Getting Started:

Open a new part, this will be needed for the settings to appear later.

Go to the options gear and start by making a backup of the default settings so that you can bulk restore the default settings if need be in the future.



When to use each feature

Constant repetitive use: Keyboard shortcut Mouse Gesture Moderate use: Shortcut bar Custom Ribbon Seldom use: Command Search (default: W)

System Options

| Enabling sketch numeric input allows you to place dimension as you make features. | System Options - Sketch System Options Document Properties General Auto-rotate view normal to sketch plane on sketch creation and sketch edit Drawings Use fully defined sketches - bisplay Style Display arc centerpoints in part/assembly sketches - Area Hatch/Fill Display entity points in part/assembly sketches - Performance Prompt to close sketch - Relations/Snaps Create sketch on new part Display Override dimensions on drag/move Selection Display plane when shaded Performance ✓ Line length measured between virtual sharps in 3d Assemblies ✓ Enable spline tangency and curvature handles Default Templates Show spline control polygon by default File Locations ✓ Ghost image on drag Spin Box Increments ✓ Scale sketch on first dimension creation Backup/Recover ✓ Enable on screen numeric input on entity creation Touch ✓ Create dimension only when value is entered |
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| Changing the snap angle from within the system options allows you to more rapidly input angles without needing to eyeball it, making dimensioning more swift later. Since Solidworks currently lacks the ability to input angles numerically this is the closest to being able to do just that. | System Options - Relations/Snaps System Options Document Properties General Drawings - Display Style - Area Hatch/Fill - Performance Colors Sketch Display Selection Performance Assemblies External References Default Templates File Locations FeatureManager Spin Box Increments View Backup/Recover Touch Hole Wizard/Toolbox File Explorer Search Collaboration Messages/Errors/Warnings Import Export |
| Auto-Rotate normal to sketch is a simple feature that makes it so that when you start a sketch you always made normal to it, similar to other modeling softwares. Depending on what you're working on this feature can be a decent time saver. | System Options - Sketch System Options Decument Properties General Drawings - Display Style - Area Hatch/Fill - Performance Colors Sketch - Relations/Snaps Override dimensions on drag/move Sketch - Relations/Snaps Override dimensions on drag/move Sketch - Breaditions/Snaps Override dimensions on drag/move Sketch - Breaditions/Snaps Override dimensions on drag/move Sketch - Breaditions/Snaps Override dimensions on drag/move Sketch - Breaditons/Snaps Override dimensions on drag/move Sketch - Breadities - Line length measured between virtual sharps in 3d Assemblies - Show spline control polygon by default File Locations - FeatureManager - Show curvature comb bounding curve View View on screen numeric input on entity creation View Scale sketch on nity whe |

| Lasso selection allows you to be more specific with what you select within a complex sketch or assembly. | System Options - Selection System Options Document Properties General Drawings Display Style Area Hatch/Fill Performance Os asso Colors Selection of hidden edges Sketch Allow selection in wireframe and HLV modes Relations/Snaps Allow selection in wireframe and HLV modes Display Selection Performance Allow selection in through transparency Selection Enhance selection precision Assemblies Enternal References Default Templates File Locations FeatureManager Spin Box Increments View Backup/Recover Touch Hole Wizard/Toolbox Hie Esplorer Search Collaboration Messages/Errors/Warnings Import Export |
|--------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|----------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| The file locations allow to control where solidworks looks for the templates for you parts, drawings and assemblies. In conjunction with using the advanced tab on part creation you can have file defaults set up for parts, drawings and assemblies, such as the unit system drafting standard and precision. Naming the files is important so that you can tell them apart at a glance. | System Options - File Locations System Options Document Properties General Drawings Display Style Area Hatch/fill Performance Colors Sketch Relations/Snaps Display Selection Performance Assemblies External References Default Templates if ite Locations View Backup/Recover Touch Hole Wizard/Toolbox File Explorer Search Collaboration Messages/Errors/Warnings Import Export |
| Using the advanced solidworks new document option allows you to easily leverage custom templates for drawings, and different unit systems. | New SOLIDWORKS Document X Templates MBD Tutorial Part 18KB 5/3/201 Assembly 16KB 5/3/201 Drawing 14KB 5/3/201 Novice Tutorials OK Cancel Help |

You can access the advanced menu by clicking the New SOLIDWORKS Document X Advanced button on the bottom of the novice version. Drawing Part mbh a 3D representation of a single design component a 3D arrangement of parts and/or other assemblies a 2D engineering drawing, typically of a part or assembly SOLIDWORKS Tutorials OK Cancel Help Advanced Feature Manager Options allows you to change what appears on the left hand feature tree, allowing you to System Options Document Properties General Scroll selected item into view show/hide or make appear only when used (Automatic) Drawings Display Style Name feature on creation various features. Arrow key navigation Area Hatch/Fill Performance Dynamic highlight In addition you have the option to enable name feature Colors Use transparent flyout FeatureManager tree in parts/assemblies Sketch Enable FeatureManager tree filter Relations/Snaps on creation which prompts you to give features a name Allow component files to be renamed from FeatureManager tree Display Selection Enable preview of hidden components after you press the green checkmark. Performance Assemblies External References Default Templates File Locations Hide/show tree items FeatureManager Spin Box Increments Blocks Automatic 🗸 🗸 ΣEquations Show Backup/Recover Automatic 🗸 🗸 Material Show Design Binder Touck Hole Wizard/Toolbox Show Annotations Hide \sim Default Planes File Explorer Search Automatic ~ Show Solid Bodies 1_Origin Collaboration Messages/Errors/Warnings Mate References Automatic Automatic 🗸 🗸 Surface Bodies Import Export ~ Automatic Tables Automatic Design Table Automatic ~ Automatic Favorites Sensors V CeDrawing Markups Automatic ~ History Hide Selection Sets Graphic Bodies Automatic Automatic ~

Shortcut bar

A mode specific toolbar that will appear at the location of the cursor when the hotkey is pressed. While flyout bars can save space they are slower to use than adding a dedicated button. Any tool is fair game for this bar, you need only find it within the list on the customize tab. Sketch, and 3D sketch do however share the same shortcut bar.

Example Part Shortcut Bar:



Example Sketch Shortcut Bar:



Keyboard shortcuts

Allows use of any command within solidworks that can normally be used, but needing only a single keystroke, or combination of keys pressed to be performed.

Works well in conjunction with tools normally meant for gaming, such as MMO mice or Macro Keyboards. Helps reduce time spent clicking up and down through menus or ribbons to get a specific tool used frequently. The defaults are a good starting point, moving them such that they make sense to you is better overall.

Defaults (Windows basic Omitted)

R: Recent documents GUI Ctrl+Q: Force Rebuild W: Search Commands F5: Show/Hide Selection Filter Toolbar F6: Toggle Selection Filter Ctrl+1~7: Standard Views Ctrl+8: Normal To Shift+C: Collapse Tree S: Shortcuts Bar L: Line Enter: Repeat Last Command F: Fit model on screen Tab: Hide Hovered-over Component/Body Shift+Tab: Show Hovered-over Component/Body* does not remove the hide tag

Non-Editable

Shift+MouseWheel: Zoom in and out* Much quicker than scrolling Ctrl+MouseWheel: Pan

Suggested

- M: Measure D: Dimension
- X: Trim

Mouse Gestures

In the same way as everything else discussed in this document through the use of mouse gestures you can add up to 12 more commands on quick access.

Using more than 8 however will take a decent amount of practice due to the density.

Ribbon Bars

In addition to all the above options, you can also customize the ribbon bars within your solidworks custom file. You can add any features to any ribbon, or preferably build a custom ribbon that allows for improved workflow.

Useful Toolbars

Unique layouts are made for each of the 3 modes (Part/Assembly/Drawing) allowing for mode specific customization. The bars are not bound and will remember their position between instances even if that means being in the middle of the screen.

General

-Selection filter -Configuration

Assembly

-Parts -Drawing

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