



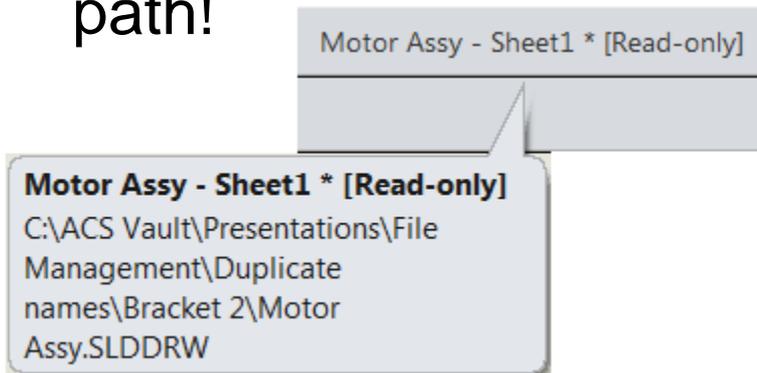
Applied CAD Solutions LLC

Tips & Tricks

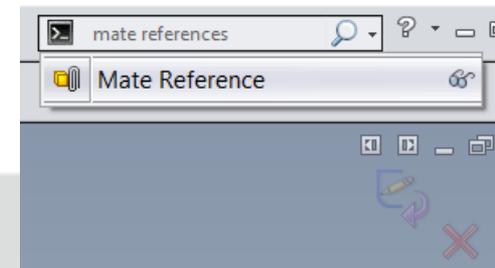
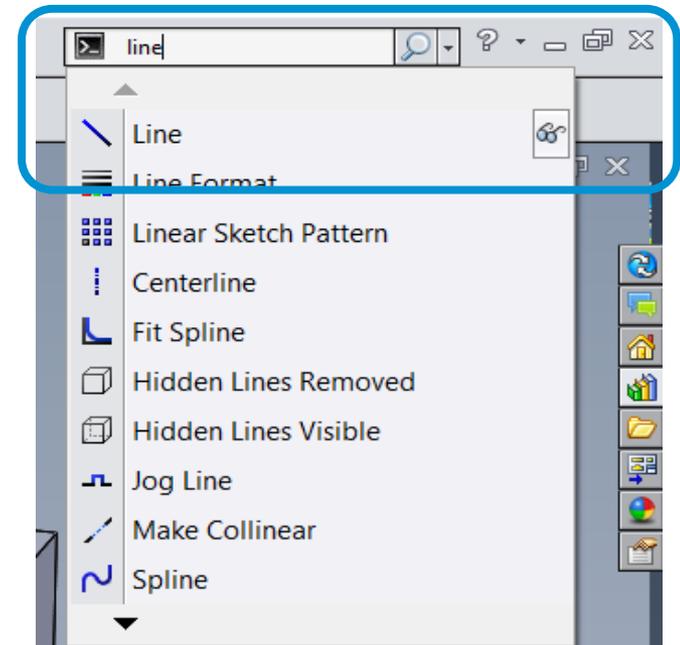
by Tom Cote

Tips & Tricks

- Most people use File > Save-As to determine file location. But you can hover over the file name to see the path!

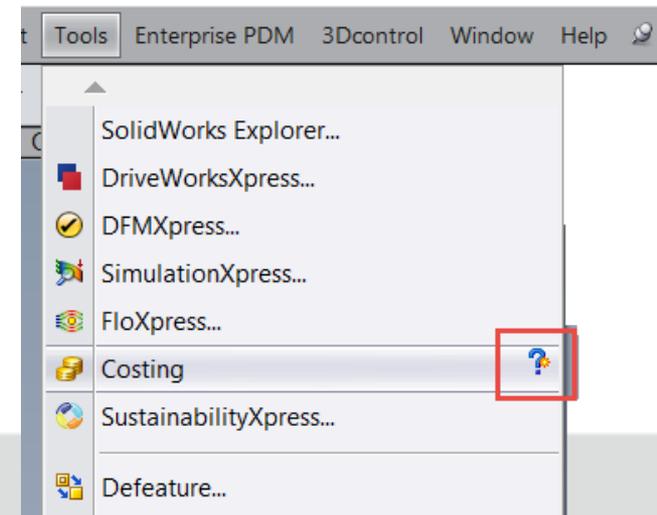
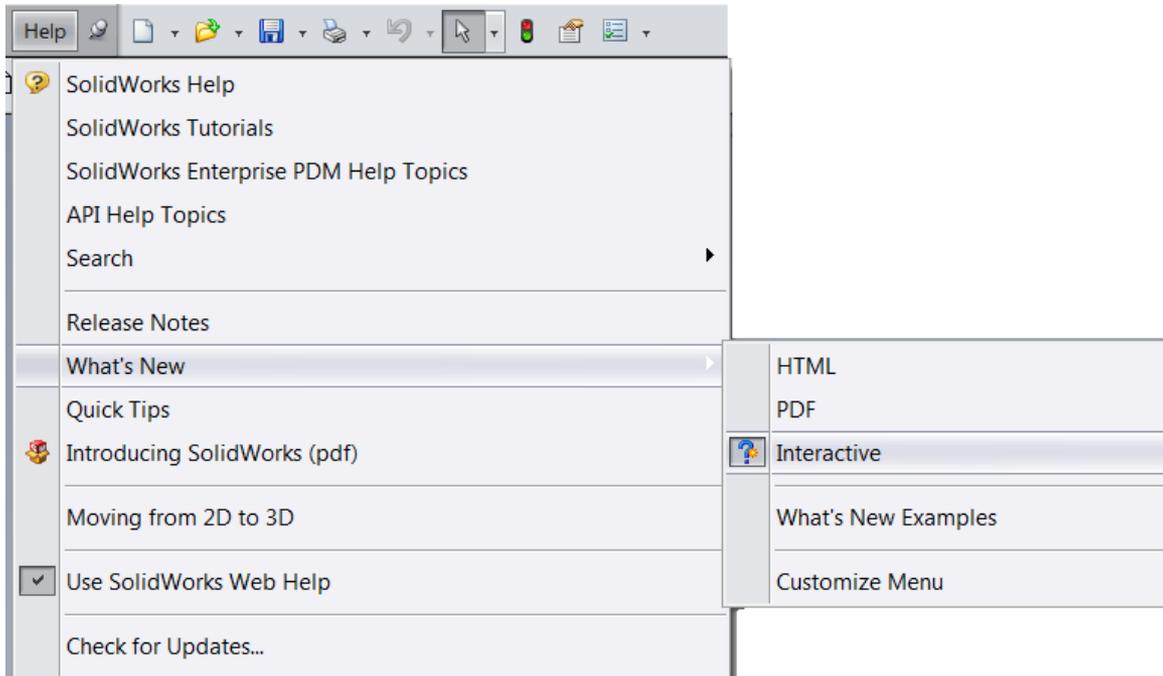


- Expand All Tree
- Using Help and showing command



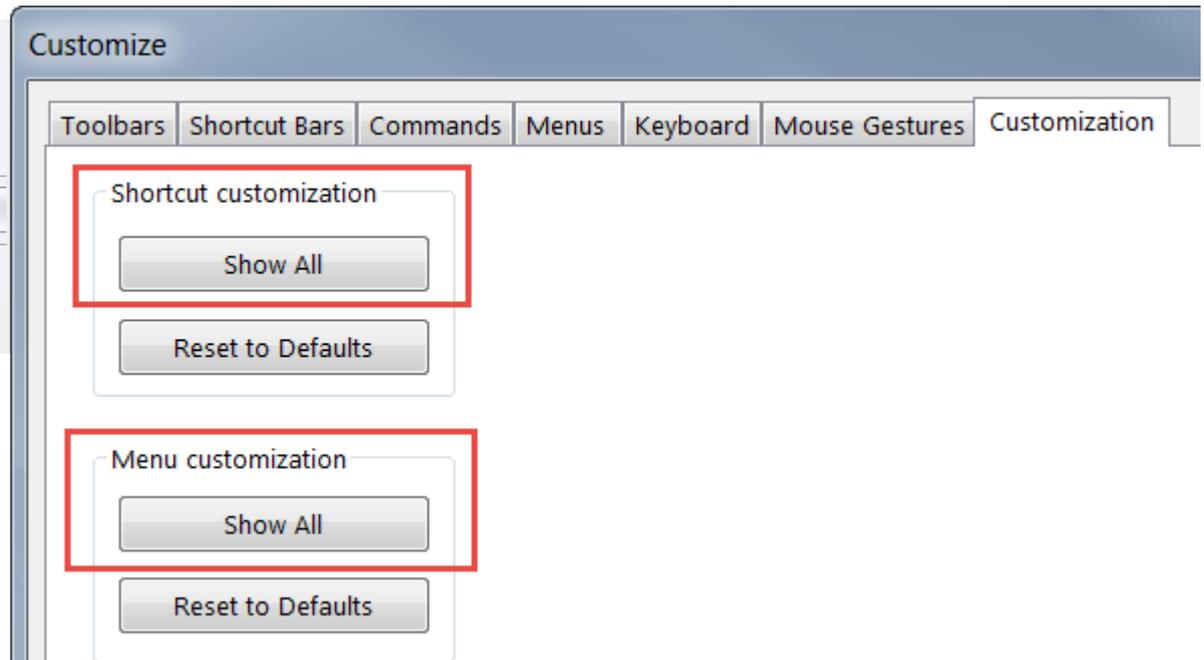
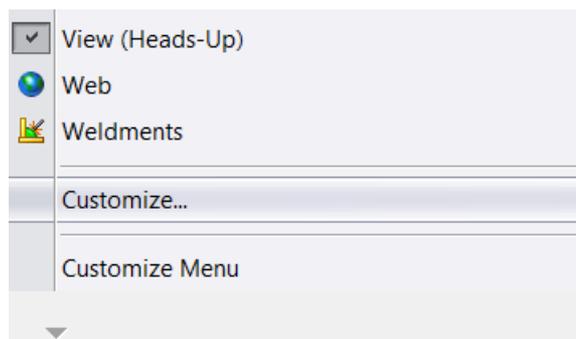
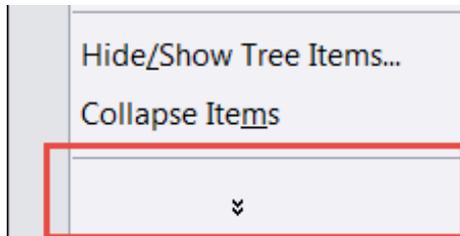
Tips & Tricks

- Interactive “What’s New” help



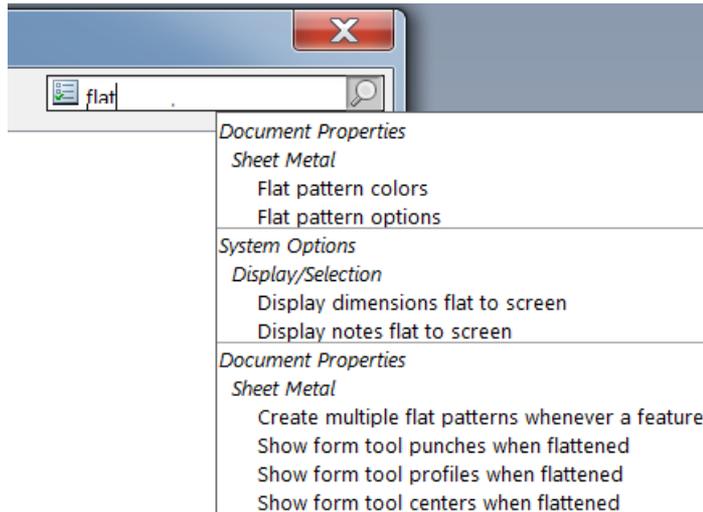
Tips & Tricks

- Eliminate the “short” menu

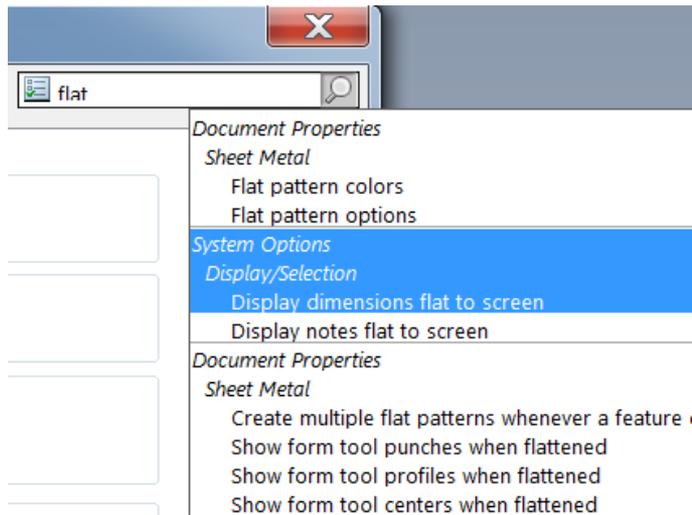


SKETCH

Search in Options



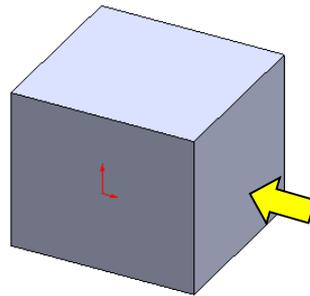
Dim's and Notes flat to Screen



- Display dimensions flat to screen
- Display notes flat to screen

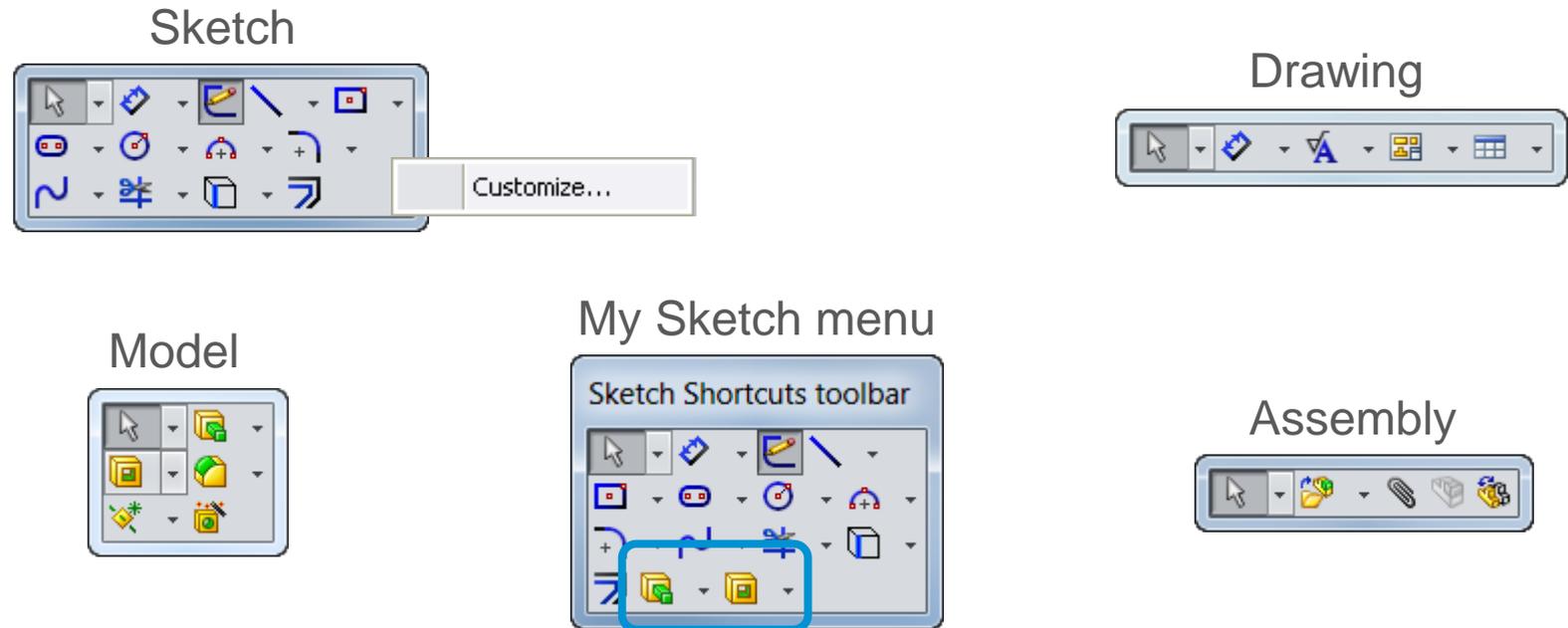
Start a Sketch on a Surface

- You already know that you can pick a face, and a little menu pops up next to the cursor and you can start a sketch
- But did you know that by picking a face, then pressing the “L” key it will start a sketch, AND start the “Line” command? This works for all sketch command shortcuts



Sketch

- Context sensitive menu's (shortcut bar), **the “S” key**
- This menu will change depending upon what you are doing inside of SolidWorks.
 - You can customize this menu, Right-Click in the command area



Sketch

- When drawing a rectangle, circle, etc... you can toggle to a different one by the “a” key
- Easily build an equation into a dimension
- When drawing a line in a sketch, press the Alt-C keys to convert the line(s) to construction
- When drawing a line in a sketch, press the Alt-H keys to convert the line(s) horizontal or Alt-V for vertical.
- Use the shift key when dimensioning to the Min or Max of a slot, two holes, etc...

Sketch

- It's can be a pain and take multiple mouse clicks to move sketch entities. It's easy by selecting the entities & holding the Ctrl key. Start “dragging” the sketch feature (it will look like they are being copied) but let go off the Ctrl key before the left mouse button.

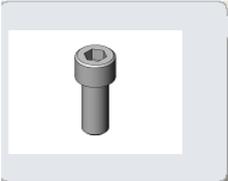
MODELS

Model

- When extruding in a model double-click on a face (surface) to automatically change end condition from “Blind” to “Up to Surface”
- Text on a face, there is a new font in 2014 that you cannot extrude but keeps file size smaller.

ToolBox Flag

- Icon is different if file is “flagged” as a ToolBox part



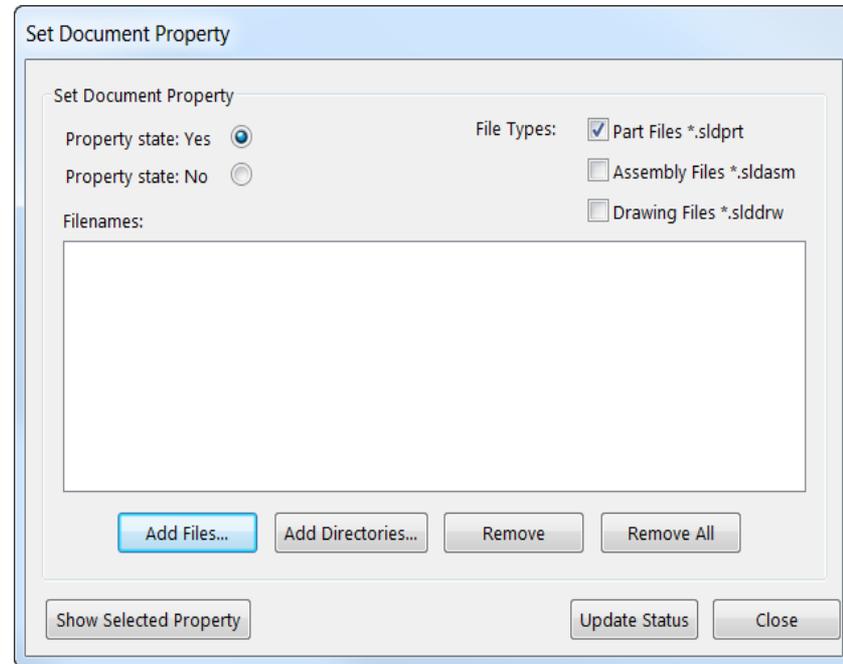
		A	B	C
1		9	44001057	SCREW, SHCS, SS, 1/2-13 X 1.25
2		8	123456	123456 test
3		7	234567	234567 Test
4		6	345678	345678 test
5		5	456789	456789 test
6		4	567890	567890 test
7		3	678901	678901 tes
8		2	789012	789012 test
9		1	8901234	8901234 test
10		ITEM NO.	PART NUMBER	DESCRIPTION
11		BILL OF MATERIAL		

Clear ToolBox Flag

- By default, models made using the SOLIDWORKS ToolBox feature have a different part type flag associated with them. The flag can/will change the way files are handled by both Workgroup (WPDM) and Enterprise PDM (EPDM) vault.
- The ToolBox flag can be added or removed to any SOLIDWORKS file by running a file called **sldsetdocprop.exe**, which is usually located inside a directory such as C:\Program Files\SolidWorks Corp 20??\SolidWorks\Toolbox\data utilities.

Clear ToolBox Flag (cont)

- To change the flag setting:
- Run the **sldsetdocprop.exe** file

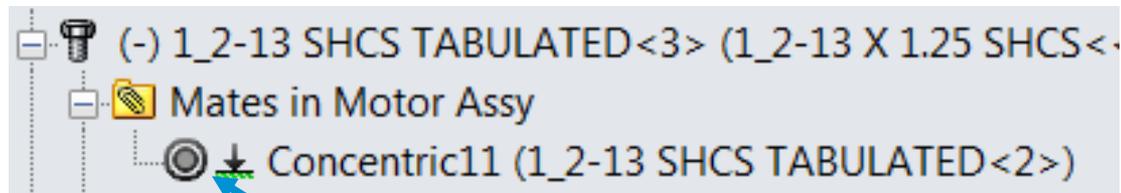
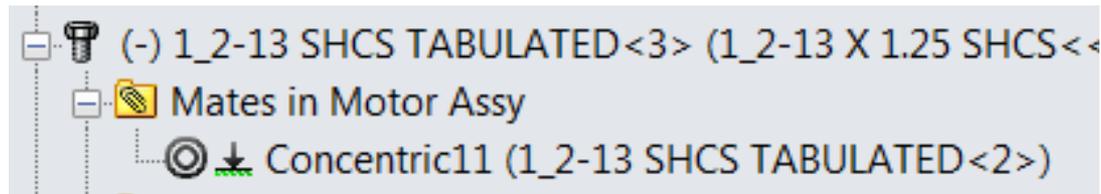


- Select the **ADD Files...** button
- Navigate to the files that you wish to change the flag on.
- Select the desired Set Document Property radio button
 - Property state: **Yes** (add the ToolBox flag)
 - Property state: **No** (remove the ToolBox flag)
- Select **Update Status**
- Then **Close**

ASSEMBLY

ASSEMBLY

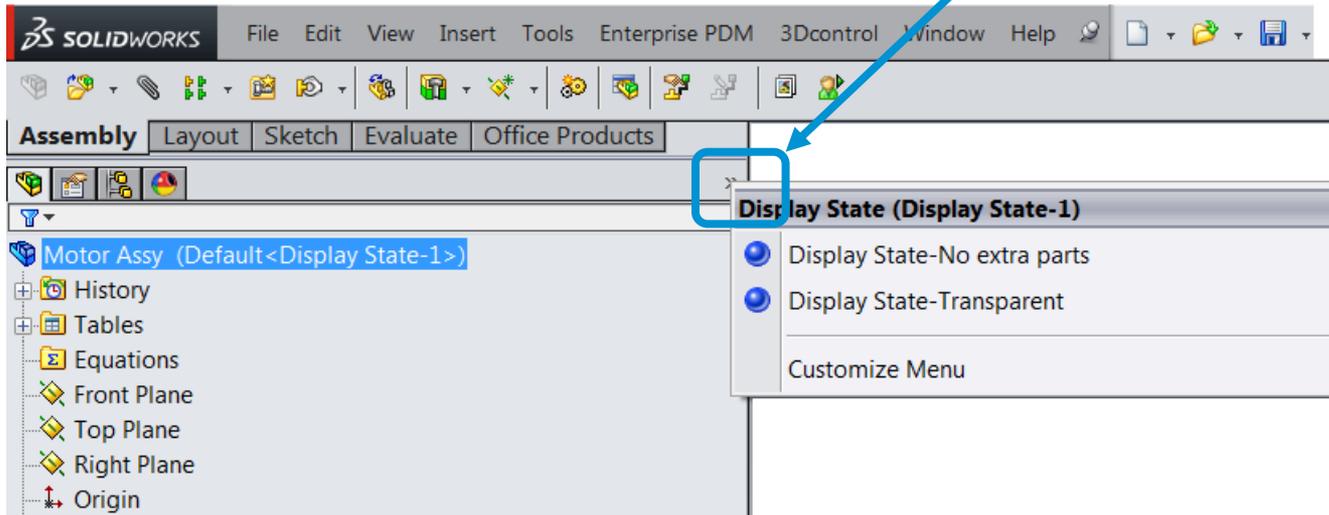
- MATES: Lock view rotation, new in 2014. There is a little “quirk” depending on how you select the mate command.



Notice the Icon Changes

Change Display State view

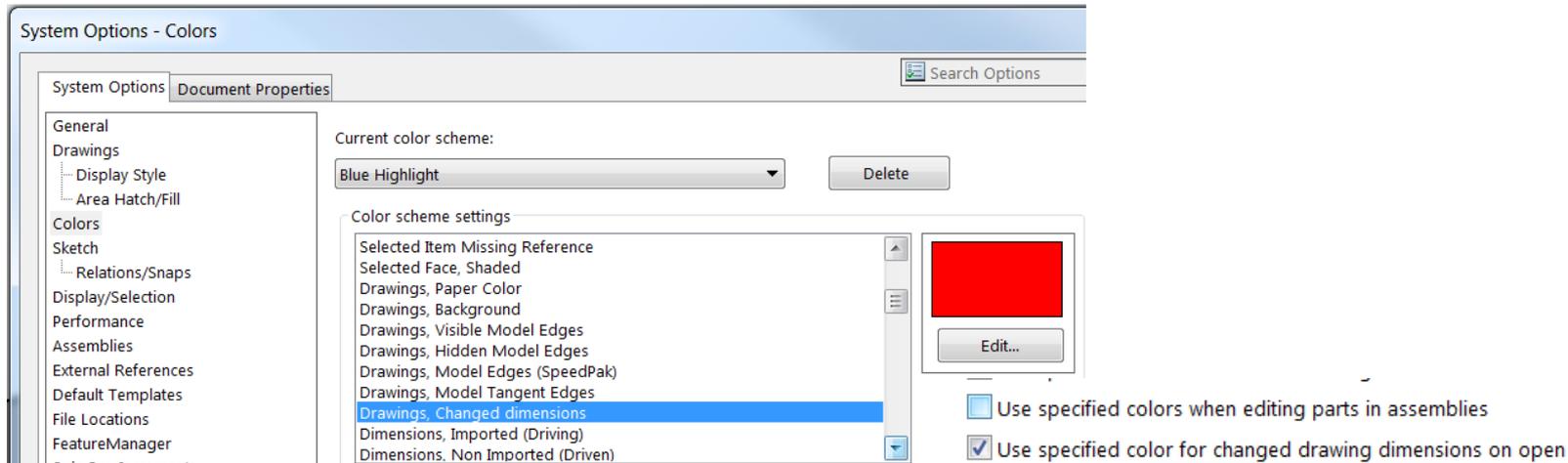
Right-Click on the >>



DRAWING

Drawing

- Use the Alt key to easily select views, BOM's, tables, etc...
- Change the color of “changed” dimensions in the options box.



- Break view alignment by holding down the Ctrl key
- Rotate view by using “relative to Model”

Drawing

- You can look for parts not ballooned in an assembly drawing by expanding the BOM on the drawing and looking for the items without a balloon symbol

Symbol indicating if item has a balloon on the drawing

		A	B	C
1	 	9	44001057	SCREW, SHCS, SS, 1/2-13 X 1.25
2	 	8	123456	123456 test
3		7	234567	234567 Test
4	 	6	345678	345678 test
5		5	456789	456789 test
6		4	567890	567890 test
7	 	3	678901	678901 tes
8	 	2	789012	789012 test
9	 	1	8901234	8901234 test
10	 	ITEM NO.	PART NUMBER	DESCRIPTION
11		BILL OF MATERIAL		



Questions

On anything I presented today?



Applied CAD Solutions LLC

Thank You!

Tom Cote

tcote@appliedcadsolutions.com

www.AppliedCADSolutions.com