SOLIDWORKS Tips and Tricks

Part 2of 2

SOLIDWORKS Tips and Tricks

16. Filter file types on open

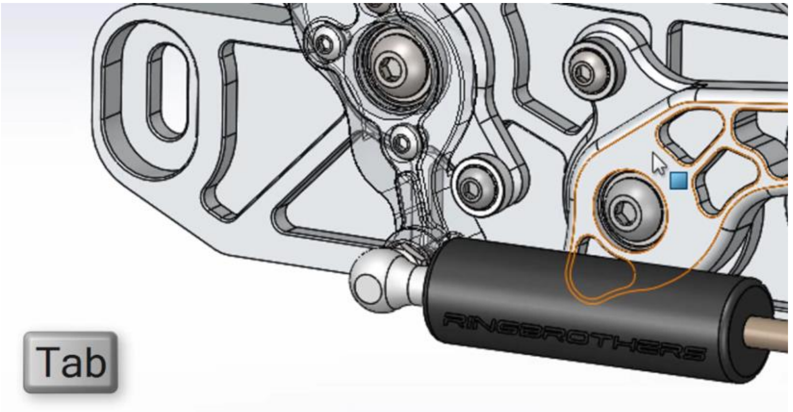
- When opening an assembly that contains lots of parts and sub-assemblies, it can be difficult to identify the correct file to open from the Windows folder.
- Use the **Quick Filter** to filter the folder display to show/hide part files, sub-assemblies, drawings, or display only the top level assemblies.
- This makes it much faster to find the file you need.

SOLIDWORKS Tips and Tricks

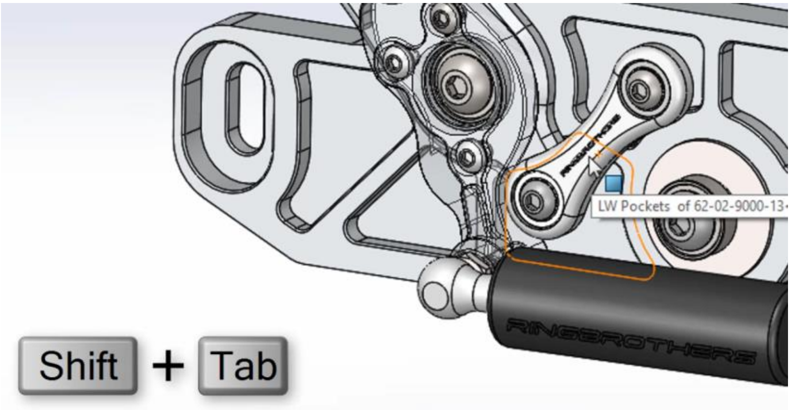
17. Assembly Shortcuts

- When working in assemblies there are some quick keyboard shortcuts that will make your life easier.
- If you wish to hide a component simply hover over it and press the TAB key on your keyboard.
- To make a hidden component visible hover over where it should be in the assembly and press SHIFT+TAB.

- HIDE:



- SHOW:

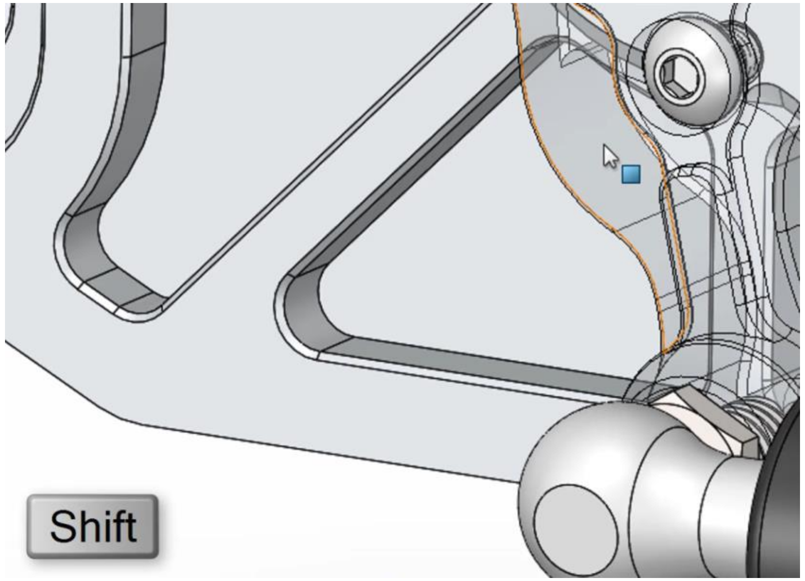


SOLIDWORKS Tips and Tricks

17. Assembly Shortcuts

- Transparent components are often hard to select as the transparency means you select through them, if you hold down shift before clicking on a transparent face then this will directly select the face regardless of transparency.

- Select Transparency:



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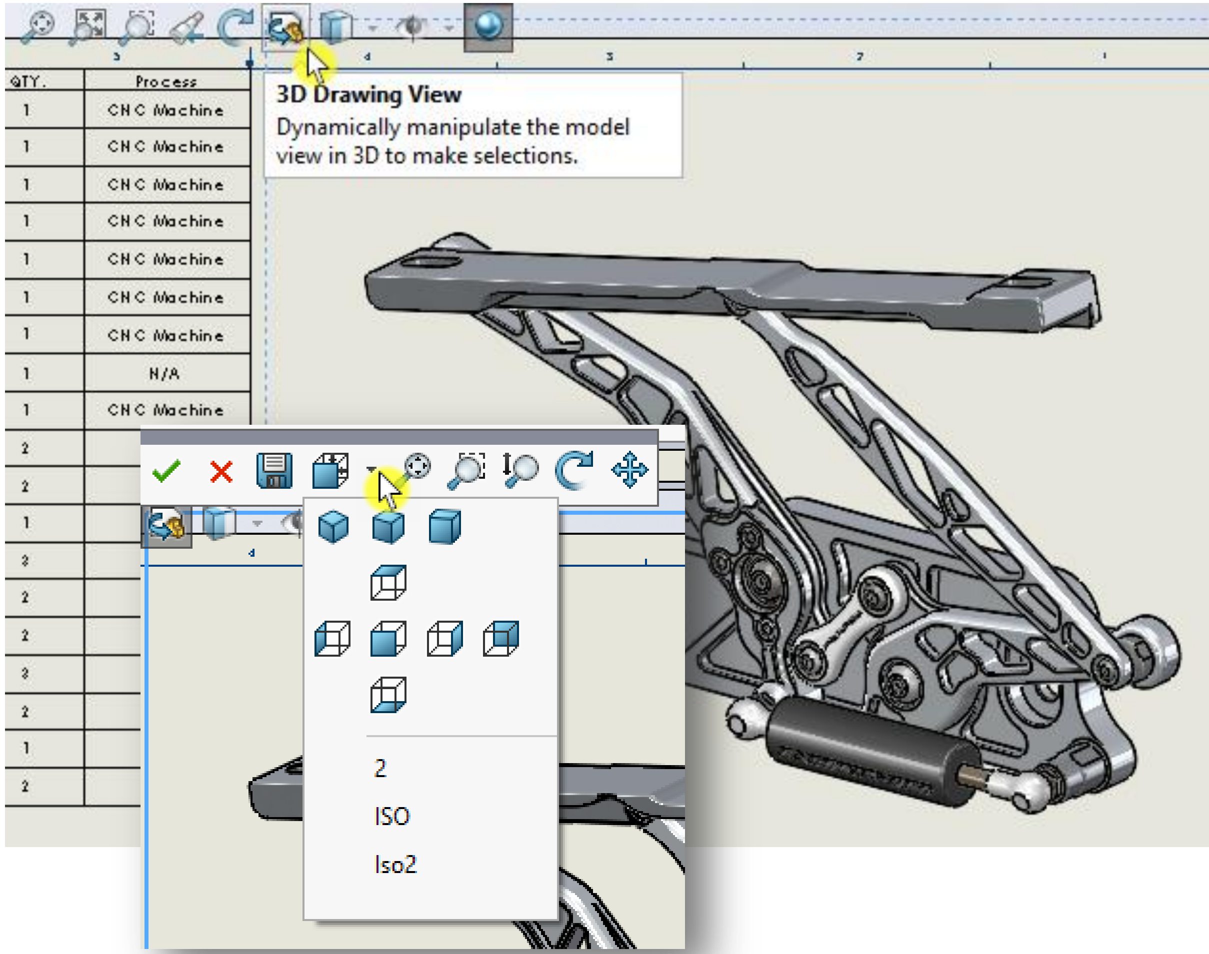
18. Save rotated view

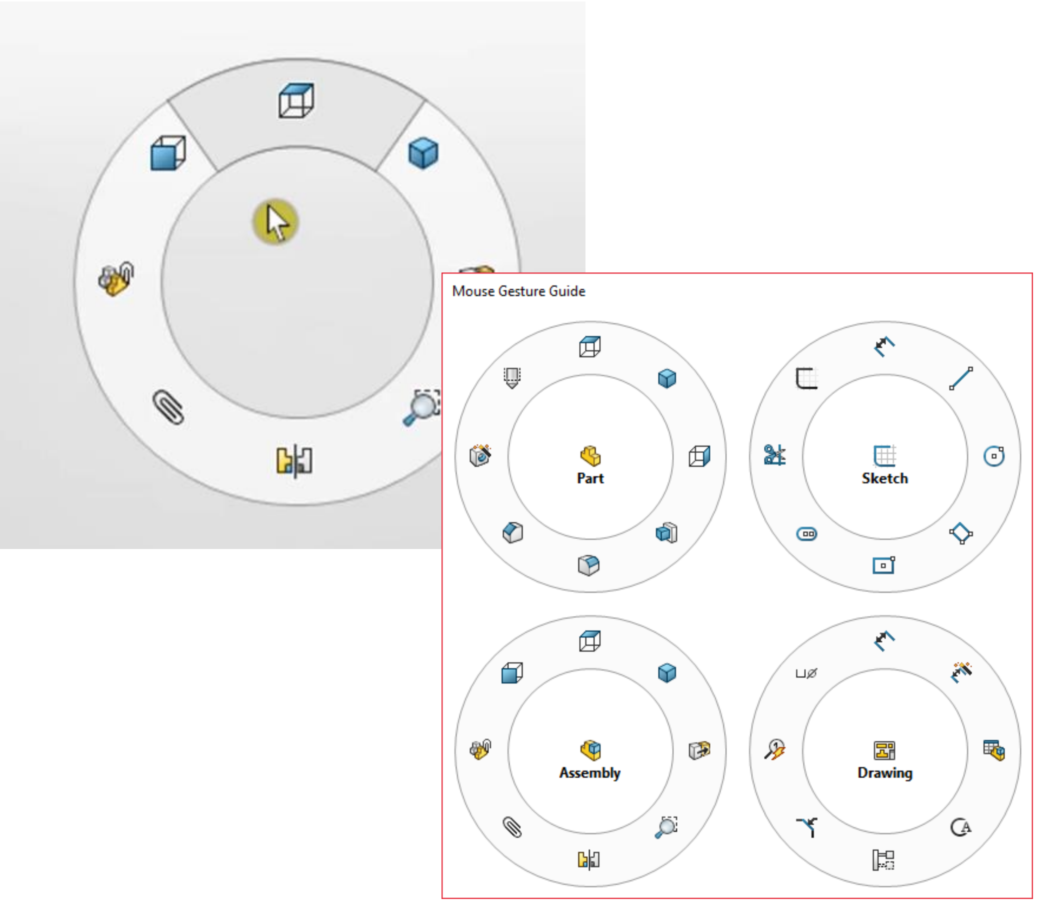
- As well as using the standard orientations in your drawing views, we can easily create our own custom 3D views.
- Click in a drawing view and select the **3D Drawing View** button where we can either select a standard view orientation, or dynamically pan and rotate to create our own custom view.
- Once happy simply hit the green tick to confirm, or select the save icon to save the new orientation as a **Named View**, so it can be applied to other drawing views.



SOLIDWORKS Tips and Tricks

19. Mouse Gestures

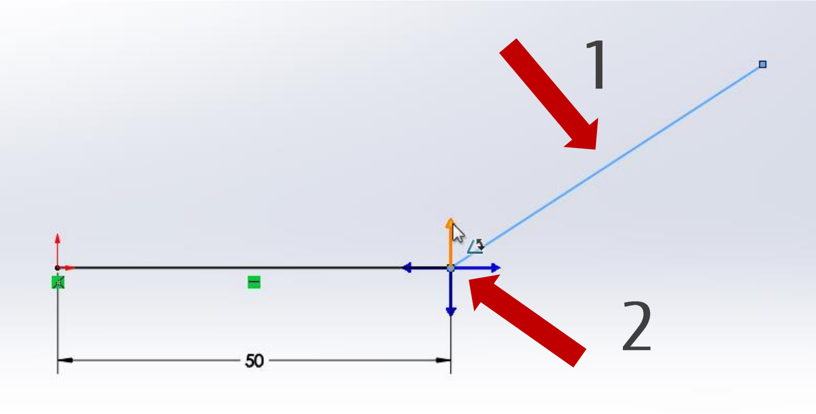
- Mouse gestures are a great way of accessing your common SOLIDWORKS functionality without the mouse movement of accessing the command manager.
- Simply hold the right mouse button down and move your mouse slightly and a guide will appear, swipe through an icon to access that tool or feature.
- You can have up to 12 gestures each for parts, assemblies, drawings and sketches and these are easily customised in SOLIDWORKS 2018 with the new gesture guide.
- Simply drag the icons into place on the guides to build your gestures, you can even print the gesture guide out to help you learn to use them.

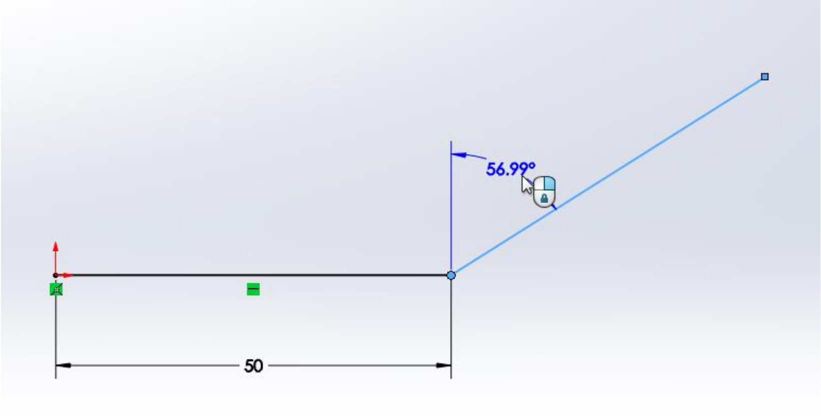


SOLIDWORKS Tips and Tricks

20. Angle Dimensions (without construction lines)

- When adding a dimension to an angle, a common method is to sketch a construction line to create an axis to take the dimension from. However, there is an easier method of doing this without using a construction line.
- When adding the angle dimension first select the line (1), then the vertex (2), then simply pick an arrow from the axis that will appear to take the dimension from.

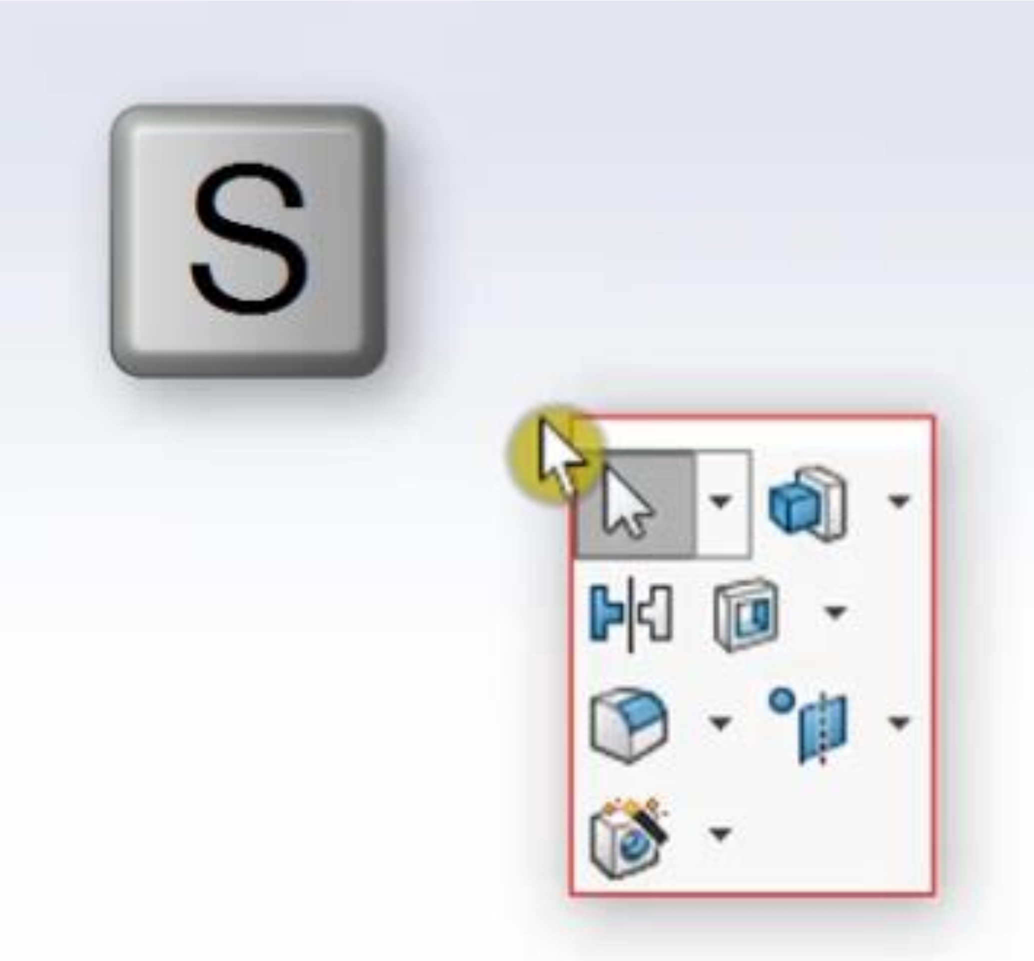




SOLIDWORKS Tips and Tricks

21. Shortcut Toolbar

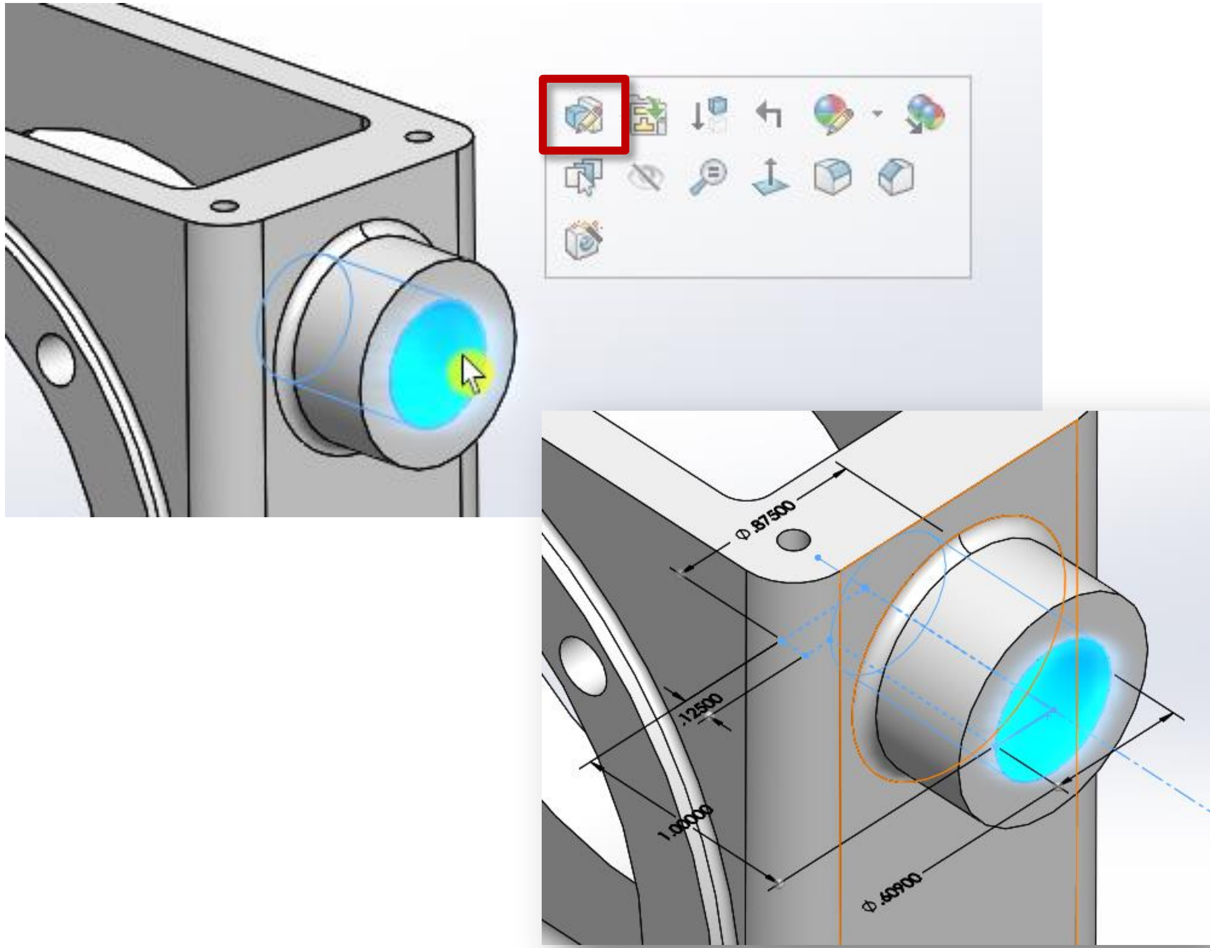
- In a similar fashion to Mouse gestures we can consider that mouse movement is wasted time, therefore if we can access the functionality that we need without excessive mouse movement (i.e. access everything we need right at our mouse pointer then this will naturally speed up our work).
- The shortcut toolbar is a great way of doing this, simply press the S key on your keyboard and a customisable toolbar appears right where your mouse is to allow you to access the tools you need.
- You can have separate shortcut toolbars for parts, assemblies, drawings and sketches and these are easily customised to only include your regularly used tools, helping to really speed up how you work.



SOLIDWORKS Tips and Tricks

22. Edit feature with FeatureWorks

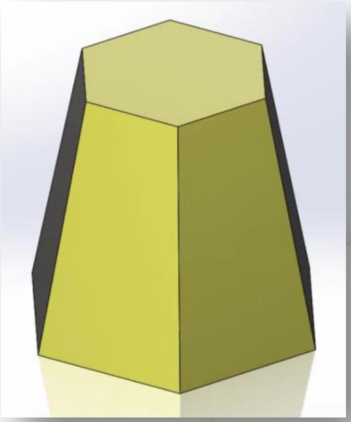
- If you need to modify imported model geometry the fastest way to do this is to simply select the model feature you wish to edit, and on the context toolbar select **Edit Feature**.
- This will automatically invoke **FeatureWorks**, which will attempt to recognise the feature as native SOLIDWORKS features and dimensioned sketches, making it much easier to modify the design.

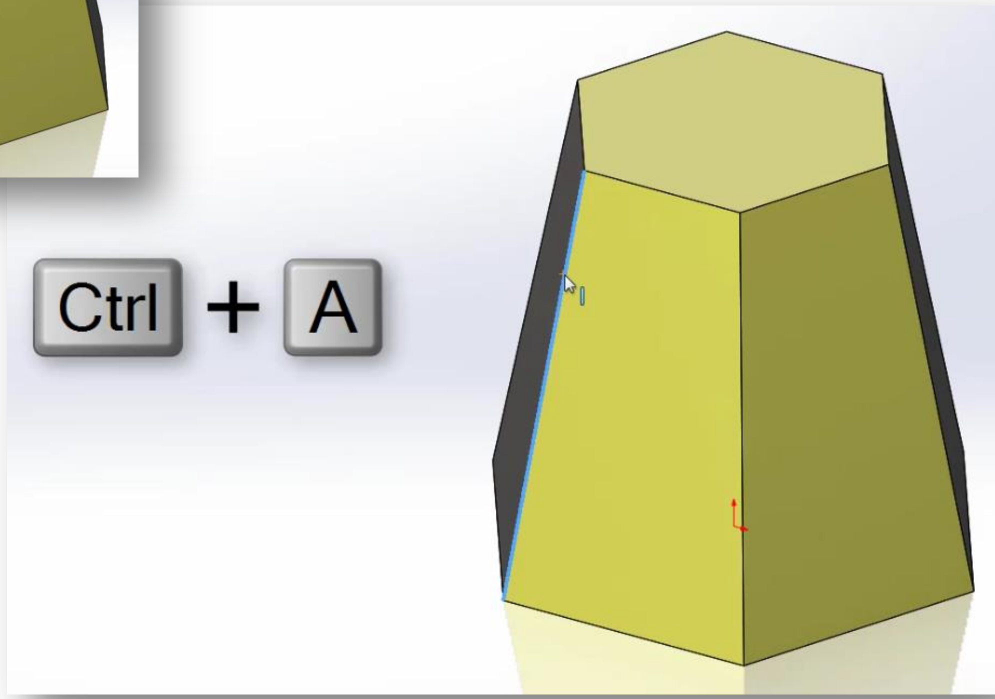


SOLIDWORKS Tips and Tricks

23. Rapid 3D sketches

- 3D sketches can often take a while to produce and inherently they take longer to produce than a 2D sketch.
- A nice technique is to solid model the geometry required for a 3D sketch (this technique works really well when creating sketches for weldment frameworks etc.), we will then convert the edges into sketch entities to use for our 3D sketch.
- The edges of this solid will form my 3D sketch. To convert the edges from a Solid simply create a 3D sketch, select a single model edge then press CTRL+A to select all edges.

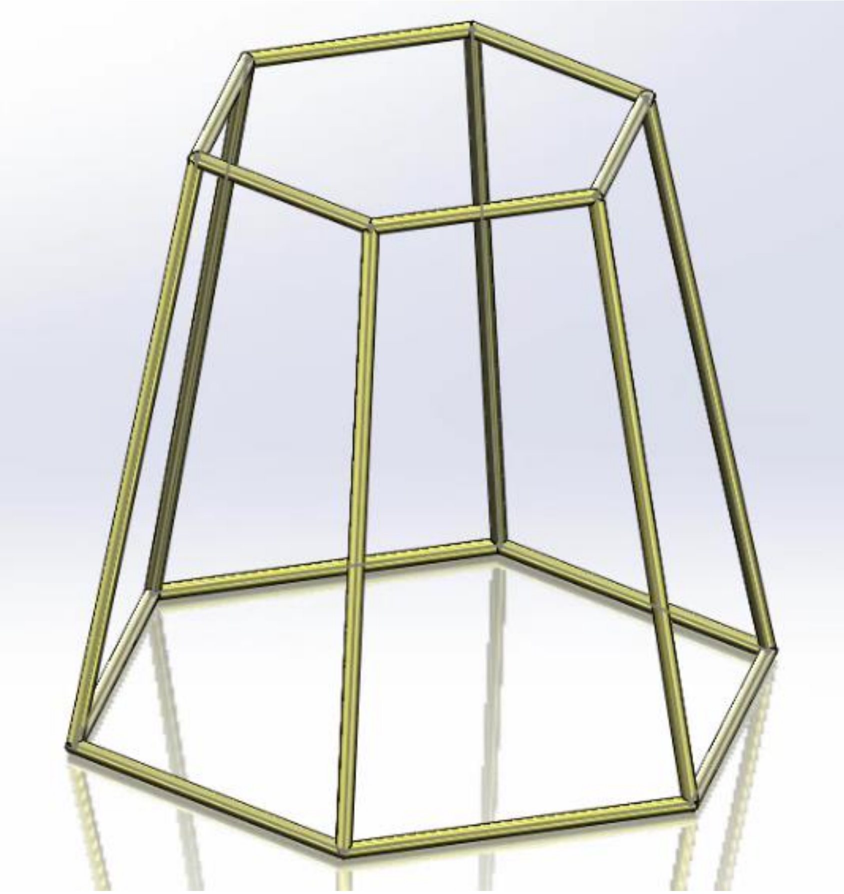




SOLIDWORKS Tips and Tricks

23. Rapid 3D sketches

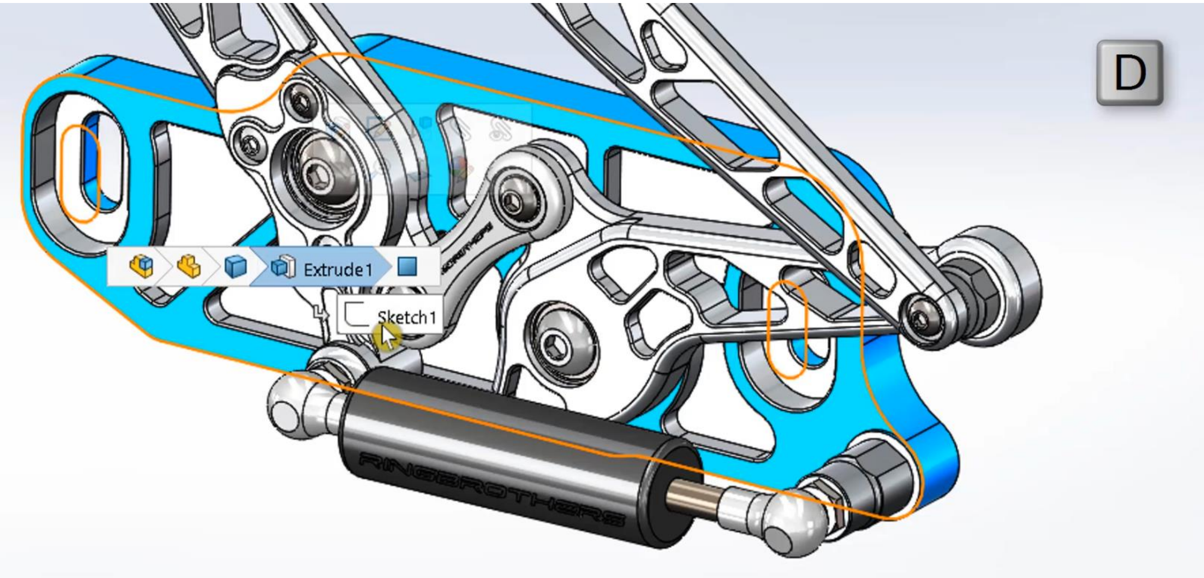
- Click **Convert Entities** to create the 3D sketch entities, hide the model and you have a 3D sketch ready for whatever you want to do next.
- It's also worth pointing out that the 3D sketch is controlled by the solid body, so if you modify the dimensions, the 3D sketch will update with it.



SOLIDWORKS Tips and Tricks

24. Breadcrumbs

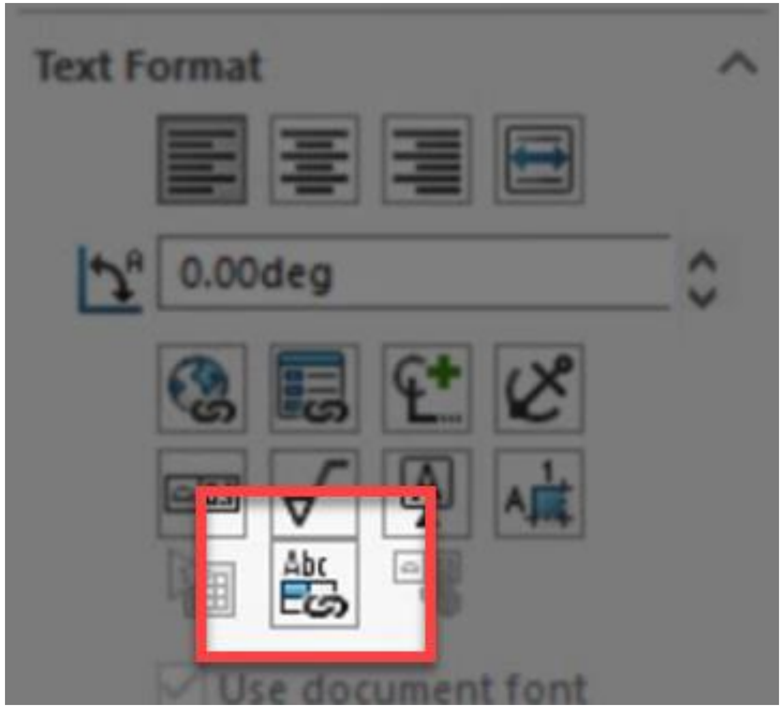
- Selection Breadcrumbs are available in part and assembly files and are a useful context based menu used to refine your selection based on the selections related features, such as the underlying sketch of a feature or the mates of a component.
- Breadcrumbs can be brought to your cursor by pressing the D key after a selection is made.
- To show Breadcrumbs enable the feature in your **System Options>Display>Show breadcrumbs on selection**.



SOLIDWORKS Tips and Tricks

25. Link Note to Table Cell

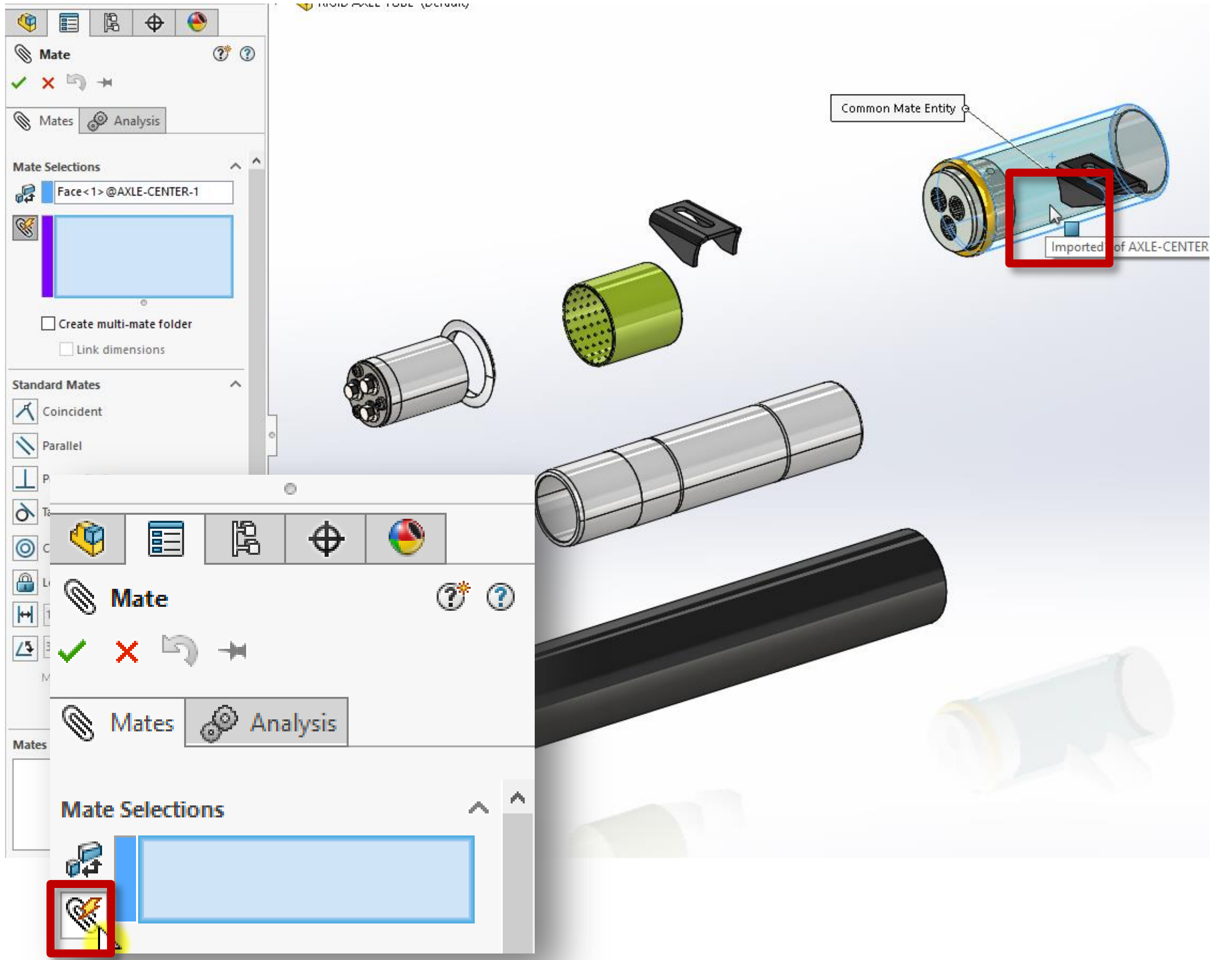
- We can directly link notes to the information within a table, simply create a note, and click the "link table cell" button in the property manager.
- Now we can directly click the cells in a table to bring the information in that cell into the note.
- Of course it's the cell that's being referenced not the text, so should the information in the cell change the note will automatically update with it.



SOLIDWORKS Tips and Tricks

26. Multiple Mate mode

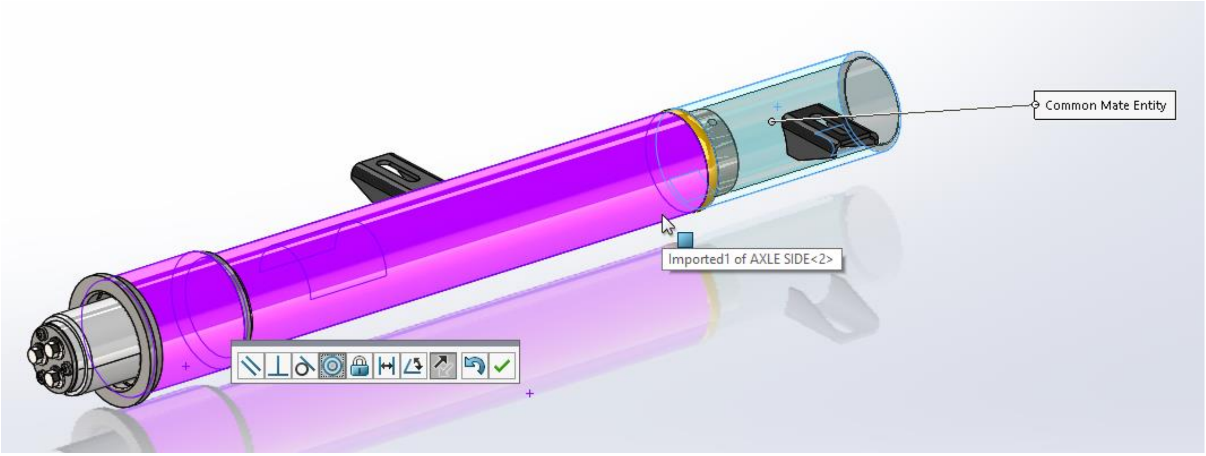
- The Multiple Mate mode in SOLIDWORKS assemblies allows user to speed up the creation of mates by first selecting a common mate reference, such as the cylindrical face shown here;



SOLIDWORKS Tips and Tricks

26. Multiple Mate mode

- Multiple entities can then be selected and will align to the common mate reference in one command. This will create the separate mates in the feature tree.



SOLIDWORKS Tips and Tricks

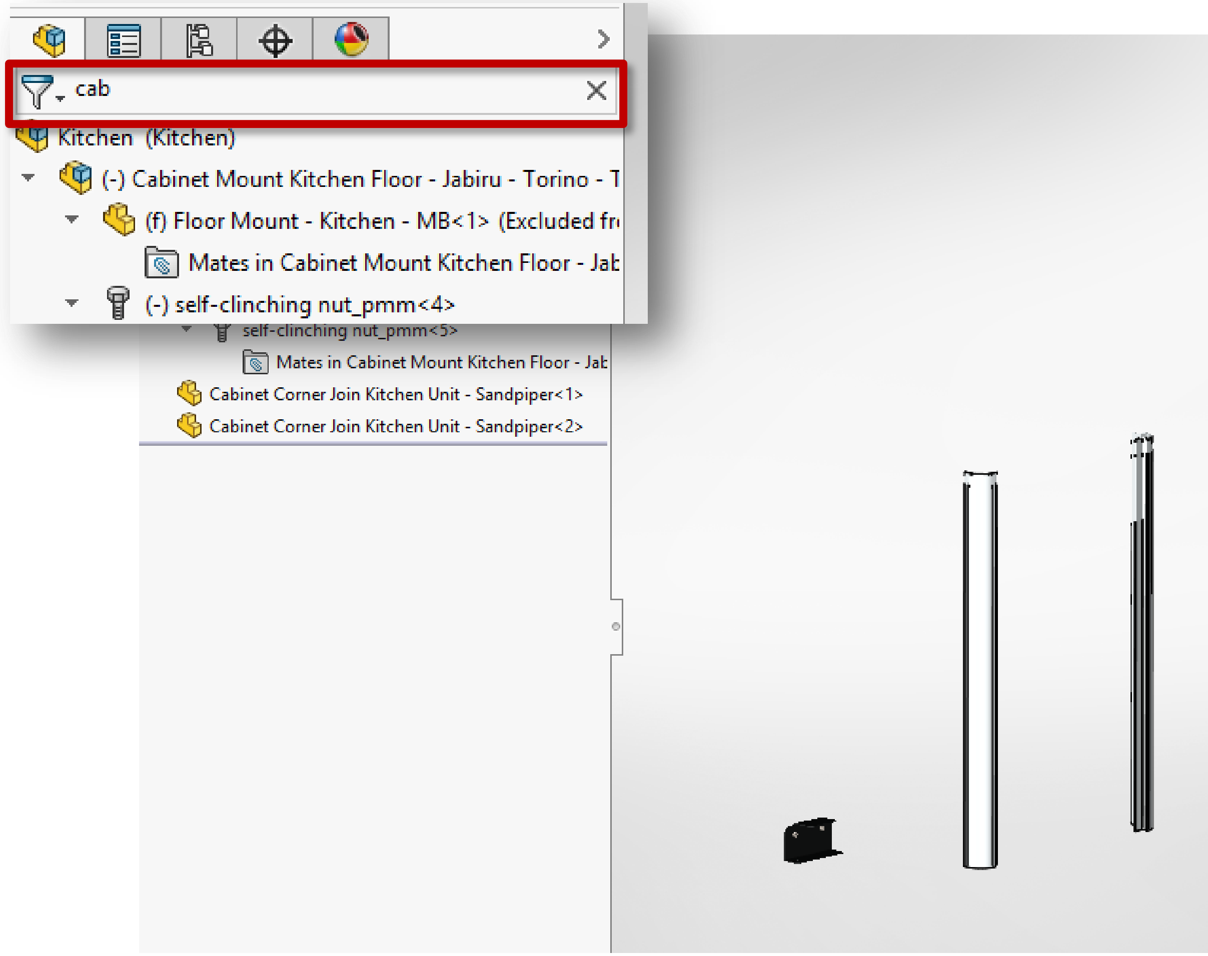
27. 'D' Key Confirmation Corner

- Confirmation corner is a great way of interacting with SOLIDWORKS however it involves moving your mouse to the top right hand corner of the graphics window and realistically this is unnecessary mouse movement.
- Instead if you press the D key, SOLIDWORKS will move the confirmation corner commands right to your mouse pointer, helping to eliminate that mouse movement.

SOLIDWORKS Tips and Tricks

28. Filter tree and collapse items

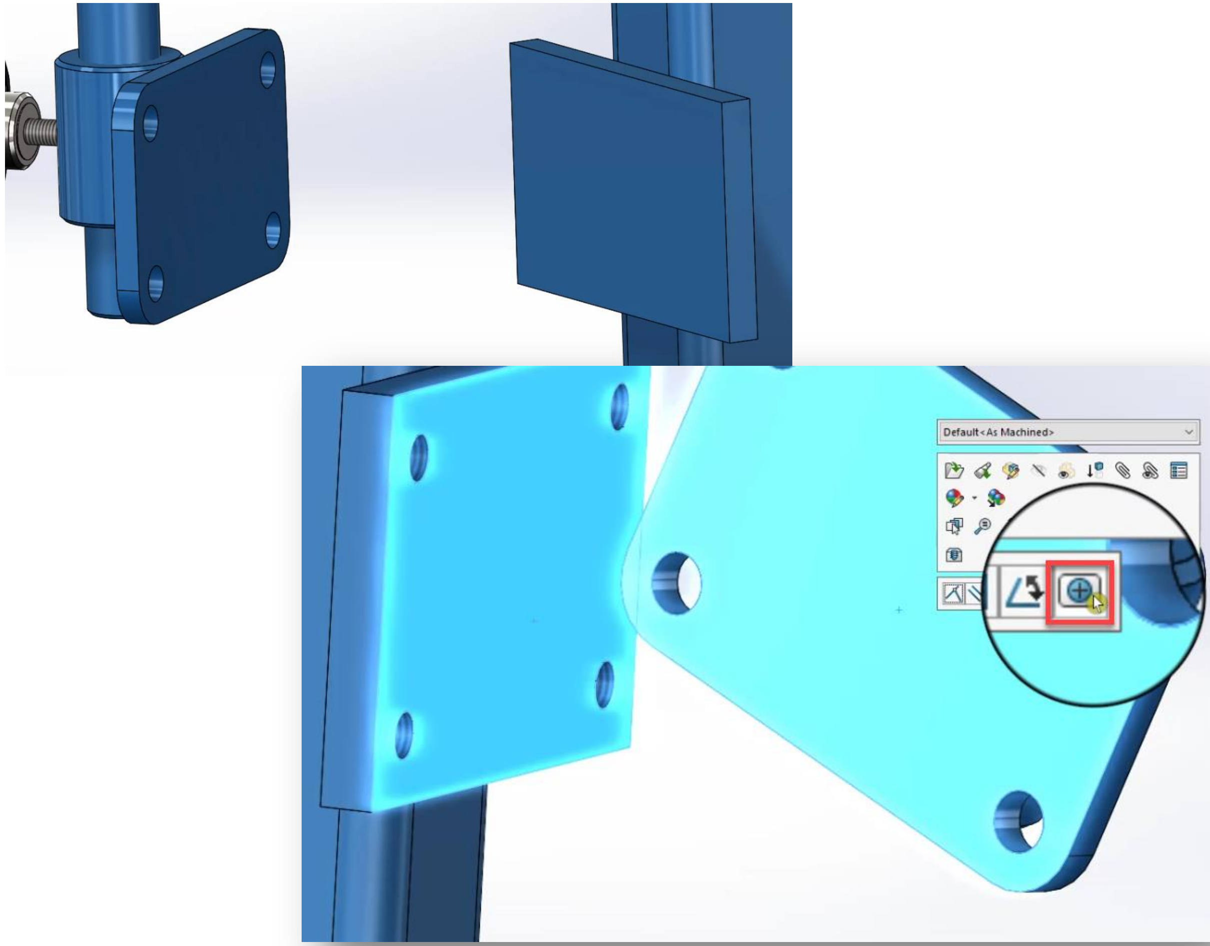
- Finding components in a large assembly tree can be time consuming.
- Using the filter box at the top of the Feature Manager Design Tree, we can start typing the name of the component or feature we are looking for and the filter isolates the results in the viewport making it really easy to find those items for editing.
- Just click on the **X** in the filter box to re-show the full assembly.
- Another quick tip: Hit **Shift-C** on your keyboard, to immediately collapse all of the expanded folders and sub-assemblies in you Feature Manager Design Tree.



SOLIDWORKS Tips and Tricks

29. Profile Center Mates

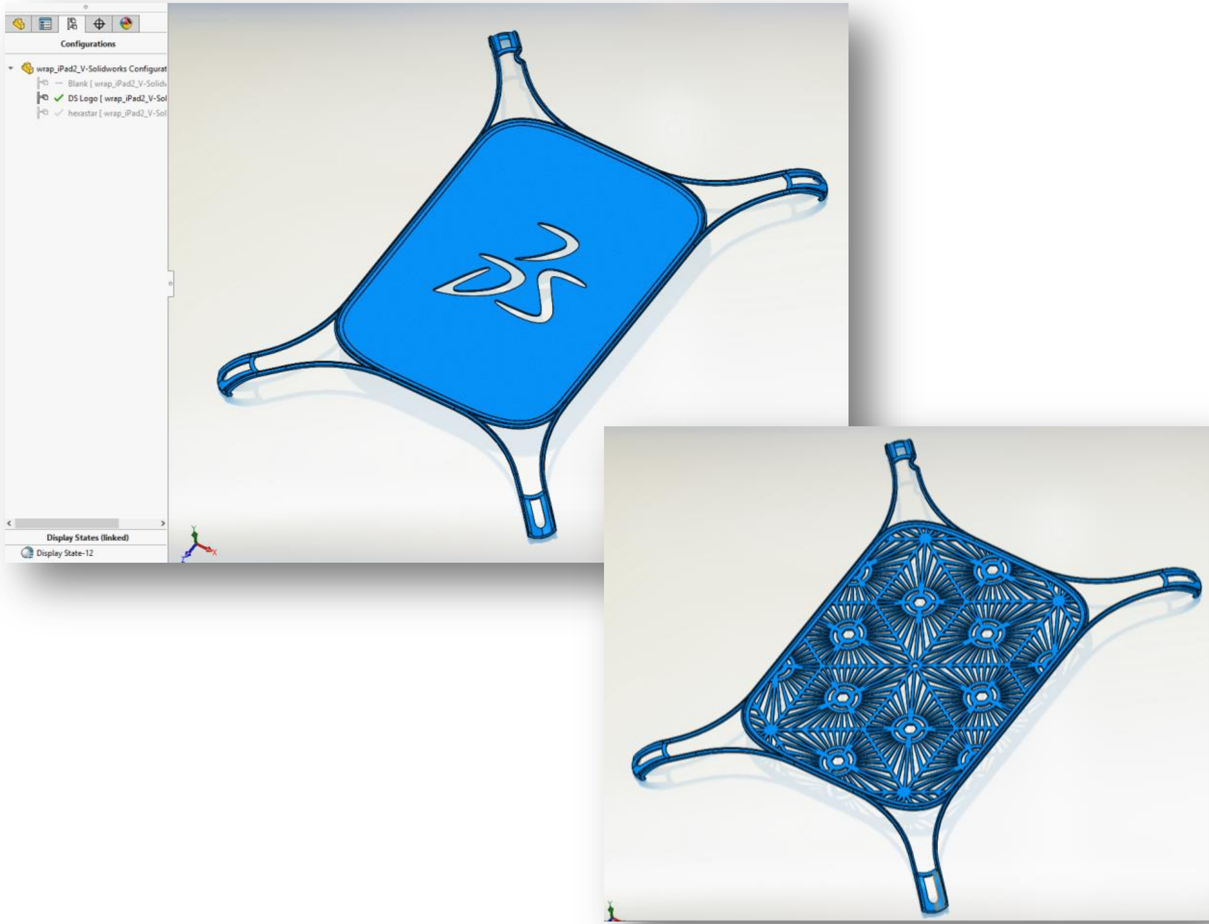
- Profile center mates are a great way of fully defining components in an assembly whilst at the same time minimising the number of mates required.
- A profile center mate will take two faces and find their geometric centres and match them together to orientate and fully define the component in a single mate operation, options are provided for offset and orientation to allow you to position the components correctly in your assembly.
- These two faces despite having dissimilar features share a geometric center that can be matched up using a profile center mate from the mates toolbar (under advanced mates) They can also be added from the quick mates toolbar.



SOLIDWORKS Tips and Tricks

30. Freeze bar and feature statistics

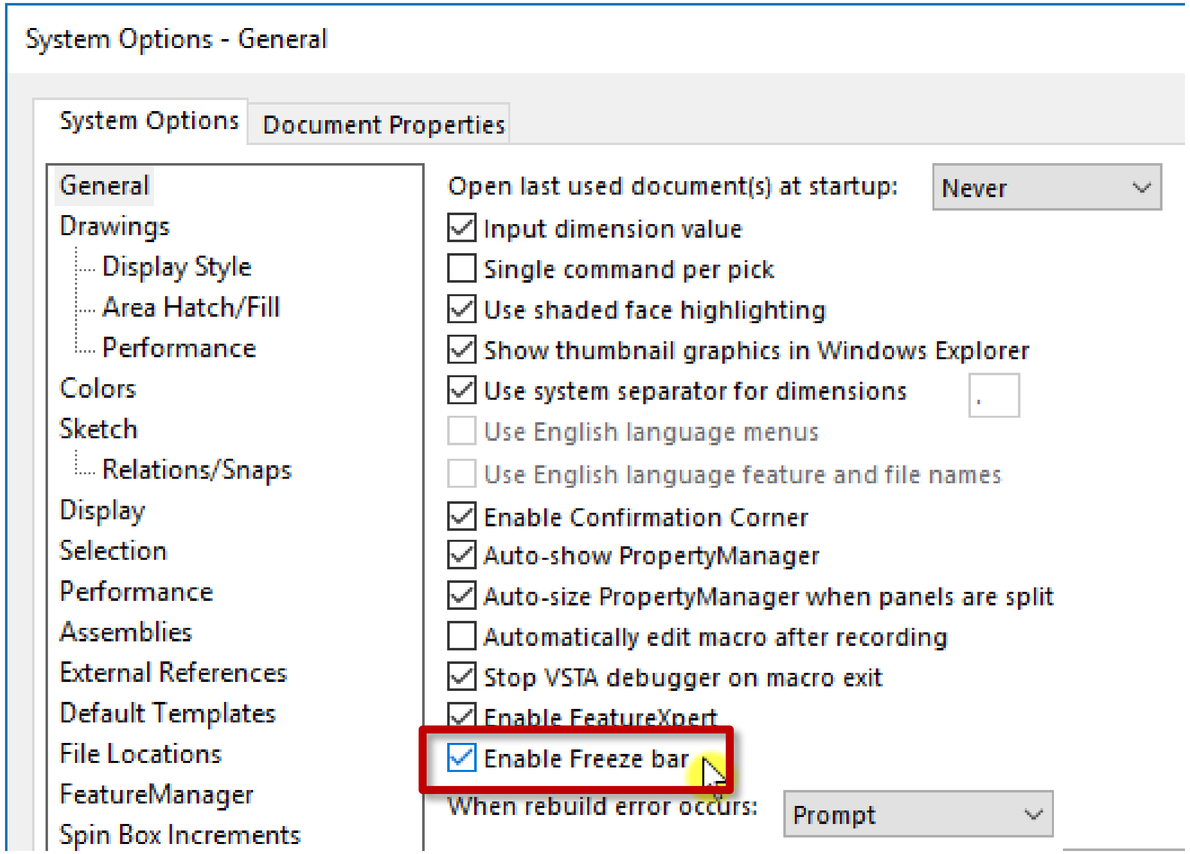
- Parts containing lots of complex features can sometimes cause model rebuild times to be high.
- This protective cover for a tablet device has several different design configurations and as we switch between configurations we get a slight delay as the model rebuilds each time. When we switch to the complex 'Hexastar' design it takes several seconds to rebuild;



SOLIDWORKS Tips and Tricks

30. Freeze bar and feature statistics

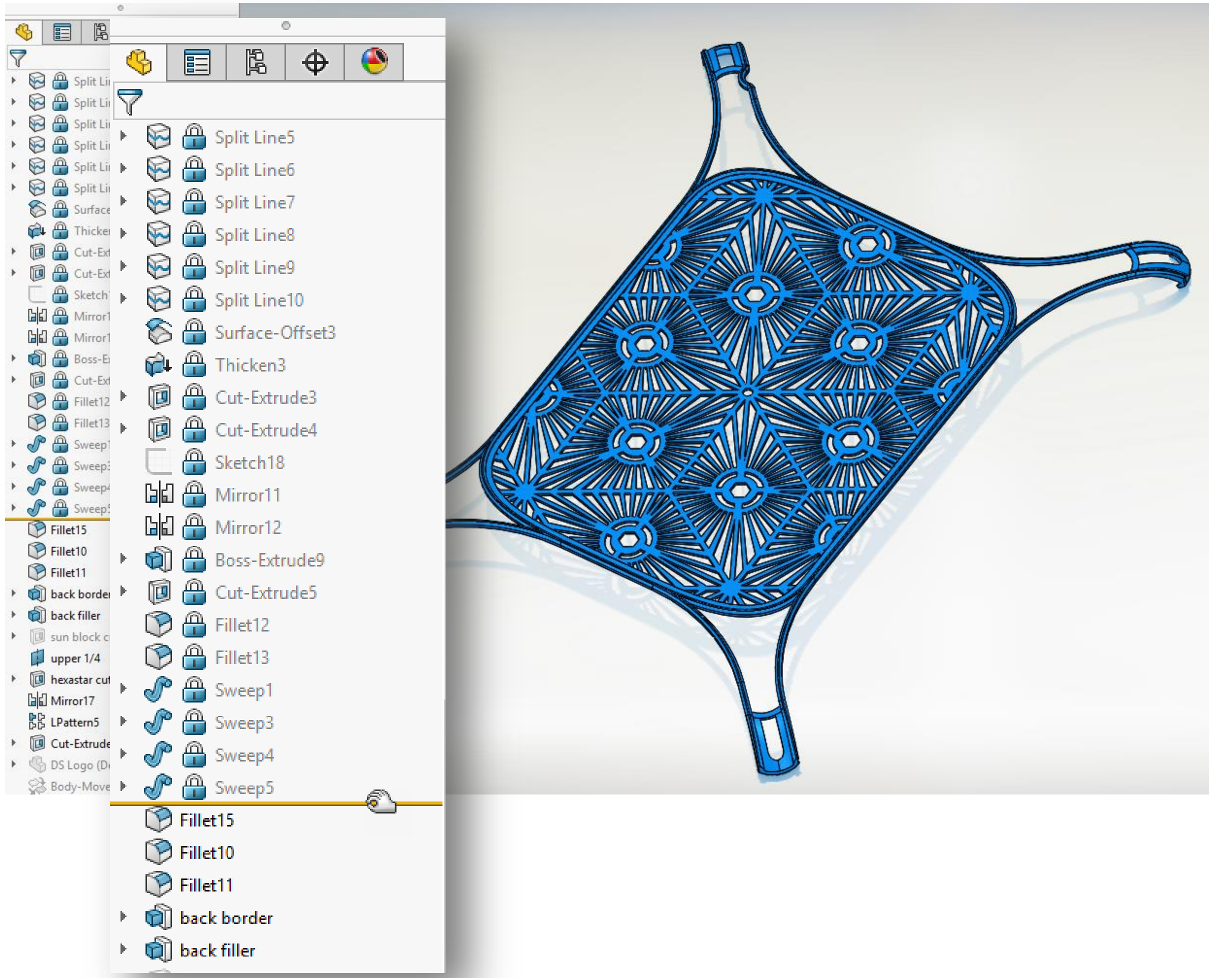
- To reduce rebuild time enable the **Freeze** bar in System Options.



SOLIDWORKS Tips and Tricks

30. Freeze bar and feature statistics

- We can drag the freeze bar to any point in the feature tree (or roll directly to the end), to stop SOLIDWORKS rebuilding any features above the bar.
- So now switching between the different configurations is almost instant.



Thank You

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