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 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).
- using the panning hand (right-click and hold to access the panning hand, then add left mouse button or Ctrl to temporarily zoom by pushing and pulling with the mouse).

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:

- Blue
- Light Green
- Cyan
- Red
- Magenta
- Yellow
- Dark Green

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.
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.

**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.

**See Also**

[Sch Editor](#)

[PCB Editor](#)

**Source URL:** https://techdocs.altium.com/display/ADRR1/Getting+to+know+your+Editor