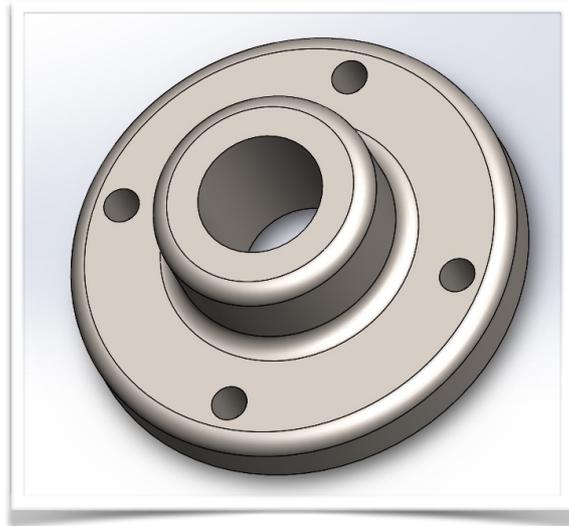# EQUATIONS

| by Jimmy Johansson |

Equations & Global variables

Equations and global variables are used to control various features of a part. With a global variable, several dimensions can be setup to be ruled by a single variable and the need to manually edit multiple dimensions disappear.

How and what?

A global variable is not bound to a single sketch or feature. Dimensions in different sketches and features can, independent of each other, be linked to the same global variable. It makes a global variable very useful for controlling features of a part with minimal effort.

An equation in SolidWorks is simply a mathematical operation to receive a value. It is created either by entering a value, a dimension, or a global variable followed by a mathematical operator, e.g. "/" or "+". A dimension can be set to "75/3" and SolidWorks will then automatically calculate the value.

Objective

The usage of equations can speed up the work process by controlling several variables with global variables and "if-statements". Using these, it's possible to modify the entire model by simply changing the value of a single variable.

The Setup

I want to create a flange with four configurations where some dimensions are governed by the base dimension of the flange. I also don't want to manually edit these dimensions for every configuration so I turn to the equations feature within SolidWorks.

To start with, I create a global variable, name it "A" and give it a value of 150 mm. I then create a profile of the flange which is to be made solid with a revolve feature. When I add the dimensions, I don't bother to change the value, I only change the name of the dimension. This make it easy for me to find the dimension in the equation table where I will create an equation based on the global variable. It is

possible to set up the equation directly in the value field, but in this case I choose not to.

Take notice of the red sum-symbols in front of the numbers. It indicates that the value is the result of an equation.
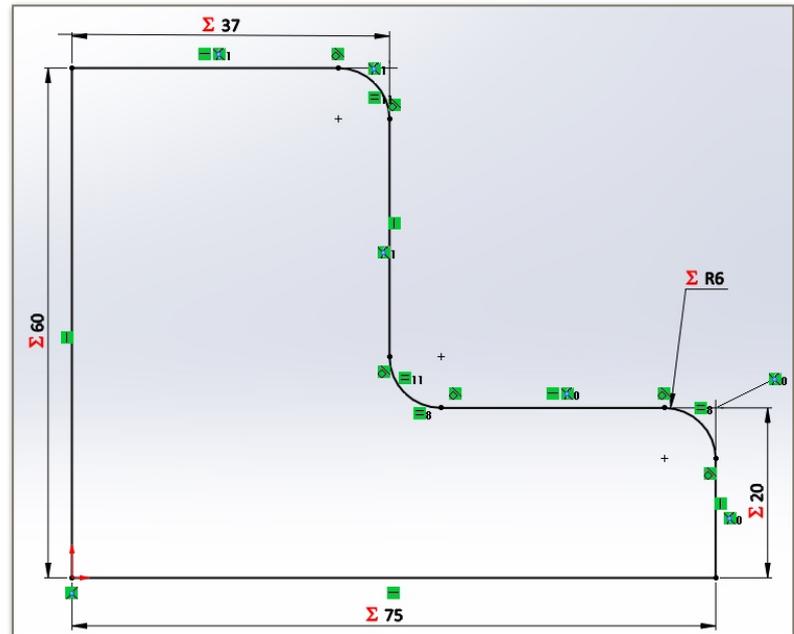
The true strength!

The true strength of global variables and equations emerge when these two features are combined. By creating a global variable with a base value from which all other dimensions are derived, one need only to change a single value and all dimensions based on that global variable will follow.

A step further is to create configurations in which the global variable is set to different values. When the value of a global variable is set, one can specify which configurations the variable should affect.

Once all dimensions have been given a unique name for identification, I exit the sketch and use the revolve feature to create a solid model. I then create a new sketch on top of the model and cut out a hole through the flange. Lastly I create a smaller hole close to the edge of the flange and use a circular pattern to obtain four instances of the small hole.

Now I'm ready to set up my equations! Using the global variable, I can create a link between the value of the dimensions in the profile sketch of the flange and the global variable by using it as the base in an equation. The equation will then automatically recalculate the output value when the global variable change, hence the dimension will be linked to the global variable.

Name	Value / Equation	Evaluates to	Comments
Global Variables			
"A"	= 150	150	Base Dimension
"B"	= 12	12	Attach Hole, Small Dimension
"C"	= 17	17	Attach Hole, Large Dimension
Add global variable			
Features			
Add feature suppression			
Dimensions			
Thickness of Base@Sketch Flange Profile	= int ("A" / 7.5)	20mm	
Base Dimension@Sketch Flange Profile	= "A" / 2	75mm	
Flange Dimension@Sketch Flange Profile	= int ("A" / 4)	37mm	
Height@Sketch Flange Profile	= "A" / 2.5	60mm	
Radius@Sketch Flange Profile	= int ("A" / 25)	6mm	
Revolve@Base Flange	360deg	360deg	
Shaft Hole@Sketch Shaft Hole	= int ("A" / 3.5)	42mm	
Guide for Attaching Holes@Sketch Attach Holes	= int ("A" - 30)	120mm	
Attaching Hole@Sketch Attach Holes	= IIF ("A" > = 160 , "C" , "B")	12mm	
Angle@Sketch Attach Holes	90deg	90deg	
Instances@Sketch Attach Holes	4	4	

By giving the dimensions unique names it's easy to find them in the equations table.

Extra features

Within the equation tool, it's also possible to enter some trigonometrical functions as well as the constant pi, the exponential function, square root, and more.

I create a global variable with the name "A" and give it a value of 150. This is my base dimension to which all other dimensions will be linked. Under the "Dimensions" tab it's possible to see all dimensions inserted in my different sketches as well as the dimensions automatically created in the features by SolidWorks. Since I've given the dimensions unique names it's easy to see in which sketch or feature contain which dimension and what they affect. To link the value of the different dimensions, I start by typing ="A" in the Value/Equation column and then the mathematical operator followed by a numerical value that gives the desired result. I do this for all dimensions.

When I'm done, I notice that some of the equations results in a decimal value. To avoid this and only obtain integers, I type 'int' ahead of the equation and add a parenthesis before and after the equation. This will round off the 'original' result of the equation to the closest integer.


Since I will create four configurations of the part, all with a different base dimension, I also want to increase the dimensions of the attaching holes when the base dimension exceed a specific value. For SolidWorks to automatically update this, I choose to use a 'if-statement'. What the if-statement does is that it evaluate if something is true or not and executes a predefined action.

For example, 'if (A > B, C, D)'. What this if-statement says is: if A is greater than B, do C, otherwise do D.
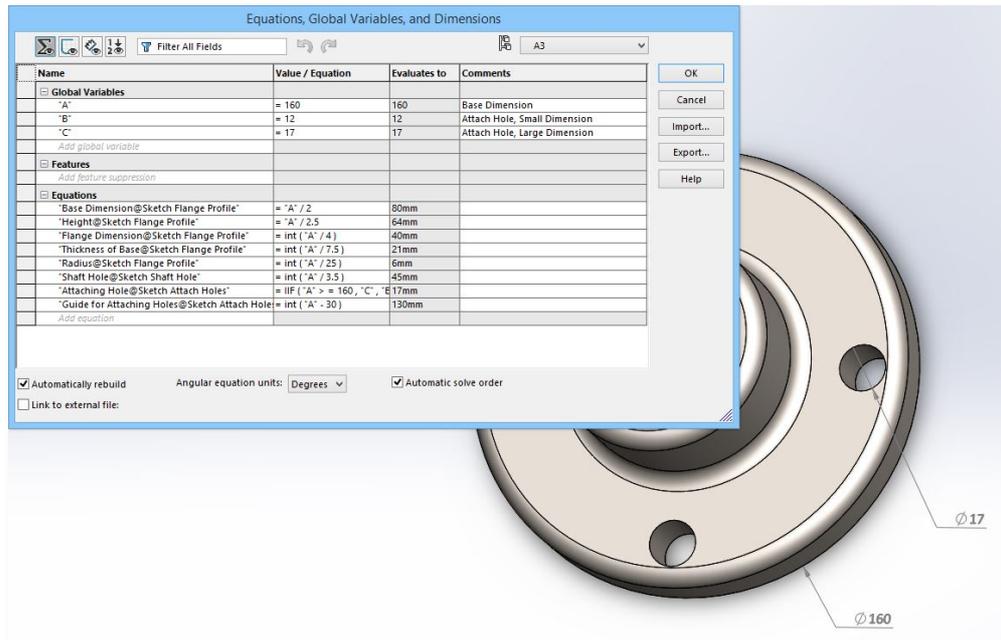
In this case, I want the diameter of the attachment holes to increase when the base dimension is equal to *or* greater than 160. Currently, they have a dimension of 12, but if the base dimension increases to 160 or more, I want the attachment holes to increase to 17 mm in diameter. I therefore must create two more global variables, "B" and "C", to which the dimension dictating the diameter of the attaching holes can be linked depending on the result of the if-statement. Said and done, two variables named "B" and "C" are created with the value of 12 and 17 respectively. I can now use them in my if-statement so I enter 'if ("A" >=160, "C", "B")' in the Value/Equation column defining the diameter of the attaching holes.

Once everything is set up, I can create three more configurations from the one I've made and the only thing I need to change for each configuration is the value of the base dimension.
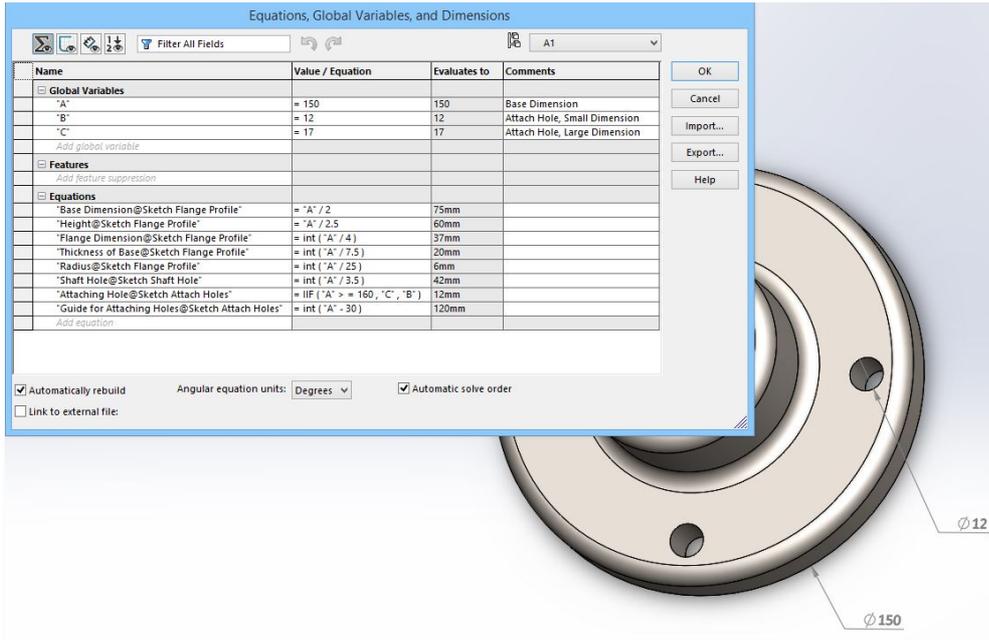
As can be seen to the top right, configuration A1 is selected. It has a base diameter of 150, hence the diameter of the attachment holes will be equal to "B", which in turn equalizes 12.

When the configurations are created, I switch to each and one of them and change the base dimension. For the third and fourth configuration (A3 and A4), I set its value to 160 and 170 respectively. This means that for A3 and A4 the base dimension exceed the base value in the if-statement (160), hence will become true. The outcome will be that the diameter of the attachment holes automatically increase when any of these two configurations are selected.

Now configuration A3 is selected and this configuration has a base diameter of 160. In return, this means that the if-statement is true and "C" is the output value which increase the dimensions of the attachment holes to 17.