## SOLIDWORKS Default Keyboard Shortcuts

<table>
<thead>
<tr>
<th>Category</th>
<th>Command</th>
<th>Shortcut(s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>File</td>
<td>New</td>
<td>Ctrl+N</td>
</tr>
<tr>
<td></td>
<td>Open</td>
<td>Ctrl+O</td>
</tr>
<tr>
<td></td>
<td>Browse Recent Documents</td>
<td>Ctrl</td>
</tr>
<tr>
<td></td>
<td>Close</td>
<td>Ctrl+W</td>
</tr>
<tr>
<td></td>
<td>Save</td>
<td>Ctrl+S</td>
</tr>
<tr>
<td></td>
<td>Print</td>
<td>Ctrl+P</td>
</tr>
<tr>
<td>Edit</td>
<td>Undo</td>
<td>Ctrl+Z</td>
</tr>
<tr>
<td></td>
<td>Redo</td>
<td>Ctrl+Y</td>
</tr>
<tr>
<td></td>
<td>Repeat Last Command</td>
<td>Enter</td>
</tr>
<tr>
<td></td>
<td>Select All</td>
<td>Ctrl+A</td>
</tr>
<tr>
<td></td>
<td>Cut</td>
<td>Ctrl+X</td>
</tr>
<tr>
<td></td>
<td>Copy</td>
<td>Ctrl+C</td>
</tr>
<tr>
<td></td>
<td>Paste</td>
<td>Ctrl+V</td>
</tr>
<tr>
<td></td>
<td>Delete</td>
<td>Ctrl+D</td>
</tr>
<tr>
<td>Edit</td>
<td>Rebuild</td>
<td>Ctrl+B</td>
</tr>
<tr>
<td></td>
<td>Rebuild All Configurations</td>
<td>Ctrl+Shift+B</td>
</tr>
<tr>
<td></td>
<td>Copy Appearance</td>
<td>Ctrl+Shift+C</td>
</tr>
<tr>
<td></td>
<td>Paste Appearance</td>
<td>Ctrl+Shift+V</td>
</tr>
<tr>
<td>View</td>
<td>Redraw</td>
<td>Ctrl+R</td>
</tr>
<tr>
<td>View</td>
<td>Orientation</td>
<td>F2 (SpaceBar)</td>
</tr>
<tr>
<td>View</td>
<td>Zoom to Fit</td>
<td>F1</td>
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<tr>
<td>View</td>
<td>Previous View</td>
<td>Ctrl+Shift-Z</td>
</tr>
<tr>
<td>View</td>
<td>Quick Snap</td>
<td>F3</td>
</tr>
<tr>
<td>View</td>
<td>FeatureManager Tree Area</td>
<td>F9</td>
</tr>
<tr>
<td>View</td>
<td>Toolbars</td>
<td>F10</td>
</tr>
<tr>
<td>View</td>
<td>Tab Pane</td>
<td>Ctrl+1</td>
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<tr>
<td>View</td>
<td>Full Screen</td>
<td>F11</td>
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<tr>
<td>Tools</td>
<td>Magnified Selection</td>
<td>G</td>
</tr>
<tr>
<td>Tools</td>
<td>Select over Geometry</td>
<td>T</td>
</tr>
<tr>
<td>Tools</td>
<td>Select All</td>
<td>Ctrl+A</td>
</tr>
<tr>
<td>Tools</td>
<td>Line</td>
<td>Ctrl+L</td>
</tr>
<tr>
<td>Help</td>
<td>Welcome to SOLIDWORKS</td>
<td>Ctrl+F2</td>
</tr>
<tr>
<td>Help</td>
<td>Help</td>
<td>H</td>
</tr>
<tr>
<td>Help</td>
<td>Commands</td>
<td>W</td>
</tr>
<tr>
<td>Help</td>
<td>Files and Models</td>
<td>I</td>
</tr>
<tr>
<td>Help</td>
<td>Knowledge Base</td>
<td>K</td>
</tr>
<tr>
<td>Help</td>
<td>Community Forums</td>
<td>O</td>
</tr>
<tr>
<td>Others</td>
<td>Front</td>
<td>Ctrl+1</td>
</tr>
<tr>
<td>Others</td>
<td>Back</td>
<td>Ctrl+2</td>
</tr>
<tr>
<td>Others</td>
<td>Left</td>
<td>Ctrl+3</td>
</tr>
</tbody>
</table>

### Others
- Right: Ctrl+4
- Top: Ctrl+5
- Bottom: Ctrl+6
- Symmetric: Ctrl+7
- Normal To: Ctrl+8
- Command option toggle: A
- Expand/Collapse Tree: Ctrl+C
- Filter Edges: E
- Find/Replace: Ctrl+F
- Next Edge: N
- Force Rebuild: Ctrl+Q
- Force Rebuild All Configurations: Ctrl+Shift+Q
- Shortcut Bar: S
- Show Flat Tree View: Ctrl+T
- Filter Vertices: V
- Scroll to FeatureManager tree bottom: End
- Toggle Notes Upper Case: Shift+F3
- Toggle Selection Filter Toolbar: F5
- Toggle Selection Filters: F6
- SpellChecker: F7
- Hide/Show Display Pane: F8
- Scroll to FeatureManager tree top: Home
- Next Command Manager Tab: Ctrl+Next
- Select Annotation View: SingleSheet
- Previous Command Manager Tab: Ctrl+Prior
- View Selector: Ctrl+SpaceBar
- Filter Faces: Ctrl+X
- Accept Edge: Y
- Zoom Out: Z
- Zoom In: Ctrl+Shift+Z
- Move Selection Breadcrumbs, Continuation Corner: D
- Hide the Hovered-over Component/Bodies: TAB
- Show the Hovered-over Component/Bodies: Shift+TAB
- Show all the Hidden Components/Bodies: Ctrl+Shift+TAB
- Select over Geometry: T
- Search SOLIDWORKS Help: H
- Search Commands: W
- Search Files and Models: I
- Search Knowledge Base: K
- Search Community Forums: O
**SOLIDWORKS DEFAULT MOUSE GESTURES**

4. Part
8. Sketch
12. Assembly

(New with SOLIDWORKS 2020)

**SOLIDWORKS REFERENCE TRIAD**

Select an axis
Select an axis that is normal to the screen
Shift + select
Ctrl + Shift + select
Alt + select
Ctrl + Alt + select

See the view normal to the screen.
Change the view direction 180 degrees.
Rotate 90 degrees about the axis.
Rotate 90 degrees in the opposite direction.
Rotate about the axis by the Arrow keys increment specified in Tools > Options > System Options > View.
Rotate in the opposite direction.

**SOLIDWORKS QUICK MENU ICONS**

### SELECTING A FEATURE

1. Edit Feature
2. Edit Sketch
3. Suppress
4. Rollback
5. Hide/Show
6. Zoom to Selection
7. Normal To
8. Appearances
9. Copy Appearance
10. Edit Sketch Plane

### SELECTING A SKETCH

2. 10
3. 4
5. 6
7
SOLIDWORKS SKETCH RELATIONS

- Coincident
- Pierce
- Horizontal
- Vertical
- Normal
- Perpendicular
- Parallel
- On Plane
- Parallel X
- Parallel Y
- Merge
- Midpoint
- Along X
- Along Y
- Along Z
- Collinear
- Intersection
- Fix
- Equal
- Coradial
- Tangent
- Concentric
- Equal Curve Length
- Symmetric

SOLIDWORKS ASSEMBLY MATES

- Angle
- Lock To Sketch
- Belt
- Parallel
- Cam
- Path Mate
- Coincident
- Perpendicular
- Concentric
- Profile Center
- Distance
- Rack and Pinion
- Gear
- Screw
- Hinge
- Slot
- In Place
- Symmetric
- Limit, Angle
- Tangent
- Limit, Distance
- Universal Joint
- Linear/Linear Coupler
- Width
- Lock

SOLIDWORKS PDM QUICK ICONS

<table>
<thead>
<tr>
<th>Status</th>
<th>Icon</th>
<th>Background Color</th>
</tr>
</thead>
<tbody>
<tr>
<td>Referencing a file version that is not the latest version</td>
<td>!</td>
<td>Red</td>
</tr>
<tr>
<td>The local file is older than the file in the vault</td>
<td></td>
<td></td>
</tr>
<tr>
<td>The local file is newer than the file in the vault</td>
<td></td>
<td></td>
</tr>
<tr>
<td>The file does not exist in the vault</td>
<td></td>
<td></td>
</tr>
<tr>
<td>The local file is of the same version as the file in the vault</td>
<td></td>
<td></td>
</tr>
<tr>
<td>The file is edited in SOLIDWORKS but not saved</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Checked in by another user during the current SOLIDWORKS session</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
SOLIDWORKS MIDDLE MOUSE BUTTON FUNCTIONS (MMB)

Zoom Out

Shift + MMB
Zoom In/Out

Zoom In

Ctrl + MMB
Pan

SUPPORT RESOURCES

GoEngineer.com
The direct source for product information, news, training and event information from GoEngineer.

YouTube
Explore hours of peer-reviewed technical videos to help you tackle your design and engineering challenges.

My.SOLIDWORKS.com
This SOLIDWORKS user portal allows you to interact with SOLIDWORKS' worldwide user base.

Support@goengineer.com
Email our dedicated support team and your information will be recorded in our case management system to best serve you. Include your screen captures and/or files and we will respond as soon as possible.