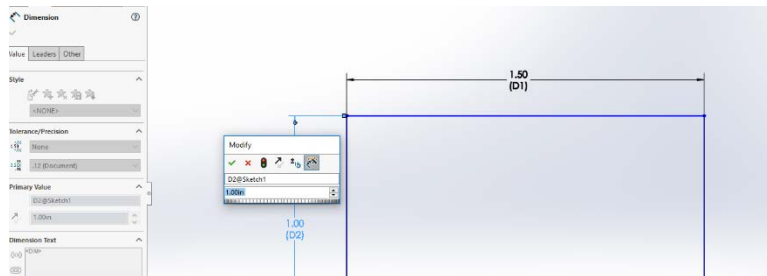


SolidWorks Intro Part 4

Dave Touretzky and Susan Finger

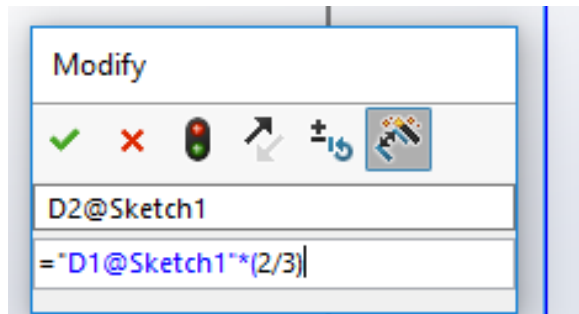
1. Entering equations

1. Start a new part on the front plane.
2. Draw a rectangle that is 1.5 inches wide and an arbitrary height.
3. Select **View** → **Hide/Show** → **Dimension Names**.
4. With Smart Dimensions on, dimension the height.
5. Double click on the Smart Dimension to get the Modify box:



6. In the modify box, **enter an = sign** and then click on the smart dimension of the width. Set the height to 2/3 of the width by entering the equation:

$$= [\text{click on dimension D1}] * (2/3)$$

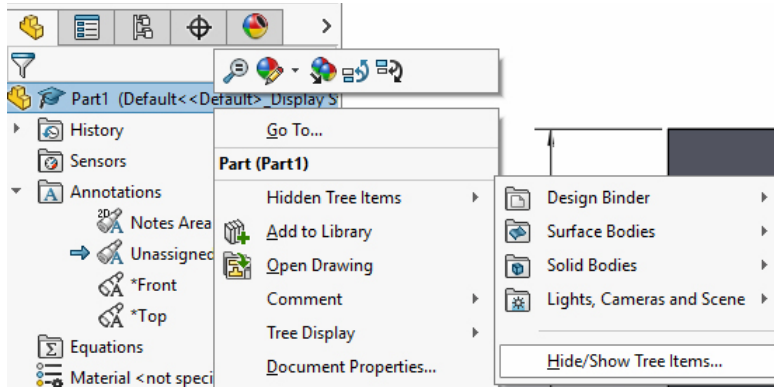


7. Click the green check mark.
8. Change the width to 2 inches and watch the height change.

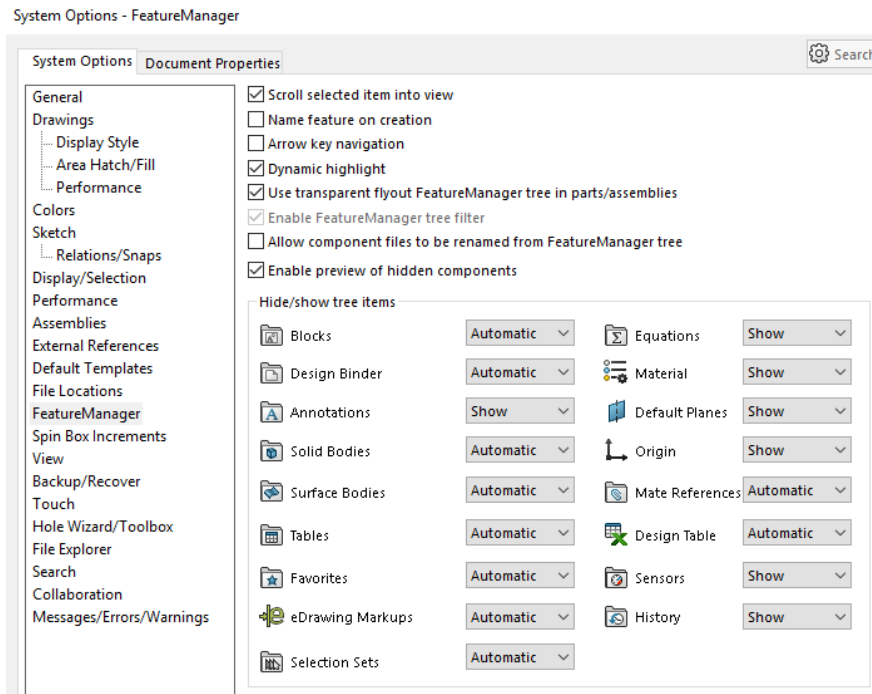
2. Creating global variables

In SolidWorks, you can also create variables to use in equations. Suppose you were making a series of parts that all had a width of 4.576.

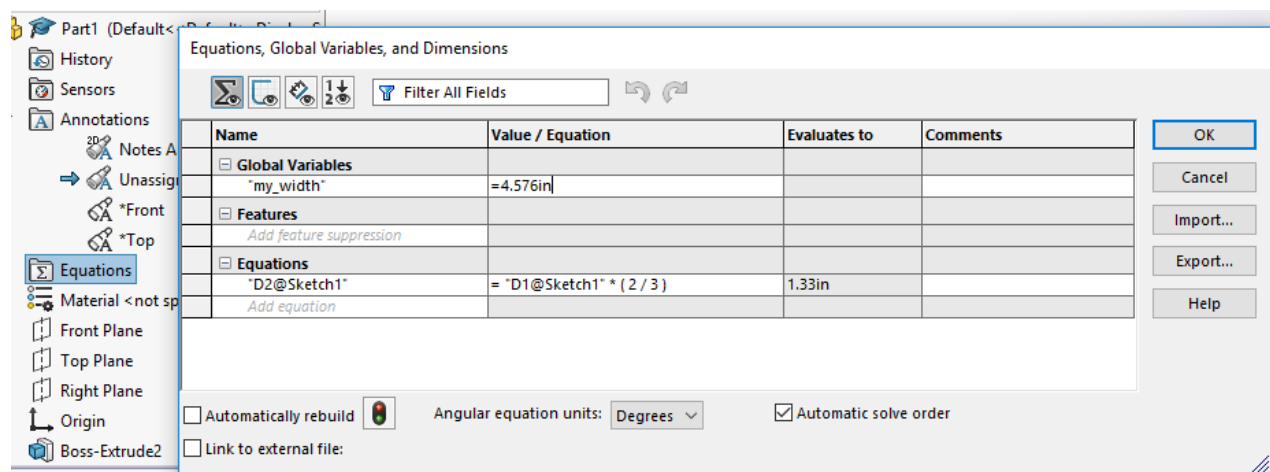
In the Feature Manager tree, right click on the part name. Select **Hidden Tree Items** → **Hide/Show Tree Items**



In the popup window, find the button at the top of the right-hand column to Show equations. While you have this window open, look at the other features of parts that you can show or hide.



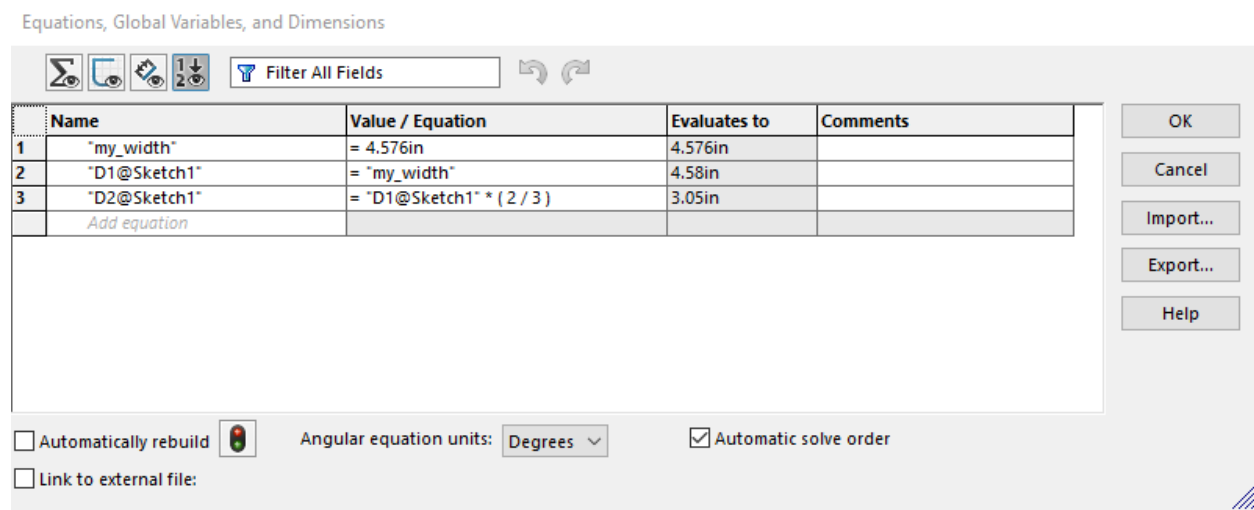
Right click on the Equations feature and select **Manage Equations**. Click in the text box under Global Variables and give your variable a name. In the screen shot below, I called my variable my_width and set its value to 4.576. Click **OK** and your variable will be defined.



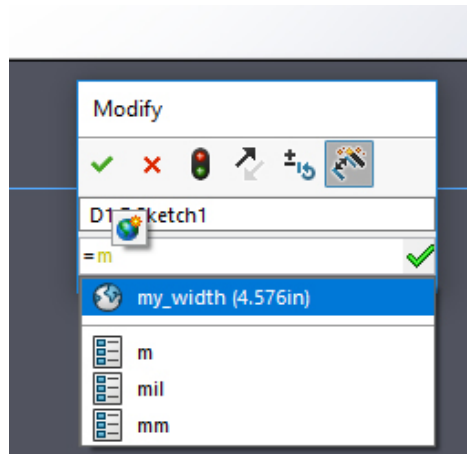
Click on the four views in the equation manager:

- Equation view
- Sketch equation view
- Dimension view
- Ordered view

Here's the ordered view, which shows all the variables that have been defined by you and by SolidWorks:



Go back to your sketch and double click on the Smart Dimension of the width. In the modify box, change the width to equal your global variable name. Enter the equal sign, =, to get started. In the screen shot below, you can see that as soon as I started to type =m, SolidWorks prompted with the name of my global variable.



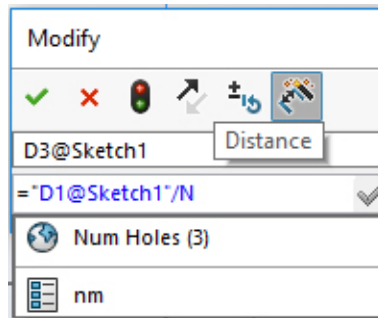
This isn't really what we want for the rest of the tutorial, so click the **red x** so that the width isn't reset.

3. Creating linear patterns

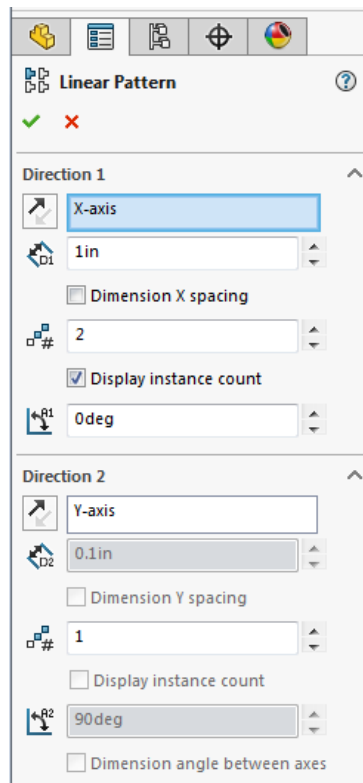
Suppose you want circles evenly spaced across your part.

1. Create a hole with a diameter of 0.1 inches in the upper left-hand corner of your part.
2. Create a horizontal centerline from the center of the circle to the left edge.
3. Using Smart Dimensions, dimension the center line and set its length to:

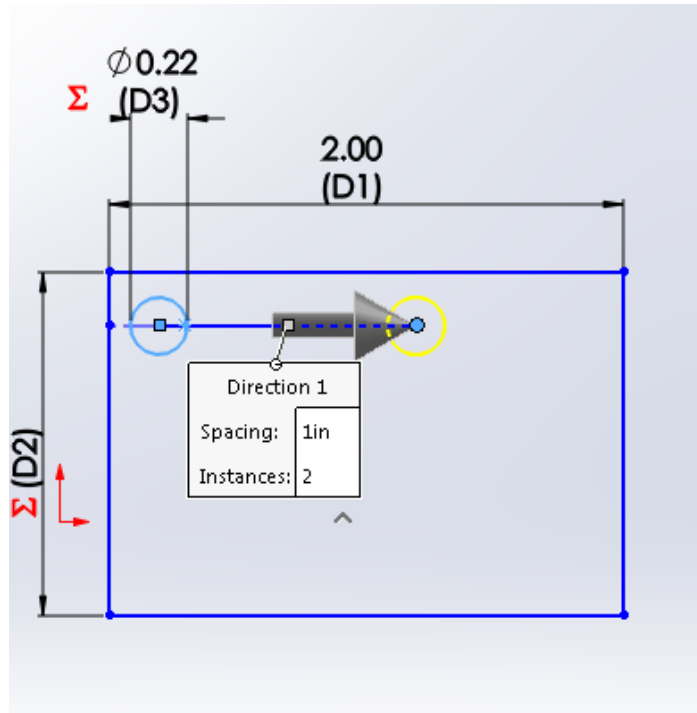
$$= [\text{dimension D1}] / [\text{Global variable "Num Holes"}] / 3$$



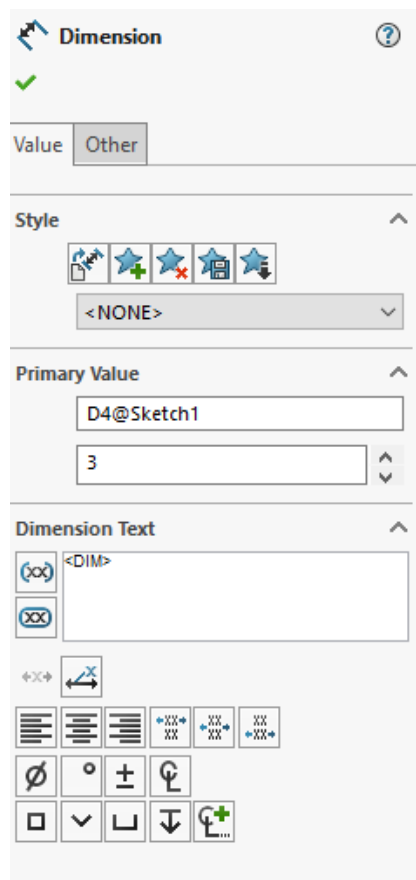
4. Select the circle and then select the **Linear Sketch Pattern** from the Sketch tool bar. We will create a linear pattern feature to replicate the hole.
5. We're going to have the holes go across the part, so Direction 1 is the X-axis, which is the default. Set the number of instances to 2 and the spacing to 1 inch.



6. After you have entered the values, but before you click the green checkmark, your workspace should look like:



7. Click OK.
8. To modify the linear array, right on one of the circles and select Edit Linear Pattern from the context sensitive menu.



9. When you click OK, the circles will go off the part, but can fix that.
10. Right click on any of the circles. From the context sensitive menu, select **Edit Linear Pattern**. You will now have a left hand panel that lets you change all of the parameters of the array.

4. Hole wizard

SolidWorks has a useful command called the Hole Wizard. Select **Insert** → **Features** → **Hole Wizard**. The left-hand panel will give you the options for many different sizes of screws, with different heads, and different ways of making the holes. For now, accept all the defaults except for the screw size.

11. Change the screw **Size** to #4
12. Click on the Position tab.
13. Click near the left edge of the part to create a screw hole.
14. Right click on Equations in the Feature Manager Tree and select Manage Equations.
15. Define another global variable "Num Holes" and set it to 3.

4.1.1 Making Basic 3D Shapes

- **Method 1:** Extruded base on an existing face of a part.
- **Method 2:** Revolved base of a cross section. Sculpture example.
- **Many additional techniques:** loft, shell, bend, etc.
- Save the 3D part or assembly as an STL file, then import into your next tool.

