1. [General Altium Environment Shortcuts](https://www.altium.com/documentation/altium-designer/shortcut-keys)

|  |  |  |
| --- | --- | --- |
| **F1** | Access technical documentation for the resource currently under the cursor in particular commands, dialogs, panels, and objects | [Resource Reference](https://www.altium.com/documentation/altium-designer/commands-reference-ad) |
| **Ctrl+O** | Open any existing document using the *Open* dialog | [Command Page](https://www.altium.com/documentation/altium-designer/workspacemanager-cmd-openobjectopenobject-ad#open_anything) |
| **Ctrl+F4** | Close active document | [Command Page](https://www.altium.com/documentation/altium-designer/workspacemanager-cmd-closeobjectcloseobject-ad#close_active_document) |
| **Ctrl+S** | Save active document | [Command Page](https://www.altium.com/documentation/altium-designer/workspacemanager-cmd-saveobjectsaveobject-ad#save_active_document) |
| **Ctrl+Alt+S** | Save and release the defined entity | [Command Page](https://www.altium.com/documentation/altium-designer/workspacemanager-cmd-filelessreleasetovaultfilelessreleasetovault-ad) |
| **Ctrl+P** | Print active document | [Command Page](https://www.altium.com/documentation/altium-designer/workspacemanager-cmd-printprint-ad#print_active_document) |
| **Alt+F4** | Exit Altium Designer | [Command Page](https://www.altium.com/documentation/altium-designer/client-cmd-quitfromedaclientquitfromedaclient-ad#exit) |
| **Ctrl+Tab** | Cycle forward to the next open tabbed document, making it the active document in the design workspace | [Command Page](https://www.altium.com/documentation/altium-designer/client-cmd-switchviewsswitchviews-ad#cycle_to_next_open_tabbed_document) |
| **Shift+Ctrl+Tab** | Cycle backward to the previous open tabbed document, making it the active document in the design workspace | [Command Page](https://www.altium.com/documentation/altium-designer/client-cmd-switchviewsswitchviews-ad#cycle_to_previous_open_tabbed_document) |
| **F4** | Toggle the display of all floating panels | [Command Page](https://www.altium.com/documentation/altium-designer/client-cmd-togglefloatingbarstogglefloatingbars-ad#toggle_floating_panels_visibility) |
| **Shift+F4** | Tile all open documents | [Command Page](https://www.altium.com/documentation/altium-designer/client-cmd-tileallopendocumentstileallopendocuments-ad#tile_all_open_documents) |
| **Shift+F5** | Toggle the focus between the last active panel and the currently active design document in the main design window | [Command Page](https://www.altium.com/documentation/altium-designer/client-cmd-switchdocumentandpanelswitchdocumentandpanel-ad#switch_focus_between_document_and_panel) |
| **Alt+Right Arrow** | Step forward to the next document in the sequence of documents that have been made active in the main design window | [Command Page](https://www.altium.com/documentation/altium-designer/client-cmd-navigatenavigate-ad#step_forward_to_next_document) |
| **Alt+Left Arrow** | Step back to the previous document, in the sequence of documents that have been made active in the main design window | [Command Page](https://www.altium.com/documentation/altium-designer/client-cmd-navigatenavigate-ad#step_back_to_previous_document) |
| **F5** | Refresh the active document when that document is a web-based document | [Command Page](https://www.altium.com/documentation/altium-designer/client-cmd-navigatenavigate-ad#refresh_current_page) |
| Hold **Ctrl**while moving a panel | Prevent automatic docking, grouping, or snapping | - |
| Drag and drop from Windows Explorer into Altium Designer | Open a document, project, or design workspace | - |
| **Shift+Ctrl+F3** | Move to the next message (down) in the [Messages panel](https://www.altium.com/documentation/altium-designer/messages-panel) and cross-probe to the object responsible for the message in the associated document (where supported) | [Command Page](https://www.altium.com/documentation/altium-designer/workspacemanager-cmd-configureconfigure-ad#messages_cross_probe_next) |
| **Shift+Ctrl+F4** | Move to the previous message (up) in the [Messages panel](https://www.altium.com/documentation/altium-designer/messages-panel) and cross-probe to the object responsible for the message in the associated document (where supported) |  |

2.[Schematic Editor & Schematic Symbol Editor Shortcuts](https://www.altium.com/documentation/altium-designer/shortcut-keys-schematic-editors)

Standard Shortcuts

The following shortcuts are standard to the Schematic and/or Schematic Symbol Editors. They are available in general and do not require you to be performing an interactive process to access them.

|  |  |
| --- | --- |
| **Selecting** | |
| **Shift+F** | Access the Find Similar Objects feature (click on an object to use as the base template). |
| **Click** | Select/deselect the object currently under the cursor. |
| **Shift+Click** | Change the selection status of the object currently under the cursor without affecting the status of other objects. |
| **Shift+Ctrl+X** | Toggle Cross Select Mode On or Off. |
| **Shift+Ctrl+Y** | Toggle the Cross Select Mode's **Reposition selected component in PCB** option On or Off. |
| **Ctrl+A** | Select all objects on the current document. |
| **Alt+Click** | On a net object to select that object, and to highlight all objects associated to that net across all sheets of the active design project. |
| **Click** (away from objects)**, Hold&Drag**(left-to-right) | Select all objects that fall completely within the bounds of the selection area. |
| **Click** (away from objects)**, Hold&Drag**(right-to-left) | Select all objects that fall completely inside the selection area or are touched by its boundary. |
| **Ctrl+n** (**n** = 1 to 8) | Store the current selection in memory location **n**. |
| **Alt+n** (**n** = 1 to 8) | Recall the selection from memory location **n**. |
| **Shift+n** (**n** = 1 to 8) | Add the current selection to the selection already stored in memory location **n**. |
| **Alt+Shift+n** (**n** = 1 to 8) | Recall selection from memory location **n** and add it to the current selection in the design space. |
| **Shift+Ctrl+n** (**n** = 1 to 8) | Apply filtering based on the selection set in memory location **n**. |
| **General Editing** | |
| **Double-Click** | Modify the properties of the object currently under the cursor. |
| **Right-Click** | Access context menu for the design space or object currently under the cursor. If currently within an interactive command, will escape from the current operation. |
| **F2** | Edit selected text object in-place (direct editing). |
| **Ctrl+C** | Copy selection |
| **Ctrl+X** | Cut selection |
| **Ctrl+V** | Paste selection |
| **Ctrl+R** | Copy selected object(s) and paste repeatedly where needed in the design space (rubber stamping). |
| **Shift+Ctrl+V** | Access the [Smart Paste dialog](https://www.altium.com/documentation/altium-designer/sch-dlg-smartpasteformsmart-paste-ad). |
| **Delete** | Delete selection |
| **Ctrl+Z** | Undo |
| **Ctrl+Y** | Redo |
| **Moving** | |
| **Click** (on an object)**, Hold&Drag** | Move the single object currently under the cursor (or group of selected objects if the object is part of that selection). If the **Always Drag** option is enabled (on the **Schematic - Graphical Editing**page of the *Preferences*dialog), then using this shortcut to move an electrical object will maintain connectivity with other electrical objects. |
| **Ctrl+Click, Hold&Drag** | Move the single object currently under the cursor (or group of selected objects if the object is part of that selection). If the **Always Drag** option is disabled (on the **Schematic - Graphical Editing**page of the *Preferences*dialog), then using this shortcut to move an electrical object will maintain connectivity with other electrical objects. |
| **Left Arrow** | Move the cursor to the left in the current document design space in increments of one snap grid unit. |
| **Shift+Left Arrow** | Move the cursor to the left in the current document design space in increments of 10 snap grid units. |
| **Right Arrow** | Move the cursor to the right in the current document design space in increments of one snap grid unit. |
| **Shift+Right Arrow** | Move the cursor to the right in the current document design space, in increments of 10 snap grid units. |
| **Up Arrow** | Move the cursor upwards in the current document design space, in increments of one snap grid unit. |
| **Shift+Up Arrow** | Move the cursor upwards in the current document design space, in increments of 10 snap grid units. |
| **Down Arrow** | Move the cursor downwards in the current document design space, in increments of one snap grid unit. |
| **Shift+Down Arrow** | Move the cursor downwards in the current document design space, in increments of 10 snap grid units. |
| **Ctrl+Left Arrow** | Move the current selection (one or more selected design objects) to the left in the current document design space in increments of one snap grid unit. |
| **Shift+Ctrl+Left Arrow** | Move the current selection (one or more selected design objects) to the left in the current document design space, in increments of 10 snap grid units. |
| **Ctrl+Right Arrow** | Move the current selection (one or more selected design objects) to the right in the current document design space, in increments of one snap grid unit. |
| **Shift+Ctrl+Right Arrow** | Move the current selection (one or more selected design objects) to the right in the current document design space, in increments of 10 snap grid units. |
| **Ctrl+Up Arrow** | Move the current selection (one or more selected design objects) upwards in the current document design space, in increments of one snap grid unit. |
| **Shift+Ctrl+Up Arrow** | Move the current selection (one or more selected design objects) upwards in the current document design space, in increments of 10 snap grid units. |
| **Ctrl+Down Arrow** | Move the current selection (one or more selected design objects) downwards in the current document design space, in increments of one snap grid unit. |
| **Shift+Ctrl+Down Arrow** | Move the current selection (one or more selected design objects) downwards in the current document design space, in increments of 10 snap grid units. |
| **Aligning** | |
| **Shift+Ctrl+L** | Align selected objects by their left edges. |
| **Shift+Ctrl+R** | Align selected objects by their right edges. |
| **Shift+Ctrl+H** | Make the horizontal spacing of selected objects equal. |
| **Shift+Ctrl+T** | Align selected objects by their top edges. |
| **Shift+Ctrl+B** | Align selected objects by their bottom edges. |
| **Shift+Ctrl+V** | Make the vertical spacing of selected objects equal. |
| **Shift+Ctrl+D** | Move selected objects to the nearest point on the current snap grid. |
| **Spacebar** | Rotate selection counterclockwise by 90°. |
| **Shift+Spacebar** | Rotate selection clockwise by 90°. |
| **Viewing the Schematic** | |
| **Ctrl+PgDn** | Display all design objects on the current document. |
| **PgUp** | Zoom-in, relative to the current cursor location.  You can also use the mouse to zoom in to a region of the document by one of the following methods (where applicable and depending on how the buttons of your mouse might be assigned):   * Hold the **Ctrl**key and roll the mouse wheel upward. Use of the **Ctrl**key is a default setting that can be changed from the [System - Mouse Wheel Configuration page](https://www.altium.com/documentation/altium-designer/system-mouse-wheel-configuration-preferences) of the [Preferences dialog](https://www.altium.com/documentation/altium-designer/client-dlg-alloptionsformpreferences-ad). * Hold both the **Ctrl**key and the right mouse button, then move the mouse forward. * Hold the right (first) and left (second) mouse buttons, then move the mouse forward. * Click and hold the mouse wheel, then move the mouse forward. |
| **PgDn** | Zoom-out, relative to the current cursor location.  You can use the mouse to zoom out from a region of the document by one of the following methods (where applicable and depending on how the buttons of your mouse might be assigned):   * Hold the **Ctrl**key and roll the mouse wheel downward. Use of the **Ctrl**key is a default setting that can be changed from the [System - Mouse Wheel Configuration page](https://www.altium.com/documentation/altium-designer/system-mouse-wheel-configuration-preferences) of the [Preferences dialog](https://www.altium.com/documentation/altium-designer/client-dlg-alloptionsformpreferences-ad). * Hold both the **Ctrl**key and the right-mouse button, then move the mouse backward. * Hold the right (first) and left (second) mouse buttons, then move the mouse backward. * Click and hold the mouse wheel, then move the mouse backward. |
| **Mouse Wheel** | Scroll vertically within the design space. This is a default setting that can be changed from the [System - Mouse Wheel Configuration page](https://www.altium.com/documentation/altium-designer/system-mouse-wheel-configuration-preferences) of the [Preferences dialog](https://www.altium.com/documentation/altium-designer/client-dlg-alloptionsformpreferences-ad). |
| **Shift+Mouse Wheel** | Scroll horizontally within the design space. This is a default setting that can be changed from the [System - Mouse Wheel Configuration page](https://www.altium.com/documentation/altium-designer/system-mouse-wheel-configuration-preferences) of the [Preferences dialog](https://www.altium.com/documentation/altium-designer/client-dlg-alloptionsformpreferences-ad). |
| **Home** | Redraw the view in the main design window, placing the location marked by the cursor - prior to launching the command - at the center of the window. |
| **End** | Refresh the screen, in effect performing a redraw of the current document, to remove any undesirable drawing update effects. |
| **Alt+End** | Redraw the current layer of the current document, to remove any undesirable drawing update effects. |
| **Alt+F5** | Toggle the display of the current document's editor between maximized and not maximized. |
| **F5** | Visually toggle the *Net Color Override* feature on or off. |
| **Shift+C** | Clear the filter that is currently being applied to the active document. |
| **Right-Click, Hold&Drag** | Display the slider (panning) hand cursor then drag to move your view of the design space. |
| **Grids** | |
| **G** | Cycle forward through your predefined snap grid settings. |
| **Shift+G** | Cycle backward through your predefined snap grid settings. |
| **Shift+Ctrl+G** | Turn the visible grid on or off in the current document. |
| **Ctrl+Shift** | Temporarily disables the grid. |
| **Shift+E** | Turn the cursor electrical grid on or off. |
| **Finding Text** | |
| **Ctrl+F** | Find text. |
| **Ctrl+H** | Find and replace text. |
| **F3** | Find next occurrence of searched text. |
| **Quick Panel Access** | |
| **F11** | Toggle the display of the [Properties panel](https://www.altium.com/documentation/altium-designer/properties-panel). |
| **F12** | Toggle the display of the [SCH Filter panel](https://www.altium.com/documentation/altium-designer/sch-filter-panel) or the [SCHLIB Filter panel](https://www.altium.com/documentation/altium-designer/schlib-filter-panel) accordingly. |
| **Shift+F12** | Toggle the display of the [SCH List panel](https://www.altium.com/documentation/altium-designer/sch-list-panel) or the [SCHLIB List panel](https://www.altium.com/documentation/altium-designer/schlib-list-panel) accordingly. |
| **Commenting** | |
| **Alt+Ctrl+C** | Add a new comment thread to a point, object, or area on the active schematic document. Before you can start using the comment feature, ensure that you have opened (checked out) a [Managed Project](https://www.altium.com/documentation/altium-designer/managed-projects-and-releasing-a-design-ad) and are working on one of its source schematic documents. |
| **Miscellaneous** | |
| **Ctrl+Double-Click** | * On a [Sheet Entry](https://www.altium.com/documentation/altium-designer/schematic-sheet-entry) to jump to the corresponding port on the sub-sheet referenced by that entry's parent [Sheet Symbol](https://www.altium.com/documentation/altium-designer/schematic-sheet-symbol). * On a [Port](https://www.altium.com/documentation/altium-designer/schematic-port) to jump to the corresponding sheet entry in the parent sheet symbol that references the sub-sheet on which the port resides. * On a port to jump to another port with the same name, on the indicated target schematic document (only available when the **Net Identifier Scope** - set on the [Options tab](https://www.altium.com/documentation/altium-designer/project-options-options) of the *Project* *Options dialog*- is set to Flat, or Global). * On a sheet symbol to descend the design hierarchy to open the child sheet referenced by that symbol. |
| **Ctrl+L** | Perform *Board Level Annotation* through use of the [Board Level Annotate dialog](https://www.altium.com/documentation/altium-designer/workspacemanager-dlg-annotationform-2board-level-annotate-ad). |
| **Ctrl+Home** | Move the cursor to the absolute origin coordinate (0,0) for the current document. |
| **Ctrl+Q** | * In the main design space to access the [Selection Memory dialog](https://www.altium.com/documentation/altium-designer/pcb-dlg-selectionmemoryformselection-memory-pcb-ad), from where you can control all aspects of the selection memory feature. * In a dialog or panel to toggle the measurement units (in the dialog or panel only), between metric (mm) and imperial (mil). |
| **Ctrl+M** | Measure and display the distance between any two points in the current document. |
| https://www.altium.com/documentation/sites/default/files/wiki_attachments/304627/CrossProbe_Icon.png , **Click** | Cross-probe from a chosen object on the current schematic document to its corresponding counterpart on the PCB document remaining in the source document (*Continuous Mode*). |
| https://www.altium.com/documentation/sites/default/files/wiki_attachments/304627/CrossProbe_Icon.png , **Ctrl+Click** | Cross-probe from a chosen object on the current schematic document to its corresponding counterpart on the PCB document making the target document the active document (*Jump To Mode*). |

Interactive Shortcuts

The following shortcuts become available when an interactive process has been launched, such as placing a new design object, or moving an existing one. Note that the shortcuts available will depend on the interactive command and the specific design object that is the focus of that command.

|  |  |
| --- | --- |
| **F1** | Access the [Graphical Editing Hot key List dialog](https://www.altium.com/documentation/altium-designer/pcb-dlg-hotkeylistformgraphical-editing-hot-key-list-pcb-ad) that provides a listing of the shortcuts available (but that cannot be launched) as well as access to the documentation for the currently running command. |
| **Shift+F1** | Access a menu that lists all valid shortcuts for the present stage of the currently running interactive command. |
| **Enter** | Acts like a **Click**of the mouse when placing/moving an object. |
| **PgUp** | Zoom in. |
| **PgDn** | Zoom out. |
| **End** | Refresh the screen. |
| **Home** | Change display to have the cursor (with the attached object being placed/moved) at the center of the design window. |
| **Left Arrow** | Move the cursor (with the attached object being placed/moved) to the left in the current document workspace, in increments of one snap grid unit. |
| **Shift+Left Arrow** | Move the cursor (with the attached object being placed/moved) to the left in the current document workspace, in increments of 10 snap grid units. |
| **Right Arrow** | Move the cursor (with the attached object being placed/moved) to the right in the current document workspace, in increments of one snap grid unit. |
| **Shift+Right Arrow** | Move the cursor (with the attached object being placed/moved) to the right in the current document workspace, in increments of 10 snap grid units. |
| **Up Arrow** | Move the cursor (with the attached object being placed/moved) upwards in the current document workspace, in increments of one snap grid unit. |
| **Shift+Up Arrow** | Move the cursor (with the attached object being placed/moved) upwards in the current document workspace, in increments of 10 snap grid units. |
| **Down Arrow** | Move the cursor (with the attached object being placed/moved) downwards in the current document workspace, in increments of one snap grid unit. |
| **Shift+Down Arrow** | Move the cursor (with the attached object being placed/moved) downwards in the current document workspace, in increments of 10 snap grid units. |
| **Tab** | Access the associated mode of the *Properties*panel in which properties for the object being placed/moved can be changed on-the-fly. |
| **X** | Mirror the object being placed/moved along the X-axis. |
| **Y** | Mirror the object being placed/moved along the Y-axis. |
| **Alt** | Constrain the direction of movement to the horizontal or vertical axis depending on the initial direction of movement. |
| **Shift** | When auto-panning while placing/moving an object, use to pan at higher speed. |
| **Esc** | Escape from the current process - either a stage of the currently running interactive command or the command itself. |
| **Spacebar** | * Rotates the object being placed/moved counterclockwise. Rotation is in increments of 90°. * Toggles start and end sub-modes when placing a [Wire](https://www.altium.com/documentation/altium-designer/schematic-wire)/[Bus](https://www.altium.com/documentation/altium-designer/schematic-bus)/[Signal Harness](https://www.altium.com/documentation/altium-designer/schematic-signal-harness) in 90 Degree or 45 Degree placement modes. * Cycles through placement modes when placing a [Line](https://www.altium.com/documentation/altium-designer/schematic-line). * Changes the wiring mode for any connected wires, buses, or signal harnesses when dragging an electrical object. |
| **Shift+Spacebar** | * Rotates the object being placed/moved clockwise. Rotation is in increments of 90°. * Cycles through placement modes when placing a [Wire](https://www.altium.com/documentation/altium-designer/schematic-wire)/[Bus](https://www.altium.com/documentation/altium-designer/schematic-bus)/[Signal Harness](https://www.altium.com/documentation/altium-designer/schematic-signal-harness). * Changes the wiring mode for any connected wires, buses, or signal harnesses when dragging an electrical object. |
| **Shift+Ctrl+Spacebar** | Rotates the object being dragged clockwise. Rotation is in increments of 90°. |
| **Insert** | * Copies attributes of the object currently under the cursor while placing an object of the same type. * Adds a vertex while placing a [Wire](https://www.altium.com/documentation/altium-designer/schematic-wire), [Line](https://www.altium.com/documentation/altium-designer/schematic-line), [Bus](https://www.altium.com/documentation/altium-designer/schematic-bus), [Signal Harness](https://www.altium.com/documentation/altium-designer/schematic-signal-harness), or [Polygon](https://www.altium.com/documentation/altium-designer/schematic-polygon). |
| **Click&Hold+Insert** | On a selected segment of a placed [Wire](https://www.altium.com/documentation/altium-designer/schematic-wire), [Line](https://www.altium.com/documentation/altium-designer/schematic-line), [Bus](https://www.altium.com/documentation/altium-designer/schematic-bus), or [Signal Harness](https://www.altium.com/documentation/altium-designer/schematic-signal-harness) or an edge of a placed [Polygon](https://www.altium.com/documentation/altium-designer/schematic-polygon) to add a vertex at that point. |
| **Click&Hold+Delete** | On a vertex of a placed [Wire](https://www.altium.com/documentation/altium-designer/schematic-wire), [Line](https://www.altium.com/documentation/altium-designer/schematic-line), [Bus](https://www.altium.com/documentation/altium-designer/schematic-bus), [Signal Harness](https://www.altium.com/documentation/altium-designer/schematic-signal-harness) or [Polygon](https://www.altium.com/documentation/altium-designer/schematic-polygon) to remove that vertex. |
| **Backspace**or**Delete** | Remove the last placed segment, while placing a [Wire](https://www.altium.com/documentation/altium-designer/schematic-wire), [Line](https://www.altium.com/documentation/altium-designer/schematic-line), [Bus](https://www.altium.com/documentation/altium-designer/schematic-bus), [Signal Harness](https://www.altium.com/documentation/altium-designer/schematic-signal-harness), or [Polygon](https://www.altium.com/documentation/altium-designer/schematic-polygon). |
| **+** (on the numeric keypad) | Enlarges the size of the [IEEE Symbol](https://www.altium.com/documentation/altium-designer/schematic-ieee-symbols) currently being placed/moved. |
| **-** (on the numeric keypad) | Reduces the size of the [IEEE Symbol](https://www.altium.com/documentation/altium-designer/schematic-ieee-symbols) currently being placed/moved. |
| **S** | Relocate the currently selected [Sheet Entry](https://www.altium.com/documentation/altium-designer/schematic-sheet-entry) (or entries) that are being moved, to the directly opposite side of the parent [Sheet Symbol](https://www.altium.com/documentation/altium-designer/schematic-sheet-symbol). |
| **V** | Reverse order while moving two or more selected [Sheet Entries](https://www.altium.com/documentation/altium-designer/schematic-sheet-entry). |
| **T** | * Toggle IO Type while moving one or more selected [Sheet Entries](https://www.altium.com/documentation/altium-designer/schematic-sheet-entry). * Toggle IO Type for all Sheet Entries while graphically resizing the parent [Sheet Symbol](https://www.altium.com/documentation/altium-designer/schematic-sheet-symbol). |

Sub-Menu Shortcuts

The following shortcuts give access to specific pop-up menus of commands, directly from within the design space itself. Some of these will be familiar as they are sub-menus found within the main menus for the editor. Each of the main menus also have a defined pop-up key for quick access from within the design space, see [Accelerator Keys](https://www.altium.com/documentation/altium-designer/shortcut-keys#accelerator-keys) for more information.

|  |  |
| --- | --- |
| **A** | the **Align**sub-menu (from the main **Edit** menu). |
| **B** | the **Toolbars**sub-menu |
| **J** | the **Jump**sub-menu (from the main **Edit** menu). |
| **K** | the **Panels** sub-menu (from the main **View**menu). |
| **M** | the **Move**sub-menu (from the main **Edit** menu). |
| **O** | the **Options**sub-menu. |
| **S** | the **Select**sub-menu (from the main **Edit** menu). |
| **X** | the **DeSelect**sub-menu (from the main **Edit** menu). |
| **Y** | the **Filter**sub-menu. |
| **Z** | a pop-up menu with zoom commands |

3.[PCB Editor & PCB Footprint Editor Shortcuts](https://www.altium.com/documentation/altium-designer/shortcut-keys-pcb-editors)

Standard Shortcuts

The following shortcuts are standard to the PCB and/or PCB Footprint Editors. They are available in general and do not require you to be performing an interactive process to access them.

|  |  |
| --- | --- |
| **Selecting** | |
| **Shift+F** | Access the Find Similar Objects feature (click on an object to use as the base template). |
| **Click** | Select/deselect the object currently under the cursor. |
| **Shift+Click** | Change the selection status of the object currently under the cursor without affecting the status of other objects. |
| **Tab** | With an initial object selected in the design, extend the selection to include the next higher-level object (or objects), based on logical hierarchy. In addition, the feature caters for selection extension across multiple objects selected across different nets in the design. |
| **Shift+Tab** | Single select the next design object in a set of co-located (overlapping) objects without utilizing a selection pop-up window. To use this command, ensure that the **Display popup selection dialog** option is disabled on the [PCB Editor - General page](https://www.altium.com/documentation/altium-designer/pcb-editor-general-preferences) of the *Preferences*dialog. |
| **Shift+Ctrl+X** | Toggle Cross Select Mode On or Off. |
| **Shift+Ctrl+Y** | Toggle the Cross Select Mode's **Reposition selected component in PCB** option On or Off. |
| **Ctrl+A** | Select all objects on the current document. |
| **Ctrl+B** | Select all objects that reside within the boundary of the defined board shape. |
| **Ctrl+H** | Select all electrical objects that are connected to the same piece of copper. |
| **Alt+Click** | On a connection to select that connection. |
| **Alt+Shift+Click** | On a connection to select that connection in addition to the connections already selected (cumulative connection selection). |
| **Click** (away from objects)**, Hold&Drag**(left-to-right) | Select all objects that fall completely within the bounds of the selection area. |
| **Click** (away from objects)**, Hold&Drag**(right-to-left) | Select all objects that fall completely inside the selection area or are touched by its boundary. |
| **Ctrl+Hold&Drag**(left-to-right) | Select all component pads that fall completely within the bounds of the selection area. |
| **Ctrl+Hold&Drag**(right-to-left) | Select all component pads that fall completely inside the selection area or are touched by its boundary. |
| **Alt+Click&Drag** (from right-to-left) | To select all connections touched by the dragged rectangle. |
| **Ctrl+n** (**n** = 1 to 8) | Store the current selection in memory location **n**. |
| **Alt+n** (**n** = 1 to 8) | Recall the selection from memory location **n**. |
| **Shift+n** (**n** = 1 to 8) | Add the current selection to the selection already stored in memory location **n**. |
| **Alt+Shift+n** (**n** = 1 to 8) | Recall selection from memory location **n** and add it to the current selection in the design space. |
| **Shift+Ctrl+n** (**n** = 1 to 8) | Apply filtering based on the selection set in memory location **n**. |
| **General Editing** | |
| **Double-Click** | Modify the properties of the object currently under the cursor. |
| **Right-Click** | Access context menu for the design space or object currently under the cursor. If currently within an interactive command, will escape from the current operation. |
| **Ctrl+C** | Copy selection |
| **Ctrl+X** | Cut selection |
| **Ctrl+V** | Paste selection |
| **Ctrl+R** | Copy selected object(s) and paste repeatedly where needed in the design space (rubber stamping). |
| **Delete** | Delete selection |
| **Ctrl+Z** | Undo |
| **Ctrl+Y** | Redo |
| **Moving** | |
| **Click** (on an object)**, Hold&Drag** | * Move the single object currently under the cursor (or group of selected objects if the object is part of that selection). * On a via in a stack of vias, use to move the entire stack to a new location. |
| **Ctrl+Click, Hold&Drag** | On a via in a stack of vias, use to move just that via to a new location (and not the entire stack). |
| **Left Arrow** | Move the cursor to the left in the current document design space in increments of one snap grid unit. |
| **Shift+Left Arrow** | Move the cursor to the left in the current document design space in increments of 10 snap grid units. |
| **Right Arrow** | Move the cursor to the right in the current document design space in increments of one snap grid unit. |
| **Shift+Right Arrow** | Move the cursor to the right in the current document design space, in increments of 10 snap grid units. |
| **Up Arrow** | Move the cursor upwards in the current document design space, in increments of one snap grid unit. |
| **Shift+Up Arrow** | Move the cursor upwards in the current document design space, in increments of 10 snap grid units. |
| **Down Arrow** | Move the cursor downwards in the current document design space, in increments of one snap grid unit. |
| **Shift+Down Arrow** | Move the cursor downwards in the current document design space, in increments of 10 snap grid units. |
| **Ctrl+Left Arrow** | Move the current selection (one or more selected design objects) to the left in the current document design space in increments of one snap grid unit. |
| **Shift+Ctrl+Left Arrow** | Move the current selection (one or more selected design objects) to the left in the current document design space, in increments of 10 snap grid units. |
| **Ctrl+Right Arrow** | Move the current selection (one or more selected design objects) to the right in the current document design space, in increments of one snap grid unit. |
| **Shift+Ctrl+Right Arrow** | Move the current selection (one or more selected design objects) to the right in the current document design space, in increments of 10 snap grid units. |
| **Ctrl+Up Arrow** | Move the current selection (one or more selected design objects) upwards in the current document design space, in increments of one snap grid unit. |
| **Shift+Ctrl+Up Arrow** | Move the current selection (one or more selected design objects) upwards in the current document design space, in increments of 10 snap grid units. |
| **Ctrl+Down Arrow** | Move the current selection (one or more selected design objects) downwards in the current document design space, in increments of one snap grid unit. |
| **Shift+Ctrl+Down Arrow** | Move the current selection (one or more selected design objects) downwards in the current document design space, in increments of 10 snap grid units. |
| **Aligning** | |
| **Shift+Ctrl+L** | Align selected objects by their left edges. |
| **Shift+Ctrl+R** | Align selected objects by their right edges. |
| **Alt+Shift+L** | Align selected design objects by their left edges while maintaining adequate spacing in observance with applicable design rules. |
| **Alt+Shift+R** | Align selected design objects by their right edges while maintaining adequate spacing in observance with applicable design rules. |
| **Shift+Ctrl+H** | Make the horizontal spacing of selected objects equal. |
| **Shift+Ctrl+T** | Align selected objects by their top edges. |
| **Shift+Ctrl+B** | Align selected objects by their bottom edges. |
| **Shift+Alt+I** | Align selected design objects by their top edges while maintaining adequate spacing in observance with applicable design rules. |
| **Shift+Alt+N** | Align selected design objects by their bottom edges while maintaining adequate spacing in observance with applicable design rules. |
| **Shift+Ctrl+V** | Make the vertical spacing of selected objects equal. |
| **Shift+Ctrl+D** | Move selected components to the nearest point on the required component placement grid. |
| **Viewing the Board** | |
| **1** | Switch the display of the PCB design space to *Board Planning Mode*. |
| **2** | Switch the display of the PCB design space to *2D Layout Mode*. |
| **3** | Switch the display of the PCB design space to *3D Layout Mode*. |
| **Ctrl+Alt+2** | Switch the display of the PCB design space to *2D Layout Mode*and see the same location and orientation of the board as you switch. |
| **Ctrl+Alt+3** | Switch the display of the PCB design space to *3D Layout Mode*and see the same location and orientation of the board as you switch. |
| **Shift+S** | Cycle through the available single layer viewing modes. The available modes are determined by enabling the corresponding options in the **Available Single Layer Modes** region on the [PCB Editor - Board Insight Display page](https://www.altium.com/documentation/altium-designer/pcb-editor-board-insight-display-preferences) of the [Preferences dialog](https://www.altium.com/documentation/altium-designer/client-dlg-alloptionsformpreferences-ad). |
| **Ctrl+PgDn** | Display all design objects on the current document. |
| **PgUp** | Zoom-in, relative to the current cursor location.  You can also use the mouse to zoom in to a region of the document by one of the following methods (where applicable and depending on how the buttons of your mouse might be assigned):   * Hold the **Ctrl**key and roll the mouse wheel upward. Use of the **Ctrl**key is a default setting that can be changed from the [System - Mouse Wheel Configuration page](https://www.altium.com/documentation/altium-designer/system-mouse-wheel-configuration-preferences) of the [Preferences dialog](https://www.altium.com/documentation/altium-designer/client-dlg-alloptionsformpreferences-ad). * Hold both the **Ctrl**key and the right mouse button, then move the mouse forward. * Hold the right (first) and left (second) mouse buttons, then move the mouse forward. * Click and hold the mouse wheel, then move the mouse forward. |
| **PgDn** | Zoom-out, relative to the current cursor location.  You can use the mouse to zoom out from a region of the document by one of the following methods (where applicable and depending on how the buttons of your mouse might be assigned):   * Hold the **Ctrl**key and roll the mouse wheel downward. Use of the **Ctrl**key is a default setting that can be changed from the [System - Mouse Wheel Configuration page](https://www.altium.com/documentation/altium-designer/system-mouse-wheel-configuration-preferences) of the [Preferences dialog](https://www.altium.com/documentation/altium-designer/client-dlg-alloptionsformpreferences-ad). * Hold both the **Ctrl**key and the right-mouse button, then move the mouse backward. * Hold the right (first) and left (second) mouse buttons, then move the mouse backward. * Click and hold the mouse wheel, then move the mouse backward. |
| **Shift+PgUp** | Zoom-in, relative to the current cursor location and in progressively smaller steps. |
| **Shift+PgDn** | Zoom-out, relative to the current cursor location and in progressively larger steps. |
| **Ctrl+PgUp** | Set the magnification of the current document to 400%. |
| **Mouse Wheel** | Scroll vertically within the design space. This is a default setting that can be changed from the [System - Mouse Wheel Configuration page](https://www.altium.com/documentation/altium-designer/system-mouse-wheel-configuration-preferences) of the [Preferences dialog](https://www.altium.com/documentation/altium-designer/client-dlg-alloptionsformpreferences-ad). |
| **Shift+Mouse Wheel** | Scroll horizontally within the design space. This is a default setting that can be changed from the [System - Mouse Wheel Configuration page](https://www.altium.com/documentation/altium-designer/system-mouse-wheel-configuration-preferences) of the [Preferences dialog](https://www.altium.com/documentation/altium-designer/client-dlg-alloptionsformpreferences-ad). |
| **Home** | Redraw the view in the main design window, placing the location marked by the cursor - prior to launching the command - at the center of the window. |
| **End** | Refresh the screen, in effect performing a redraw of the current document, to remove any undesirable drawing update effects. |
| **Alt+End** | Redraw the current layer of the current document, to remove any undesirable drawing update effects. |
| **Alt+F5** | Toggle the display of the current document's editor between maximized and not maximized. |
| **F5** | Visually toggle the *Net Color Override* feature on or off. |
| **Shift+Z** | Toggle the 3D model visibility in the current PCB document. |
| **L** | Access the **Layers And Colors** tab of the [View Configuration panel](https://www.altium.com/documentation/altium-designer/view-configuration-panel-pcb) in which you can configure the display of layers for the board and the colors assigned to those layers. |
| **Ctrl+D** | Access the**View Options** tab of the [View Configuration panel](https://www.altium.com/documentation/altium-designer/view-configuration-panel-pcb) in which you can configure the mode used to display each of the various design items within the design space. |
| **Right-Click, Hold&Drag** | Display the slider (panning) hand cursor then drag to move your view of the design space. |
| **Board Insight Display** | |
| **Shift+H** | Toggle the Heads Up Display on or off. |
| **Shift+G** | Toggle Heads Up Display tracking on or off. |
| **Insert** | Resets the Delta Origin point for the Heads Up Display feature to 0,0. |
| **Shift+D** | Toggle the display of the *Delta* coordinates within the Heads Up Display. |
| **Shift+V** | Access the *Board Insight* pop-up, listing all violations (of defined [Design Rules](https://www.altium.com/documentation/altium-designer/pcb-design-rule-types)) currently under the cursor. |
| **Shift+X** | Access the *Board Insight* pop-up, listing all components and/or net objects currently under the cursor. |
| **Grids** | |
| **Shift+E** | Cycle to the next mode of object *Hotspot Snapping*. |
| **Ctrl+G** | Access the dedicated grid editor dialog for the snap grid currently under the cursor. |
| **Shift+Ctrl+G** | Set the X (horizontal) and Y (vertical) step values - for the default Global Board Snap Grid - simultaneously to a chosen value. |
| **Ctrl+Shift** | Temporarily disables the grid. |
| **Shift+H** | Toggle on/off whether the cursor on a grid will snap to the active grid. Once enabled, the cursor will pull or snap to the nearest snap group location. |
| **Viewing Differences (Legacy Collaborative Design)** | |
| **Shift+O** | Toggle the display of the Difference Map Overlay in the main design space On or Off. Note that this command is available only provided a comparison has been performed from the [Collaborate, Compare and Merge panel](https://www.altium.com/documentation/altium-designer/collaborate-compare-and-merge-panel). |
| **F6** | Toggle the state of the current cell containing detected differences between checked and unchecked when using Altium Designer's Collaborative PCB Design functionality (where a comparison has been performed from the [Collaborate, Compare and Merge panel](https://www.altium.com/documentation/altium-designer/collaborate-compare-and-merge-panel)). |
| **F7** | Navigate to the previous cell containing one or more detected differences when using Altium Designer's Collaborative PCB Design functionality (where a comparison has been performed from the [Collaborate, Compare and Merge panel](https://www.altium.com/documentation/altium-designer/collaborate-compare-and-merge-panel)). |
| **F8** | Navigate to the next cell containing one or more detected differences when using Altium Designer's Collaborative PCB Design functionality (where a comparison has been performed from the [Collaborate, Compare and Merge panel](https://www.altium.com/documentation/altium-designer/collaborate-compare-and-merge-panel)). |
| **Highlighting** | |
| **Ctrl+Click** | * On a net object to highlight the entire routed net. * On a layer tab to highlight all content on that layer. * In free space to clear current highlighting. |
| **Shift+Ctrl+Click** | * On a net object to highlight the entire routed net in addition to the routed nets already highlighted (cumulative routed net highlighting). * On a layer tab to highlight all content on that layer in addition to the content already highlighted on other layers (cumulative layer highlighting). |
| **Alt+Ctrl** | While hovering the cursor over a layer tab to highlight the content of that layer only. |
| **]** | Increase the masking level, which decreases the extent of dimming in the design space for all objects not falling under the scope of the currently applied filter. Note that this shortcut is only effective when the highlighting method for objects not passing the applied filter is set to Dim. Unlike the Mask highlighting method, such objects can still be selected and edited. |
| **[** | Decrease the masking level, which increases the extent of dimming in the design space for all objects not falling under the scope of the currently applied filter. Note that this shortcut is only effective when the highlighting method for objects not passing the applied filter is set to Dim. Unlike the Mask highlighting method, such objects can still be selected and edited. |
| **Shift+C** | Clear the filter that is currently being applied to the active document. |
| **Quick Panel Access** | |
| **F11** | Toggle the display of the [Properties panel](https://www.altium.com/documentation/altium-designer/properties-panel). |
| **F12** | Toggle the display of the [PCB Filter panel](https://www.altium.com/documentation/altium-designer/pcb-filter-panel) or the [PCBLIB Filter panel](https://www.altium.com/documentation/altium-designer/pcblib-filter-panel) accordingly. |
| **Shift+F12** | Toggle the display of the [PCB List panel](https://www.altium.com/documentation/altium-designer/pcb-list-panel) or the [PCBLIB List panel](https://www.altium.com/documentation/altium-designer/pcblib-list-panel) accordingly. |
| **Layer Switching** | |
| **+** (on numeric keypad) | Switch to the next enabled layer. |
| **-** (on numeric keypad) | Switch to the previous enabled layer. |
| **\*** (on numeric keypad) | Switch to the next enabled signal layer. |
| **Shift+\*** (on numeric keypad) | Switch to the previous enabled signal layer. |
| **Routing/Track Modification** | |
| **Shift+A** | ActiveRoute selected connections. |
| **Shift+Ctrl+Click&Hold** | Create a vertex (or break) in a track segment at the current cursor position. |
| **Backspace** | Delete a single, selected end-of-route object (component-free track, arc, via, or pad). The singular routing object connected to the deleted object will be automatically selected and ready for subsequent deletion. By using the command repeatedly, you are able to incrementally unwind the routed path in the same way as using the **Backspace** shortcut while interactively routing. |
| **Ctrl+Delete** | Delete one or more selected routing objects (component-free tracks, arcs, vias, and pads) on the current document. All routing objects connected to those deleted will be automatically selected and ready for subsequent deletion. By using the command repeatedly, you are able to incrementally unwind the routed path in both directions. |
| **Ctrl+Alt+G** | Use to improve the quality of the selected routes by reducing the overall length and number of corners. |
| **Commenting** | |
| **Alt+Ctrl+C** | Add a new comment thread to a point, object, or area on the active PCB document. Before you can start using the comment feature, ensure that you have opened (checked out) a [Managed Project](https://www.altium.com/documentation/altium-designer/managed-projects-and-releasing-a-design-ad) and are working on its PCB document. |
| **Miscellaneous** | |
| **Alt+Insert** | Paste objects onto the current layer regardless of their original layer assignments. |
| **Ctrl+Home** | Move the cursor to the absolute origin at the lower-left corner of the design space. |
| **Ctrl+End** | Move the cursor to the relative origin of the current document (PCB document), or the location of the component reference point (PCB Footprint document). |
| **Ctrl+Q** | * In the main design space to access the [Selection Memory dialog](https://www.altium.com/documentation/altium-designer/pcb-dlg-selectionmemoryformselection-memory-pcb-ad), from where you can control all aspects of the selection memory feature. * In a dialog or panel to toggle the measurement units (in the dialog or panel only), between metric (mm) and imperial (mil). |
| **Alt+Left Arrow** | Jump to, and make active, the previous component in the current library document. Note that after reaching the first component in the list, the command will not cycle to the last component again. |
| **Alt+Right Arrow** | Jump to, and make active, the next component in the current library document. Note that after reaching the last component in the list, the command will not cycle to the first component again. |
| **Q** | Toggle the measurement units for the current document between metric (mm) and imperial (mil). |
| **Ctrl+M** | Measure and display the distance between any two points in the current document. |
| https://www.altium.com/documentation/sites/default/files/wiki_attachments/304647/CrossProbe_Icon.png , **Click** | Cross-probe from a chosen object on the current PCB document to its corresponding counterpart on the relevant schematic source document while remaining in the PCB document (*Continuous Mode*) |
| https://www.altium.com/documentation/sites/default/files/wiki_attachments/304647/CrossProbe_Icon.png , **Ctrl+Click** | Cross-probe from a chosen object on the current PCB document to its corresponding counterpart on the relevant schematic document, making the target schematic document the active document (*Jump To Mode*). |

General Interactive Shortcuts

The following shortcuts become available when an interactive process has been launched, such as placing a new design object, or moving an existing one. Note that the shortcuts available will depend on the interactive command and the specific design object that is the focus of that command.

|  |  |
| --- | --- |
| **F1** | Access the [Graphical Editing Hot key List dialog](https://www.altium.com/documentation/altium-designer/pcb-dlg-hotkeylistformgraphical-editing-hot-key-list-pcb-ad) that provides a listing of the shortcuts available (but that cannot be launched) as well as access to the documentation for the currently running command. |
| **Shift+F1** | Access a menu that lists all valid shortcuts for the present stage of the currently running interactive command. |
| **Enter** | Acts like a **Click**of the mouse when placing/moving an object. |
| **Tab** | Access the associated mode of the *Properties*panel in which properties for the object being placed/moved can be changed on-the-fly. |
| **X** | Mirror the object being placed/moved along the X-axis. |
| **Y** | Mirror the object being placed/moved along the Y-axis. |
| **L** | Flip the object being placed/moved to the other side of the board. |
| **Alt** | Constrain the direction of movement to the horizontal or vertical axis depending on the initial direction of movement. |
| **Shift** | When auto-panning while placing/moving an object, use to pan at higher speed. |
| **Esc** | Escape from the current process - either a stage of the currently running interactive command or the command itself. |
| **Spacebar** | Rotates the object being placed/moved counterclockwise. Rotation is in accordance with the value for the **Rotation Step** defined on the [PCB Editor – General page](https://www.altium.com/documentation/altium-designer/pcb-editor-general-preferences) of the [Preferences dialog](https://www.altium.com/documentation/altium-designer/client-dlg-alloptionsformpreferences-ad). |
| **Shift+Spacebar** | Rotates the object being placed/moved clockwise. Rotation is in accordance with the value for the **Rotation Step** defined on the [PCB Editor – General page](https://www.altium.com/documentation/altium-designer/pcb-editor-general-preferences) of the [Preferences dialog](https://www.altium.com/documentation/altium-designer/client-dlg-alloptionsformpreferences-ad). |

4. [Component Editor Shortcuts](https://www.altium.com/documentation/altium-designer/shortcut-keys-component-editor)

|  |  |
| --- | --- |
| **Single Component Editing Mode** | |
| **Ctrl+X** | Clear the contents of the **Name**, **Description**, or **Id**field and add it to the clipboard. |
| **Ctrl+C** | * Copy the content of the **Name**, **Description**, or **Id**field to the clipboard. * Copy the content of the **Name** and/or **Value**of selected cell(s) in the **Parameters**region to the clipboard. |
| **Ctrl+V** | * Paste content from the clipboard into the **Name**, **Description**, or **Id**field. * Paste content from the clipboard into the **Value** field for a parameter in the **Parameters**region. Note: ensure the target **Value**field is ready to accommodate the paste (select the field, then click again to enter editing mode). |
| **Shift+Ctrl+M** | Access the [Manufacturer Part Search](https://www.altium.com/documentation/altium-designer/manufacturer-part-search-panel) dialog in which you can browse for a required component across enabled supplier's online part databases. Once you select a component and click **OK**, the *Use Component Data* dialog will appear, allowing you to determine what data from that component you would like to bring in for the component you are defining (name, description, parameters, symbol, models, datasheets). Any defined Part Choices for the selected component will be brought in as well. |
| **F5** | Refresh the Component Editor. This is useful if, for any reason, you have lost connection with your connected Workspace. It ensures model links are refreshed and also, if the Component Editor is referencing a Component Template, it allows the data defined in that template to be brought in again, which is very helpful if you have inadvertently cleared content from read-only cells! |
| **Batch Component Editing Mode** | |
| **Ctrl+X** | Clear the content of the currently selected (and editable) cell(s) - in the component definitions area - and copy that content to the clipboard. |
| **Ctrl+C** | Copy the content of the currently selected (and editable) cell(s) - in the component definitions area - to the clipboard. |
| **Ctrl+V** | Paste content from the clipboard into the currently selected (and editable) cell(s) in the component definitions area. |
| **Delete** | Clear the content of the currently selected (and editable) cell(s) in the component definitions area. |
| **Ctrl+Delete** | Remove the currently selected component definition(s) from the document. |
| **Shift+Ctrl+X** | Cut the selected component definition(s) from the document and add them to the clipboard. |
| **Shift+Ctrl+C** | Copy the selected component definition(s) to the clipboard. |
| **Ctrl+D** | Clone the selected component definition. |
| **Shift+Ctrl+V** | Paste component definition(s) from the clipboard into the component definitions area. |
| **Shift+Ctrl+M** | Access the [Manufacturer Part Search dialog](https://www.altium.com/documentation/altium-designer/manufacturer-part-search-panel) in which you can browse for a required component across enabled supplier's online part databases. A selected component will be used to create a new component definition in the Component Editor. |
| **F5** | Refresh the Component Editor. This is useful if, for any reason, you have lost connection with your connected Workspace. It ensures model links are refreshed and also, if the Component Editor is referencing a Component Template, it allows the data defined in that template to be brought in again, which is very helpful if you have inadvertently cleared content from read-only cells! |
| **Ctrl+Z** | Restore the Component Editor to its state previous to the last operation. Multiple levels of undo are supported. |
| **Ctrl+Y** | Restore changes made by the Undo feature in the Component Editor. Multiple levels of Redo are supported. |

5. [Output Job Editor Shortcuts](https://www.altium.com/documentation/altium-designer/shortcut-keys-output-job-editor)

|  |  |
| --- | --- |
| **General Editing** | |
| **Ctrl+X** | * Clear the selected output(s) and copy them to the Output Job Editor's clipboard. * Clear the selected output container/print job, and copy it to the Output Job Editor's clipboard. |
| **Ctrl+C** | * Copy the selected output(s) to the Output Job Editor's clipboard. * Copy the selected output container/print job to the Output Job Editor's clipboard |
| **Ctrl+V** | * Place the contents of the Output Job Editor's clipboard into the active Output Job file * Place the output container/print job from the OutputJob Editor's clipboard into the relevant output media section of the active Output Job file |
| **Delete** | * Remove the selected output(s). * Remove the selected output container/print job. |
| **Working with Outputs** | |
| **Ctrl+D** | Make an identical copy (duplicate) of the selected output. If multiple outputs are selected, the output that has focus (distinguished by a dotted outline around its Name cell) will be duplicated. |
| **Alt+Enter** | Launch the associated configuration dialog (where available) for the selected output, in which you can define exactly how, and what, you wish to be generated. If multiple outputs are selected, the output that has focus (distinguished by a dotted outline around its Name cell) will be configured. |
| **Ctrl++** (on numeric keypad) | Enable all selected outputs for generation, connecting them to the currently selected output container/print job. |
| **Ctrl+-** (on numeric keypad) | Disable all selected outputs from generation, disconnecting them from the currently selected output container/print job. |
| **Working with Output Containers/Print Jobs** | |
| **Drag and Drop** | * One or more selected applicable (printable) outputs into the **Hard Copy** region to create a new Print Job. The output(s) will automatically be linked. * One or more selected applicable outputs into the **Output Containers** region to create a new PDF output container. The output(s) will automatically be linked. * One or more selected PCB3D Video outputs into the **Output Containers** region to create a new Video output container. The output(s) will automatically be linked. * One or more selected applicable outputs into the **Output Containers** region to create a new Folder Structure output container. The output(s) will automatically be linked. |
| **Shift+Ctrl+O** | Configure the properties of the currently selected output container/print job. |
| **F9** | Generate the outputs that are enabled and linked to the selected output container/print job. |

6.[Draftsman Editor Shortcuts](https://www.altium.com/documentation/altium-designer/shortcut-keys-draftsman-editor)

Standard Shortcuts

The following shortcuts are standard to the Draftsman Editor. They are available in general and do not require you to be performing an interactive process to access them.

|  |  |
| --- | --- |
| **Selecting** | |
| **Click** | Select the object currently under the cursor. |
| **Shift+Click** | On an object to add it to the current selection (cumulative selection). |
| **Ctrl+A** | Select all objects on the current document. |
| **Click** (away from objects)**, Hold&Drag**(left-to-right) | Select all objects that fall completely within the bounds of the selection area. |
| **Click** (away from objects)**, Hold&Drag**(right-to-left) | Select all objects that fall completely inside the selection area or are touched by its boundary. |
| **Shift+Tab** | Switch between selection of a component and its overlapping designator in a Board Assembly View. |
| **General Editing** | |
| **Double-Click** | Modify the properties of the object currently under the cursor. |
| **Right-Click** | Access context menu for the design space. If currently within an interactive command, will escape from the current operation. |
| **Ctrl+C** | Copy selection. |
| **Ctrl+X** | Cut selection. |
| **Ctrl+V** | Paste selection. |
| **Delete** | Delete selection. |
| **Ctrl+Z** | Undo. |
| **Ctrl+Y** | Redo. |
| **Moving** | |
| **Click** (on an object)**, Hold&Drag** | Move the single object currently under the cursor (or group of selected objects if the object is part of that selection). |
| **Left Arrow** | Move the current selection (one or more selected design objects) to the left in the current document design space in increments of one snap grid unit. |
| **Shift+Left Arrow** | Move the current selection (one or more selected design objects) to the left in the current document design space, in increments of 10 snap grid units. |
| **Right Arrow** | Move the current selection (one or more selected design objects) to the right in the current document design space, in increments of one snap grid unit. |
| **Shift+Right Arrow** | Move the current selection (one or more selected design objects) to the right in the current document design space, in increments of 10 snap grid units. |
| **Up Arrow** | Move the current selection (one or more selected design objects) upwards in the current document design space, in increments of one snap grid unit. |
| **Shift+Up Arrow** | Move the current selection (one or more selected design objects) upwards in the current document design space, in increments of 10 snap grid units. |
| **Down Arrow** | Move the current selection (one or more selected design objects) downwards in the current document design space, in increments of one snap grid unit. |
| **Shift+Down Arrow** | Move the current selection (one or more selected design objects) downwards in the current document design space, in increments of 10 snap grid units. |
| **Viewing the Draftsman Sheet** | |
| **Ctrl+PgDn** | Change the view in the main design window so that the active Draftsman document is made completely visible (where possible). If the document contains multiple sheets, fitting the entire document into view especially if there are a great many sheets, may not be possible. In this case, the software will attempt to fit what it can with the center of the document at the center of the main design window. |
| **PgUp** | Zoom-in, relative to the current cursor location.  You can also use the mouse to zoom in to a region of the document by one of the following methods (where applicable and depending on how the buttons of your mouse might be assigned):   * Hold the **Ctrl**key and roll the mouse wheel upward. Use of the **Ctrl**key is a default setting that can be changed from the [System - Mouse Wheel Configuration page](https://www.altium.com/documentation/altium-designer/system-mouse-wheel-configuration-preferences) of the [Preferences dialog](https://www.altium.com/documentation/altium-designer/client-dlg-alloptionsformpreferences-ad). * Hold both the **Ctrl**key and the right mouse button, then move the mouse forward. * Click and hold the mouse wheel, then move the mouse forward. |
| **PgDn** | Zoom-out, relative to the current cursor location.  You can use the mouse to zoom out from a region of the document by one of the following methods (where applicable and depending on how the buttons of your mouse might be assigned):   * Hold the **Ctrl**key and roll the mouse wheel downward. Use of the **Ctrl**key is a default setting that can be changed from the [System - Mouse Wheel Configuration page](https://www.altium.com/documentation/altium-designer/system-mouse-wheel-configuration-preferences) of the [Preferences dialog](https://www.altium.com/documentation/altium-designer/client-dlg-alloptionsformpreferences-ad). * Hold both the **Ctrl**key and the right-mouse button, then move the mouse backward. * Click and hold the mouse wheel, then move the mouse backward. |
| **Mouse Wheel** | Scroll vertically within the design space. This is a default setting that can be changed from the [System - Mouse Wheel Configuration page](https://www.altium.com/documentation/altium-designer/system-mouse-wheel-configuration-preferences) of the [Preferences dialog](https://www.altium.com/documentation/altium-designer/client-dlg-alloptionsformpreferences-ad). |
| **Shift+Mouse Wheel** | Scroll horizontally within the design space. This is a default setting that can be changed from the [System - Mouse Wheel Configuration page](https://www.altium.com/documentation/altium-designer/system-mouse-wheel-configuration-preferences) of the [Preferences dialog](https://www.altium.com/documentation/altium-designer/client-dlg-alloptionsformpreferences-ad). |
| **Right-Click, Hold&Drag** | Pan the view within the main design window. |
| **Grids** | |
| **G** | Access the *Snap Grid* dialog, to set the required grid size. |
| **Shift+Ctrl+G** | Turn the visible grid on or off in the current document. |
| **Ctrl+Shift** | Temporarily disables the grid when placing/moving an object. |
| **Shift+E** | Toggle the snapping functionality on or off. |
| **Finding Text** | |
| **Ctrl+F** | Find text. |
| **F3** | Find next occurrence of searched text. |
| **Miscellaneous** | |
| **Shift+Ctrl+C** | Copy the formatting from the currently selected text object (title or note). Alternatively, select the text object whose formatting you wish to copy and launch the **Format Painter** command from the main **Edit** menu. |
| **Shift+Ctrl+V** | Paste the formatting copied from a previously selected text object to the currently selected text object (title or note). Alternatively, if you have launched the **Format Painter** command from the main **Edit** menu, just click on each additional title/note object to apply the 'copied' formatting. |
| **F11** | Toggle the display of the [Properties](https://www.altium.com/documentation/altium-designer/properties-panel-draftsman) panel. |

Interactive Shortcuts

The following shortcuts become available when an interactive process has been launched, such as placing a new design object, or moving an existing one. Note that the shortcuts available will depend on the interactive command and the specific design object that is the focus of that command.

|  |  |
| --- | --- |
| **PgUp** | Zoom in. |
| **PgDn** | Zoom out. |
| **X** | Mirror the object being moved along the X-axis. |
| **Y** | Mirror the object being moved along the Y-axis. |
| **Esc** | Escape from the current process - either a stage of the currently running interactive command or the command itself. |
| **Spacebar** | Rotates the object being moved counterclockwise. Rotation is in increments of 90°. |
| **Shift+Spacebar** | Rotates the object being moved clockwise. Rotation is in increments of 90°. |
| **Backspace** | Remove the last placed segment, while placing a region. |

Sub-Menu Shortcuts

The following shortcuts give access to specific pop-up menus of commands, directly from within the design space itself. Some of these will be familiar as they are sub-menus found within the main menus for the editor. Each of the main menus also have a defined pop-up key for quick access from within the design space, see [Accelerator Keys](https://www.altium.com/documentation/altium-designer/shortcut-keys#accelerator-keys) for more information.

|  |  |
| --- | --- |
| **B** | the **Toolbars**sub-menu |
| **K** | the **Panels** sub-menu (from the main **View**menu). |
| **M** | a pop-up menu of move commands. |
| **S** | the **Select**sub-menu (from the main **Edit** menu). |
| **Z** | a pop-up menu with zoom commands |

7.[Multi-board Editor Shortcuts](https://www.altium.com/documentation/altium-designer/shortcut-keys)

| SHORTCUT | DESCRIPTION | MORE INFORMATION... |
| --- | --- | --- |
| **Ctrl+P** | Print the currently active multi-board schematic document | [Command Page](https://www.altium.com/documentation/altium-designer/pcbdrawing-cmd-printprint-ad) |
| **Ctrl+Z** (or**Alt+Backspace**) | Restore the active Multi-board document to its state previous to the last operation. Multiple levels of undo are supported | [Command Page](https://www.altium.com/documentation/altium-designer/pcbdrawing-cmd-undoundo-ad) |
| **Ctrl+Y** (or**Ctrl+Backspace**) | Restore changes made by the Undo feature in the active Multi-board document. Multiple levels of redo are supported | [Command Page](https://www.altium.com/documentation/altium-designer/pcbdrawing-cmd-redoredo-ad) |
| **Ctrl+X** | Clear the selected object(s) from the active Multi-board document and copy them to the Multi-board Editor's clipboard | [Command Page](https://www.altium.com/documentation/altium-designer/pcbdrawing-cmd-cutcut-ad) |
| **Ctrl+C** | Copy the selected object(s) from the active Multi-board document to the Multi-board Editor's clipboard | [Command Page](https://www.altium.com/documentation/altium-designer/pcbdrawing-cmd-copycopy-ad) |
| **Ctrl+V** | Place the last content cut/copied to the Multi-board Editor's clipboard into any open Multi-board document | [Command Page](https://www.altium.com/documentation/altium-designer/pcbdrawing-cmd-pastepaste-ad) |
| **Ctrl+E** | Enter editing mode for the currently selected part (PCB) in the active Multi-board Assembly document | [Command Page](https://www.altium.com/documentation/altium-designer/multiboardassembly-cmd-editselectedparteditselectedpart-ad) |
| **Ctrl+K** | Check for collisions between the various constituent entities of the multi-board assembly | [Command Page](https://www.altium.com/documentation/altium-designer/multiboardassembly-cmd-checkcollisioncheckcollision-ad) |
| **Ctrl+M** | Measure distances between 3D bodies in the active Multi-board Assembly document | [Command Page](https://www.altium.com/documentation/altium-designer/multiboardassembly-cmd-measuredistance3dmeasuredistance3d-ad) |
| **Delete** | Remove the selected object(s) from the active Multi-board document | [Command Page](https://www.altium.com/documentation/altium-designer/pcbdrawing-cmd-deleteselecteddrawingitemsdelete-ad) |
| **Q** | Toggle between the use of Imperial and Metric measurement units for the active Multi-board document | [Command Page](https://www.altium.com/documentation/altium-designer/pcbdrawing-cmd-togglemeasurementunitstogglemeasurementunits-ad) |
| **Shift+E** | Toggle the snapping functionality on or off for the active Multi-board document | [Command Page](https://www.altium.com/documentation/altium-designer/pcbdrawing-cmd-togglesnappingtogglesnapping-ad) |
| **F11** | Toggle the display of the [Properties](https://www.altium.com/documentation/altium-designer/properties-panel-draftsman) panel accordingly | [Command Page](https://www.altium.com/documentation/altium-designer/interactiveproperties-cmd-displayinteractivepropertiespaneldisplayinteractivepropertiespanel-ad) |

8. [CAM Editor Shortcuts](https://www.altium.com/documentation/altium-designer/shortcut-keys)

| SHORTCUT | DESCRIPTION | MORE INFORMATION... |
| --- | --- | --- |
| **Ctrl+Z** (or **Alt+Backspace**) | Restore the current document to its state previous to the last operation. Multiple levels of undo are supported | [Command Page](https://www.altium.com/documentation/altium-designer/camtastic-cmd-editundoeditundo-ad) |
| **Ctrl+Y** (or **Ctrl+Backspace**) | Restore changes made by the Undo feature in the current document. Multiple levels of Redo are supported | [Command Page](https://www.altium.com/documentation/altium-designer/camtastic-cmd-editredoeditredo-ad) |
| **Ctrl+X** | Clear the selected object(s) from the current document and copy them to the CAMtastic Editor's clipboard | [Command Page](https://www.altium.com/documentation/altium-designer/camtastic-cmd-editcuteditcut-ad) |
| **Ctrl+C** (or **Ctrl+Insert**) | Copy the selected object(s) from the current document to the CAMtastic Editor's clipboard | [Command Page](https://www.altium.com/documentation/altium-designer/camtastic-cmd-editcopyeditcopy-ad) |
| **Ctrl+V** (or **Shift+Insert**) | Paste the contents of the CAMtastic Editor's internal clipboard into a CAM document | [Command Page](https://www.altium.com/documentation/altium-designer/camtastic-cmd-editpasteeditpaste-ad) |
| **Ctrl+M** | Mirror selected objects about a defined mirror line on the current document | [Command Page](https://www.altium.com/documentation/altium-designer/camtastic-cmd-editmirroreditmirror-ad) |
| **Ctrl+R** | Rotate selected objects about a specified rotation 'base' point in the current document | [Command Page](https://www.altium.com/documentation/altium-designer/camtastic-cmd-editrotateeditrotate-ad) |
| **Ctrl+L** | Align layers in the current document based on chosen layer objects | [Command Page](https://www.altium.com/documentation/altium-designer/camtastic-cmd-editaligneditalign-ad) |
| **L** | Create a new layer by merging existing layers in the current document | [Command Page](https://www.altium.com/documentation/altium-designer/camtastic-cmd-buildmergebuildmerge-ad) |
| **Alt+C** | Select design objects within a user-defined area. The objects do not have to be wholly inside the defined boundaries of the area | [Command Page](https://www.altium.com/documentation/altium-designer/camtastic-cmd-selmenucrossselmenucross-ad) |
| **Alt+P** | Reselect the objects that were part of the previous selection | [Command Page](https://www.altium.com/documentation/altium-designer/camtastic-cmd-selmenuprevselmenuprev-ad) |
| **Alt+O** | Select objects in the workspace one at a time | [Command Page](https://www.altium.com/documentation/altium-designer/camtastic-cmd-selmenuentityselmenuentity-ad) |
| **Ctrl+F** | Toggle the Flash/Point selection mode On or OFF | [Command Page](https://www.altium.com/documentation/altium-designer/camtastic-cmd-selmenuflashselmenuflash-ad) |
| **Ctrl+T** | Toggle the Draws selection mode On or OFF | [Command Page](https://www.altium.com/documentation/altium-designer/camtastic-cmd-selmenutraceselmenutrace-ad) |
| **Ctrl+A** | Copy selected objects and place them in a defined array within the current document | [Command Page](https://www.altium.com/documentation/altium-designer/camtastic-cmd-editarrayeditarray-ad) |
| **Ctrl+D** | Change the shape and/or layer for selected objects in the current document | [Command Page](https://www.altium.com/documentation/altium-designer/camtastic-cmd-editmodifyeditmodify-ad) |
| **Ctrl+I** | Set a new origin point for the current document | [Command Page](https://www.altium.com/documentation/altium-designer/camtastic-cmd-editoriginusereditoriginuser-ad) |
| **Ctrl+U** | Scale selected objects in the current document based on a specified base point and scale factor | [Command Page](https://www.altium.com/documentation/altium-designer/camtastic-cmd-editscaleeditscale-ad) |
| **Home** | Display all objects on all enabled layers in the current document | [Command Page](https://www.altium.com/documentation/altium-designer/camtastic-cmd-zoomallzoomall-ad) |
| **Shift+P** | Redefine the display area about a specified point in the current document | [Command Page](https://www.altium.com/documentation/altium-designer/camtastic-cmd-zoompanzoompan-ad) |
| **PgUp** | Bring the design closer to you relative to the cursor position in the current document | [Command Page](https://www.altium.com/documentation/altium-designer/camtastic-cmd-zoominzoomin-ad) |
| **PgDn** | Move the design away from you relative to the cursor position in the current document | [Command Page](https://www.altium.com/documentation/altium-designer/camtastic-cmd-zoomoutzoomout-ad) |
| **Shift+V** | Return the display to the previous view of the screen in the current document | [Command Page](https://www.altium.com/documentation/altium-designer/camtastic-cmd-zoompreviouszoomprevious-ad) |
| **End** | Refresh the workspace, in effect performing a redraw of the current document to remove any undesired drawing update effects | [Command Page](https://www.altium.com/documentation/altium-designer/camtastic-cmd-redrawredraw-ad) |
| **D** | Control the panning action in the current document dynamically using the mouse | [Command Page](https://www.altium.com/documentation/altium-designer/camtastic-cmd-zoomdynamicpanzoomdynamicpan-ad) |
| **Alt+Home** | View and highlight all objects drawn with the current Dcode | [Command Page](https://www.altium.com/documentation/altium-designer/camtastic-cmd-zoomdcodezoomdcode-ad) |
| **Shift+E** | Toggle the display of the Extents Box. This box is used to display the drawing extents within the main design workspace | [Command Page](https://www.altium.com/documentation/altium-designer/camtastic-cmd-viewextentsviewextents-ad) |
| **Shift+F** | Toggle the view of the current document between active fill and outline sketch modes | [Command Page](https://www.altium.com/documentation/altium-designer/camtastic-cmd-viewfillviewfill-ad) |
| **Shift+H** | Toggle the highlighting of objects based on the current Dcode. In this way, you can interrogate exactly wit which objects a particular Dcode is associated | [Command Page](https://www.altium.com/documentation/altium-designer/camtastic-cmd-highlightallhighlightall-ad) |
| **N** | Toggle the display of the current document between Negative and Standard views | [Command Page](https://www.altium.com/documentation/altium-designer/camtastic-cmd-drawnegativedrawnegative-ad) |
| **Shift+T** | Toggle translucent display mode ON or OFF. This mode displays objects in the current document with a transparent effect, allowing you to see objects partially or entirely overlapped by other objects | [Command Page](https://www.altium.com/documentation/altium-designer/camtastic-cmd-viewtranslucentviewtranslucent-ad) |
| **Shift+B** | Toggle the display of the current film box | [Command Page](https://www.altium.com/documentation/altium-designer/camtastic-cmd-viewboundboxviewboundbox-ad) |
| **Shift+G** | Access the **CAM Editor - Drawing Modes** page of the [Preferences dialog](https://www.altium.com/documentation/altium-designer/client-dlg-alloptionsformpreferences-ad) | - |
| **Shift+Y** | Converts a closed polyline to a solid polygon | - |
| **Shift+Z** | Converts a closed polyline to a CutOut polygon | - |
| **Shift+X** | Converts a closed polyline to a solid polygon region | - |
| **Q** | Obtain information with respect to a single selected object in the current document | [Command Page](https://www.altium.com/documentation/altium-designer/camtastic-cmd-infoqueryinfoquery-ad) |
| **Shift+N** | Obtain information with respect to a single selected net in the current document | [Command Page](https://www.altium.com/documentation/altium-designer/camtastic-cmd-infoquerynetinfoquerynet-ad) |
| **Shift+M** | Measure and display the distance between any two points in the current document | [Command Page](https://www.altium.com/documentation/altium-designer/camtastic-cmd-infomeasureinfomeasure-ad) |
| **Shift+A** | Access the Edit Apertures dialog in which you can create new aperture shapes and edit, or delete, existing ones | [Command Page](https://www.altium.com/documentation/altium-designer/camtastic-cmd-editapertureseditapertures-ad) |
| **K** | Access the Set Layers OFF dialog in which you can specify which layers you wish to turn OFF | [Command Page](https://www.altium.com/documentation/altium-designer/camtastic-cmd-viewsetlayoffviewsetlayoff-ad) |
| **Alt+K** | Access the Set Layers ON dialog in which you can specify which layers you wish to turn ON | [Command Page](https://www.altium.com/documentation/altium-designer/camtastic-cmd-viewsetlayonviewsetlayon-ad) |
| **Shift+S** | Toggle between the Off snap mode and the previously selected snap mode (other than Off) | [Command Page](https://www.altium.com/documentation/altium-designer/camtastic-cmd-snapchangesnapchange-ad) |
| **Esc** | Cancel the current command  If the current command uses a dialog at any stage, this feature will only cancel out of the dialog. You will need to use the feature again to completely abort the command. | [Command Page](https://www.altium.com/documentation/altium-designer/camtastic-cmd-escapebreakescapebreak-ad) |
| **Ctrl+Home** | Change the display of the main design window to show the current film box and its contents | [Command Page](https://www.altium.com/documentation/altium-designer/camtastic-cmd-zoomfilmboxzoomfilmbox-ad) |
| **Ctrl+Delete** (or **Ctrl+E**) | Delete selected objects from the current document | [Command Page](https://www.altium.com/documentation/altium-designer/camtastic-cmd-editeraseediterase-ad) |
| **+** (on numeric keypad) | Change the current layer for the design to the next layer in the Layers list | [Command Page](https://www.altium.com/documentation/altium-designer/camtastic-cmd-setcurrentlayerexsetcurrentlayerex-ad#cycle_to_next_layer) |
| **-** (on numeric keypad) | Change the current layer for the design to the previous layer in the Layers list | [Command Page](https://www.altium.com/documentation/altium-designer/camtastic-cmd-setcurrentlayerexsetcurrentlayerex-ad#cycle_to_previous_layer) |
| **\*** (on numeric keypad) | Change the current layer for the design to the next signal layer in the Layers list | [Command Page](https://www.altium.com/documentation/altium-designer/camtastic-cmd-setcurrentlayerexsetcurrentlayerex-ad#cycle_to_next_signal_layer) |
| **Shift+F9** | Execute the current function after all required objects involved in the function have been selected | [Command Page](https://www.altium.com/documentation/altium-designer/camtastic-cmd-executefunctionexecutefunction-ad) |
| **Shift+F4** | Presents all open design documents in their own individually-tiled regions within the main application design window | - |
| **Shift+Ctrl+R** | Repeat the previous placement/editing command | [Command Page](https://www.altium.com/documentation/altium-designer/camtastic-cmd-repeatfunctionrepeatfunction-ad) |
| **Ctrl+G** | Set the X (horizontal) and Y (vertical) step values - for the Snap Grid - simultaneously to a chosen value | [Command Page](https://www.altium.com/documentation/altium-designer/camtastic-cmd-snapgridsnapgrid-ad#set_snap_grid) |

##### **Sub-Menu Shortcuts**

| SHORTCUT | GIVES ACCESS TO... |
| --- | --- |
| **C** | the **Circle**sub-menu |
| **B** | the **Toolbars**sub-menu |
| **G** | a pop-up menu with snap grid commands |
| **I** | the **Import** sub-menu |
| **O** | the **Objects**sub-menu |
| **S** | the **Selection**sub-menu |
| **X** | the **Export**sub-menu |
| **Z** | a pop-up menu with zoom commands |

9. [SimData Editor Shortcuts](https://www.altium.com/documentation/altium-designer/shortcut-keys)

[SimData Editor Shortcuts](https://www.altium.com/documentation/altium-designer/shortcut-keys)Click to Collapse

| SHORTCUT | DESCRIPTION | MORE INFORMATION... |
| --- | --- | --- |
| **Ctrl+X** (or **Shift+Delete**) | Clear the selected waveform from the Waveform Analysis window and copy it to the SimData Editor's internal clipboard | [Command Page](https://www.altium.com/documentation/altium-designer/simview-cmd-cutcut-ad) |
| **Ctrl+C** | Copy the selected waveform from the Waveform Analysis window to the SimData Editor's internal clipboard | [Command Page](https://www.altium.com/documentation/altium-designer/simview-cmd-copycopy-ad) |
| **Ctrl+Z** | Used to restore the active SimData Editor to its state previous to the last operation | - |
| **Ctrl+Y** | Used to restore changes made by the Undo feature, in the active SimData Editor | - |
| **Ctrl+V** (or **Shift+Insert**) | Place the current contents of the SimData Editor's internal clipboard into a new or existing wave plot of the current chart in the Waveform Analysis window | [Command Page](https://www.altium.com/documentation/altium-designer/simview-cmd-pastepaste-ad) |
| **Shift+Ctrl+C** | Used to clear any filtering that is currently applied to the active document | - |
| **Ctrl+A** | Use to select all contents in the SimData Editor |  |
| **Ctrl+H** | Use to quickly find specific or partial text in accordance with defined search options | - |
| **F3** | Use to find the next occurrence of the last text search that was specified | - |
| **Delete** | Clear all waveforms from the active wave plot in the Waveform Analysis window | [Command Page](https://www.altium.com/documentation/altium-designer/simview-cmd-clearclear-ad) |
| **Ctrl+PgDn** | Fit all waveforms in the current chart, in their entirety within the Waveform Analysis window | [Command Page](https://www.altium.com/documentation/altium-designer/simview-cmd-fitwaveformsfitwaveforms-ad) |
| **End** | Refresh the screen, in effect performing a redraw of the active simulation analysis chart to remove any undesired drawing update effects | [Command Page](https://www.altium.com/documentation/altium-designer/simview-cmd-zoomzoom-ad#refresh) |
| **Esc** (or **Shift+C**) | Clear the existing waveform filter that is currently being applied in the active analysis chart | [Command Page](https://www.altium.com/documentation/altium-designer/simview-cmd-clearfilterclearfilter-ad) |
| **Up Arrow** | Scroll vertically-upwards through the data in the active simulation analysis chart one wave plot at a time | [Command Page](https://www.altium.com/documentation/altium-designer/simview-cmd-scrollscroll-ad#scroll_data_vertically_up_by_plot) |
| **Down Arrow** | Scroll vertically-downwards through the data in the active simulation analysis chart one wave plot at a time | [Command Page](https://www.altium.com/documentation/altium-designer/simview-cmd-scrollscroll-ad#scroll_data_vertically_down_by_plot) |
| **Left Arrow** | Scroll horizontally-left through the data in the active simulation analysis chart one major X-axis division at a time | [Command Page](https://www.altium.com/documentation/altium-designer/simview-cmd-scrollscroll-ad#scroll_data_horizontally_left_by_major_x_axis_division) |
| **Right Arrow** | Scroll horizontally-right through the data in the active simulation analysis chart one major X-axis division at a time | [Command Page](https://www.altium.com/documentation/altium-designer/simview-cmd-scrollscroll-ad#scroll_data_horizontally_right_by_major_x_axis_division) |
| **Shift+Up Arrow** | Scroll vertically-upwards through the data in the active simulation analysis chart one page at a time | [Command Page](https://www.altium.com/documentation/altium-designer/simview-cmd-scrollscroll-ad#scroll_data_vertically_up_by_page) |
| **Shift+Down Arrow** | Scroll vertically-downwards through the data in the active simulation analysis chart one page at a time | [Command Page](https://www.altium.com/documentation/altium-designer/simview-cmd-scrollscroll-ad#scroll_data_vertically_down_by_page) |
| **Shift+Left Arrow** | Scroll horizontally-left through the data in the active simulation analysis chart one page at a time | [Command Page](https://www.altium.com/documentation/altium-designer/simview-cmd-scrollscroll-ad#scroll_data_horizontally_left_by_page) |
| **Shift+Right Arrow** | Scroll horizontally-right through the data in the active simulation analysis chart one page at a time | [Command Page](https://www.altium.com/documentation/altium-designer/simview-cmd-scrollscroll-ad#scroll_data_horizontally_right_by_page) |
| **Ctrl+Up Arrow (or Ctrl+Home)** | Scroll to the top of the active simulation analysis chart | [Command Page](https://www.altium.com/documentation/altium-designer/simview-cmd-scrollscroll-ad#scroll_data_vertically_to_top) |
| **Ctrl+Down Arrow (or Ctrl+End)** | Scroll to the bottom of the active simulation analysis chart | [Command Page](https://www.altium.com/documentation/altium-designer/simview-cmd-scrollscroll-ad#scroll_data_vertically_to_bottom) |
| **Ctrl+Left Arrow** | Scroll to the start of the X-axis in the active simulation analysis chart | [Command Page](https://www.altium.com/documentation/altium-designer/simview-cmd-scrollscroll-ad#scroll_data_horizontally_to_start_of_x_axis) |
| **Ctrl+Right Arrow** | Scroll to the end of the X-axis in the active simulation analysis chart | [Command Page](https://www.altium.com/documentation/altium-designer/simview-cmd-scrollscroll-ad#scroll_data_horizontally_to_end_of_x_axis) |
| **+**(on numeric keypad) | Make the next available analysis chart the current chart in the Waveform Analysis window | [Command Page](https://www.altium.com/documentation/altium-designer/simview-cmd-setcurrentanalysissetcurrentanalysis-ad#next_analysis_chart) |
| **-**(on numeric keypad) | Make the previous available analysis chart the current chart in the Waveform Analysis window | [Command Page](https://www.altium.com/documentation/altium-designer/simview-cmd-setcurrentanalysissetcurrentanalysis-ad#previous_analysis_chart) |
| **PgUp** | Bring the waveform(s) closer to you relative to the cursor position in the active simulation analysis chart | [Command Page](https://www.altium.com/documentation/altium-designer/simview-cmd-zoomzoom-ad#zoom_in_at_cursor) |
| **PgDn** | Move the waveform(s) away from you relative to the cursor position in the active simulation analysis chart | [Command Page](https://www.altium.com/documentation/altium-designer/simview-cmd-zoomzoom-ad#zoom_out_at_cursor) |

11. [Text-based Document Editor Shortcuts](https://www.altium.com/documentation/altium-designer/shortcut-keys)

|  |  |  |
| --- | --- | --- |
| **Ctrl+Z** | Restore a document to its state previous to the last operation. Multiple levels of undo are supported | [Command Page](https://www.altium.com/documentation/altium-designer/textedit-cmd-undoundo-ad#undo) |
| **Ctrl+Y** | Restore changes made by the Undo feature. Multiple levels of redo are supported | [Command Page](https://www.altium.com/documentation/altium-designer/textedit-cmd-redoredo-ad#redo) |
| **Ctrl+X** | Remove all selected text from the current document. A copy of the selection is placed on the standard Windows clipboard | [Command Page](https://www.altium.com/documentation/altium-designer/textedit-cmd-cutcut-ad#cut) |
| **Ctrl+C** | Copy all selected text from the current document to the standard Windows clipboard | [Command Page](https://www.altium.com/documentation/altium-designer/textedit-cmd-copycopy-ad#copy) |
| **Ctrl+V** | Place the current contents of the standard Windows clipboard into the active text-based document | [Command Page](https://www.altium.com/documentation/altium-designer/textedit-cmd-pastepaste-ad#paste) |
| **Shift+Ctrl+C** | Clear any filtering that is currently applied to the active document | [Command Page](https://www.altium.com/documentation/altium-designer/textedit-cmd-clearbrowsermarksclearbrowsermarks-ad#clear_browser_marks) |
| **Ctrl+F** | Access the [Find Text dialog](https://www.altium.com/documentation/altium-designer/textedit-dlg-findrepldlgformfind-replace-text-ad) in which you can configure a search for specific text located in the current document, all text-based documents in the active project, all open text-based documents, or all text-based documents in a specified directory | [Command Page](https://www.altium.com/documentation/altium-designer/textedit-cmd-findfind-ad#find_text) |
| **Ctrl+H** | Access the [Replace Text dialog](https://www.altium.com/documentation/altium-designer/textedit-dlg-findrepldlgformfind-replace-text-ad) in which you can configure a search to locate and replace specific text - located in the current document, all text-based documents in the active project, all open text-based documents, or all text-based documents in a specified directory | [Command Page](https://www.altium.com/documentation/altium-designer/textedit-cmd-findandreplacefindandreplace-ad#find_and_replace_text) |
| **F3** | Find the next occurrence of the last text search that was specified using the [Find Text dialog](https://www.altium.com/documentation/altium-designer/textedit-dlg-findrepldlgformfind-replace-text-ad) | [Command Page](https://www.altium.com/documentation/altium-designer/textedit-cmd-findnextfindnext-ad#find_next_instance_of_searched_text) |
| **Shift+Ctrl+F** | Find the next occurrence of the currently selected text in the active document | [Command Page](https://www.altium.com/documentation/altium-designer/textedit-cmd-findnextselectedfindnextselected-ad#find_next_instance_of_selected_text) |
| **Ctrl+A** | Select all of the text in the current document | [Command Page](https://www.altium.com/documentation/altium-designer/textedit-cmd-selectallselectall-ad#select_all) |
| **Delete** | Delete a single character to the right of the cursor  If one or more characters in the document are selected, the command will delete the selection. | [Command Page](https://www.altium.com/documentation/altium-designer/textedit-cmd-deletecharacterdeletecharacter-ad#delete_character_to_right) |
| **Backspace** | Delete a single character to the left of the cursor  If one or more characters in the document are selected, the command will delete the selection. | [Command Page](https://www.altium.com/documentation/altium-designer/textedit-cmd-deletecharactertoleftdeletecharactertoleft-ad#delete_character_to_left) |
| **Ctrl+Backspace** | Delete all characters back to the beginning of the first word immediately to the left of the current cursor position | [Command Page](https://www.altium.com/documentation/altium-designer/textedit-cmd-deletetobeginningofworddeletetobeginningofword-ad#delete_to_beginning_of_word) |
| **Ctrl+Q+Y** | Delete all characters from the current cursor position to the end of the current line | [Command Page](https://www.altium.com/documentation/altium-designer/textedit-cmd-deletetoendoflinedeletetoendofline-ad#delete_to_end_of_line) |
| **Ctrl+T** | Delete all characters up to the beginning of the first word immediately to the right of the current cursor position | [Command Page](https://www.altium.com/documentation/altium-designer/textedit-cmd-deletetoendofworddeletetoendofword-ad#delete_to_end_of_word) |
| **Alt+Shift+PgDn** | Extend a column selection by one page down from the current cursor position | [Command Page](https://www.altium.com/documentation/altium-designer/textedit-cmd-extendselectioncolumnonepagedownextendselectioncolumnonepagedown-ad#extend_selection_column_one_page_down) |
| **Alt+Shift+PgUp** | Extend a column selection by one page up from the current cursor position | [Command Page](https://www.altium.com/documentation/altium-designer/textedit-cmd-extendselectioncolumnonepageupextendselectioncolumnonepageup-ad#extend_selection_column_one_page_up) |
| **Alt+Shift+Ctrl+Left Arrow** | Extend a column selection by one word to the left of the current cursor position | [Command Page](https://www.altium.com/documentation/altium-designer/textedit-cmd-extendselectioncolumnonewordleftextendselectioncolumnonewordleft-ad#extend_selection_column_one_word_left) |
| **Alt+Shift+Ctrl+Right Arrow** | Extend a column selection by one word to the right of the current cursor position | [Command Page](https://www.altium.com/documentation/altium-designer/textedit-cmd-extendselectioncolumnonewordrightextendselectioncolumnonewordright-ad#extend_selection_column_one_word_right) |
| **Alt+Shift+Ctrl+PgDn** | Extend a column selection to the bottom of the current document window from the current cursor position | [Command Page](https://www.altium.com/documentation/altium-designer/textedit-cmd-extendselectioncolumntobottomofwindowextendselectioncolumntobottomofwindow-ad#extend_selection_column_to_bottom_of_window) |
| **Alt+Shift+Ctrl+Home** | Extend a column selection to the beginning of the current document from the current cursor position | [Command Page](https://www.altium.com/documentation/altium-designer/textedit-cmd-extendselectioncolumntofilebeginextendselectioncolumntofilebegin-ad#extend_selection_column_to_beginning_of_file) |
| **Alt+Shift+Ctrl+End** | Extend a column selection to the end of the current document from the current cursor position | [Command Page](https://www.altium.com/documentation/altium-designer/textedit-cmd-extendselectioncolumntofileendextendselectioncolumntofileend-ad#extend_selection_column_to_end_of_file) |
| **Alt+Shift+Home** | Extend a column selection to the beginning of the current line from the current cursor position | [Command Page](https://www.altium.com/documentation/altium-designer/textedit-cmd-extendselectioncolumntolinebeginextendselectioncolumntolinebegin-ad#extend_selection_column_to_beginning_of_line) |
| **Alt+Shift+End** | Extend a column selection to the end of the current line from the current cursor position | [Command Page](https://www.altium.com/documentation/altium-designer/textedit-cmd-extendselectioncolumntolineendextendselectioncolumntolineend-ad#extend_selection_column_to_end_of_line) |
| **Alt+Shift+Ctrl+PgUp** | Extend a column selection to the top of the current document window from the current cursor position | [Command Page](https://www.altium.com/documentation/altium-designer/textedit-cmd-extendselectioncolumntotopofwindowextendselectioncolumntotopofwindow-ad#extend_selection_column_to_top_of_window) |
| **Shift+Left Arrow** | Extend a selection by one character to the left of the current cursor position | [Command Page](https://www.altium.com/documentation/altium-designer/textedit-cmd-extendselectionleftonecharacterextendselectionleftonecharacter-ad#extend_selection_left_one_character) |
| **Shift+Ctrl+Left Arrow** | Extend a selection by one word to the left of the current cursor position | [Command Page](https://www.altium.com/documentation/altium-designer/textedit-cmd-extendselectionleftonewordextendselectionleftoneword-ad#extend_selection_left_one_word) |
| **Alt+Shift+Down Arrow** | Extend a column selection by one line down from the current cursor position | [Command Page](https://www.altium.com/documentation/altium-designer/textedit-cmd-extendselectiononecolumndownextendselectiononecolumndown-ad#extend_selection_column_one_line_down) |
| **Alt+Shift+Left Arrow** | Extend a selection column by one column to the left of the current cursor position | [Command Page](https://www.altium.com/documentation/altium-designer/textedit-cmd-extendselectiononecolumnleftextendselectiononecolumnleft-ad#extend_selection_one_column_left) |
| **Alt+Shift+Right Arrow** | Extend a selection column by one column to the right of the current cursor position | [Command Page](https://www.altium.com/documentation/altium-designer/textedit-cmd-extendselectiononecolumnrightextendselectiononecolumnright-ad#extend_selection_one_column_right) |
| **Alt+Shift+Up Arrow** | Extend a column selection by one line up from the current cursor position | [Command Page](https://www.altium.com/documentation/altium-designer/textedit-cmd-extendselectiononecolumnupextendselectiononecolumnup-ad#extend_selection_column_one_line_up) |
| **Shift+PgDn** | Extend a selection by one page down from the current cursor position | [Command Page](https://www.altium.com/documentation/altium-designer/textedit-cmd-extendselectiononepagedownextendselectiononepagedown-ad#extend_selection_one_page_down) |
| **Shift+Right Arrow** | Extend a selection by one character to the right of the current cursor position | [Command Page](https://www.altium.com/documentation/altium-designer/textedit-cmd-extendselectionrightonecharacterextendselectionrightonecharacter-ad#extend_selection_right_one_character) |
| **Shift+Ctrl+Right Arrow** | Extend a selection by one word to the right of the current cursor position | [Command Page](https://www.altium.com/documentation/altium-designer/textedit-cmd-extendselectionrightonewordextendselectionrightoneword-ad#extend_selection_right_one_word) |
| **Shift+Ctrl+Home** | Extend a selection to the beginning of the current document from the current cursor position | [Command Page](https://www.altium.com/documentation/altium-designer/textedit-cmd-extendselectiontobeginningoffileextendselectiontobeginningoffile-ad#extend_selection_to_beginning_of_file) |
| **Shift+Home** | Extend a selection to the beginning of the current line from the current cursor position | [Command Page](https://www.altium.com/documentation/altium-designer/textedit-cmd-extendselectiontobeginningoflineextendselectiontobeginningofline-ad#extend_selection_to_beginning_of_line) |
| **Shift+Ctrl+PgDn** | Extend a selection to the bottom of the current document window from the current cursor position | [Command Page](https://www.altium.com/documentation/altium-designer/textedit-cmd-extendselectiontobottomofwindowextendselectiontobottomofwindow-ad#extend_selection_to_bottom_of_window) |
| **Shift+Ctrl+End** | Extend a selection to the end of the current document from the current cursor position | [Command Page](https://www.altium.com/documentation/altium-designer/textedit-cmd-extendselectiontoendoffileextendselectiontoendoffile-ad#extend_selection_to_end_of_file) |
| **Shift+End** | Extend a selection to the end of the current line from the current cursor position | [Command Page](https://www.altium.com/documentation/altium-designer/textedit-cmd-extendselectiontoendoflineextendselectiontoendofline-ad#extend_selection_to_end_of_line) |
| **Shift+PgUp** | Extend a selection by one page up from the current cursor position | [Command Page](https://www.altium.com/documentation/altium-designer/textedit-cmd-extendselectiontoonepageupextendselectiontoonepageup-ad#extend_selection_one_page_up) |
| **Shift+Down Arrow** | Extend a selection by one line down from the current cursor position | [Command Page](https://www.altium.com/documentation/altium-designer/textedit-cmd-extendselectiontosamecolumnonnextlineextendselectiontosamecolumnonnextline-ad#extend_selection_one_line_down) |
| **Shift+Up Arrow** | Extend a selection by one line up from the current cursor position | [Command Page](https://www.altium.com/documentation/altium-designer/textedit-cmd-extendselectiontosamecolumnonpreviouslineextendselectiontosamecolumnonpreviousline-ad#extend_selection_one_line_up) |
| **Shift+Ctrl+PgUp** | Extend a selection to the top of the current document window from the current cursor position | [Command Page](https://www.altium.com/documentation/altium-designer/textedit-cmd-extendselectiontotopofwindowextendselectiontotopofwindow-ad#extend_selection_to_top_of_window) |
| **Alt+Click&Drag** | Full control of column selection | - |
| **Click&Drag** | Full control of text selection | - |
| **Ctrl+N** | Insert a new line at the current cursor position | [Command Page](https://www.altium.com/documentation/altium-designer/textedit-cmd-insertlineinsertline-ad#insert_new_line) |
| **Enter** | Insert a carriage return at the current cursor position | [Command Page](https://www.altium.com/documentation/altium-designer/textedit-cmd-insertreturninsertreturn-ad#insert_carriage_return) |
| **Tab** | Insert a tab at the current cursor position | [Command Page](https://www.altium.com/documentation/altium-designer/textedit-cmd-inserttabinserttab-ad#insert_tab) |
| **Shift+Tab** | Tab backwards from the current cursor position | [Command Page](https://www.altium.com/documentation/altium-designer/textedit-cmd-insertshifttabinsertshifttab-ad#tab_backwards) |
| **Down Arrow** | Move the text cursor downwards one line at a time | [Command Page](https://www.altium.com/documentation/altium-designer/textedit-cmd-movecursordownmovecursordown-ad#move_cursor_down) |
| **Left Arrow** | Move the text cursor to the left in the current line and by one character at a time | [Command Page](https://www.altium.com/documentation/altium-designer/textedit-cmd-movecursorleftmovecursorleft-ad#move_cursor_left) |
| **Ctrl+Left Arrow** | Move the text cursor to the left one word at a time | [Command Page](https://www.altium.com/documentation/altium-designer/textedit-cmd-movecursorleftonewordmovecursorleftoneword-ad#move_cursor_left_one_word) |
| **Right Arrow** | Move the text cursor to the right one character at a time | [Command Page](https://www.altium.com/documentation/altium-designer/textedit-cmd-movecursorrightmovecursorright-ad#move_cursor_right) |
| **Ctrl+Right Arrow** | Move the text cursor to the right one word at a time | [Command Page](https://www.altium.com/documentation/altium-designer/textedit-cmd-movecursorrightonewordmovecursorrightoneword-ad#move_cursor_right_one_word) |
| **Home** | Move the text cursor to the beginning of the current line | [Command Page](https://www.altium.com/documentation/altium-designer/textedit-cmd-movecursortobeginningoflinemovecursortobeginningofline-ad#move_cursor_to_beginning_of_line) |
| **Ctrl+End** | Move the text cursor to the end of the file | [Command Page](https://www.altium.com/documentation/altium-designer/textedit-cmd-movecursortobottomofdocumentmovecursortobottomofdocument-ad#move_cursor_to_end_of_document) |
| **Ctrl+PgDn** | Move the text cursor to the bottom of the document window | [Command Page](https://www.altium.com/documentation/altium-designer/textedit-cmd-movecursortobottomofwindowmovecursortobottomofwindow-ad#move_cursor_to_bottom_of_window) |
| **End** | Move the text cursor to the end of the current line | [Command Page](https://www.altium.com/documentation/altium-designer/textedit-cmd-movecursortoendoflinemovecursortoendofline-ad#move_cursor_to_end_of_line) |
| **Ctrl+Home** | Move the text cursor to the beginning of the file | [Command Page](https://www.altium.com/documentation/altium-designer/textedit-cmd-movecursortotopofdocumentmovecursortotopofdocument-ad#move_cursor_to_start_of_document) |
| **Ctrl+PgUp** | Move the text cursor to the top of the document window | [Command Page](https://www.altium.com/documentation/altium-designer/textedit-cmd-movecursortotopofwindowmovecursortotopofwindow-ad#move_cursor_to_top_of_window) |
| **Up Arrow** | Move the text cursor upwards one line at a time | [Command Page](https://www.altium.com/documentation/altium-designer/textedit-cmd-movecursorupmovecursorup-ad#move_cursor_up) |
| **Ctrl+Enter** | Automatically open a document that the cursor is currently over  The feature will only open documents that are stored in the same location on the hard disk as the text document making the call.  The feature will only open documents that have no spaces in their filenames. For example, Example\_Design\_File.SchDoc and ExampleDesignFile.SchDoc will both open using this feature, but Example Design File.SchDoc will not. | [Command Page](https://www.altium.com/documentation/altium-designer/textedit-cmd-opendocumentundercursoropendocumentundercursor-ad#open_document_under_cursor) |
| **PgDn** | Scroll down one page | [Command Page](https://www.altium.com/documentation/altium-designer/textedit-cmd-pagedownpagedown-ad#page_down) |
| **PgUp** | Scroll up one page | [Command Page](https://www.altium.com/documentation/altium-designer/textedit-cmd-pageuppageup-ad#page_up) |
| **Ctrl+Down Arrow** | Scroll down one line | [Command Page](https://www.altium.com/documentation/altium-designer/textedit-cmd-scrolldownonelinescrolldownoneline-ad#scroll_down_one_line) |
| **Ctrl+Up Arrow** | Scroll up one line | [Command Page](https://www.altium.com/documentation/altium-designer/textedit-cmd-scrolluponelinescrolluponeline-ad#scroll_up_one_line) |
| **Insert** | Toggle the text mode between Insert and Overwrite | [Command Page](https://www.altium.com/documentation/altium-designer/textedit-cmd-toggleinsertmodetoggleinsertmode-ad#toggle_insert_mode) |
| **Shift+Ctrl+n** (**n**=1-9) | Store the current location of the text cursor on the current document into location marker **n**  A new location assigned to the location marker will overwrite a previously defined location. | [Command Page](https://www.altium.com/documentation/altium-designer/textedit-cmd-setlocationmarksetlocationmark-ad#set_location_mark_n) |
| **Ctrl+n** (**n**=1-9) | Move the text cursor to a predefined location on the current document, as stored in location marker **n**  If the applicable location marker has not been set, the text cursor will remain at its current location. | [Command Page](https://www.altium.com/documentation/altium-designer/textedit-cmd-jumplocationmarkjumplocationmark-ad#jump_location_mark_n) |

##### **Scripting Document Shortcuts**

The following additional shortcuts are available when working with scripting text-based documents:

| SHORTCUT | DESCRIPTION | MORE INFORMATION... |
| --- | --- | --- |
| **F9** | Run the current script. If a run script is not defined, nominate a startup script procedure to execute in the the Select Item to Run dialog | - |
| **Ctrl+F9** | Run the current script up to the line where the text cursor is currently positioned, then pause | - |
| **F5** | Toggle an enabled breakpoint for the current line  Clicking in the gutter will also toggle the breakpoint for the current line. | - |
| **Ctrl+F7** | Open the script Evaluate dialog to see the current value (result) of the expression located at the cursor  This command is only available while the script is running or being debugged (stepped through). | - |
| **F7** | Step into and execute the next line of code. Use to execute the current script one line at a time | - |
| **F8** | Step over (execute without stopping) a called procedure. If the line statement is not a called procedure, step into and execute the line as normal | - |
| **Ctrl+F3** | Halt (stop) a currently running script procedure | - |
| **Ctrl+F5** | Access the Add New Watch dialog in which you can define a new watch expression for the current script | - |
| **Ctrl+Click** on variable/method | Jump the cursor to the respective variable/method declaration point | - |
| **Ctrl+J** | Pop-up the Statement Templates list window | - |
| **Shift+Ctrl+Space** | Activate the Method Parameters pop-up window | - |
| **Alt+Ctrl+B** | Open the Breakpoints panel | - |
| **Alt+Ctrl+E** | Open the Code Explorer panel | - |
| **Alt+Ctrl+I** | Open the Object Inspector panel | - |
| **Alt+Ctrl+P** | Open the Tool Palette panel | - |
| **Alt+Ctrl+S** | Open the Call Stack panel | - |
| **Alt+Ctrl+W** | Open the Watch List panel | - |
| **F12** | Toggle between the **Code**and **Form**view for the active script document | - |

12. [Schematic Keyboard Sequences](https://www.altium.com/documentation/altium-designer/shortcut-keys)

|  |  |
| --- | --- |
| **V, D** | Fit document |
| **V, F** | Fit board |
| **S, Y** | Select all on current layer |
| **X, A** | Deselect all |
| **O, D** | Access the **View Options** tab of the [View Configuration panel](https://www.altium.com/documentation/altium-designer/view-configuration-panel-pcb) |
| **G, G, 5, Enter** | Set default Global Board Snap Grid to 5mil (both **Step X** and **Step Y**) when the board measurement units are Imperial.  **G, G** accesses the [Snap Grid (1..1000) dialog](https://www.altium.com/documentation/altium-designer/pcb-dlg-inputcoorddialog-coordeditsnap-grid1-1000-ad). Depending on the grid size required and the current measurement units, change the sequence after this and before the final **Enter** as needed. |
| **T, M** | Reset error markers |
| **D, R** | Access the [PCB Rules and Constraints Editor dialog](https://www.altium.com/documentation/altium-designer/pcb-dlg-form-designrulespcb-rules-and-constraints-editor-ad) |
| **T, G, M** | Access the [Polygon Pour Manager dialog](https://www.altium.com/documentation/altium-designer/pcb-dlg-polygonmanagerformpolygon-pour-manager-ad) |
| **T, D, R** | Run Batch Design Rule Check |
| **P, S** | Place string |
| **P, T** | Start interactive routing |
| **M, M** | Move object |
| **M+X** | Opens the *Get X-Y Offsets* dialog to move the current selection by a specified distance in the horizontal (X) and/or vertical (Y) planes. |
| **T, P** | Access the [PCB Editor - General page](https://www.altium.com/documentation/altium-designer/pcb-editor-general-preferences) of the [Preferences dialog](https://www.altium.com/documentation/altium-designer/client-dlg-alloptionsformpreferences-ad) |

13. [Draftsman Keyboard Sequences](https://www.altium.com/documentation/altium-designer/shortcut-keys)

|  |  |
| --- | --- |
| **V, D** | Fit document |
| **V, F** | Fit selected sheet |
| **P, S** | Place Board Assembly View |
| **P, V, E** | Place Board Realistic View |
| **T, I** | Import changes from the PCB document |
| **T, A** | Add new sheet |
| **M, S** | Move selected object(s) |

14. [PCB Keyboard Sequences](https://www.altium.com/documentation/altium-designer/shortcut-keys)

|  |  |
| --- | --- |
| **V, D** | Fit document |
| **V, F** | Fit board |
| **S, Y** | Select all on current layer |
| **X, A** | Deselect all |
| **O, D** | Access the **View Options** tab of the [View Configuration panel](https://www.altium.com/documentation/altium-designer/view-configuration-panel-pcb) |
| **G, G, 5, Enter** | Set default Global Board Snap Grid to 5mil (both **Step X** and **Step Y**) when the board measurement units are Imperial.  **G, G** accesses the [Snap Grid (1..1000) dialog](https://www.altium.com/documentation/altium-designer/pcb-dlg-inputcoorddialog-coordeditsnap-grid1-1000-ad). Depending on the grid size required and the current measurement units, change the sequence after this and before the final **Enter** as needed. |
| **T, M** | Reset error markers |
| **D, R** | Access the [PCB Rules and Constraints Editor dialog](https://www.altium.com/documentation/altium-designer/pcb-dlg-form-designrulespcb-rules-and-constraints-editor-ad) |
| **T, G, M** | Access the [Polygon Pour Manager dialog](https://www.altium.com/documentation/altium-designer/pcb-dlg-polygonmanagerformpolygon-pour-manager-ad) |
| **T, D, R** | Run Batch Design Rule Check |
| **P, S** | Place string |
| **P, T** | Start interactive routing |
| **M, M** | Move object |
| **M+X** | Opens the *Get X-Y Offsets* dialog to move the current selection by a specified distance in the horizontal (X) and/or vertical (Y) planes. |
| **T, P** | Access the [PCB Editor - General page](https://www.altium.com/documentation/altium-designer/pcb-editor-general-preferences) of the [Preferences dialog](https://www.altium.com/documentation/altium-designer/client-dlg-alloptionsformpreferences-ad) |

15.The following table summarizes the accelerator and pop-up keys assigned to various main menus available across editors:

The following table summarizes the accelerator and pop-up keys assigned to various main menus available across editors:

| MENU | ACCELERATOR KEY | POPUP KEY |
| --- | --- | --- |
| **File** | F | F |
| **Edit** | E | E |
| **View** | V | V |
| **Project** | C | C |
| **Place** | P | P |
| **Design** | D | D |
| **Tools** | T | T |
| **Route** | U | U |
| **Reports** | R | R |
| **Window** | W | W |
| **Help** | H | H |

Link: https://www.altium.com/documentation/altium-designer/shortcut-keys.