

# SOLIDWORKS Tips and Tricks

Part 1 of 2



 **SOLIDWORKS**

 **SOLIDSOLUTIONS**  
SUPPORTING EXCELLENCE

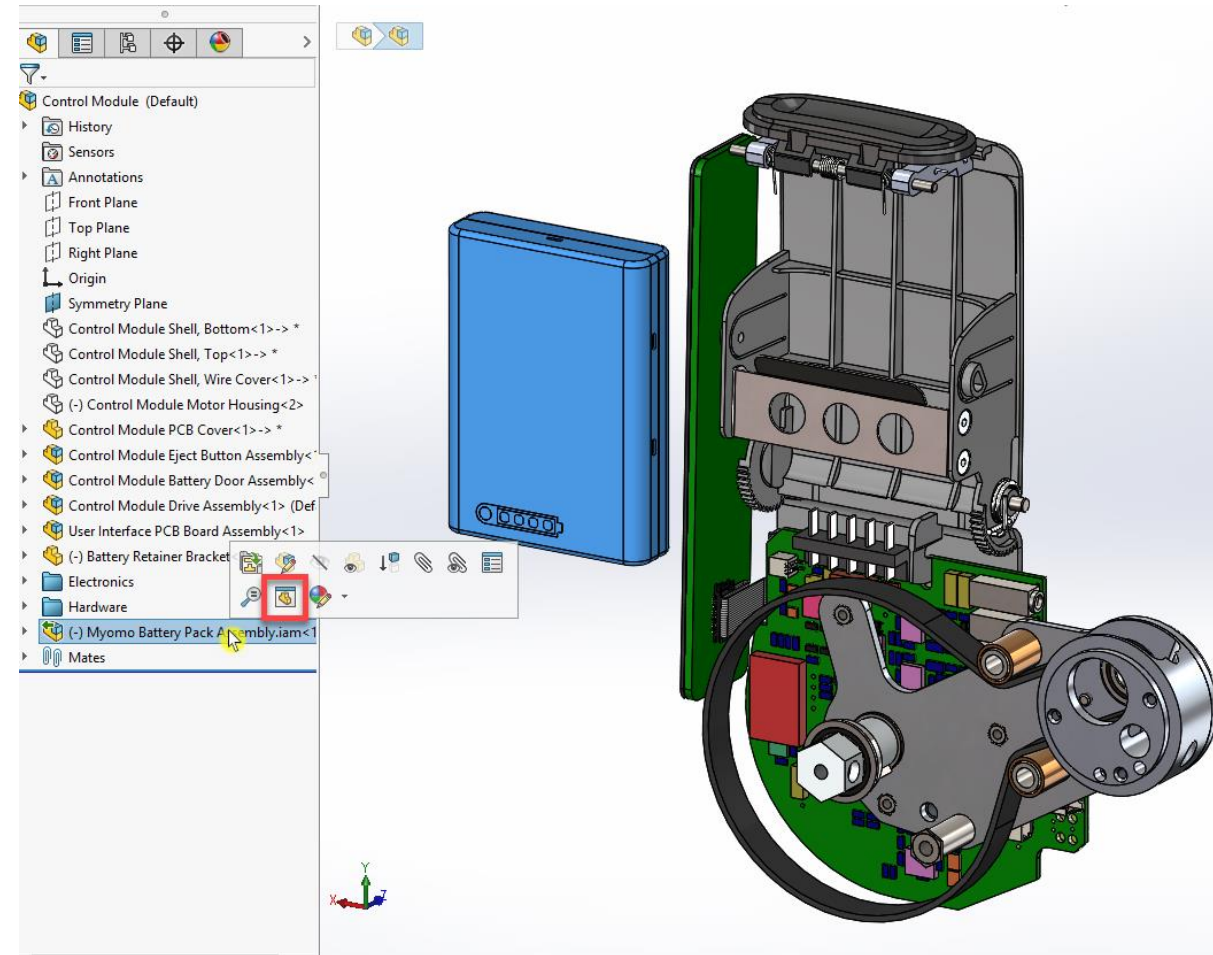
 nt  
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 CADTEK

# SOLIDWORKS Tips and Tricks

## 01. Component Preview Windows

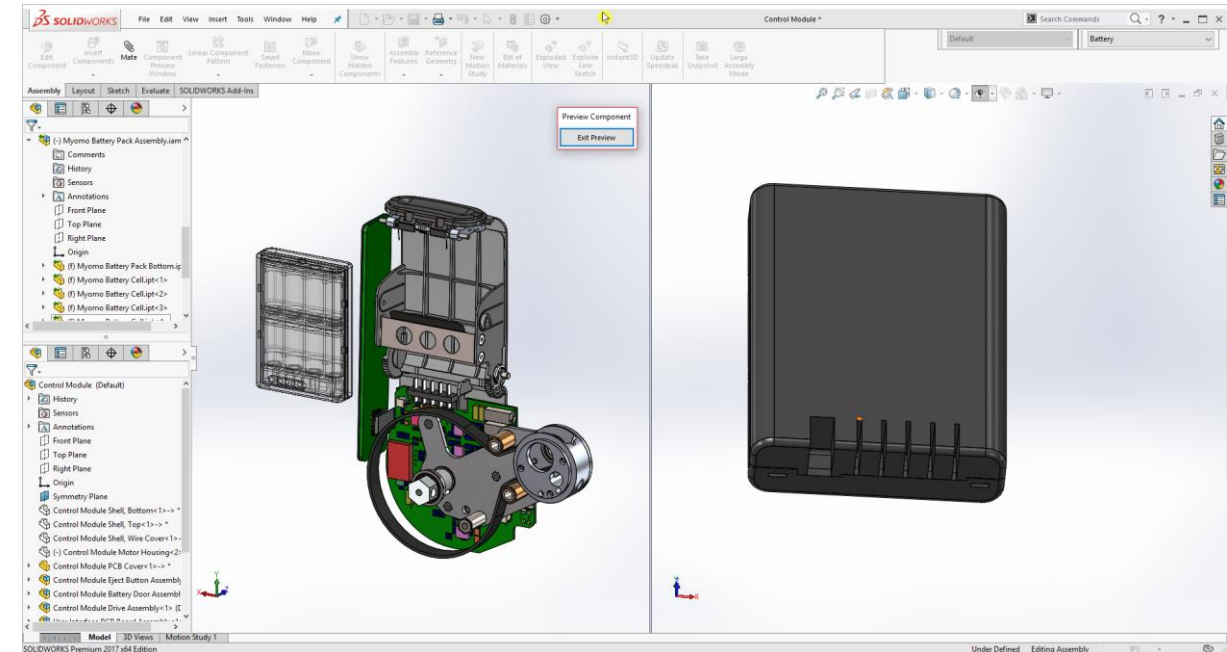
- ▶ When mating components in an assembly it is often difficult to access the desired faces for the mates as they might be obscured by other geometry.
- ▶ In the past we have resorted to using Select Other, transparency or simply rotating the model to an awkward angle to access the faces.
- ▶ By right clicking on the component in the graphics area or in the Feature Manager you can open up a component preview window;



# SOLIDWORKS Tips and Tricks

## 01. Component Preview Windows

- ▶ The Component Preview Window allows you to manipulate the component separately from the main assembly in its own window, making applying mates much simpler.

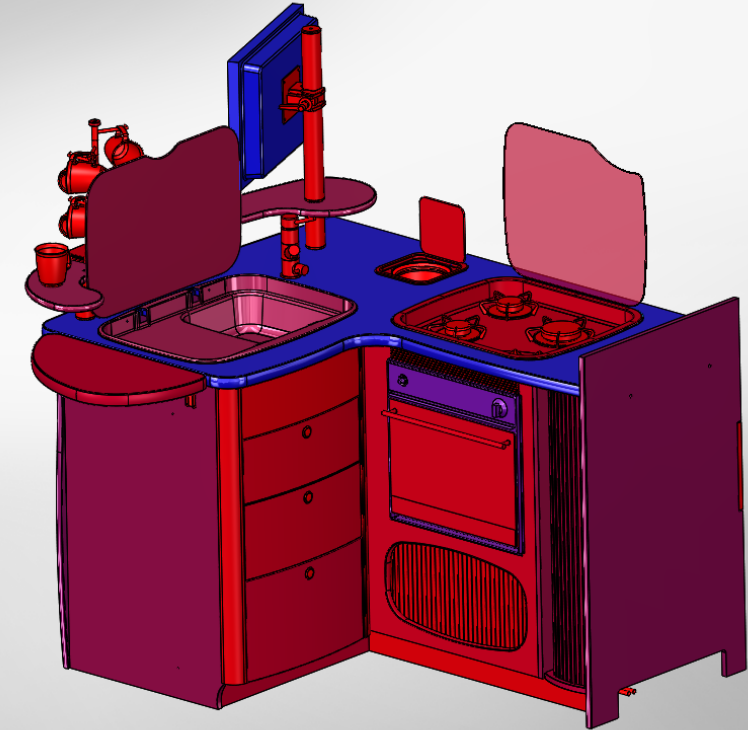


# SOLIDWORKS Tips and Tricks

## 02. Assembly Visualization

- ▶ We can use **Assembly Visualization** to rank our sub-assemblies and components based on any custom file property.
- ▶ This feature is found on the Evaluate tab in assemblies.
- ▶ Here we can see components ranked by their mass property from high to low, using colour coding and value bars in the graphics display.
- ▶ We can toggle sort values from high to low by clicking on a properties title bar.

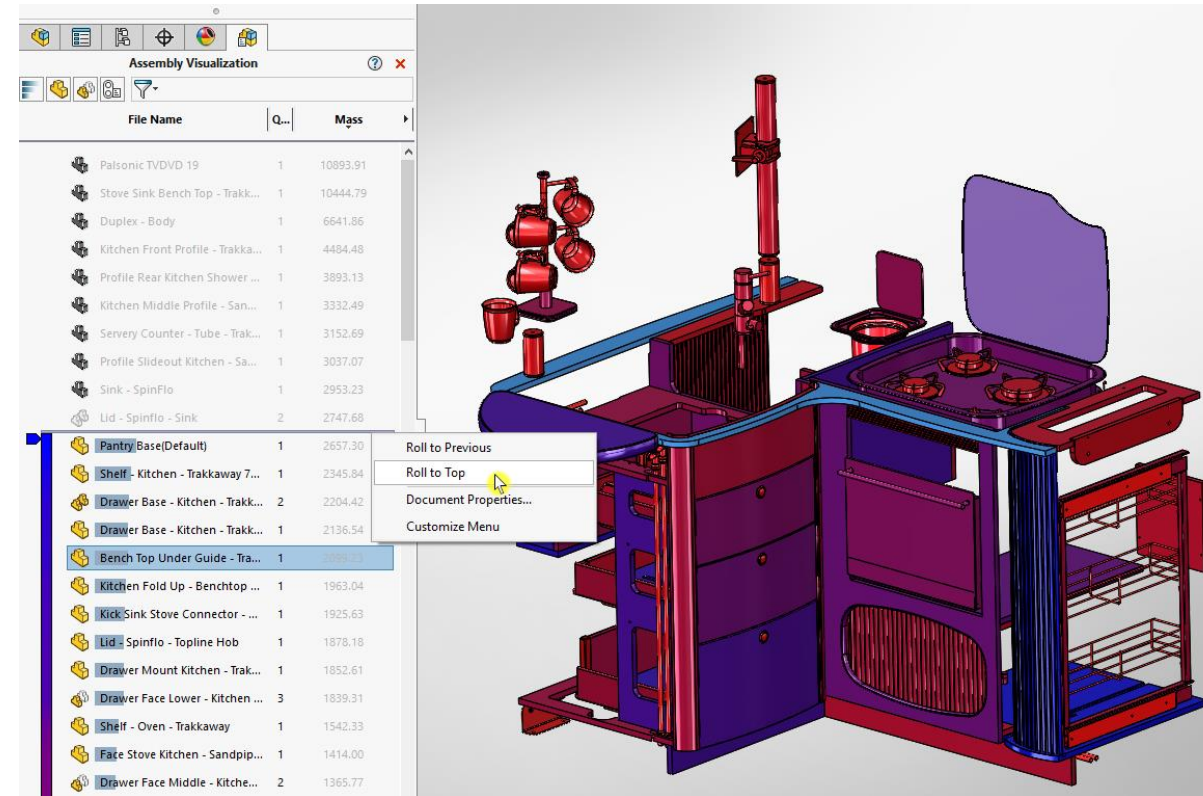
File Name	Q...	Mass
Palsonic TVDVD 19	1	10893.91
Stove Sink Bench Top - Trakk...	1	10444.79
Duplex - Body	1	6641.86
Kitchen Front Profile - Trakka...	1	4484.48
Profile Rear Kitchen Shower ...	1	3893.13
Kitchen Middle Profile - San...	1	3332.49
Servery Counter - Tube - Trak...	1	3152.69
Profile Slideout Kitchen - Sa...	1	3037.07
Sink - SpinFlo	1	2953.23
Lid - Spinflo - Sink	2	2747.68
Pantry Base(Default)	1	2657.30
Shelf - Kitchen - Trakkaway 7...	1	2345.84
Drawer Base - Kitchen - Trakk...	2	2204.42
Drawer Base - Kitchen - Trakk...	1	2136.54
Bench Top Under Guide - Tra...	1	2099.23
Kitchen Fold Up - Benchtop ...	1	1963.04
KickSink Stove Connector - ...	1	1925.63
Lid - Spinflo - Topline Hob	1	1878.18
Drawer Mount Kitchen - Trak...	1	1852.61
Drawer Face Lower - Kitchen ...	3	1839.31
Shelf - Oven - Trakkaway	1	1542.33
Face Stove Kitchen - Sandpip...	1	1414.00
Drawer Face Middle - Kitche...	2	1365.77
Stove - Topline - SpinFlo	1	1280.00
Profile Sink Base - Sandpip...	1	1271.70



# SOLIDWORKS Tips and Tricks

## 02. Assembly Visualization

- ▶ Any selected components are highlighted in the graphics area, and the roll bar allows us to show components above or below a certain value.



# SOLIDWORKS Tips and Tricks

## 02. Assembly Visualization

- ▶ With a right click we can roll back to the top to show all components.
- ▶ Add extra columns as desired, and choose to show any available file property.

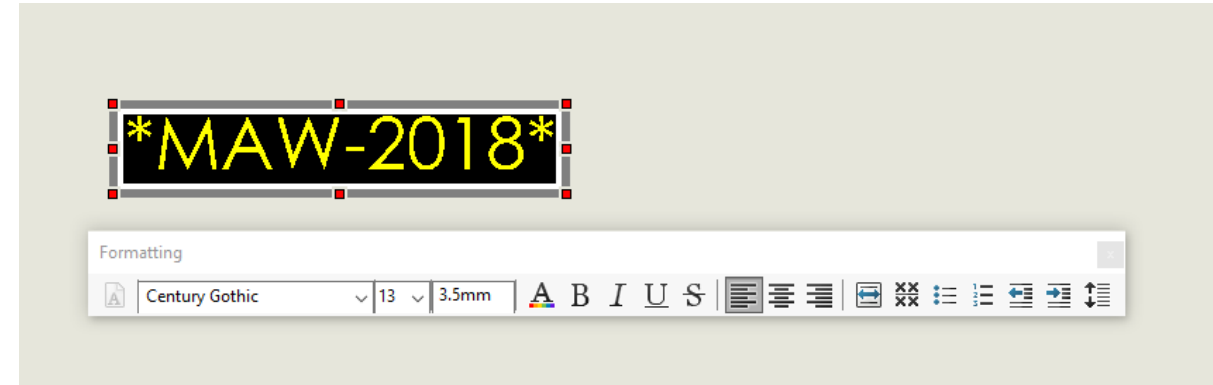
The screenshot shows the 'Assembly Visualization' window in SolidWorks. The table displays a list of components with columns for 'File Name', 'Q...', 'Mass', and 'Total'. A right-click context menu is open over the 'Total' column header, which is highlighted with a red box. The menu includes options such as 'Total Weight', 'Mass', 'Density', 'Volume', 'Surface Area', 'More...', 'Add Column', 'Delete Current', 'Unit Precision', 'Value Bars', 'Add Display State', 'Load Style', 'Save Style', 'Save As ...', and 'Performance Analysis'. The 'Add Column' option is highlighted with a red box and a yellow mouse cursor.

File Name	Q...	Mass	Total
Palsonic TVDVD 19	1	10893.91	10...
Stove Sink Bench Top - ...	1	10444.79	10...
Duplex - Body	1	6641.86	66...
Kitchen Front Profile - T...	1	4484.48	44...
Profile Rear Kitchen Sh...	1	3893.13	38...
Kitchen Middle Profile - ...	1	3332.49	33...
Servery Counter - Tube ...	1	3152.69	31...
Profile Slideout Kitchen...	1	3037.07	30...
Sink - SpinFlo	1	2953.23	29...
Lid - Spinflo - Sink	2	2747.68	54...
Pantry Base(Default)	1	2657.30	26...
Shelf - Kitchen - Trakka...	1	2345.84	23...

# SOLIDWORKS Tips and Tricks

## 03. Adding Barcodes to Drawing Sheets

- ▶ Adding Barcodes to a drawing sheet in SOLIDWORKS is really simple and allows for documents to be tracked once they have been printed.
- ▶ Simply create a note featuring the text you wish to have in the barcode such as part or drawing number;



# SOLIDWORKS Tips and Tricks

## 03. Adding Barcodes to Drawing Sheets

- ▶ Change the text into a barcode font (which you can download from the internet- look out for a code 39 barcode font) and you now have a working barcode on your drawing sheet. (Barcodes often use a \* as the start/finish character in a string as a check digit)
- ▶ Pro Tip- if you incorporate into your sheet format then the barcode will manage itself!

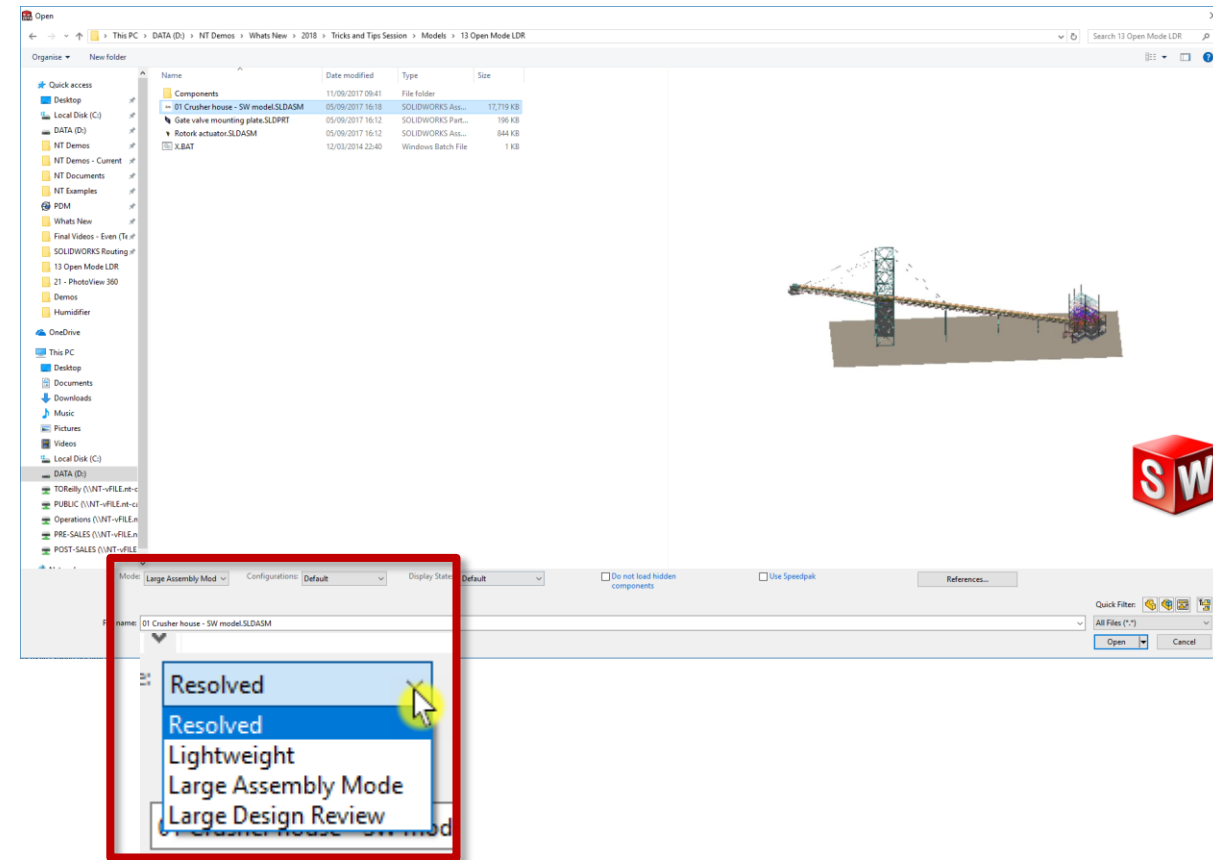




# SOLIDWORKS Tips and Tricks

## 04. Open Mode Options

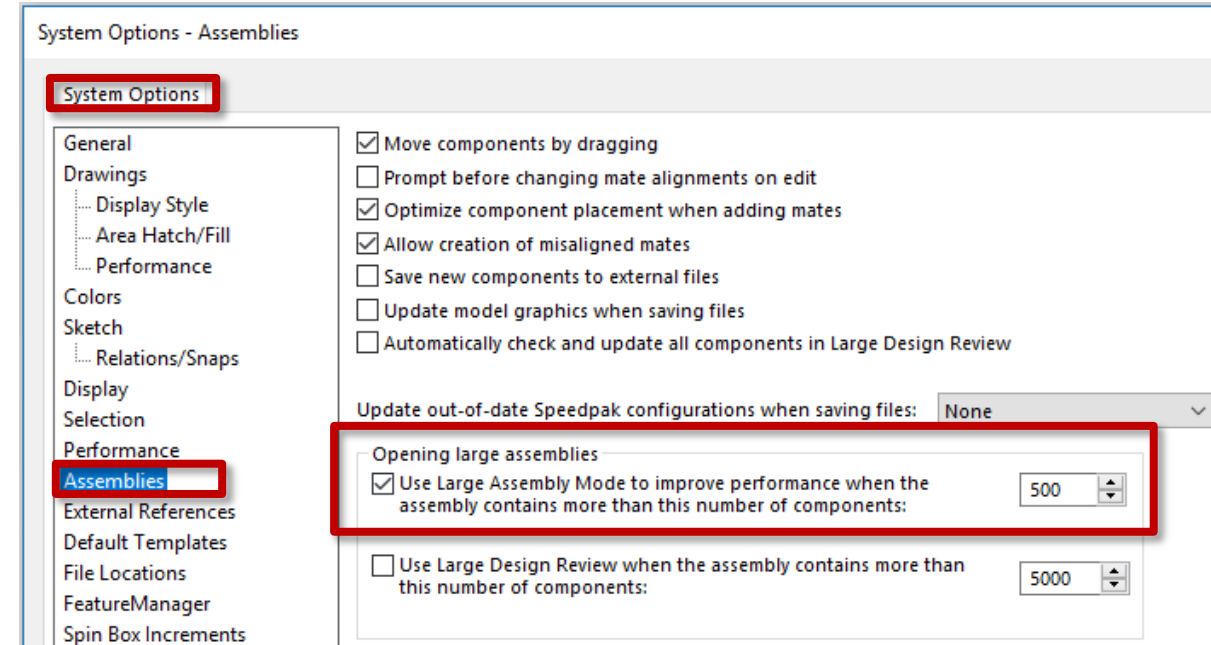
- ▶ Choosing the right mode for opening an assembly can have a significant effect on the opening time.
- ▶ **Resolved** mode will open the model and load all of its model data including sketches, dimensions and features.
- ▶ **Lightweight** mode will only open up a sub-set of the models data, so files open much faster.
- ▶ Model data can simply be loaded on demand (Resolved) when changes to sketches and features are required, simply right-click on a component in the Feature Manager Design Tree and choose the required option.



# SOLIDWORKS Tips and Tricks

## 04. Open Mode Options

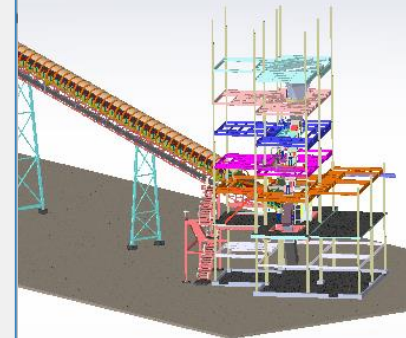
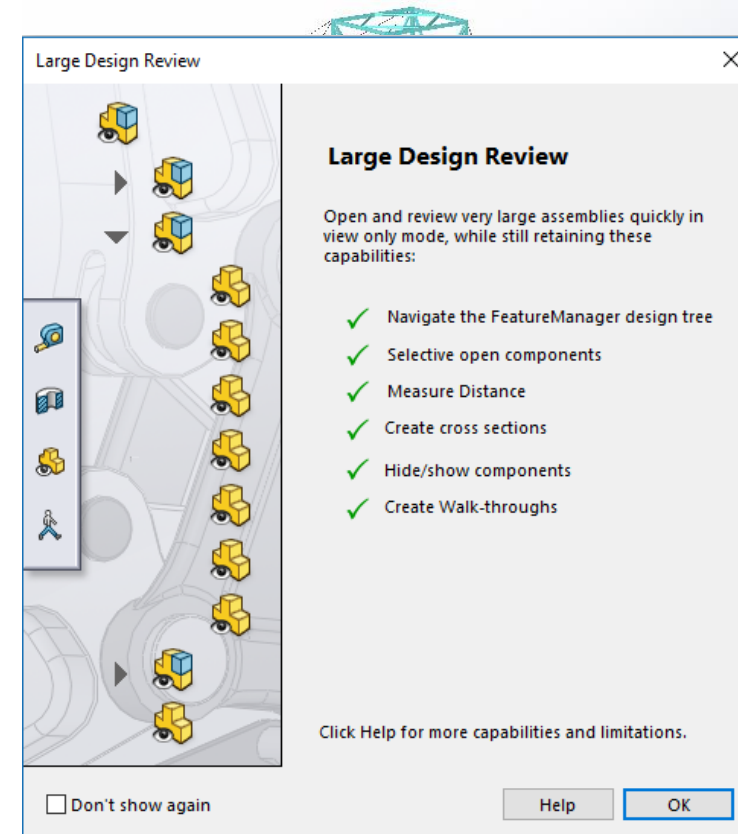
- ▶ Large Assembly Mode will automatically force lightweight mode dependant on the number of components in the assembly. It is set by default to any assembly over 500 components, but we can adjust this number in system options;



# SOLIDWORKS Tips and Tricks

## 04. Open Mode Options

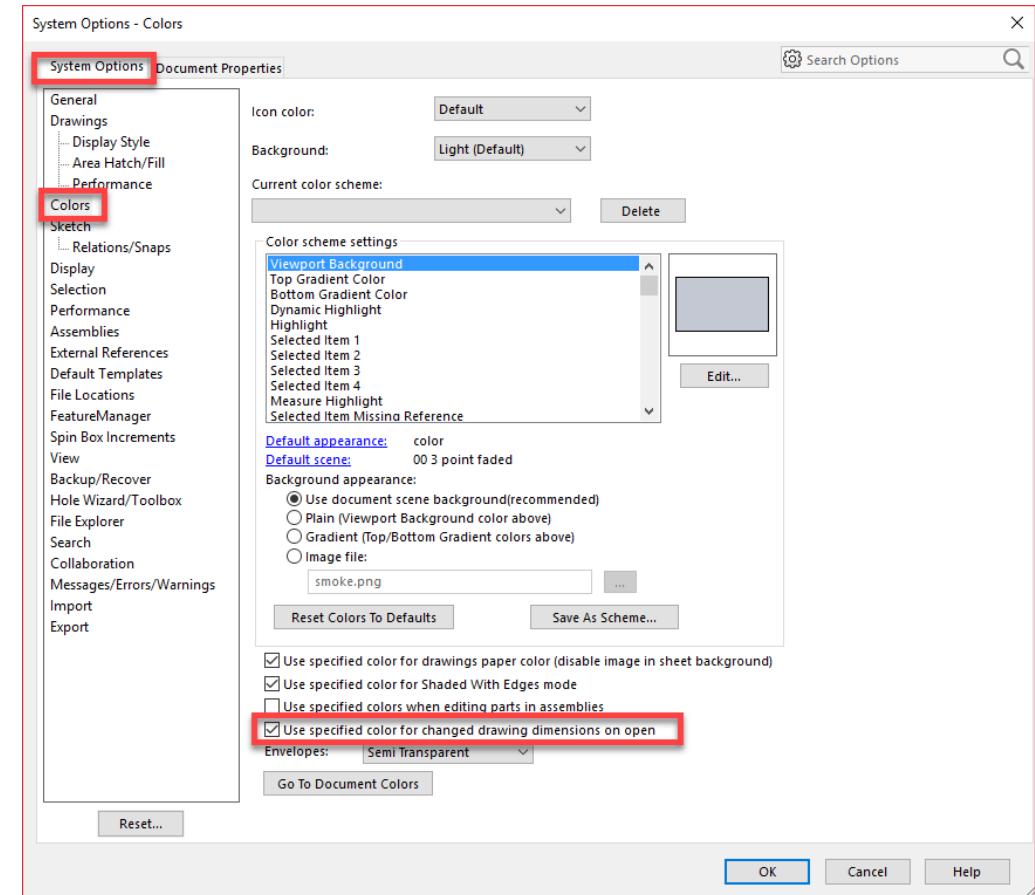
- ▶ **Large Design Review** mode lets you open very large assemblies quickly, while still retaining capabilities that are useful when conducting design reviews of assemblies such as navigating the Feature Manager Design Tree, hide and show, creating sections, measuring and creating walkthrough animations.
- ▶ This is a great tool for reviewing large models, or to quickly identify a model within a very large assembly. As with Lightweight mode we can still load-in the model information (Resolve) on demand.



# SOLIDWORKS Tips and Tricks

## 05. View Changed dimensions in drawings

- ▶ Due to design intent a small dimensional change in a model could alter a large number of dimensions on a drawing, which may subsequently be difficult to identify on the drawing sheet.
- ▶ Under tools>options>colours there is a setting called “Use specified colours for changed dimensions in drawings”

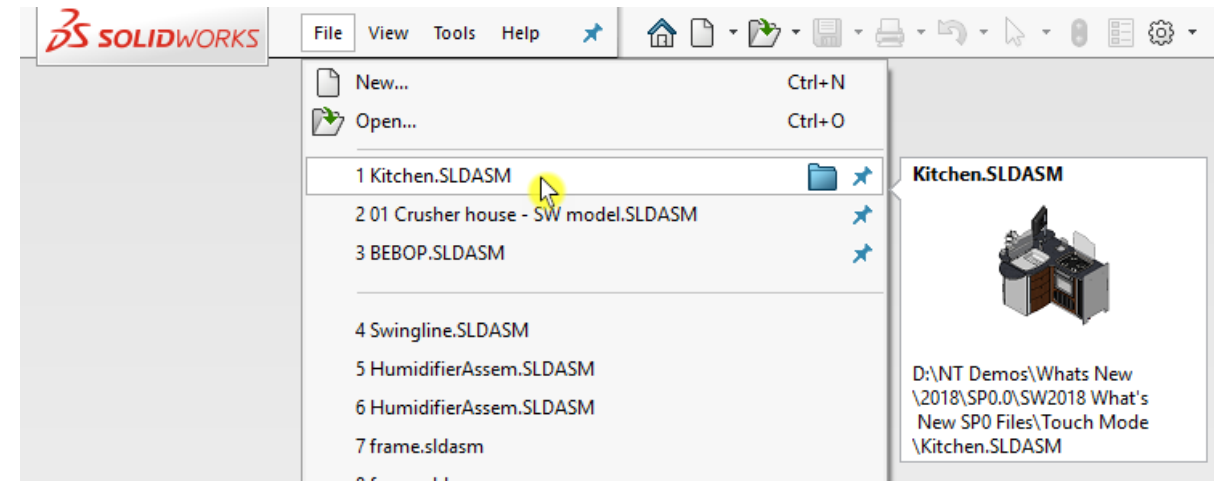




# SOLIDWORKS Tips and Tricks

## 06. Recent and pinned documents

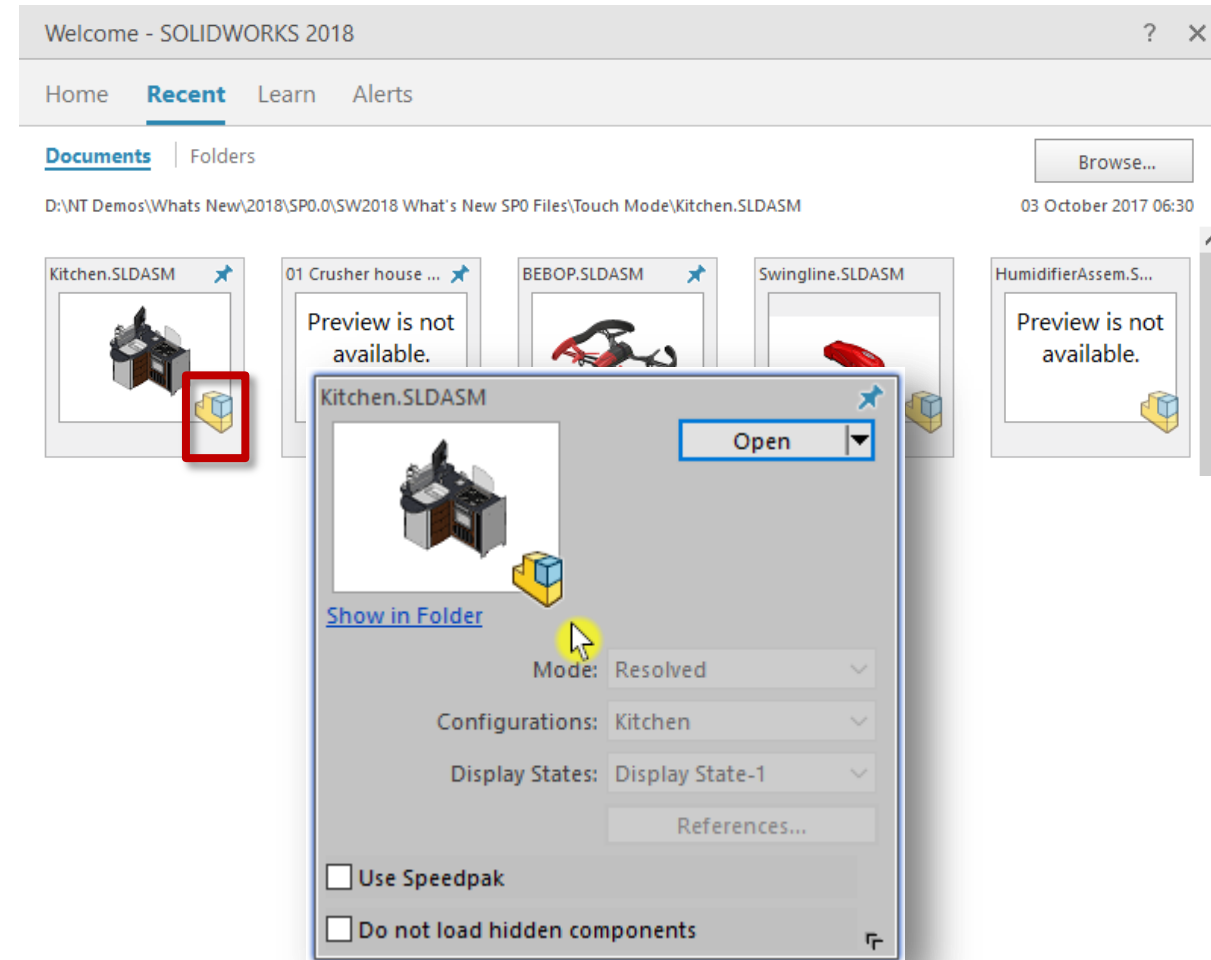
- ▶ To access your recent documents in SOLIDWORKS simply navigate through the **File** menu drop-down where we can see a thumbnail preview, show the file in its folder or pin it down for later quick access.



# SOLIDWORKS Tips and Tricks

## 06. Recent and pinned documents

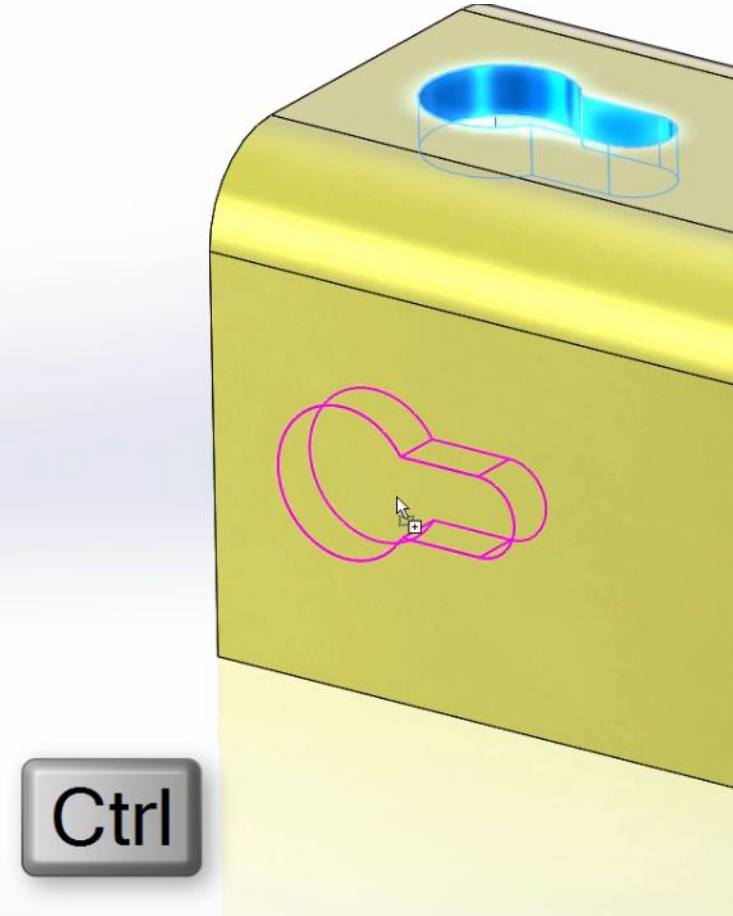
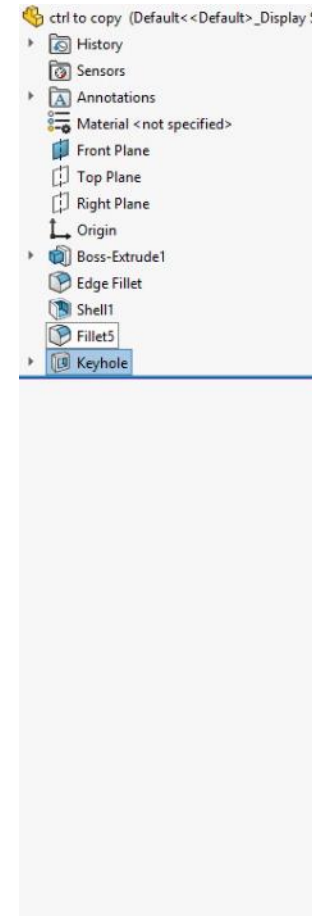
- ▶ For more options use the **R** key on your keyboard, this will open the **Recent** documents tab on the SOLIDWORKS welcome screen.
- ▶ Expand the options to choose different opening modes such as Resolved, Lightweight, Large Assembly Mode and Large Design Review, and also the option to open directly into a Speedpak configuration (which is particularly useful for opening large assemblies)
- ▶ We can also pre-select the configuration and display state to open directly into, rather than waiting for a file to open before switching to another configuration of the model.



# SOLIDWORKS Tips and Tricks

## 07. Copy Features Using CTRL

- ▶ The Ctrl Key acts as a copy command, so dragging a feature from the feature tree back onto your model with CTRL held down will result in a copy of that feature being applied to the face you dragged it to.
- ▶ You can use this really quickly on the fly by dragging a fillet from one edge to another whilst holding control to quickly create a fillet on the new edge. It even works to copy planes and feature in the graphics area (plus in an assembly dragging a component with CTRL held down results in a new instance of that component)
- ▶ SHIFT acts as a move command so grabbing a feature from the tree with shift held down will move it to a different face of the model – much quicker than 'edit sketch plane'

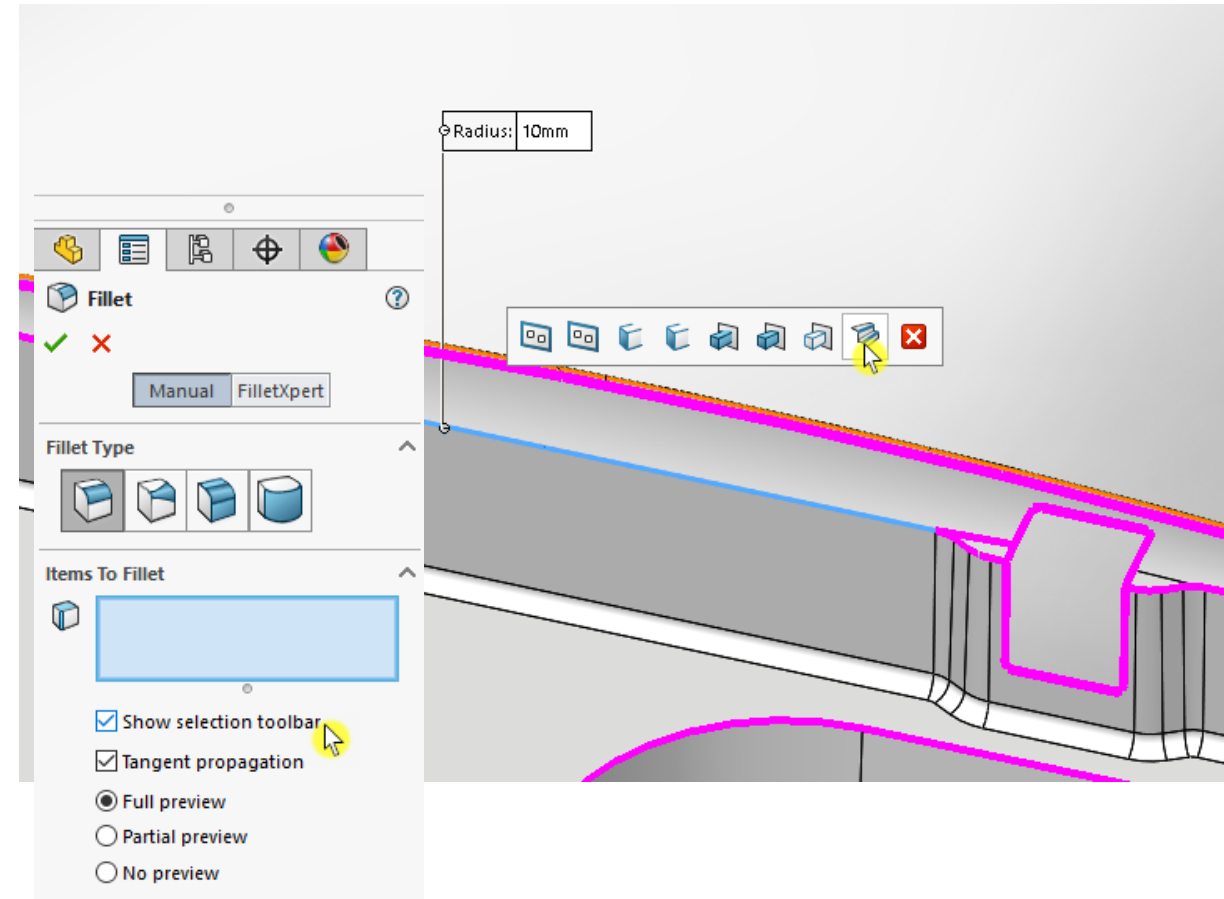




# SOLIDWORKS Tips and Tricks

## 08. Quick Selection Toolbars for Fillets

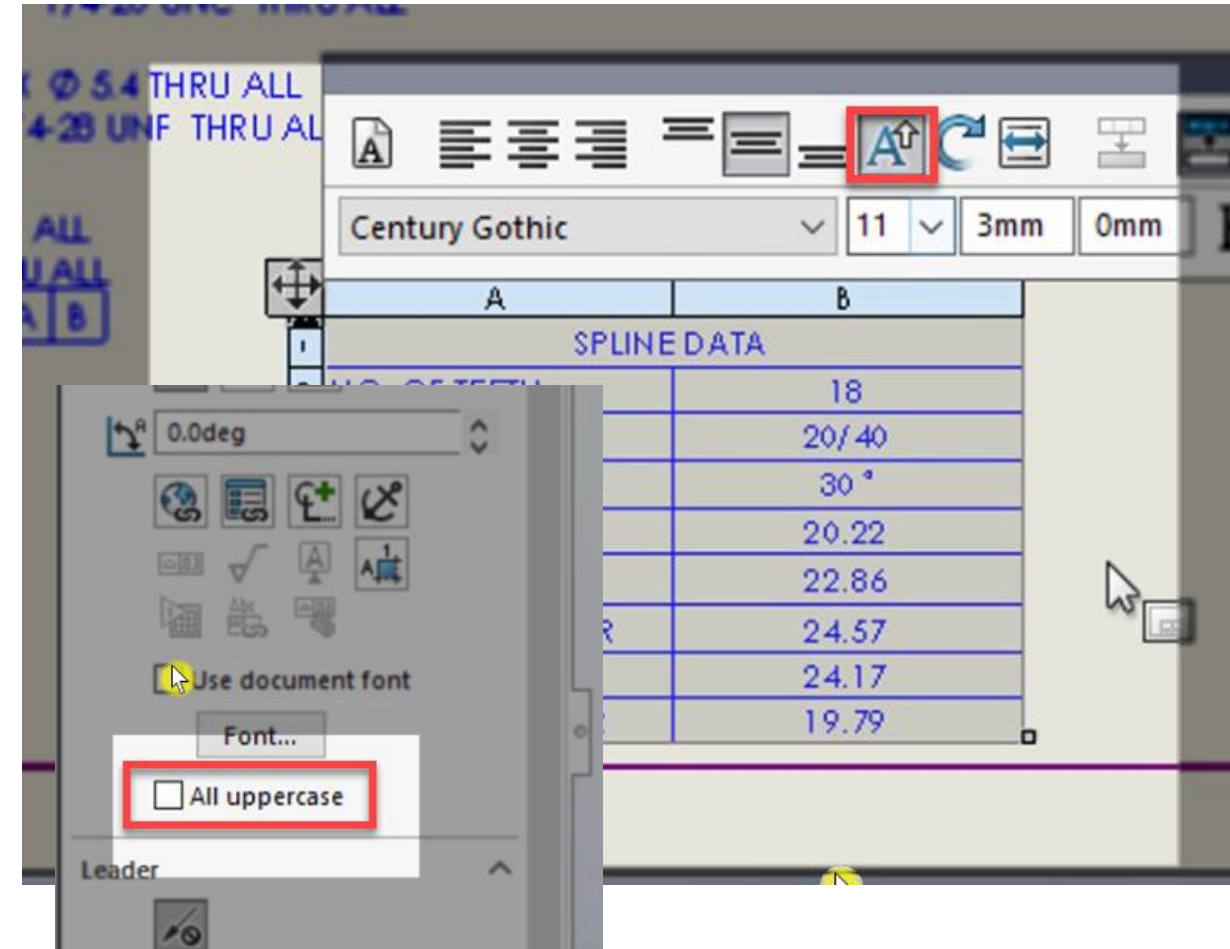
- ▶ When filleting a model check the **Show Selection Toolbar** option on the fillet property manager to enable fast selection of multiple edges for Constant or Variable radius fillets.
- ▶ The pop-up selection toolbar shows us which edge selections are available and the number of edges.
- ▶ Here we see options to select loops of edges, connected edges and edges that are convex to the selection, allowing us to fillet multiple edges in one simple operation.



# SOLIDWORKS Tips and Tricks

## 09. All Uppercase Text

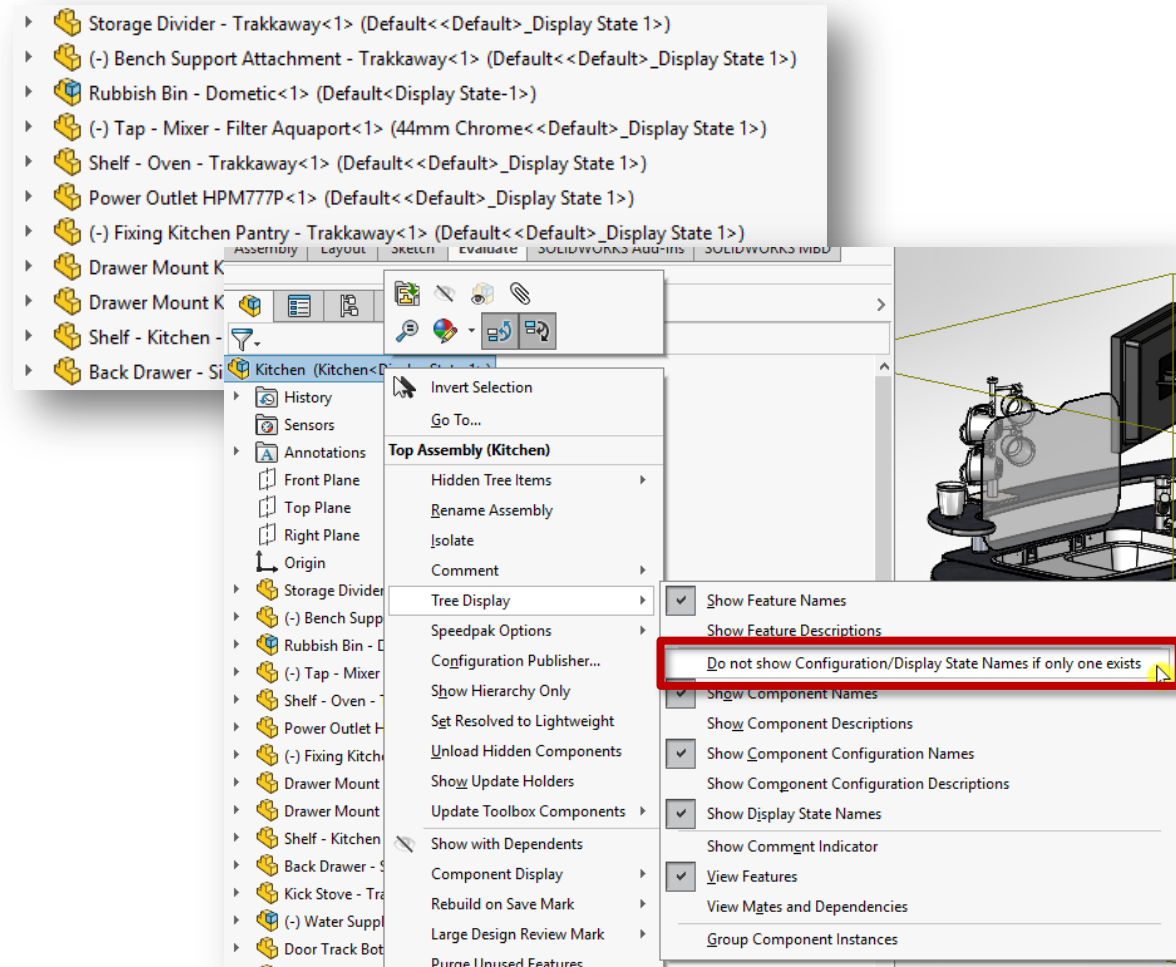
- ▶ Lower case text is traditionally not used on engineering drawings and some industries strictly forbid its use. This can pose a problem if text is formed from custom properties that may be time consuming to rectify.
- ▶ New for SW2018 is a table option to convert the text to uppercase at table level, rectifying this potential problem instantly!
- ▶ For note text there is a property manager button to convert text to uppercase which again is a quick way of solving what could be a time consuming rectification process.



# SOLIDWORKS Tips and Tricks

## 10. Feature tree Display

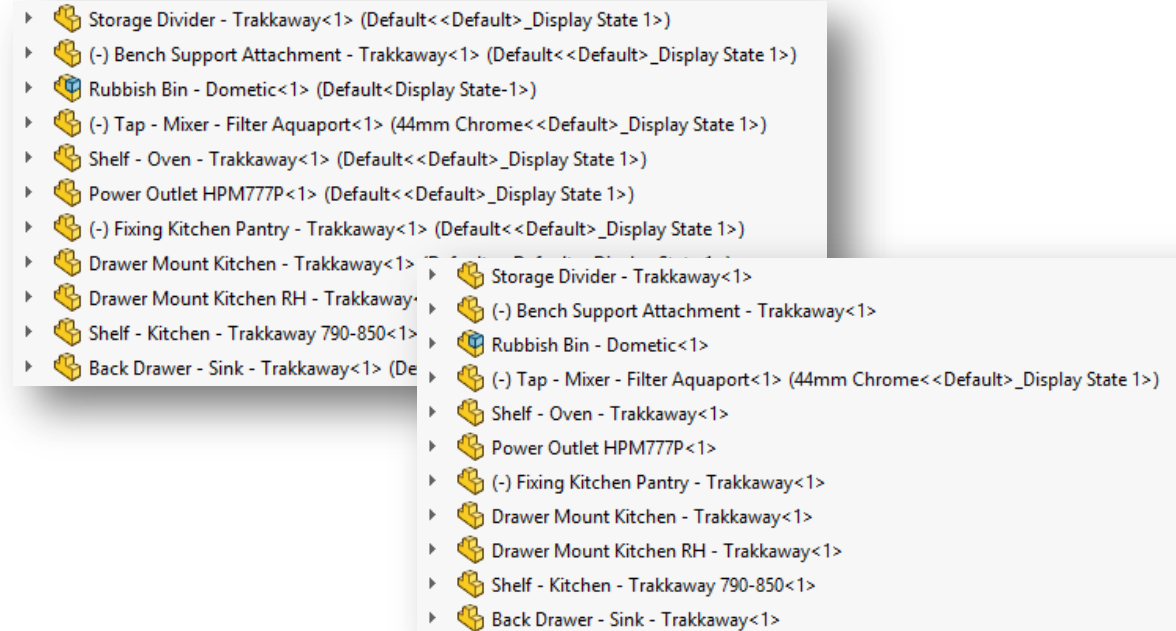
- ▶ SOLIDWORKS shows the Configuration and Display State name for each of the model items in the Feature Manager Design Tree.
- ▶ Whilst this is important information it can make the tree information look cluttered.
- ▶ Right click on the top level assembly in the Feature Manager Design Tree and in **Tree Display** options we can choose **Do not show Configuration/Display State Names if only one exists**.



# SOLIDWORKS Tips and Tricks

## 10. Feature tree Display

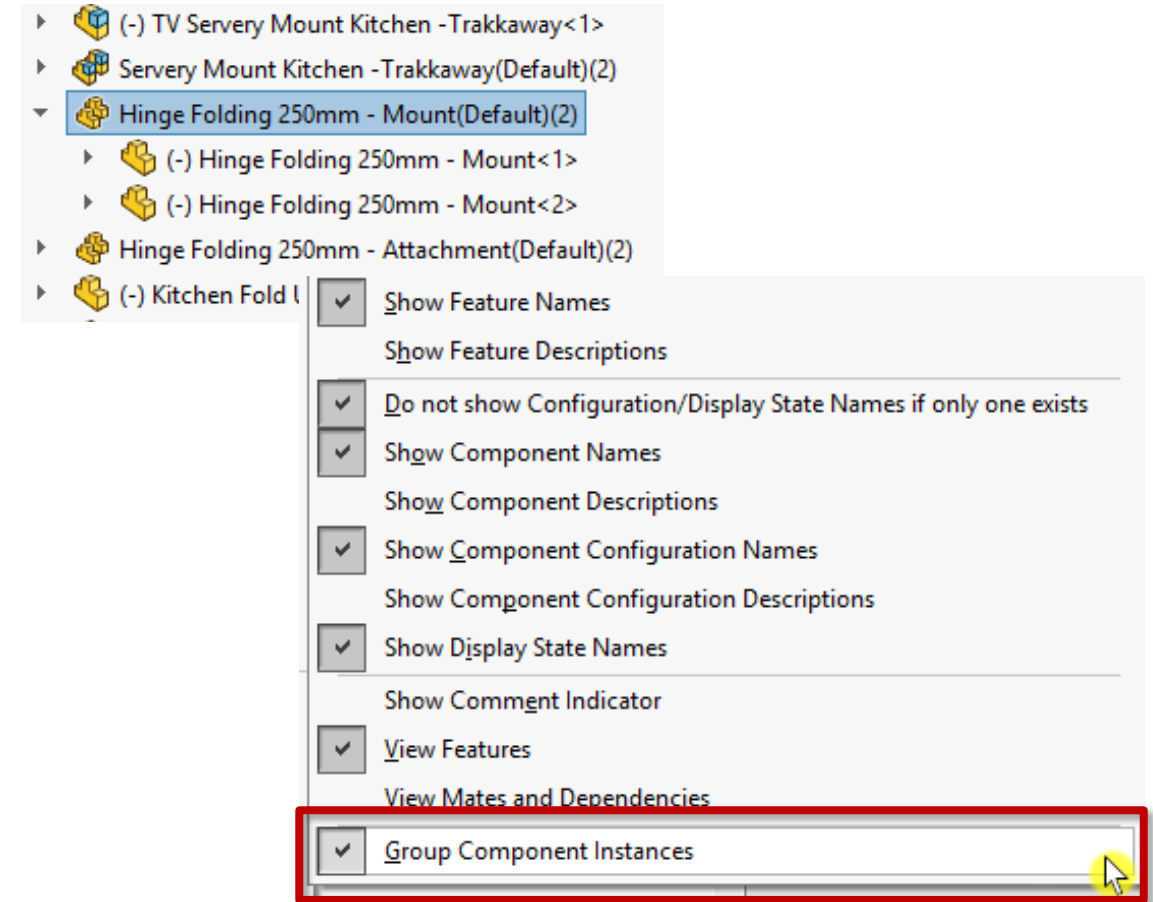
- ▶ This option makes for a much tidier tree that is easier to understand.



# SOLIDWORKS Tips and Tricks

## 10. Feature tree Display

- ▶ Another useful Tree Display option is to Group Component Instances.
- ▶ Components of the same configuration and display state will be listed together in a folder structure in the Feature Manager Design Tree, rather than being listed separately, making them much easier to find and shortening the tree.



# SOLIDWORKS Tips and Tricks

## 11. Magnifying Glass

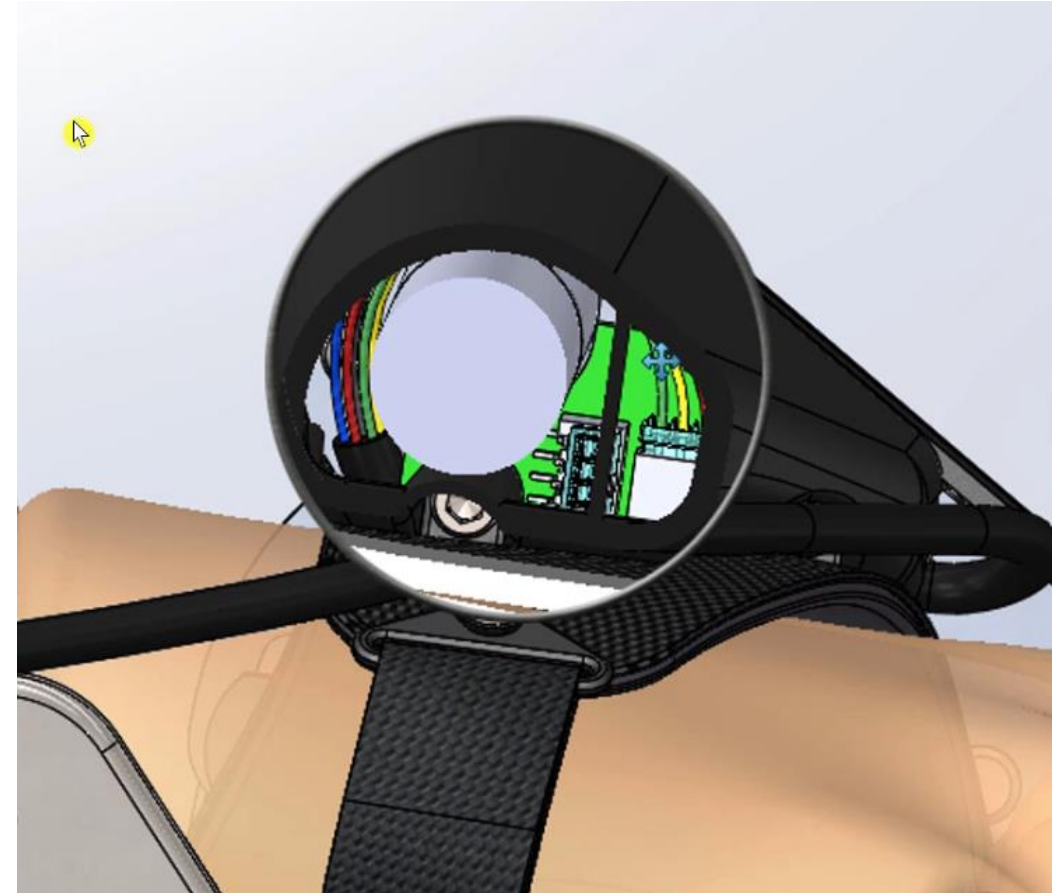
- ▶ The G Key in SOLIDWORKS opens up a magnifying glass allowing you to zoom, pan and rotate within a magnified area of the screen using your standard mouse interactions.
- ▶ This is incredibly useful when trying to make selections of small edges or faces on a model without zooming the entire model. Its also great for assisting with adding dimensions to edges in a drawing.



# SOLIDWORKS Tips and Tricks

## 11. Magnifying Glass

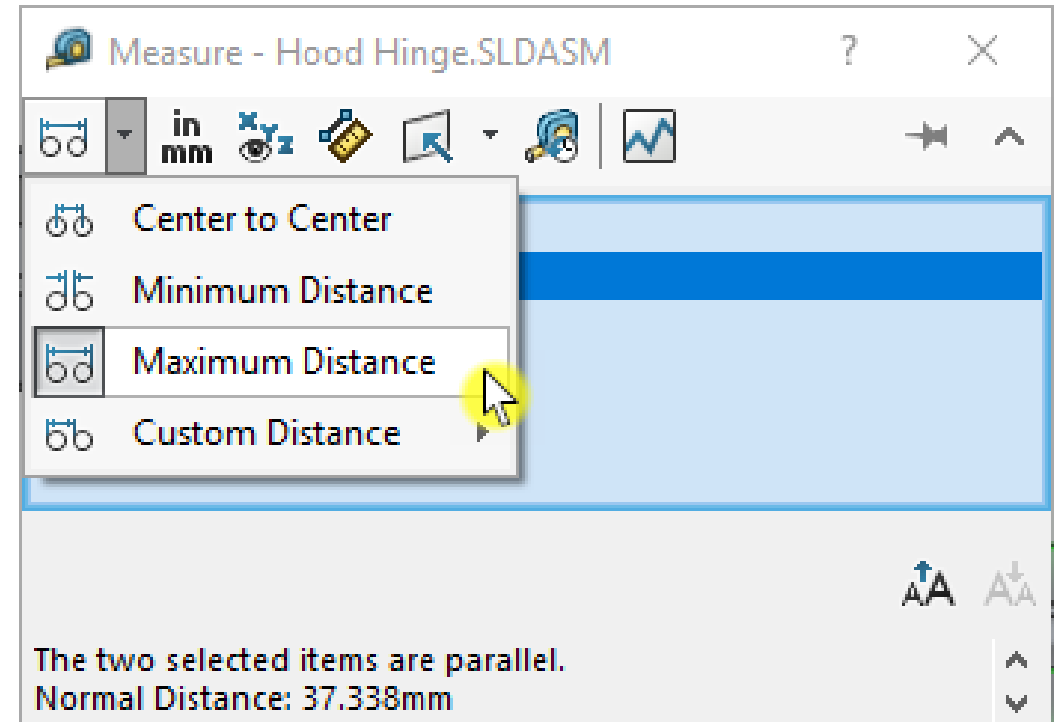
- ▶ Holding the ALT key down in the magnifying glass creates a dynamic section normal to the screen to give you a great insight into the model without resorting to a full section view.
- ▶ Click off the geometry and the magnifying glass will disappear.



# SOLIDWORKS Tips and Tricks

## 12. Measure tool options

- ▶ The measure tool has several options found under the drop-down menus.
- ▶ Users can choose to measure selected entities from Centre to Centre, or measure the Minimum or Maximum Distance between faces, edges or vertex rather than the normal distance.

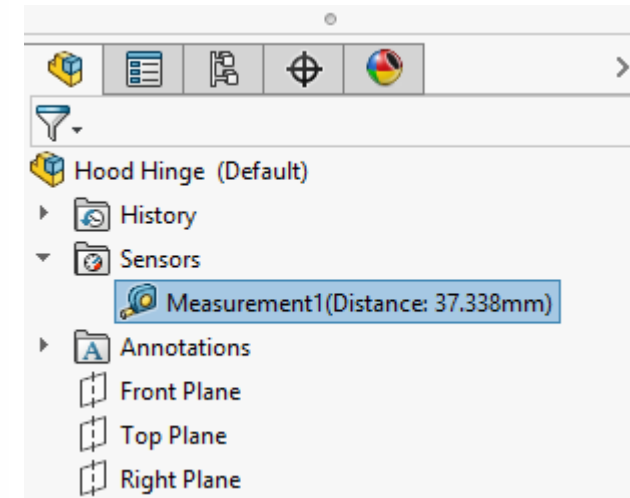
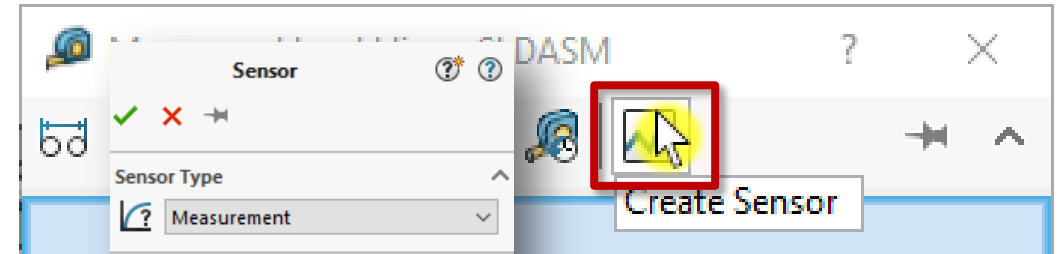




# SOLIDWORKS Tips and Tricks

## 12. Measure tool options

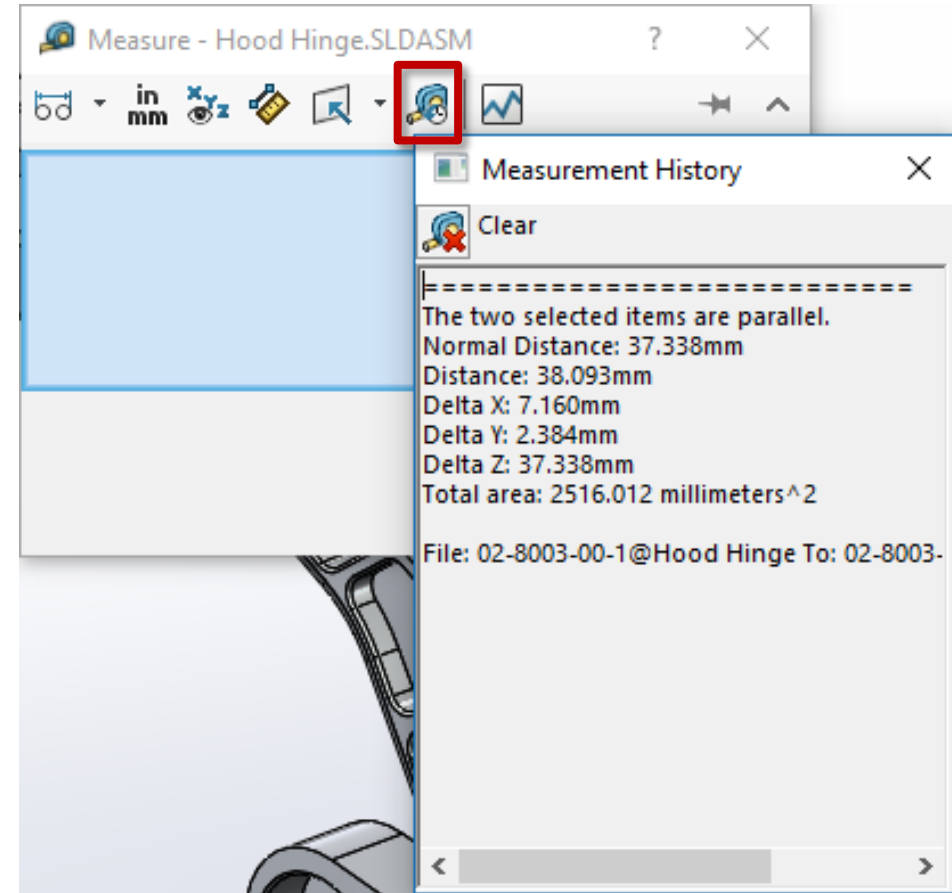
- ▶ We can also create a Sensor from the measurement that has just been taken, for example to alert us if a measurement is outside of a certain range of allowable values.



# SOLIDWORKS Tips and Tricks

## 12. Measure tool options

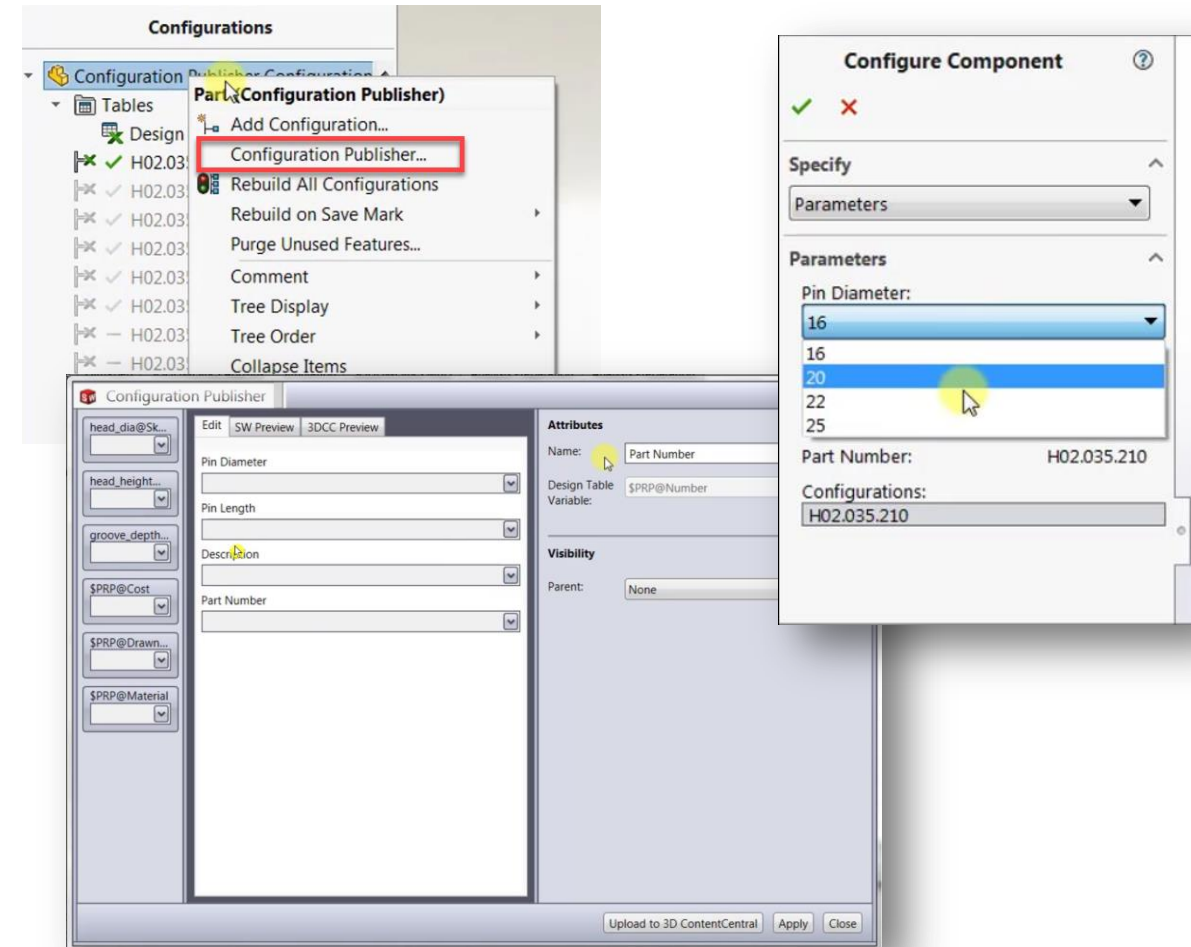
- ▶ ...and a full history of the measurements taken is available to view for reference.



# SOLIDWORKS Tips and Tricks

## 13. Configure Components

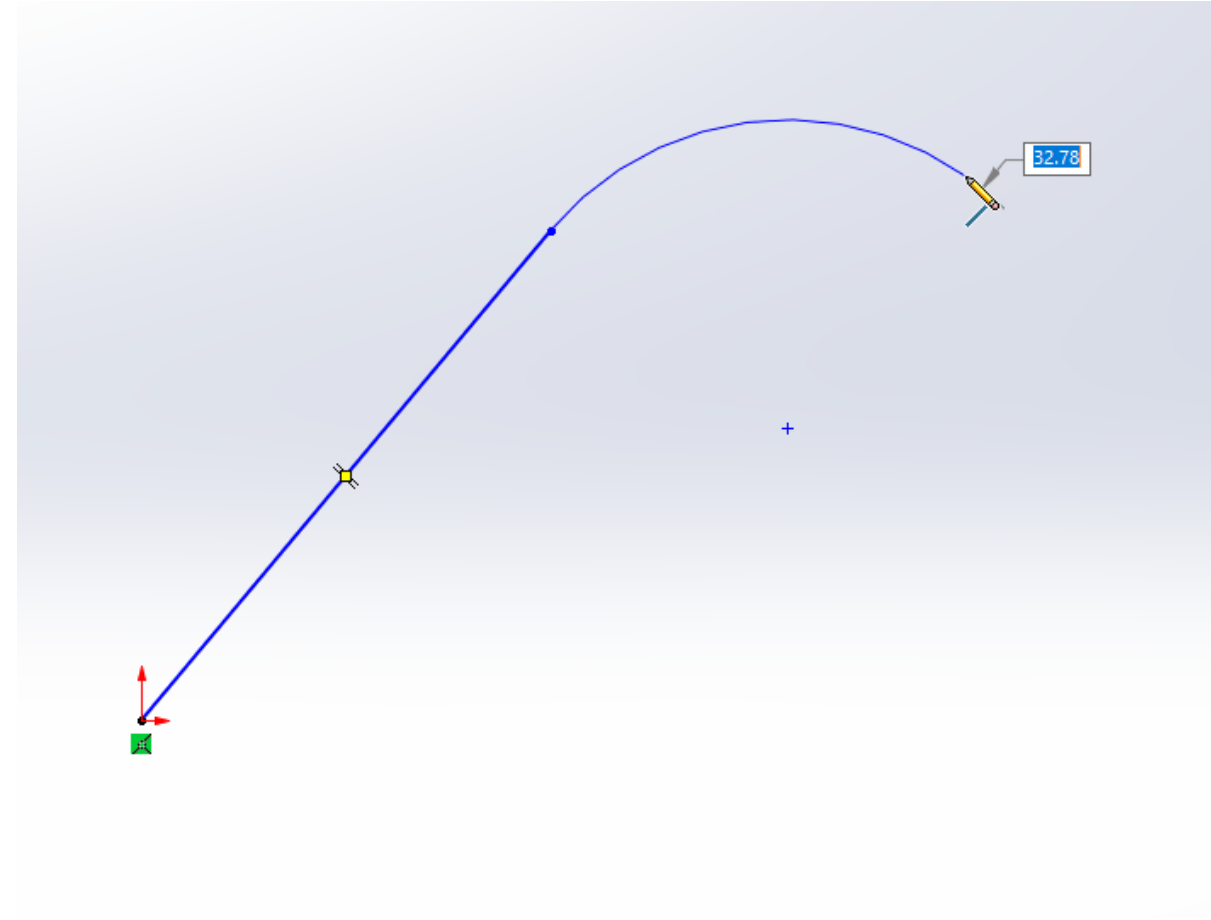
- ▶ Often when working with configurations we use configuration or part numbers to identify the configurations, however this doesn't always help us to identify what the configuration actually is with regards to dimensions and geometry.
- ▶ Using Configure Component we can create an easy to use form to allow us to select our requirements (for example length and diameter) and let SOLIDWORKS pair this up with an available configuration.
- ▶ With this form setup when you add that component to an assembly instead of just using the active configuration, configure component displays the form and allows us to select our requirements.



# SOLIDWORKS Tips and Tricks

## 14. Auto-transitioning with the A key

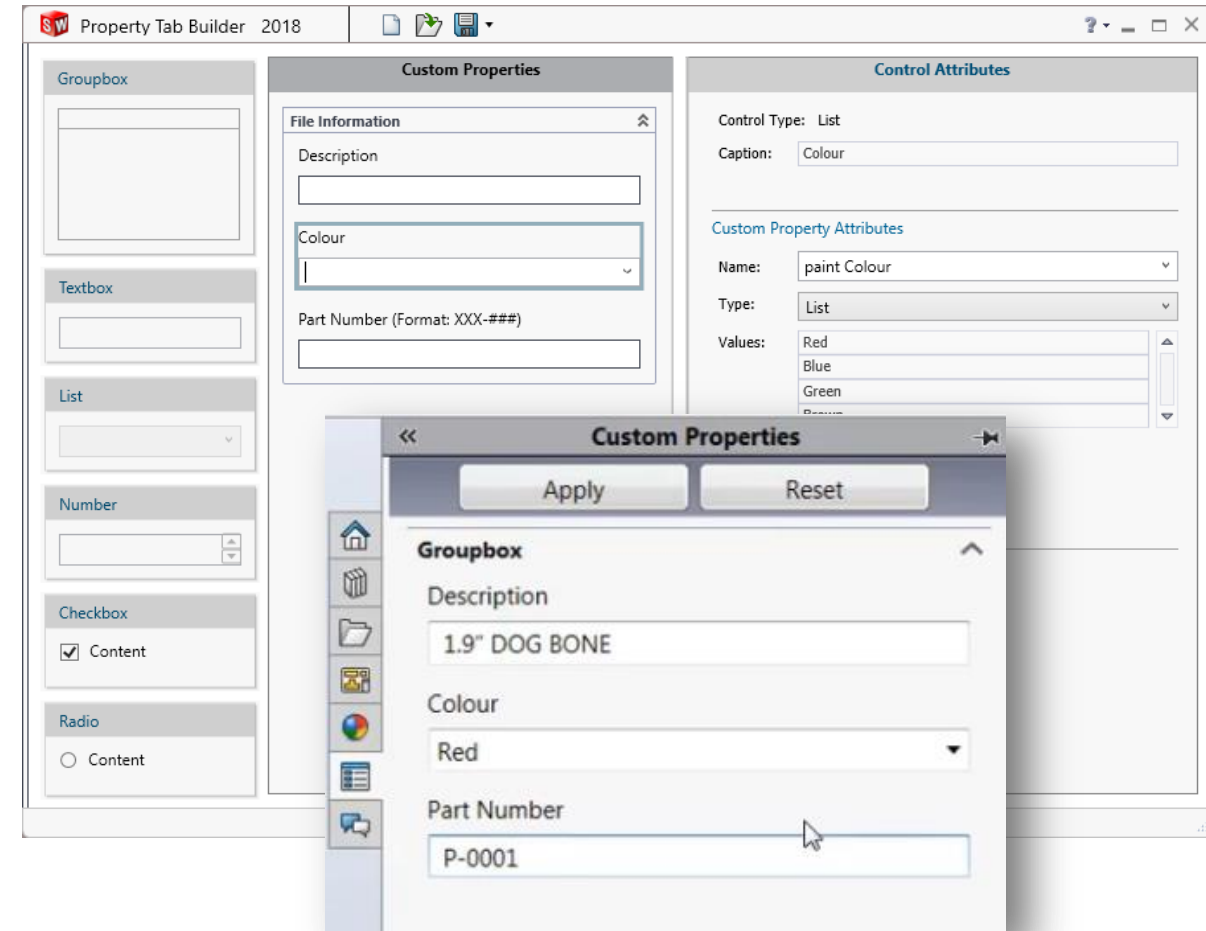
- ▶ When drawing lines and arcs in a sketch, you can easily toggle between the two different entities using the **A** key as a shortcut.
- ▶ This allows us to create lines with tangent arcs with minimal effort.
- ▶ Just hit the **A** key to switch from arc to line, and vice versa.



# SOLIDWORKS Tips and Tricks

## 15. Property Tab Builder

- ▶ The property tab builder is a great way of creating forms that sit in the task pane and make populating custom properties easy and user friendly.
- ▶ These are created in a standalone application called the Property Tab Builder (found in the Windows start menu SOLIDWORKS folders).
- ▶ Here we can assign text boxes, list boxes (you can even link directly to an excel table of available options) to create the form we want.
- ▶ Once saved the property tab sits in the task pane and allows us to populate our custom properties quickly and easily.



# Thank You



*3D* SOLIDWORKS

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