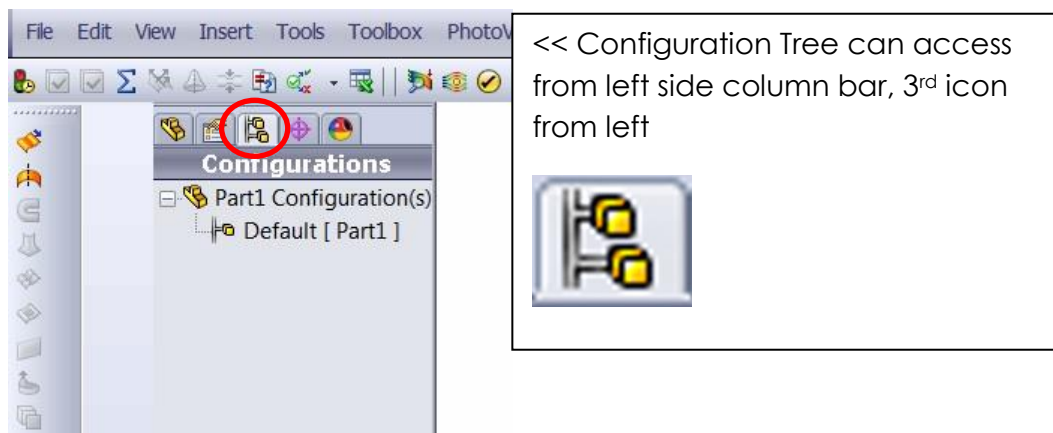
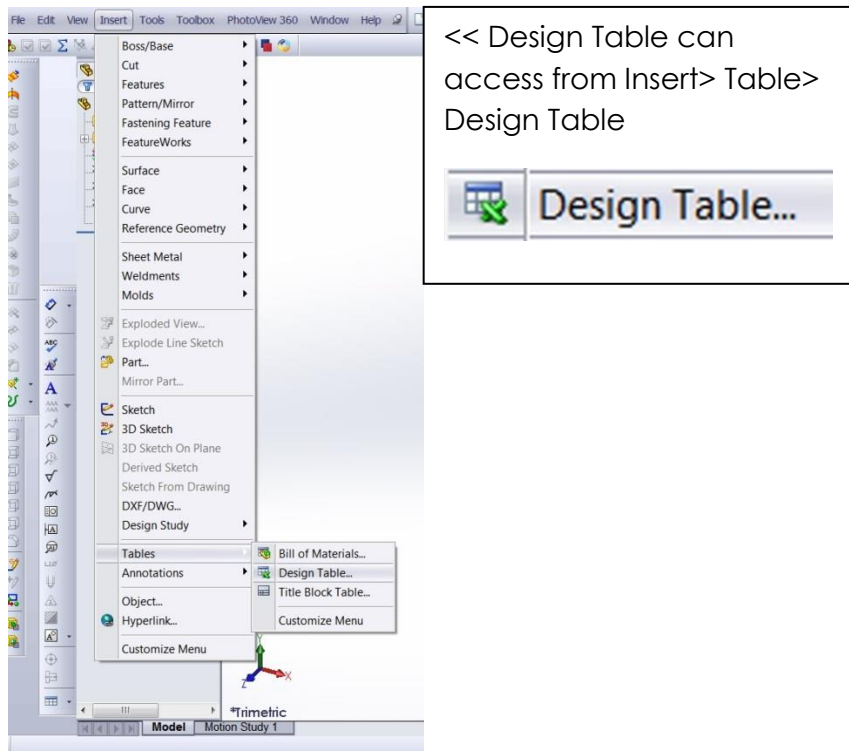


Tutorial>> Solidworks™ >> Design Table & Configuration.

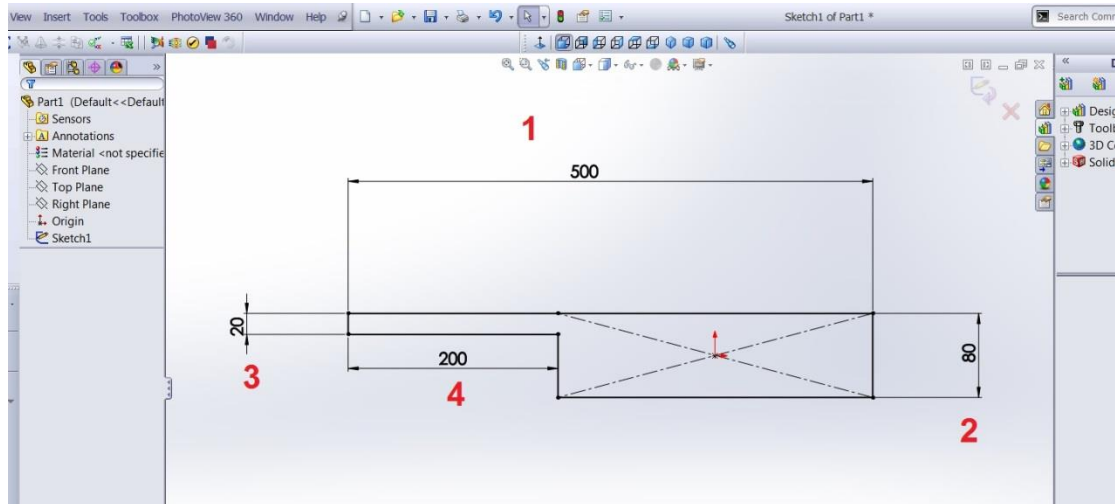
What is Design Table & Configuration?

Design Table & Configuration is one of the features in Solidworks™ which helping users to create a several versions, sizes, design etc., just in single file either Part (.SLDPRT), Assembly (.SLDASM) or Drawing (.SLDDRW) files.

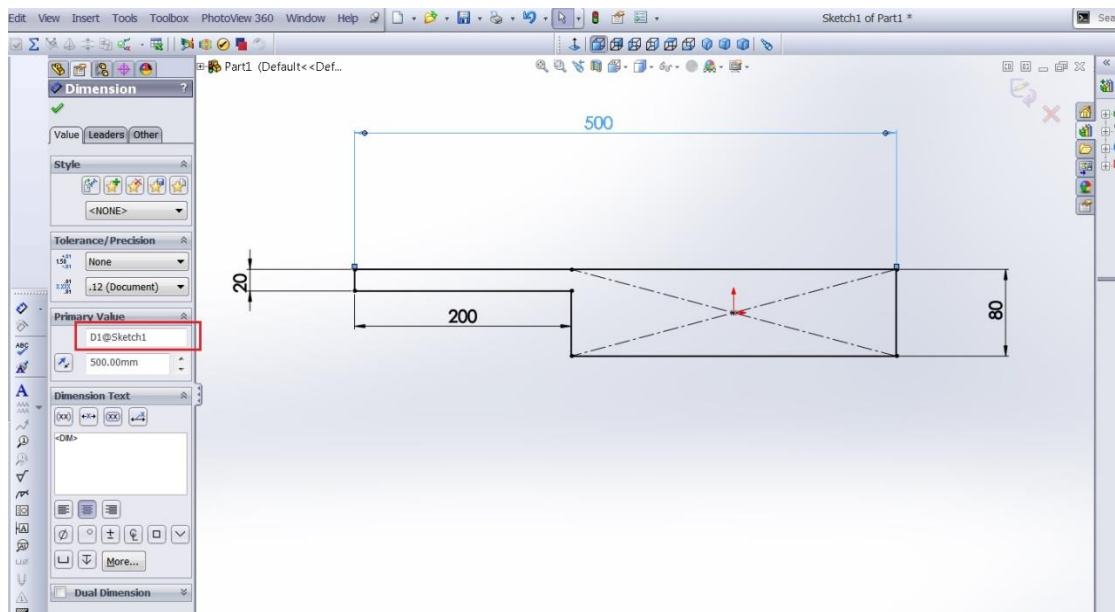
To put in simple words, you have 20 sizes of wrench sets to draw, but the only thing you need to do is just make a single Part file, but in drawing later, you can blow out 20 wrench sizes just in a minutes. Take a look on picture below to understand the workspace in Solidworks™.



Now, we try out this exercise. What we will do in this exercise is to create a simple knife. Let's do this sketch on Front Plane, and follow dimensioning order. This is critical part, why you need to follow my dimensioning order? Because it will reflex data input in design table. Before I go further, Design Tables works better with Microsoft Excel 2008 onwards (not working with Home & Student version). Okay, follow these steps:



Now we have 4 dimensioning, please follow the orders. Here is the reason why:



If we click on dimension 1 (500mm), please check on the left value check box, go to **Primary Value** menu, you will notice in the first value box stated – **D1@Sketch1**-, take note this value name, if you click on the dimension number 2, its value name is –**D2@Sketch1**-. So, value “D1 will increase as many as dimension has.

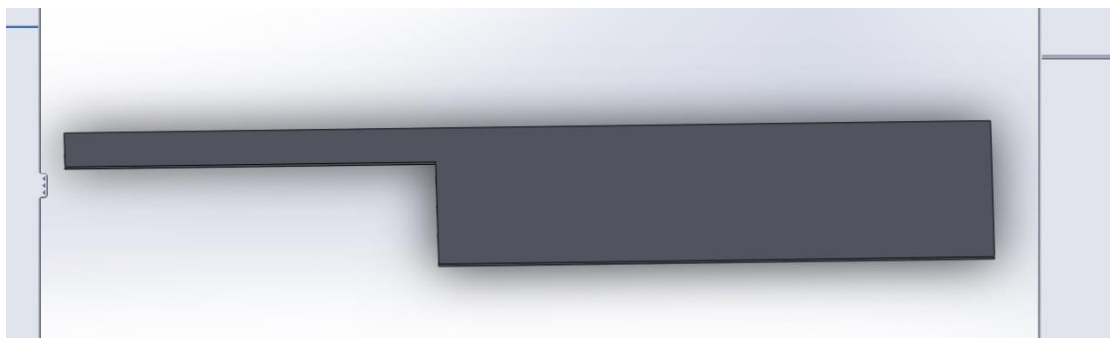
Of course, you can rename this as you like, but for this instruction, please rename this value name accordingly, but symbol @ remain and so "Sketch1" because it reflex to Sketch name at design tree bar. Follow these changes:

Original	Changed
D1@Sketch1	Length@Sketch1
D2@Sketch1	Width@Sketch1
D3@Sketch1	HandleLength@Sketch1
D4@Sketch1	HandleWidth@Sketch1

So the current Sketch1 value is like this:

Dimension Name	Dimension Value (mm)
Length@Sketch1	500
Width@Sketch1	80
HandleLength@Sketch1	200
HandleWidth@Sketch1	20

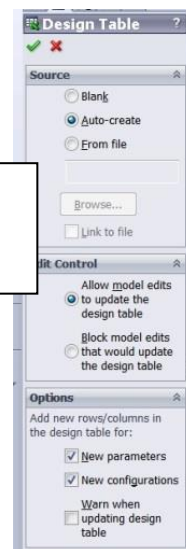
Now, after that, please Extrude Boss 3mm thickness

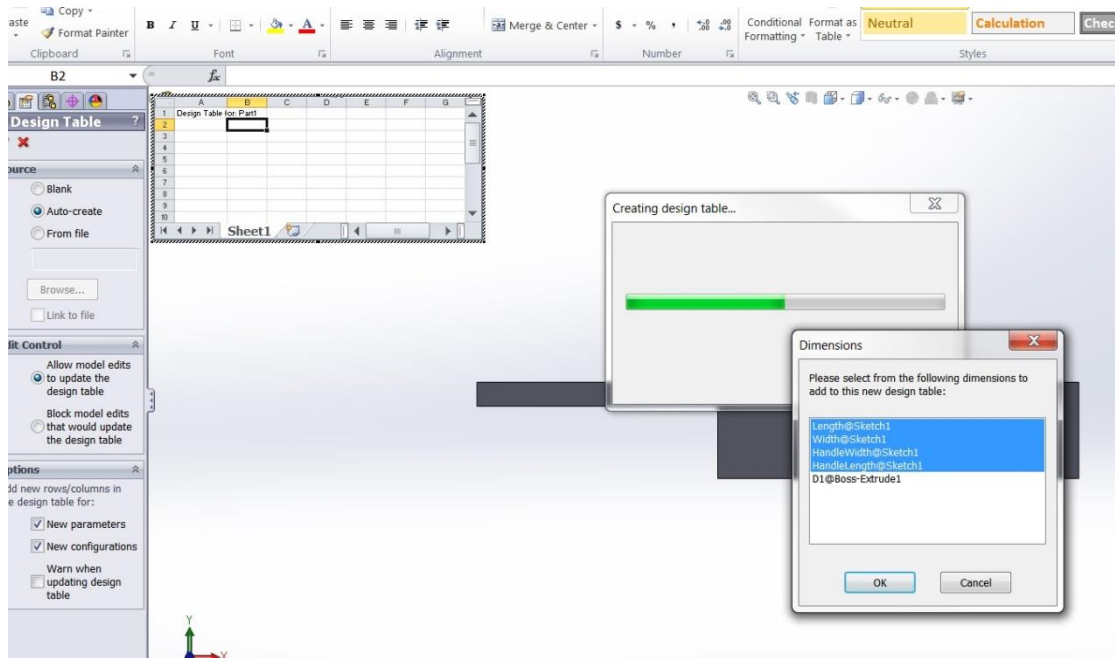


Next, go to Insert> Table> Design Table or pick this icon

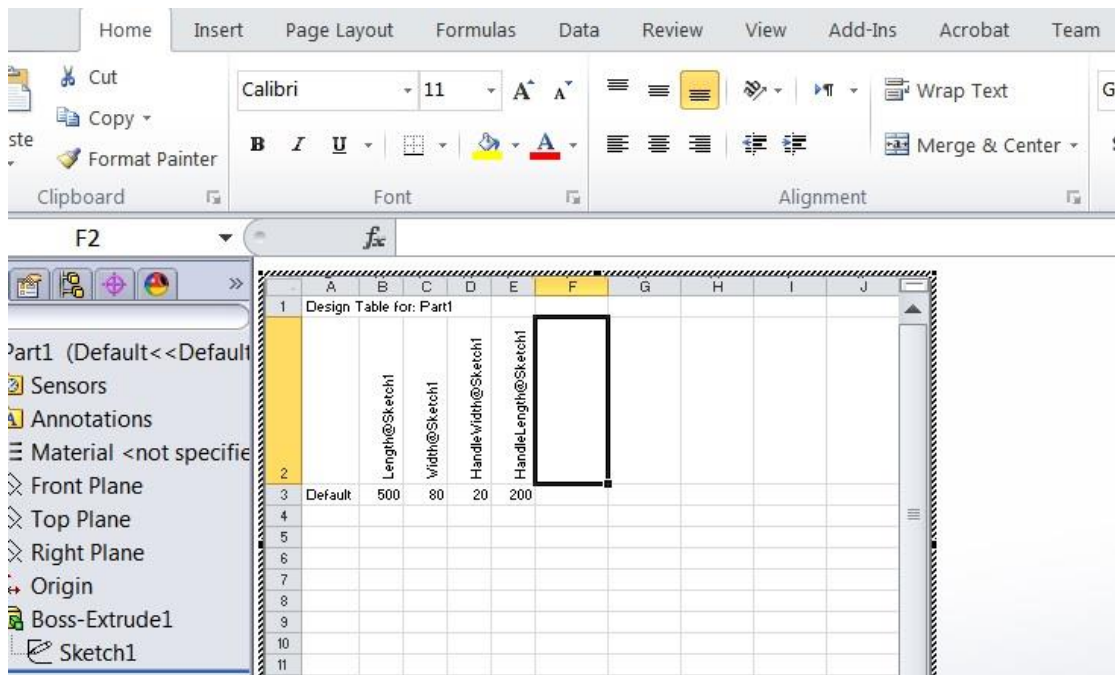


This is default setting, leave it like this and click OK.





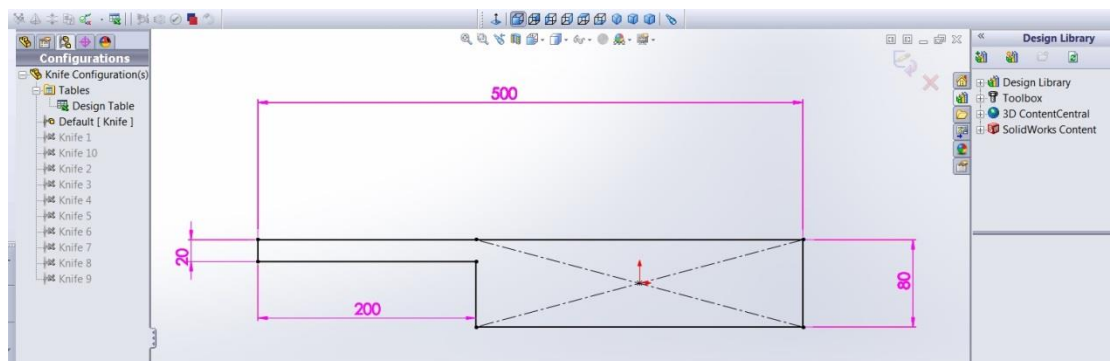
This Dimension Pop Up Window will appear, and select Value Name required (Highlighted in blue is value has been selected) and click OK



As you noticed, Part layout automatically changed to Excel layout, and at top right corner, and Excel Sheet appear, and you will noticed this data arrangement. Start at B2-E2 and onwards is Dimension Value Name. Start from A3 and downwards is Configuration name. "Default" name you can change it as you like.

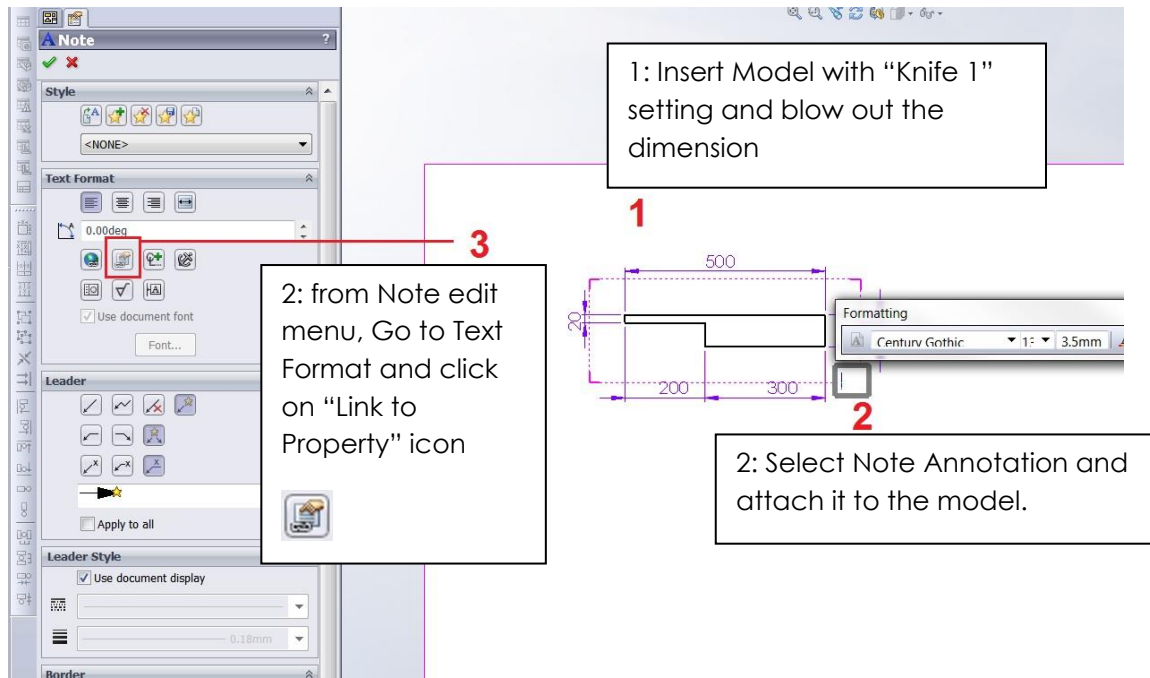
	A	B	C	D	E	F	G	H	I	J
1	Design Table for: Part1									
2		Length@Sketch1	width@Sketch1	HandleWidth@Sketch1	HandleLength@Sketch1					
3	Knife 1	500	80	20	200					
4	Knife 2	520	100	40	220					
5	Knife 3	540	120	60	240					
6	Knife 4	560	140	80	260					
7	Knife 5	580	160	100	280					
8	Knife 6	600	180	120	300					
9	Knife 7	620	200	140	320					
10	Knife 8	640	220	160	340					
11	Knife 9	660	240	180	360					
12	Knife 10	680	260	200	380					
13										
14										

Please fill in the table like above picture; we are making 10 sizes of Knives. I hope you are well in Excel interface. Make the increment value 20mm each, please use SUM factor for fast and easy edit, because if you do that, next time when changes required, you only changing on top value only, the rest will change accordingly. After that save it (Ctrl+S) or just click anywhere in modeling space. A pop up window will appear and just click OK.

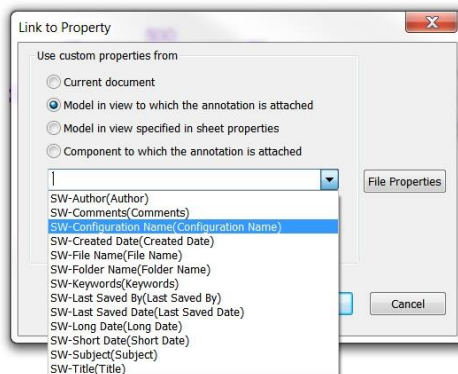


Go to design tree and click edit sketch, you will notice Dimension in Pink Color, that's mean you dimension already automated in Design Table. Now check the Configuration Tree, you will noticed under Part Icon, top level is Excel Design Table, below that there is 11 configuration, 10 with Design Table and 1 by "Default". Click and check every Configuration and the model will change accordingly. CONGRAGULATIONS!!!. Save it as Knife.SLDRPT.

Now we go to Drawing interface.



In Drawing Layout, follow instruction above.

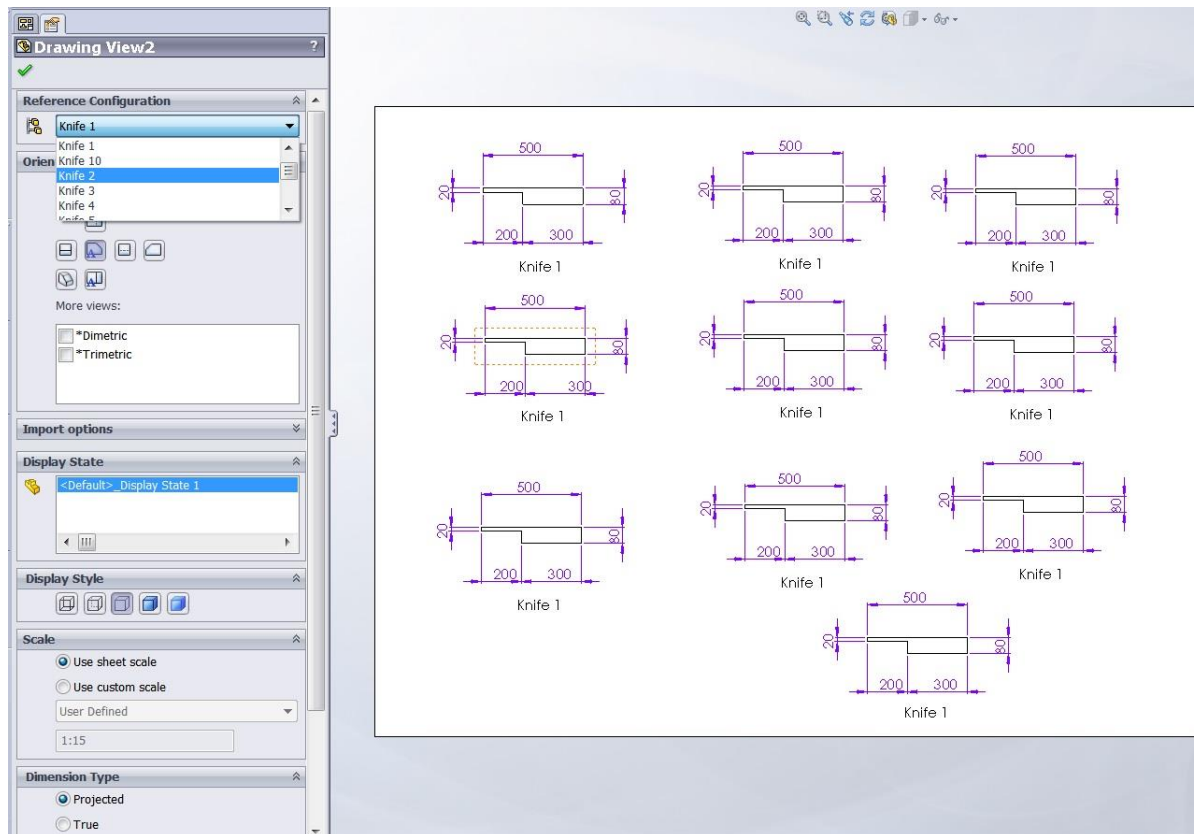


<<In Link To Property pop up window, please select the second option and from drop down menu, please select>

SW-Configuration(Configuration Name)

Then click OK

After that, you will notice the note automatically insert the Configuration Name, that means you don't need to retype again in the next stage.



In Drawing View, click on the Model View> copy (Ctrl+C) and paste it to make 10 model view from same Part model. And each model view, reconfigure the Configuration Name by click drop down menu on the model view tool bar at left side, in the drop down menu choose Configuration name. Do it same to the rest Model View, and below you will notice the sizes change accordingly and also annotation name also changed automatically.

CONGRAGULATIONS!!!

