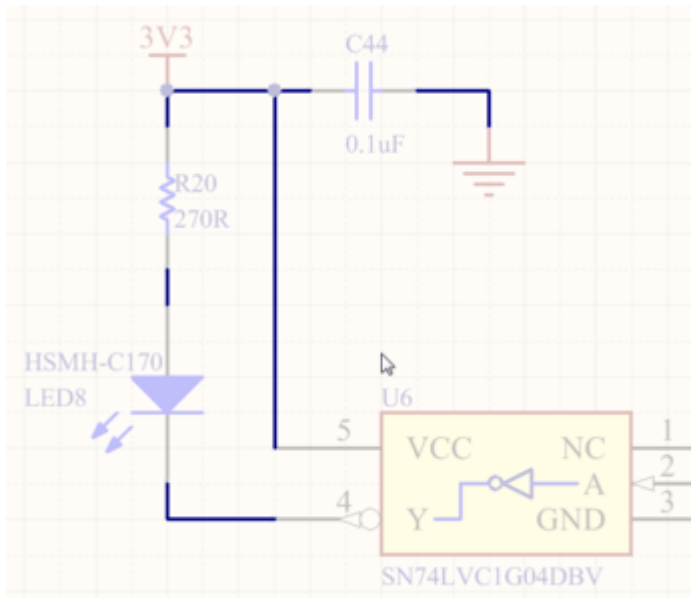


Wire

Modified by Jason Howie on 8-Sep-2014

Parent page: [Schematic Objects](#)




Wires are used to create electrical connectivity in a schematic.

Summary

A wire is a polyline electrical design primitive that is used to form electrical connections between points on a schematic. It is analogous to a physical wire.

Availability

Wires are only available for placement in the Schematic Editor, by clicking **Home | Circuit Elements** |  **Wire** from the main menus.

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:

1. Click or press **Enter** to anchor the starting point for the wire.
2. Position the cursor and click or press **Enter** to anchor a series of vertex points that define the shape of the wire.
3. After placing the final vertex point, right-click or press **Esc** to complete placement of the wire.
4. Continue placing further wire objects, or right-click or press **Esc** to exit placement mode.

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

While attributes can be modified during placement (**Tab** to bring up associated properties dialog), bear in mind that these will become the default settings for further placement.

Placement Modes

When placing a wire there are 3 placement modes, 2 of which have Start and End sub-modes. The mode specifies how corners are created when placing wires and the angles at which wires can be placed. During placement:

- Press **Shift+Spacebar** to cycle through the 90 Degree, 45 Degree and Any Angle modes.
- While in the 90 Degree or 45 Degree mode (known as true orthogonal modes), press **Spacebar** to cycle between the Start and End sub-modes.
- During placement, the current placement mode is displayed in the Status bar. You can change modes at any time during wire placement.
- In 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.



45 degree mode



90 degree mode



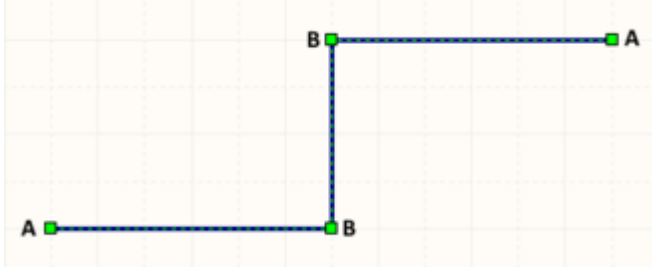
Any angle mode

Press **Shift+Spacebar** to cycle through the different placement modes.

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.

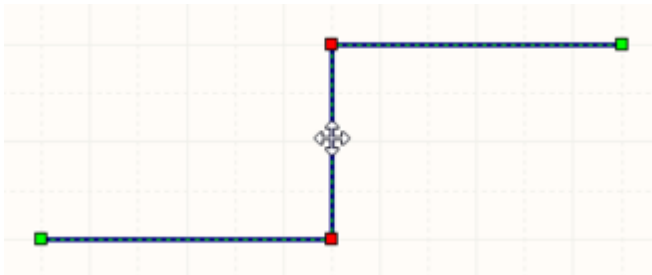
When a wire object is selected, the following editing handles are available:



Selected Wire, ready for graphical editing.

- Click and drag **A** to reposition the end points of the wire.
- Click and drag **B** to move a wire vertex. The end points will remain anchored.
- Click and drag on 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 nth vertex selected ready for editing.
- Click and hold on a wire segment, then press **Insert** on the keyboard to add a vertex at that point.
- Click and hold on a vertex, then press **Delete** on the keyboard to remove that vertex.

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



Individual segment sub-selection.

The associated vertices for the segment can then be edited directly using the *SCH Inspector* panel, with any changes appearing immediately on the schematic.



To move an entire wire, click and hold on the un-selected wire, then move to the new location.

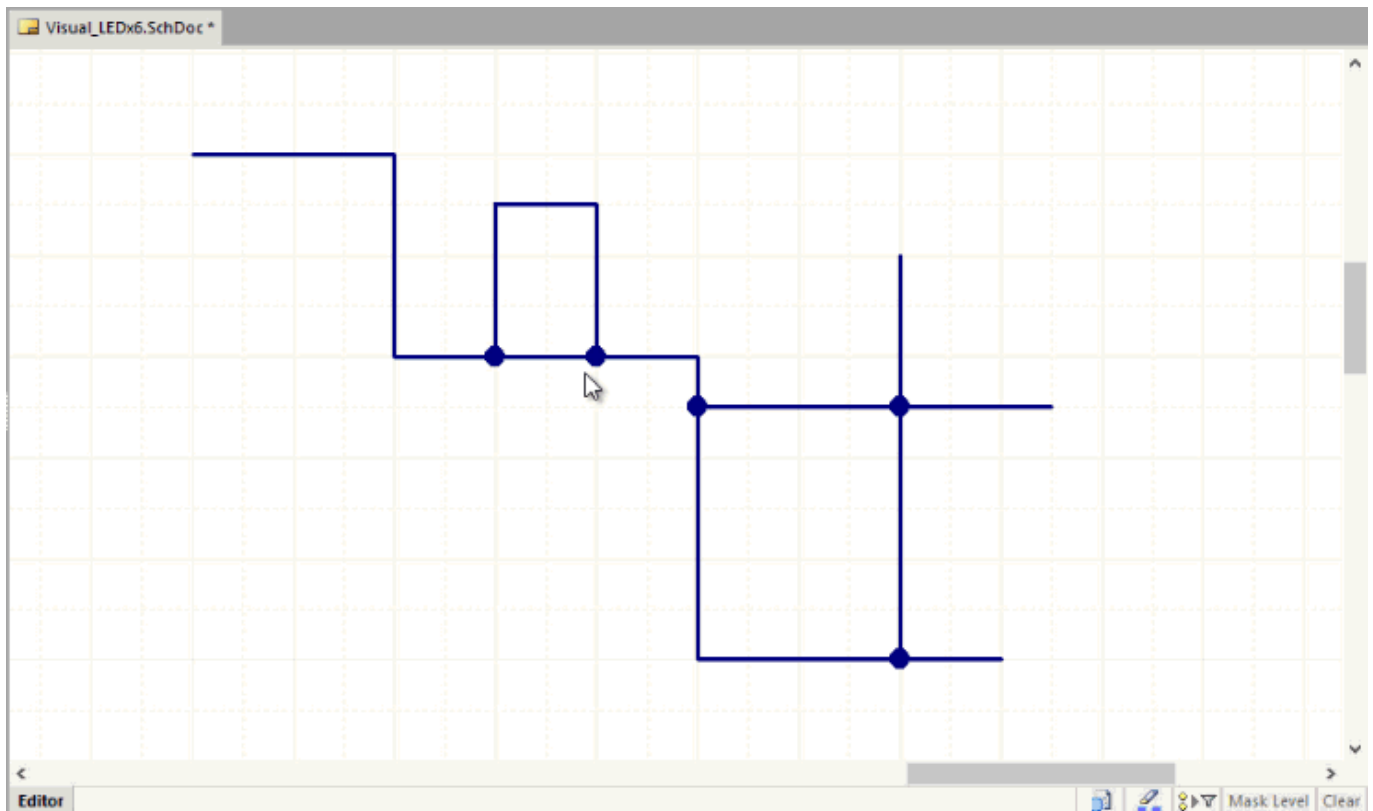
An object that has its **Locked** property enabled cannot be selected or graphically edited. Double click on the locked object directly and disable the **Locked** property, to graphically edit the object.

Deleting a Wire Segment

You can remove a wire segment, graphically, simply by selecting that segment, and pressing the **Delete** key. Auto-junctions are also accounted for - allowing you to remove a segment of a wire up to a specific junction only (and including that junction if only two other wire segments would otherwise remain connected to it).

Considering a T-junction, which is formed of three wire segments and a junction, removal of one wire segment will result in the removal of the junction. The remaining two wire segments will simply be merged to form a single segment.

Remember to click twice (with a pause in between) on a particular segment of wire to select it, denoted by its end-point editing handles turning red. You can delete multiple segments across different wires, ensure that each is selected (**Shift**+click twice on each subsequent segment to include it in the overall segment selection).



Delete selected track segments, including attached autojunctions where applicable, with a press of the **Delete** key.



Where multiple different objects are currently selected on a schematic, the first use of the **Delete** key will delete all selected wire segments (indicated by their red editing handles). Subsequent use of the **Delete** key will then cause all other objects in the original selection to be deleted.



A wire segment can also be removed through use of the Break Wire feature, with the **Cutting Length** option set to Snap To Segment.

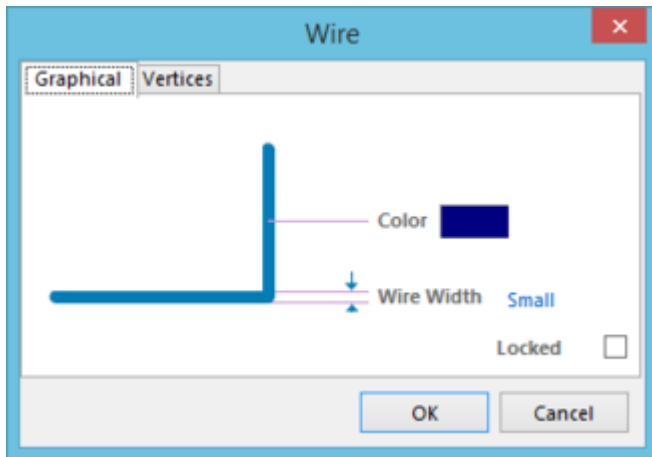
Non-Graphical Editing

The following methods of non-graphical editing are available:

Via an Associated Properties Dialog

Dialog page: [Wire](#)

This method of editing uses the *Wire* dialog to modify the properties of a Wire object.



The *Wire* dialog.

The dialog can be accessed during placement by pressing the **Tab** key.

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

- Double-clicking on the placed Wire object.
- Placing the cursor over the Wire object, right-clicking and choosing **Properties** from the context menu.



The *Wire* dialog includes a **Vertices** tab, where you can edit the individual vertices of the currently selected wire object.

Via the SCH Inspector Panel

Panel page: [SCH Inspector](#)

The *SCH Inspector* panel enables the designer to interrogate and edit the properties of one or more design objects in the active document. Used in conjunction with the [Find Similar Objects dialog](#), the panel can be used to make changes to multiple objects of the same kind, from one convenient location.

Source URL: [http://documentation.circuitmaker.com/display/CMAK/Sch_Obj-Wire\(\(Wire\)\)_CM](http://documentation.circuitmaker.com/display/CMAK/Sch_Obj-Wire((Wire))_CM)