailed information about each of the options available. Click on the question mark button at the top right of the dialog and then click over a field or option to pop-up information specific to that field or option.

The **Probe** dialog can be accessed prior to entering placement mode, from the **Schematic - Default Primitives** page of the **Preferences dialog (Tools » Schematic Preferences)**. This allows you to change the default properties for the probe directive, which will be applied when placing subsequent probe directives.

During placement, the **Probe** dialog can be accessed by pressing the **Tab** key.

After placement, the **Probe** dialog can be accessed in one of the following ways:

- double-clicking on the placed probe directive
- selecting the probe directive and choosing **Properties** from the right-click pop-up menu
- choosing the **Change** command from the **Edit** menu and then clicking once over the placed probe directive.

### Editing via the SCH Inspector panel

The **SCH Inspector** panel enables you to interrogate and edit the properties of one or more design objects in the active document. Used in conjunction with appropriate filtering, the panel can be used to make changes to multiple objects of the same kind, from one convenient location.

 For more information on the **SCH Inspector** panel, press **F1** when the cursor is over the panel.

### Editing via the SCH List panel

The **SCH List** panel allows you to display design objects from one or more documents in tabular format, enabling you to quickly inspect and modify object attributes. When used in conjunction with the **SCH Filter** panel, it enables you to display just those objects falling under the scope of the active filter - allowing you to target and edit multiple design objects with greater accuracy and efficiency.

 For more information on the **SCH List** and **SCH Inspector** panels, press **F1** when the cursor is over a panel.

### Graphical editing

This method of editing allows you to select a placed probe directive directly in the workspace and change its location graphically. Probe directives are fixed with respect to their size and shape. As such, editing handles are not available when the probe directive is selected:



Click anywhere inside the dashed box and drag to reposition the probe directive as required. The probe directive can be rotated or flipped while dragging.

If you attempt to graphically modify a probe object that has its **Locked** property enabled, a dialog will appear asking for confirmation to proceed with the edit.

If the **Protect Locked Objects** option is enabled in the **Schematic – Graphical Editing** page of the *Preferences* dialog (**Tools** » **Schematic Preferences**), and the **Locked** option for this design object is enabled as well then this object cannot be selected or graphically edited.

You will have to double click on this locked object directly and disable the **Locked** property or disable the **Protect Locked Objects** option in the **Schematic – Graphical Editing** Preferences dialog to graphically edit this object.

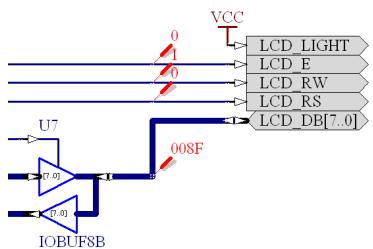
If you attempt to select locked objects along with other objects, only those objects that are unlocked can be selected and moved as a group if the **Protect Locked Objects** option is enabled. Otherwise if the **Protect Locked Objects** option is disabled you will be prompted for confirmation to proceed with the edit of the objects including locked objects.

### Notes

For an FPGA design, the status of FPGA device pins can be monitored directly from the schematic sheet by placing a probe directive on any net that connects to an FPGA pin, as illustrated in the image below.

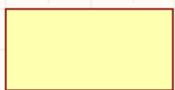
**Note:** For this feature to function, the source FPGA design must be downloaded to the physical FPGA device and the associated pin states panel for that device must be open and remain open.

The pin states panel for a physical device is accessed by clicking the **Show Pins Panel** button on the associated instrument panel for that device. The latter is loaded into the **Instrument Rack – Hard Devices** panel upon double-clicking the entry for the device, in the Hard Devices chain of the **Devices** view.



Any changes made to object properties during placement will cause the default properties for the object to be updated, unless the **Permanent** option - on the **Schematic - Default Primitives** page of the *Preferences* dialog - is enabled. When this option is enabled, changes made will affect only the object being placed and subsequent objects placed during the same placement session.

## Rectangle



### Description

A rectangle is a non-electrical drawing primitive. It is a graphic element that can be placed on a schematic sheet and can be filled or unfilled.

### Availability

Rectangles are available for placement in both Schematic and Schematic Library Editors:

#### Schematic Editor

- Choose **Place » Drawing Tools » Rectangle [P, D, R]** from the main menus
- Click the button on the **Utility Tools** drop-down of the **Utilities** toolbar

#### Schematic Library Editor

- Choose **Place » Rectangle [P, R]** from the main menus
- Click the button on the **Utility Tools** drop-down of the **Utilities** toolbar

### Placement

After launching the command, the cursor will change to a cross-hair and you will enter rectangle placement mode. Placement is made by performing the following sequence of actions:

- click or press **Enter** to anchor the first corner of the rectangle
- move the cursor to adjust the size of the rectangle, then click or press **Enter** to anchor the diagonally-opposite corner and thereby complete placement of the rectangle.

Continue placing further rectangles, or right-click or press **Esc** to exit placement mode.

The rectangle object can be rotated or flipped while in placement mode and before the first corner of the rectangle is anchored:

- Press the **Spacebar** to rotate the rectangle. Rotation is anti-clockwise and in steps of 90°.
- Press the **X** or **Y** keys to flip the rectangle along the X-axis or Y-axis respectively.

### Editing

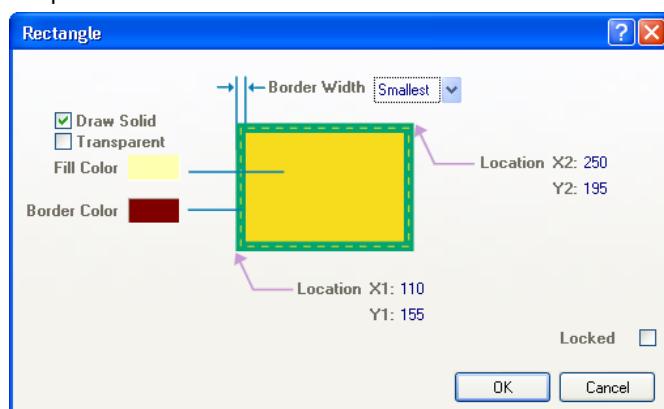
The properties of a rectangle object can be modified before, during and after placement. Editing itself falls into two categories - graphical and non-graphical.

The following three methods of non-graphical editing are available:

#### Editing via an associated properties dialog

This method of editing uses the following dialog to modify the properties of a rectangle object.

Use the dialog's 'What's This Help' feature to obtain detailed information about each of the options available. Click on the question mark button at the top right of the dialog and then click over a field or option to pop-up information specific to that field or option.



The **Rectangle** dialog can be accessed prior to entering placement mode, from the **Schematic - Default Primitives** page of the **Preferences** dialog (**Tools » Schematic Preferences**). This allows you to change the default properties for the rectangle object, which will be applied when placing subsequent rectangles.

During placement, the **Rectangle** dialog can be accessed by pressing the **Tab** key.

After placement, the **Rectangle** dialog can be accessed in one of the following ways:

- double-clicking on the placed rectangle object
- selecting the rectangle object and choosing **Properties** from the right-click pop-up menu (Schematic Editor only)
- choosing the **Change** command from the **Edit** menu and then clicking once over the placed rectangle object.

#### Editing via the SCH Inspector panel

The **SCH Inspector** panel enables you to interrogate and edit the properties of one or more design objects in the active document. Used in conjunction with appropriate filtering, the panel can be used to make changes to multiple objects of the same kind, from one convenient location.

 For more information on the **SCH Inspector** panel, press **F1** when the cursor is over the panel.

#### Editing via the SCH List panel

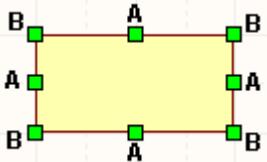
The **SCH List** panel allows you to display design objects from one or more documents in tabular format, enabling you to quickly inspect and modify object attributes. When used in conjunction with the **SCH Filter** panel, it enables you to display just those objects falling under the scope of the active filter - allowing you to target and edit multiple design objects with greater accuracy and efficiency.

 For more information on the **SCH List** and **SCH Inspector** panels, press **F1** when the cursor is over a panel.

#### Graphical editing

This method of editing allows you to select a placed rectangle object directly in the workspace and change its size, shape or location, graphically.

When a rectangle object is selected, the following editing handles are available:



Click and drag **A** to resize the rectangle in the vertical and horizontal directions separately.

Click and drag **B** to resize the rectangle in the vertical and horizontal directions simultaneously.

Click anywhere on the rectangle - away from editing handles - and drag to reposition it. The rectangle can be rotated or flipped while dragging.

If you attempt to graphically modify a rectangle object that has its **Locked** property enabled, a dialog will appear asking for confirmation to proceed with the edit.

If the **Protect Locked Objects option** is enabled in the **Schematic – Graphical Editing** page of the **Preferences** dialog (**Tools » Schematic Preferences**), and the **Locked** option for this design object is enabled as well then this object cannot be selected or graphically edited.

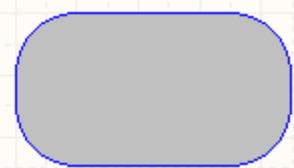
You will have to double click on this locked object directly and disable the **Locked** property or disable the **Protect Locked Objects** option in the **Schematic – Graphical Editing** Preferences dialog to graphically edit this object.

If you attempt to select locked objects along with other objects, only those objects that are unlocked can be selected and moved as a group if the **Protect Locked Objects** option is enabled. Otherwise if the **Protect Locked Objects** option is disabled you will be prompted for confirmation to proceed with the edit of the objects including locked objects.

#### Notes

Any changes made to object properties during placement will cause the default properties for the object to be updated, unless the **Permanent** option - on the **Schematic - Default Primitives** page of the **Preferences** dialog - is enabled. When this option is enabled, changes made will affect only the object being placed and subsequent objects placed during the same placement session.

## Round Rectangle



### Description

A round rectangle is a non-electrical drawing primitive. Essentially a rectangle object with rounded corners, it is a graphic element that can be placed on a schematic sheet - filled or unfilled.

### Availability

Round rectangles are available for placement in both Schematic and Schematic Library Editors:

#### Schematic Editor

- Choose **Place » Drawing Tools » Round Rectangle [P, D, O]** from the main menus
- Click the button on the **Utility Tools** drop-down of the **Utilities** toolbar

#### Schematic Library Editor

- Choose **Place » Round Rectangle [P, O]** from the main menus
- Click the button on the **Utility Tools** drop-down of the **Utilities** toolbar

### Placement

After launching the command, the cursor will change to a cross-hair and you will enter round rectangle placement mode. Placement is made by performing the following sequence of actions:

- click or press **Enter** to anchor the first corner of the round rectangle
- move the cursor to adjust the size of the round rectangle, then click or press **Enter** to anchor the diagonally-opposite corner and thereby complete placement of the rectangle.

Continue placing further round rectangles, or right-click or press **Esc** to exit placement mode.

The round rectangle object can be rotated or flipped while in placement mode and before the first corner of the rectangle is anchored:

- Press the **Spacebar** to rotate the round rectangle. Rotation is anti-clockwise and in steps of 90°.
- Press the **X** or **Y** keys to flip the round rectangle along the X-axis or Y-axis respectively.

### Editing

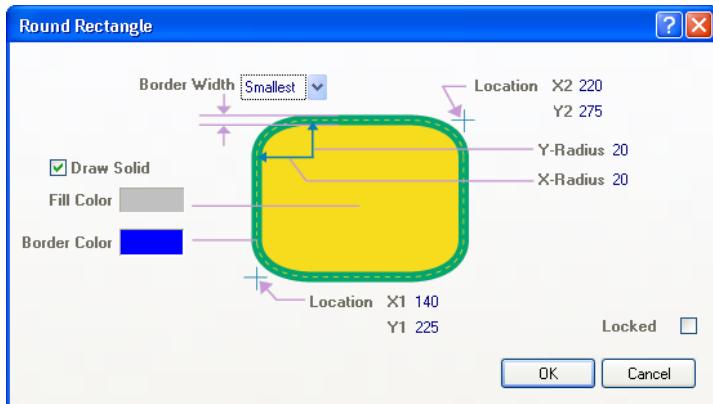
The properties of a round rectangle object can be modified before, during and after placement. Editing itself falls into two categories - graphical and non-graphical.

The following three methods of non-graphical editing are available:

#### Editing via an associated properties dialog

This method of editing uses the following dialog to modify the properties of a round rectangle object.

Use the dialog's 'What's This Help' feature to obtain detailed information about each of the options available. Click on the question mark button at the top right of the dialog and then click over a field or option to pop-up information specific to that field or option.



The **Round Rectangle** dialog can be accessed prior to entering placement mode, from the **Schematic - Default Primitives** page of the Preferences dialog (**Tools » Schematic Preferences**). This allows you to change the default properties for the round rectangle object, which will be applied when placing subsequent round rectangles.

During placement, the **Round Rectangle** dialog can be accessed by pressing the **Tab** key.

After placement, the **Round Rectangle** dialog can be accessed in one of the following ways:

- double-clicking on the placed round rectangle object
- selecting the round rectangle object and choosing **Properties** from the right-click pop-up menu (Schematic Editor only)
- choosing the **Change** command from the **Edit** menu and then clicking once over the placed round rectangle object.

#### Editing via the SCH Inspector panel

The **SCH Inspector** panel enables you to interrogate and edit the properties of one or more design objects in the active document. Used in conjunction with appropriate filtering, the panel can be used to make changes to multiple objects of the same kind, from one convenient location.

For more information on the **SCH Inspector** panel, press **F1** when the cursor is over the panel.

#### Editing via the SCH List panel

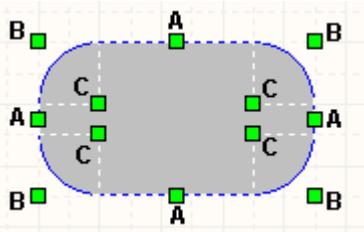
The **SCH List** panel allows you to display design objects from one or more documents in tabular format, enabling you to quickly inspect and modify object attributes. When used in conjunction with the **SCH Filter** panel, it enables you to display just those objects falling under the scope of the active filter - allowing you to target and edit multiple design objects with greater accuracy and efficiency.

For more information on the **SCH List** and **SCH Inspector** panels, press **F1** when the cursor is over a panel.

#### Graphical editing

This method of editing allows you to select a placed round rectangle object directly in the workspace and change its size, shape or location, graphically.

When a rounded rectangle object is selected, the following editing handles are available:



Click and drag **A** to resize the round rectangle in the vertical and horizontal directions separately.

Click and drag **B** to resize the round rectangle in the vertical and horizontal directions simultaneously.

Click and drag **C** to change the curvature of the corners. This affects all corners equally, regardless of which editing handle is chosen.

Click anywhere on the round rectangle - away from editing handles - and drag to reposition it. The round rectangle can be rotated or flipped while dragging.

If you attempt to graphically modify a rounded rectangle object that has its **Locked** property enabled, a dialog will appear asking for confirmation to proceed with the edit.

If the **Protect Locked Objects option** is enabled in the **Schematic – Graphical Editing** page of the *Preferences* dialog (**Tools** » **Schematic Preferences**), and the **Locked** option for this design object is enabled as well then this object cannot be selected or graphically edited.

You will have to double click on this locked object directly and disable the **Locked** property or disable the **Protect Locked Objects** option in the **Schematic – Graphical Editing** Preferences dialog to graphically edit this object.

If you attempt to select locked objects along with other objects, only those objects that are unlocked can be selected and moved as a group if the **Protect Locked Objects** option is enabled. Otherwise if the **Protect Locked Objects** option is disabled you will be prompted for confirmation to proceed with the edit of the objects including locked objects.

### Notes

Any changes made to object properties during placement will cause the default properties for the object to be updated, unless the **Permanent** option - on the **Schematic - Default Primitives** page of the *Preferences* dialog - is enabled. When this option is enabled, changes made will affect only the object being placed and subsequent objects placed during the same placement session.

## Sheet Entry



### Description

A sheet entry is an electrical design primitive that belongs within a sheet symbol. It is placed within a sheet symbol to designate input/output ports for the symbol. The sheet entries correspond to ports placed in the source schematic sub-sheet that the symbol represents.

### Availability

Sheet entries are available for placement in the Schematic Editor only. Use one of the following methods to access the placement command:

- choose **Place » Add Sheet Entry [P, A]** from the main menus
- click the button on the **Wiring** toolbar.

### Placement

After launching the command, the cursor will change to a cross-hair and you will enter sheet entry placement mode. Placement is made by performing the following sequence of actions:

- the sheet symbol is implicitly chosen by the position of the new sheet entry by the mouse on the schematic sheet (this sheet needs to have at least one sheet symbol).
- move the cursor to adjust the position of the sheet entry in relation to any edge of the sheet symbol, then click or press **Enter** to anchor the sheet entry and complete placement.

Continue placing further sheet entries, or right-click or press **Esc** to exit placement mode.

### Editing

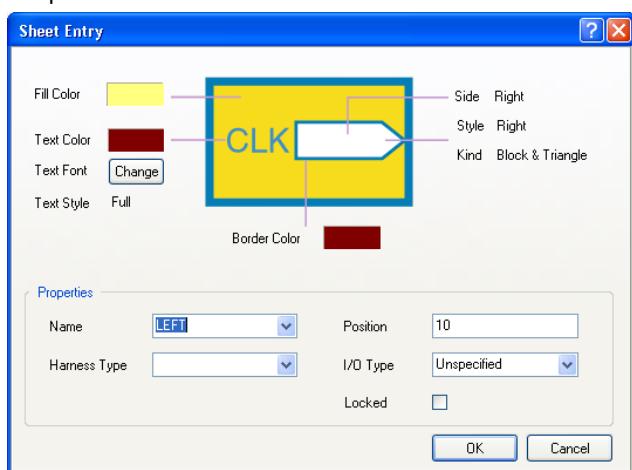
The properties of a sheet entry object can be modified before, during and after placement. Editing itself falls into two categories - graphical and non-graphical.

The following three methods of non-graphical editing are available:

#### Editing via an associated properties dialog

This method of editing uses the following dialog to modify the properties of a sheet entry object.

Use the dialog's 'What's This Help' feature to obtain detailed information about each of the options available. Click on the question mark button at the top right of the dialog and then click over a field or option to pop-up information specific to that field or option.



The **Sheet Entry** dialog can be accessed prior to entering placement mode, from the **Schematic - Default Primitives** page of the **Preferences** dialog (**Tools** » **Schematic Preferences**). This allows you to change the default properties for the sheet entry object, which will be applied when placing subsequent sheet entries.

During placement, the **Sheet Entry** dialog can be accessed by pressing the **Tab** key.

After placement, the **Sheet Entry** dialog can be accessed in one of the following ways:

- double-clicking on the placed sheet entry object
- selecting the sheet entry object and choosing **Properties** from the right-click pop-up menu
- choosing the **Change** command from the **Edit** menu and then clicking once over the placed sheet entry object.

#### **Editing via the SCH Inspector panel**

The **SCH Inspector** panel enables you to interrogate and edit the properties of one or more design objects in the active document. Used in conjunction with appropriate filtering, the panel can be used to make changes to multiple objects of the same kind, from one convenient location.

 For more information on the **SCH Inspector** panel, press **F1** when the cursor is over the panel.

#### **Editing via the SCH List panel**

The **SCH List** panel allows you to display design objects from one or more documents in tabular format, enabling you to quickly inspect and modify object attributes. When used in conjunction with the **SCH Filter** panel, it enables you to display just those objects falling under the scope of the active filter - allowing you to target and edit multiple design objects with greater accuracy and efficiency.

 For more information on the **SCH List** and **SCH Inspector** panels, press **F1** when the cursor is over a panel.

#### **Graphical editing**

This method of editing allows you to select a placed Sheet Entry object directly in the workspace and change its location graphically. The Sheet Symbol itself gets resized automatically when you attempt to move the existing Sheet Entry beyond the current extends of the symbol. To move an existing Sheet Entry (or a group of sheet entries) to a different Sheet Symbol, press the **CTRL** key while dragging the Sheet Entry.

Sheet Entry colors help you identify whether you are making correct placement or not. If you move the Sheet Entry outside of a Sheet Symbol using the **CTRL** key, the Sheet Entry changes to a gray color to indicate incorrect Sheet Entry placement and you will not be able to place the Sheet Entry. A blue Sheet Entry indicates correct placement and you will be able to place the Sheet Entry.

Sheet entries can only be adjusted with respect to their shape by changing their I/O Type (accessed through the **Sheet Entry** dialog). As such, editing handles are not available when the Sheet Entry object is selected:



Click anywhere inside the dashed box and drag to reposition the sheet entry within the sheet symbol as required.

If the **Enable In-Place Editing** option is enabled on the **Schematic - General** page of the **Preferences** dialog (**Tools** » **Schematic Preferences**), you will be able to edit the name for the sheet entry directly in the workspace. Select the sheet entry object and then click once to invoke the feature. Type the new name as required and then click away from the sheet entry object or press **Enter** to effect the change.

#### **Sheet Entry Actions**

Right-clicking over a placed sheet entry will pop-up a context-sensitive menu, from which a variety of commands are available that act on that sheet entry (or on all selected sheet entries where applicable). The following sections detail each of these commands.

**Note:** Many of the following commands are also available from the Schematic Editor's main menus. Commands on the main menus apply to the selected sheet entry(ies) or allow you to choose the sheet entry on which the command will act, rather than just the sheet entry under the cursor. Where such commands exist, reference to their access is made.

## Jumping to the Corresponding Port

This command is used to jump to the chosen sheet entry's corresponding port on the sub-sheet referenced by that entry's parent sheet symbol. The command is accessed by right-clicking over the required sheet entry and choosing **Sheet Entry Actions** » **Jump to Port PortName** from the menu that appears.

## Toggling Sheet Entry I/O Type

This command is used to toggle the I/O Type for the currently selected sheet entries, irrespective of sheet symbol to which they are associated, on the current schematic sheet.

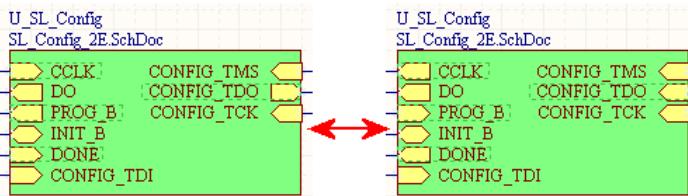
The command can be accessed either by:

- choosing **Edit** » **Move** » **Toggle Selected Sheet Entries IO Type** from the main menus
- right-clicking over the required sheet entry (or one sheet entry in a selection of sheet entries) and choosing **Sheet Entry Actions** » **Toggle Selected Sheet Entries IO Type** from the menu that appears.

If you wish to toggle the I/O Type for the sheet entry under the cursor, use the second method of access, without prior selection.

If you wish to toggle the I/O Type for multiple sheet entries, ensure they are first selected (in their respective sheet symbols) before launching the command.

After launching the command, the I/O Type defined for each sheet entry in the selection will be toggled, where applicable.



The actual change depends on the current I/O Type as follows:

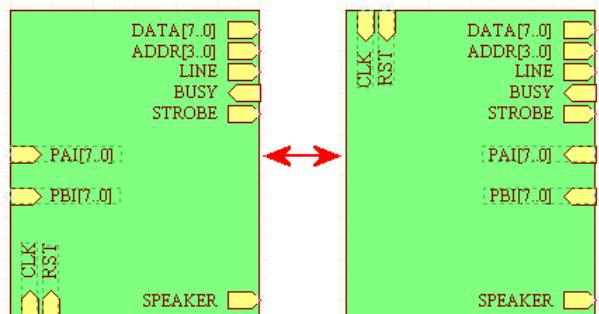
- Unspecified remains Unspecified
- Output changes to Input
- Input changes to Output
- Bidirectional remains Bidirectional.

## Swapping Sheet Entry Side

This command is used to relocate a selected sheet entry to the side of its parent sheet symbol that is directly opposite to its current position. The sheet entry's I/O Type is not changed by the swap. The command acts on all selected sheet entries, irrespective of sheet symbol to which they are associated, on the current schematic sheet.

The command can be accessed either by:

- choosing **Edit** » **Move** » **Swap Selected Sheet Entries Side** from the main menus
- right-clicking over the required sheet entry (or one sheet entry in a selection of sheet entries) and choosing **Sheet Entry Actions** » **Swap Selected Sheet Entries Side** from the menu that appears.



If you wish to swap sides for the sheet entry under the cursor, use the second method of access, without prior selection. If you wish to swap sides for multiple sheet entries, ensure they are first selected (in their respective sheet symbols) before launching the command.

After launching the command, each sheet entry in the selection will be swapped to the opposite side of its sheet symbol.

## Reversing Sheet Entry Order

This command allows you to reverse the order that selected sheet entries appear along a side of a parent sheet symbol. The I/O Type of a sheet entry is not changed by the reordering.

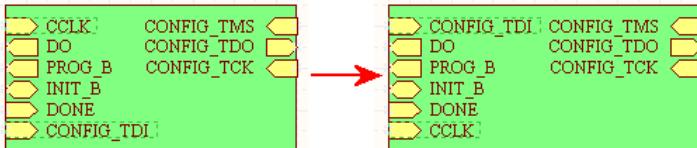
The command can be accessed by choosing **Edit** » **Move** » **Reverse Selected Sheet Entries Order** from the Schematic Editor's main menus.

Ensure that all sheet entries that you wish to reorder are selected prior to launching the command. Two or more sheet entries must be selected for a particular side of a sheet symbol in order for the command to have effect. You can simultaneously

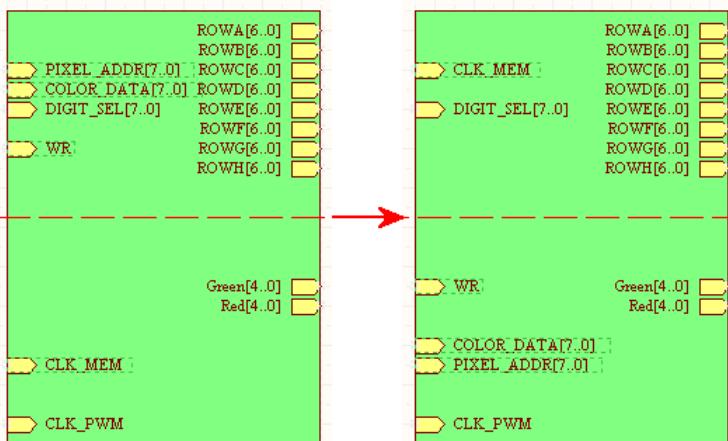
reorder sheet entries along different sides of the same parent sheet symbol and across different sheet symbols on the active schematic sheet.

After launching the command, the reordering will take place. The reordering is achieved by mirroring the positions of the selected sheet entries - along a particular sheet symbol side - about an imaginary line at the mid point of the distance between the extents of the two most-outer selected sheet entries.

The following image shows reordering for two selected sheet entries - whereby they exchange positions.



The next image shows reordering for four selected sheet entries. The position of the center line used for the reordering is indicated to better show the mirroring effect on their pre- and post-reordering positions.



## Notes

Different arrow kinds or shapes can be defined for the sheet entries. The following arrow kinds are:

**Arrow:** Display as an arrow. The direction of the arrow is dependent on the I/O type setting.

**Triangle:** Display as a triangle. The direction of the triangle is dependent on the I/O type setting.

**Block & Triangle:** Display as a standard entry symbol. The direction of the entry is dependent on the I/O type setting.

---

When a Sheet Entry is connected to a Signal Harness, the Sheet Entry becomes a Harness object. By default, the Sheet Entry will change color to match the color of the Signal Harness. Disable the option, **Sheet Entries and Ports use Harness Color** in **Schematic – Graphical Editing** page of the *Preferences* dialog to specify your own color for Sheet Entries or to use the default color.

---

When a Sheet Entry is connected to a Harness Connector by a Signal Harness, the **Harness Type** in the *Sheet Entry* dialog is automatically populated with the **Harness Type** of the Harness Connector. When a Sheet Entry is connected to a Port by a Signal Harness and the Port has a Harness Type declared, the Sheet Entry will become a Harness object and change to the color of the Signal Harness. If you move the Sheet Entry away from the Harness Connector and the **Harness Type** field is not populated, the Sheet Entry will revert back to the default color.

---

Should you need to negate (include a bar over the top of) a sheet entry name, this can be done in one of two ways:

- Include a backslash character after each character in the name (e.g. E\N\A\B\L\E\)
- Enable the **Single '\ Negation** option on the **Schematic - Graphical Editing** page of the *Preferences* dialog, then include one backslash character at the start of the name (e.g. \ENABLE).

When instantiating multiple channels from the same sheet symbol, certain signals are repeated and sent individually to each instantiated channel. With respect to a sheet entry, a signal is repeated by using the **Repeat** keyword in the sheet entry's name (e.g. `Repeat(Headphone)`) as illustrated in the example image below:



The sheet entry is wired to a bus, which in turn carries the individual signals to their correspondingly instantiated destinations.

Any changes made to object properties during placement will cause the default properties for the object to be updated, unless the **Permanent** option - on the **Schematic - Default Primitives** page of the *Preferences* dialog - is enabled. When this option is enabled, changes made will affect only the object being placed and subsequent objects placed during the same placement session.

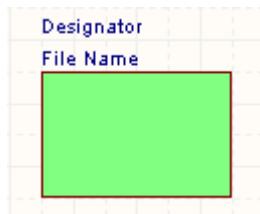
If you attempt to graphically modify a sheet entry object that has its **Locked** property enabled, a dialog will appear asking for confirmation to proceed with the edit.

If the **Protect Locked Objects** option is enabled in the **Schematic – Graphical Editing** page of the *Preferences* dialog (**Tools** » **Schematic Preferences**), and the **Locked** option for this design object is enabled as well then this object cannot be selected or graphically edited.

You will have to double click on this locked object directly and disable the **Locked** property or disable the **Protect Locked Objects** option in the **Schematic – Graphical Editing** Preferences dialog to graphically edit this object.

If you attempt to select locked objects along with other objects, only those objects that are unlocked can be selected and moved as a group if the **Protect Locked Objects** option is enabled. Otherwise if the **Protect Locked Objects** option is disabled you will be prompted for confirmation to proceed with the edit of the objects including locked objects.

## Sheet Symbol



### Description

A sheet symbol is an electrical design primitive. It is used to represent a sub-sheet in a multi-sheet hierarchical design. Sheet symbols include sheet entry symbols, which provide a connection point for signals between the parent and child sheets, similar to the way that Ports provide connections between sheets in a flat-sheet design.

### Availability

Sheet symbols are available for placement in the Schematic Editor only. Use one of the following methods to access the placement command:

- choose **Place » Sheet Symbol [P, S]** from the main menus
- click the  button on the **Wiring** toolbar.

### Placement

After launching the command, the cursor will change to a cross-hair and you will enter sheet symbol placement mode.

Placement is made by performing the following sequence of actions:

- click or press **Enter** to anchor the top-left corner of the sheet symbol
- move the cursor to adjust the size of the sheet symbol, then click or press **Enter** to anchor the diagonally-opposite corner and thereby complete placement of the sheet symbol.

Continue to place other sheet symbols, or right-click or press **Esc** to exit placement mode.

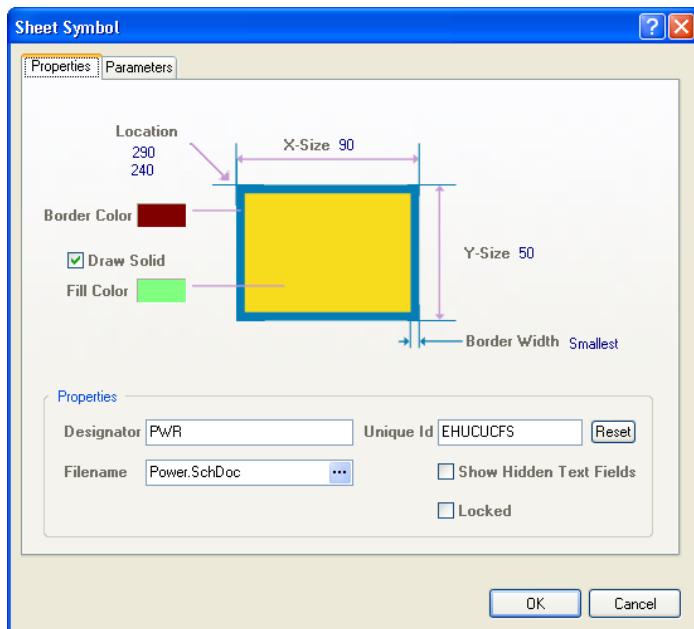
### Editing

The properties of a sheet symbol object can be modified before, during and after placement. Editing itself falls into two categories - graphical and non-graphical.

The following three methods of non-graphical editing are available:

#### Editing via an associated properties dialog

This method of editing uses the following dialog to modify the properties of a sheet symbol object.



Use the dialog's 'What's This Help' feature to obtain detailed information about each of the options available. Click on the question mark button at the top right of the dialog and then click over a field or option to pop-up information specific to that field or option.

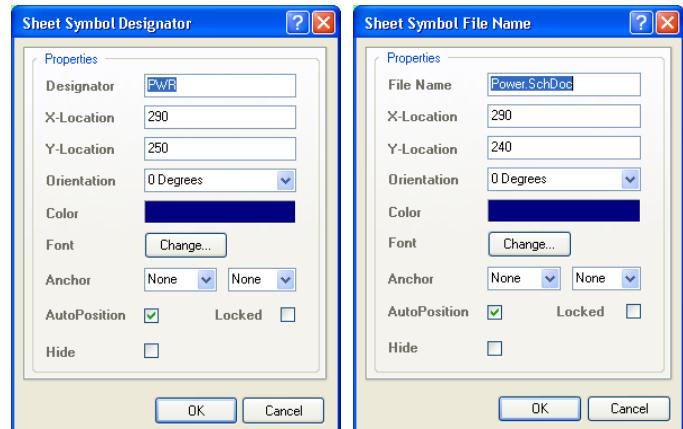
The **Sheet Symbol** dialog can be accessed prior to entering placement mode, from the **Schematic - Default Primitives** page of the **Preferences** dialog (**Tools** » **Schematic Preferences**). This allows you to change the default properties for the sheet symbol object, which will be applied when placing subsequent sheet symbols.

During placement, the **Sheet Symbol** dialog can be accessed by pressing the **Tab** key.

After placement, the **Sheet Symbol** dialog can be accessed in one of the following ways:

- double-clicking on the placed sheet symbol object
- selecting the sheet symbol object and choosing **Properties** from the right-click pop-up menu
- choosing the **Change** command from the **Edit** menu and then clicking once over the placed sheet symbol object.

The sheet symbol **Designator** and **Filename** text fields can be formatted independently of the sheet symbol itself. The corresponding properties dialogs for each - the **Sheet Symbol Designator** and **Sheet Symbol File Name** dialogs respectively - can be accessed using the three methods described above (replacing sheet symbol with the relevant object whose properties you wish to view/modify).



### Editing via the SCH Inspector panel

The **SCH Inspector** panel enables you to interrogate and edit the properties of one or more design objects in the active document. Used in conjunction with appropriate filtering, the panel can be used to make changes to multiple objects of the same kind, from one convenient location.

For more information on the **SCH Inspector** panel, press **F1** when the cursor is over the panel.

### Editing via the SCH List panel

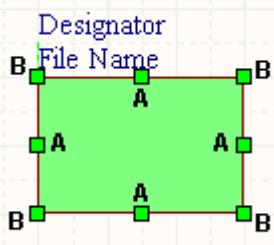
The **SCH List** panel allows you to display design objects from one or more documents in tabular format, enabling you to quickly inspect and modify object attributes. When used in conjunction with the **SCH Filter** panel, it enables you to display just those objects falling under the scope of the active filter - allowing you to target and edit multiple design objects with greater accuracy and efficiency.

For more information on the **SCH List** and **SCH Inspector** panels, press **F1** when the cursor is over a panel.

## Graphical editing

This method of editing allows you to select a placed sheet symbol object directly in the workspace and change its size, shape or location, graphically.

When a sheet symbol object is selected, the following editing handles are available:



Click and drag **A** to resize the sheet symbol in the vertical and horizontal directions separately.

Click and drag **B** to resize the sheet symbol in the vertical and horizontal directions simultaneously.

Resizing the sheet symbol will not affect the absolute positions of any defined sheet entries within.

Click anywhere on the sheet symbol - away from editing handles - and drag to reposition it.

The sheet symbol's Designator and File Name text fields can only be adjusted with respect to their size by changing the size of the font used (accessed through the *Sheet Symbol Designator* and *Sheet Symbol File Name* dialogs, respectively). As such, editing handles are not available when either of these objects are selected:



Click anywhere inside the dashed box and drag to reposition the text object as required. The object can be rotated or flipped while dragging:

- Press the **Spacebar** to rotate the text. Rotation is anti-clockwise and in steps of 90°.
- Press the **X** or **Y** keys to flip the text along the X-axis or Y-axis respectively.

If the **Enable In-Place Editing** option is enabled on the **Schematic - General** page of the *Preferences* dialog (**Tools » Schematic Preferences**), you will be able to edit the name for the Designator or File Name directly in the workspace. Select the text object and then click once to invoke the feature. Type the new name as required and then click away from the text object or press **Enter** to effect the change.

## Sheet Symbol Actions

Right-clicking over a placed sheet symbol will pop-up a context-sensitive menu, from which a variety of commands are available that act on that sheet symbol (or on all selected sheet symbols where applicable). The following sections detail each of these commands.

**Note:** Many of the following commands are also available from the Schematic Editor's main menus. Commands on the main menus apply to the selected sheet symbol(s) or allow you to choose the sheet symbol on which the command will act, rather than just the sheet symbol under the cursor. Where such commands exist, reference to their access is made.

### Opening a Selected Sheet Symbol's Sub-Sheet

This command applies only to the sheet symbol under the cursor and is accessed by right-clicking and choosing **Sheet Symbol Actions » Open SubSheet "SheetName.SchDoc"** from the menu that appears.

The child sheet for the symbol will be opened (if not already) and made the active document in the main design window.

### Creating a Schematic Sheet directly from a Sheet Symbol

This command is used to create a new schematic document from a sheet symbol and add ports to that document corresponding to each of the sheet entries on the symbol. In this way, you can automatically create the sub-sheets for a multi-sheet schematic design, based on the sheet symbols you have created and placed on the top sheet.

The command can be accessed either by:

- choosing **Design » Create Sheet From Symbol** from the main menus. You will be prompted to choose a sheet symbol
- right-clicking over the required sheet symbol and choosing **Sheet Symbol Actions » Create Sheet From Symbol** from the menu that appears.

After launching the command (and choosing the sheet symbol if applicable), a dialog will appear asking whether you wish to reverse input/output directions. This option changes the electrical type for a port on the newly created sheet to be the opposite of that for the corresponding sheet entry that it represents. Choose **Yes** or **No** as required - the schematic document will be created and opened as the active document. The matching ports for the sheet entries on the symbol will be located in the bottom left-hand corner of the new document.

The schematic document that is created is named using the entry in the sheet symbol's **Filename** field. You can either enter the intended name for the document in this field before launching the command, complete with extension (i.e. DocumentName.SchDoc), or leave the name blank and enter the name when saving the generated document at a later stage.

Care should be taken when creating a sheet from a sheet symbol and a sheet with that filename already exists. A new sheet with the same filename will be created. The duplication can be resolved when saving, by either saving the new sheet with a different name, or overwriting the existing sheet if required.

### Creating a VHDL File directly from a Sheet Symbol

This command is used to create a new VHDL document (\*.Vhd) from a placed sheet symbol on the current schematic document.

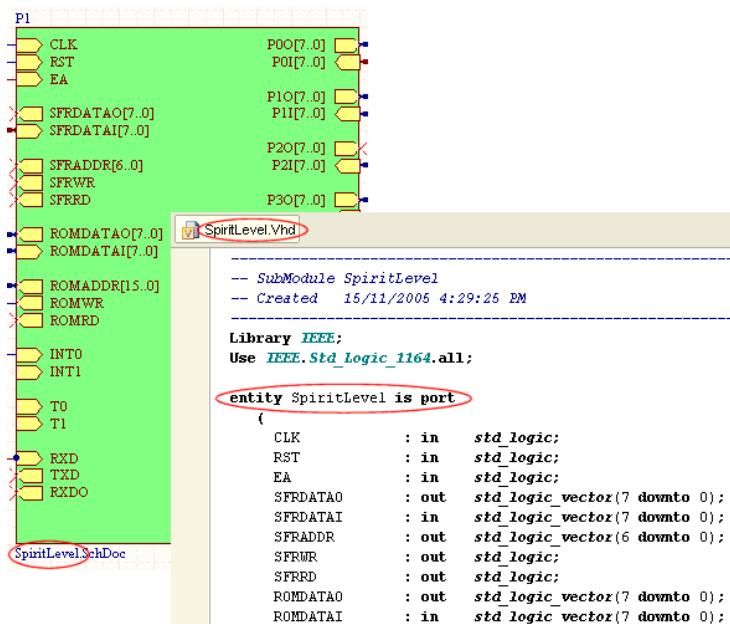
The command can be accessed either by:

- choosing **Design » Create HDL File From Symbol » Create VHDL File From Symbol** from the main menus. You will be prompted to choose a sheet symbol
- right-clicking over the required sheet symbol and choosing **Sheet Symbol Actions » Create VHDL File From Symbol** from the menu that appears.

After launching the command (and choosing the sheet symbol if applicable), the VHDL document will be created and opened as the active document. The sheet entries on the symbol will be included as declared ports in the document's entity definition.

The template for the architecture will be defined, but the component and signal declarations are left ready for you to code in.

The VHDL document is named using the entry in the sheet symbol's **Filename** field. This entry is also used as the name for the document's entity.



### Creating a Verilog File directly from a Sheet Symbol

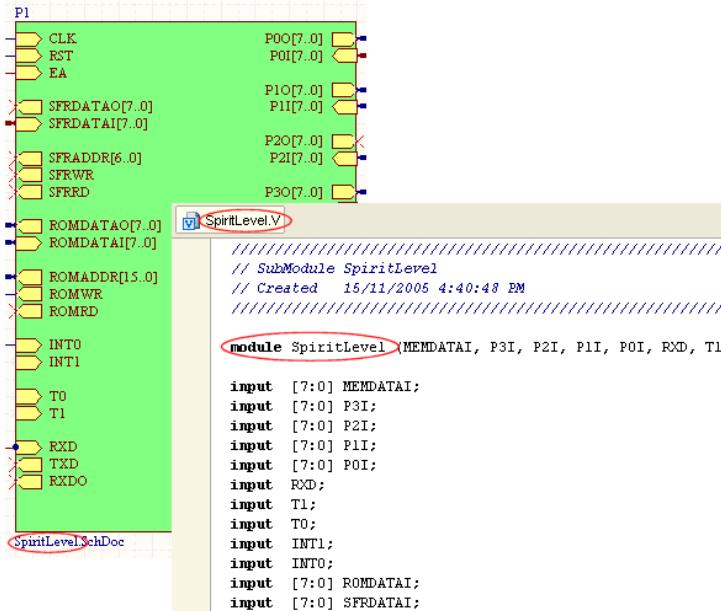
This command is used to create a new Verilog document (\*.v) from a placed sheet symbol on the current schematic document.

The command can be accessed either by:

- choosing **Design » Create HDL File From Symbol » Create Verilog File From Symbol** from the main menus. You will be prompted to choose a sheet symbol
- right-clicking over the required sheet symbol and choosing **Sheet Symbol Actions » Create Verilog File From Symbol** from the menu that appears.

After launching the command (and choosing the sheet symbol if applicable), the Verilog document will be created and opened as the active document. The sheet entries on the symbol will be included in the document's module definition.

The Verilog document is named using the entry in the sheet symbol's **Filename** field. This entry is also used as the name for the document's module.



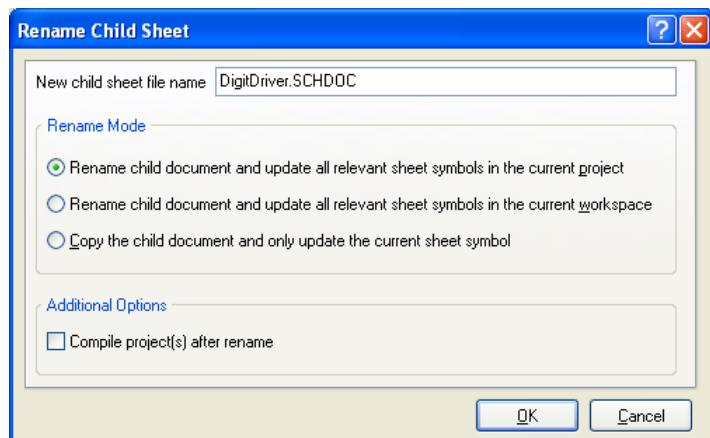
### Renaming a Sheet Symbol's Child Sheet

This command provides a quick and efficient facility for renaming the child schematic sheet referenced by a selected sheet symbol.

The command can be accessed either by:

- choosing **Design » Rename Child Sheet** from the main menus. You will be prompted to choose a sheet symbol
- right-clicking over the required sheet symbol and choosing **Sheet Symbol Actions » Rename Child Sheet** from the menu that appears.

After launching the command (and choosing the sheet symbol if applicable), the *Rename Child Sheet* dialog will appear.



Initially, the **New child sheet file name** field will contain the current name for the document. Type the new name for the document as required - ensuring that the **.SCHDOC** extension remains.

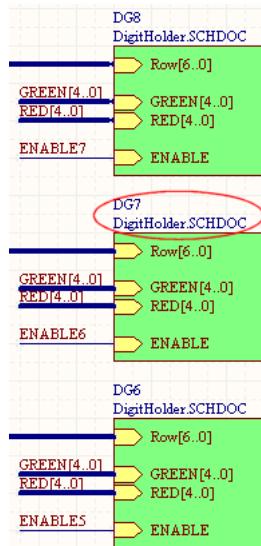
The **Rename Mode** region of the dialog allows you to determine how the renaming should proceed. The first two options, as their names suggest, rename the child sheet. The options differ in the scope of the update with respect to the sheet symbols that point to this sheet - either in the active project or across all projects in the active design workspace. In each case, the **Filename** field for the sheet symbol will be updated to reflect the newly named child sheet.

The third option takes a copy of the child sheet before renaming the original. Only the current sheet symbol is updated using this option. This is useful when the current child sheet is referenced by multiple sheet symbols, and one sheet symbol needs to reference a modified version of the circuitry contained on that sheet. You still want to keep the original sheet - you are creating a renamed copy of this sheet with which to point to from a single sheet symbol. You can then modify the content of the sheet as required.

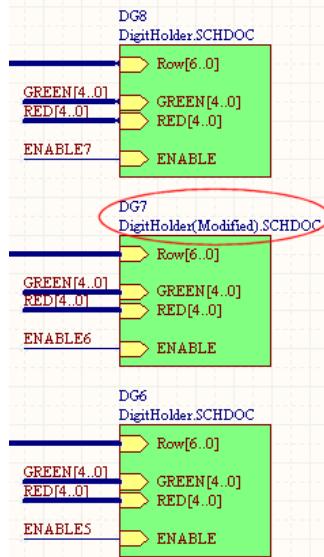
Enabling the **Compile project(s) after rename** option will ensure that the newly named child sheet is correctly inserted into the design hierarchy, which will be reflected on the **Projects** panel.

## Example

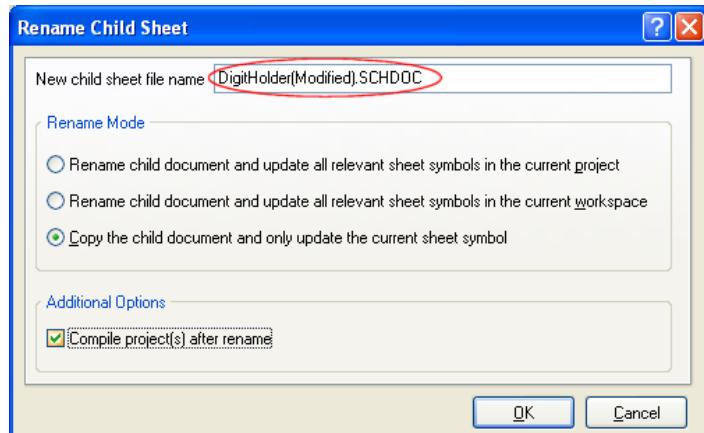
Consider the three sheet symbols in the adjacent image, each one pointing to the `DigitHolder.SchDoc` sub-sheet.



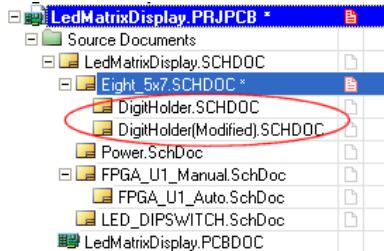
Consider now that the middle symbol is required to point to a slightly modified version of the underlying schematic. The canvas for the modified sheet can be quickly generated and this sheet symbol made to point to it, by using the Rename Child Sheet feature, and selecting the third option - to copy the original sheet and update only the current sheet symbol.



After defining the required rename options, clicking **OK** will proceed with the rename, updating the sheet symbol's **Filename** field to point to the newly named document.



As can be seen in the image and the image below, the original schematic document has not been deleted and the other sheet symbols still point to it. The copy of the original document - renamed to `DigitHolder (Modified).SchDoc` - is referenced only by the intended sheet symbol. The **Projects** panel is updated to reflect the two child sheets that are accessed by sheet symbols on the parent sheet.



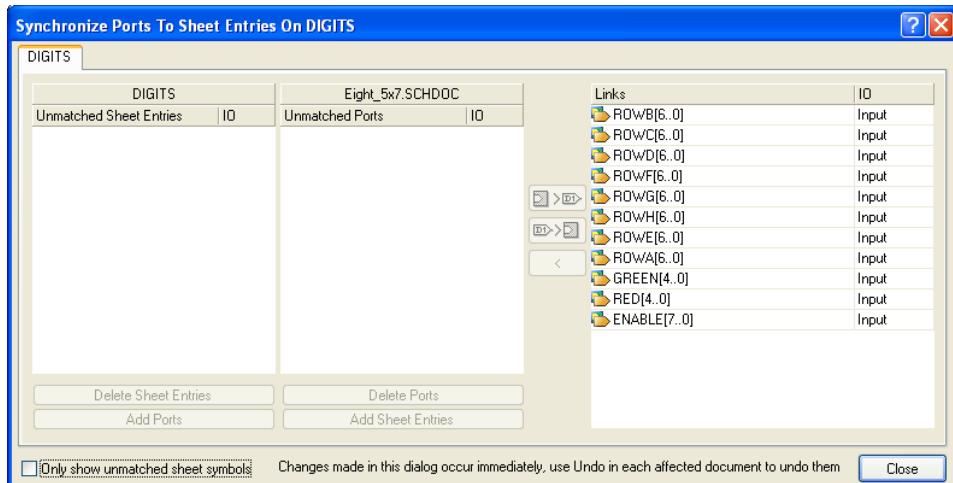
### Synchronizing Sheet Entries and Ports

The Schematic Editor provides a facility for maintaining synchronization between the sheet entries on a sheet symbol and the corresponding ports on the referenced child sheet below. Two commands are used to implement this facility, differing only in their scope.

#### Synchronization for Current Sheet Symbol

Allows you to synchronize the sheet entries and sub-sheet ports for the sheet symbol currently under the cursor. The command can be accessed by right-clicking over the required sheet symbol and choosing **Sheet Symbol Actions** » **Update ports and sheet entries on the chosen sheet symbol** from the menu that appears.

After launching the command, the *Synchronize Ports To Sheet Entries On SheetSymbolDesignator* dialog will appear:

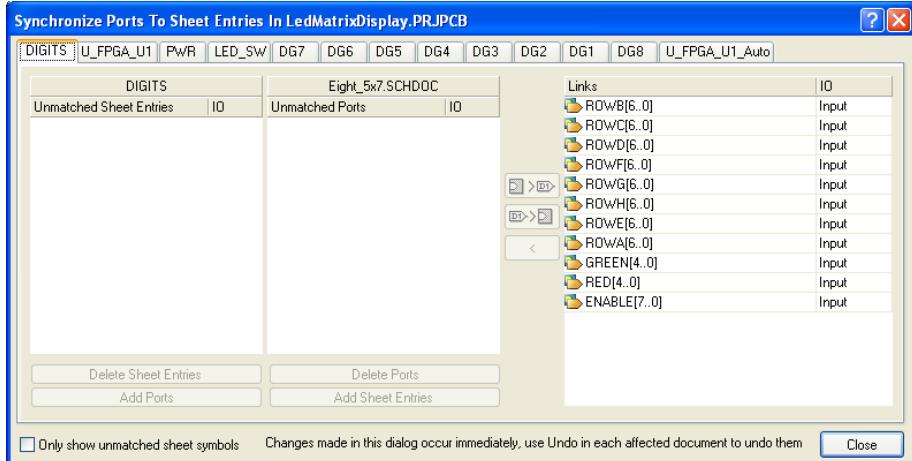


This incarnation of the Synchronize dialog includes a single tab for the sheet symbol under the cursor.

#### Synchronization for all Sheet Symbols

Allows you to synchronize the sheet entries and sub-sheet ports for each sheet symbol in the active design project. The command can be accessed by choosing **Design** » **Synchronize Sheet Entries And Ports** from the main menus.

After launching the command, the *Synchronize Ports To Sheet Entries In ActiveProjectName* dialog will appear:



This incarnation of the Synchronize dialog includes a separate tab for each sheet symbol found in the project. The ordering of the tabs depends on the order in which documents have been added to the project and the order in which sheet symbols were added to each document.

### Using the Dialog

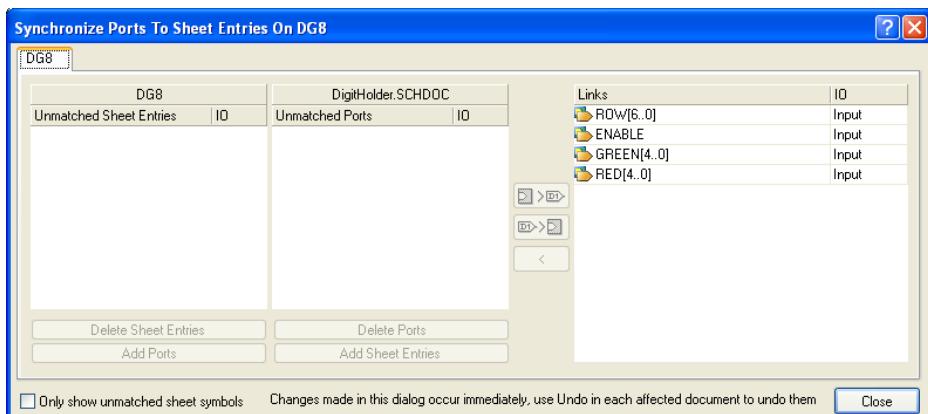
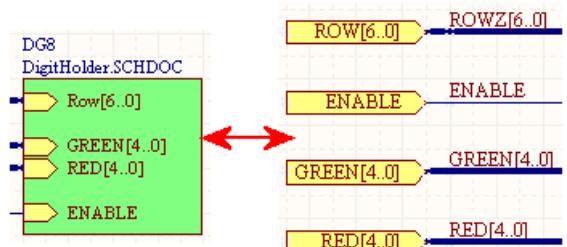
The aim, when using the Synchronize dialog, is to ensure that all sheet entries on a sheet symbol are matched to ports on the referenced child sheet both in terms of name and I/O Type. The left-hand side of the dialog provides two regions:

- one listing all currently unmatched sheet entries associated to the chosen sheet symbol. The Designator of the sheet symbol appears as the header for the region. Each sheet entry in this region is listed in terms of its Designator and I/O Type
- one listing all currently unmatched ports on the schematic sheet referenced by the sheet symbol. The document name appears as the header for the region. Each port entry in this region is listed in terms of its Name and I/O Type.

The right-hand side of the dialog lists all currently linked (or matched) sheet entry-port pairings. Each entry shows the common name used by both the sheet entry as its Designator and the port as its Name. The **IO** column shows the current I/O Type set for each pairing. Both name and I/O Type can be modified from this dialog - click on the required field entry and enter the new name/choose the required I/O Type. Any modification will be passed immediately to both the sheet entry and port.

The simplest way to explain the workings of the dialog, in terms of matching sheet entries with ports, is to use an underlying example. Consider the sheet symbol and corresponding ports in the image to the right.

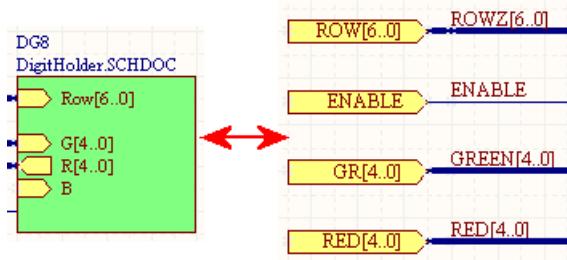
Each sheet entry in the symbol corresponds, both in name and I/O Type, to a port in the referenced schematic sub-sheet. Accessing the Synchronize dialog for this sheet symbol, there are no unmatched entries.



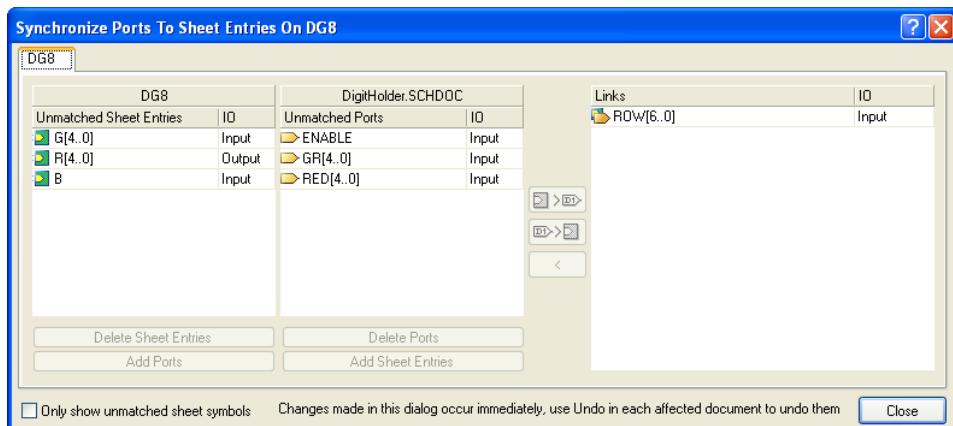
Now consider the same sheet symbol, but with a few intentional changes made within the design, as listed below and summarized in the following image:

- sheet entry designator changed to G [ 4 .. 0 ]

- sheet entry designator changed to R[4..0] and I/O Type changed to Output
- extra sheet entry added, with designator B and I/O Type Input
- sheet entry for ENABLE removed
- port name changed to GR[4..0]



This time when the Synchronize dialog is accessed, the changes introduced cause links between sheet entries and ports to be broken and the unmatched lists become populated.



The two unmatched regions must now be used to resolve the current unsynchronized state that exists between the sheet entries of the symbol and the ports of the sub-sheet. Below each region are two buttons, the use of which will involve making one or more of the following matching decisions:

- If there are sheet entries that you no longer require, select each (use **Ctrl+click**, **Shift+click** or click-and-drag for multi-select) and click the **Delete Sheet Entries** button. Each sheet entry in the selection will be removed from the sheet symbol.
- If there are ports that you no longer require, select each and click the **Delete Ports** button. Each port in the selection will be removed from the child sheet.
- If there are existing sheet entries that you need to keep but no corresponding ports for these entries, you can automatically create ports with the same names and I/O Types by selecting each sheet entry and clicking the **Add Ports** button. You will be taken to the child sheet, with the port(s) floating on the cursor ready for initial placement. Click or press **Enter** to place the port(s). The Synchronize dialog will reappear, with an entry for each sheet entry-port pairing automatically entered into the **Links** region of the dialog.
- If there are existing ports that you need to keep but no corresponding sheet entries for these ports, you can automatically create sheet entries with the same names and I/O Types by selecting each port and clicking the **Add Sheet Entries** button. You will be taken to the sheet symbol, with the sheet entry(ies) floating on the cursor ready for initial placement. Click or press **Enter** to place the sheet entry(ies). The Synchronize dialog will reappear, with an entry for each sheet entry-port pairing automatically entered into the **Links** region of the dialog.

All other residual unmatched entries will either be due to name or I/O Type mismatching. For these, manual matching can be carried out. This is done by selecting a sheet entry in one region and a port in the other and then using one of the following two buttons to determine which name and I/O Type attributes are used when linking:

- Use this button to link the selected sheet entry with the selected port, using the name and I/O Type defined for the sheet entry. The port will be renamed and/or its I/O type changed.
- Use this button to link the selected sheet entry with the selected port, using the name and I/O Type defined for the port. The sheet entry will be renamed and/or its I/O type changed.

If the name or I/O Type is not as you require, remember that these can be changed from the Synchronize dialog, after the unmatched entries have been linked.

If you want to break up a linked entry, select it in the **Links** region of the dialog and click on the  button.

The actual process of matching will vary from sheet symbol to sheet symbol, depending on the discrepancies involved and your preferred matching techniques. Considering the point reached with the underlying example:

DG8		DigitHolder.SCHDOC	
Unmatched Sheet Entries	IO	Unmatched Ports	IO
G[4..0]	Input	ENABLE	Input
R[4..0]	Output	GR[4..0]	Input
B	Input	RED[4..0]	Input

the sheet entries and ports could be quite easily matched and re-synchronized back to their original states by performing the following actions:

- Selecting the entry for port **ENABLE** and clicking the **Add Sheet Entries** button
- Selecting the entry for sheet entry **B** and clicking the **Delete Sheet Entries** button
- Selecting the entries for sheet entry **R[4..0]** and port **RED[4..0]** and clicking the  button
- Selecting the entries for sheet entry **G[4..0]** and port **GR[4..0]** and clicking the  button This keeps the I/O Type setting of **Input**, as required. Then, once linked, change the name to **GREEN[4..0]** in the **Links** region of the dialog

The approach to matching is not set in stone - the tools are provided to allow you to match as quickly and as efficiently as possible. For example, the first two actions in the list above could be achieved by a single, alternative action - to select the entries for sheet entry **B** and port **ENABLE** and click on the  button.

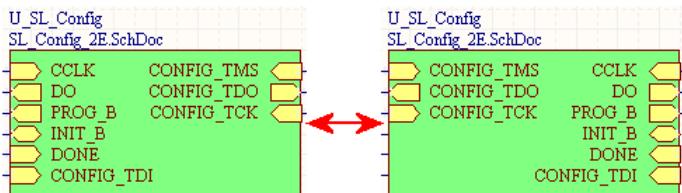
### Flipping Selected Sheet Symbols along the X-Axis

This command allows you to flip selected sheet symbols along the X-axis.

The command can be accessed either by:

- selecting the required symbols and choosing **Edit** » **Move** » **Flip Selected Sheet Symbols Along X** from the main menus
- right-clicking over the required sheet symbol (or a symbol in a selected group of symbols) and choosing **Sheet Symbol Actions** » **Flip Selected Sheet Symbols Along X** from the menu that appears.

After launching the command the selected sheet symbols will be flipped. The sheet entries associated with a symbol will essentially be swapped to the opposite side of the symbol (in the horizontal plane) - those on the left will be repositioned on the right and vice-versa.



**Note:** When flipping multiple selected sheet symbols, the symbols will be flipped about an imaginary vertical line which is located mid-way between the bounding extents of the symbols in the selection.

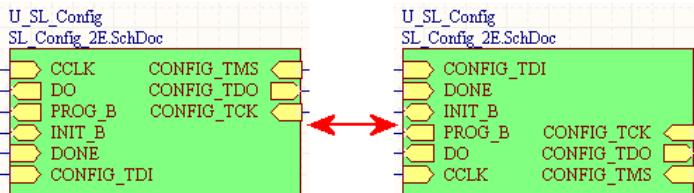
### Flipping Selected Sheet Symbols along the Y-Axis

This command allows you to flip selected sheet symbols along the Y-axis.

The command can be accessed either by:

- selecting the required symbols and choosing **Edit** » **Move** » **Flip Selected Sheet Symbols Along Y** from the main menus
- right-clicking over the required sheet symbol (or a symbol in a selected group of symbols) and choosing **Sheet Symbol Actions** » **Flip Selected Sheet Symbols Along Y** from the menu that appears.

After launching the command the selected sheet symbols will be flipped. The sheet entries associated with a symbol will essentially be swapped to the opposite side of the symbol (in the vertical plane) - those at the top will be repositioned at the bottom and vice-versa.



**Note:** When flipping multiple selected sheet symbols, the symbols will be flipped about an imaginary horizontal line which is located mid-way between the bounding extents of the symbols in the selection.

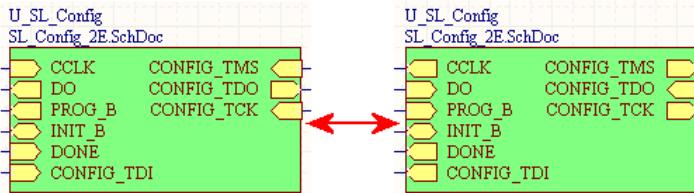
### Toggling Sheet Entry IO Type in a Selected Sheet Symbol

This command allows you to toggle the I/O Type for all sheet entries in all selected sheet symbols, simultaneously.

The command can be accessed either by:

- selecting the required symbols and choosing **Edit » Move » Toggle All Sheet Entries IO Type In Selected Sheet Symbol** from the main menus
- right-clicking over the required sheet symbol (or a symbol in a selected group of symbols) and choosing **Sheet Symbol Actions » Toggle All Sheet Entries IO Type In Selected Sheet Symbol** from the menu that appears.

After launching the command, the I/O Type defined for each sheet entry will be toggled, where applicable.



The actual change depends on the current I/O Type as follows:

- Unspecified remains Unspecified
- Output changes to Input
- Input changes to Output
- Bidirectional remains Bidirectional.

### Notes

The **Filename** property of the sheet symbol, set on the **Properties** tab of the **Sheet Symbol** dialog, must be set to the file name of the schematic sheet that the symbol represents.

---

If a group of sheet entries is pasted into a selected sheet symbol and those entries fall outside the current bounds of the symbol, it will automatically be resized to accommodate them.

---

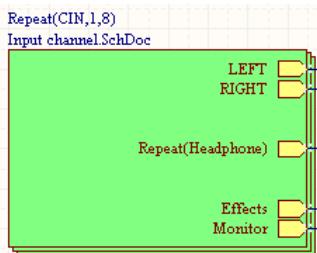
By using sheet symbol instantiation, multiple channels on the same sub-sheet can be referenced from a single sheet symbol.

The syntax used involves the use of the **Repeat** keyword in the sheet symbol's Designator field and takes the form:

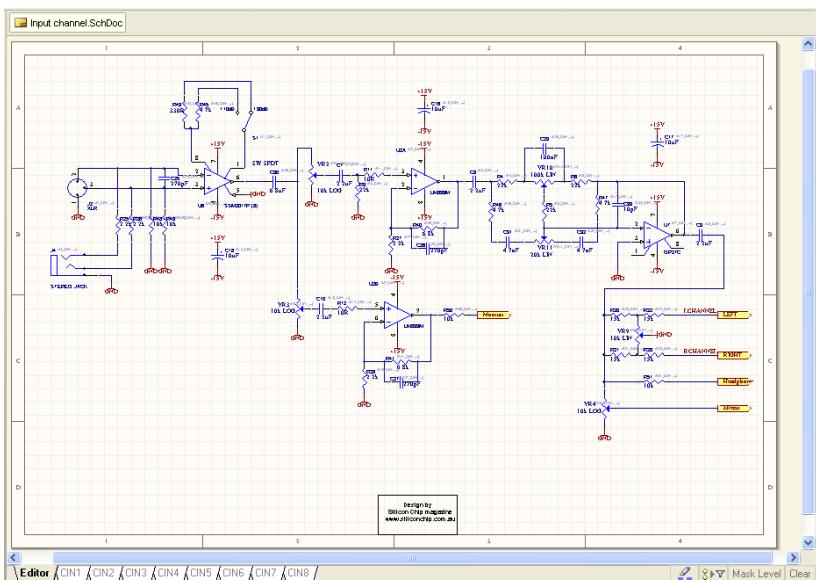
**Repeat**(SheetSymbolDesignator, FirstInstance, LastInstance),

where **SheetSymbolDesignator** is the base name for the sheet symbol and **FirstInstance** and **LastInstance** together define the number of channels to be instantiated. The **FirstInstance** parameter must start at a value of one (1) onwards.

The following example image illustrates the use of the **Repeat** keyword to instantiate 8 input channels for an audio mixer.



When the project is built, the Compiler instantiates the channel the required number of times as it builds the internal compiled model, using a chosen annotation scheme to uniquely identify each component in each channel. The channel sub-sheet is not duplicated. Instead, once compiled, a Compiled Document Tabs appear at the bottom of the sub-sheet document in the main design window for each channel on that sheet, as shown in the example image below.



Multiple sub-sheets may be referenced by a single sheet symbol. Separate each filename by a semi-colon in the **Filename** field. With the effective use of off-sheet connectors placed on the sub-sheets, you can effectively spread a section of your design over multiple sheets, treated as though they were one giant (flat) sheet. Note however, that use of off-sheet connectors is only possible for sheets referenced by the same sheet symbol.

Any changes made to object properties during placement will cause the default properties for the object to be updated, unless the **Permanent** option - on the **Schematic - Default Primitives** page of the *Preferences* dialog - is enabled. When this option is enabled, changes made will affect only the object being placed and subsequent objects placed during the same placement session.

If you attempt to graphically modify a sheet symbol object that has its **Locked** property enabled, a dialog will appear asking for confirmation to proceed with the edit.

If the **Protect Locked Objects** option is enabled in the **Schematic – Graphical Editing** page of the *Preferences* dialog (**Tools** » **Schematic Preferences**), and the **Locked** option for this design object is enabled as well then this object cannot be selected or graphically edited.

You will have to double click on this locked object directly and disable the **Locked** property or disable the **Protect Locked Objects** option in the **Schematic – Graphical Editing** Preferences dialog to graphically edit this object.

If you attempt to select locked objects along with other objects, only those objects that are unlocked can be selected and moved as a group if the **Protect Locked Objects** option is enabled. Otherwise if the **Protect Locked Objects** option is disabled you will be prompted for confirmation to proceed with the edit of the objects including locked objects.

## Sheet Symbol Designator



### Description

The sheet symbol designator is a non-electrical child object of an electrical design primitive. It is used to provide a sheet symbol with a meaningful name that will distinguish it from other sheet symbols placed on the same schematic sheet. Typically the name will reflect the overall function of the schematic sub-sheet that the symbol represents.

### Availability and Placement

The sheet symbol designator is automatically placed when the parent sheet symbol object is placed. As such, it is not a design object that can be accessed and placed by the user.

### Editing

The properties of a sheet symbol designator object can be modified before and after placement of the parent sheet symbol. Editing itself falls into two categories - graphical and non-graphical.

The following three methods of non-graphical editing are available:

#### Editing via an associated properties dialog

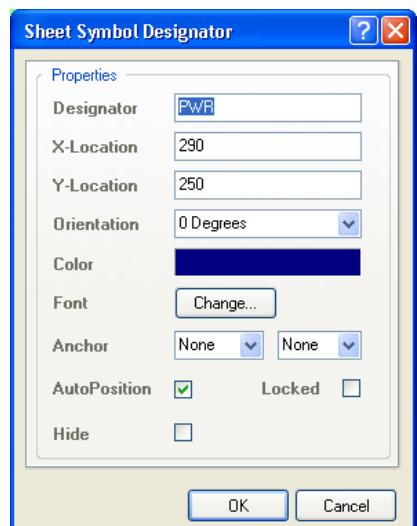
This method of editing uses the following dialog to modify the properties of a sheet symbol designator object, independently of the parent sheet symbol object.

Use the dialog's 'What's This Help' feature to obtain detailed information about each of the options available. Click on the question mark button at the top right of the dialog and then click over a field or option to pop-up information specific to that field or option.

The *Sheet Symbol Designator* dialog can be accessed prior to entering sheet symbol placement mode, from the **Schematic - Default Primitives** page of the *Preferences* dialog (**Tools » Schematic Preferences**). This allows you to change the default properties for the sheet symbol designator object, which will be applied when placing subsequent sheet symbols.

After placement, the *Sheet Symbol Designator* dialog can be accessed in one of the following ways:

- double-clicking on the designator field of the placed sheet symbol object
- selecting the designator field of the sheet symbol object and choosing **Properties** from the right-click pop-up menu
- choosing the **Change** command from the **Edit** menu and then clicking once over the designator field of the placed sheet symbol object.



#### Editing via the SCH Inspector panel

The **SCH Inspector** panel enables you to interrogate and edit the properties of one or more design objects in the active document. Used in conjunction with appropriate filtering, the panel can be used to make changes to multiple objects of the same kind, from one convenient location.

For more information on the **SCH Inspector** panel, press **F1** when the cursor is over the panel.

#### Editing via the SCH List panel

The **SCH List** panel allows you to display design objects from one or more documents in tabular format, enabling you to quickly inspect and modify object attributes. When used in conjunction with the **SCH Filter** panel, it enables you to display just those objects falling under the scope of the active filter - allowing you to target and edit multiple design objects with greater accuracy and efficiency.

For more information on the **SCH List** and **SCH Inspector** panels, press **F1** when the cursor is over a panel.

## Graphical editing

This method of editing allows you to select a sheet symbol designator object directly in the workspace and change its location graphically. Sheet symbol designators can only be adjusted with respect to their size by changing the size of the font used (accessed through the *Sheet Symbol Designator* dialog). As such, editing handles are not available when the sheet symbol designator object is selected:

**Designator**

Click anywhere inside the dashed box and drag to reposition the sheet symbol designator object as required. The object can be rotated or flipped while dragging:

- Press the **Spacebar** to rotate the designator. Rotation is anti-clockwise and in steps of 90°.
- Press the **X** or **Y** keys to flip the designator along the X-axis or Y-axis respectively.

If the **Enable In-Place Editing** option is enabled on the **Schematic - General** page of the *Preferences* dialog (**Tools » Schematic Preferences**), you will be able to edit the name for the sheet symbol designator directly in the workspace. Select the designator and then click once to invoke the feature. Type the new name as required and then click away from the designator field or press **Enter** to effect the change.

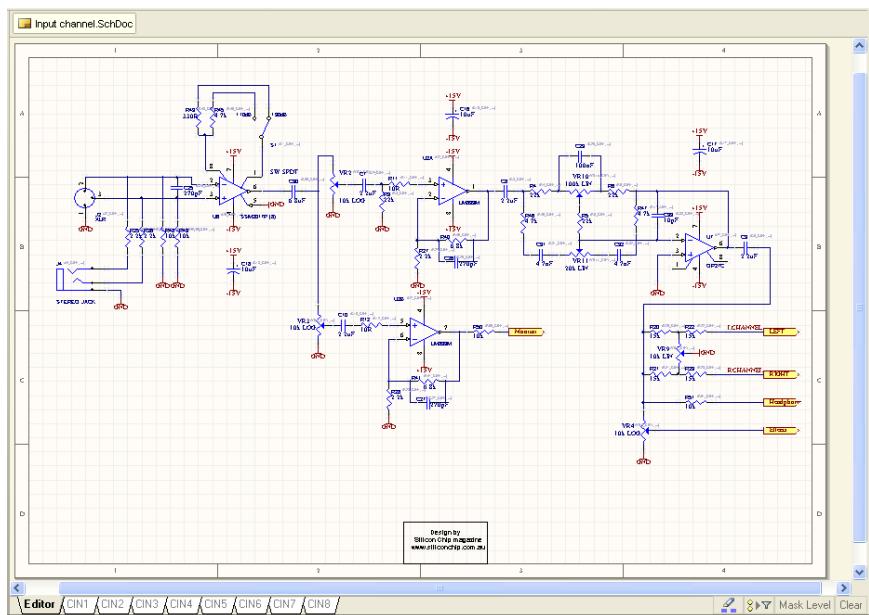
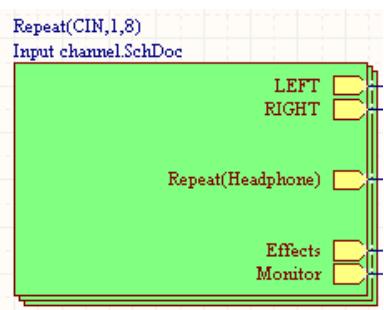
## Notes

By using sheet symbol instantiation, multiple channels on the same sub-sheet can be referenced from a single sheet symbol. The syntax used involves the use of the **Repeat** keyword in the sheet symbol's Designator field and takes the form:

**Repeat**(*SheetSymbolDesignator*, *FirstInstance*, *LastInstance*),

where *SheetSymbolDesignator* is the base name for the sheet symbol and *FirstInstance* and *LastInstance* together define the number of channels to be instantiated. The *FirstInstance* parameter should start at 1 or greater. The example image below illustrates the use of the **Repeat** keyword to instantiate 8 input channels for an audio mixer.

When the project is built, the Compiler instantiates the channel the required number of times as it builds the internal compiled model, using a chosen annotation scheme to uniquely identify each component in each channel. The channel sub-sheet is not duplicated. Instead, once compiled, a separate tab appears at the bottom of the sub-sheet document in the main design window, for each channel on that sheet, as shown in the example image below.



Any changes made to the Designator field during sheet symbol placement will cause the default properties for the sheet symbol designator object to be updated, unless the **Permanent** option - on the **Schematic - Default Primitives** page of the

Preferences dialog - is enabled. When this option is enabled, changes made will affect only the designator of the sheet symbol object being placed and subsequent sheet symbol objects placed during the same placement session.

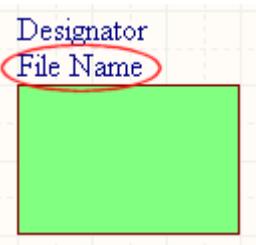
If you attempt to graphically modify a sheet symbol designator object that has its **Locked** property enabled, a dialog will appear asking for confirmation to proceed with the edit.

If the **Protect Locked Objects** option is enabled in the **Schematic – Graphical Editing** page of the *Preferences* dialog (**Tools** » **Schematic Preferences**), and the **Locked** option for this design object is enabled as well then this object cannot be selected or graphically edited.

You will have to double click on this locked object directly and disable the **Locked** property or disable the **Protect Locked Objects** option in the **Schematic – Graphical Editing** Preferences dialog to graphically edit this object.

If you attempt to select locked objects along with other objects, only those objects that are unlocked can be selected and moved as a group if the **Protect Locked Objects** option is enabled. Otherwise if the **Protect Locked Objects** option is disabled you will be prompted for confirmation to proceed with the edit of the objects including locked objects.

## Sheet Symbol Filename



### Description

The sheet symbol filename is a non-electrical child object of an electrical design primitive. It provides the link between the sheet symbol and the schematic sub-sheet that the symbol represents.

### Availability and Placement

The sheet symbol filename is automatically placed when the parent sheet symbol object is placed. As such, it is not a design object that can be accessed and placed by the user.

### Editing

The properties of a sheet symbol filename object can be modified before and after placement of the parent sheet symbol. Editing itself falls into two categories - graphical and non-graphical.

The following three methods of non-graphical editing are available:

#### Editing via an associated properties dialog

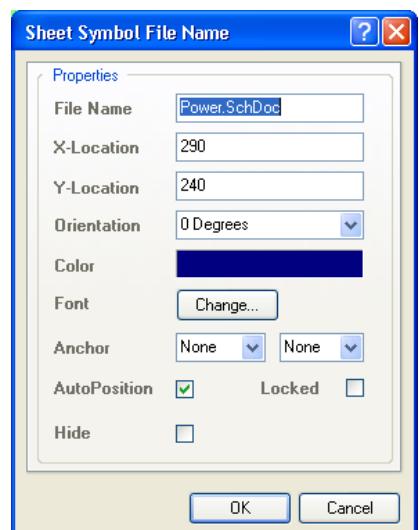
This method of editing uses the following dialog to modify the properties of a sheet symbol filename object, independently of the parent sheet symbol object.

Use the dialog's 'What's This Help' feature to obtain detailed information about each of the options available. Click on the question mark button at the top right of the dialog and then click over a field or option to pop-up information specific to that field or option.

The *Sheet Symbol File Name* dialog can be accessed prior to entering sheet symbol placement mode, from the **Schematic - Default Primitives** page of the *Preferences* dialog (**Tools » Schematic Preferences**). This allows you to change the default properties for the sheet symbol filename object, which will be applied when placing subsequent sheet symbols.

After placement, the *Sheet Symbol File Name* dialog can be accessed in one of the following ways:

- double-clicking on the filename field of the placed sheet symbol object
- selecting the filename field of the sheet symbol object and choosing **Properties** from the right-click pop-up menu
- choosing the **Change** command from the **Edit** menu and then clicking once over the filename field of the placed sheet symbol object.



#### Editing via the SCH Inspector panel

The **SCH Inspector** panel enables you to interrogate and edit the properties of one or more design objects in the active document. Used in conjunction with appropriate filtering, the panel can be used to make changes to multiple objects of the same kind, from one convenient location.



For more information on the **SCH Inspector** panel, press **F1** when the cursor is over the panel.

#### Editing via the SCH List panel

The **SCH List** panel allows you to display design objects from one or more documents in tabular format, enabling you to quickly inspect and modify object attributes. When used in conjunction with the **SCH Filter** panel, it enables you to display just those objects falling under the scope of the active filter - allowing you to target and edit multiple design objects with greater accuracy and efficiency.

 For more information on the **SCH List** and **SCH Inspector** panels, press **F1** when the cursor is over a panel.

### Graphical editing

This method of editing allows you to select a sheet symbol filename object directly in the workspace and change its location graphically. Sheet symbol filenames can only be adjusted with respect to their size by changing the size of the font used (accessed through the *Sheet Symbol File Name* dialog). As such, editing handles are not available when the sheet symbol filename object is selected:



File Name

Click anywhere inside the dashed box and drag to reposition the sheet symbol filename object as required. The object can be rotated or flipped while dragging:

- Press the **Spacebar** to rotate the filename. Rotation is anti-clockwise and in steps of 90°.
- Press the **X** or **Y** keys to flip the filename along the X-axis or Y-axis respectively.

If the **Enable In-Place Editing** option is enabled on the **Schematic - General** page of the *Preferences* dialog (**Tools** » **Schematic Preferences**), you will be able to edit the entry for the sheet symbol filename directly in the workspace. Select the filename field and then click once to invoke the feature. Type the new entry as required and then click away from the filename field or press **Enter** to effect the change.

If you attempt to graphically modify a sheet symbol filename object that has its **Locked** property enabled, a dialog will appear asking for confirmation to proceed with the edit.

If the **Protect Locked Objects** option is enabled in the **Schematic – Graphical Editing** page of the *Preferences* dialog (**Tools** » **Schematic Preferences**), and the **Locked** option for this design object is enabled as well then this object cannot be selected or graphically edited.

You will have to double click on this locked object directly and disable the **Locked** property or disable the **Protect Locked Objects** option in the **Schematic – Graphical Editing** Preferences dialog to graphically edit this object.

If you attempt to select locked objects along with other objects, only those objects that are unlocked can be selected and moved as a group if the **Protect Locked Objects** option is enabled. Otherwise if the **Protect Locked Objects** option is disabled you will be prompted for confirmation to proceed with the edit of the objects including locked objects.

### Notes

The **Filename** property of the sheet symbol, which can also be set on the **Properties** tab of the *Sheet Symbol* dialog, must be set to the file name of the schematic sub-sheet that the symbol represents.

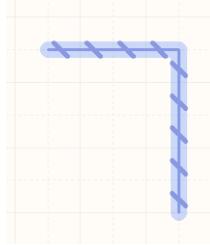
---

Multiple sub-sheets may be referenced by a single sheet symbol. Separate each filename by a semi-colon, in the **Filename** field. With the effective use of off-sheet connectors, placed on the sub-sheets, you can effectively spread a section of your design over multiple sheets, treated as though they were one giant (flat) sheet. Note however, that use of off-sheet connectors is only possible for sheets referenced by the same sheet symbol.

---

Any changes made to the **Filename** field during sheet symbol placement will cause the default properties for the sheet symbol filename object to be updated, unless the **Permanent** option - on the **Schematic - Default Primitives** page of the *Preferences* dialog - is enabled. When this option is enabled, changes made will affect only the filename field of the sheet symbol object being placed and subsequent sheet symbol objects placed during the same placement session.

## Signal Harness



### Description

A Signal Harness is an electrical design primitive. It is an abstract connection which combines different signals.

### Availability

Signal Harnesses are available for placement in the Schematic Editor only. Use one of the following methods to access the placement command:

- choose **Place » Harness » Signal Harness [P, H, H]** from the main menus
- click the  button on the **Wiring** toolbar.

### Placement

After launching the command, the cursor will change to a cross-hair and you will enter Signal Harness placement mode. Placement is made by performing the following sequence of actions:

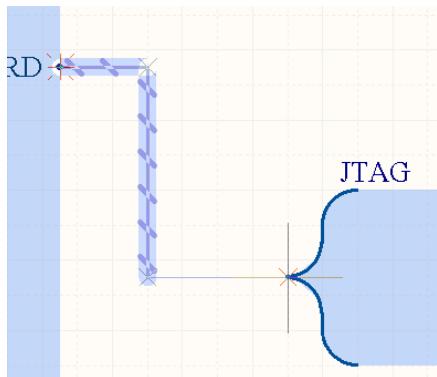
- click or press **Enter** to anchor the starting point for the Signal Harness
- position the cursor and click or press **Enter** to anchor a series of vertex points that define the shape of the signal harness
- after placing the final vertex point, right-click or press **Esc** to complete placement of the Signal Harness.

Continue placing further Signal Harness objects, or right-click or press **Esc** to exit placement mode.

Use the **Backspace** or **Delete** keys to remove the last Signal Harness segment placed.

### Guided wiring of a Signal Harness

Schematics have a definable electrical grid that makes it easy to define electrical connections between objects. As you are placing a Signal Harness, when the Signal Harness falls within the electrical grid range of another electrical object the cursor will snap to the fixed object and a Hot Spot (red cross) will appear.



The Hot Spot guides you to where a valid connection can be made and automatically snaps the cursor to electrical connection points.

The electrical grid can be defined on the **Sheet Options** tab of the *Document Options* dialog (**Design » Document Options**). It is recommended that you set the electrical grid to be slightly smaller than the current snap grid, or it becomes difficult to position electrical objects one snap grid apart.

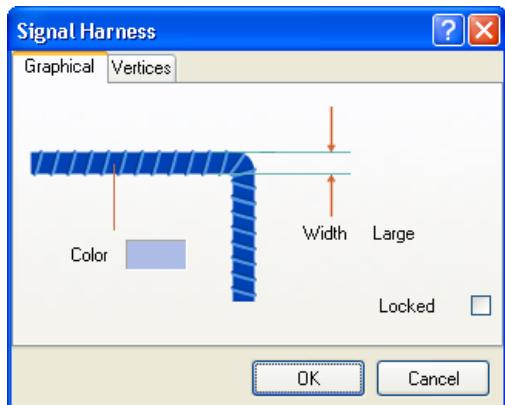
### Editing

The properties of a Signal Harness object can be modified before, during and after placement. Editing itself falls into two categories - graphical and non-graphical.

The following three methods of non-graphical editing are available:

### Editing via an associated properties dialog

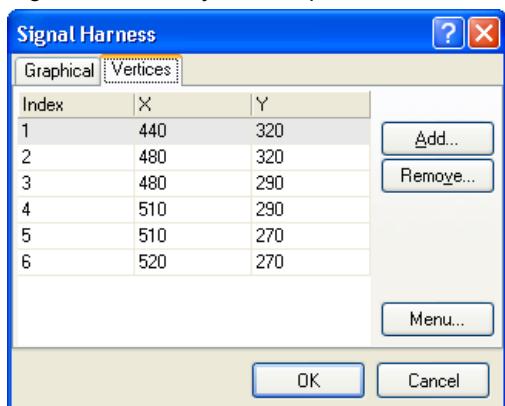
This method of editing uses the following dialog to modify the properties of a Signal Harness object.



Use the dialog's 'What's This Help' feature to obtain detailed information about each of the options available. Click on the question mark button at the top right of the dialog and then click over a field or option to pop-up information specific to that field or option.

### Editing Vertices

The *Signal Harness* dialog provides a **Vertices** tab, from where you can edit the individual vertices of the currently selected Signal Harness object as required.



The main region of the tab lists all of the vertex points currently defined for the Signal Harness. You can add new vertices to the Signal Harness, edit the coordinates of existing vertices, or remove selected vertices altogether.

Click the **Menu** button or right-click within the main list region to access a pop-up menu containing the following commands:

- **Edit** - right click on a coordinate cell (X or Y) for a vertex and use this command to edit the value in that cell. Alternatively, click directly on the cell
- **Add** - use this command to add a new vertex point. The new vertex will be added below the currently focused vertex entry (as distinguished by a dotted outline around a cell in its row) and will initially have the same coordinates as the focused entry
- **Remove** - use this command to remove the currently selected vertex entries in the list. This command will be unavailable if there are only two vertices present for the wire
- **Copy** - use this command to copy the content of the selected cells in the list to the clipboard (alternatively use **Ctrl+C**)
- **Paste** - use this command to paste the content of the clipboard into the list, starting at the selected cell (alternatively use **Ctrl+V**)
- **Select All** - use this command to quickly select the entire grid contents of the list
- **Select Column** - use this command to quickly select the entire column in which the currently focused cell resides
- **Move Up** - use this command to move the selected vertex upward in the list
- **Move Down** - use this command to move the selected vertex downward in the list

- **Move Signal Harness By XY** - use this command to move the entire signal harness object. The **Move Signal Harness By** dialog will appear, from where you can enter the increment value to be applied to each vertex point's X and Y coordinates.

#### Dialog access

The *Signal Harness* dialog can be accessed prior to entering placement mode, from the **Schematic - Default Primitives** page of the *Preferences* dialog (**Tools** » **Schematic Preferences**). This allows you to change the default properties for the signal harness object, which will be applied when placing subsequent signal harnesses.

During placement, the *Signal Harness* dialog can be accessed by pressing the **Tab** key.

After placement, the *Signal Harness* dialog can be accessed in one of the following ways:

- double-clicking on the placed Signal Harness object
- selecting the Signal Harness object and choosing **Properties** from the right-click pop-up menu
- choosing the **Change** command from the **Edit** menu and then clicking once over the placed Signal Harness object.

#### Editing via the SCH Inspector panel

The **SCH Inspector** panel enables you to interrogate and edit the properties of one or more design objects in the active document. Used in conjunction with appropriate filtering, the panel can be used to make changes to multiple objects of the same kind, from one convenient location.

 For more information on the **SCH Inspector** panel, press **F1** when the cursor is over the panel.

#### Editing via the SCH List panel

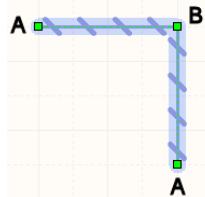
The **SCH List** panel allows you to display design objects from one or more documents in tabular format, enabling you to quickly inspect and modify object attributes. When used in conjunction with the **SCH Filter** panel, it enables you to display just those objects falling under the scope of the active filter - allowing you to target and edit multiple design objects with greater accuracy and efficiency.

 For more information on the **SCH List** and **SCH Inspector** panels, press **F1** when the cursor is over a panel.

#### Graphical editing

This method of editing allows you to select a placed Signal Harness object directly in the workspace and change its size and/or shape, graphically.

When a Signal Harness object is selected, the following editing handles are available:

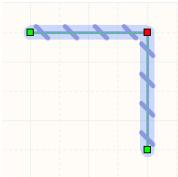


Click and drag **A** to reposition the end points of the Signal Harness.

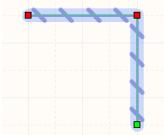
Click and drag **B** to move a vertex. The end points will remain anchored.

Click and drag near the center of a Signal Harness segment to "grab" that segment and reposition it. The end points and other vertices will remain anchored.

With the Signal Harness selected, click on a vertex or segment to individually select that vertex or segment. This Signal Harness 'sub-selection' is distinguished by the associated editing handles becoming red in color.



Individual vertex sub-selection.



Individual segment sub-selection.

The associated vertex (or vertices for a segment) can then be edited directly using the **SCH Inspector** or **SCH List** panels, with any changes appearing immediately on the schematic.

If you attempt to graphically modify a signal harness object that has its **Locked** property enabled, a dialog will appear asking for confirmation to proceed with the edit.

If the **Protect Locked Objects** option is enabled in the **Schematic – Graphical Editing** page of the *Preferences* dialog (**Tools** » **Schematic Preferences**), and the **Locked** option for this design object is enabled as well then this object cannot be selected or graphically edited.

You will have to double click on this locked object directly and disable the **Locked** property or disable the **Protect Locked Objects** option in the **Schematic – Graphical Editing** Preferences dialog to graphically edit this object.

If you attempt to select locked objects along with other objects, only those objects that are unlocked can be selected and moved as a group if the **Protect Locked Objects** option is enabled. Otherwise if the **Protect Locked Objects** option is disabled you will be prompted for confirmation to proceed with the edit of the objects including locked objects.

 Please refer to the [Using Signal Harnesses](#) document for more information on how to use Signal Harnesses.

## Notes

When placing a Signal Harness, various placement modes are available: 90 Degree, 45 Degree, Any Angle and Auto Wire. The mode specifies how corners are created when placing wires and the angles at which wires can be placed. The 90 Degree and 45 Degree modes (true orthogonal modes) both have Start and End sub-modes.

---

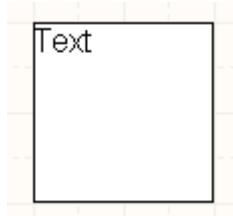
When placing a Signal Harness, press **Shift+Spacebar** to cycle through the wire placement modes. Press **Spacebar** to toggle between the Start and End sub-modes (when in 90 Degree or 45 Degree modes), or between Any Angle and Auto Wire modes (when either of these modes is active). The current placement mode is displayed in the status bar. You can change modes at any time during Signal Harness placement.

In all modes other than Any Angle, the Signal Harness segment attached to the cursor is a "look ahead" segment. The segment you are actually placing precedes this look ahead segment.

---

Any changes made to object properties during placement will cause the default properties for the object to be updated, unless the **Permanent** option - on the **Schematic - Default Primitives** page of the *Preferences* dialog - is enabled. When this option is enabled, changes made will affect only the object being placed and subsequent objects placed during the same placement session.

## Text Frame



### Description

A text frame is a non-electrical drawing primitive. It is used to define an area on a schematic to contain textual information. The frame is a resizable rectangular area that can contain multiple lines of text and can automatically wrap and clip text to keep it within the bounds of the frame.

### Availability

Text frames are available for placement in the Schematic Editor only. Use one of the following methods to access the placement command:

- choose **Place » Text Frame [P, F]** from the Schematic Editor main menus
- click the button on the **Utility Tools** drop-down of the **Utilities** toolbar.

### Placement

After launching the command, the cursor will change to a cross-hair and you will enter text frame placement mode. Placement is made by performing the following sequence of actions:

- click or press **Enter** to anchor the first corner of the text frame
- move the cursor to adjust the size of the text frame, then click or press **Enter** to anchor the diagonally-opposite corner and thereby complete placement of the text frame.

Continue placing further text frames, or right-click or press **Esc** to exit placement mode.

The text frame object can be rotated or flipped while in placement mode and before the first corner of the frame is anchored:

- Press the **Spacebar** to rotate the text frame. Rotation is anti-clockwise and in steps of 90°.
- Press the **X** or **Y** keys to flip the text frame along the X-axis or Y-axis respectively.

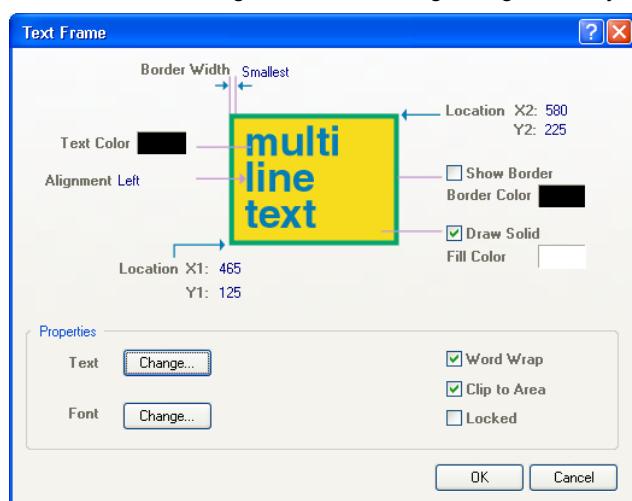
### Editing

The properties of a text frame object can be modified before, during and after placement. Editing itself falls into two categories - graphical and non-graphical.

The following three methods of non-graphical editing are available:

#### Editing via an associated properties dialog

This method of editing uses the following dialog to modify the properties of a text frame object.



Use the dialog's 'What's This Help' feature to obtain detailed information about each of the options available. Click on the question mark button at the top right of the dialog and then click over a field or option to pop-up information specific to that field or option.

The *Text Frame* dialog can be accessed prior to entering placement mode, from the **Schematic - Default Primitives** page of the *Preferences* dialog (**Tools » Schematic Preferences**). This allows you to change the default properties for the text frame object, which will be applied when placing subsequent text frames.

During placement, the *Text Frame* dialog can be accessed by pressing the **Tab** key.

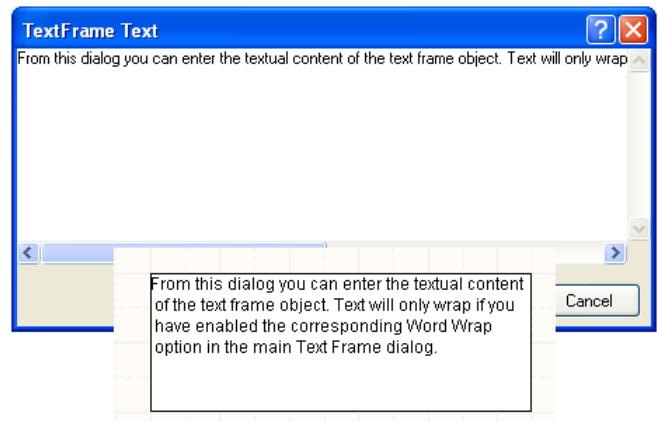
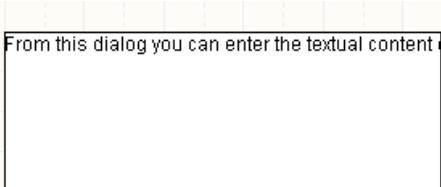
After placement, the *Text Frame* dialog can be accessed in one of the following ways:

- double-clicking on the placed text frame object
- selecting the text frame object and choosing **Properties** from the right-click pop-up menu
- choosing the **Change** command from the **Edit** menu and then clicking once over the placed text frame object.

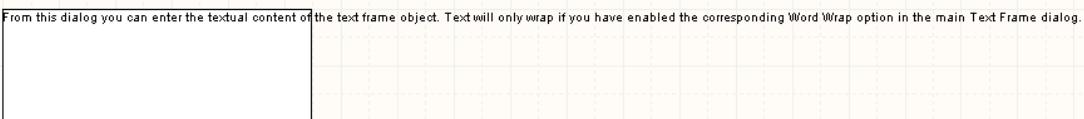
The actual text of the frame can be entered/edited via the **Text Frame** dialog. Press the **Change** button associated with the frame's **Text** property to access the **TextFrame Text** dialog.

In addition to providing a **Word Wrap** option, the main **Text Frame** dialog provides a **Clip to Area** option. With this option enabled, text will be kept within the bounds of the frame. When disabled, text will spill out of the frame onto the schematic sheet.

For example, consider the text frame in the previous image. Both **Word Wrap** and **Clip to Area** options were enabled. If word wrapping is disabled, leaving only the clipping option enabled, only text that fits within the text frame will remain displayed:



However, if the option to clip the text is disabled, the text is free to be displayed beyond the constraints of the frame's boundary:



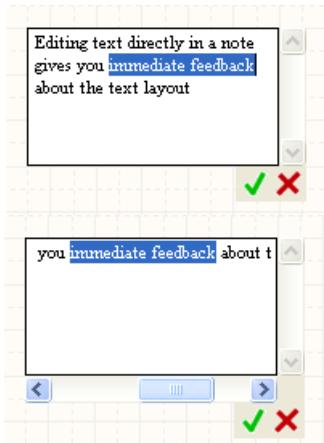
### In-Place Editing

If the **Enable In-Place Editing** option is enabled on the **Schematic - General** page of the **Preferences** dialog (**Tools** » **Schematic Preferences**), you will be able to edit the textual content of the text frame directly in the workspace. Select the text frame and then click once to invoke the feature.

Make changes to the text as required. The available right-click menu provides standard editing commands such as cut, copy, paste and delete.

To effect a change either click away from the text frame or press the green tick button to effect the change. If you decide the change made is not needed, press the red cross button to discard the change.

If the **Word Wrap** option is disabled in the **Text Frame** dialog, a horizontal scroll bar will also be available when editing the text in-situ.



### Editing via the SCH Inspector panel

The **SCH Inspector** panel enables you to interrogate and edit the properties of one or more design objects in the active document. Used in conjunction with appropriate filtering, the panel can be used to make changes to multiple objects of the same kind, from one convenient location.

For more information on the **SCH Inspector** panel, press **F1** when the cursor is over the panel.

### Editing via the SCH List panel

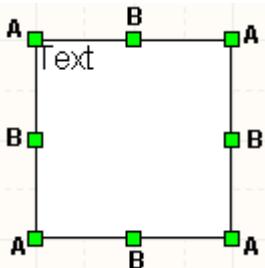
The **SCH List** panel allows you to display design objects from one or more documents in tabular format, enabling you to quickly inspect and modify object attributes. When used in conjunction with the **SCH Filter** panel, it enables you to display just those objects falling under the scope of the active filter - allowing you to target and edit multiple design objects with greater accuracy and efficiency.

For more information on the **SCH List** and **SCH Inspector** panels, press **F1** when the cursor is over a panel.

## Graphical editing

This method of editing allows you to select a placed text frame object directly in the workspace and change its size, shape or location, graphically.

When a text frame object is selected, the following editing handles are available:



Click and drag **A** to resize the text frame in the vertical and horizontal directions simultaneously.

Click and drag **B** to resize the text frame in the vertical and horizontal directions separately.

Click anywhere on the text frame - away from editing handles - and drag to reposition it. The text frame can be rotated or flipped while dragging.

If you attempt to graphically modify a text frame object that has its **Locked** property enabled, a dialog will appear asking for confirmation to proceed with the edit.

If the **Protect Locked Objects option** is enabled in the **Schematic – Graphical Editing** page of the *Preferences* dialog (**Tools** » **Schematic Preferences**), and the **Locked** option for this design object is enabled as well then this object cannot be selected or graphically edited.

You will have to double click on this locked object directly and disable the **Locked** property or disable the **Protect Locked Objects** option in the **Schematic – Graphical Editing** Preferences dialog to graphically edit this object.

If you attempt to select locked objects along with other objects, only those objects that are unlocked can be selected and moved as a group if the **Protect Locked Objects** option is enabled. Otherwise if the **Protect Locked Objects** option is disabled you will be prompted for confirmation to proceed with the edit of the objects including locked objects.

## Notes

While text frames can be rotated or flipped along the x or y axis, this has no effect on the orientation of the text.

---

For simple one line text annotations, use the Text String object.

---

Any changes made to object properties during placement will cause the default properties for the object to be updated, unless the **Permanent** option - on the **Schematic - Default Primitives** page of the *Preferences* dialog - is enabled. When this option is enabled, changes made will affect only the object being placed and subsequent objects placed during the same placement session.

## Text String



### Description

A text string (also referred to as an annotation) is a non-electrical drawing primitive. It is a single line of free text that can be placed on a schematic sheet. Uses might include section headings, revision history, timing information or some other descriptive or instructive text.

### Availability

Text strings are available for placement in both Schematic and Schematic Library Editors by:

- Choosing **Place » Text String [P, T]** from the main menus
- Clicking the A button on the **Utility Tools** drop-down of the **Utilities** toolbar.

### Placement

After launching the command, the cursor will change to a cross-hair and you will enter text string placement mode. Position the cursor on the sheet and click or press **Enter** to effect placement of the string.

Continue placing further text strings or right-click or press **Esc** to exit placement mode.

Press the **Spacebar** while in placement mode to rotate the text string. Rotation is anti-clockwise and in steps of 90°.

Press the **X** or **Y** keys while in placement mode to flip the text string along the X-axis or Y-axis respectively.

### Editing

The properties of a text string object can be modified before, during and after placement. Editing itself falls into two categories - graphical and non-graphical.

The following three methods of non-graphical editing are available:

#### Editing via an associated properties dialog

This method of editing uses the following dialog to modify the properties of a text string object.

Use the dialog's 'What's This Help' feature to obtain detailed information about each of the options available. Click on the question mark button at the top right of the dialog and then click over a field or option to pop-up information specific to that field or option.

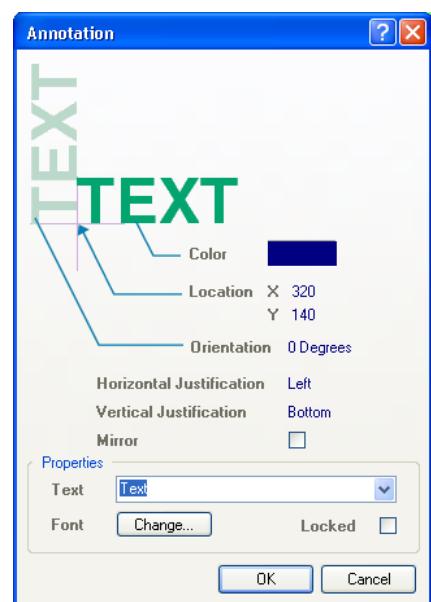
The **Annotation** dialog can be accessed prior to entering placement mode, from the **Schematic - Default Primitives** page of the *Preferences* dialog (**Tools » Schematic Preferences**). This allows you to change the default properties for the text string object, which will be applied when placing subsequent text strings.

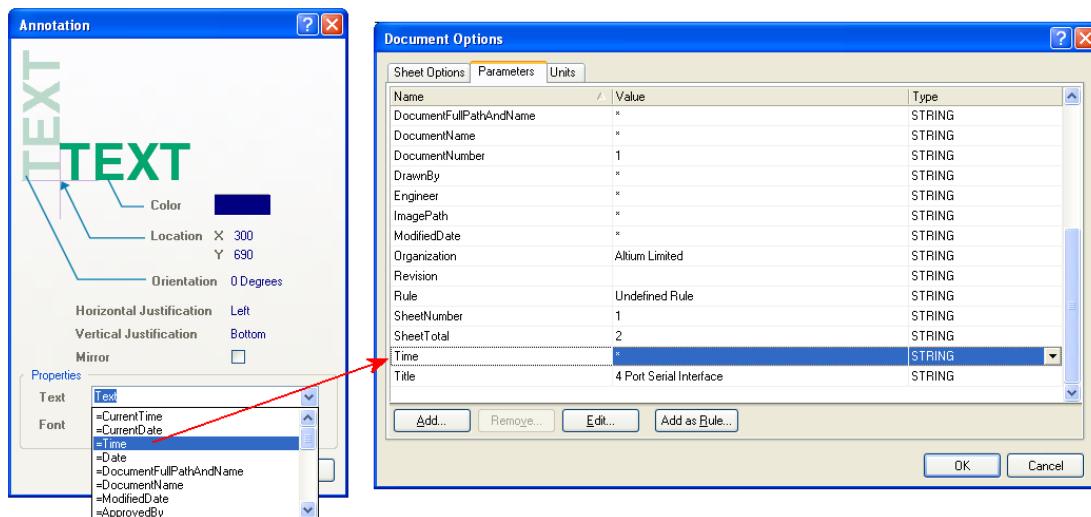
During placement, the **Annotation** dialog can be accessed by pressing the **Tab** key.

After placement, the **Annotation** dialog can be accessed in one of the following ways:

- double-clicking on the placed text string object
- selecting the text string object and choosing **Properties** from the right-click pop-up menu (Schematic Editor only)
- choosing the **Change** command from the **Edit** menu and then clicking once over the placed text string object.

The actual content of the text string can be entered/edited via the **Annotation** dialog using the **Text** field. Type the required text directly into the field. The **Text** field also provides a drop-down list of special strings that link to parameter information defined for the active document on the **Parameters** tab of the *Document Options* dialog (**Design » Document Options**).





## Special Strings

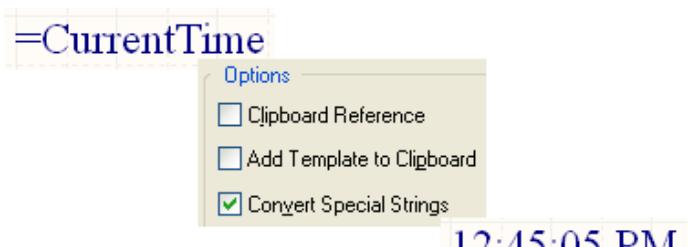
Special strings are text strings which are recognized and interpreted when the sheet is printed or plotted. Certain special strings provide current information, such as date and time, which are inserted at the time of printing.

Some special strings can be displayed on the document prior to printing, by enabling the Convert Special Strings option on the **Schematic - Graphical Editing** page of the *Preferences* dialog (**Tools** » **Schematic Preferences**).

Special Strings are preceded by the '=' character. Navigate to the **Parameters** tab in the *Document Options* dialog from the **Design** menu to check the available Special Strings. You can set up different special strings for each Schematic Document using the *Document Options* dialog.

The following lists the defined set of special schematic strings:

- =**CurrentTime** – the current time, automatically calculated from your system settings and in the format hh:mm:ss, updated upon editing your schematic or on redraw. For example, this special string is converted to 2:39:47 PM on the schematic.
- =**CurrentDate** – the current date, automatically calculated from your system settings and in the format dd/mm/yyyy, updated upon editing your schematic or on redraw. For example this special string is converted to 24/04/2008 on the schematic.
- =**DocumentFullPathAndName** – the full path and name of the document. For example, this special string is converted to C:\Program Files\Altium Designer 6\Examples\Reference Designs\Multi-Channel Mixer\Mixer.SchDoc on the schematic.
- =**DocumentName** – the schematic's file name only (without the file path). For example, this special string is converted to 4 Port UART and Line Drivers.SchDoc on the schematic.
- =**ModifiedDate** – the modified date stamp of the schematic, automatically populated. For example, this special string is converted to as 2/04/2008 on the schematic.
- =**Time** – user defined field. Enter the value in the *Document Options* dialog
- =**Date** – user defined field. Enter the value in the *Document Options* dialog
- =**ApprovedBy** – user defined field. Enter the value in the *Document Options* dialog
- =**CheckedBy** – user defined field. Enter the value in the *Document Options* dialog
- =**Author** – user defined field. Enter the value in the *Document Options* dialog
- =**CompanyName** – user defined field. Enter the value in the *Document Options* dialog
- =**DrawnBy** – user defined field. Enter the value in the *Document Options* dialog
- =**Engineer** – user defined field. Enter the value in the *Document Options* dialog
- =**Organization** – user defined field. Enter the value in the *Document Options* dialog
- =**Address1** – user defined field. Enter the value in the *Document Options* dialog



- =Address2 – user defined field. Enter the value in the *Document Options* dialog
- =Address3 – user defined field. Enter the value in the *Document Options* dialog
- =Address4 – user defined field. Enter the value in the *Document Options* dialog
- =Title – user defined field. Enter the value in the *Document Options* dialog
- =DocumentNumber – user defined field. Enter the value in the *Document Options* dialog
- =Revision – user defined field. Enter the value in the *Document Options* dialog
- =SheetNumber – the sheet number of the current schematic. This value is calculated when you perform **Number Schematic Sheets** or **Annotate Compiled Sheets** from the Tools menu. To display your Compiled Sheet Number, navigate to **Tools** » **Schematic Preferences** » **Schematic** » **Compiler** and enable the flag to expand **Sheet Number** in the **Compiled Names Expansion** section.
- =SheetTotal – the sheet total for the project. This value is calculated when you perform **Number Schematic Sheets** or **Annotate Compiled Sheets** from the Tools menu. To display your Compiled Sheet Number, navigate to **Tools** » **Schematic Preferences** » **Schematic** » **Compiler** and enable the flag to expand **Sheet Number** in the **Compiled Names Expansion** section.
- =ImagePath – user defined field.
- =VariantName – this is the name of your variant specified in **Project** » **Assembly Variants**. The value of this special string is dependent on the Variant selected when printing or creating a PDF, therefore it is not interpreted until output.
- =[VariantParameterName] – enter the actual name of your Variant Parameter as the special string name. The value of the variant parameter displayed is dependent on the Variant selected. You can specify the same variant parameter name for all of your variants and enter a different value for each. As with VariantName, this special string is not interpreted until output.
- Custom special strings that can be added to the **Parameters** tab of either the *Document Options* or *Project Options* dialog.

### Editing via the SCH Inspector panel

The **SCH Inspector** panel enables you to interrogate and edit the properties of one or more design objects in the active document. Used in conjunction with appropriate filtering, the panel can be used to make changes to multiple objects of the same kind, from one convenient location.

 For more information on the **SCH Inspector** panel, press **F1** when the cursor is over the panel.

### Editing via the SCH List panel

The **SCH List** panel allows you to display design objects from one or more documents in tabular format, enabling you to quickly inspect and modify object attributes. When used in conjunction with the **SCH Filter** panel, it enables you to display just those objects falling under the scope of the active filter - allowing you to target and edit multiple design objects with greater accuracy and efficiency.

 For more information on the **SCH List** and **SCH Inspector** panels, press **F1** when the cursor is over a panel.

### Graphical editing

This method of editing allows you to select a placed text string object directly in the workspace and change its location graphically. Text strings can only be adjusted with respect to their size by changing the size of the font used (accessed through the *Annotation* dialog). As such, editing handles are not available when the text string object is selected:



Click anywhere inside the dashed box and drag to reposition the text string as required. The text string can be rotated or flipped while dragging.

If the **Enable In-Place Editing** option is enabled on the **Schematic - General** page of the *Preferences* dialog (**Tools** » **Schematic Preferences**), you will be able to edit the content for the text string directly in the workspace. Select the text string and then click once to invoke the feature. Type the new text as required and then click away from the text string or press **Enter** to effect the change.

If you attempt to graphically modify a text string object that has its **Locked** property enabled, a dialog will appear asking for confirmation to proceed with the edit.

If the **Protect Locked Objects** option is enabled in the **Schematic – Graphical Editing** page of the *Preferences* dialog (**Tools** » **Schematic Preferences**), and the **Locked** option for this design object is enabled as well then this object cannot be selected or graphically edited.

You will have to double click on this locked object directly and disable the **Locked** property or disable the **Protect Locked Objects** option in the **Schematic – Graphical Editing** Preferences dialog to graphically edit this object.

If you attempt to select locked objects along with other objects, only those objects that are unlocked can be selected and moved as a group if the **Protect Locked Objects** option is enabled. Otherwise if the **Protect Locked Objects** option is disabled you will be prompted for confirmation to proceed with the edit of the objects including locked objects.

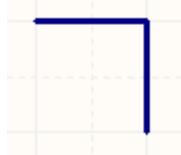
### Notes

A newly placed text string will initially have a default name of **Text**. Edit the properties of the text string to change the text to that required.

---

Any changes made to object properties during placement will cause the default properties for the object to be updated, unless the **Permanent** option - on the **Schematic - Default Primitives** page of the *Preferences* dialog - is enabled. When this option is enabled, changes made will affect only the object being placed and subsequent objects placed during the same placement session.

## Wire



### Description

A wire is an electrical design primitive. It is a polyline object that forms an electrical connection between points on a schematic and is analogous to a physical wire.

### Availability

Wires are available for placement in the Schematic Editor only. Use one of the following methods to access the placement command:

- choose **Place » Wire [P, W]** from the main menus
- click the  button on the **Wiring** toolbar.

### Placement

After launching the command, the cursor will change to a cross-hair and you will enter wire placement mode. Placement is made by performing the following sequence of actions:

- click or press **Enter** to anchor the starting point for the wire
- position the cursor and click or press **Enter** to anchor a series of vertex points that define the shape of the wire
- after placing the final vertex point, right-click or press **Esc** to complete placement of the wire.

Continue placing further wire objects, or right-click or press **Esc** to exit placement mode.

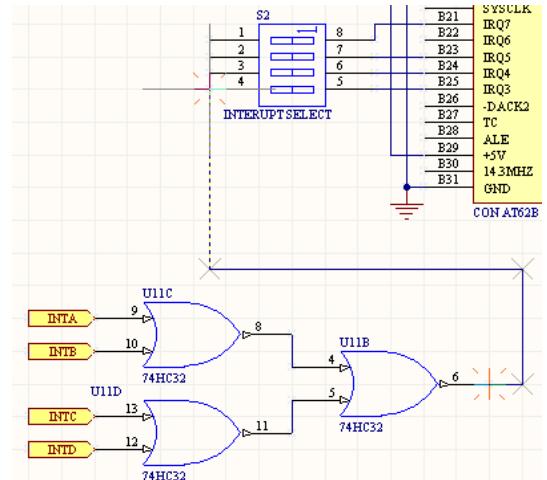
Use the **Backspace** or **Delete** keys to remove the last wire segment placed.

### Guided wiring

Schematics have a definable electrical grid that makes it easy to define electrical connections between objects. As you are placing a wire, when the wire falls within the electrical grid range of another electrical object the cursor will snap to the fixed object and a Hot Spot (red cross) will appear.

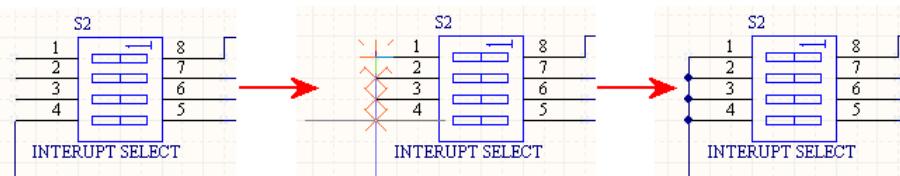
The Hot Spot guides you to where a valid connection can be made and automatically snaps the cursor to electrical connection points.

The electrical grid can be defined on the **Sheet Options** tab of the **Document Options** dialog (**Design » Document Options**). It is recommended that you set the electrical grid to be slightly smaller than the current snap grid, or it becomes difficult to position electrical objects one snap grid apart.



### Auto-junctioning

The schematic auto-junctioning feature places an electrical junction (compiler generated junction) when two wires are connected in a T-type fashion, or when a wire connects orthogonally to a pin or power port.



This feature allows you to easily create electrical connections at junction points without the need to manually define the connection (through placement of a manual junction). Wires that cross away from their end points do not have a junction automatically inserted.

Display of auto-junctions on the schematic sheet, with respect to wires, can be controlled from the **Schematic - Compiler** page of the **Preferences** dialog (**Tools » Schematic Preferences**). Additional options provide control over junction size and color.

## Editing

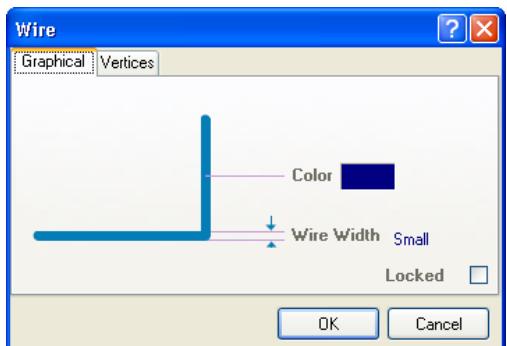
The properties of a wire object can be modified before, during and after placement. Editing itself falls into two categories - graphical and non-graphical.

The following three methods of non-graphical editing are available:

### Editing via an associated properties dialog

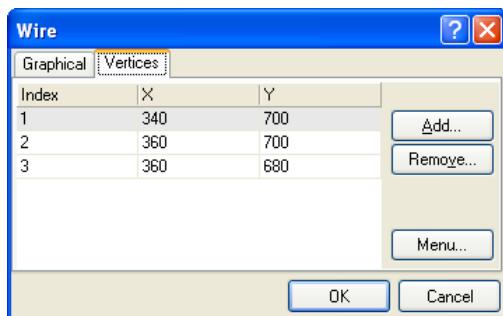
This method of editing uses the following dialog to modify the properties of a wire object.

Use the dialog's 'What's This Help' feature to obtain detailed information about each of the options available. Click on the question mark button at the top right of the dialog and then click over a field or option to pop-up information specific to that field or option.



### Editing Vertices

The **Wire** dialog provides a **Vertices** tab, from where you can edit the individual vertices of the currently selected wire object as required.



The main region of the tab lists all of the vertex points currently defined for the wire. You can add new vertices to the wire, edit the coordinates of existing vertices, or remove selected vertices altogether.

Click the **Menu** button or right-click within the main list region to access a pop-up menu containing the following commands:

- **Edit** - right click on a coordinate cell (X or Y) for a vertex and use this command to edit the value in that cell. Alternatively, click directly on the cell
- **Add** - use this command to add a new vertex point. The new vertex will be added below the currently focused vertex entry (as distinguished by a dotted outline around a cell in its row) and will initially have the same coordinates as the focused entry
- **Remove** - use this command to remove the currently selected vertex entries in the list. This command will be unavailable if there are only two vertices present for the wire
- **Copy** - use this command to copy the content of the selected cells in the list to the clipboard (alternatively use **Ctrl+C**)
- **Paste** - use this command to paste the content of the clipboard into the list, starting at the selected cell (alternatively use **Ctrl+V**)
- **Select All** - use this command to quickly select the entire grid contents of the list
- **Select Column** - use this command to quickly select the entire column in which the currently focused cell resides
- **Move Up** - use this command to move the selected vertex upward in the list
- **Move Down** - use this command to move the selected vertex downward in the list
- **Move Wire By XY** - use this command to move the entire wire object. The Move Wire By dialog will appear, from where you can enter the increment value to be applied to each vertex point's X and Y coordinates.

### Dialog access

The **Wire** dialog can be accessed prior to entering placement mode, from the **Schematic - Default Primitives** page of the **Preferences** dialog (**Tools » Schematic Preferences**). This allows you to change the default properties for the wire object, which will be applied when placing subsequent wires.

During placement, the **Wire** dialog can be accessed by pressing the **Tab** key.

After placement, the **Wire** dialog can be accessed in one of the following ways:

- double-clicking on the placed wire object
- selecting the wire object and choosing **Properties** from the right-click pop-up menu
- choosing the **Change** command from the **Edit** menu and then clicking once over the placed wire object.

### Editing via the SCH Inspector panel

The **SCH Inspector** panel enables you to interrogate and edit the properties of one or more design objects in the active document. Used in conjunction with appropriate filtering, the panel can be used to make changes to multiple objects of the same kind, from one convenient location.

 For more information on the **SCH Inspector** panel, press **F1** when the cursor is over the panel.

### Editing via the SCH List panel

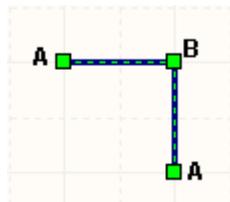
The **SCH List** panel allows you to display design objects from one or more documents in tabular format, enabling you to quickly inspect and modify object attributes. When used in conjunction with the **SCH Filter** panel, it enables you to display just those objects falling under the scope of the active filter - allowing you to target and edit multiple design objects with greater accuracy and efficiency.

 For more information on the **SCH List** and **SCH Inspector** panels, press **F1** when the cursor is over a panel.

### Graphical editing

This method of editing allows you to select a placed wire object directly in the workspace and change its size and/or shape, graphically.

When a wire object is selected, the following editing handles are available:



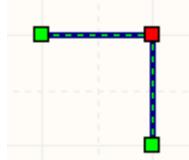
Click and drag **A** to reposition the end points of the wire.

Click and drag **B** to move a vertex. The end points will remain anchored.

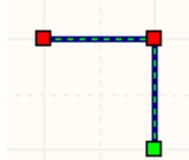
Click and drag near the center of a wire segment to "grab" that segment and reposition it. The end points and other vertices will remain anchored.

Right-click on a vertex point and choose the **Edit Wire Vertex n** command to access the **Vertices** tab of the **Wire** dialog, with the entry for the **n**th vertex selected ready for editing.

With the wire selected, click on a vertex or segment to individually select that vertex or segment. This wire 'sub-selection' is distinguished by the associated editing handles becoming red in color.



Individual vertex sub-selection.



Individual segment sub-selection.

The associated vertex (or vertices for a segment) can then be edited directly using the **SCH Inspector** or **SCH List** panels, with any changes appearing immediately on the schematic.

If you attempt to graphically modify a wire object that has its **Locked** property enabled, a dialog will appear asking for confirmation to proceed with the edit.

If the **Protect Locked Objects** option is enabled in the **Schematic – Graphical Editing** page of the *Preferences* dialog (**Tools** » **Schematic Preferences**), and the **Locked** option for this design object is enabled as well then this object cannot be selected or graphically edited.

You will have to double click on this locked object directly and disable the **Locked** property or disable the **Protect Locked Objects** option in the **Schematic – Graphical Editing** Preferences dialog to graphically edit this object.

If you attempt to select locked objects along with other objects, only those objects that are unlocked can be selected and moved as a group if the **Protect Locked Objects** option is enabled. Otherwise if the **Protect Locked Objects** option is disabled you will be prompted for confirmation to proceed with the edit of the objects including locked objects.

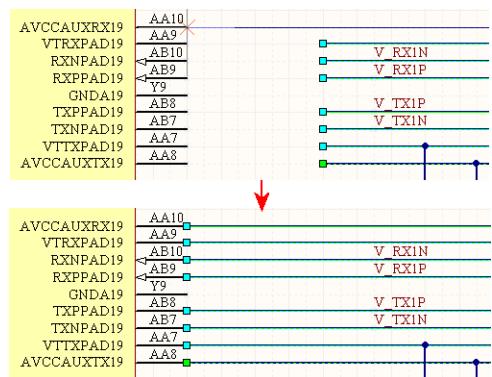
### Multi-wire Editing

If multiple selected wires share a coordinate for their end vertex, then you can drag the vertex of one wire and the vertices of all other wires in the selection will be moved to keep the wire ends aligned.

This becomes an important productivity enhancer for those times when you need to quickly extend a group of wires – for example when connecting to the pins of a placed component.

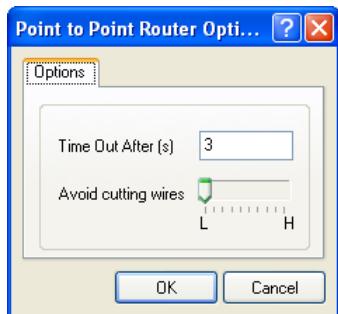
### Notes

A wire is an electrical object and should ONLY be used to make electrical connections between points on a schematic. To draw a non-electrical line on a schematic, use the Line object.



When placing a wire, various placement modes are available: 90 Degree, 45 Degree, Any Angle and Auto Wire. The mode specifies how corners are created when placing wires and the angles at which wires can be placed. The 90 Degree and 45 Degree modes (true orthogonal modes) both have Start and End sub-modes.

The Auto Wire mode is a special mode that allows you to automatically connect between two points on the schematic. The Autowirer will automatically route the bus around obstacles. When in this mode, press the **Tab** key to set the Autowirer options in the subsequent *Point to Point Router Options* dialog:



When placing a wire, press **Shift+Spacebar** to cycle through the wire placement modes. Press **Spacebar** to toggle between the Start and End sub-modes (when in 90 Degree or 45 Degree modes), or between Any Angle and Auto Wire modes (when either of these modes is active). The current placement mode is displayed in the status bar. You can change modes at any time during wire placement.

In all modes other than Any Angle, the wire segment attached to the cursor is a "look ahead" segment. The segment you are actually placing precedes this look ahead segment.

When launching the command to place a wire, the initial placement mode is dependant on the placement mode last used when previously placing wires or lines. Therefore, if the last line you placed used 90 Degree Start, then the next wire or line you place will have 90 Degree Start as the initial placement mode. If you last placed a wire using Auto Wire mode, placement of a subsequent line will be confined to Any Angle mode only. You will need to place a wire in any mode other than Auto Wire in order for line placement to have all placement modes fully restored and functional.

Any changes made to object properties during placement will cause the default properties for the object to be updated, unless the **Permanent** option - on the **Schematic - Default Primitives** page of the *Preferences* dialog - is enabled. When this option is enabled, changes made will affect only the object being placed and subsequent objects placed during the same placement session.

## Revision History

Date	Version No.	Revision
01-Dec-2004	1.0	New product release
07-Mar-2005	1.1	Updates for SP2
01-Apr-2005	1.2	Updated for SP3
09-Jun-2005	1.3	Updated for Altium Designer SP4
30-Sep-2005	1.4	Updated information for Bus, Net Label and Parameter objects.
22-Nov-2005	1.5	Updated for Altium Designer 6
09-Nov-2007	1.6	Updated for Altium Designer 6.8 – Information for Sheet Entries, Schematic Components, locking objects, moving selected objects with arrow keys and new Zoom functionality. New objects dialogs screenshots.
20-Mar-2008	1.7	Updated Page Size to A4.
06-May-2008	1.8	Updated document with info on special strings, netlabels and formatting updates.
06-Aug-2008	1.9	Added sections for C Code Symbol and C Code Entry primitives. Changed formatting slightly to have each primitive start on a new page.
16-Mar-2011	-	Updated template.

Software, hardware, documentation and related materials:

Copyright © 2011 Altium Limited.

All rights reserved. You are permitted to print this document provided that (1) the use of such is for personal use only and will not be copied or posted on any network computer or broadcast in any media, and (2) no modifications of the document is made. Unauthorized duplication, in whole or part, of this document by any means, mechanical or electronic, including translation into another language, except for brief excerpts in published reviews, is prohibited without the express written permission of Altium Limited. Unauthorized duplication of this work may also be prohibited by local statute. Violators may be subject to both criminal and civil penalties, including fines and/or imprisonment.

Altium, Altium Designer, Board Insight, DXP, Innovation Station, LiveDesign, NanoBoard, NanoTalk, OpenBus, P-CAD, SimCode, Situs, TASKING, and Topological Autorouting and their respective logos are trademarks or registered trademarks of Altium Limited or its subsidiaries. All other registered or unregistered trademarks referenced herein are the property of their respective owners and no trademark rights to the same are claimed.