
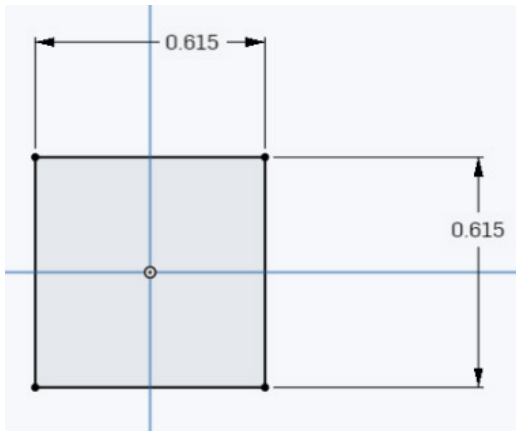



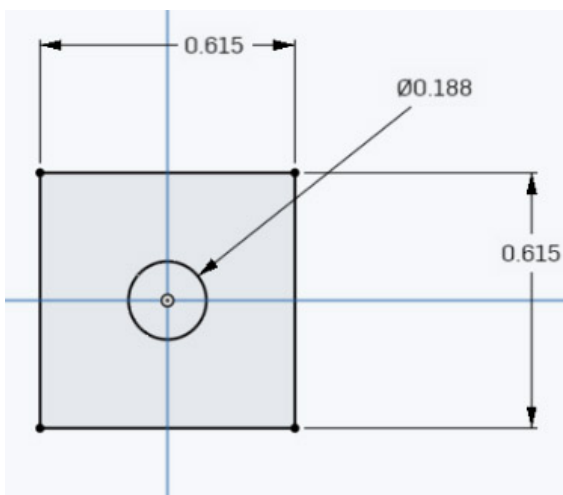
Onshape Lesson 9a


In this lesson you will create a nut to work with the Acme threaded rod you created in Lesson 9.

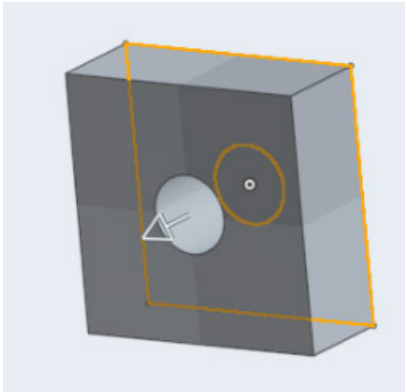
1. Open Lesson 9 in Onshape. Then click on the plus button near bottom left of screen and select **Create Part Studio**. Then rename this part studio "Nut."
2. Create a new sketch on the top plane and set the view normal to the sketch plane.
3. Use the Center point rectangle tool  to draw a rectangle centered on the top plane. Use the Dimension tool to make the sides of the rectangle 0.615 inches.



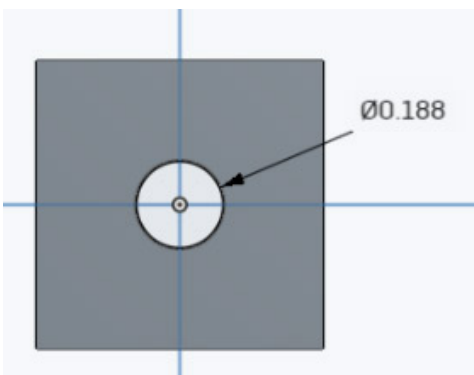
4. Use the Circle tool  to draw a circle centered on the top plane. Use the Dimension tool to set the circle diameter to 0.1875 inches. Close the sketch.



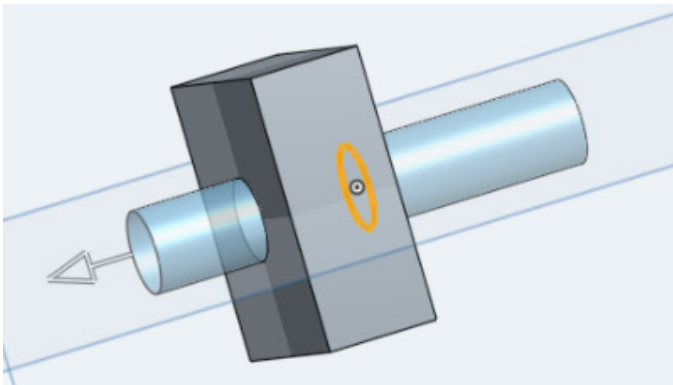
5. Use the Extrude tool  to extrude the nut to a thickness of 0.25 inches. Close the extrude.




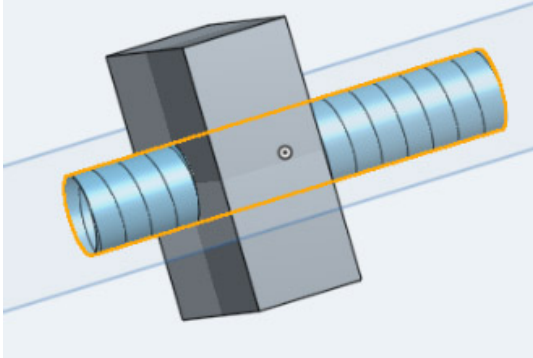
6. Create a new sketch on the top plane and set the view normal to the sketch plane. Use the Circle tool to draw a circle centered on the top plane. Use the Dimension tool to set the diameter of the circle to 0.1875 inches. Close the sketch.




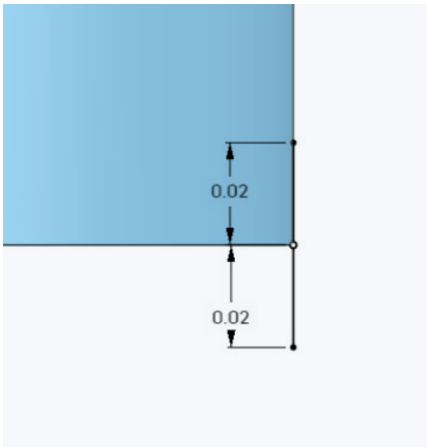
7. Use the Extrude tool to extrude the circle created in step 6. In the extrude box set the extrude to **Surface, symmetric** and **1 inch** depth. Close the extrude.



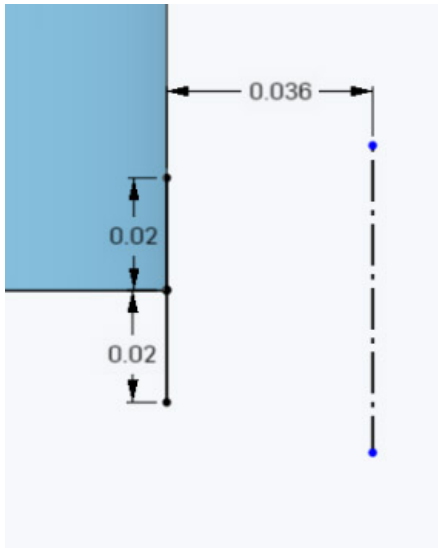
8. Use the Helix tool  to apply a helix to the surface extrude you made in step 7. The helix should have 16 revolutions (16 threads per inch – the extrude is one inch long). Close the helix.



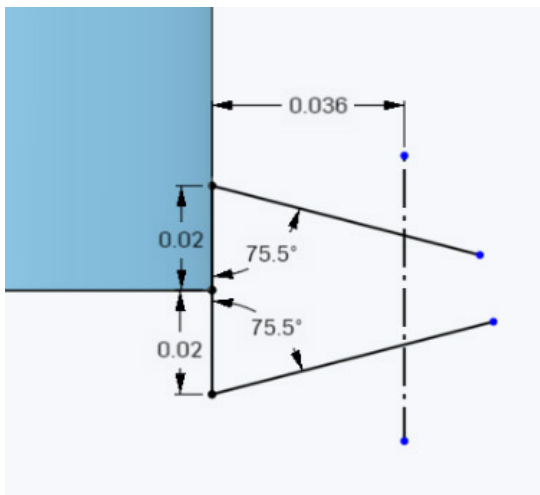
9. Create a new sketch on the front plane and make the view normal to the sketch plane. Then zoom in to the right-bottom corner of the cylindrical surface. Select the Use tool . Click on the corner of the cylinder to set a point. Use the Line tool to draw two vertical lines as seen below. Use the dimension tool to set the length of each line to 0.01965 inches.




10. Draw a vertical construction line as seen below. Use the Dimension tool to position the line 0.0362 inches to the right of the cylinder.

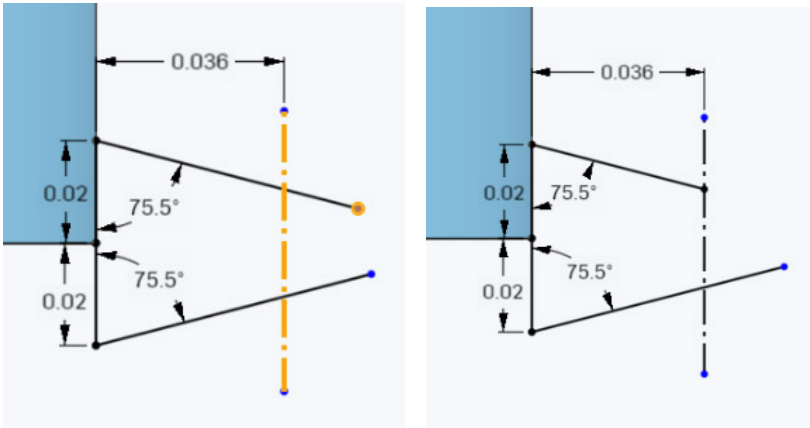


11. Draw two lines as seen below and use the Dimension tool to set their angles to 75.5 degrees.

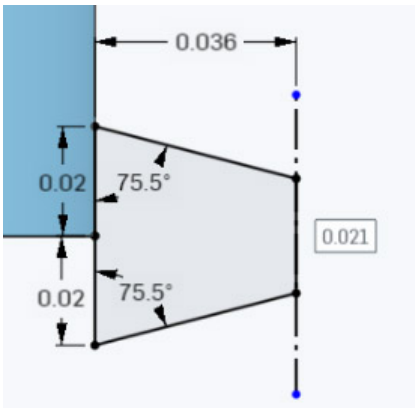



12. Make sure there are no tools selected. Click on the end point of one of the lines to select it. Then click on the construction line to select it as well (see image on left below). Now click on

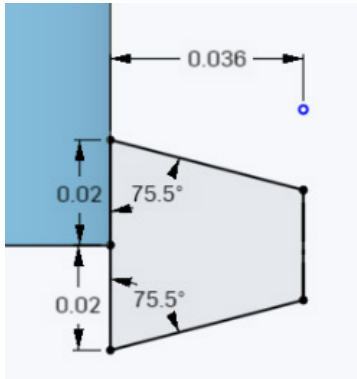
the Coincident tool  to connect the end of the line to the construction line as seen in image on the right below. Repeat procedure for the other 75.5 degree line.




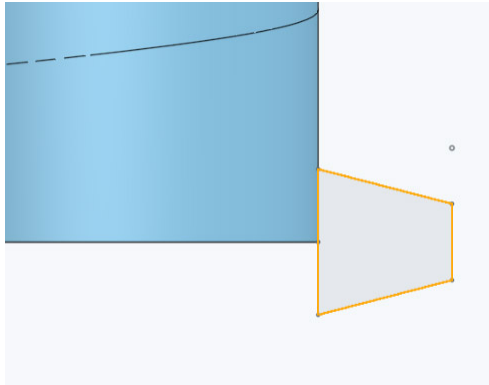
13. Use the line tool to draw a line connecting the two 75.5 degree lines as seen below.



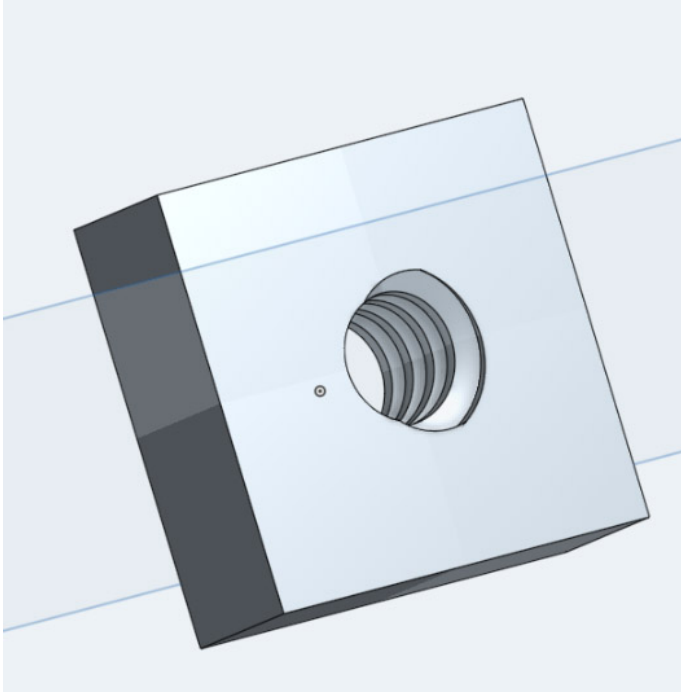
14. Use the Trim tool  to remove the remains of the construction line. You have now completed the thread-cutting profile for the nut. Close the sketch.



15. Select the Sweep tool . In the sweep box, select **Remove**. Click in the “Faces and sketch regions to sweep” slot. Then click on the thread-cutting profile you finished in step 14 to select it as the object to sweep. In the Sweep box click into the sweep path slot then click on the helix to select it as the sweep path. You have just completed the thread cutting process. Close the sweep.



16. In order to see the threads cut in the nut, you will need to hide two elements, the cylinder surface and the helix. Position the mouse over Surface1 in the list on the left side of the window and click on its eye to make it hidden. Do the same for the helix. Then you can see the nut as pictured below.



Congratulations! You have finished this lesson. In the next lesson you will learn how to make an assembly of the threaded rod and nut.