

## Tulane MakerSpace: Autodesk Fusion 360 Guide

In this training, we are going to learn enough about computer-aided design (CAD) to be able to digitally create a 3D object. This file that we create can then be sent to a 3D printer, in the Tulane MakerSpace or elsewhere. This is a screenshot of a 3D print of the object we'll be creating:

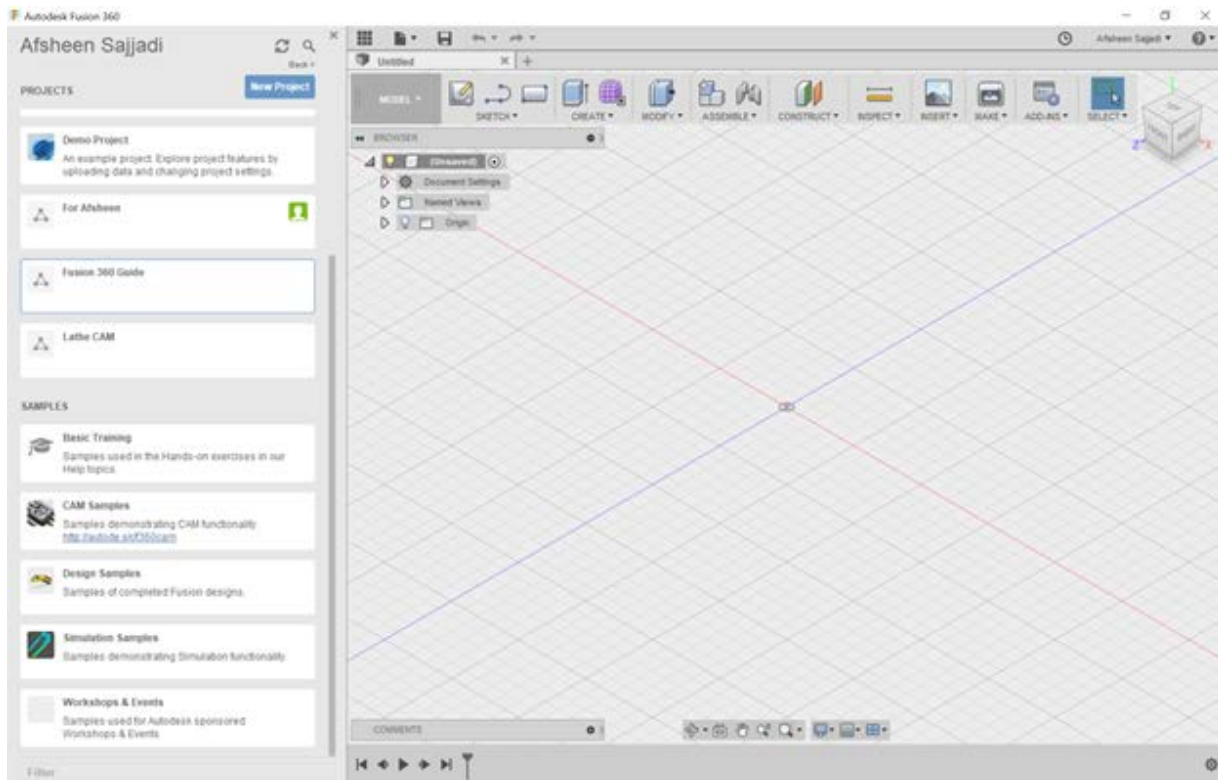


The object will be designed with the help of Fusion 360, a free 3D modeling software. If you have not downloaded Fusion 360 already, you may do so by navigating to the following web address:

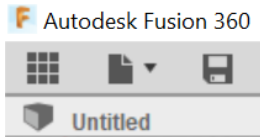
<https://www.autodesk.com/products/fusion-360/students-teachers-educators>


Since computer screens are 2D, the guide will start with navigating 3D space on a 2D screen and end with successfully making a part that is ready to be post-processed into a tangible object. It is expected that you could complete this guide on your own, but in today's training you may ask a Maker Ninja if you get stuck on any parts.

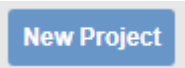
To begin, open Fusion 360. You will be greeted by a gridded plane. This is called the workspace.

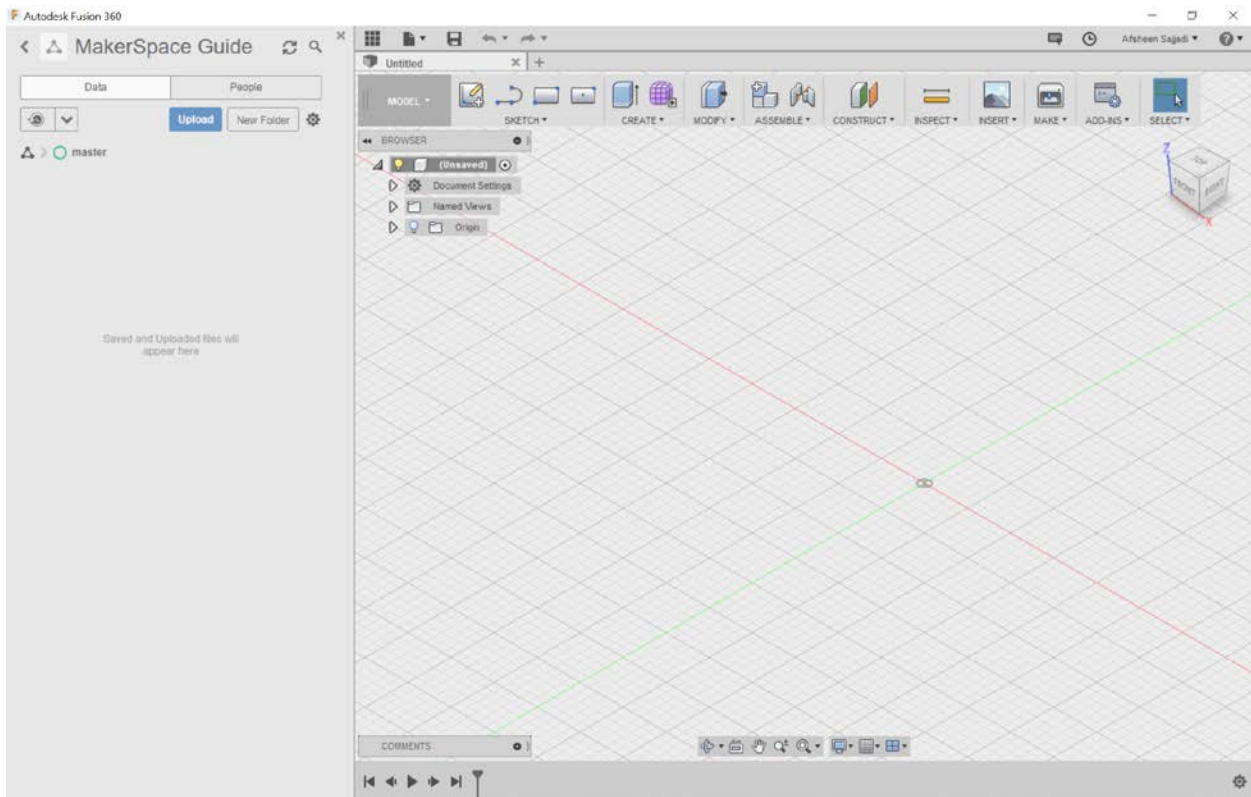
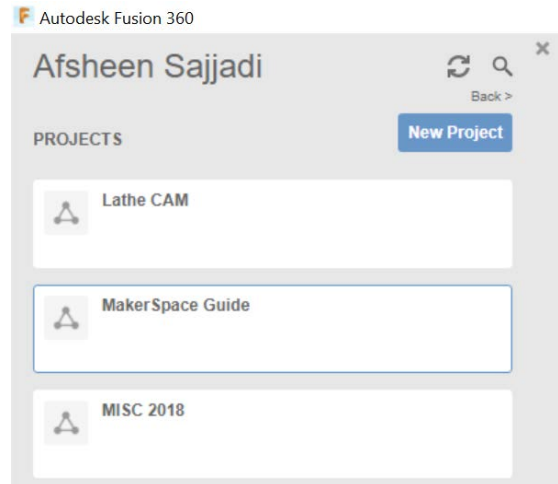


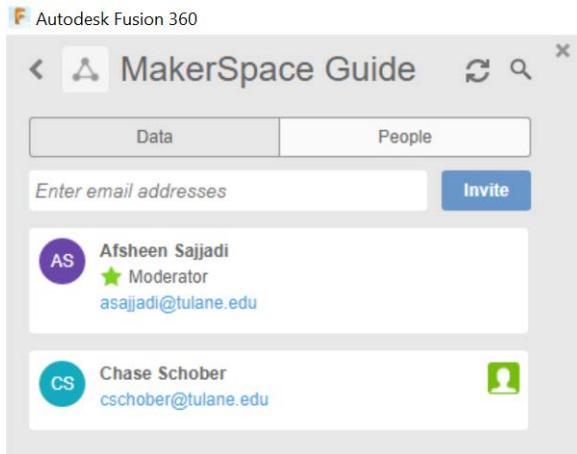
On the left, there is a sidebar called the Data Panel which contains your project library, where you can organize your different projects by folder. This sidebar also contains several additional samples and tutorials. At the upper left corner of the Data Panel, your name should be shown.




If the sidebar isn't shown, but the gridded plane is visible, click on the Data Panel button , which looks like a grid of 9 squares. This button is found in the upper left corner of the program.



Click on the blue  button to create a new folder. Name this folder "MakerSpace Guide" for this walk-through. Double click on the your newly created project folder, visible in the sidebar, to show the empty file directory. Here you can upload any files that may be relevant to the project to keep organized.

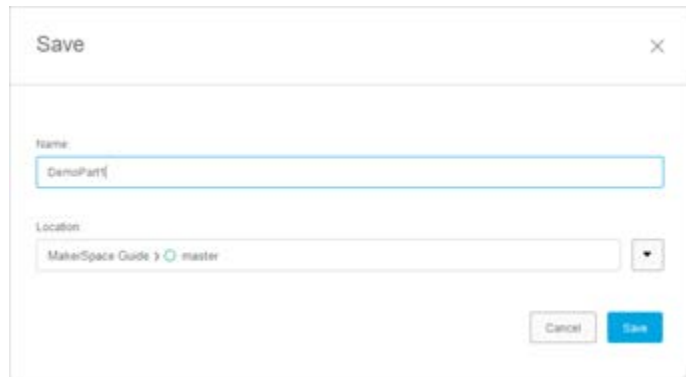




It is important to note that you can actively share your project for group collaboration efforts by clicking the “People” tab next to “Data”. Here you can invite the emails associated with your group member’s Fusion 360 accounts. Let’s close out the project sidebar/data panel so we can use the entire screen for viewing the model. Click on the data panel button  in the upper left of

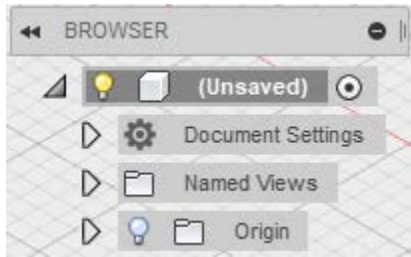
your workspace to hide the data panel.

Next, click on the file button  just to the right of the data panel button  in the upper left corner of the program. Select on save to bring up a popup. Save the current file as DemoPart1, making sure that the location shows MakerSpace Guide.



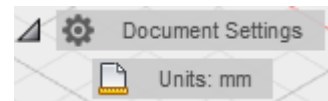
This is the project we just created. All changes made to the model are actively saved to the cloud if you are connected to the internet. Otherwise, the changes to the model will upload to the cloud next time that you open Fusion 360 when you are connected to the internet.


Before we get started with our model, let’s make sure our system of units is what we desire. For this model, we will be working in imperial (English) units, with our unit of measure being inches. Why inches? Because that is what most machinists in the America and their equipment use and it makes compatibility easier with commonly sold parts like screws.

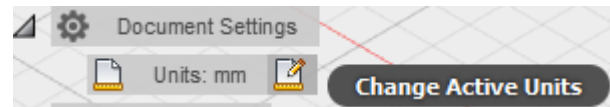



To verify our system of unit, look under the floating browser menu in the upper left corner. There will be an option for document settings. Click on the arrow to the left of this option to expand its menu.

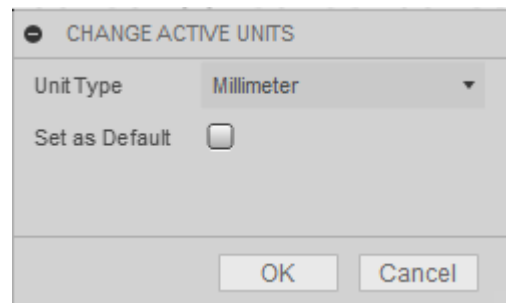
The Units option should now be shown along with the current measurement system for length. In my case, this says “mm,” but your units may be different. We want to change this to say “in” so that inches are our base unit for our measurement.







If you hover over the words “Units: mm”, a button  will appear to the right of the word.




Click on this button  and a window should appear called “Change Active Units”. Click the dropdown menu next to unit type, and select “Inch”. Click ok, and your units will now be set to inches.



Now let’s get oriented with moving around in the workspace. Use your mouse to zoom in and out by scrolling. If you have a laptop with a trackpad, you can use pinch to zoom.

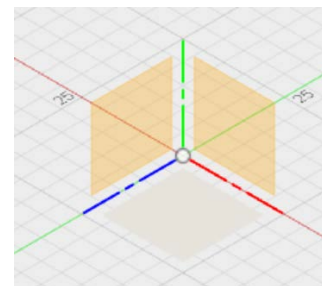
You have several options to change the view. The first option is to use the free orbit tool  located in the Navigation Bar at the bottom of your screen.  Click on the free orbit tool and then click and drag anywhere on the gridded workspace to observe the changes in the view. The pan tool  will allow you to basically slide your view from left to right as well as up and down. The pan tool  mirrors the motion of a person when they turn

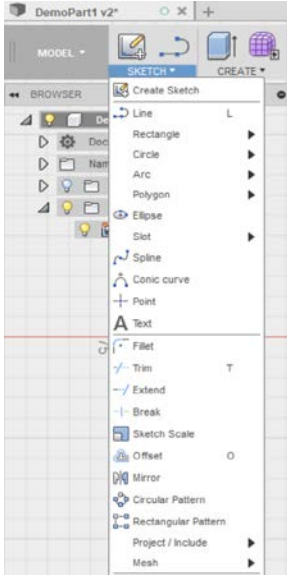
their head on their neck from left to right. The second option you may use to change your view requires the use of an external mouse with scroll wheel and middle mouse button. This is the option that most people prefer. Click and hold the middle mouse button while dragging to pan your view of the gridded work-space. You can also pivot the view if you hold the Shift key while using the middle mouse button. The ViewCube in the upper right corner allows you to quickly change your perspective to certain views, as well as freely manipulate your view. In the upper right corner, hover over the ViewCube and click the home  button to reset your view of the 3D workspace. To see other perspectives, left- click and drag along the faces, edges, or corners of the ViewCube to center your view along that plane. By clicking and dragging on the box, you can pivot your view about the origin (the dot in the middle) which indicates the x, y, and z coordinates to be zero.

Now that we can move around, let's start building our model. First, we'll create a cylinder with a height of  $\frac{1}{2}$  inch and a diameter of 2 inch. Click on the create sketch button on the leftmost part of the sketch tab on the upper toolbar.



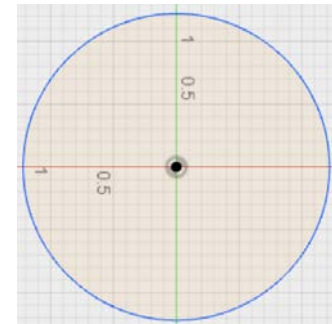
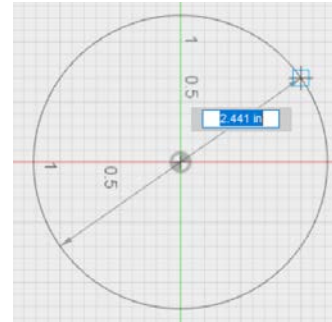
A sketch is a 2-dimensional object. Fusion 360 can place our 2-dimensional object in any of three planes, but we need to tell it which plane to work in. The three squares representing the 3 different default sketch planes should show up in the middle of the screen. Click on the lowermost sketch plane. This means we will start building our model from bottom to top. The view should automatically change to an overhead view typically denoted as "top view".






Next click on the sketch drop down menu. As you can see there are a lot of options to draw your design. Let's begin with a center-diameter circle by hovering over the circle option. Any circle type will work, but for our purposes, we want our model to be built centered around the origin point. The other options are primarily for designs where we are creating circles in reference to other shapes and objects.

Next move your cursor to the center point. Click and drag your mouse outward and see the circle expand. You also will see a highlighted box with a diameter measurement appear. To finish the circle, click anywhere on the screen to generate the circle. We want to dimension the circle to get the specific diameter we desire. Dimensioning the circle makes the diameter a parameter that can easily be adjusted in the future.

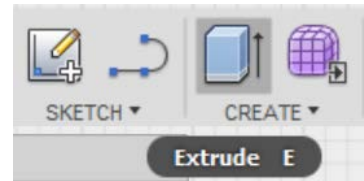


Click on the sketch drop down menu, and at the bottom, click on sketch dimension. Next, click on circle's blue border to get the dimension and then

click outside of the circle to verify the dimension. This will allow you to type a diameter of 2 inches and hit enter to set it. Please note that you can type simple math in this box to get half of a dimension or adding 3 inches, which can be useful in some cases.

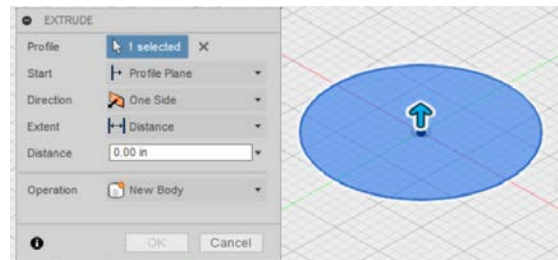
Go back to perspective view by clicking on the Home button. This is  found by hovering over the ViewCube in the upper right corner.

Next, we will take our 2D sketch and turn it into a 3D model by extruding it. When you look at the end of an open tube of toothpaste, you see a 2D circle. By squeezing the tube, you



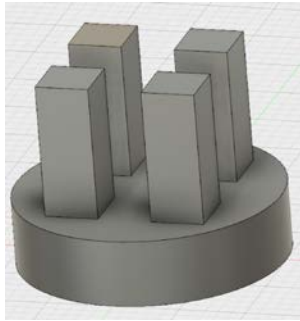
extrude the toothpaste into a 3D cylinder. Click on the Create menu and Extrude tool will be activated. Then click anywhere in the circle to select it. An arrow will appear along with the extrude menu.

The distance option in the extrude menu allows for precise control over how tall a structure is created from the 2D sketch. Instead, here we will click on the arrow and drag the mouse upwards to extrude the model until the box on the right reads 0.5 inches. Hit enter to finish the extrusion. Your circle is now a shaded gray cylinder.



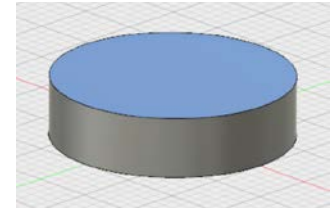
Notice how at the bottom of the screen there is a row of icons that acts as a timeline. You can move the progress bar from the right to go back along your design and hide features that you made. If something is built atop another, you can also delete features that you may no longer desire. Right-clicking on a feature allows you to change values and make various edits. For example, if you want to increase the height of the base cylinder, then right click on the first extrude feature to change it. The play pause forward backward buttons allow you to watch how your model was built up, step by step.



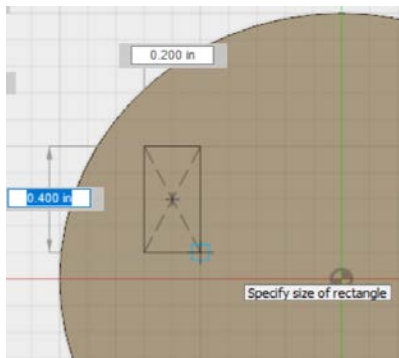


Now that we have a 3D object, we will use this cylinder as a base from which to build up the rest of our 3D model. Next, we are going to add some pillars to the cylinder.

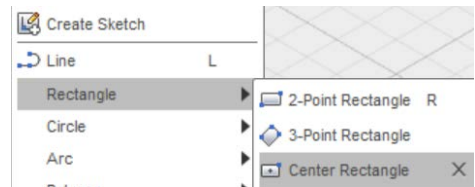
To continue, we need to select a surface from which we can continue to build up our model.



Click on the top face of our cylinder and then go to the sketch menu and select Center Rectangle under the

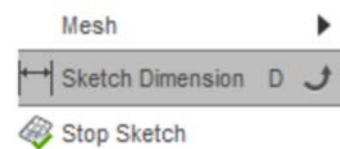


Rectangle option. The view will change to the top face.

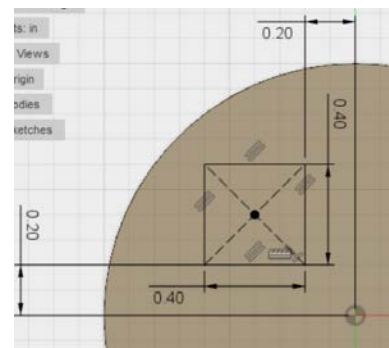


Click somewhere in the circle other than the origin to set your center point and click somewhere else to set your rectangle.

We will now use the dimension tool to make this freehand rectangle into a square in the desired location. To do this, we will move the lower left corner of the square to be 0.2 inches in the x direction and 0.2 inches from the y direction relative to the origin (center of our circle). TO accomplish this, we will create a dimension between the right side and the origin for the horizontal distance and a dimension between the bottom side and the origin for the vertical distance. First, click on the dimension tool found at the bottom of the sketch menu, then select the right side of the rectangle and select the center of the circle. This allows us to position our rectangle.



Click anywhere and set the dimension to a value of 0.2 in. Repeat this step with the bottom side of the rectangle. This locks the lower right corner relative to the origin, but the lengths of those sides can still be changed. Next, dimension just the bottom side to be 0.4 inches and then separately dimension the right side to 0.4 inches as well to make it square.

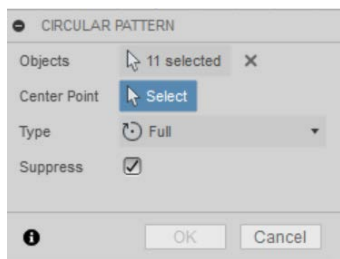




We want to make 4 identical symmetric pillars, but instead of repeating this process 4 times, we will use the circular pattern tool under the sketch menu. We'll need to tell Fusion to replicate the rectangle; to create a total of 4 rectangles; and to center the 4 rectangles evenly around the center of the circle.

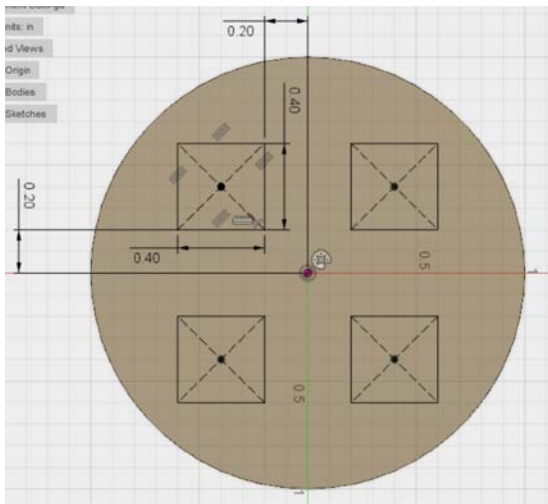



With the circular pattern menu open, click and drag diagonally from just outside the top left of the square to just outside the bottom right of the circle. This should fully encompass the square

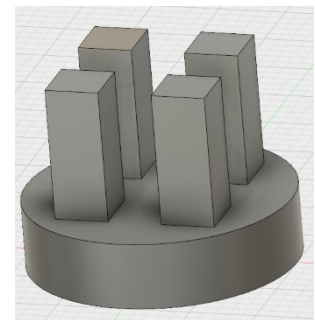


and the circular pattern menu should say "objects: 11 selected".

Next, click on the cursor next to the center point option on the circular pattern menu and then click on the origin point at the center of the 2-inch radius circle in your workspace.

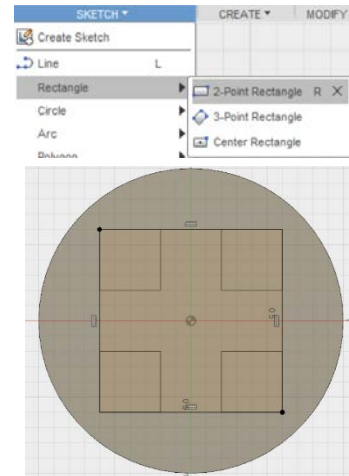


A few squares should appear rotated about the center point. Change the quantity in the circular pattern menu to 4 and hit enter. We now have 4 equivalent squares that are all equidistant from the origin. Next, click extrude and select all 4 squares to extrude them to a value of 1 inch to create the 4 pillars. Reset the perspective view with the home button  next to

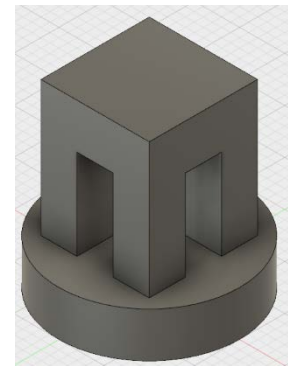


the ViewCube, and you should see 4 pillars sticking out of the cylinder.

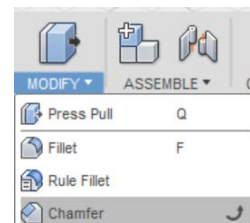
We want to create a square platform that rests on the 4 pillars. To continue building our model click on the top of any of the 4 pillars to choose our sketch plane. Go to the sketch menu and select the 2-point rectangle option. Click on the outermost corner of the square and then click on the outermost corner of the square diagonal to the original square. A large square encompassing the 4 smaller squares can be seen.



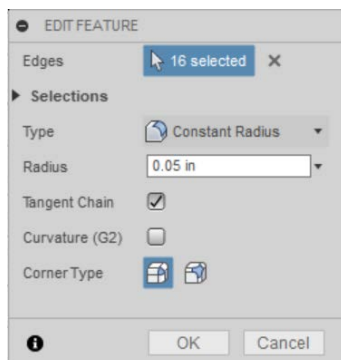
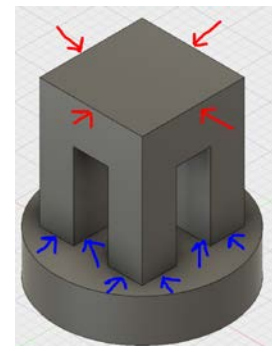
Now we want to give this platform a height of a half inch. Since our 2D sketch plane is based off the top of one of the pillars, the larger square is divided into 2 selectable parts: the original and the rest. When extruding you must select both. Shift click on each selectable part to have all selected at the same time, then hit extrude. Let's extrude this large square upwards to a height of 0.5 inches.



We have a lot of sharp edges in our model. We will round off these sharp edges using fillets and chamfers. A fillet (pronounced "fill-it") is a rounded edge. A chamfer is a flat angled edge. Also, it's aesthetically pleasing to add these features. Click on the Chamfer tool under the modify menu.

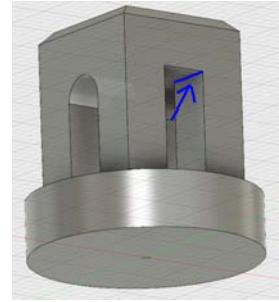


With the chamfer menu now open, click on the top four edges of the model. Each edge should turn blue as you select them. Type a value of 0.1 inches for the chamfer distance and hit enter.



Next, click Fillet under the modify menu. With the fillet menu now open, select all 16 bottom edges of the pillars one by one. You may need to pivot your view as we learned earlier in the guide. With constant radius selected in the fillet menu, use a radius of 0.05 inches and hit enter.

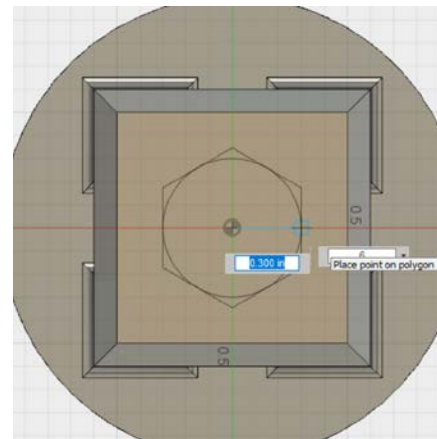
Let's turn the pillars into arches by using the fillet tool. Click fillet under the modify menu and then select the 8 inner edges of the top of the pillars. Use a fillet radius of 0.2 inches to get the desired arch effect. Finally, let's add a fillet to the top edge of the base cylinder using a radius of 0.05 inches.



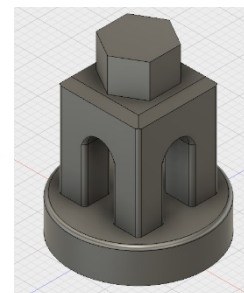
If you make a mistake, you can use the undo button in the upper left corner. This will undo features, sketches, and dimension changes, but not necessarily an accidental edge selection. In those instances, if you are in the fillet or chamfer menu, you can just click the edge again to deselect it.



We now have a very nice platform, but nothing that sits on it. Click on the top of the platform. Let's add a circumscribed polygon from under the sketch menu. Click on the center of our square to place our polygon's center point. Without clicking anywhere in your workspace, click on the number box that does not have units. If you hover over this box, the label "Edge Number" will appear. Use the number 6 to use 6 sides to make the model a hexagon. Hover along either the x or y axis. I

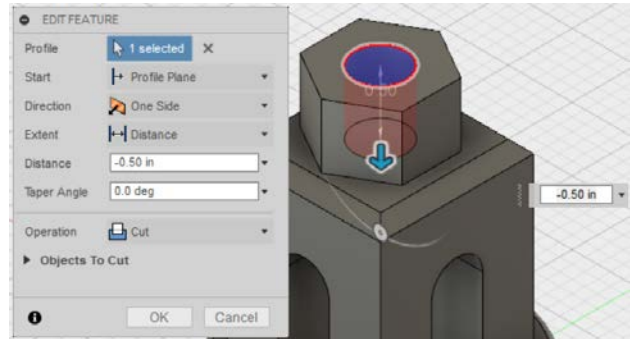


chose the x axis (in red). Click along this axis to set the hexagon position. Now we need to set the size of the hexagon by dimensioning it. We will dimension the hexagon from the center to the polygon's edge. Click on the dimension tool underneath the sketch menu. Click on the center point and then click on the right edge. Type a value of 0.4 inches to set its size. Extrude this hexagon to a value of 0.5 inches using the extrude tool under the create menu.

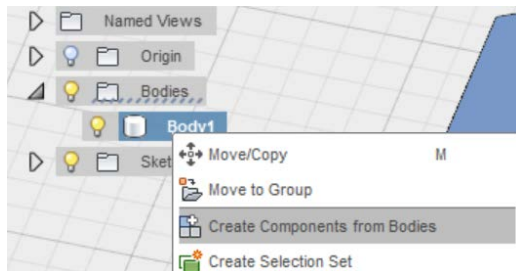
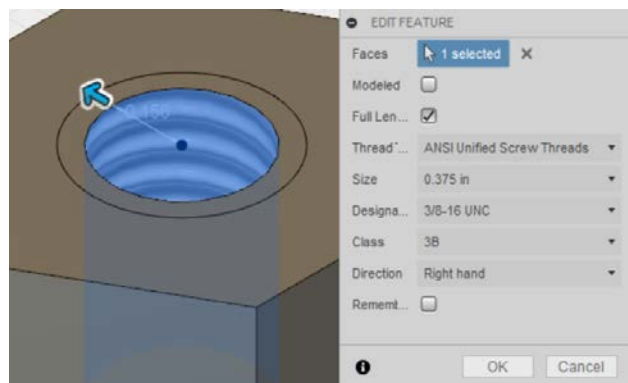


We are now going to add a hole with threads in this hexagon essentially making a nut for another component to screw into.

Sketch a center-diameter circle on the top face of the hexagon with a diameter of 0.375 inches (3/8 of an inch). This value will change automatically when we add the thread. We will use the press pull tool to make this hole. The press pull tool can be used for both extruding out as well as cutting into a body. Select our circle then click on Press Pull under the modify menu. Click and drag down on the arrow until you get to a depth of -0.5 inches and hit enter. This length is for a threaded custom component and is not based on any length standards.



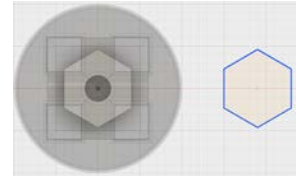
Now go to the thread option under the create menu and select the interior of our hole. Choose ANSI Unified Screw Threads under Thread Type in the Thread menu. Use a size of 0.375 for the size. We will use 3/8-16 UNC for Designation. Hit enter and you can see the generated thread.



This part of our design is now complete. In order to allow this part to interface with others, we will convert it from a body to a component. Go to the floating browser menu in the upper left side and right click on Body 1. Click on Create Components from Bodies.

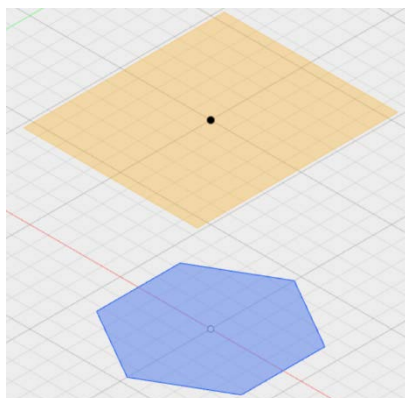
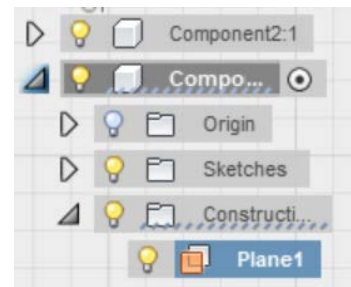
The body option will disappear underneath a component option that appears. We now have an isolated part that can be moved and manipulated without the internal references of the part changing.

We will now make a second part that will interface with our component via the threaded hexagon. Click on New Component under the create menu and then select our part as the parent. Then click on create sketch and let's use XY plane. The view should change to the top plane. Let's create a hexagon with the same dimensions of the hexagon in our first component so that they fit together seamlessly. Put the center point away from our first component, which is now opaque so it is easier to work with.

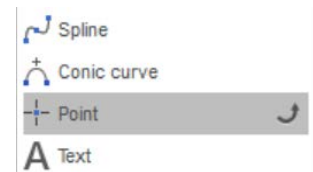


The hexagon will become a 1" tall tapered spire that has a threaded bottom component, to couple to our first model. There are two primary ways of making a tapered element. The first way is simply using the push pull tool with a taper angle, in this case of about -22 degrees (negative because it's tapering smaller as we rise away from the plane). The second way, which is slightly more parametric, involves creating an offset plane at the 1-inch height and using "loft" between the hexagon and a point on the offset plane. In this tutorial, we will use the second way.

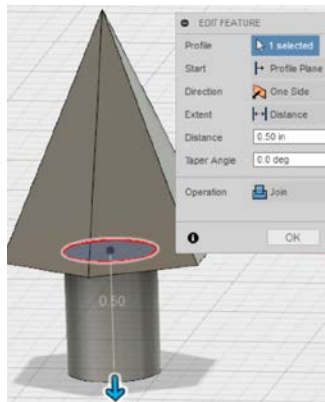
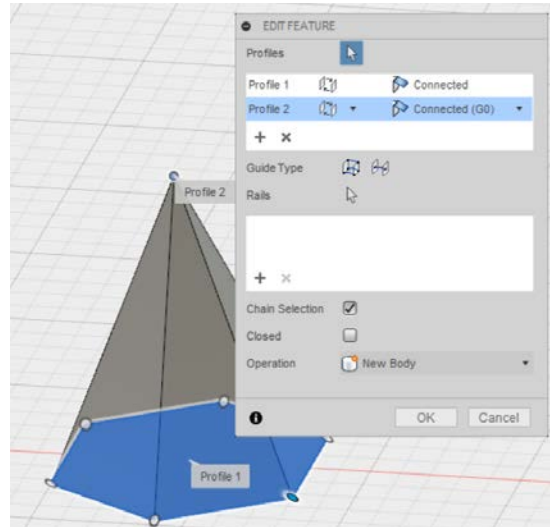
Under the construct tab, click on offset plane. To select a plane, go under the origin dropdown menu under the floating browser menu in the upper left. Find and select the XZ plane and set an offset distance of 1 inch. There should now be a construction option with the new plane under the second component in the browser. If you click on the home button, you can see this new plane floating an inch above your 2D hexagon. Select this new plane and click create a



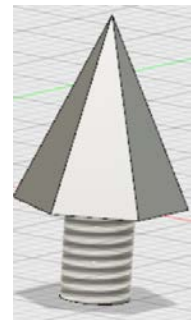
new sketch. Under the sketch menu choose Point and center the point on the center of our hexagon. Now we have a point floating 1 inch above our hexagon sketch.



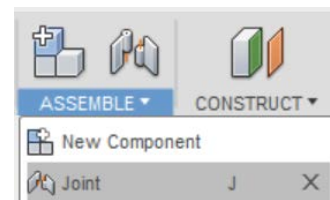
Next, click on Loft under the Create tab. The view will become isometric again. Select our hexagon which becomes Profile 1 and then select our floating point which become Profile 2. A tapered hexagonal cone will immediately be generated. In the popup menu, ensure the Operation is “New Body”. Hit enter to finish the loft.



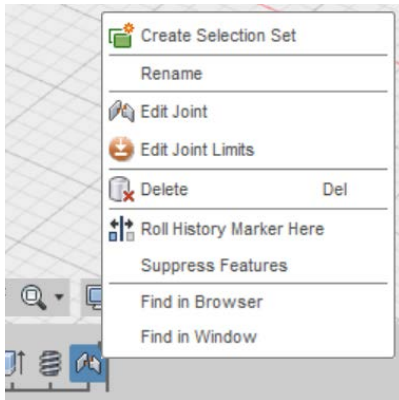
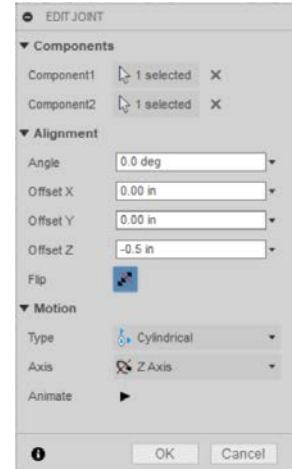
Now pivot to the bottom of our cone to select the bottom face as our new sketch plane. Sketch a 0.375-inch diameter circle using the center of the hexagon. Use the push pull tool to extrude a 0.5-inch-long cylinder. Next, click on Thread underneath the Create tab and use the same parameters as before: ANSI Unified Screw Threads, size 0.375, and 3/8-16 UNC designation. Our second component is now complete.



We have 2 components, which can now be 3d printed separately and assembled if desired. However, we may want to assemble the pieces digitally for several reasons: to see what it looks like, to animate it, or to perform various simulations such as performance under static stresses. To do this, we will create a joint. Click on Joint under the Assemble tab to get started.

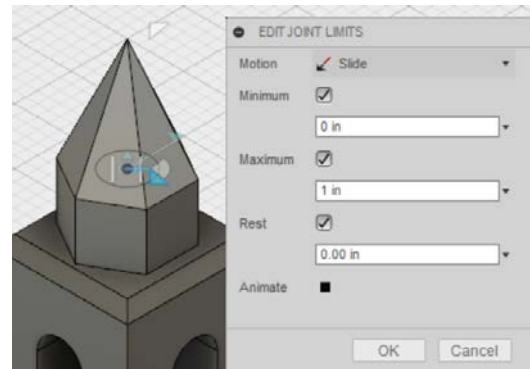


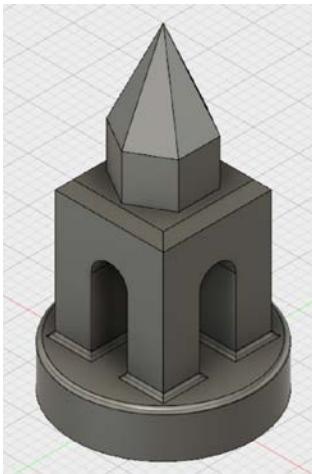
First, select the threaded part of our second piece as our “Component 1” and the center of the top face of the threaded hole on the first piece as our “Component 2”. The reason for this switch in component names is that “component 1” will move to “component 2” and we want our second piece to move while our first piece is rigid. In the Joint popup menu, set an Offset Z value of -0.5 inches under the alignment section and choose cylindrical for the type underneath the motion section. Ensure the Z axis is chosen. If you click animate, you will see the second part move in and out of the first part, but there are no limits to prevent them from overlapping each other. Hit enter to keep these settings.



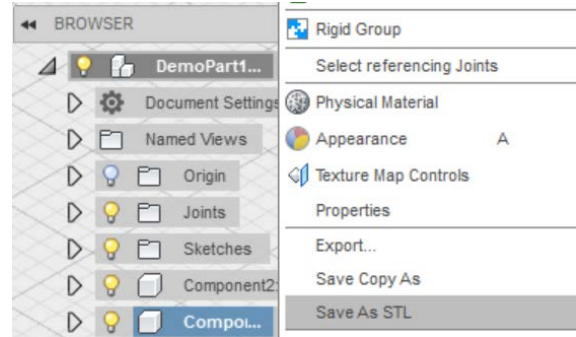
To add joint limits, right click on the joint feature in the timeline on the bottom bar of the screen and select Edit Joint Limits for a popup menu to appear. Under motion choose slide to put limits on that aspect. Select rest, and use a value of 0 inches to have the resting position be when the parts are fully interfaced. Similarly,

use 0 inch for the minimum and 1 inch for the maximum to have a 1 inch range of motion for the joint. If you click animate, you can see the limits in action.

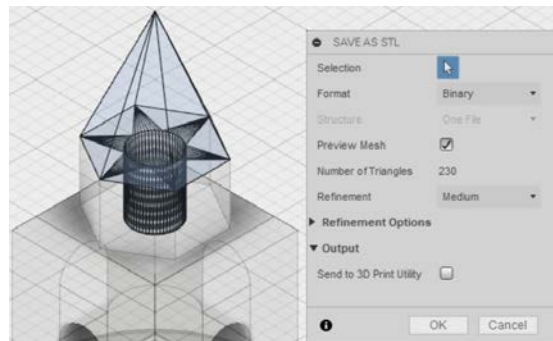




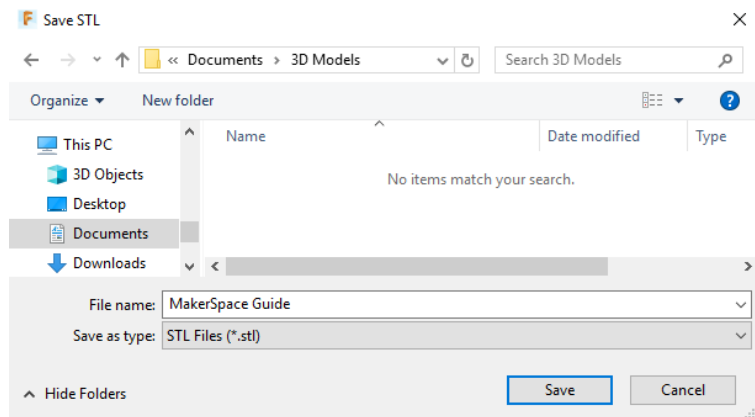
You now have a completed part ready for simulation, animation, or just presentation. To export your parts as a 3D printable file, right click on the desired component or body from among the list of items on the left side of the screen, and left-click select Save as STL.



This will save your part as a mesh file which is composed of many triangles as you can see by clicking Preview Mesh. For our purposes the default settings are fine. Uncheck "Send to 3D Print Utility" under the "Output" tab if it is checked. We do not want to send the file to another program, rather, we want to save it as a 3D printable file. Click ok, and a window will pop up asking you where to save the file.



Using this save window, navigate to the location that you want to save it. In my case, I will save it to a folder called 3D Models in My Documents to keep my 3D printable files from Fusion 360 organized. Once you have picked a location, click ok and the 3D printable file should immediately



save to the location you chose. Notice that 3D printable files have a file extension of .STL similar to how Microsoft Word documents use a file extension of .docx. This .STL file can be now be moved to a USB and imported into Cura in the MakerSpace to prepare it for 3D printing.