

# Tutorial OnShape Revolve Solid Model

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Brighton High School

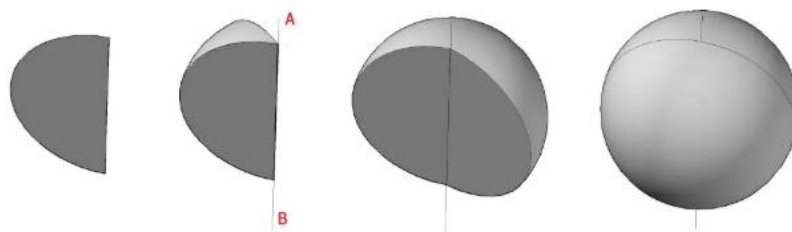
Brighton, MI

1. Navigate to [brightonk12.onshape.com](http://brightonk12.onshape.com) > Sign in
2. Navigate to Class Folder > Copy Assignment > Move Assignment to Personal Student Folder

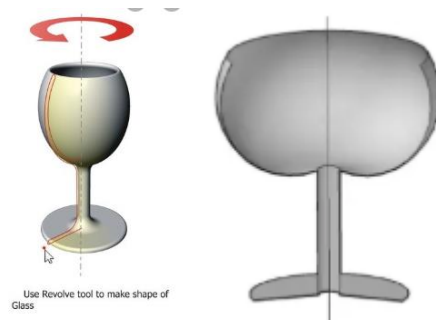
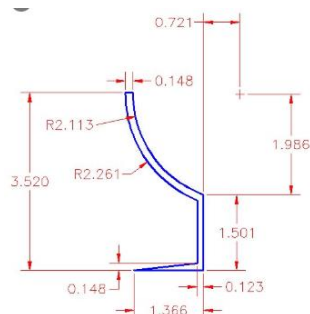
**Revolve Modeling:** allows the user to create a profile and spin that profile around an axis line (As opposed to extruding where the user extrudes in a direct linear direction (up/down, left/right, or forward/backward). Revolve parts are used for symmetrically cylindrical parts (i.e cups, pots, crazy straws, tires, etc.)

Initial revolve part needs 2 things

1. Axis Line
2. Profile drawn only on one side of the axis line. The reason for this is if there was part of the profile on both sides when the part is spun around the axis the profiles would intersect each other after reaching the 180 degree mark. Users are allowed to touch the axis line, just NOT go over it. A number of revolved parts will have a hole in the middle to either allow fluid to pass through/hold (i.e. glass, cup, etc.) or to have a mechanical shaft to transmit power from point A to B (i.e. drive axel in a car)
  - a. Profile that touches the axis line creates solid material
  - b. Profile set off the axis line (see below) creates a hole in the center
  - c. Or a combination of both



Profile Touching Axis Line



Profile NOT Touching Axis Line

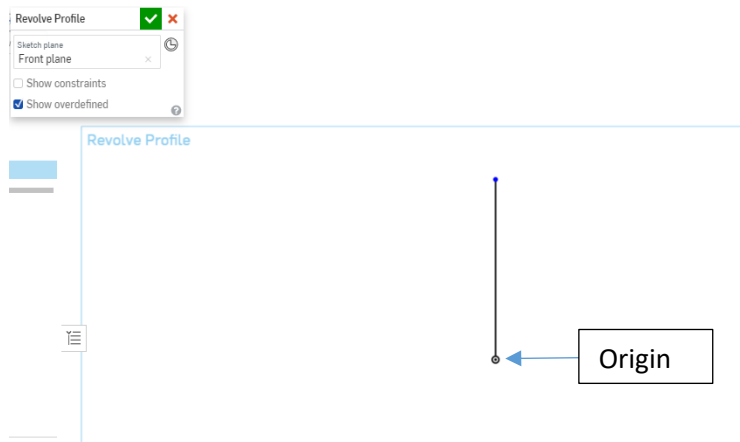
## 1. Creating Profile and Axis Line

- a. Select Sketch Icon > Select Front Datum (Plane) > Press n key (rotates screen normal (perpendicular) to viewer) > Press p key (turns datums (planes) OFF) > Rename Sketch1 to Revolve Profile

- b. Axis Line

It is recommended for any revolve part that is being created the user should draw the axis line first so there is reference to where the part is being spun around. NOTE: this is not required, but HIGHLY RECOMMENDED

- i. Select the line tool and draw the following line (NOTE: the length of the axis line DOES NOT matter; can be any length, the current line is 3" long if needed for reference)

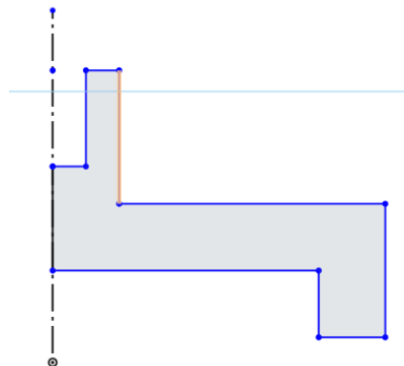


- ii. Convert the line to a construction line > Select the line > Select construction line icon



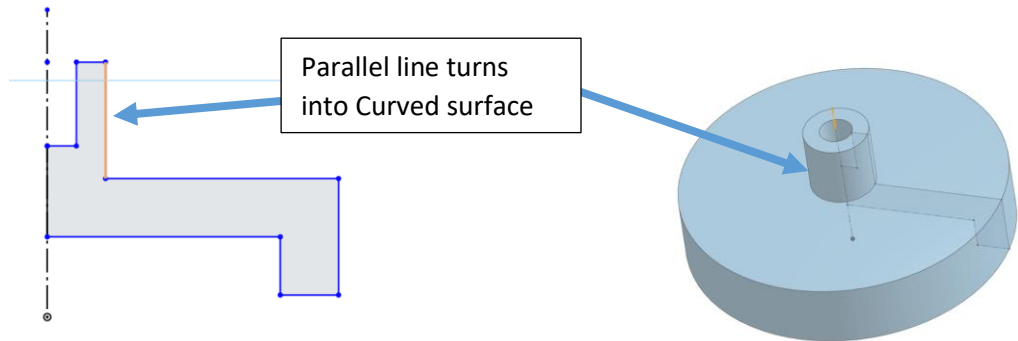
- c. Profile

- i. Draw the following profile > DO NOT set any dimensions



## ii. Setting Dimensions

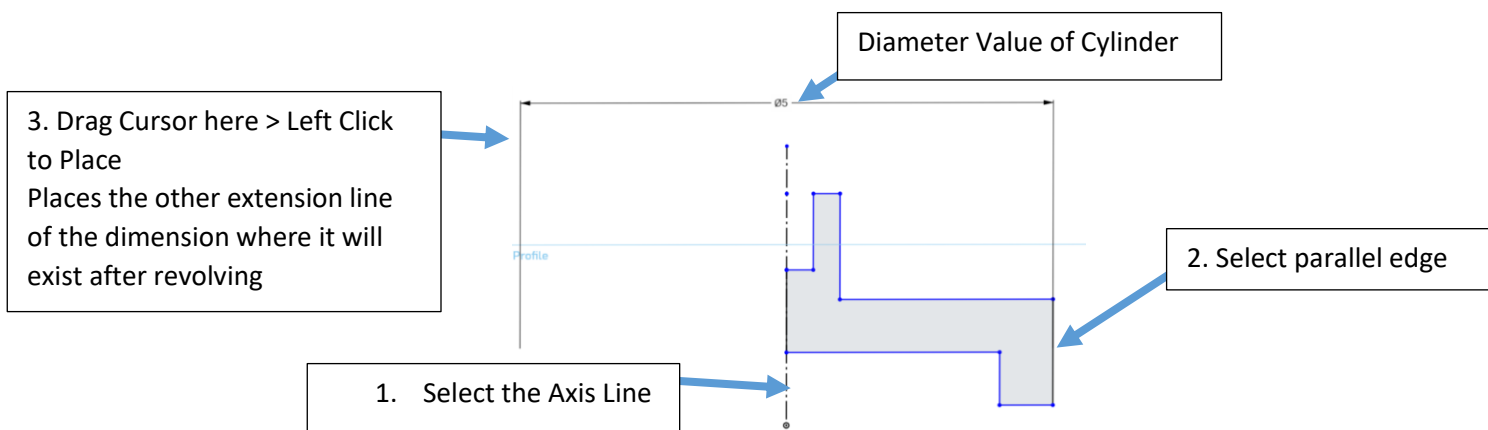
When looking at the profile user needs to recognize some of the edges shown will not be sharp corner edges but will form the curved surface of the cylinder. Rule thumb is if a line is parallel to the axis line then that line will become a curved surface



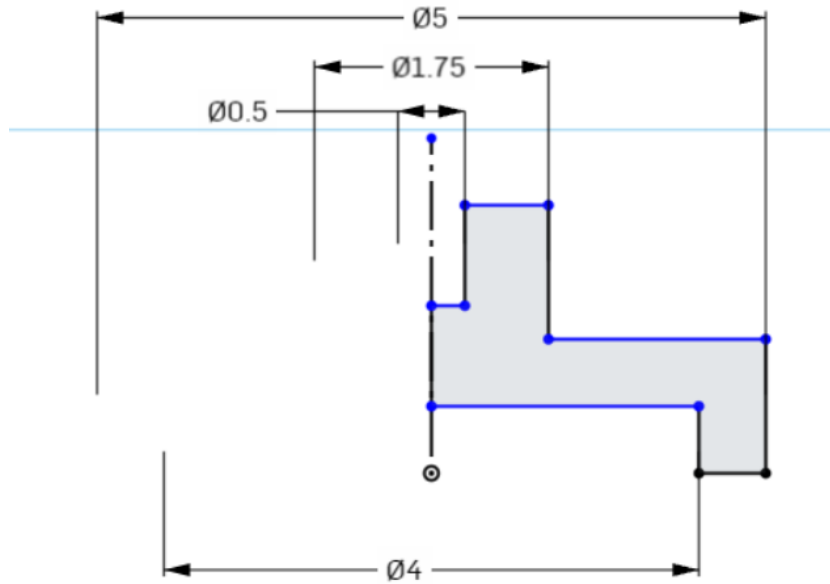
NOTE: when looking at drawing these features will be dimensioned with a diameter of the cylinder and linear (length/height of cylinder) dimension, so the user needs to be careful when setting dimensions

### 1. Setting Diameter Dimensions

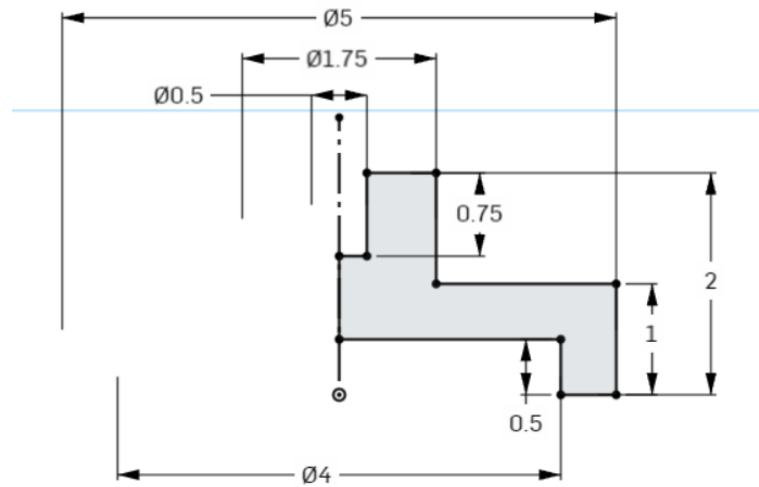
- a. Select the Dimension Tool
- b. Left Click on Axis Line > Left Click Once on Parallel Edge (maybe a point; beginners stick to edges) > Place Cursor on the side of the axis line where the profile does not drawn > Left Click to Place > Extension line will be created where the other side of the cylinder will appear after revolving



- c. Set the following Dimensions > Change Values as shown  
Recommended to set smaller diameters first then move out to larger diameters.



- d. Linear Dimensions > Set the Linear Dimensions as follows



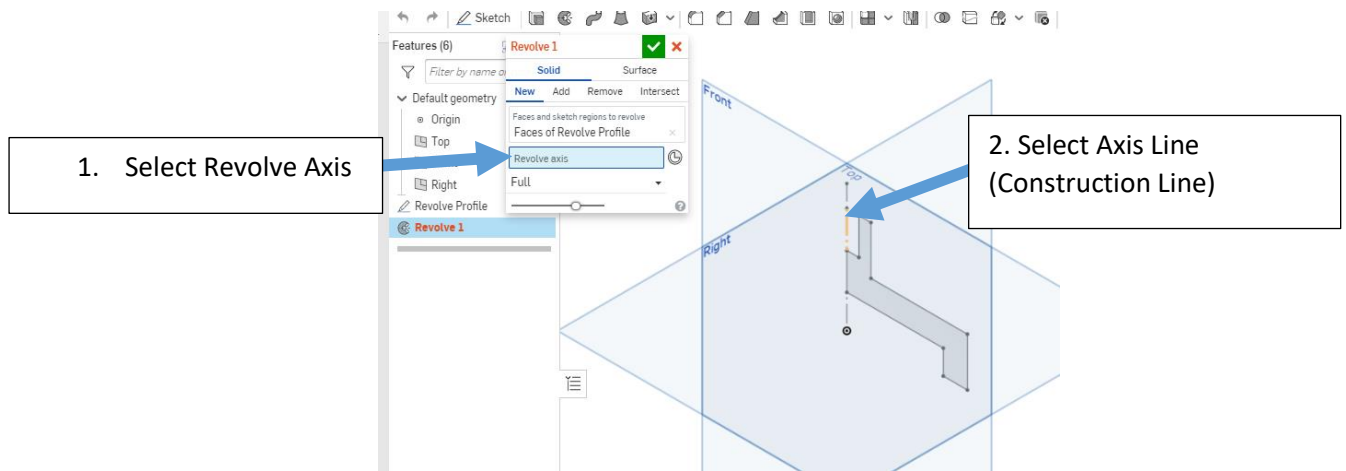
- iii. Green Check to Accept the Sketch

d. Creating the Revolve

- i. Press Shift + 7 to rotate to Isometric
- ii. Select Revolve Profile from Model Tree or Workspace > Select Revolve Icon

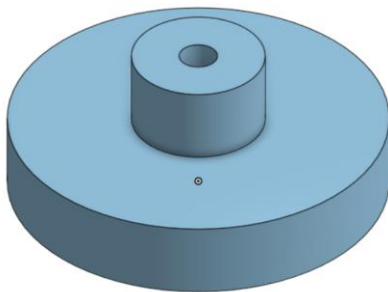


- iii. Select Revolve Axis from Revolve Properties > Select Axis Line of part (NOTE: An edge of the profile can be an axis line, be sure to select the construction line that represents the axis line)

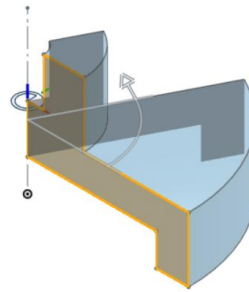


- iv. Keep Revolve Type: Full.

NOTE: All parts that this class is working on will be revolved 360 degrees  
Parts are typically spun 360 Degrees (Full), but can be spun at any number of degrees. Also, Revolves can subtract material, this more advanced and is not needed as often as adding material)



Completed Part



Example of Partial Rotation

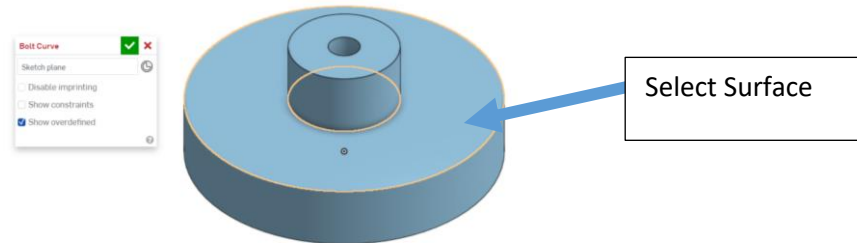
- v. Green Check to accept Revolve

## 2. Patterning Holes

A number revolved parts will have a series of holes to be used as attachment or fastener holes. These holes need to be equally spaced to help dispense rotational load equally across each fastener within the hole to one fastener does not accept more mechanical load than another and snap off during operation.

To complete this we will only draw one hole then allow the computer to pattern the number of degrees we desire between the total number of iterations (holes) that is desired. This allows for the user to quickly change the properties (diameter, location, total number, degrees between, etc.) without have to set and change a large number of dimensions.

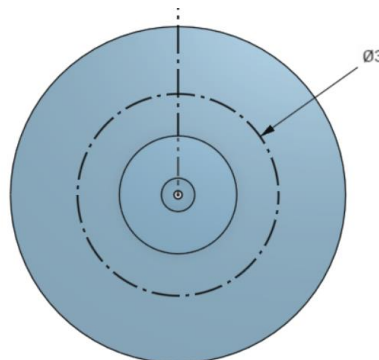
- a. Select Sketch Icon > Select the Top Plate of the Revolve Part > Rename Sketch > Bolt Holes > Select n Key to rotate perpendicular to the selected face



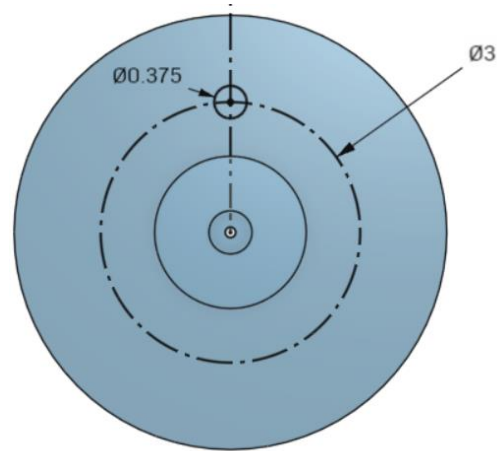
- b. Drawing Bolt Curve.

Bolt Curve is a circle that will represent the distance from the center of the part to the center of the first hole. This will eliminate 1 of the 2 locating dimensions for the hole, making it simpler to adjust the location of the hole, having to change only one dimension.

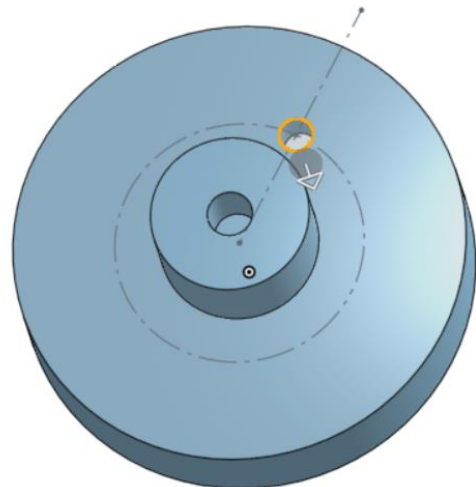
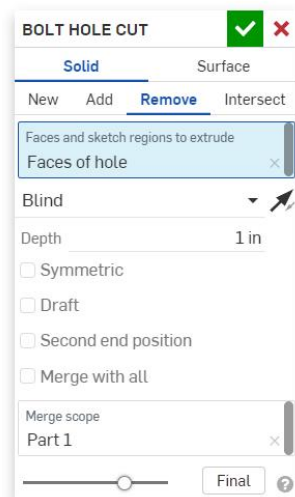
Draw the Following Construction Circle and Construction Vertical Line (NOTE: the Construction Tool can be selected before or after the feature is created) > Set the Diameter to 3; Length of Vertical Line does not matter as long as we have an intersection point to the circle



- c. Draw a circle (NOTE: this is NOT a construction circle) at the intersection of the construction line and circle (Coincident Constraint is applied so if the diameter of the circle changes then the hole (circle) will follow it down or up the vertical line because the center of the hole is constrained to the line) > Set Diameter to .500 > Green Check to Accept the Sketch



- d. Select the Sketch Bolt Holes > Select Extrude > Rename Extrude to Bolt Hole Cut > Select Remove > Cut thru the plate > Green Check to Accept



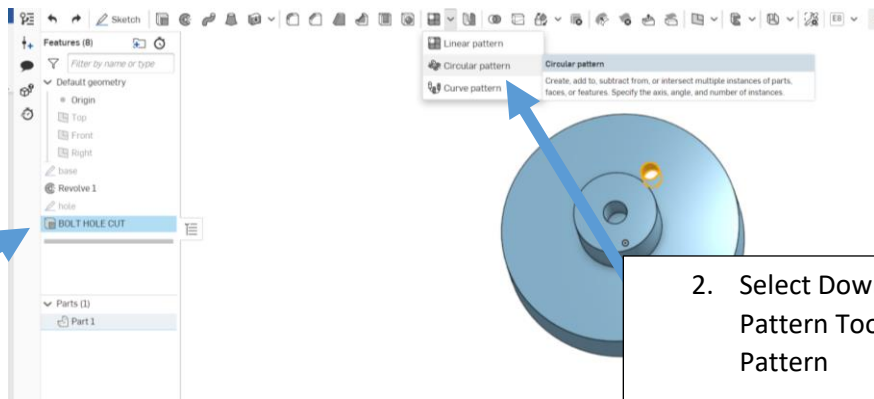
- e. Pattern Hole: Circular Pattern (also known as axis pattern)  
As discussed in pervious tutorials. Patterning is a useful tool to copy the same feature multiple times and place it in an array or in specified locations. Here an Circular Pattern is to be used to place the created iterations based on the center of the part along the bolt curve (Circular Construction Sketch)

Lugnuts equally spaced to dispense mechanical load evenly



Select Down Arrow Next to Pattern Tool > Select Circular Pattern

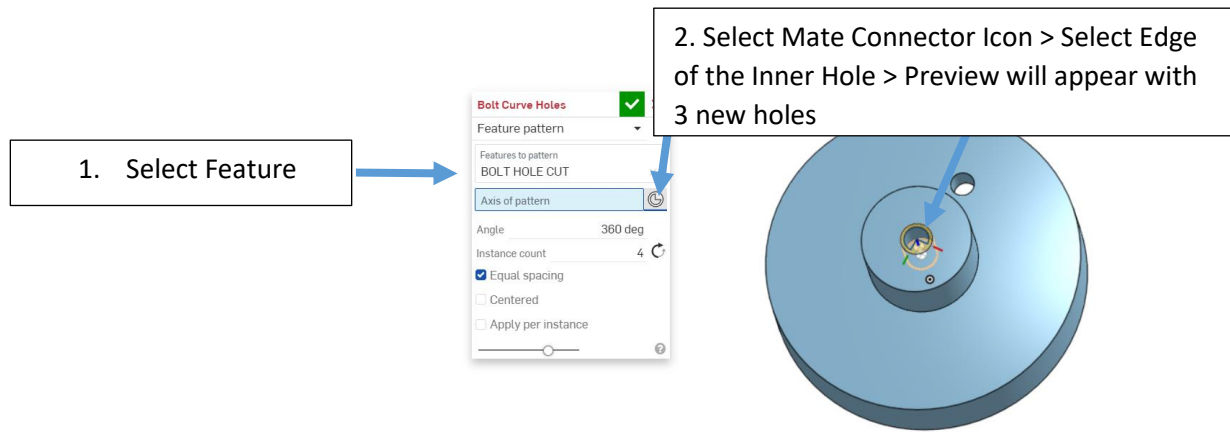
1. Select Bolt Hole Cut



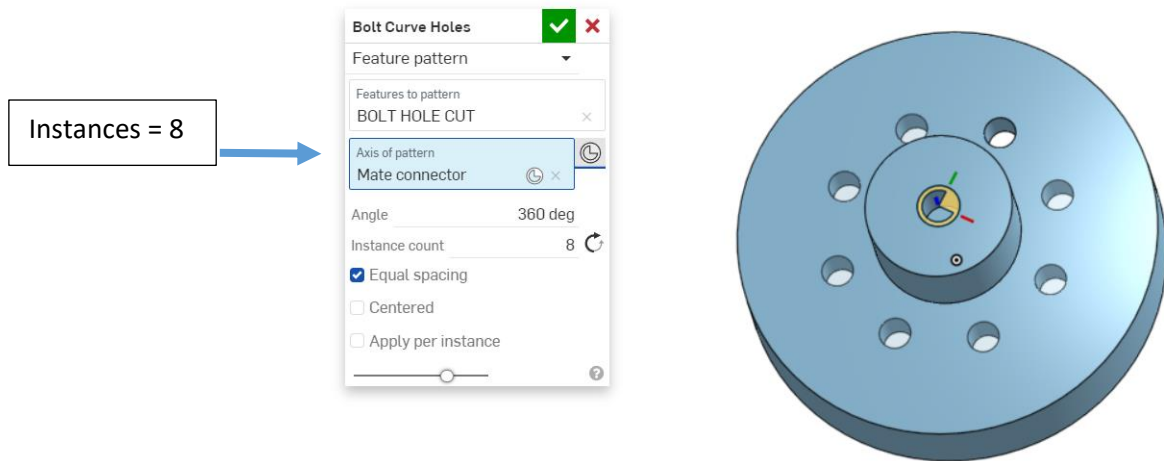
2. Select Down Arrow next to Pattern Tool > Select Circular Pattern



- f. From the Pattern Properties
  - i. Rename Pattern: Bolt Holes
  - ii. Change Pattern Type > Feature Pattern
  - iii. Select Extrude Bolt Hole Cut > Select Mate Connector Icon next to Axis Pattern > Select inner Hole of the part > This will snap to the center of the part, which is the center axis that we want to spin around > Preview will show 3 new iterations of the hole

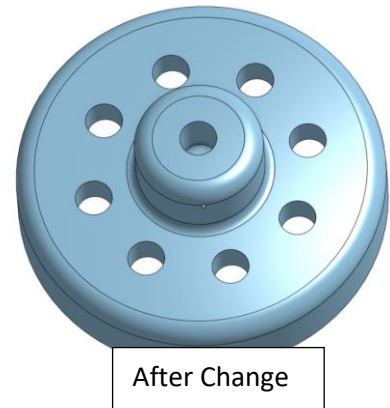
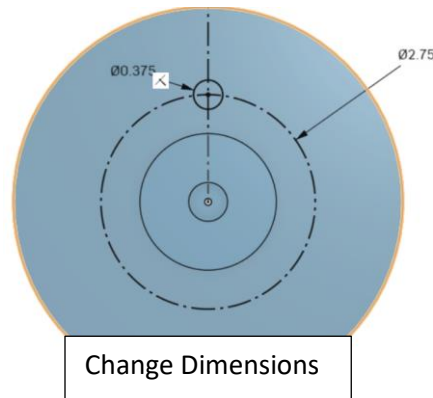


- iv. Change the Instance count to 8 in the Pattern Properties Menu > Software will automatically update the drawing > Green Check to Accept Pattern



g. Changing the Diameter of the 1<sup>st</sup> Hole (Parent)

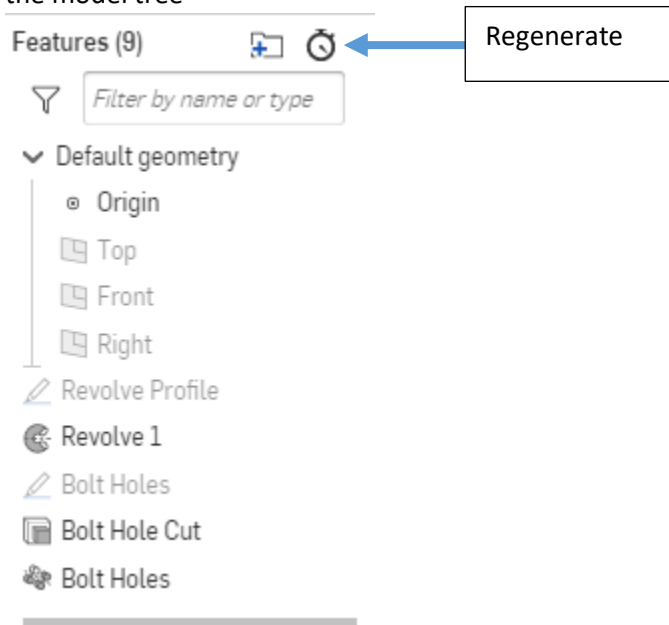
- i. Double Click on Sketch Bolt Holes > Change the Diameter of the hole to .375 > Change the Bolt Curve Diameter to 2.75 > Green Check to Accept > Notice the part does not update (Holes are not rendered)



ii. Change Bolt Curve Diameter back to 3.00

iii. Update Screen

Screen should update. IF it does not press the Regenerate icon located on top of the model tree



h. Add the following Rounds/Fillets

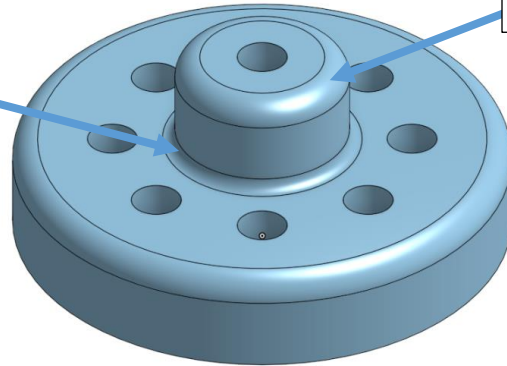
i. Rounds = .250

ii. Fillets = .125

Change Center  
Cylinder to diameter  
1.75

Fillet

Rounds



Final Part

Submission: Share Document to  
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