



# Schematic Editing Essentials

## Summary

Application Note

AP0109 (v2.1) November 29, 2005

This application note looks at the placement and editing of schematic objects in Altium.

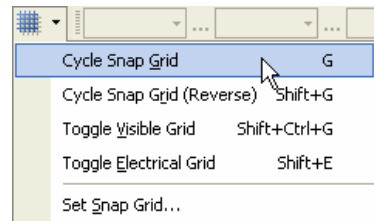
This application note provides a general overview of design object placement and editing methods used in the Schematic Editor. More detailed explanations of placing and editing some of the more complex objects, such as wires and parts, are also included.

## Fundamentals of object placement

### Grids and cursors

Before placing objects in the Schematic Editor, set the grids to enable easier placement. Altium offers three grid types: visible grids for navigation; snap grids for placement and electrical grids for aiding the creation of connections. Grids are document options, meaning that they are saved with the individual design, so grid settings may differ between one design document and the next. Set your grids initially in the *Document Options* dialog (**Design » Document Options**).

Visible grids appear whenever the zoom level will allow them to be sufficiently spaced, displayed as either lines or dots. The Snap grid is the grid that the cursor is locked to when placing or moving schematic design objects. Overriding Snap grids are Electrical grids, which allow connections to be made to off-grid parts. When moving an electrical object in the workspace and it falls within the electrical grid range of another electrical object that you could connect to, it will snap to the fixed object and a hotspot (red cross) will appear. The Electrical grid should be set slightly lower than the current Snap grid or else it becomes difficult to position electrical objects one snap grid apart.



Grids can be quickly modified, or toggled between enabled and disabled, through keyboard or mouse shortcuts, for example, press **G** to cycle through the Snap grid settings of 1, 5 and 10. You can also use the **View » Grids** submenu or the **Grids** right-click menu. Use the **Schematic – Grids** page of the *Preferences* dialog (**Tools » Schematic Preferences** or shortcut **T, P**) to set Imperial and Metric grid presets.

You can also change the Cursor type to suit your needs in the Cursor section of the **Schematic – Graphical Editing** page of the *Preferences* dialog. For example, a large 90 degree cross that extends

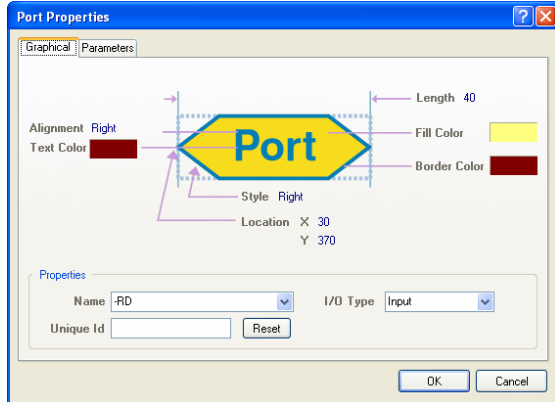
## Schematic Editing Essentials

to the edges of the design window (Large Cursor 90 option) can be handy when placing and aligning design objects.

### Placing design objects

The basic steps for placing schematic design objects are outlined below.

1. Select the object type that you want to place. You can do this by selecting an object type from the Place menu (e.g. **Place** » **Wire**) or by clicking on one of icons on the placement toolbars. Shortcut keys for placement are also available (e.g. **P**, **W** to place a wire). To place components (parts), you can also click the **Place** button in the Library panel, or select the component name from an available library in the Library panel and drag it into the document.
2. When an object is selected for placement, the cursor will change to a crosshair, indicating that you are in editing mode, and, if relevant, the object will appear "floating" on the cursor.
3. Press the **TAB** key to edit the properties of the object before placing it. This will open the *Properties* dialog for the particular object, allowing you to change various options.



Once you have finished setting the properties, click **OK** to return to placement mode.

The advantages of editing during placement are that objects that have a numeric identifier, such as a designator, will auto-increment. Also, changes made during placement can become the defaults for that type of 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 (**T**, **P**) is enabled.

4. Position the cursor and left-click, or press **ENTER**, to place the object. For complex objects, such as wires or polygons, you must continue the position-and-click procedure to place all vertices of the object.

Note: If autopanning is active, you can move around the document by simply moving the cursor past the edge of the editing window in the direction that you wish to go. You can set the style and speed of autopanning in the **Schematic – Graphical Editing** page of the *Preferences* dialog.

The AutoFocus options in the **Schematic – AutoFocus** page of the *Preferences* dialog control the state of the schematic display, e.g. it can be configured to automatically zoom in when placing or editing connected objects or dim all wiring not related to the wire currently being placed.

Other zooming and panning options are available using the shortcut keys or mouse wheel. Use **Ctrl+wheel mouse** to zoom in and out when placing. You can setup the behaviour of your mouse in the **Schematic – Mouse Configuration** page of the *Preferences* dialog.

5. After placing an object you will remain in placement mode (indicated by the crosshair cursor), allowing you to place another object of the same type immediately.
6. To end placement mode, right-click or press the **ESC** key. In some cases, such as placing a polygon, you may need to do this twice; once to finish placing the object and once to exit placement mode. When you exit placement mode, the cursor will return to its default shape.



For more information about specific design objects press **F1** when the cursor is over an object in the Schematic Editor, information about the object will appear in the **Knowledge Center** panel.

## Re-entrant editing

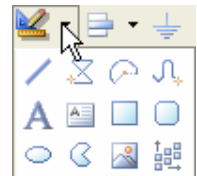
The Schematic Editor includes a powerful feature called re-entrant editing which allows you to perform a second operation using the keyboard shortcuts without having to quit from the operation you are currently carrying out. For example, pressing the **SPACEBAR** when placing a part will rotate the object but will not disrupt the placement process. Once you place the part, another part will appear ready upon your cursor, already rotated.

Another example of when re-entrant editing is very useful is if you start placing a wire that it needs to be connected to a port which you have not placed yet. There is no need to exit Place Wire mode; just press the Place Port shortcut keys (**P, R**), place the port, press **Esc** to exit Place Port mode and then connect the wire to the port.

## Placing graphical objects

Schematic objects are divided into two groups: graphical and electrical.

To place graphical objects, such as lines, arcs and text, use the Drawing Tools toolbar (available from the Utilities toolbar (**View » Toolbars » Utilities**)). Drawing toolbar functions can also be accessed through the **Place » Drawing Tools** menu, except for Paste Array (**Edit » Paste Array**).




## Placing electrical objects

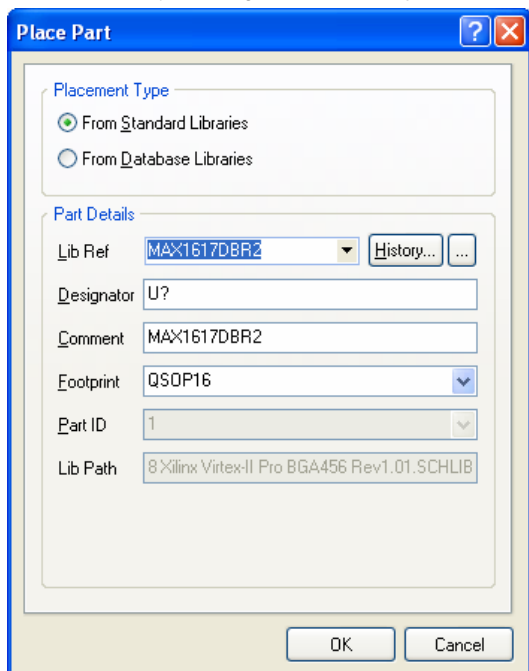
Schematic electrical design objects define the physical circuit you are capturing. Electrical objects include components (parts) and connective elements, such as wires, buses and ports. Use the Place menu or the Wiring toolbar (**View » Toolbars » Wiring**) to place electrical objects.



The following sections detail the placement of two commonly used object types – parts (components) and wires.

### Placing Parts


When **Place » Part (PP)** is selected, or you click on  in the Wiring toolbar, the *Place Part* dialog is displayed. You can enter the name of the component in the Lib Ref field or you can click on the Browse button (...) to locate the part by searching for and adding the required library. You can find previously placed parts by clicking on the **History** button.

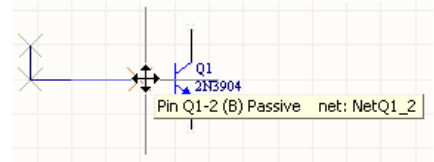


Parts can also be placed using the **Place** button in the Library panel or the Schematic Library Editor. Alternatively, select a component name in the **Libraries** panel and drag it into the document where it will appear floating on the cursor ready for placement. Click to place.

When placing parts, use a snap grid that will cause the pin ends to fall on a grid point, e.g. 10. Remember that you can press **G** to cycle through the Snap grid settings of 1, 5 and 10.

## Placing Wires

Wires are used to represent an electrical connection between points. When placing wires, be careful to use the **Place » Wire** command  and not use the Line command by mistake. The Place Wire command is also available from the right-click menu when you are in a schematic document or the **Wiring** toolbar.



A wire end must fall on the connection point of an electrical object to be connected to it, e.g. the end of a wire must fall on the hot end of a pin to connect. 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. This hot spot guides you to where a valid connection can be made and automatically snaps the cursor to electrical connection points. The wire will be automatically terminated when it finishes on a hot spot.

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.

If you wish to place a wire that does not yet connect to another electrical object, right-click (or press **ESC**) to terminate the wire. Right-click, or press **ESC**, to exit wire placement mode.

While placing wires, use the **BACKSPACE** key to delete the last vertex placed.

## Wire placement modes

When placing a wire, press **SHIFT+SPACEBAR** to cycle through the wire placement modes. Various placement modes are available:

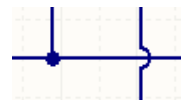
- 90 Degree
- 45 Degree
- Any Angle
- Auto Wire.

The mode specifies how corners are created when placing wires and the angles at which wires can be placed. 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 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 wire around obstacles. When in this mode, press the **TAB** key to set the Autowirer options in the *Point to Point Router Options* dialog.

## Junctions

Wires have the Auto Junction feature, which automatically inserts a Junction object (dot) if a wire starts or ends on another wire or runs across a pin. The display, junction size and color of auto-junctions and manual junctions can be controlled from the **Schematic - Compiler** page of the **Preferences** dialog. You can also set wire cross-overs and convert existing wires to cross-overs using options in the **Schematic - General** page of this dialog.



See [Editing placed wires](#), later in this application note, for tips on how to modify a placed wire.

## Editing schematic design objects


---

Any design objects you have placed in a schematic can be modified in a variety of ways. Objects can be moved around within the document, and cut, copied and pasted within and between documents. Also, you can edit the properties of objects to change their colors, designator or net assignment, etc. With some objects, such as polyline shapes (for example buses, wires, polygons and lines), you can graphically change the shape of the object after it has been placed.

You can change just one object, or extend changes across your entire design using powerful editing options, such as queries. Several other useful tools will help you work with groups of objects at a time, including the *Find Similar Objects* dialog, the List panel and the Inspector panel.

 See the [Editing Multiple Objects](#) tutorial for more information.

Placed components (parts) and footprints can be edited through their properties' dialogs or changed in their Library Editors and updated. Pins can be edited by using the Component Pin Editor, available from the schematic *Component Properties* dialog (**Edit Pins** button).

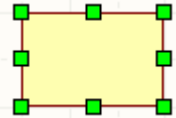
 Refer to the [Creating Library Components](#) tutorial for more information about creating or editing parts and footprints using the Library Editors.

## Graphically editing placed objects

---

To edit the look of an object in the workspace it is generally easier to do it graphically. To do this, you must first select the object(s).

When an object is selected, you can move the object, or edit its graphical characteristics. Click on an object to select it and its 'handles' or vertices are displayed. To graphically change a selected object, click and hold on an editing handle. That point of the object will then become attached to the cursor, so simply move the mouse to a new location and release to resize. Click anywhere on a selected object to move it, or press the **Delete** key to delete it.

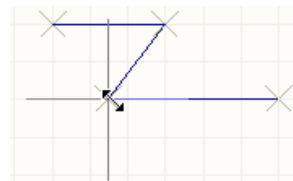


## Editing placed wires

There are several ways you can change a placed wire — by moving a vertex, moving a segment, moving the entire wire or extending the wire to a new location. You can also edit, add or remove vertices through the **Vertices** tab in the wire's *Properties* dialog. See the [Editing vertices from the Properties dialog](#) section for more information.

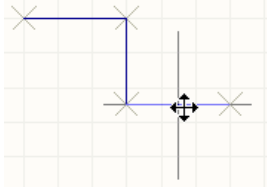
### Editing by moving a wire's vertex

To edit a selected wire by moving a vertex, click on the wire to select it. Position the cursor on the vertex you want to move until the cursor changes to a double arrow. Then click and drag the vertex to its new location, as shown below.



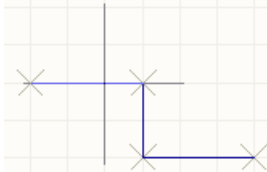
## Moving a wire segment

You can edit a selected wire by moving a segment of the wire. Select the wire and move the cursor over the segment until it changes to a 'quad' arrow. Then click and drag the segment to its new location, as shown below.



## Moving an entire wire

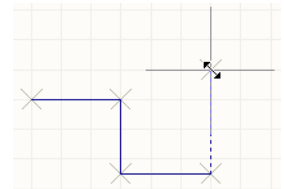
To move the entire wire without modifying it, you do not have to select it first — simply click and drag.



## Extending a wire to a new location

Placed wires can be extended, or added to. Select the wire and position the cursor on the end you want to move until the cursor changes to a double arrow. Click on the endpoint and drag the 'look ahead' segment to the new location and click to place. You can change the placement mode while moving the cursor to the new location by using **SHIFT + SPACEBAR**.

To make a wire longer by extending it in the same direction, hold down the **ALT** key as you drag the selected endpoint.



## Breaking wires

Use the **Edit » Break Wire** command to break the wire segment into two pieces.

This command is also available from the right-click menu when the cursor is over a wire. A 'cutter box' displays (by default) which is placed on the wire to be cut. The section to be cut out is masked as shown. The cutting length is the space between the two new segments. Press **SPACEBAR** to cycle through the three cutting length modes (snap to segment, snap grid size multiple or fixed length). Press **TAB** to set the fixed cutting length and any other cutter options. Click to break the wire. Right-click, or press **ESC**, to exit break wire mode. Break Wire options can also be set in the **Schematic – Break Wire** page of the *Preferences* dialog.



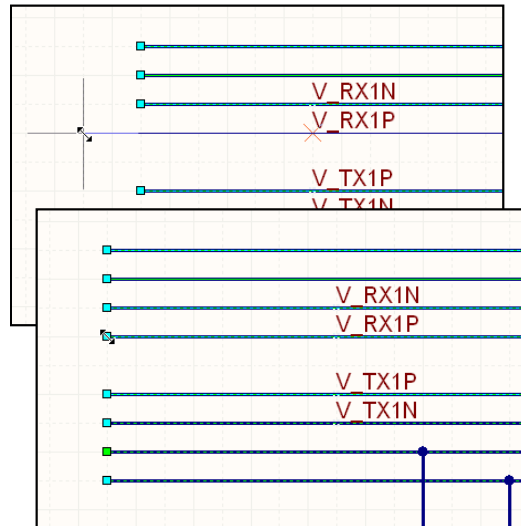
Note that you can also set the **Components Cut Wires** option in the **Schematic – General** page of the *Preferences* dialog. When this option and the **Optimize Wires and Buses** option are enabled, you can place a component onto a wire and the wire is cut into two segments which are automatically terminated onto any two 'hot' pins in the component.

### Multi-wire

Do you find that you often need to extend a group of wires? The multi-wire editing mode in the schematic editor supports this. If multiple parallel wires share a coordinate for their end vertex, then when you click and drag to move the end of one wire vertex the end vertex of all other selected wires will also move, keeping the wire ends aligned.

### Moving and dragging schematic objects

To move an object in Altium is to reposition it without regard to objects in contact with it. Moving a component, for example, will not move any of the wires connected to it. Dragging the component, on the other hand, stretches the wires out after it, maintaining connectivity. If you wish electrical wiring to always stay connected when moving selected objects, enable the **Always Drag** option in the **Schematic – Graphical Editing** page of the *Preferences* dialog.



*Drag and release one vertex, and all aligned vertices will move to remain aligned.*

### Moving objects

You can move single, non-selected objects or multiple, selected objects, using the mouse to click-and-drag. You could also use the **Edit » Move** commands, especially if you wish to move the objects to the front or back of other placed objects.

You do not need to select objects first to move them around the sheet. Just click and hold until the electrical hotspot markers appear and drag the component/object to the new location. The cursor will move to the nearest electrical hotspot, e.g. the pin closest to the cursor position when you first clicked on the object. The **Object's Electrical Hot Spot** option (default) or **Center of Object** option in the **Schematic – Graphical Editing** page of the *Preferences* dialog will determine where the object is held by during moving or dragging.

While objects are being moved:

- press **SPACEBAR** to rotate the object. Rotation is anti-clockwise and in steps of 90°. Press **SHIFT+SPACEBAR** to rotate the object clockwise.
- press the **X** or **Y** keys to flip the object along the X-axis or Y-axis respectively.
- hold down the **ALT** key to constrain the direction of movement to the horizontal or vertical axis.

### Dragging objects

The **Edit » Move » Drag** command enables you to move any object, e.g. a component (part), port, wire or bus, and any connected wires are moved with the object being dragged so that the connectivity of the schematic is maintained. The cursor changes to a crosshair, which you position over an object you



want to drag, and click, or press **ENTER**. Move the object to the required position and click, or press **ENTER**, to place. Continue moving other objects, or right-click or press **ESC** to exit dragging mode.

To drag multiple selected objects and maintain connectivity, use the **Edit » Move » Drag Selection** command.

Alternatively, you can drag objects using a shortcut key. Just hold the **CTRL** key as you click, hold and move the mouse. You can then release the **CTRL** key once you start dragging. This also works with multiple, selected objects. Note that using the **CTRL** key temporarily toggles the setting of the **Always Drag** option set in the *Preferences* dialog.

While dragging the object:

- press the **SPACEBAR** or **SHIFT+SPACEBAR** while dragging to change the wiring mode.
- press **CTRL+SPACEBAR** while in movement mode to rotate the object. Rotation is anti-clockwise and in steps of 90°.
- press the **X** or **Y** keys while in movement mode to flip the object along the X-axis or Y-axis respectively.
- press **SPACEBAR** while moving to toggle the orthogonal wire mode for any wires connected to the object.
- press the **ALT** key while moving, to constrain the direction of movement to the horizontal or vertical axis, depending on the initial direction of movement.

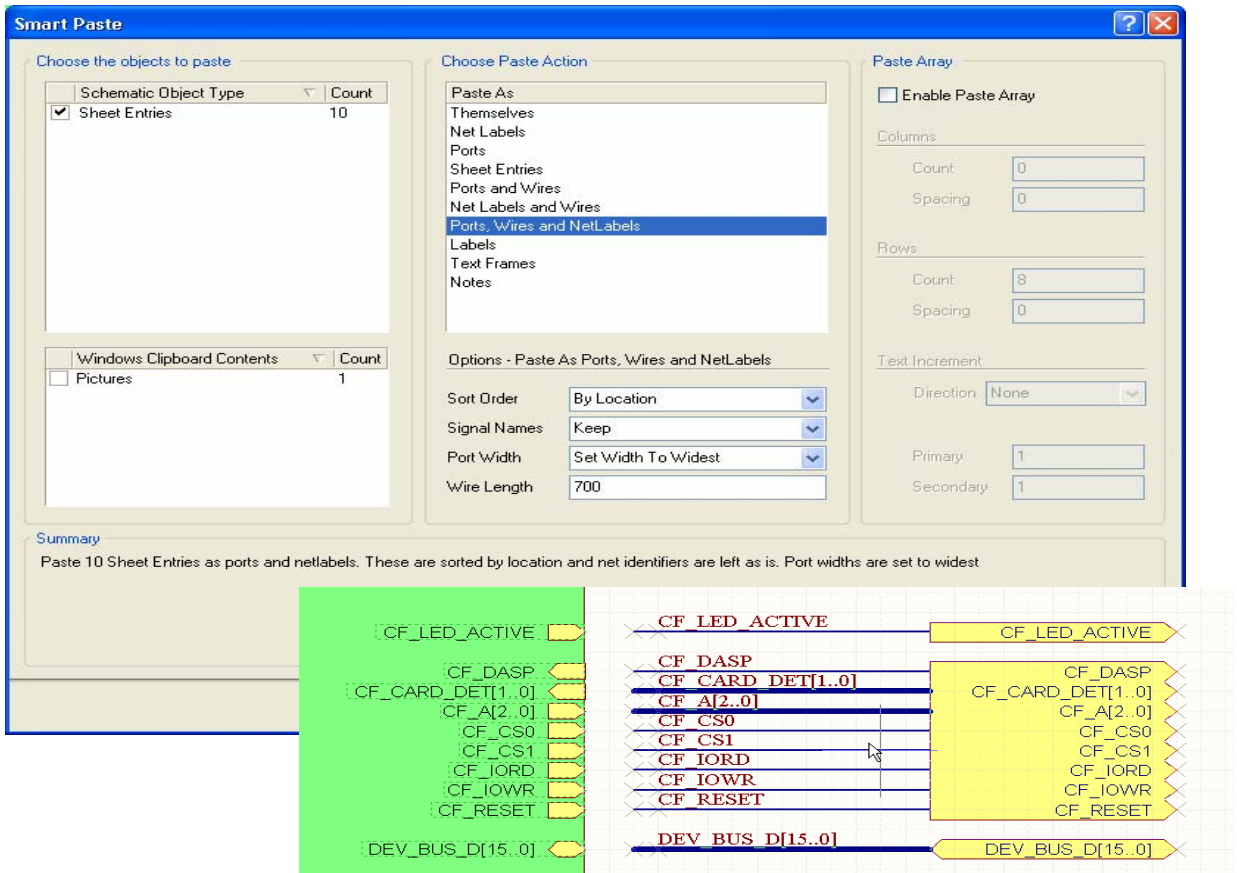
## Using Copy and Paste

In the Schematic Editor, you can copy and paste objects within or between schematic documents, e.g. component(s) from a schematic can be copied into another schematic document. You can copy objects to the Windows clipboard and paste them into other documents. Text can be copied from the Windows clipboard into a schematic text frame. You can also directly copy/paste a table-type selection from another application such as Microsoft Excel, or from any grid style control within Altium Designer. More advanced copy/paste actions can be performed using the Smart Paste feature.

Select the object(s) you want to copy, select **Edit » Copy (Ctrl+C)** and click to set a copy reference point on the object which will be used to accurately position the object during pasting. Note that you will only be prompted to click to set a reference point if the **Clipboard Reference** option is enabled in the **Schematic – Graphical Editing** page of the *Preferences* dialog.

## Using Smart Paste

The Schematic Editor's Smart Paste feature allows you to *transform* the copy of the selected objects into other objects as you paste them. For example you could copy a selection of Net Labels, and Smart Paste them as Ports, or the selected Sheet Entries could be pasted as Ports+Wires+Net Labels, all in a single paste action.



Sheet Entries copied and then transformed into Wires+Ports+Net Labels as they are Smart Pasted.

You also have complete control over which of the objects in the selection set you want to paste, simply clear the checkbox to ignore specific object kinds when you Smart Paste your selection.

Another handy feature is the ability to paste the selected circuitry as a graphic. Using this you can easily include a graphic of section of circuit on another sheet, and size it as required. Select **Edit » Smart Paste** to transform the clipboard objects as you paste them.

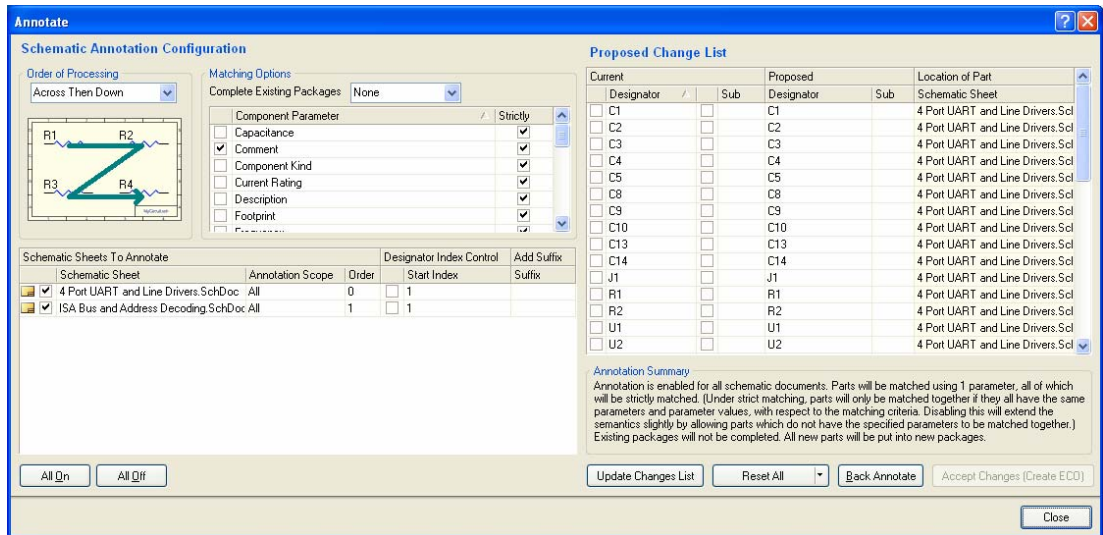
### On-sheet text editing

Text strings can be edited directly on the schematic sheet. Click once to select a text string, text frame or note, then click a second time to edit the string directly on the schematic sheet (or press the **F2** shortcut key). This behavior can be disabled by turning off the **Enable In-place Editing** option in the **Schematic – General** page of the *Preferences* dialog.

## Annotation and Re-annotation

Both the Schematic Editor and the PCB Editor offer a positional annotation option. The schematic annotation feature allows you to match components by parameters and offers designator index control and suffix options for schematic sheets.

Use the **Tools » Annotate** command to open the *Annotate* dialog, from where you can reassign the designators for all parts in selected sheets of the current project, so that they are unique and in consecutive order.



Press **F1** when the *Annotate* dialog is open for more information about this process.

You can also use the **Tools » Annotate Quiet** command to assign a unique designator to any component that currently does not have a designator, without launching the *Annotate* dialog.

If necessary, you can use the **Tools » Reset Designators** command to reset all component designators in the active project, or reset duplicate designators (**Tools » Reset Duplicates**).

To re-annotate all component designators in accordance with the positional annotation scheme defined in the *Annotate* dialog, use the **Tools » Force Annotate All** command.

Back Annotating (**Tools » Back Annotate**) will update the designators of the components in the source schematic sheets of the active project, using a WAS-IS file generated when re-annotating designators in the PCB document.

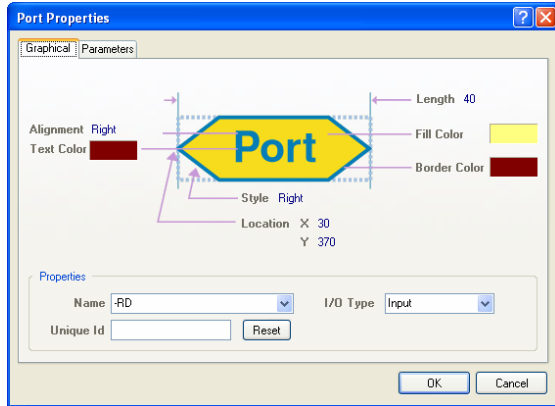
## Editing an object's properties

To view or edit the properties of any object, open its associated *Properties* dialog in one of the following ways:

- when an object is 'floating' on the cursor during the placement procedure, press the **TAB** key to open its *Properties* dialog.
- double-click on a placed object to open its properties dialog directly.

## Schematic Editing Essentials

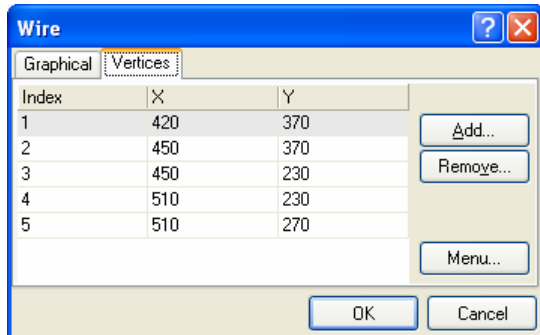
- select **Edit » Change** from the menus to enter the object change mode. Click on the object that you want to edit. Right-click or press **ESC** to exit object change mode.



- click on a placed object to select it, then edit its properties in the Inspector or List panel, as described below.

## Editing vertices from the Properties dialog

You can edit the co-ordinates of vertices for bus, wire, polyline and polygon objects via the **Vertices** tab of their *Properties* dialog. For example, the *Wire* properties dialog has 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 from where you can edit, add or remove vertices or copy, paste, select or move cells. The **Move Wire By XY** command will 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.

## Editing objects in the Inspector panel

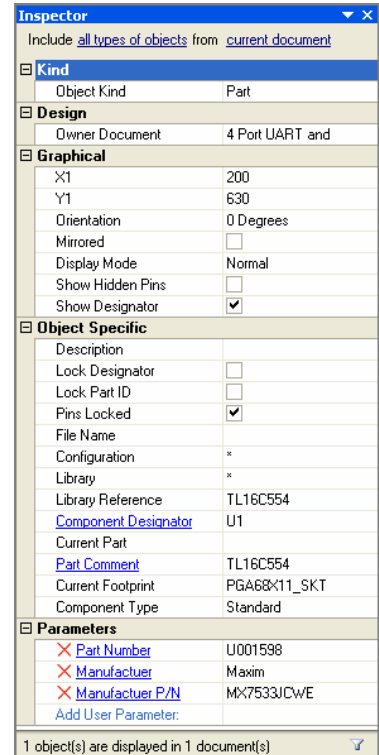
The Inspector panel enables you to interrogate and edit the properties of one or more design objects in the current or open document(s). When used in conjunction with the **Filter** panel (F12) or the Find Similar Objects command (**Shift+F**, or right-click on a selected object and choose **Find Similar Objects**), you can make changes to multiple objects of the same kind.

Select one or more objects and press **F11** to display the Inspector panel, or click on the **Inspector** tab. If the panel is not visible, click on the **SCH** button in the Status bar and select **Inspector**, or select **View » Workspace Panels » SCH » Inspector**. You can also set the **Double Click Runs Inspector** option in the **Schematic – Graphical Editing** page of the *Preferences* dialog to bring up the Inspector panel instead of the object's *Properties* dialog when you double-click on a design object.

Note that the Inspector panel will only show properties that are common to all selected objects. The attributes listed can be modified in the Inspector panel. Type in the new attribute, click on a check box or select from a drop-down list, as applicable. Press **Enter** or click on another cell in the panel to implement the change.

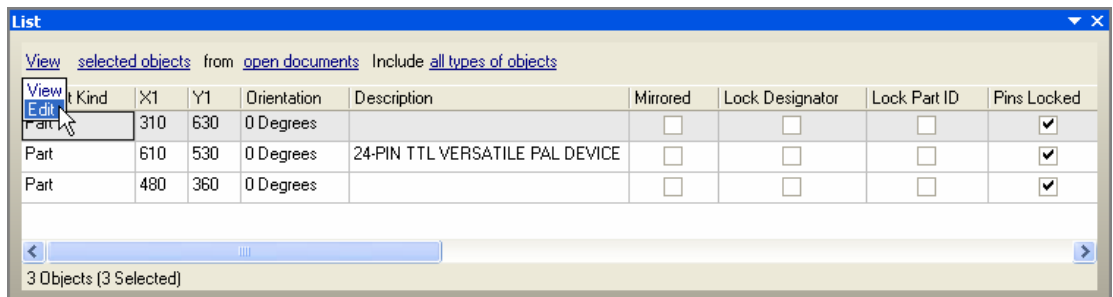


For more information about editing multiple objects, see the [Editing Multiple Objects](#) tutorial.



## Editing objects in the List panel

Select an object (or objects) and press **Shift+F12** to display the List panel. When used in conjunction with the **Filter** panel (F12) or the **Find Similar Objects** command (**Shift+F**), you can target and edit multiple design objects. From the List panel you can change the properties of the object by selecting **Edit** from the View/Edit drop-down menu at the top of the panel. To change the location of a part, for example, select the X or Y coordinates and type in the new position. Press **Enter** to set the change.



Double-click on the object in the Object Kind list of the List panel to display its *Properties* dialog.

## Revision History

---

Date	Version No.	Revision
05-Jan-2004	1.0	New product release
24-Mar-2005	2.0	Renamed from <i>Object Placement and Editing</i> . Concentrates on Schematic Editor objects only in Altium SP3.
29-Nov-2005	2.1	Updated for Altium Designer 6

Software, hardware, documentation and related materials:

Copyright © 2005 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 , CAMtastic, Design Explorer, DXP, LiveDesign, NanoBoard, Nexar, nVisage, P-CAD, Protel, 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.