

9-10-2017

# Creating a Water Turbine in SolidWorks

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## Recommended Citation

Haight, April; Lennex, Lesia; and Elswick, Justin, "Creating a Water Turbine in SolidWorks" (2017). *Department of Middle Grades and Secondary Education Research & Publication Archives*. 4.  
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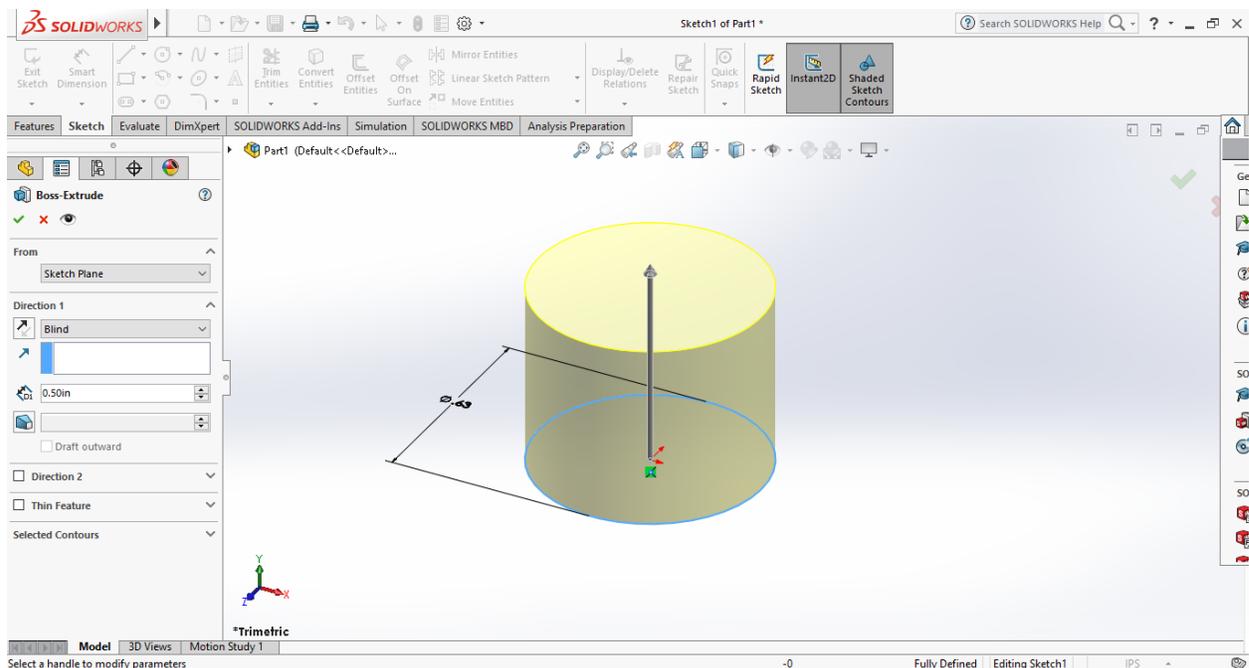
## Creating a Water Turbine in SolidWorks

### Open SolidWorks:

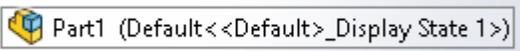
- When SolidWorks fully loads, you may need to select **New** at the very top left of the screen, or click on the piece of paper. When prompted, select the **Part** option to create a new part.

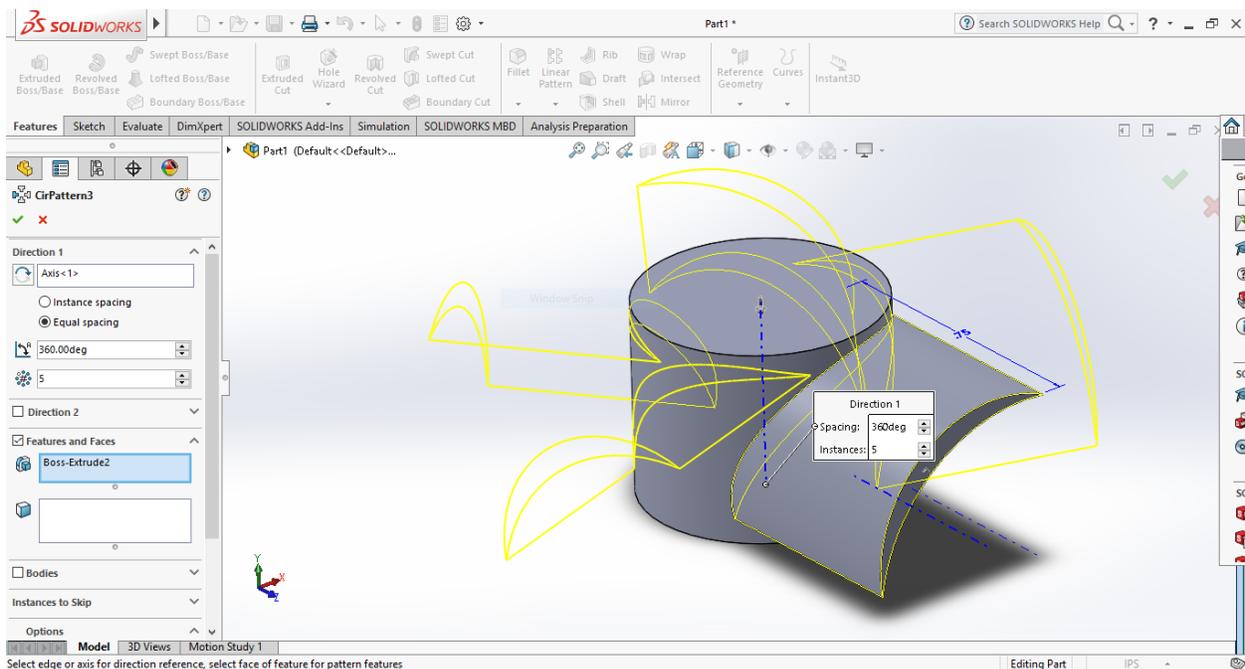
### Constructing the central cylinder:

- At the top left of the screen, under the **Features** tab, click on **Extrude Boss/Base**. Select the **Top Plane**. This will allow you to draw on the top plane.
- Select the **Sketch** tab and click on the **Circle** tool (which looks like this: ). Then, construct a circle by *clicking on the center of the origin* in the arrows located in the middle of the screen and then *dragging your mouse outward*. At this point, the size of the circle doesn't matter, but make sure it's big enough so that you can see it.
- Click on **Smart Dimension**. This will allow you to resize your circle. Click on the outer ring of the circle you drew, move your mouse somewhere else close to the circle, and click again. A window will open that will prompt you to enter in a size
- For the purposes of this demonstration, enter **5/8**. Underneath this measurement, you should see a box that says **Units**. Hover over this and select **in**, for inches. This is the only time you'll have to select inches, the program will automatically set all future measurements to be in inches as well.
- Once you've resized the circle, click on **Exit Sketch**. You will see the circle rotate and an arrow coming out from it. On the left-hand side of the screen, you will see a toolbar. About halfway down the screen, you will see a line that looks like:  and a measurement. Enter **1/2**. This will make your cylinder a half inch tall. When you've finished, hit the **green check mark**.



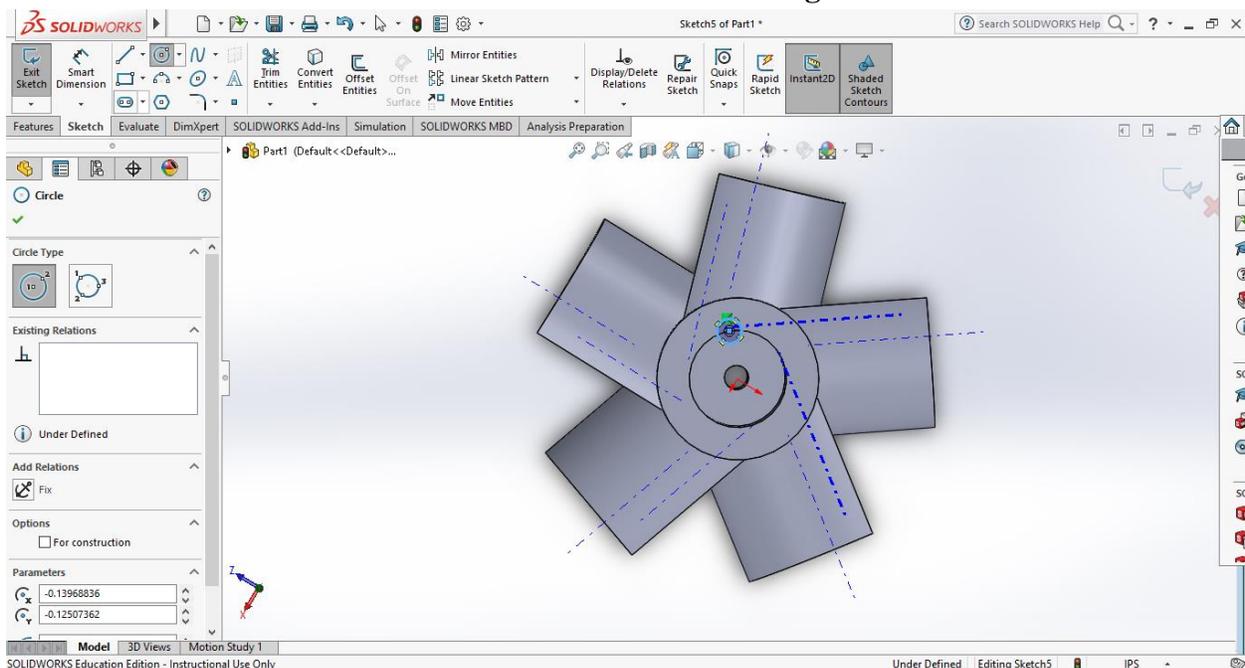
## Constructing the turbine blades:

- To construct the turbine blades, select **Extruded Boss/Base**. At the top right of the work area you should see something like this:  Click on the arrow at the left and select either **Front Plane**. Then, click on the **View Orientation** icon (which looks like this: ). You should see a cube representing different views of the model, click on the plane that is facing you, to the left. You should now be looking at the side of your model.  
  
Click on the **Arc** tool (which looks like this: ) and click **3 Point Arc**.
- Draw the shape of your turbine. This is entirely up to you but remember the principles of how turbine blades work. You want to have a crescent shaped blade with a bowl in it, you'll want it to be at a slant and not too vertical or horizontal. You will need to draw two arcs in order to give the blade a thickness. Make sure the two arcs are joined at the ends. When you are finished, **Exit Sketch**.
- For the dimension of the Boss-Extrude (recall that it looks like this: ) enter **3/4**. Then select the **green check mark**. Click in on your mouse's scroll wheel and move your mouse to rotate the model, and make sure the blade is a desired shape.
- Next, click on the arrow beside the **SolidWorks logo**, click on **View**, hover over **Hide / Show**, then click on **Temporary Axes**. Blue dotted lines will appear in your model; this will be useful for the next step.
- On the **Features** tab, select the arrow underneath **Linear Pattern** and select **Circular Pattern**. Click on the blade you made, and then click on the blue dotted line located in the middle of your cylinder.
- For the toolbar on the left, make sure that **Equal Spacing** is selected. Underneath it, besides the logo, enter the number of turbine blades you want the turbine to have (in the screenshot I have 5 blades ready to go, but the number that the turbine has is at your discretion), then click on the **green check mark**.



### Constructing the center key.

- Click on **View Orientation** again and click on the view such that you are looking at the very top of your turbine. Click on **Extruded Cut** and select the top face of the central cylinder.
- Next, click on the **Circle Tool** and draw any size circle that is contained within the central cylinder. Make sure your circle is centered at the origin, like at the very beginning. Click on **Smart Dimension**, and then click on the outer edge of the circle you just drew. In the window that pops up, enter **3/8** and then click the **green check mark**.
- Click on **Exit Sketch**. Like when you were constructing Extruded Boss-Bases, you will see the option to enter a dimension. This is how far into the cylinder the program will carve out. Enter **1/4** and then click the **green check mark**.
- Orient the view so that you are again looking at the top of the turbine. Do the same thing as before, draw a circle at the origin, and this time when you click on **Smart Dimension** you will enter **3/32**. When you click **Exit Sketch**, there should be nothing for you to modify, so just click the **green check mark**.
- Next, select **Extruded Boss/Base** and click on the very top of the turbine cylinder, on the “outer ring” that is formed. Click on the **Circle Tool** and draw a circle centered at the edge of the cut you made (you will see this in the screen-capture at the bottom). Click on **Smart Dimension** and resize that circle to be **3/32** inches. Click **Exit Sketch**.
- Make sure that the dimension of the Boss-Extrude is **.25in (1/4)**. Click on the **Reverse Direction** button (which looks like this: ) and click the **green check mark**.
- Select **Circular Pattern**. Like before, click the bump you just made and the blue dotted line at the very center of the turbine. *Be careful here, as the center axis may be hard to see.* Make sure that **2** is entered next to the  icon and click on the **green check mark**.



**Congratulations**, you've successfully completed your turbine!

For more information about our project, scan this QR code to access our website:

