SolidWorks
Tips & Tricks

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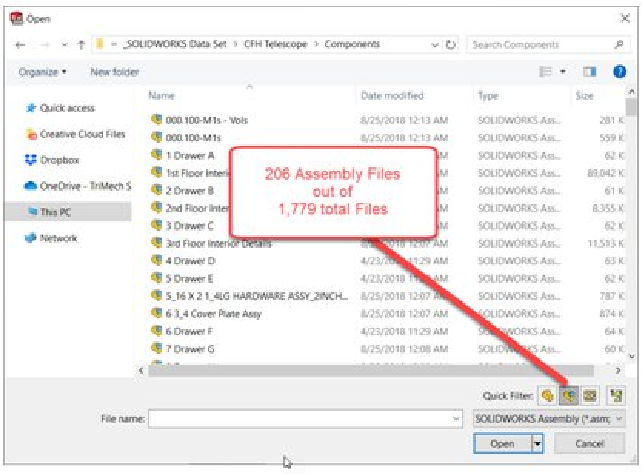
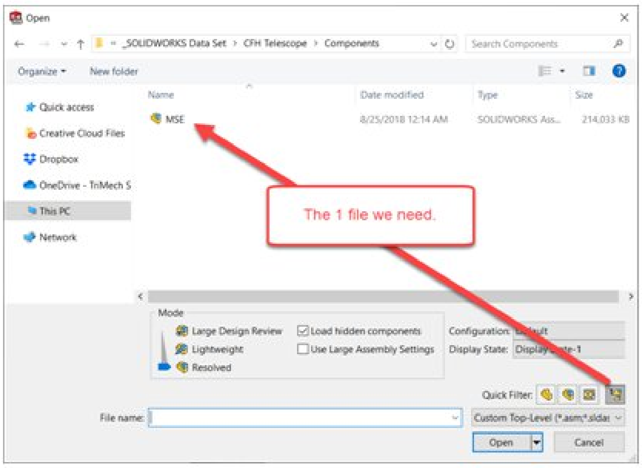
Tip 1: Filter Top Level Assembly—the easy button for opening files

The filter top-level assembly button was added to SOLIDWORKS in 2013 with the addition of Quick Filters. This has been my low-key personal favorite enhancement over the past decade.

The Quick Filters are the four buttons at the lower-right corner of the open dialog box that allows you to quickly filter for parts, assemblies, drawings and top-level assemblies. When you are searching for a file in a directory, the odds are you are going to want to open the top-level assembly. That's why I call this the easy button for opening files.





Sure, you could filter for assemblies and then sort by various things like the largest disc size or longest SW Open Time (See tip #2). However, that's two steps and plus the largest assembly is not guaranteed to be the top-level assembly because of virtual assemblies, simulation, and other information embedded in the file.

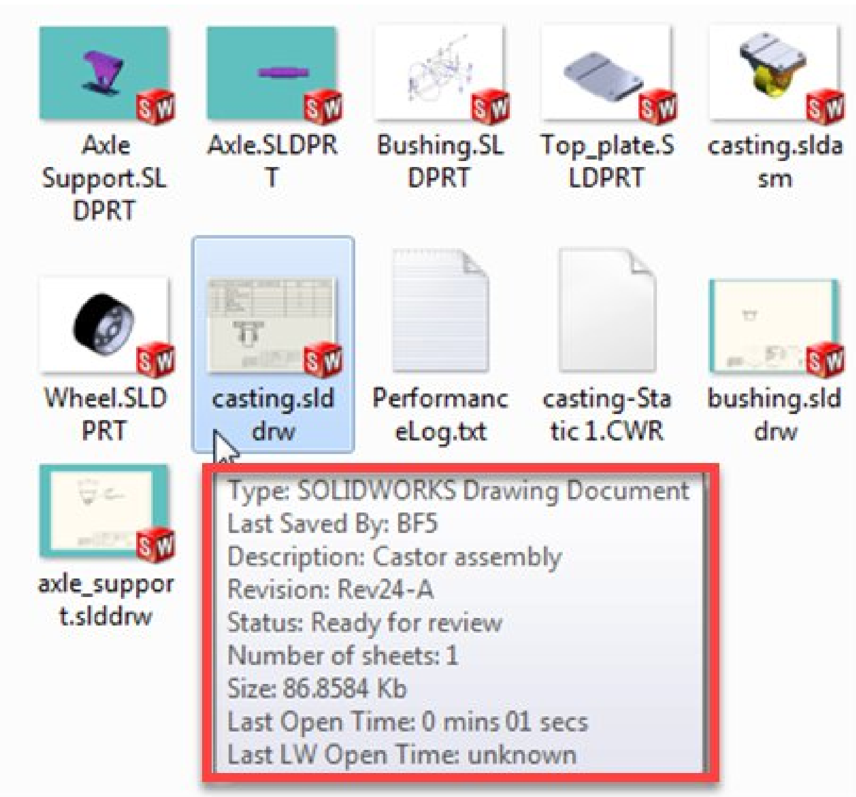
With the filter top-level assembly button on, you will see only top-level assemblies and no subassemblies. This is what I use to open a file a client sends me. By using the top-level assembly Quick Filters, I waste no time opening the correct file, so I can get to work right away.

<u>Tip</u> 2. Windows/File Explorer—Display key SOLIDWORKS information directly in File Explorer

As you browse through Windows Explorer, which was renamed File Explorer in Windows 10, you can learn key information about your files from the Tool Tip, thumbnail preview and even the details listed about the files. Here are three ways to make this work for you:

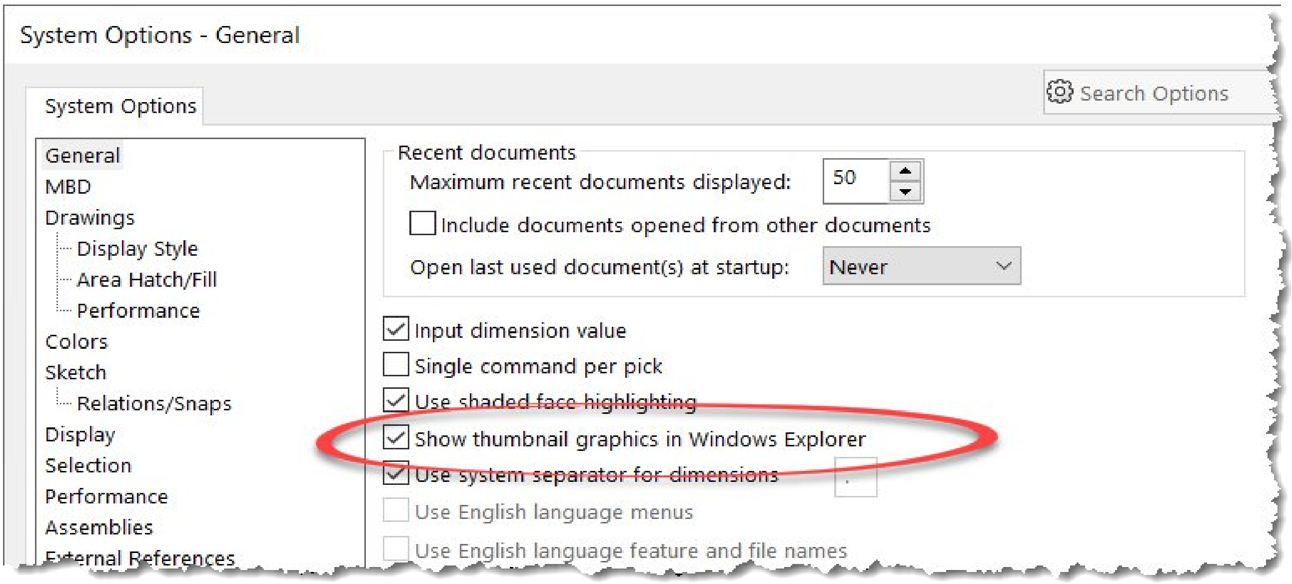
Tool Tip:

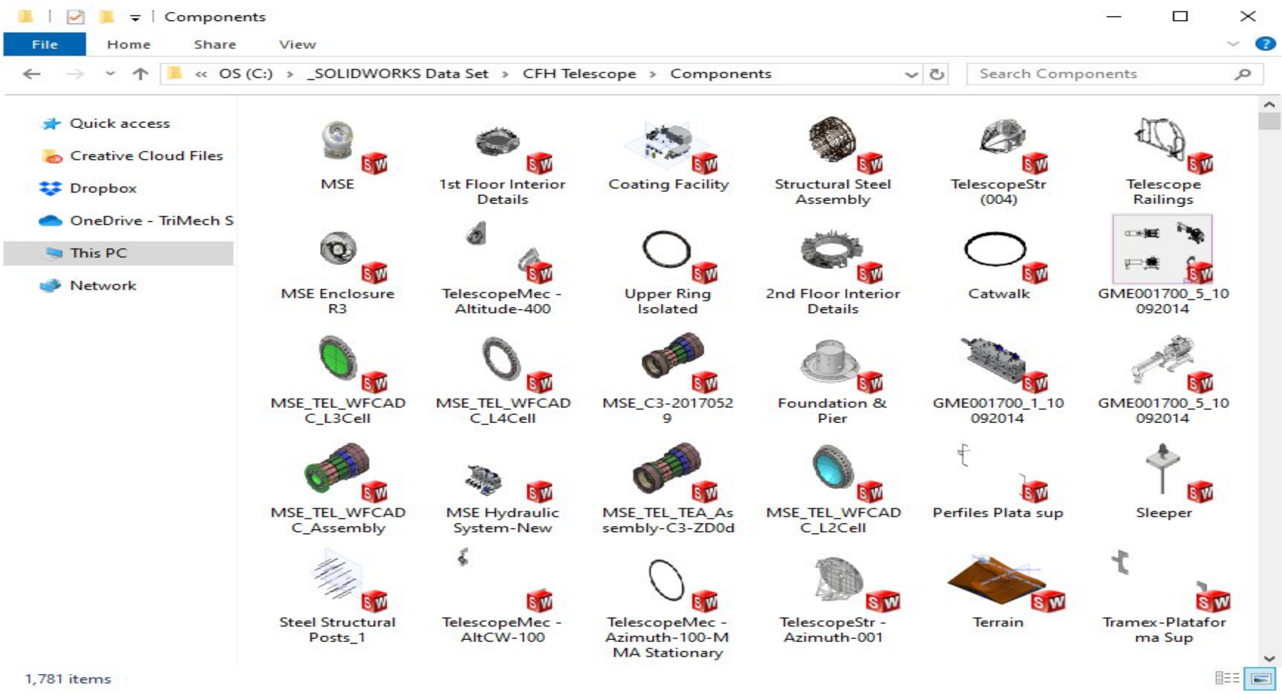
This one is straight forward—just hover your mouse over the file and you will see information such as custom properties, including Description, Revision, and Status, depending on what is relevant. You will also automatically see Type, Size, Last Saved By, Number of Sheets, Last Open Time, and Last LW Open Time. The Last Open Time indicates how long it took to open the file resolved, while the Last LW Open Time indicates how long it took to open the file in Lightweight mode.



Thumbnail Preview:

In File Explorer you can browse through your files while seeing a thumbnail preview of the file. This is an option that might not be turned on. If it's turned off, you will see the "LEGO block icons" of SOLIDWORKS files. To turn it on, check the box in the general system options called "Show thumbnail graphics in Windows Explorer." Then you can see the thumbnail preview of your model displayed in an isometric view, zoomed to fit, with a white background.

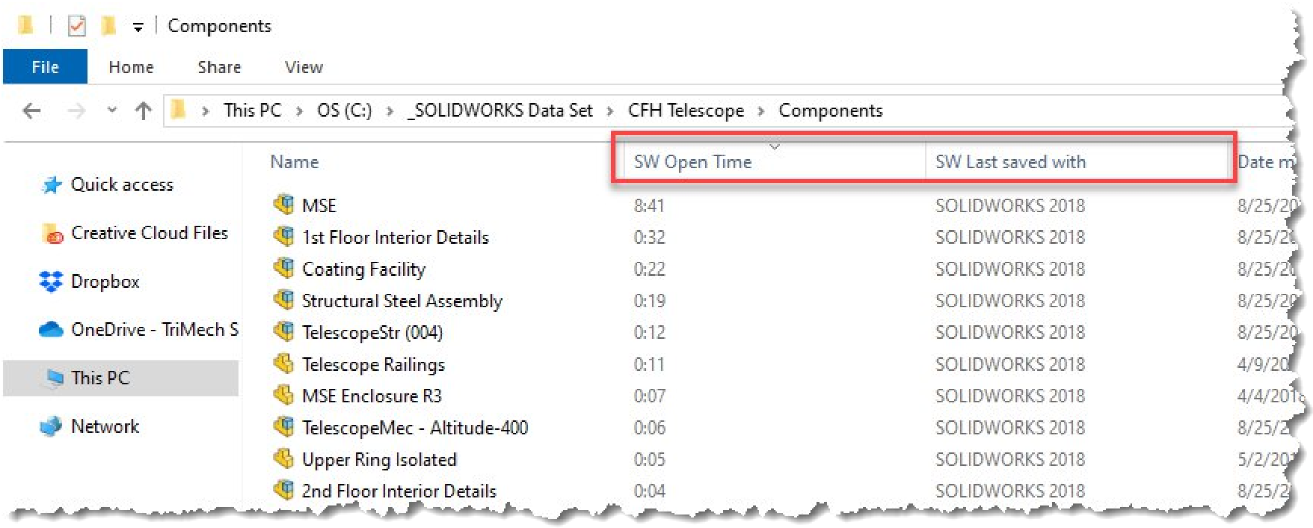


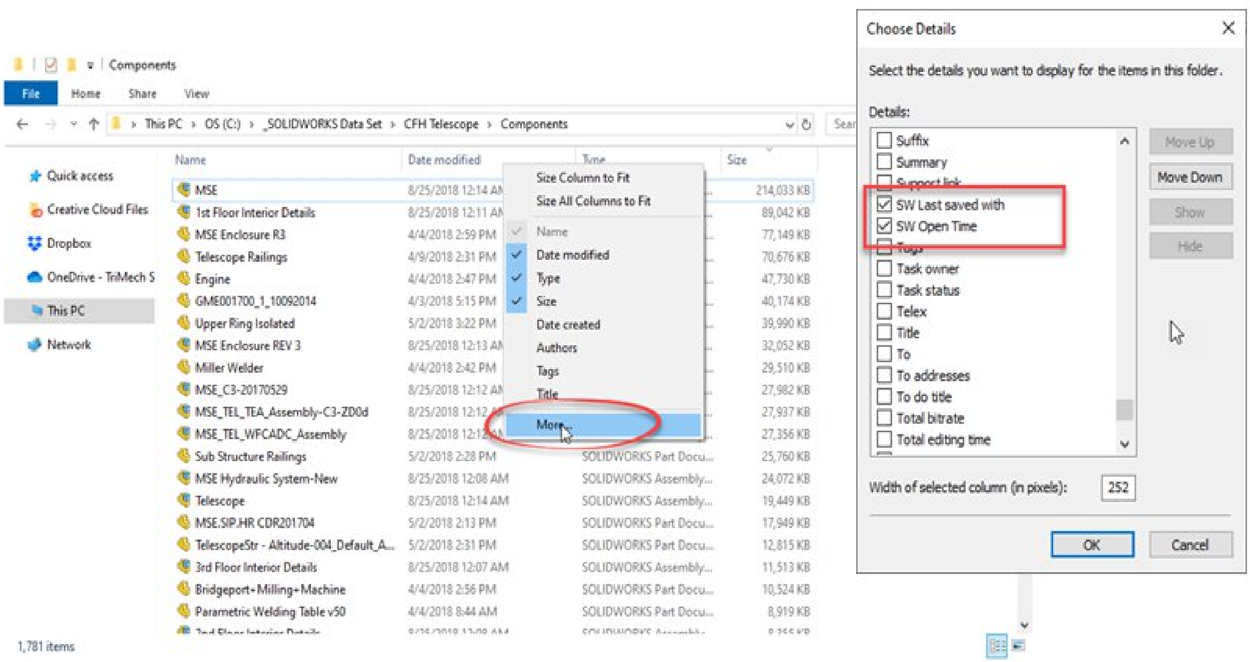


File Details:

In just a few steps, you can customize File Explorer to display details about the SOLIDWORKS files as header columns such as SW Open Time and SW Last Saved With in addition to standard Windows things like Name, Type and Size. You can then filter this information or just browse quickly through to understand key information about your files.

To turn this on, right-click on the Windows header and select more. Under details, you will check the box for SW Last saved with and SW Open Time. Now you can gain some valuable information about your files displayed in File Explorer.

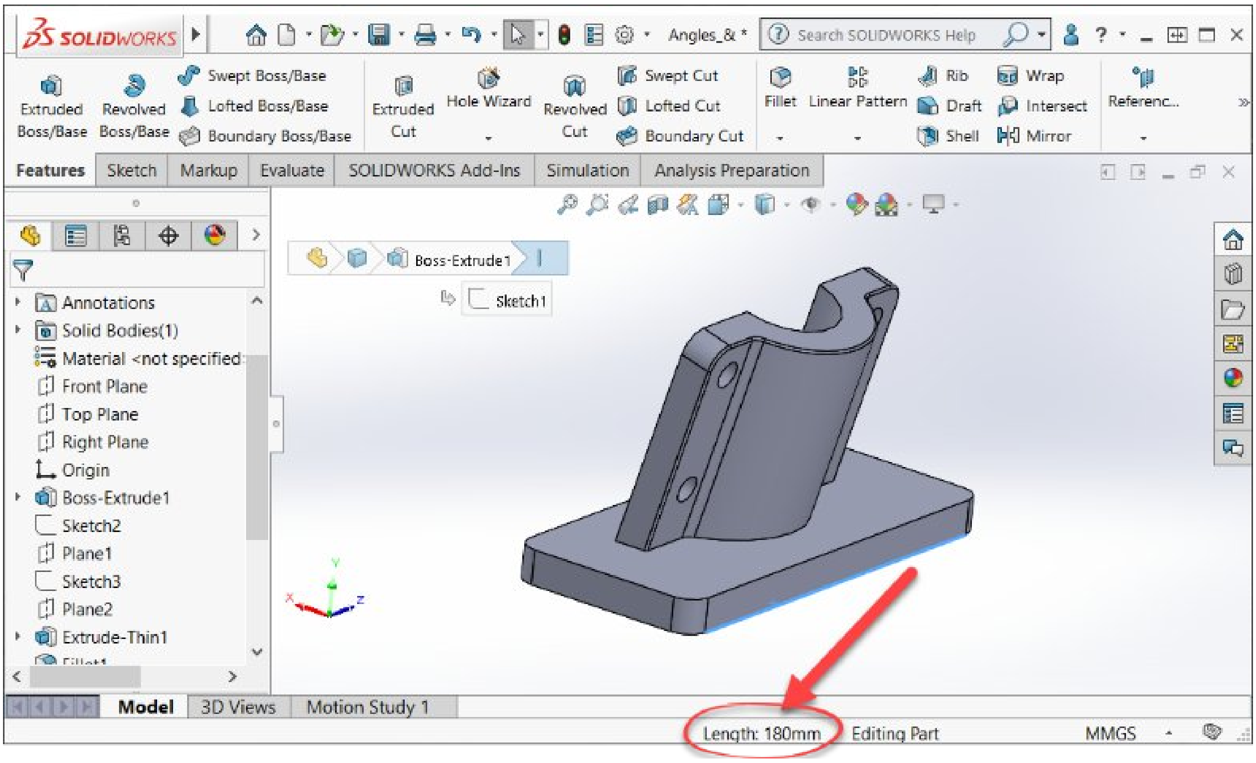


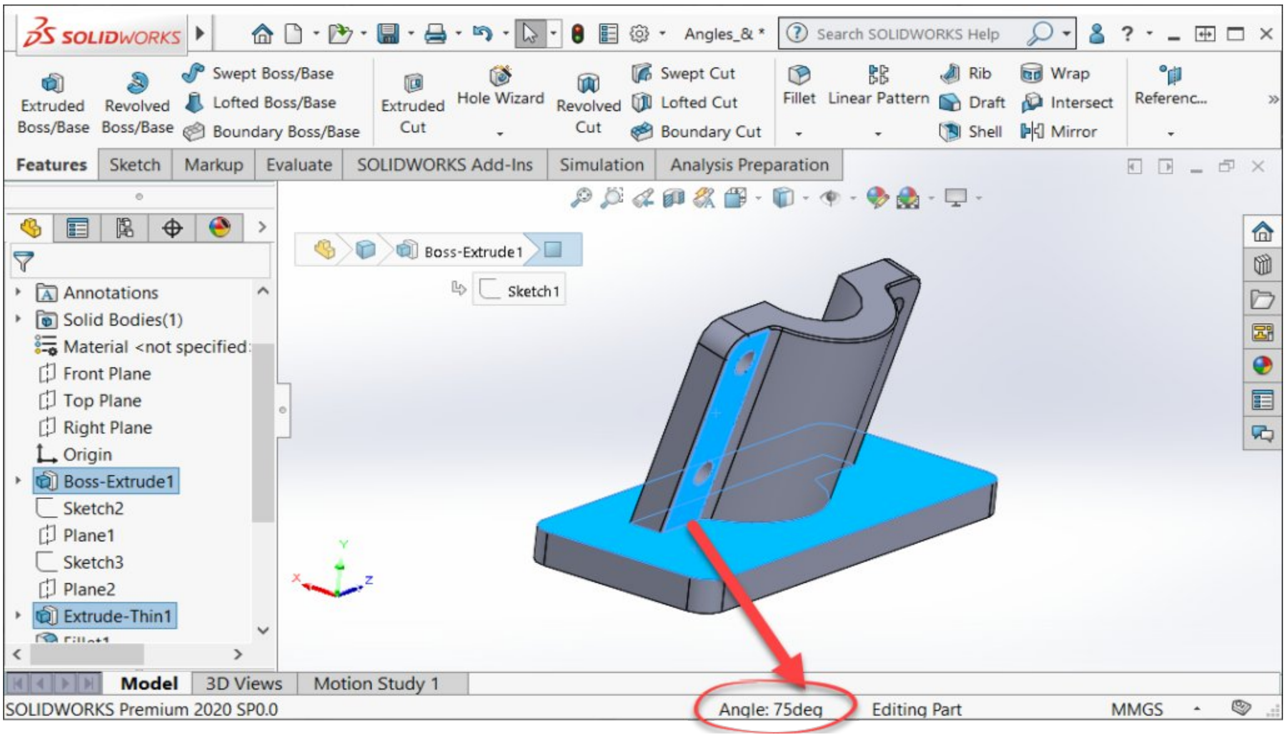


Tip 3. Status Bar—The quickest way to measure 14 ways in SOLIDWORKS

The status bar is the lower-right corner of the SOLIDWORKS interface and offers information about your model, but the most useful thing it provides is a quick way to take measurements of your model. Using the status bar, you can take measurements of your geometry such as:

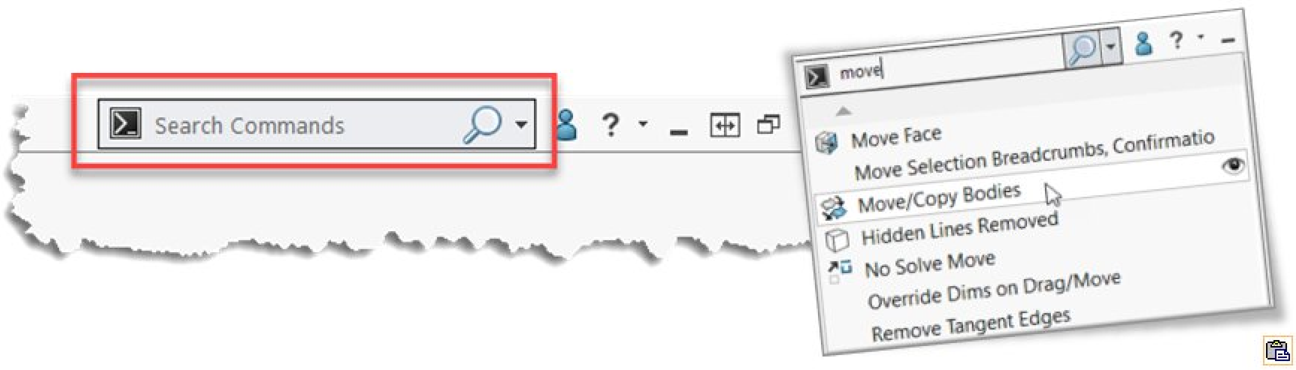
- Coordinates of a **Vertex**
- Normal distance between a **Vertex** and a **Line**
- Distance and delta X, Y, Z between two **Vertices**
- Length of an **Edge**
- Normal distance and total combined length of **Two Parallel Edges**
- Angle between **Two Non-Parallel Edges**
- Total length of **Multiple Edges**
- Radius and center of a **Circular Arc Edge**
- Diameter and center of a **Circular Edge**
- Normal distance between **Two Parallel Planar Faces**
- Angle between **Two Non–Parallel Planar Faces**
- Radius of a **Cylindrical Arc Face**
- Diameter of a **Cylindrical Face**
- Distance between axes of **Two Cylindrical Axes**





Tip 4. Search Commands—Your life line for finding commands

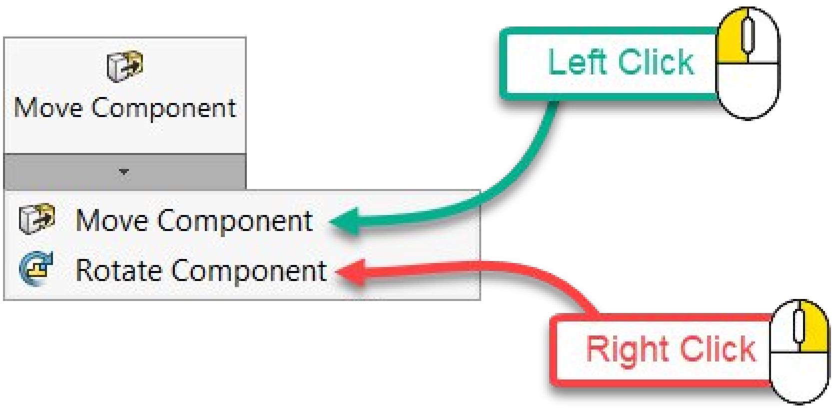
It wouldn't be a SOLIDWORKS tips blog written by me without this one. This is my all-time favorite SOLIDWORKS tip and I am not shy about sharing it. The Search Commands is the quickest way to find and launch any command you might be looking for. Instead of manually digging through the menus to find something like, for example, "Cosmetic Thread," you can go to the Search Commands and launch it that way. I use Search Commands daily.



To use Search Commands, just start typing in the upper-right corner of the screen, and SOLIDWORKS will begin to populate a list of commands. Click on the command to launch it or click on the eye icon to have SOLIDWORKS show you where the command resides. If you remember just one tip from this article, make sure it's this one.

Tip 5. Advance Move Components—Right-click to rotate components

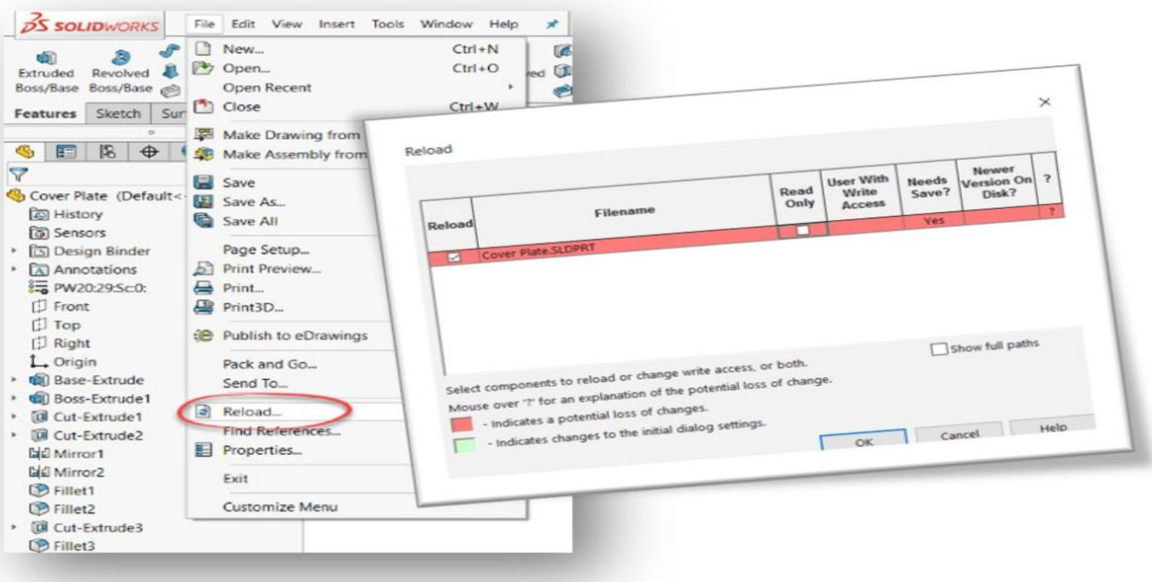
When you are working on an assembly and want to move components around the space, you can left-click on a component and drag to move it around in space. An extension of this functionality is right-clicking the mouse to rotate the component. This is a quick way to get components oriented just right as you build an assembly. Think of this as a breakdown of the Move Components command shown in the image below.



Tip 6. Reload—The "oops" button

The Reload command will automatically close without saving, and then reopen the model. I know we've all been there.

You open a file and make a mistake, so you close, hit don't save, and then reopen the file. To automate this and save time, you can use the Reload command to make that process much faster. SOLIDWORKS will even highlight any files that have been modified so you're aware of any potential data loss.



Tip 7 Force Rebuild—The duct tape of SOLIDWORKS

Have you ever called tech support for something you spent hours trying to fix but to no avail? Only to have tech support fix it in two seconds? Well that's because of Force Rebuild

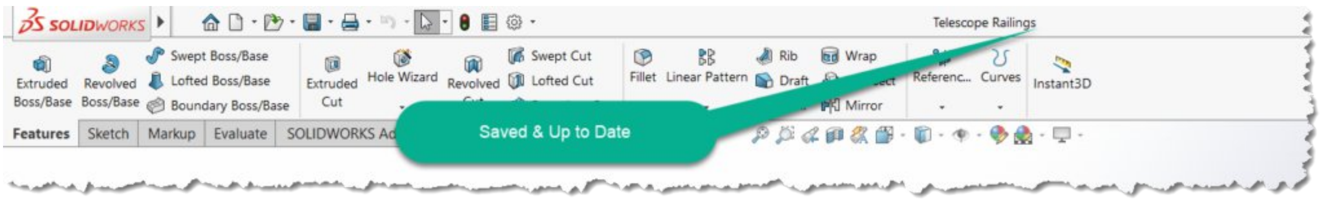
I'm only partly joking. We use this option in tech support to fix a lot of issues.

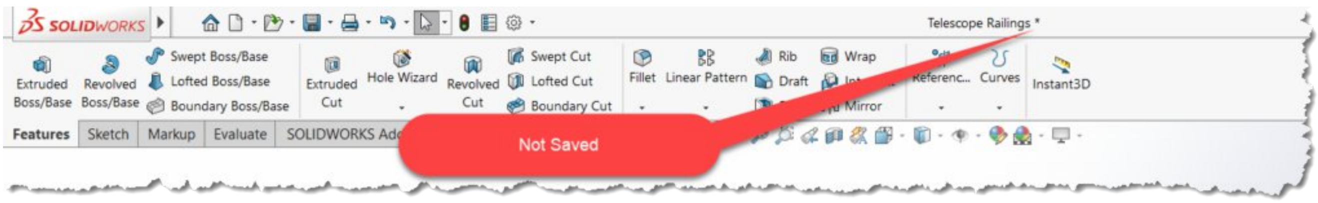
I call Force Rebuild the duct tape of SOLIDWORKS because it fixes everything, and we use it all the time. Force Rebuild is the command we use to force the regeneration of all the SOLIDWORKS geometry. This is your secret button to fix anything “wonky or strange” in your model. To use Force Rebuild, press Ctrl + Q.

The difference between this option and the regular Rebuild (Ctrl + B) is that Force Rebuild command rebuilds every piece of geometry from start to finish, while standard Rebuild only regenerates items that need to be rebuilt.

Tip 7. Did You Save?—No more uncertainty with the Asterisk

At the top of the SOLIDWORKS interface, in the middle of the screen, you’ll the name of the file. If you see an asterisk next to the file name, it means that there are changes to the design that have not been saved. Use this asterisk to identify whether you’ve saved your work.





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