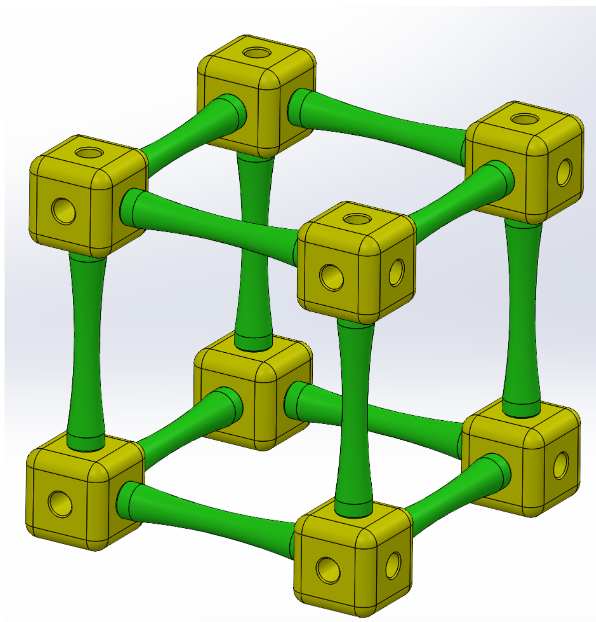
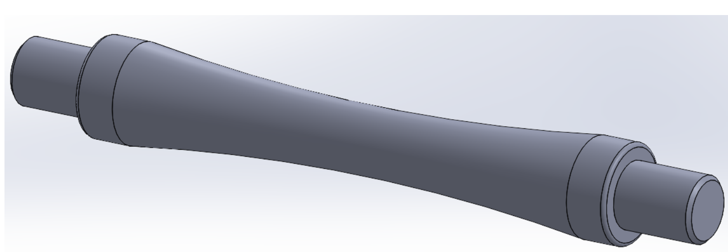
Solidworks Primer/Tutorial

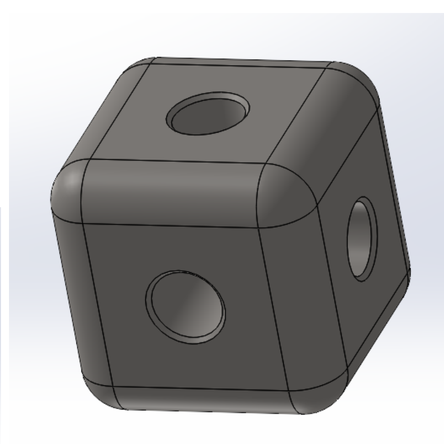
Introduction

This primer will introduce you to the modeling, visualization and design tools in Solidworks.

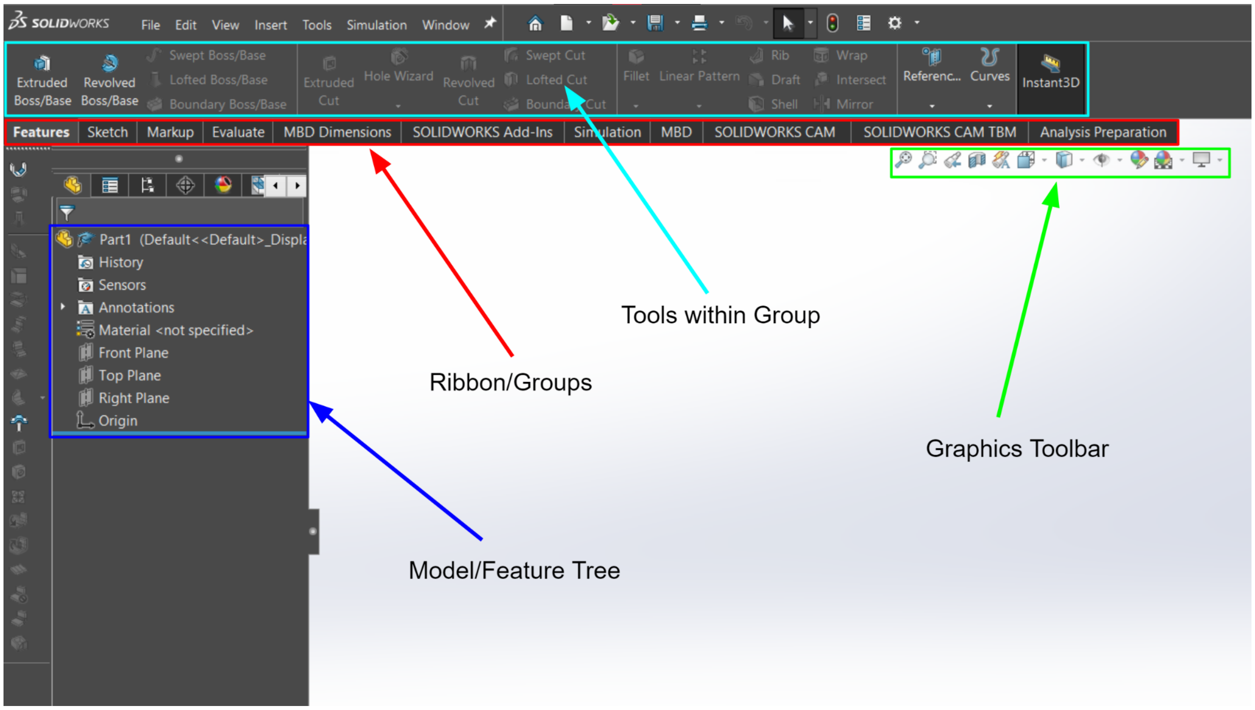


You will be taught how to use Solidworks to model two components for a construction kit - a cube and a strut.
You will then be shown how to put these together in an assembly.





Understanding the Solidworks Interface

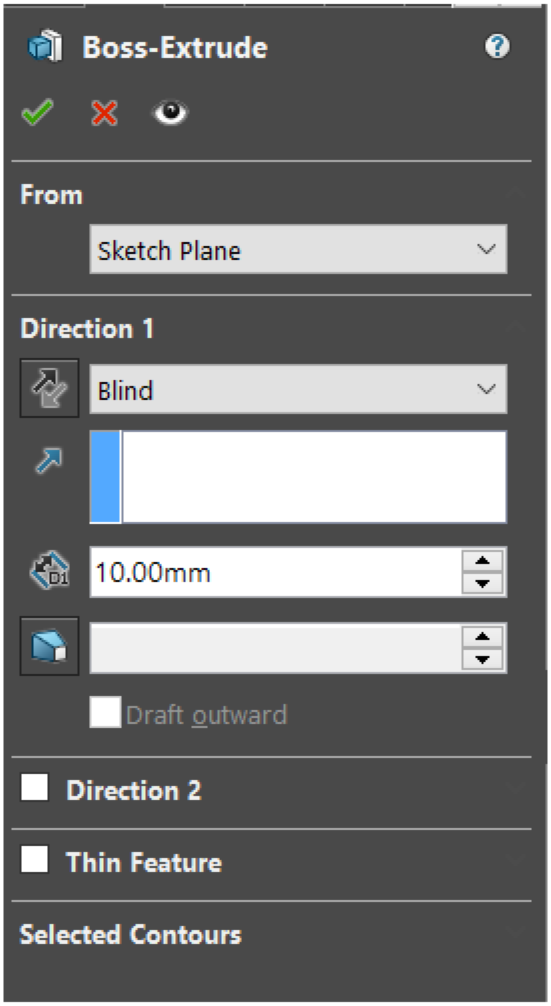


Quick Access Toolbar: For Creating new files, opening existing files, saving, and printing. Also has the “rebuild” (stoplight) icon which is useful in more complex parts and assemblies.

Ribbon Tabs: Groups the tools together so it is easier to find what you are looking for.

Graphics Area: The area that you work in.

Tool Parameters/Dashboard: Appears on the left side of the screen when you are using a tool.


Boss-Extrude
From
Sketch Plane
Direction 1
Blind
10.00mm
Draft outward
Direction 2
Thin Feature
Selected Contours

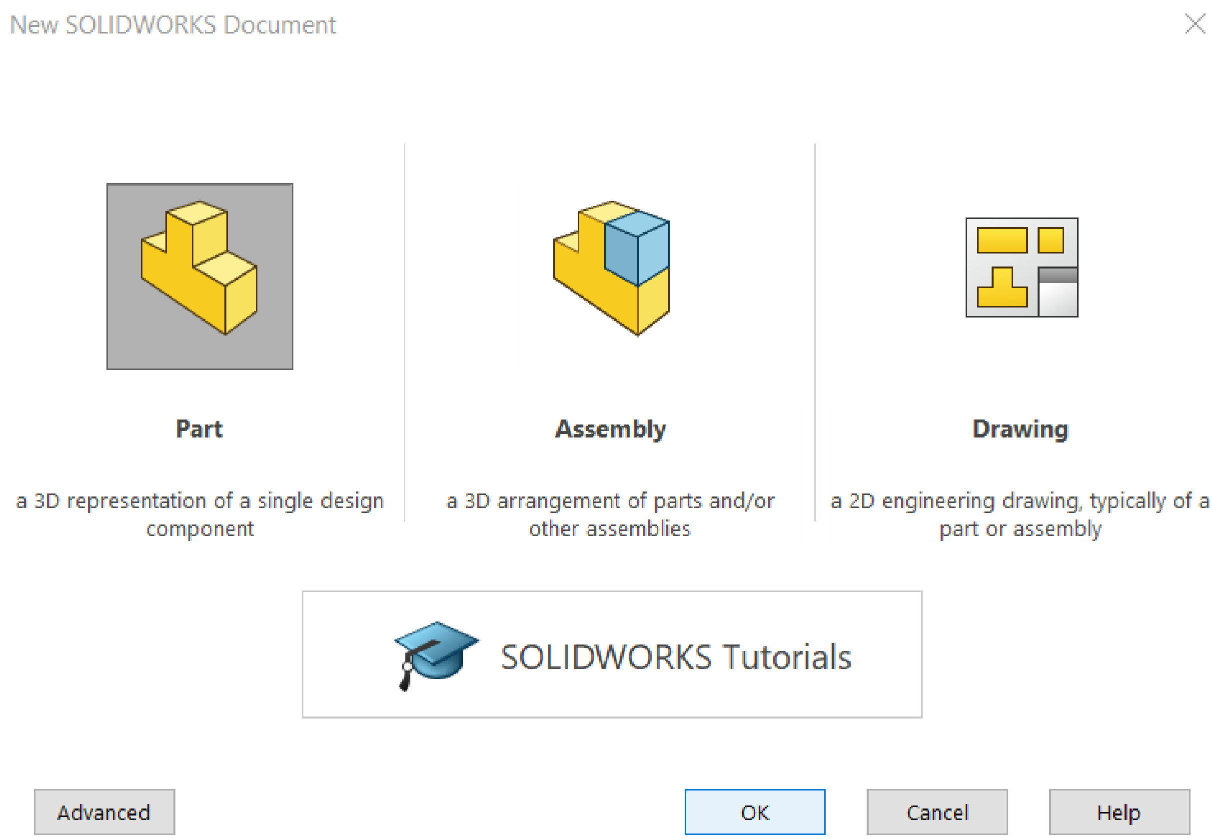
Opening, Saving, and Working from One Folder

Opening and Saving Files: Opening and saving files is just like any other software. There are icons for opening and saving in the title bar of the window. You can also use File->Save, File->Save As, File->Save All, or File->Open.

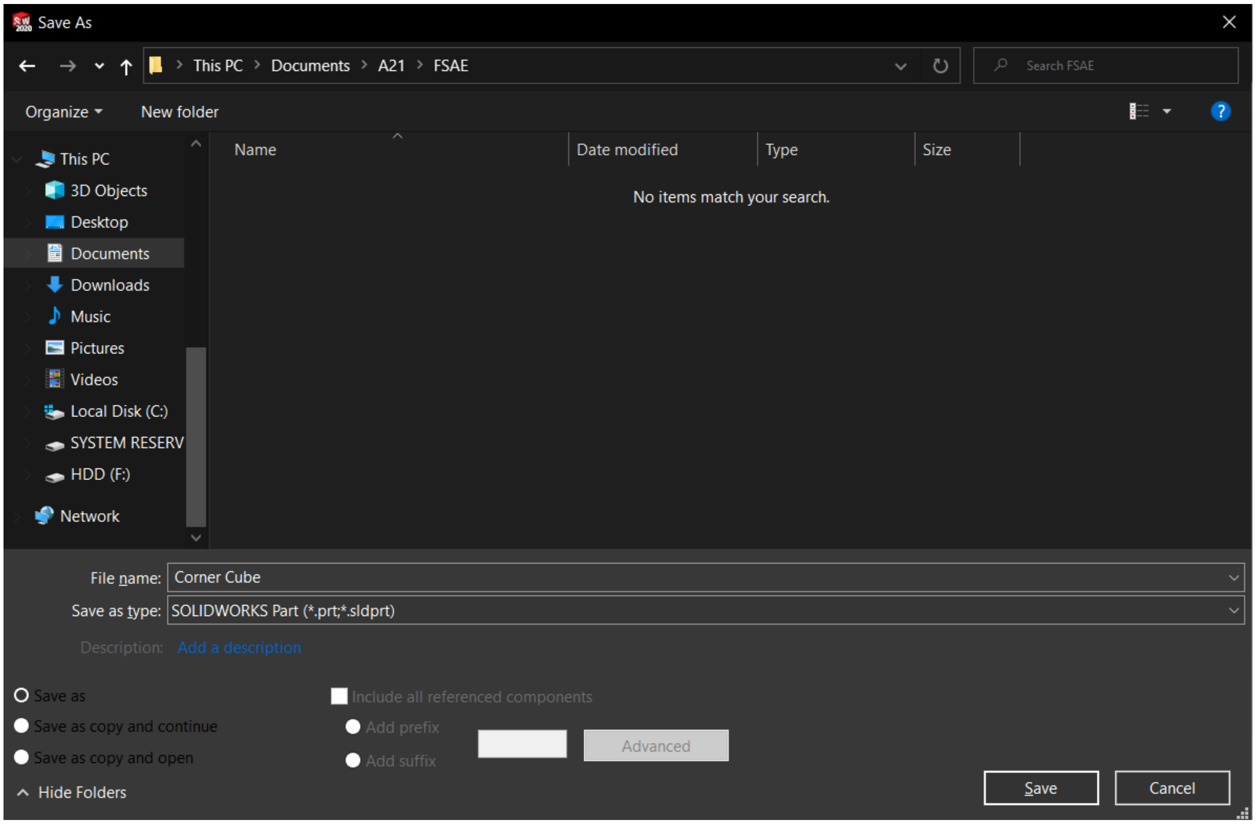
Working from One Folder: It is generally a good practice to keep all the files related to a project within the same main folder. It is up to you to tell solidworks where to save files, and maintain all of the files in the correct directory.

Creating the Corner Cube

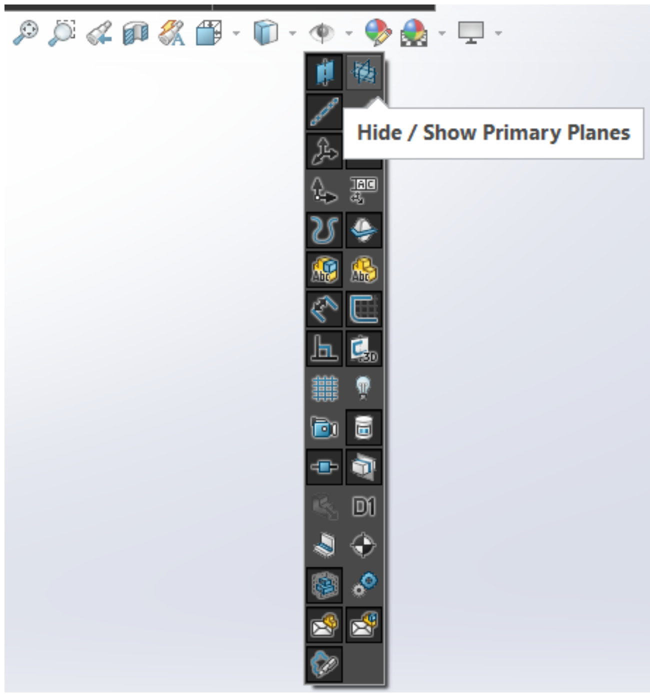
1. **Create a new part**
    a. **Click File -> New, then click part, then press OK.**



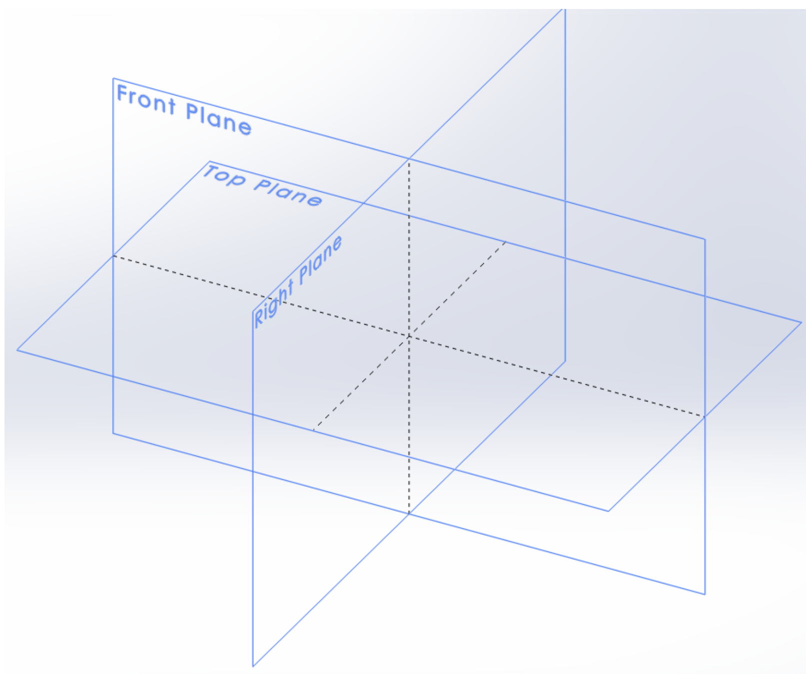
2. **It is a good idea to save and name the part next. You can do this by pressing ctrl+s, File -> Save, or the save icon in the quick access toolbar.**
    a. **Name the file *Corner Cube* and hit save.**



3. **Changing Display of Planes**
    a. **In the graphics toolbar, click the down arrow next to the eye icon. You will see planes and axis among other things that you can show/hide. Click on the "Hide/Show Primary Planes" button to show the primary planes.**



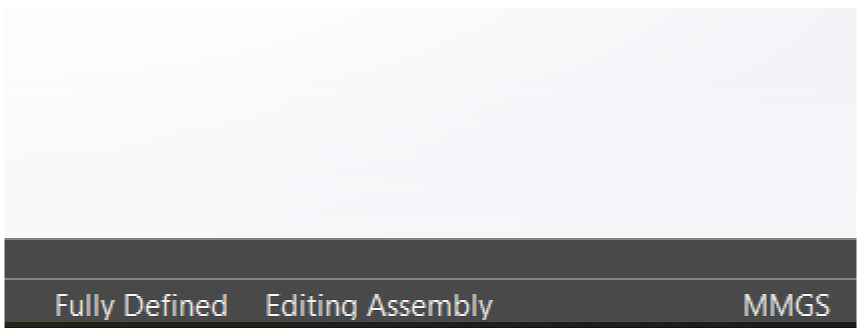
b. You should be met with this:



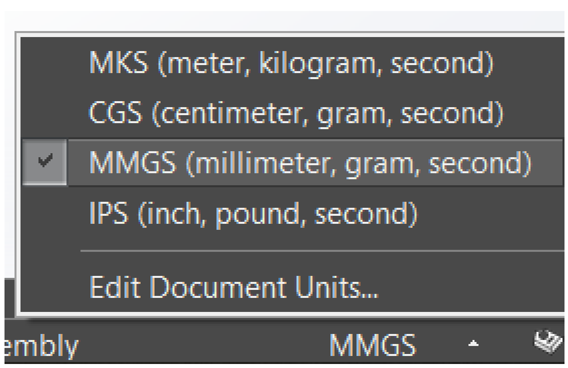
c. You can hover over the other icons in that menu to see what they do.
d. If you want to hide all planes, axis etc, you can click on the eye itself (rather than the dropdown) and

hide all things shown in the dropdown menu at once.

4. **Maneuvering in solidworks**
    a. You always want to be using a mouse
    b. Middle click and hold allows you to orbit
    c. Scroll allows you to zoom in and out
    d. Ctrl+Middle click and hold allows you to pan
5. **Checking Units**
    a. Look in the bottom right corner of the solidworks window and check to see what units are being used



    b. We want to be in MMGS for this tutorial. If you ever want to switch, click on the MMGS or whatever yours says, and choose the correct one.



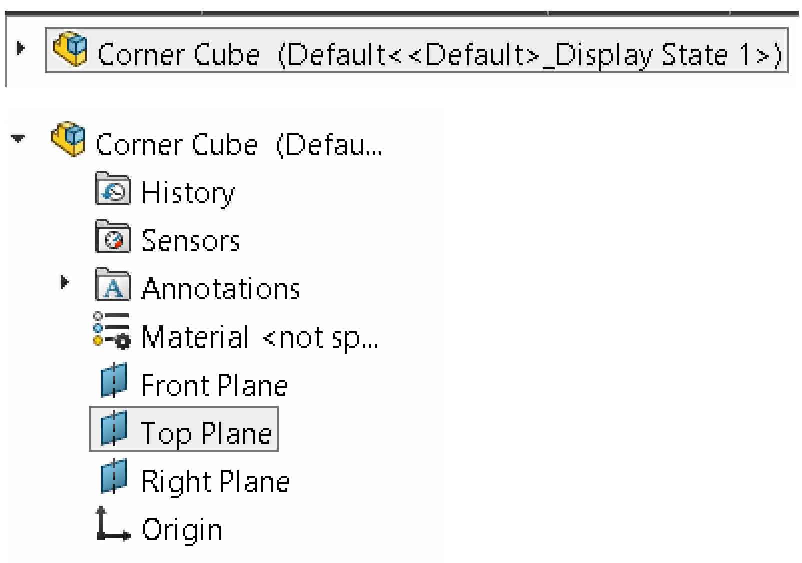
6. **Start an extrude**
    a. Extrudes are the most basic and most common type of feature used in solidworks. An extrude takes a 2 dimensional sketch and “pulls” it into

three dimensions. Think taking a sketch of a circle and pulling it into the third dimension as a cylinder.

b. To start an extrude, go to the "Features" tab, and click on "Extruded Boss/Base"

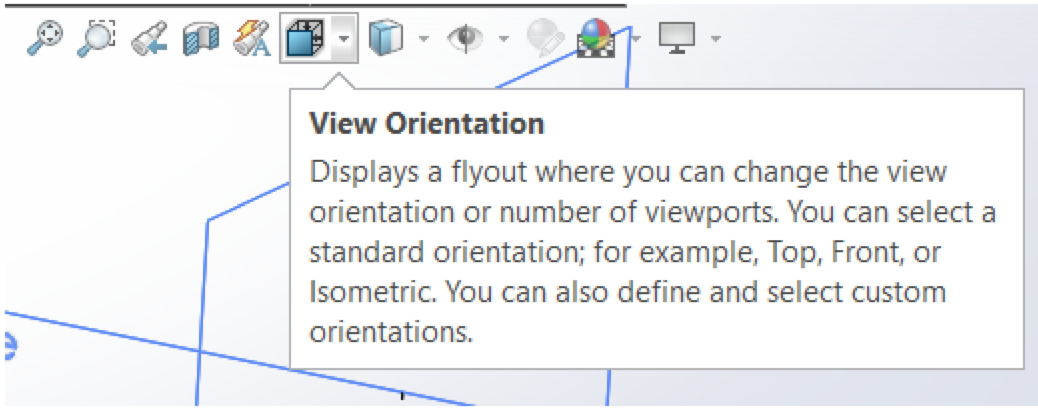
c. Next, Select a plane to sketch on. This can be done either from the planes visible in the Graphics Area, or using the Model Tree dropdown that is visible next to the Tool Parameters/Dashboard. We will use the Model Tree dropdown as this will become more useful as you start making more complex models.

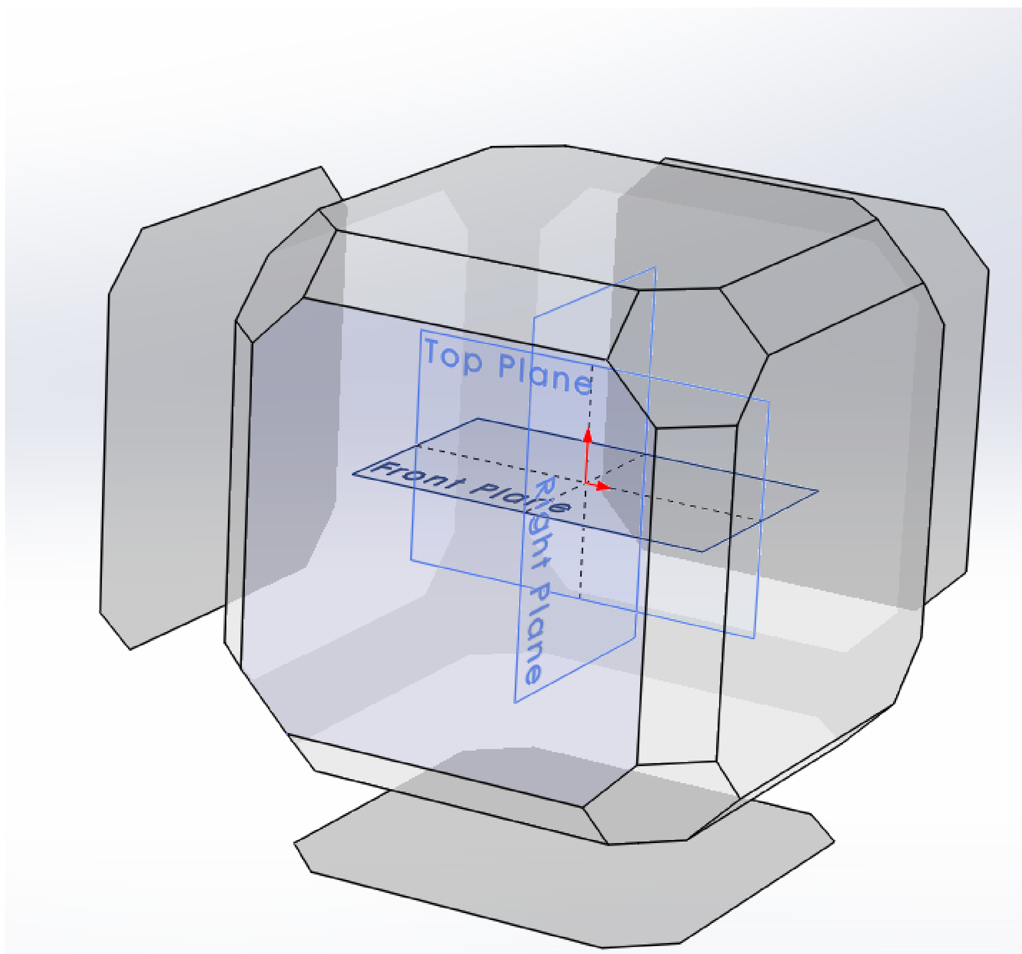
d. Click the dropdown for the model tree, and select the top plane.



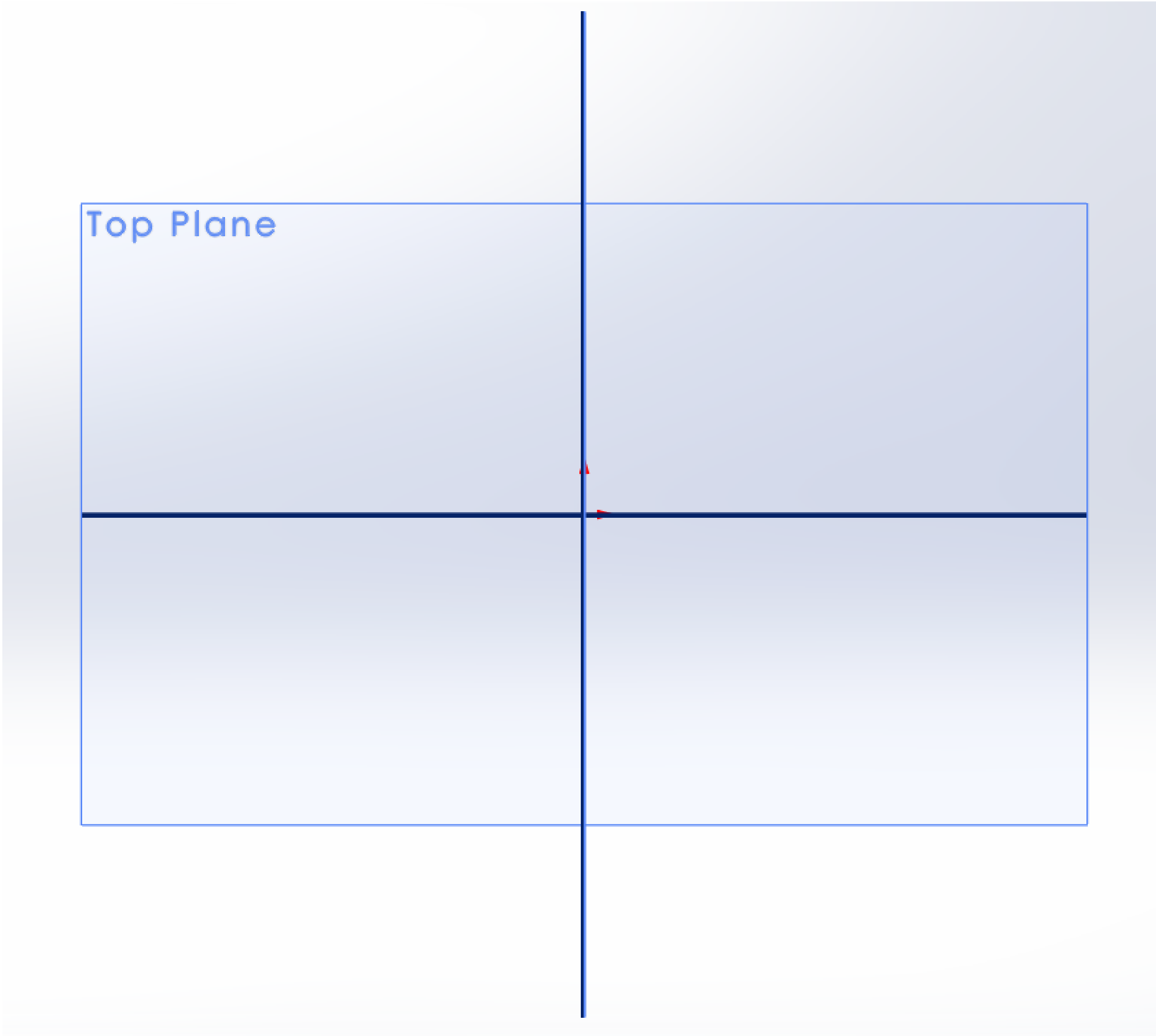
e. If solidworks does not automatically reorient you to so the top plane is parallel with your screen, click on the View Orientation button in the Graphics

Toolbar, then click on the surface shown that is parallel with the top plane.

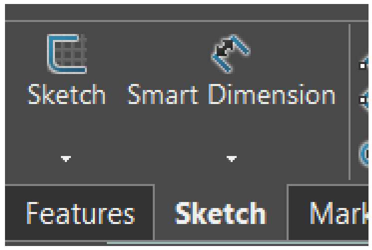




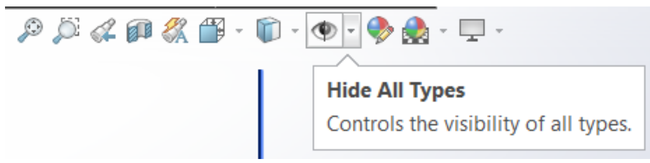
i. It should look like this when you are done:



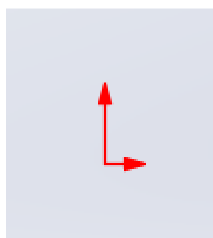
f. You are now in a sketch. A sketch is the basic building block in solidworks, which allows you to sketch in two dimensions before creating a 3D model referencing your 2D sketch. You can create sketches by themselves (without clicking extrude and then a surface) by going to the sketch tab in the Ribbon, clicking sketch, and then selecting the sketch plane as you just did.

i. 

g. The next step is optional. I do not like sketching with the planes displayed, so you can click the eye (Hide all types) button in the graphics toolbar to hide them if you wish.

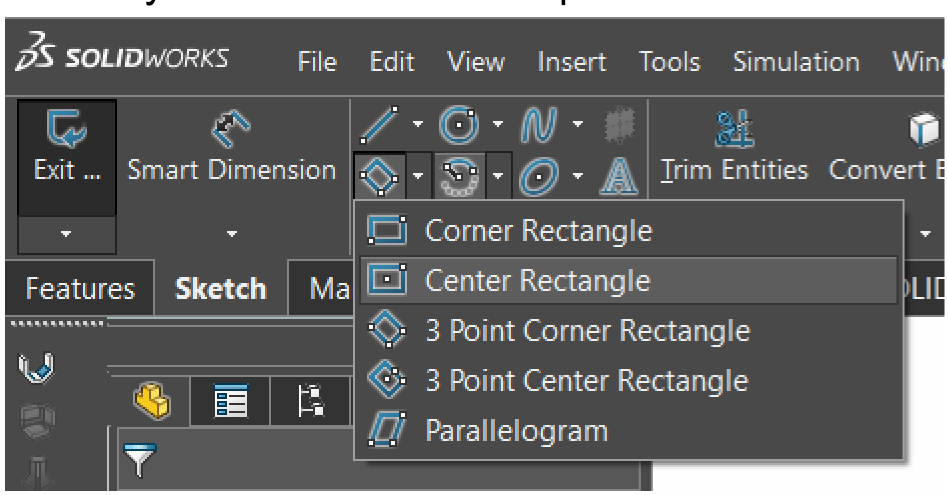
i. 

ii. Note that without the planes, there is still a red origin visible.

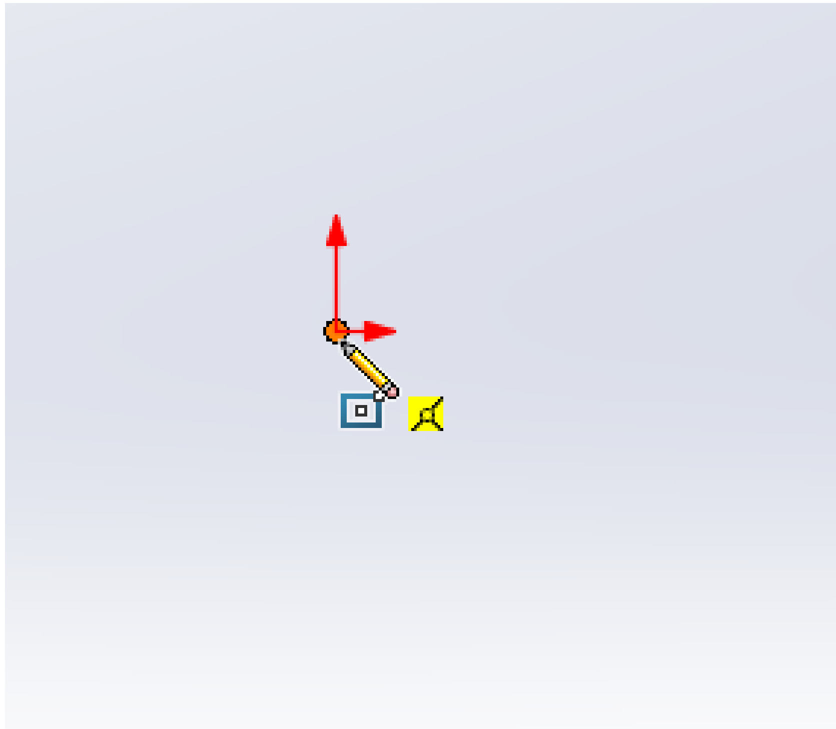


h. Sketch a rectangle centered on the origin.

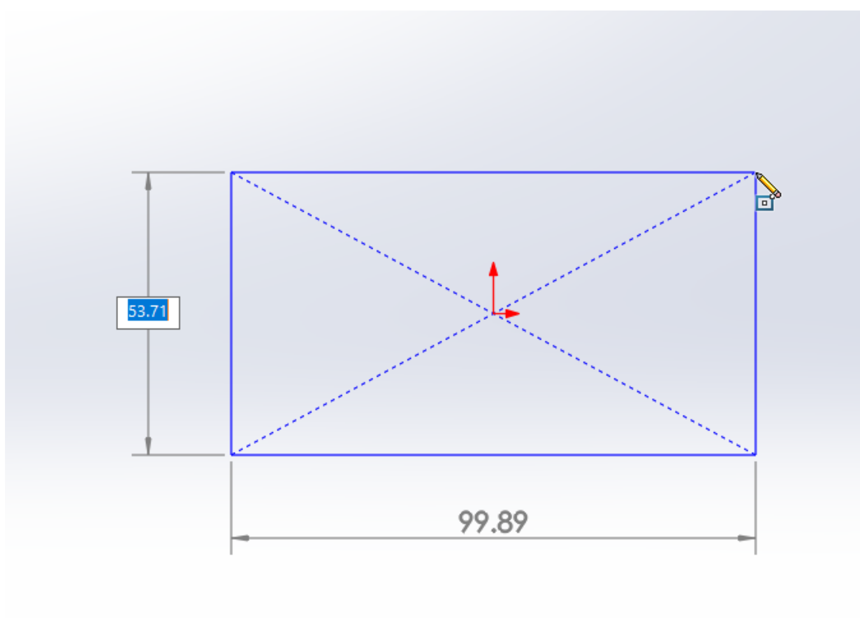
i. In the Toolbar, select the “center rectangle” tool. You may need to click the dropdown to select it.



ii. Then click on the origin.



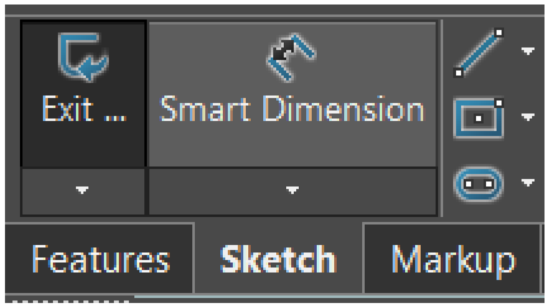
iii. You can now move your mouse cursor, and click again to create the rectangle. Do not worry about the size or shape of it yet.



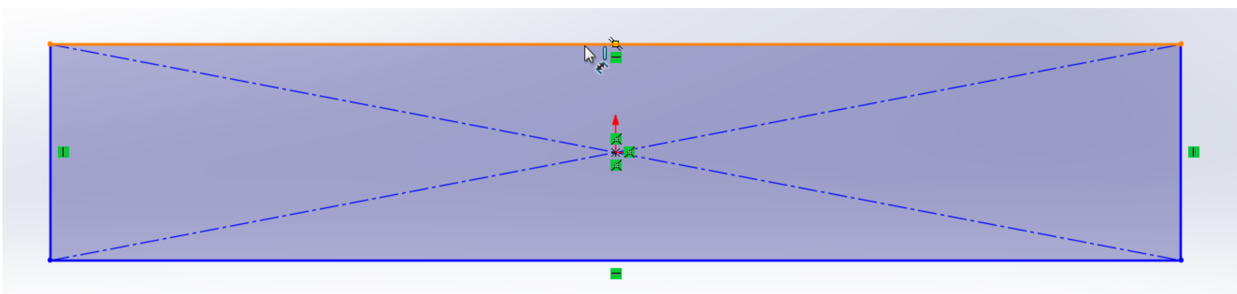
iv. After the rectangle is drawn, you can press the escape key to stop using the rectangle tool. This is a useful keystroke to remember.

i. Next, Dimension the rectangle.

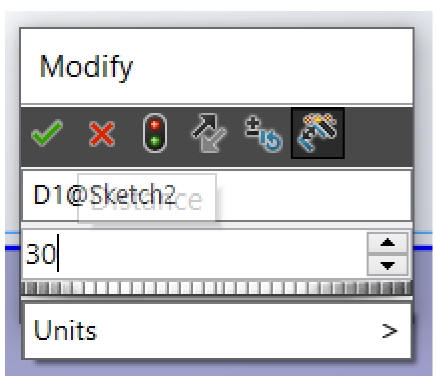
i. Click on the Smart Dimension tool



ii. Click on one of the edges of the rectangle to dimension it.



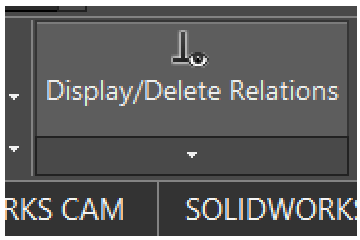
iii. Click again somewhere in space to place the dimension, then type in 30mm to make that side 30mm long, and press enter.



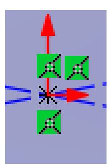
iv. That rectangle side should have resized itself to 30mm.

j. **For the other side, we will add a sketch relation rather than dimension it. Sketch relations are useful to make two entities (lines in this case) parallel, perpendicular, equal length, collinear, tangent, concentric, etc.**

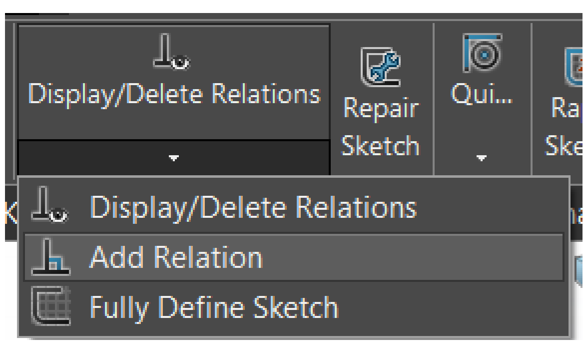
   i. Click the "Display/Delete Relations" Tool in the toolbar.



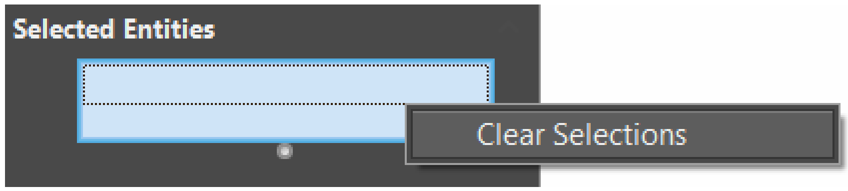
   ii. On the left hand side, all of the current relations will be shown. You can see that the rectangle you just drew has already added some relations (all the little green squares showing in the sketch).

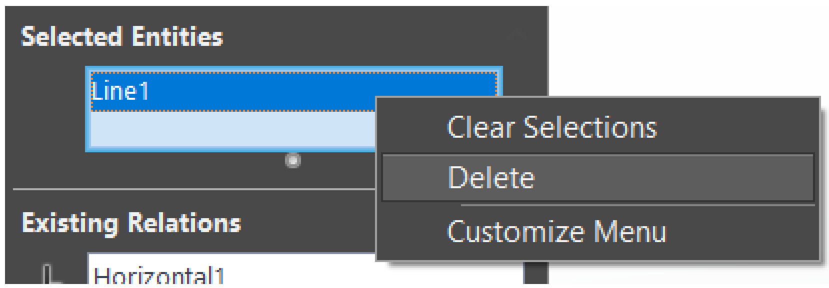


   iii. Press escape to get out of the "Display/Delete Relations tool, and click on the drop down right below it, then click "Add Relation."

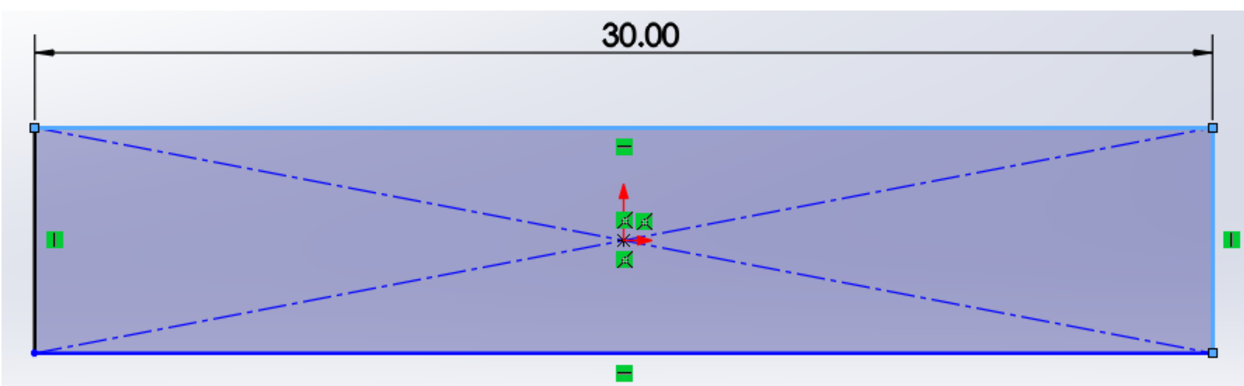


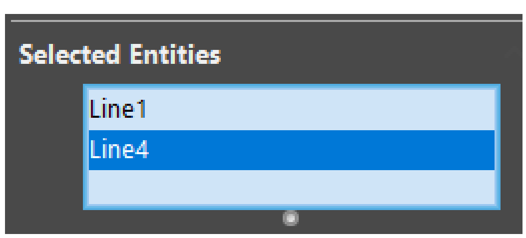
   iv. The left hand side of your screen will show any selected entities. If the box is not blank, right click on it, and click clear selections. If you accidentally select the wrong entity at any point, you can right click on it and delete just it from the selection, or clear the entire selection. Note that this does not actually delete the line, just removes it from the current selection.





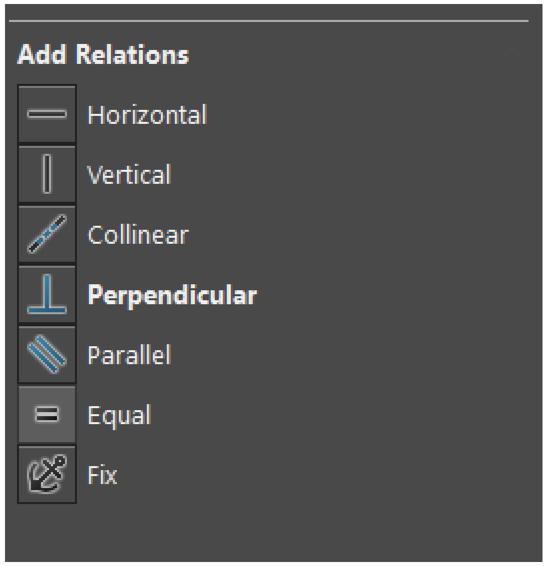
v. Select the top line and right side line of the rectangle. They both should show up in the selected entities box.





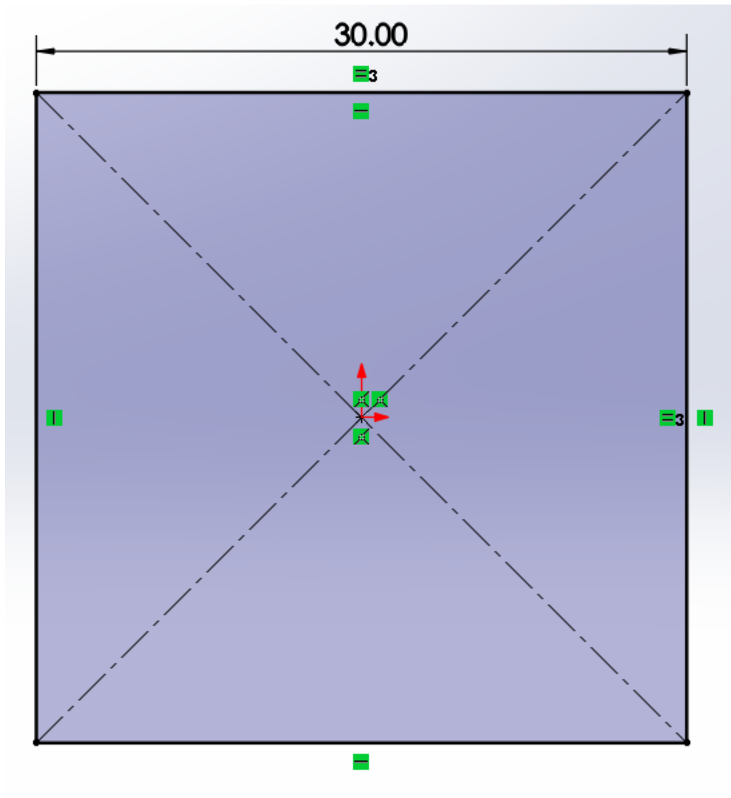
vi. Towards the bottom of the Tool Dashboard on the left hand side, you have an “Add Relations”

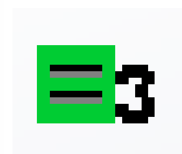
Section. This allows you to add any relations between the two lines you have selected. Click on "Equal" to make them both equal.



vii. You should see the rectangle turn into a square. You can now press either escape, or click the checkmark at the top of the Tool Dashboard.

viii. You can see your "equal" relation next to both of the lines you set to be equal (Equals sign in green box at center of top and right lines, with number 3 next to them). The number represents the set of lines that is equal. All lines with the equals Constraint/relation with a three next to it are the same length.



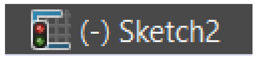


ix. If you ever want to delete a relation, you can left click on it, then press the delete key on your keyboard. Do this for the equal constraints now so you can learn the shortcut for adding relations.

k. Fully defined vs Partially defined.

i. Before adding the relation back, there is one property of sketches that is very important. Whether it is fully defined or not. Any fully defined entities are Black (see the black lines on the two sides of your square). Blue entities are not defined, and thus can be dragged to be any size because there are no dimensions or constraints limiting them in a certain direction.

ii. Undefined sketches also show up in the model tree with a “(-)” next to them.



iii. Undefined sketches are not ideal, as it means that there is something in your sketch that could be any size/location. You should try to make all of your sketches fully defined by adding either dimensions or constraints.

l. **Relations the fast and easy way.**

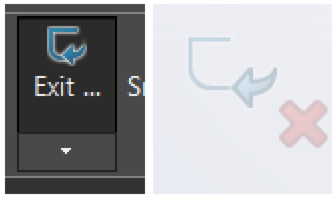
i. The fastest, and easiest way to add relations is to ctrl+click on the entities you wish to add a relation between. You do not need to hold ctrl when clicking on the first entity, but it also doesn't hurt.

ii. Hold Ctrl, and click on the top line and the right line. You will see that the same add relation menu shows up on the left side of your screen. Click equal as you did before, and the same equal relations should show up that you had before.

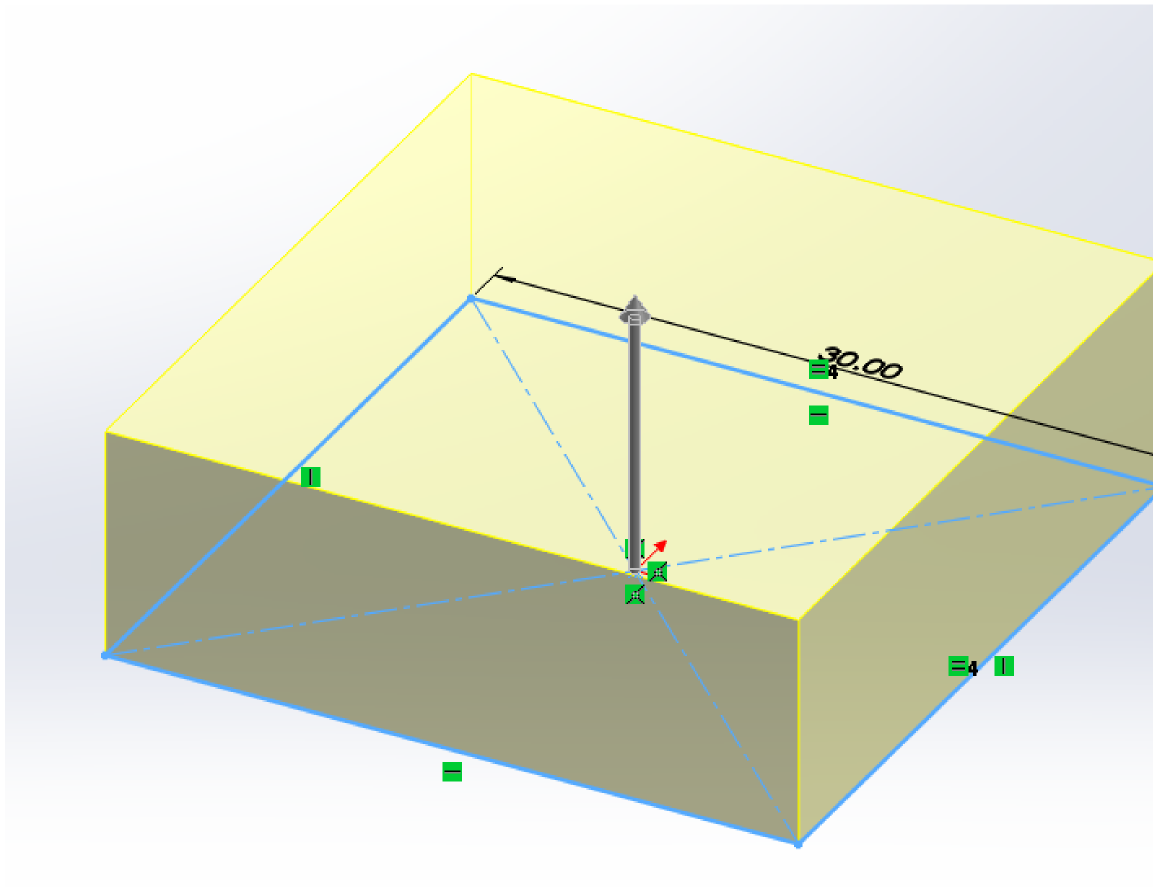
iii. You can add relations to points, lines, circles, arcs, and any other sketch entities. The type of sketch entities selected will determine the available relation types.

m. **Finish the extrude**

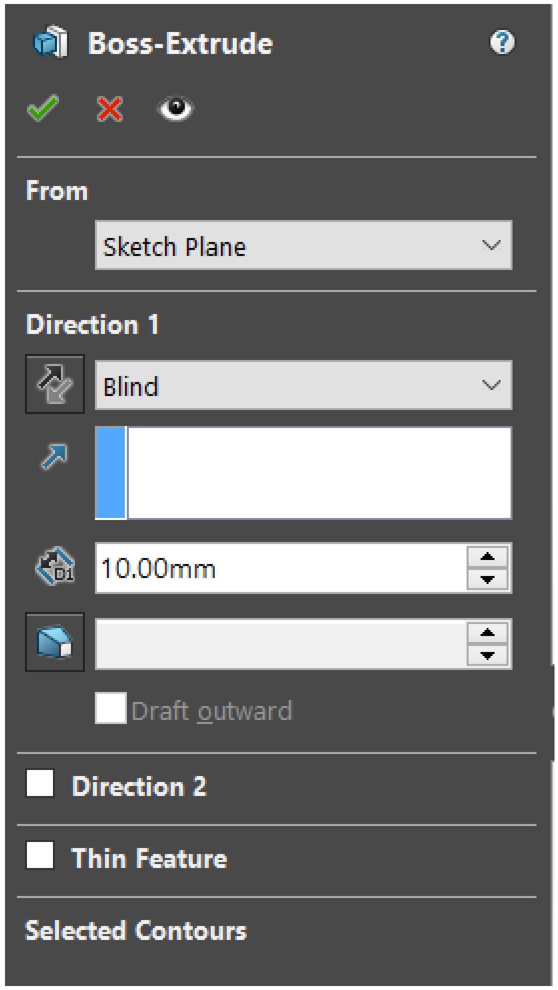
i. Click the exit sketch button. There is one in the top left of the toolbar in the sketch tab, as well as one in the top right of the graphics area. There is also a red X in the top right of the graphics area, which is to discard changes. Notice how solidworks put it right next to the exit sketch button so you'll accidentally click it after making a super complicated sketch and lose all your work? Make sure you're careful clicking the exit sketch button!



ii. Solidworks will show your square being extruded a set distance at first.



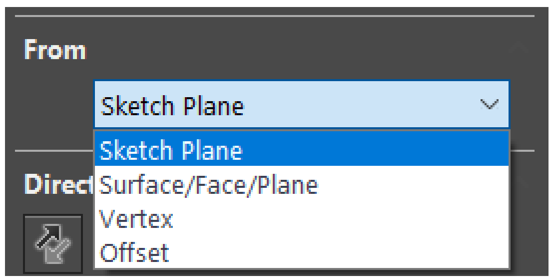
iii. Notice the options in the Tool Dashboard on the left.



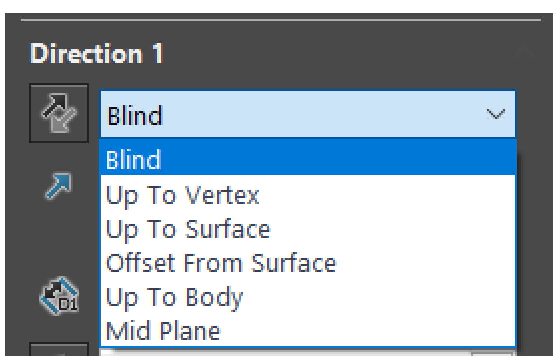
1. Select the surface from which to start the extrude. The sketch plane we selected was the top plane.
    a. Sketch Plane extrudes from the previously selected sketch plane.
    b. Surface/Face/Plane extrudes from a selected Surface/Face/Plane. This is useful when you have more complex geometries, or when you accidentally sketch on a plane parallel but offset from the one you wanted. You would select the Surface/Face/Plane in the blue box

that shows up once you select the Surface/Face/Plane option. Note that the surface selected does NOT have to be flat.

   c. Vertex is similar, but you select a vertex (point) instead of a Surface.
   d. Offset offsets the start of the extrusion a set distance from the sketch plane.

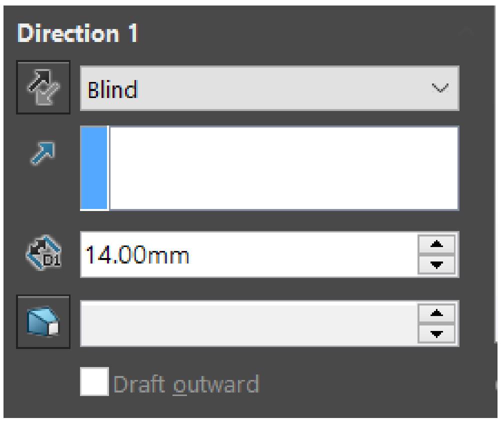


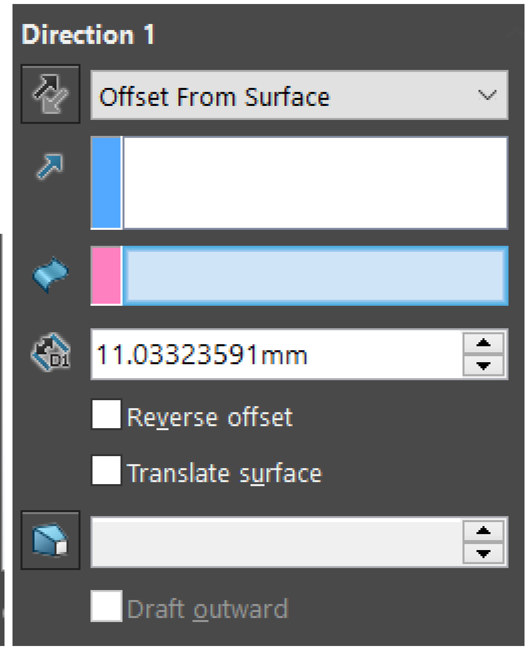
2. Blind/Up to Vertex/Up to Surface/Offset from Body/Up to Body/Mid Plane
   a. Blind extrudes a set distance from the selected “from” surface
   b. Up to Vertex extrudes to a vertex
   c. Up to Surface extrudes up to a surface. Note that this surface does NOT have to be flat.



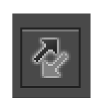
3. Notice that you can type in a distance as long as blind is selected. Different choices require

different inputs (ex. Offset from surface requires a surface as well as a distance).

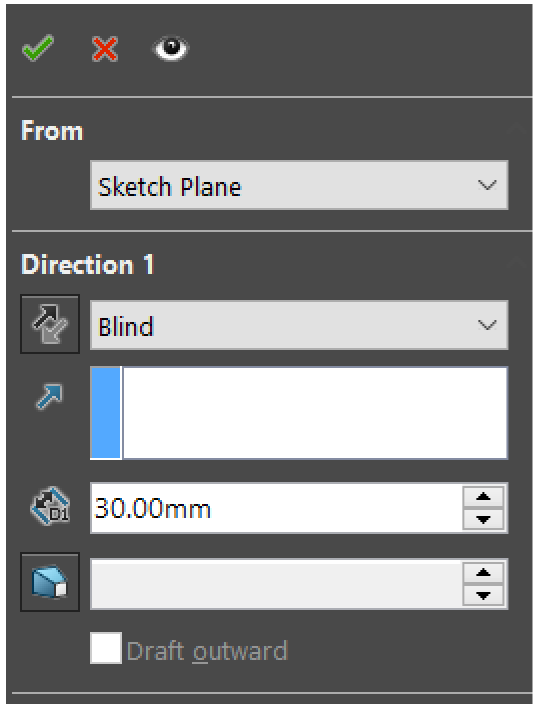




a. To select the surface in a window such as this, you would click the box that is highlighted blue in the Offset From Surface example, then click on the surface in the model you wish to use.

4. There are also more dropdowns for “Direction 2” which allows you to extrude both directions from the sketch plane (or selected “From” surface), and to do it to different distances.
5. It is also worthwhile to note that you can reverse the direction of extrusion with this button next to where it says “blind”: 

iv. We will be using “Sketch Plane”, “Blind” and setting the distance to 30mm.

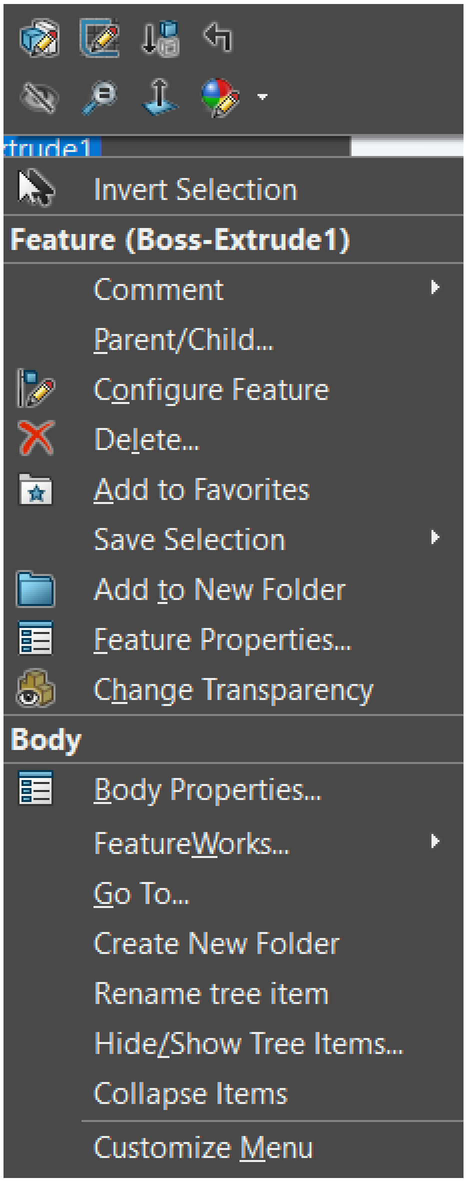


v. Press the green checkmark either at the top of the Tool Dashboard or in the top right of the Graphics Area, and you should have a grey cube.

7. Modify an Existing Feature

a. You will often need to go back and edit a feature. To do this, find the feature in the model tree. Right now we have just one called “Boss-Extrude1”
b. It is good practice to rename these to be something descriptive. To rename them, left click on the feature and press F2, or Left Click on it once to highlight it, pause, then again to allow you to edit it. You can also right click on it and then click on “Rename Tree Item” if you want. We will call the Cube feature “Cube”
c. Now, lets edit the feature. Right clicking on the feature yields a very large menu. You can hover over each icon to show what it does. Starting with the top row from left to right, the icons do the following:

i. Edit Feature: Edit extrude settings (blind, depth, etc)
ii. Edit sketch: Edit the drawing we made (the square)
iii. Suppress: Essentially turns off this feature/acts like it doesnt exit.
iv. Rollback: Allows you to act like this feature hasnt been created yet so you can add stuff “before” it in the model tree.
v. Hide: Hides the feature
vi. Zoom to selection: Zooms so the feature is centered in the Graphic Area.
vii. Normal To: Makes the sketch plane normal to the Graphics Area.
viii. Appearances: Allows you to add appearances/colors to the features.

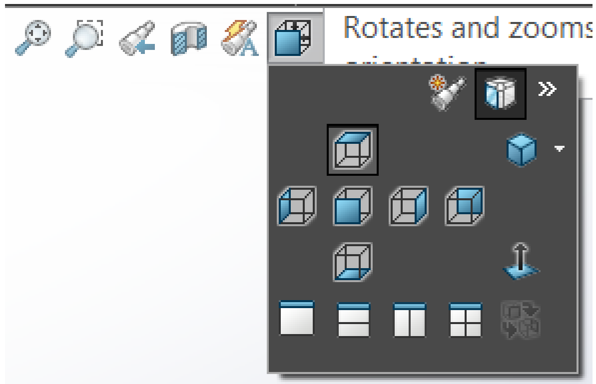


d. We want to select the first icon, Edit Feature.
    i. Notice that you can now modify any of the properties of the feature you wish to.
    ii. Press the green checkmark to exit the feature.

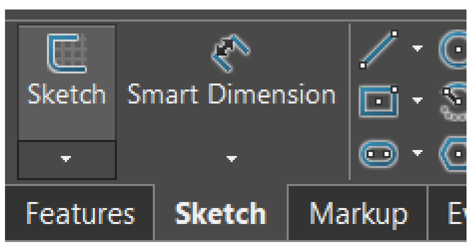
8. Create Holes in the part

a. Select the top face of the cube to sketch on.
    i. If you are not sure which face is the top, click the View Orientation dropdown from the graphics menu, and hover over the option to find "Top." You

can also press spacebar, and hover over the surfaces of the imaginary cube shown to find the top surface, then click on it.

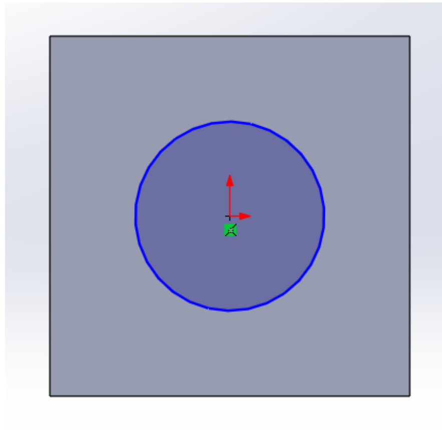


ii. After the top surface is selected, go into the sketch tab, and select sketch. Selecting the surface first allows you to go directly into the sketch. It is also possible to click sketch, and then select the surface you wish to sketch on.



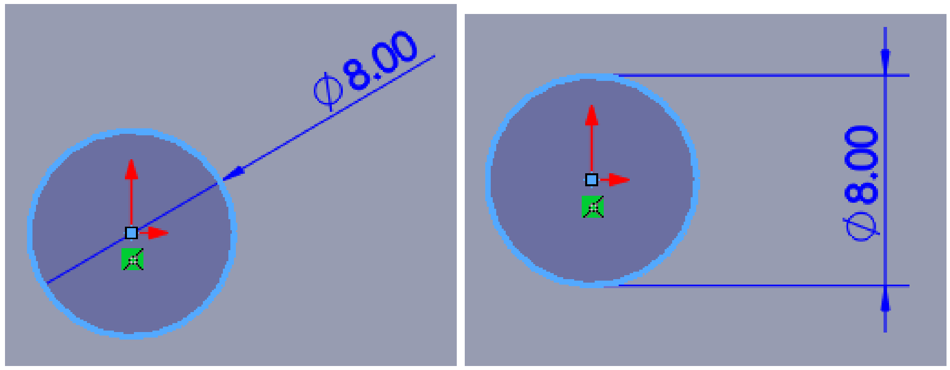
iii. Notice the same Origin as we had before when drawing the cube? Now we want to draw a circle with a diameter of 8mm.

iv. Select the Circle tool, then click on the origin and click again to create the circle, similar to how you made the rectangle.



v. Press escape to exit the circle tool.
vi. Notice that the center of the circle is fully defined (black) as it is on the origin. The circumference is blue because the diameter is not defined yet. You can click and drag on it to see this.
vii. Click the smart dimension tool then click on the circumference of the circle. You should see a little lock symbol by your cursor. If you move your mouse around, you should see that there are two different styles of dimensions that you can use in this scenario. If you want one of them, you can move to where it shows up that way, and click the RIGHT mouse button to lock the type of dimension, then drag it wherever you want to place it and click the LEFT mouse button to place the dimension. You do not have to do this, and you can just place it wherever it is that type of dimension, but in some cases this can be very useful. Specifically when you have a line that is not vertical or horizontal, if you wish to dimension it along its length vs its horizontal or vertical height, you can “select” how you wish to

dimension it this way, and place it anywhere you want.

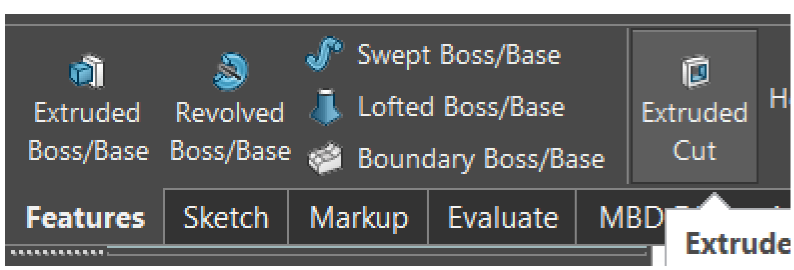


viii. After placing the dimension, type 8, and hit enter to make the circle 8mm.

ix. With the circle fully defined, click exit sketch.

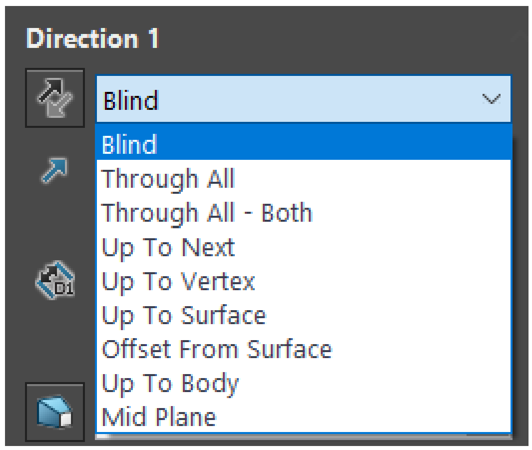
x. You may not currently be able to view the sketch. If this is the case, check that the "Show all types" eye symbol is not hiding everything. Click it to show everything. You may want to rehide the planes that we previously made visible.

xi. Click on the sketch we just made in the model tree. Note that it does NOT have a "(-)" next to it to show that it is undefined. With the sketch highlighted in the model tree, click on the features tab, and then click on extruded cut.



xii. You can orbit around the part to better see what will be cut away. Notice that you have very similar controls in the Tool Dashboard. The notable differences are the following in the direction dropdown:

1. Through All: Goes to infinity to cut away material in just direction 1
2. Through All - Both: Goes to infinity in both directions
3. Up to Next: Cuts until it hits the next surface
4. Up to Body: Similar to up to surface, but you select a separate body rather than a surface.

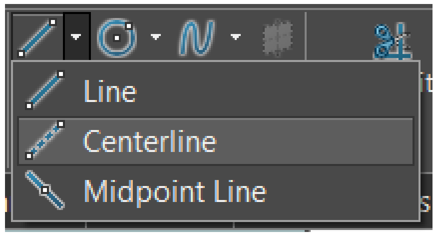


xiii. The best option for us is “Through All.” Select this, and press the green checkmark to complete the feature.

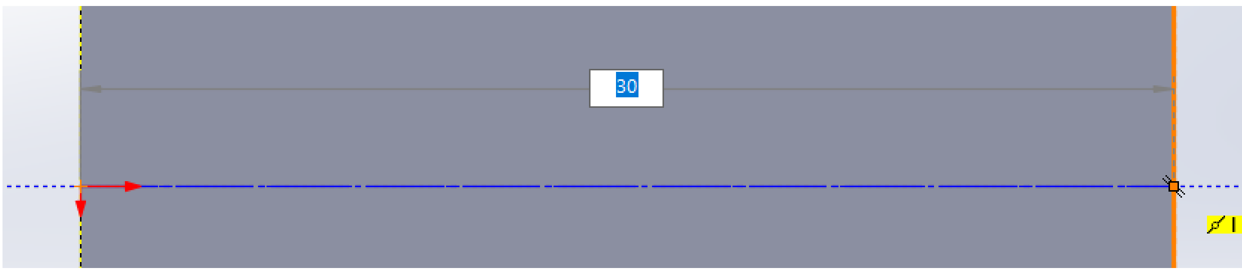
b. Now start a sketch on one of the other surfaces.

i. Notice that the origin is no longer centered on the cube. You have to create either construction lines or reference geometry to put the hole on center. We will do both for the two remaining holes.

ii. Starting with construction lines, click on the dropdown for the line tool, and click on “Centerline”



iii. Click on the origin, and then on the other side of the cube so that the line is horizontal, then click again.



iv. Press escape to exit the line tool. Notice that the line is black meaning it is fully defined.

v. It is worth noting that if you draw a regular line (or any other entity in a sketch: Arc, Circle, ellipse, etc.) you can click on it after drawing it and select “For Construction” from the Tool Dashboard. This allows you to convert lines to and from regular lines to construction lines.

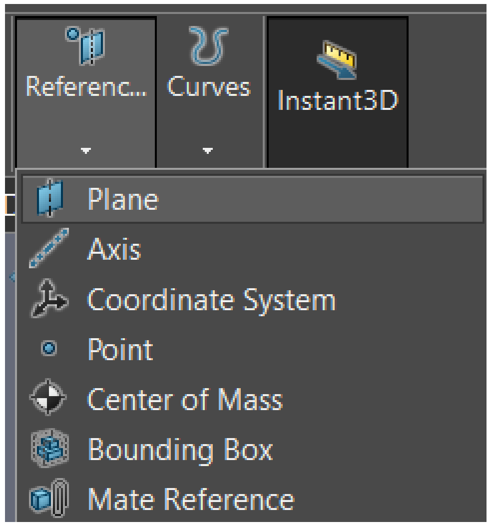
vi. Click on the circle tool, then hover near the center of the line. Notice how a dot shows up at the center of the line, which you can click on. This adds a midpoint relation (Which you can also do manually by selecting the line and the center point of a circle after drawing it).

vii. Dimension the circle (8mm), double check that it is fully defined, and exit the sketch.

viii. You can now extrude cut through all like we did with the previous hole.

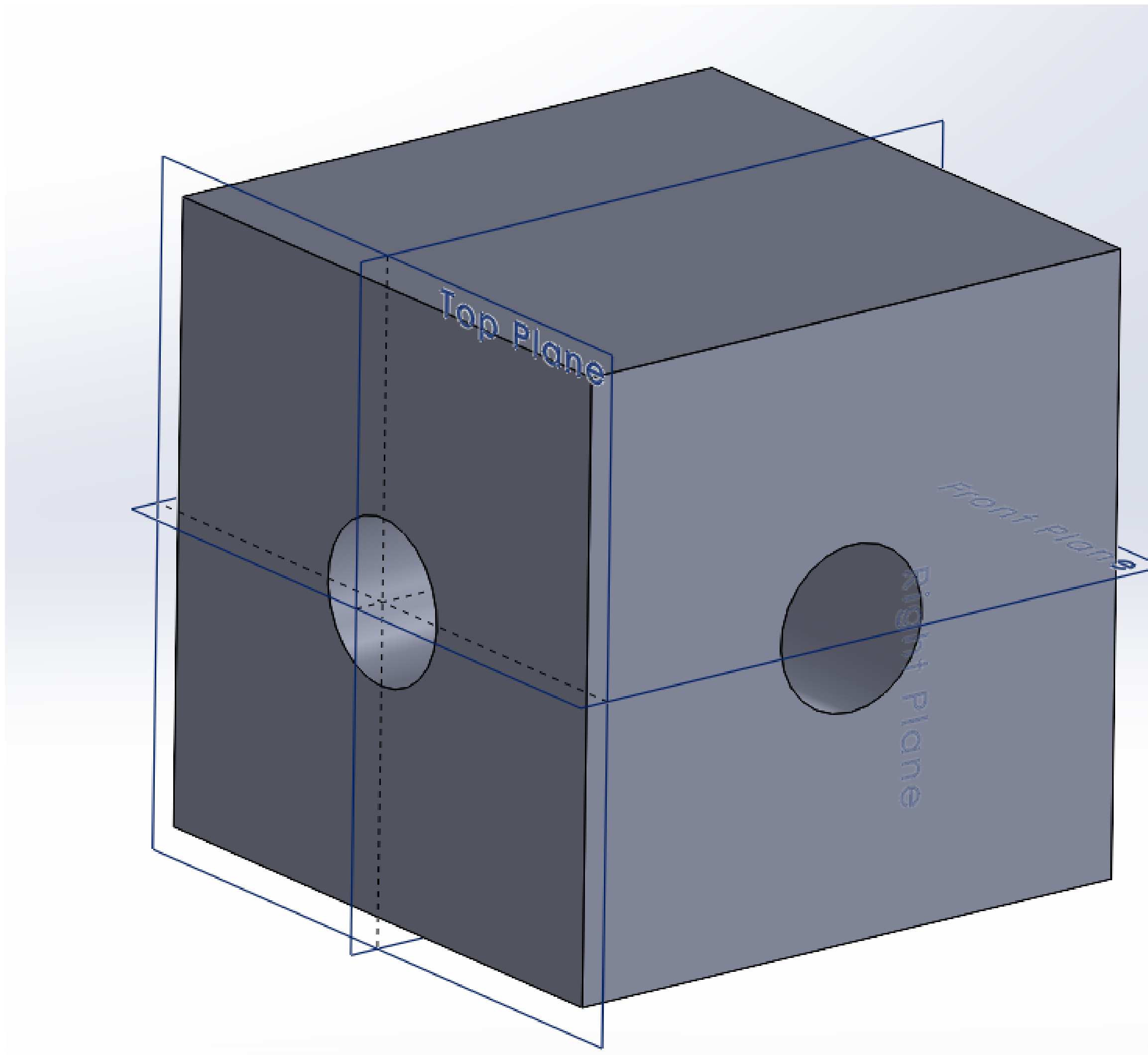
c. Create a reference plane

   i. Click on the “Reference Geometry” dropdown in the Features tab, and select “Plane”



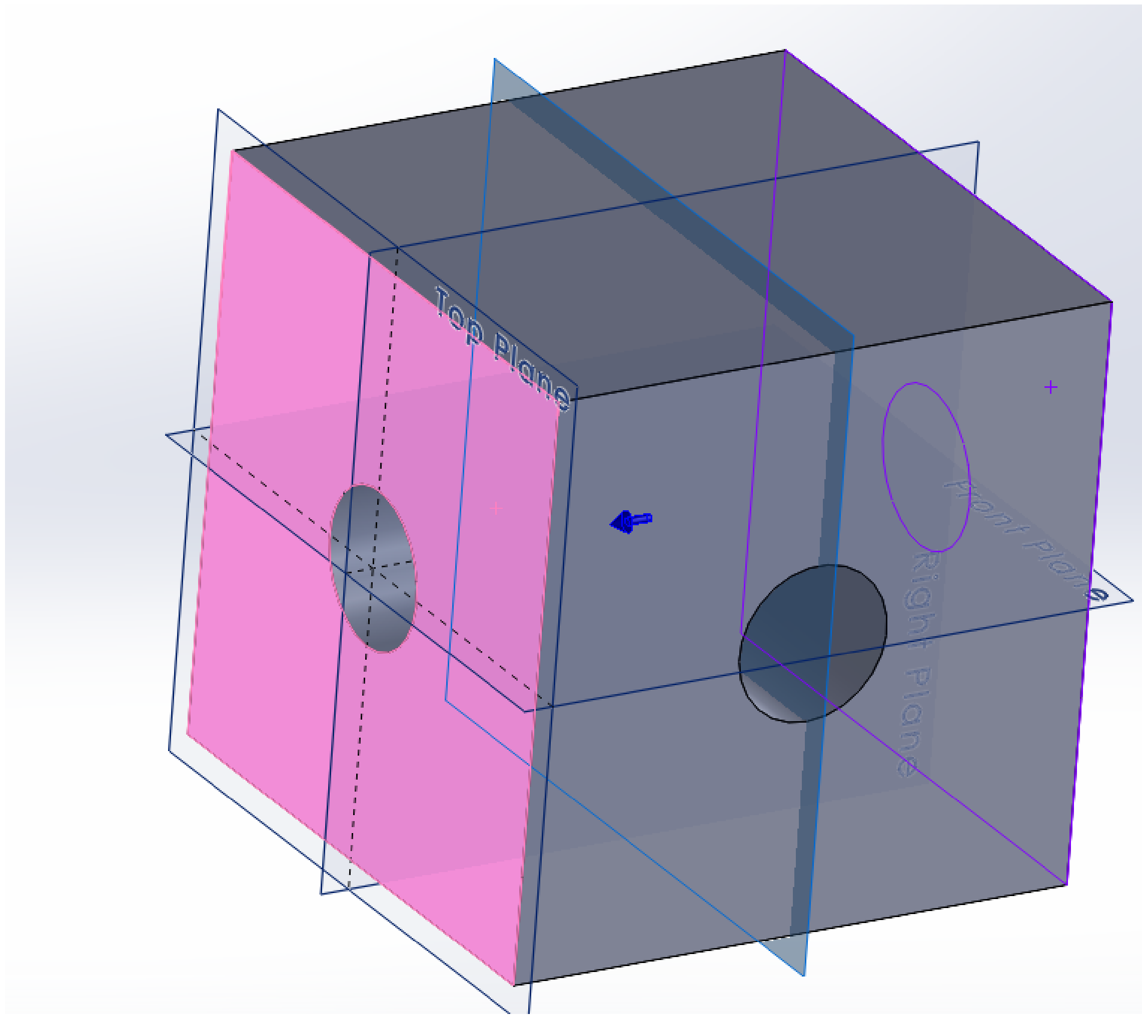
   ii. Notice that you can add up to three references in the Tool Dashboard to define the plane. We will select two parallel planes to define it. It is a good idea to make the primary planes visible again.

   iii. We want to make a plane that is parallel to the top plane, but centered on the cube.



iv. For the first reference, select the surface of the cube along the top plane. For the second, select the surface of the cube opposite the top plane.

v. Solidworks should show the plane halfway between the two surfaces, right where we want it. It should also say “Fully defined” in the Tool Dashboard.

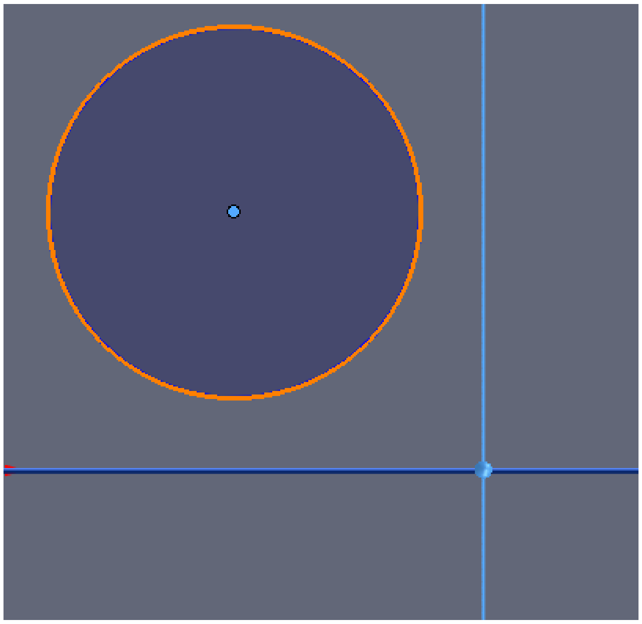


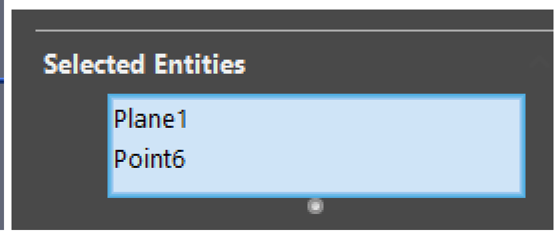
vi. Note that you can change the relations between the references to achieve different results. This can be useful to make more complex parts.

vii. Press the checkmark to complete the reference plane.

d. Start a sketch on one of the sides that doesnt have holes.

i. For practice, draw the circle without the center on any of the visible planes, and use relations to make the center of the circle coincident with the planes that form a cross over the center of the cube.





ii. Make sure you dimension the circle to me 8mm, and exit the sketch, then do a cut extrude feature through all again.

9. Fillets and Chamfers

a. Next, we will make the edges of the cube softer.

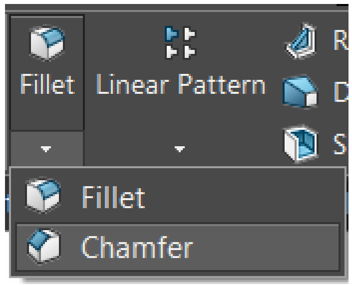
i. In the features tab, click on the fillet tool.

ii. Select all of the edges of the cube so they show up in the "Items to fillet" box in the Tool Dashboard. If you cannot select anything, first click in the box so that it is highlighted, then click on the edges.

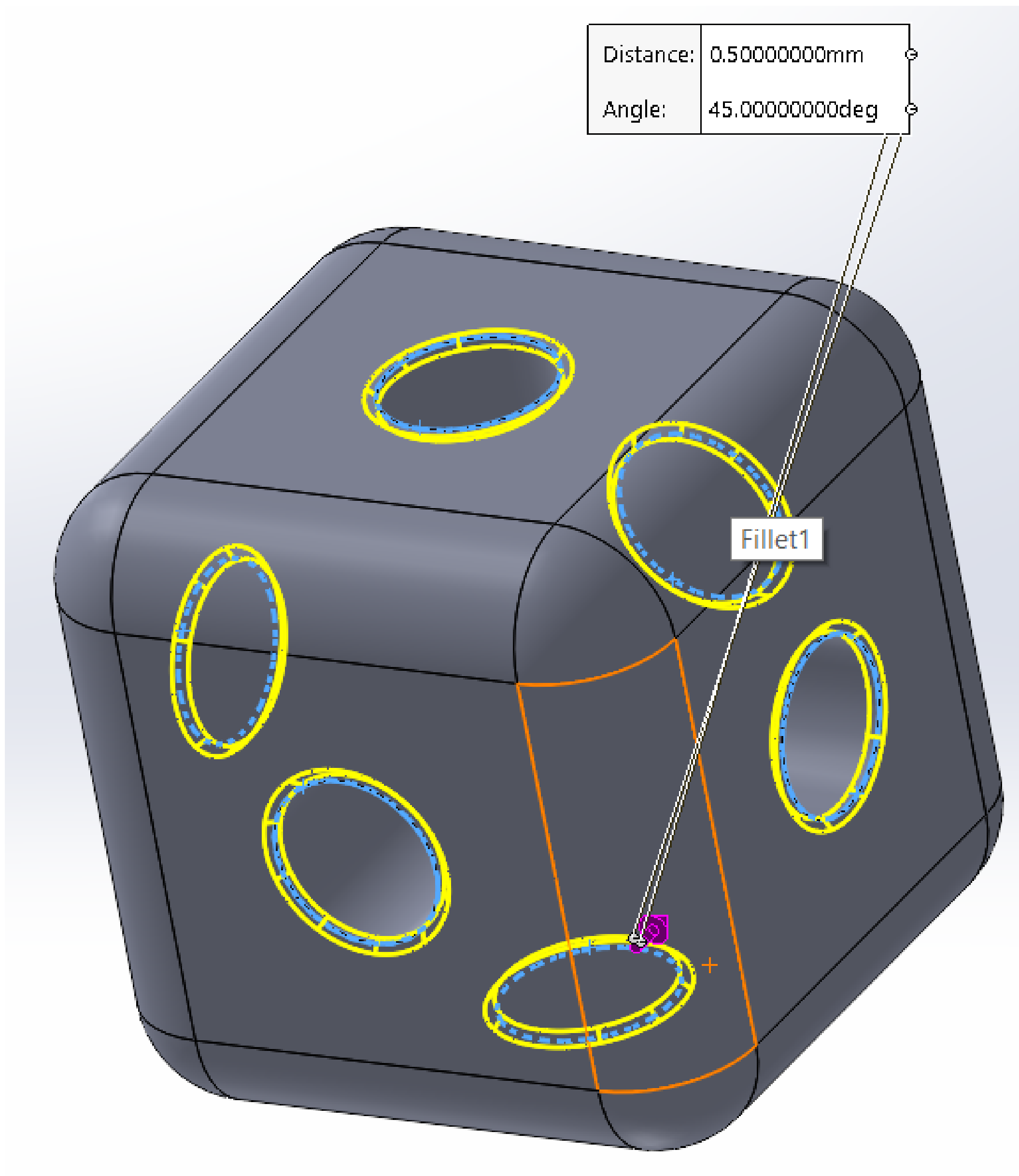
iii. Make the radius 5mm, then click on the check mark. Notice how much nicer the cube looks.

b. Next we will add a chamfer to the edges of the holes.

i. Click the dropdown on the fillet tool in the features tab, and click on chamfer.

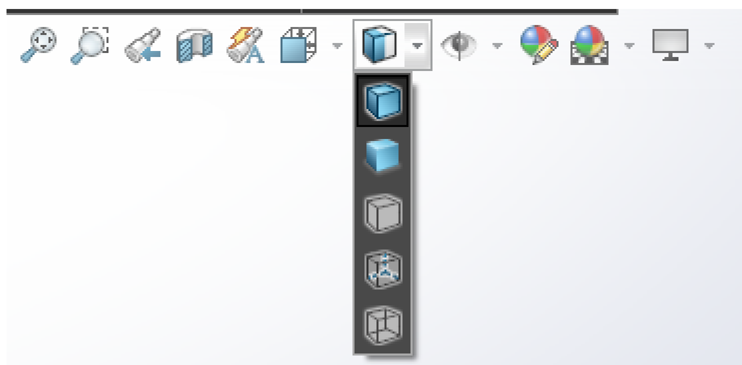


ii. Select all of the inside edges of the holes



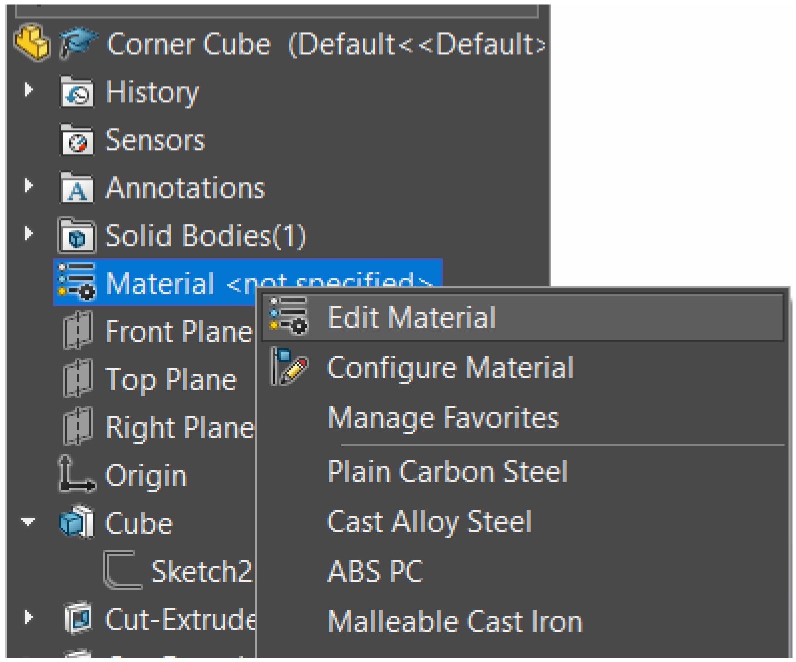
iii. Make the size of the chamfer 0.5mm at 45 degrees, and press the check mark.

10. You have completed this part! Make sure you hit save.
11. Before moving on, you should learn about different display styles, and selecting the part material.
    a. Display styles:
        i. In the Graphics Toolbar, select the display styles dropdown.

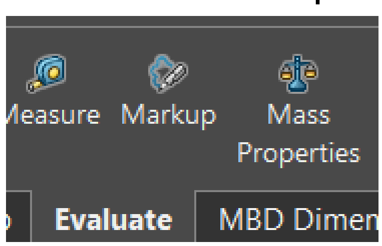


        ii. Click on each of the settings to see what it shows. With the wireframe views, you can see inside of the part at the internal geometry.

    b. Material selection:
        i. In the model tree, there is a line labelled “Material <not specified>”
        ii. By right clicking and clicking on “Edit Material” you can select the material of your choice.



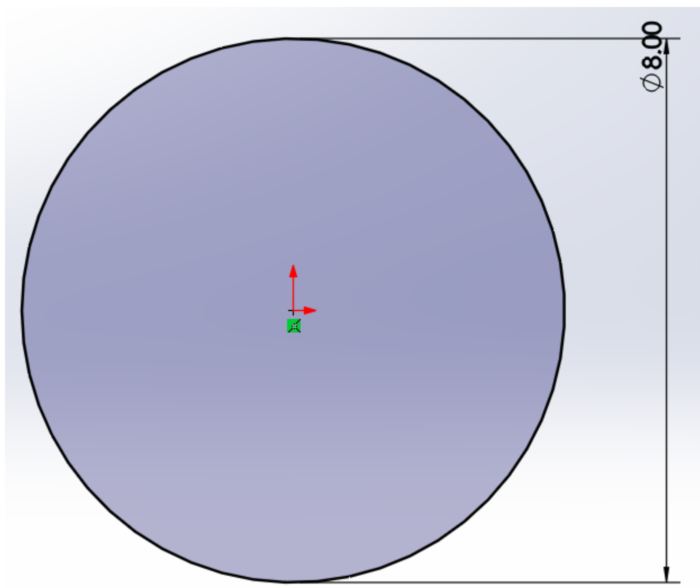
iii. Search in the window that shows up for a material you like, and click apply, then click close. Notice that the color of your part changed to be more similar to that material, and the material is now shown in the model tree. As well as this, density and other stats of the model updated as well, which can be seen in the evaluate tab if you click on the Mass Properties tool.



12. Let's begin the second part!
    a. Make sure you save the cube, and then create a new part and name it “Strut”.
    b. Create the main cylinder
        i. Start a sketch on the right plane one of three ways:

1. Select the plane from the model tree, then click “sketch” in the sketch tab
2. Click sketch, then select the right plane from the dropdown next to the tool dashboard
3. Click sketch, and show the primary planes, then click on the right plane.

ii. Sketch a circle centered on the origin, and dimension it to be 8mm.

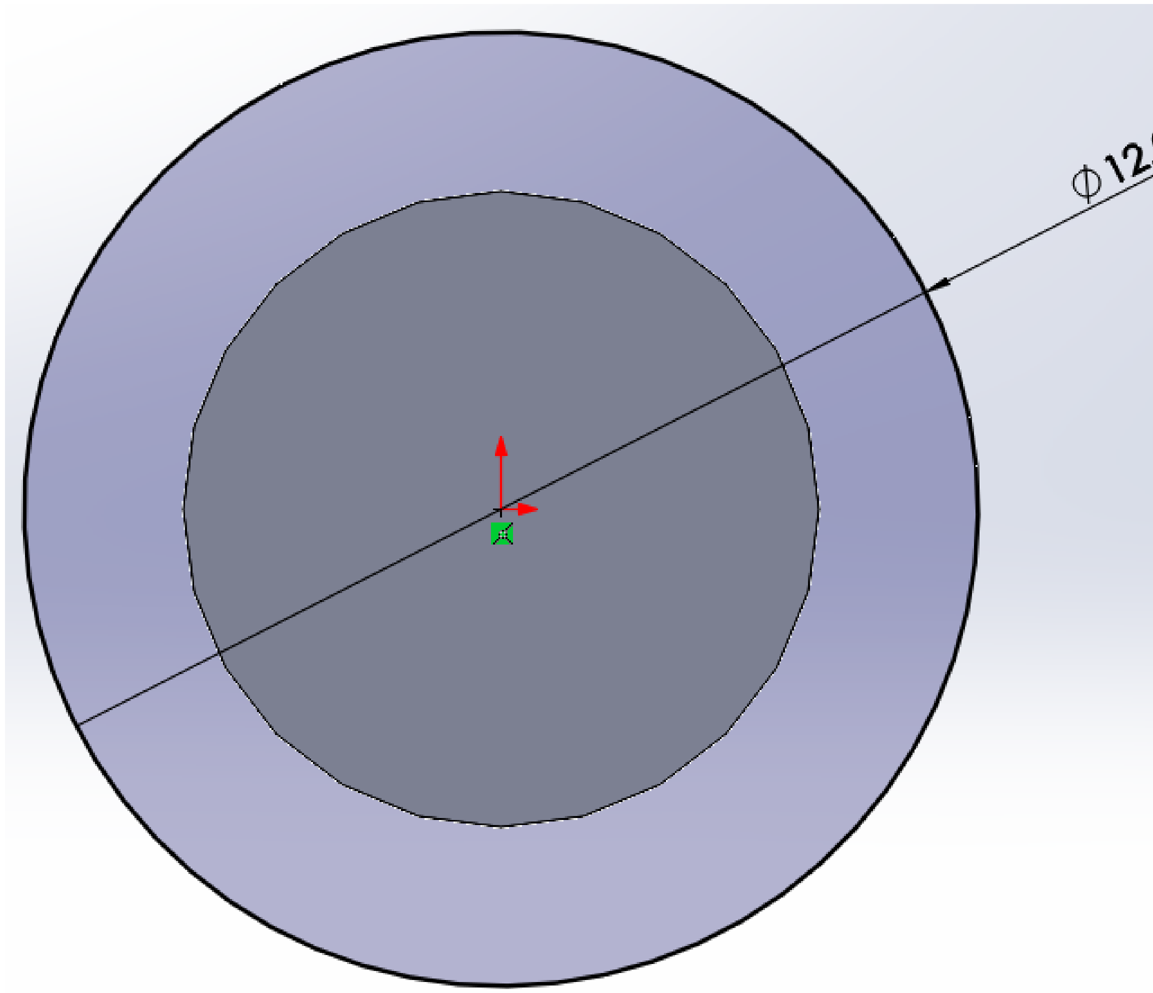


iii. Without exiting the sketch, click on the features tab, then click Extruded Boss/Base. (Engineers love shortcuts)

iv. Extrude this cylinder midplane (change blind to midplane) and 90mm, then press the check mark.

c. Start a new sketch on the right plane, and draw another circle centered on the origin, this time 12mm.

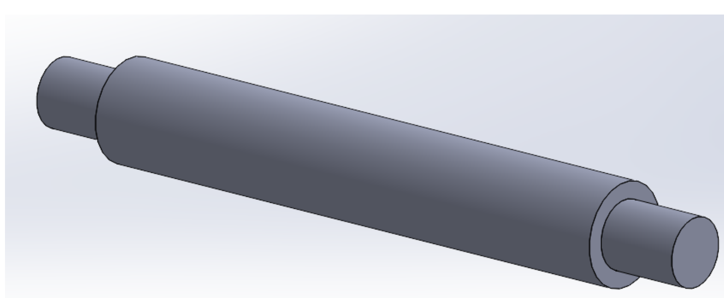
i. Note that it is very useful to orient the part so your sketch plane is parallel with your monitor. Press the spacebar and select the correct face of the imaginary cube, and the model will orient itself to make your life easier.



ii. Once you have the circle, click Extruded Boss/Base again.

iii. This cylinder will be extruded midplane 70mm.

iv. The part should look like this once you have finished that.

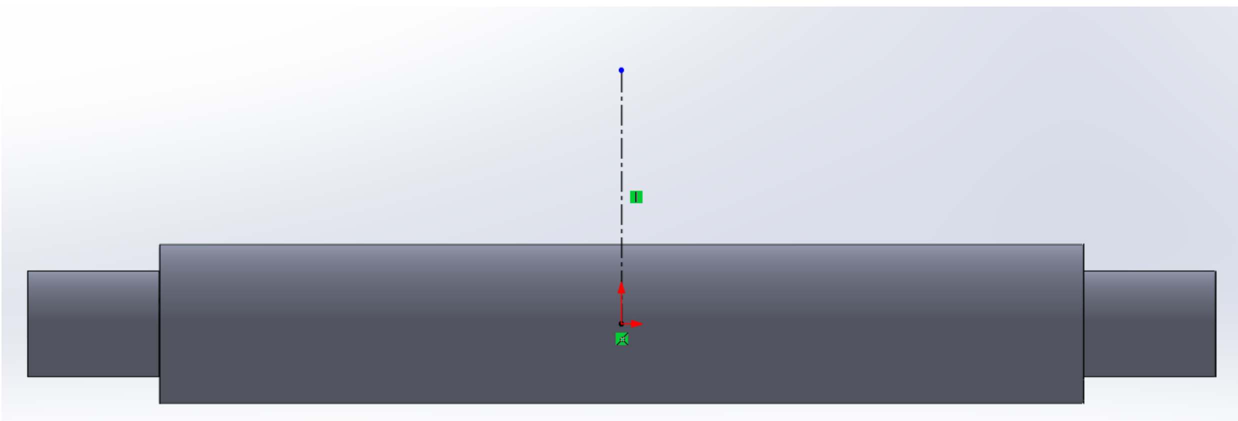


d. Create a Sketch to perform a Revolved cut

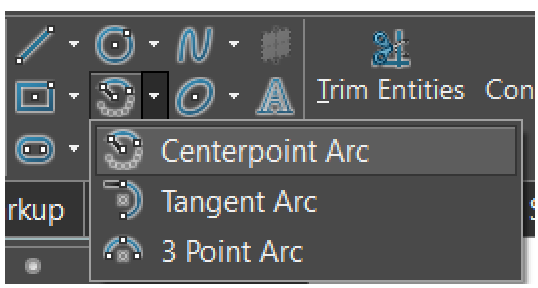
i. A revolve in solidworks is taking a two dimensional slice, and rotating it about an axis to create a solid part. If you have taken Calc 3, you have used this method to find the volume of mathematical functions in three dimensions.

ii. Create a sketch on the Front plane, and orient your part so you are looking directly at your sketch plane.

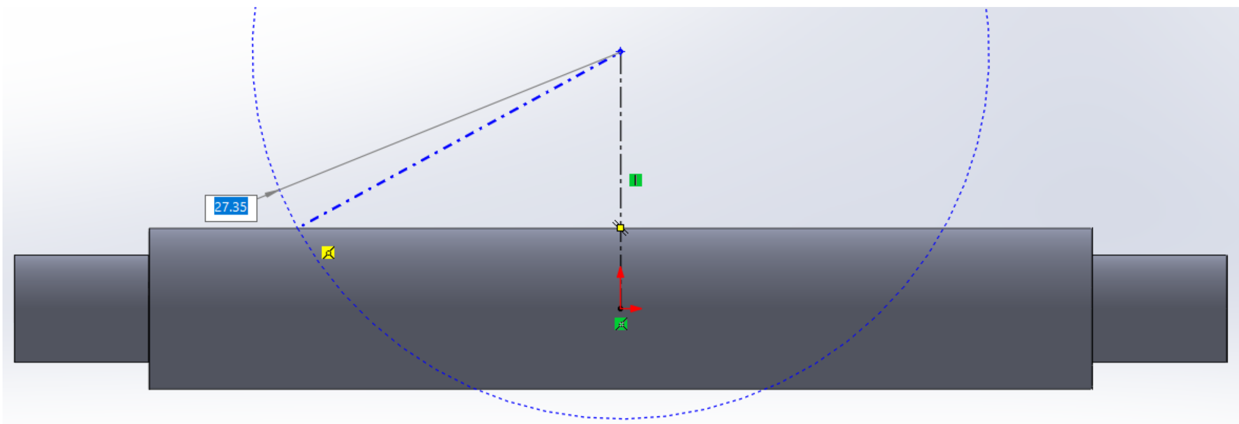
iii. Create a construction line that is vertical from the origin.

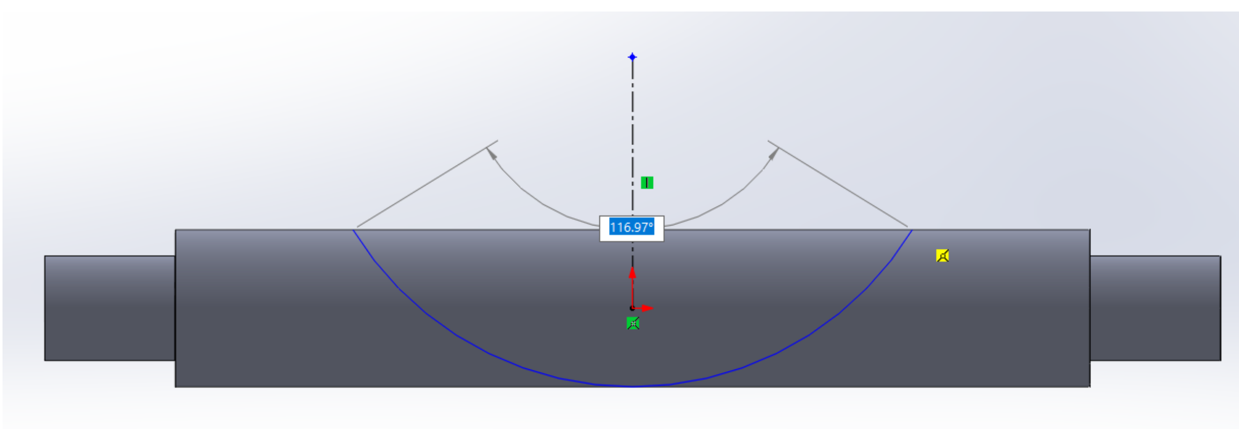


iv. Select the centerpoint arc tool.

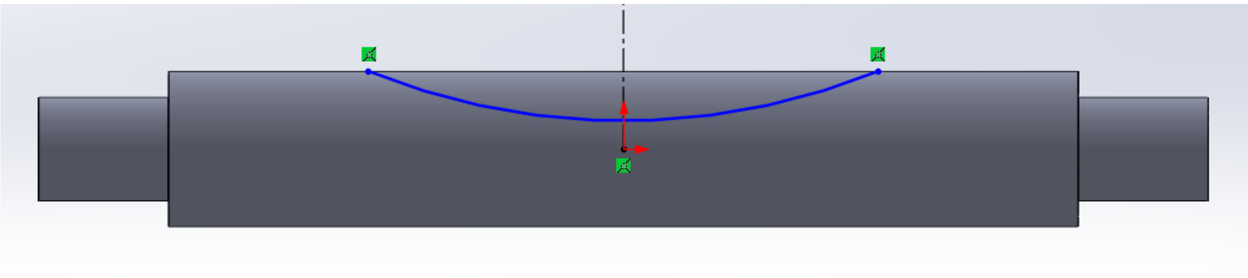


v. Click on the top endpoint of the line, then on the left side of the top edge of the larger cylinder (Second click shown below), then on a symmetric part on the other side (Third click shown below second). Note that you make need to move your mouse around to get the arc to be the direction it is shown below (so arc is overlapping existing cylinders). Also, ensure that the “point on entity” constraints pop up before you click. They are the yellow squares that look like relations (because they are). If these do not show up, you can manually add a relation between the horizontal line of the cylinder and the endpoint of the arc.

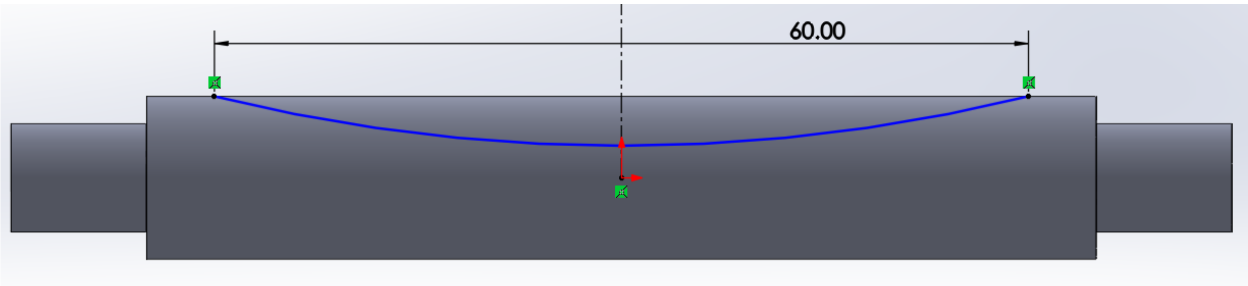




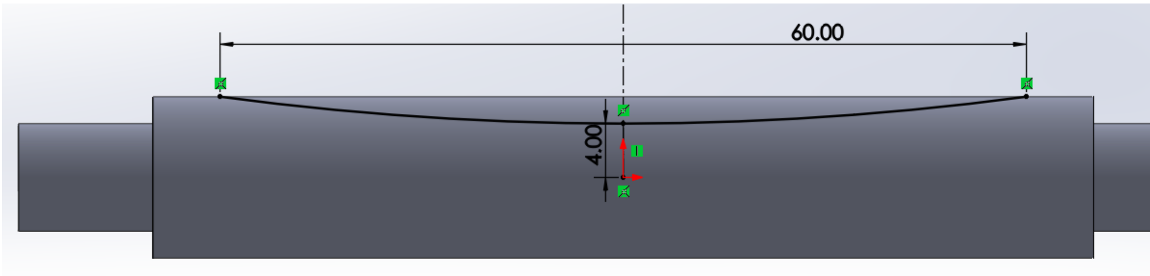
vi. Click and drag the bottom of the arc and endpoints to get it positioned similar to what is shown below.



vii. Dimension horizontally between the two endpoints of the arc so it is 60mm long.

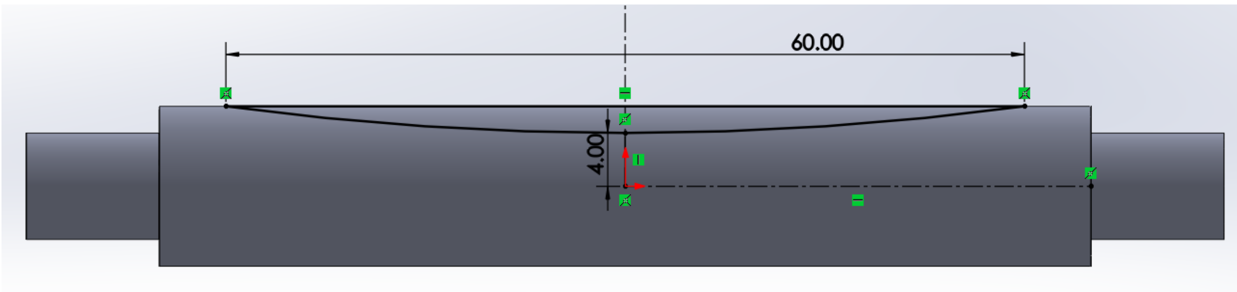


viii. Create a construction line that goes between the bottom/midpoint of the arc, and the origin, and dimension it to be 4mm. When dimensioning, you may need to select the endpoints of the line rather than the line itself since there are two overlapping lines there. The entire sketch should be defined.



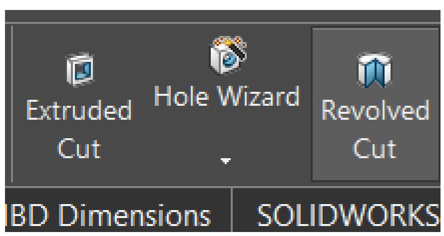
ix. You want to close the sketch by adding a line between the two endpoints of the arc, and add a centerline (construction line) along the center axis of the cylinder. It does not need to be the full distance, but you can check the “Infinite length” attribute in the Tool Dashboard with the line selected if you wish. Make sure that it is defined, and add any necessary constraints (To the origin and Horizontal) if it is not.

x. You can then exit the sketch.

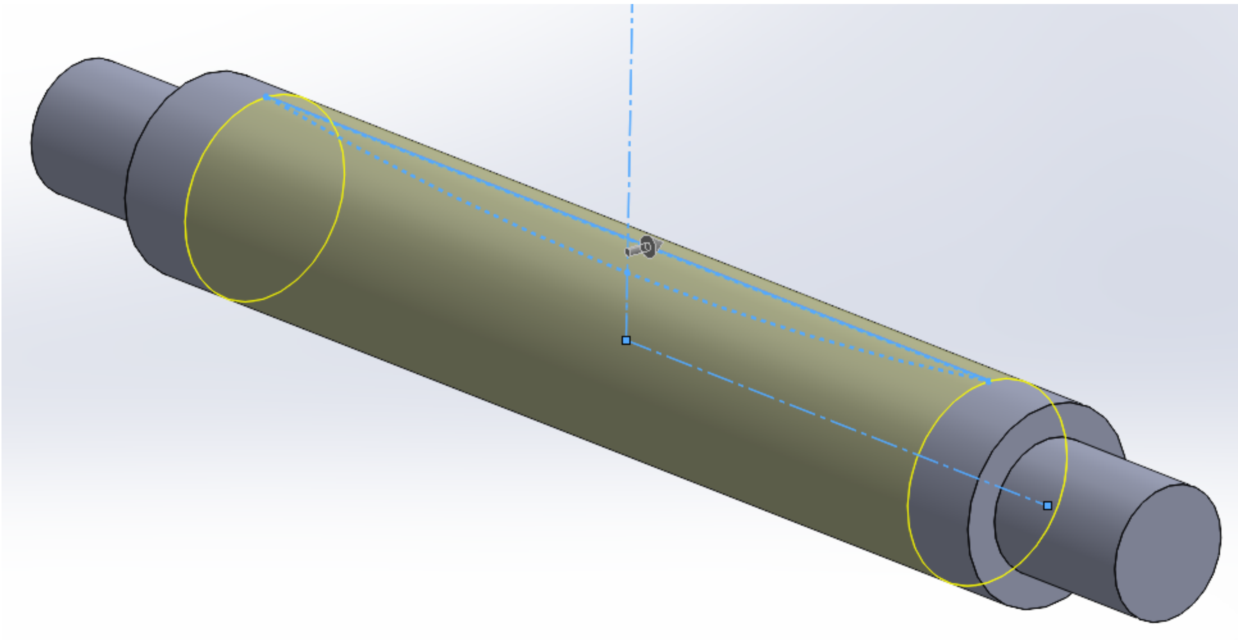


e. Create the revolved cut Feature

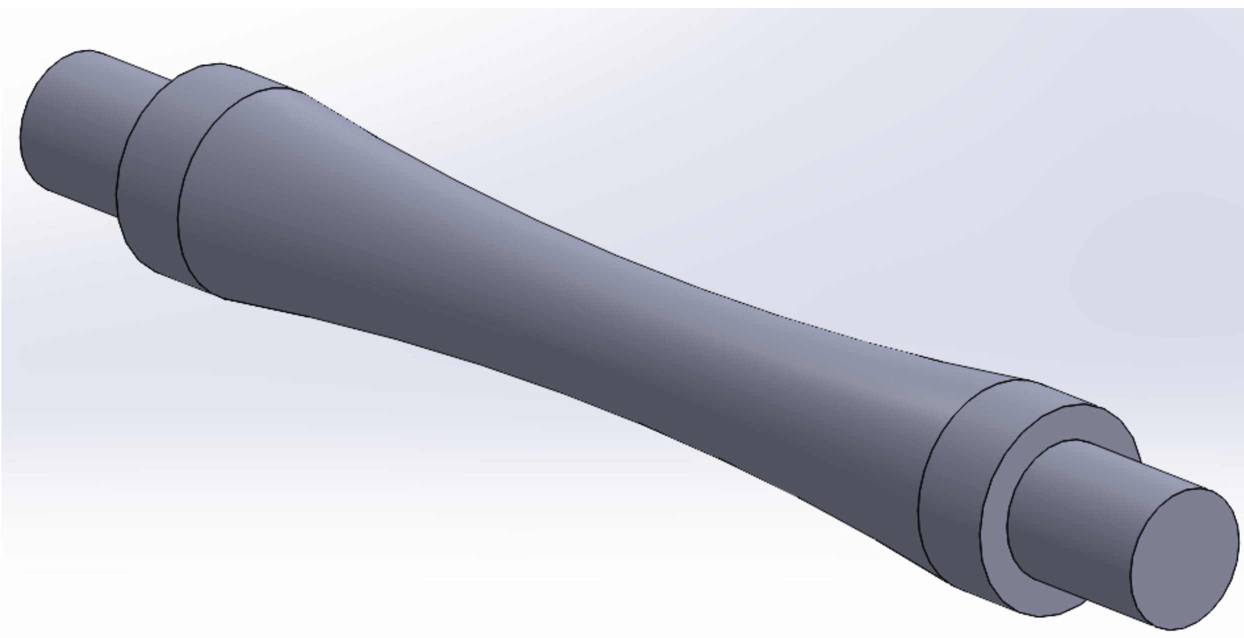
i. Select the most recent sketch from the model tree, then go to the feature tab and click on Revolved Cut.



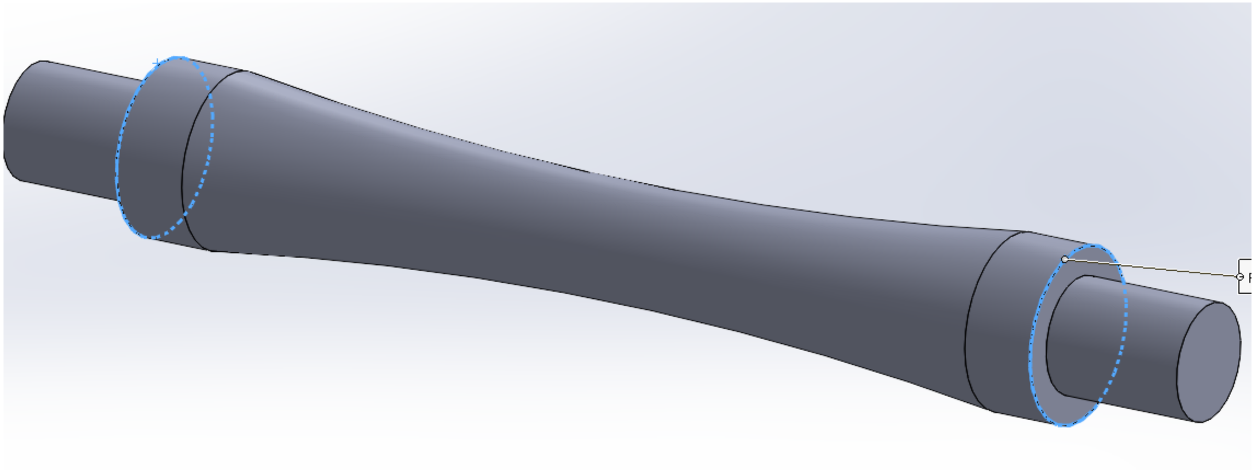
ii. The first thing the Tool Dashboard asks for is the axis of revolution. Solidworks allows you to use either an axis or construction line in a sketch for this. You can create an axis similarly to how we made a reference plane before (From the Reference Geometry dropdown). We will use the construction line we just created. Click on the Construction line in the sketch that is the axis of rotation, and make sure that the revolved cut is set to 360 degrees. Then press the checkmark.



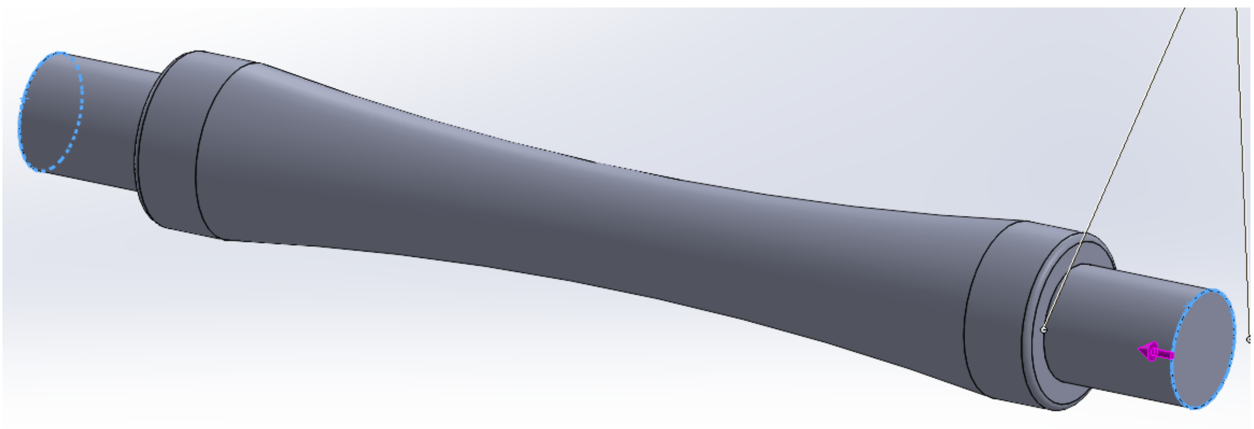
iii. After pressing the checkmark, the part should look like this:



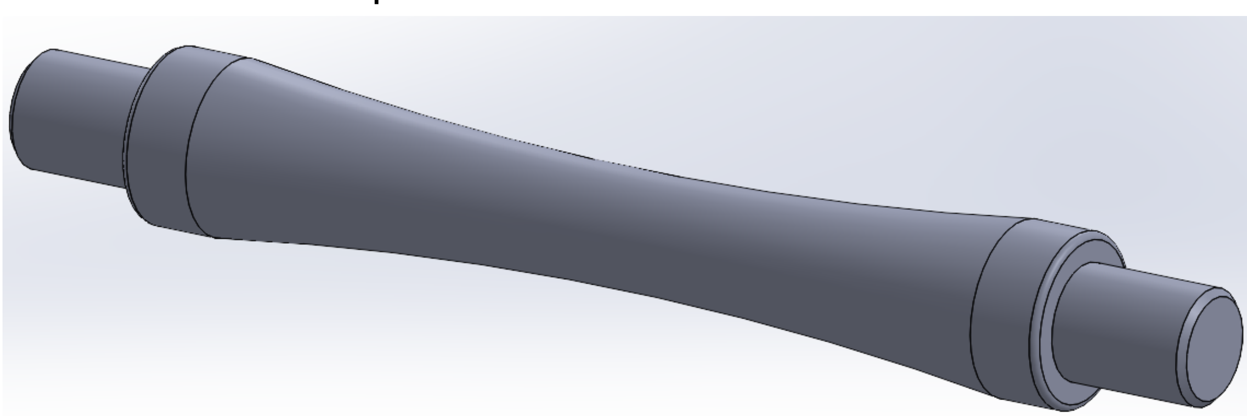
f. Adding Fillets and Chamfers
   i. We will Fillet the edges shown below with a 0.5mm fillet:



ii. We will Chamfer the following edges with a 0.5mm 45deg Chamfer

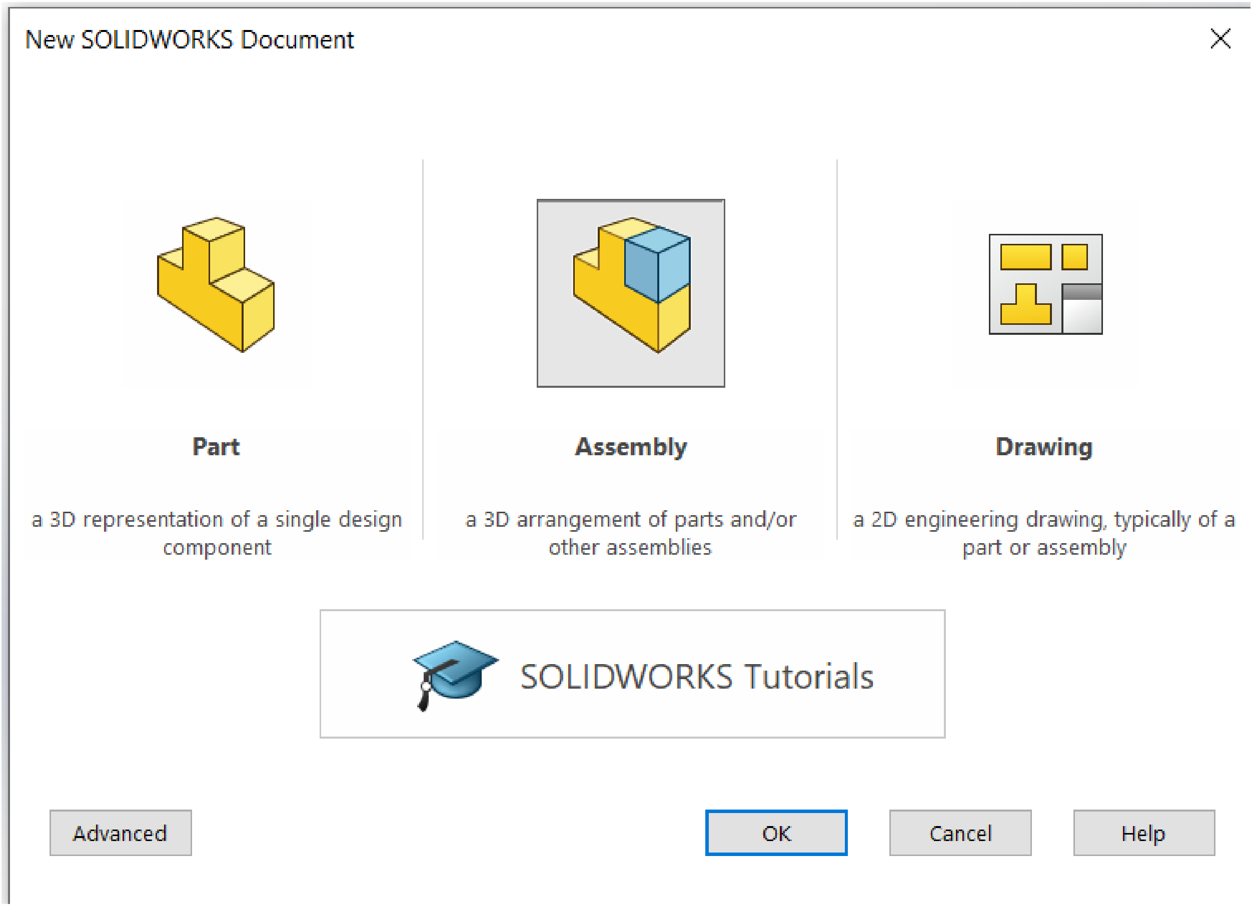


iii. The part should look like this when done:



13. Assembly Time

a. Create a new file, this time an assembly.



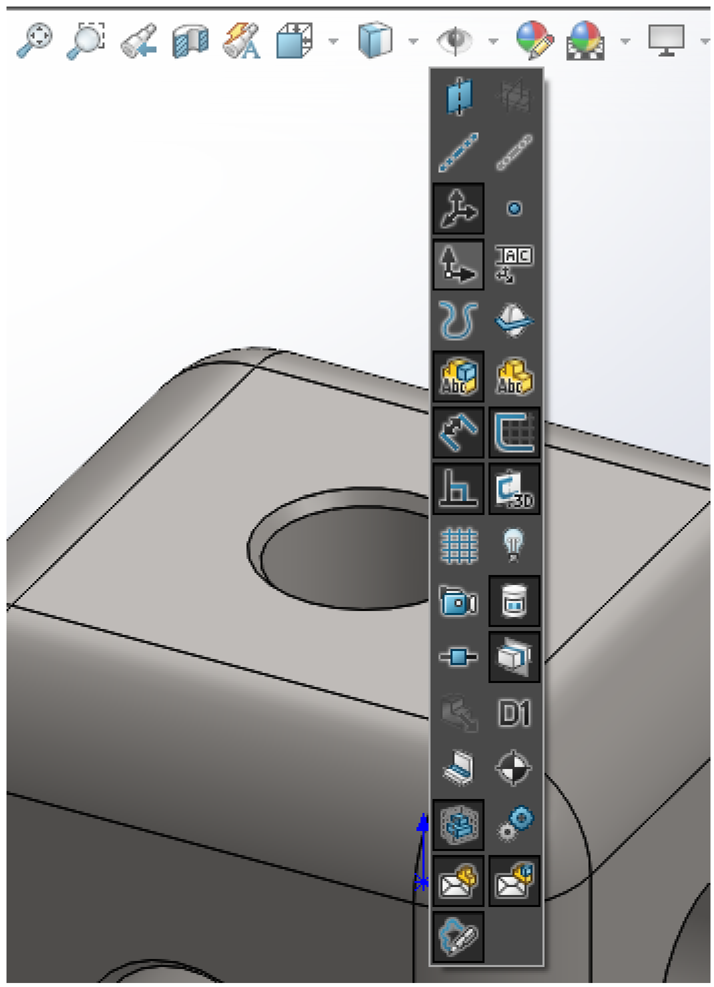
b. The Tool Dashboard will ask you for a first part to insert into the assembly. If you still have the two parts open, they will show up under "open documents" otherwise you can click browse, and locate the parts. Click on the Corner Cube, move your mouse into the Graphics Area and left click to place the cube.

c. Notice that there is now an "Assembly" tab in the ribbon. This is where all the tools we will be using right now are located.

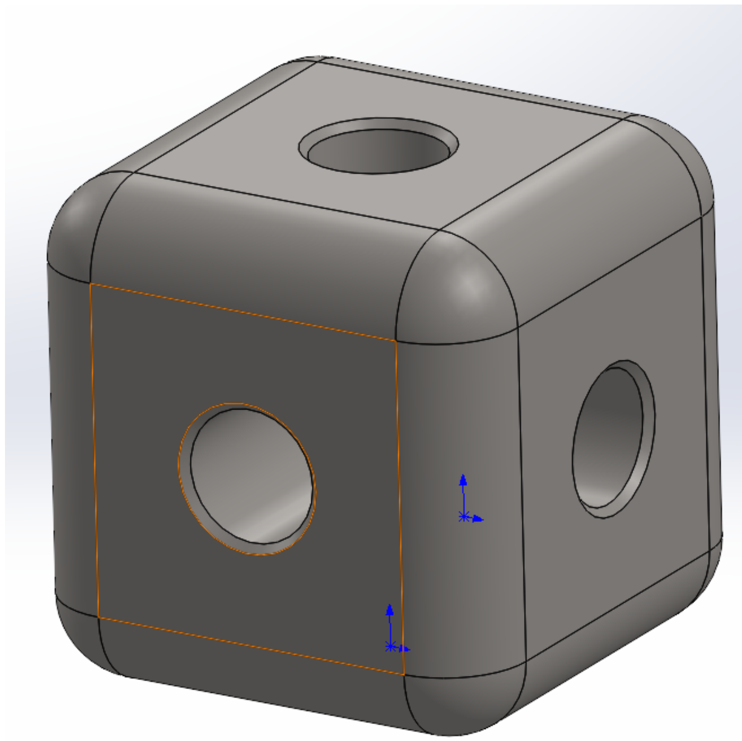
d. Relocating the Cube to the Origin of the Assembly

   i. This step is not necessary most of the time, but is still good to be aware of.

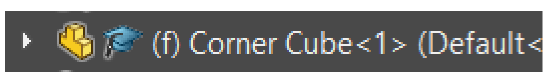
ii. Click on the Hide All Types drop down in the Graphics Toolbar (Eye), and select both of the coordinates options so you can see where the coordinate of the assembly is relative to the coordinates of the cube.



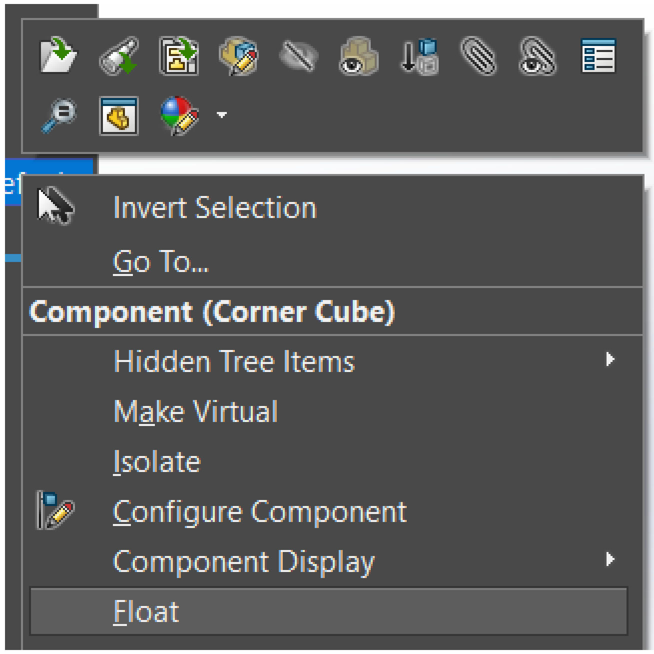
iii. Notice how they are not centered on each other.



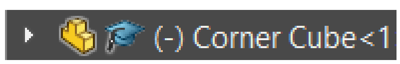
iv. This is because solidworks just fixes the first component in an assembly in space. This is also shown in the model tree, denoted by an (f).



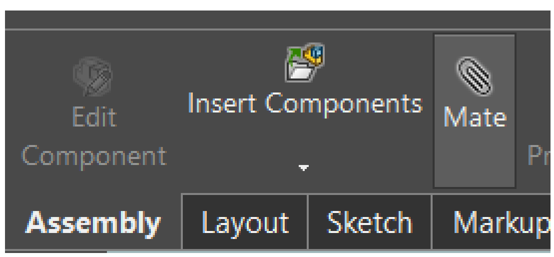
v. To resolve this, and align the axis, right click on the Corner Cube in the model tree, and click “float”



vi. You will notice that instead of an (f) there is now a (-) meaning it is undefined. You can click and drag the cube around in the assembly too.

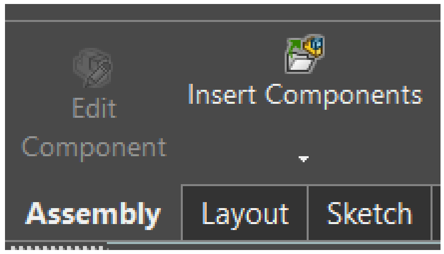


vii. To align the origins of the cube and the assembly, click on Mate in the Assembly tab.

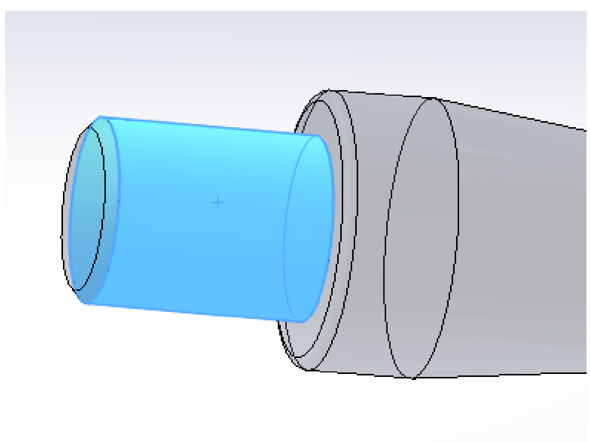


viii. With the mate tool, you can select two mating surfaces, edges, points, origins, axis, etc. and pick the type of mate between them. In this case, we will click on one of the blue origins and then the other, and Solidworks will know we want to place them on top of eachother and align the axis (look at the checkbox in the Tool Dashboard). Once the axis are aligned, you can press the green checkmark (once to complete the mate, and again to exit the mate tool).

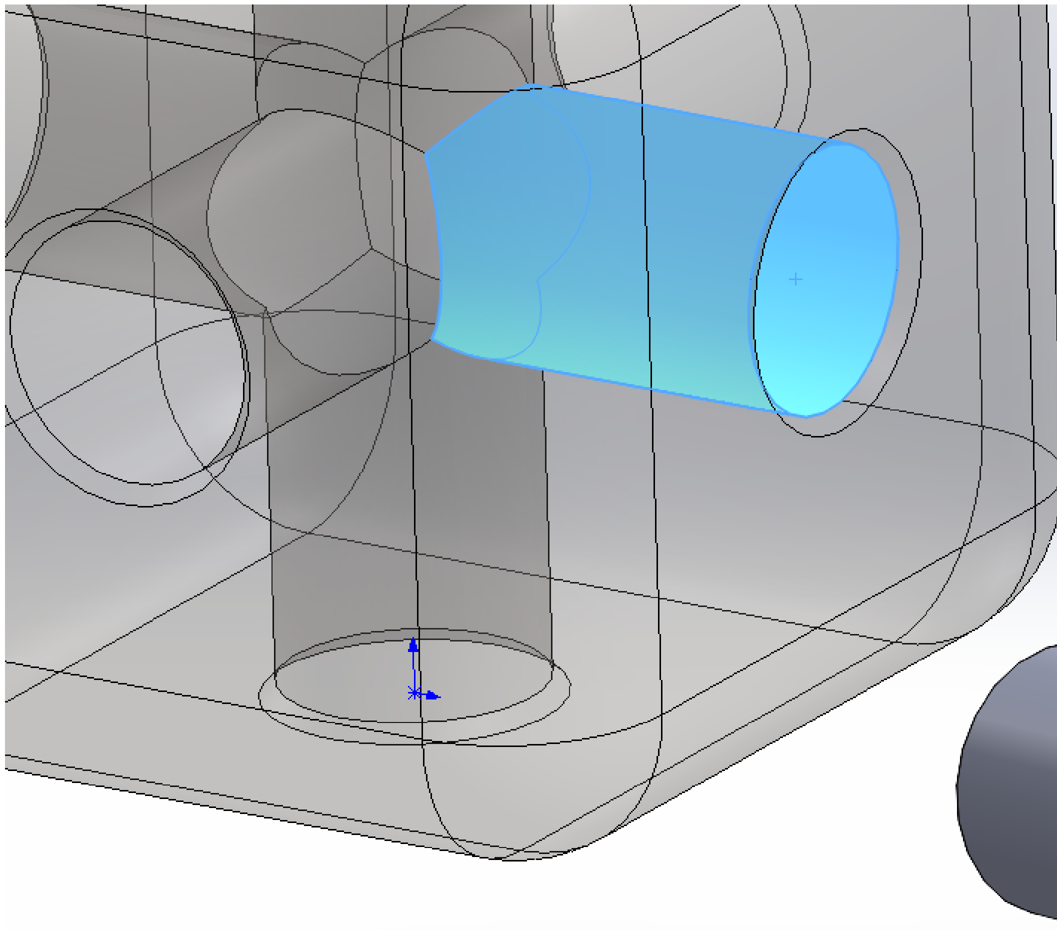
e. We will now insert the next part.
   i. Click “Insert Components” in the Assembly tab.



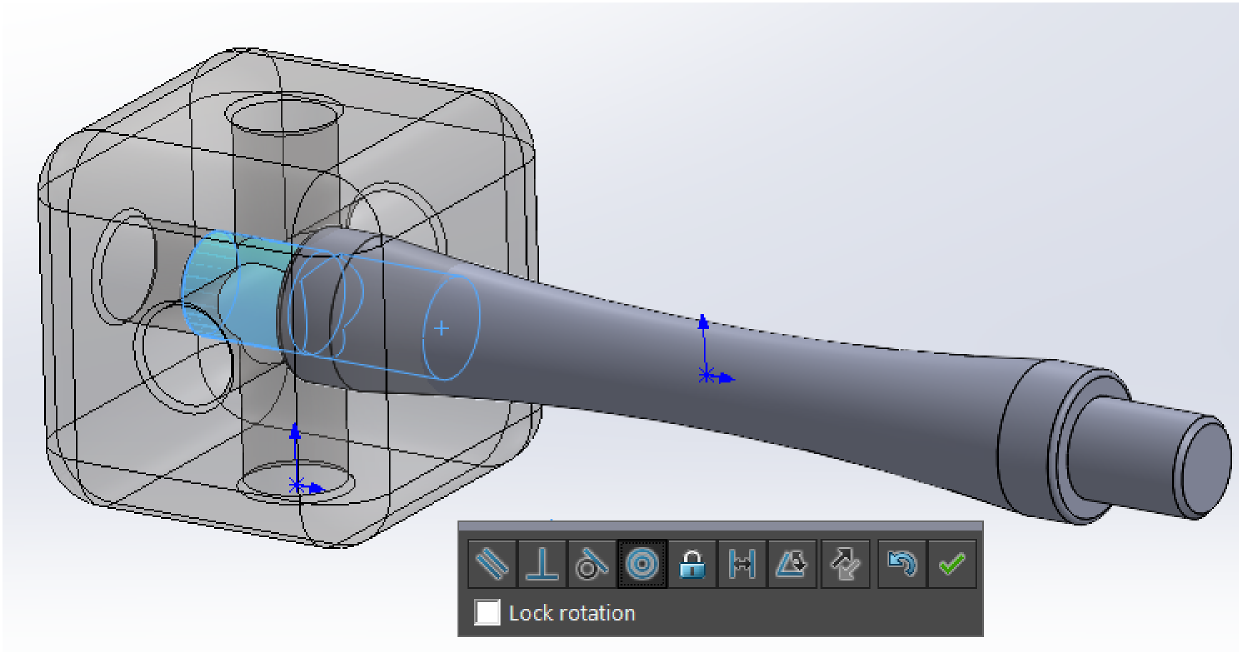
   ii. You can now select the strut and place it how we did the cube before (by clicking in the Graphics Area)
   iii. After placing it, try clicking and dragging with a left mouse click vs a right mouse click. Notice that a left mouse click allows you to translate it while a right mouse click allows you to rotate it.
   iv. We will click on the Mate tool again.
      1. Select the End cylinder of the strut



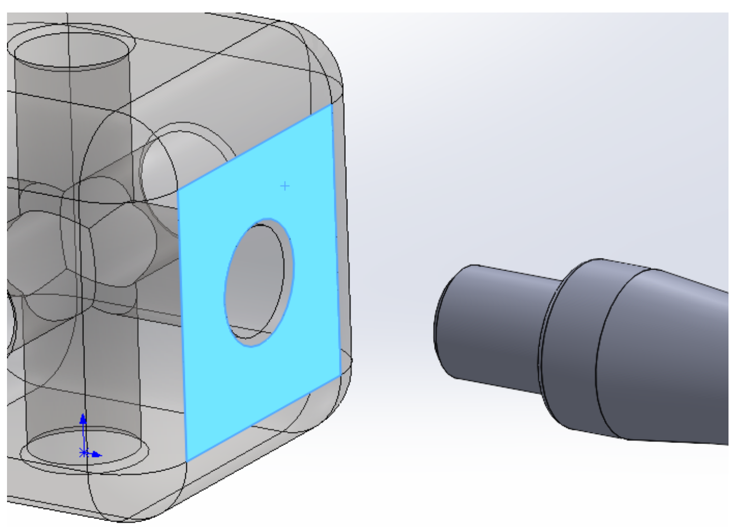
      2. Then select one of the inner cylinders of the cube



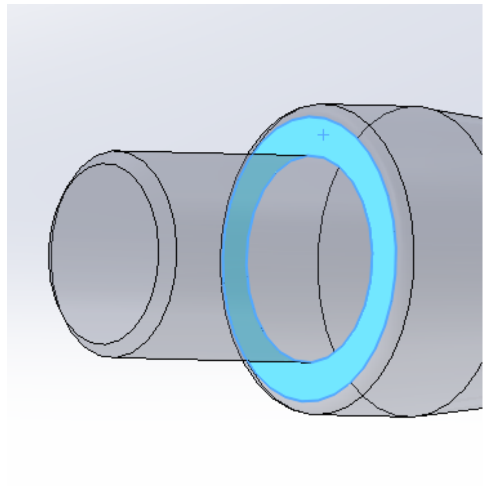
3. Notice that solidworks assumed you wanted a concentric mate, and aligned the parts correctly. We do want to click the check box for “Lock Rotation” as well though.



4. Click the green checkmark on this mate. Dont worry if the strut is inside of the cube right now.
5. If the strut is inside of the cube, left click and drag it out
6. Select the flat surface of the cube right next to the cylinder you selected before for this new mate.



7. Then select the flat surface on the strut that will sit flat on that.

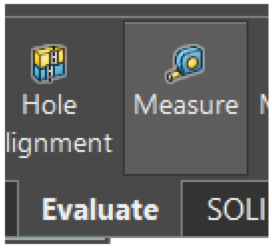


8. The Parts should mate together with a Coincident mate. If they do not, you can manually select that type of mate from the Tool Dashboard

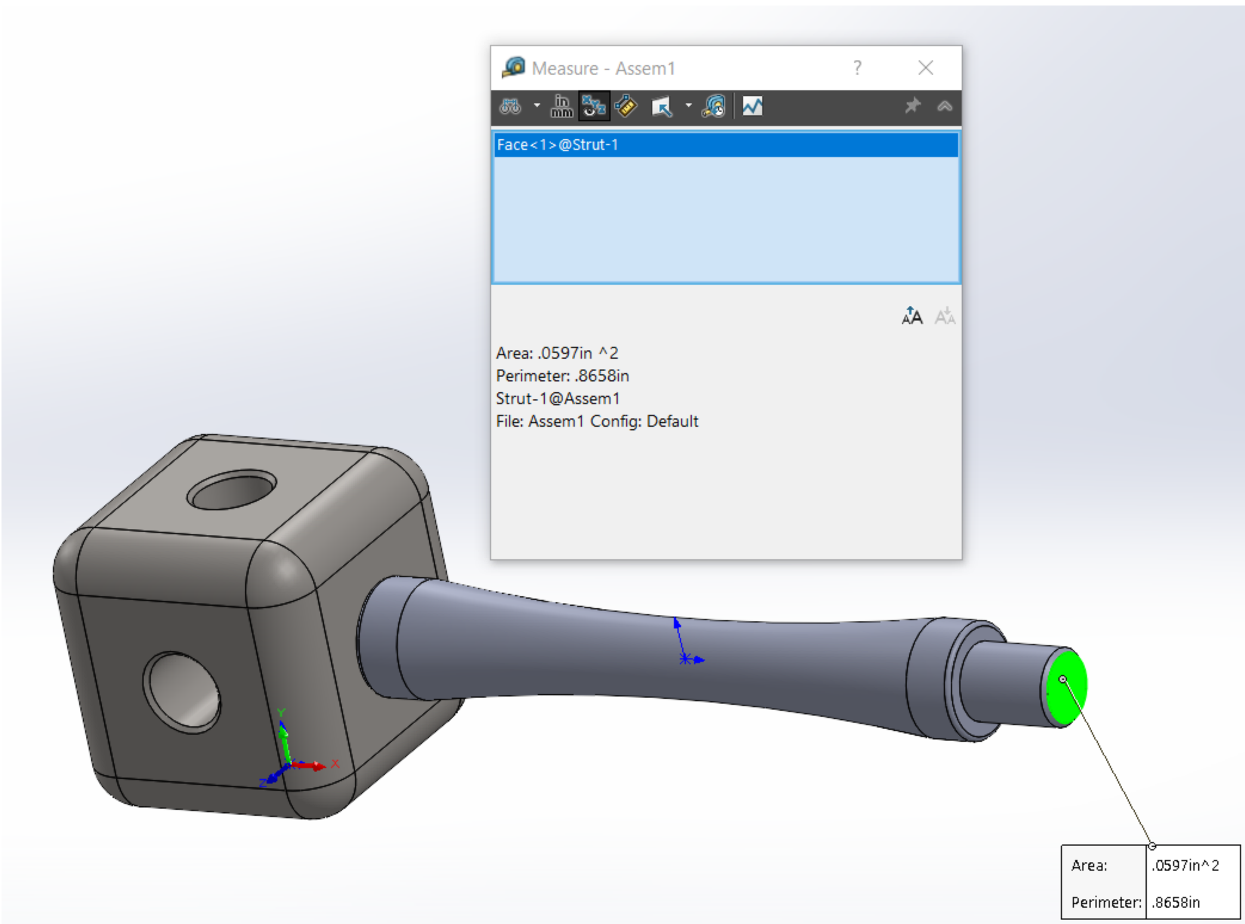
f. Add in the rest of the parts to make the cube. Note that instead of using the “Insert Components” tool, you can copy and paste parts by selecting them in the model tree, pressing Ctrl+C and then Ctrl+V. You can even copy multiple parts at once, maintaining the mates between them to save time.
    i. You now have all the skills you need to finish the cube. Feel free to finish it, or not if you think you have learned everything you wish to know.

14. Measure Tool

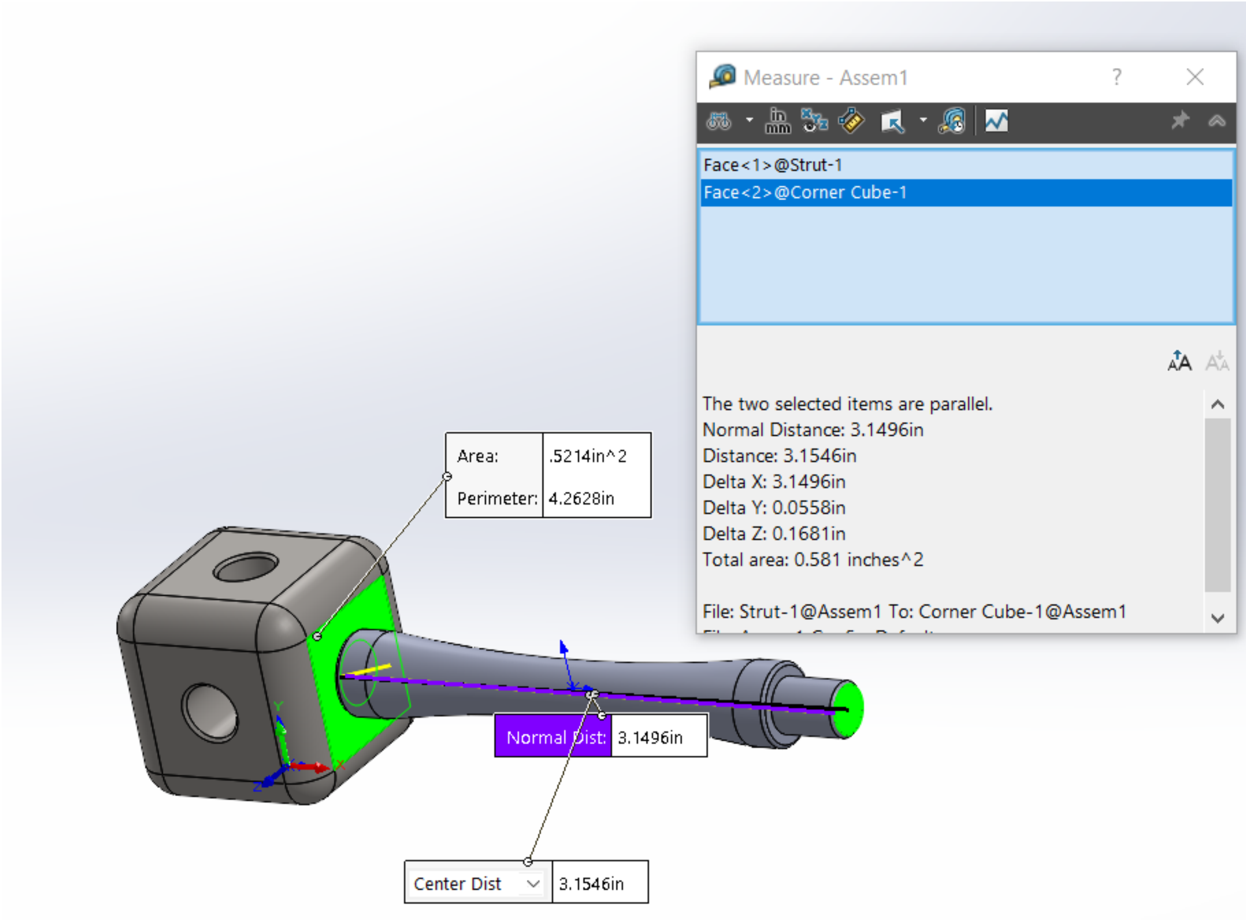
a. One of the most important tools in solidworks is the measuring tool.
    i. Go to the Evaluate tab in either an assembly or a part.
    ii. Click on the Measure tool



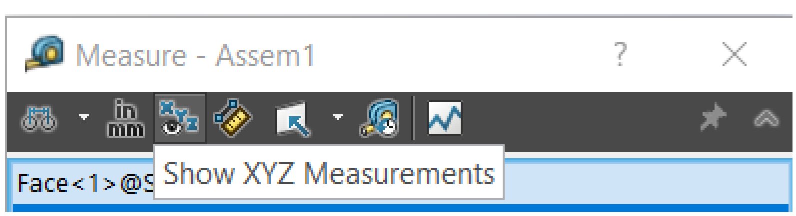
iii. Notice the blank window that shows up. Click on a surface and see what it shows.



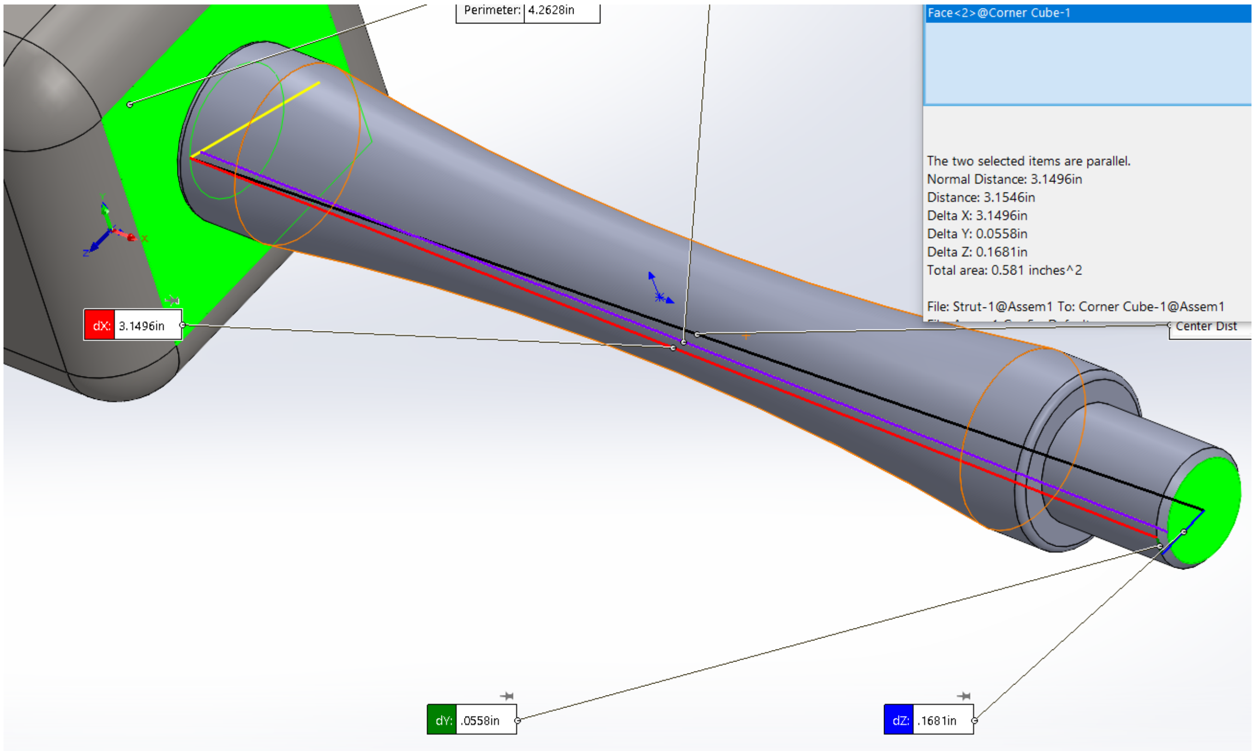
iv. Click on another parallel surface now.



v. You can select surfaces, edges, points, planes, etc. and measure between them. It is also sometimes useful to turn on the XYZ Measurements to show the amount of distance in the XYZ individually rather than the standard distance. This can be done by clicking this button in the window.



vi. Note that it now shows you XYZ distances as well as a normal distance. This isnt useful in this example but can be very useful at times.


Perimeter: 4.2628in
Face<2>@Corner Cube-1
The two selected items are parallel.
Normal Distance: 3.1496in
Distance: 3.1546in
Delta X: 3.1496in
Delta Y: 0.0558in
Delta Z: 0.1681in
Total area: 0.581 inches^2
File: Strut-1@Assem1 To: Corner Cube-1@Assem1
Center Dist
dX: 3.1496in
dY: .0558in
dZ: .1681in

vii.