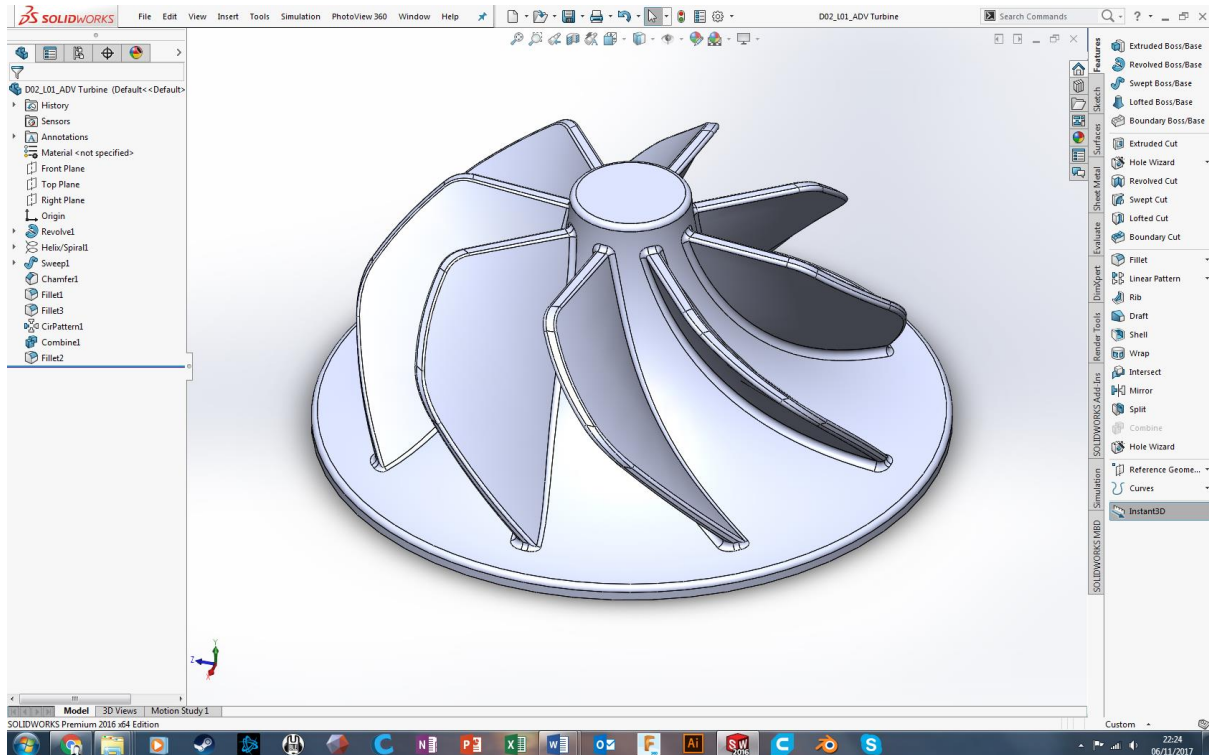
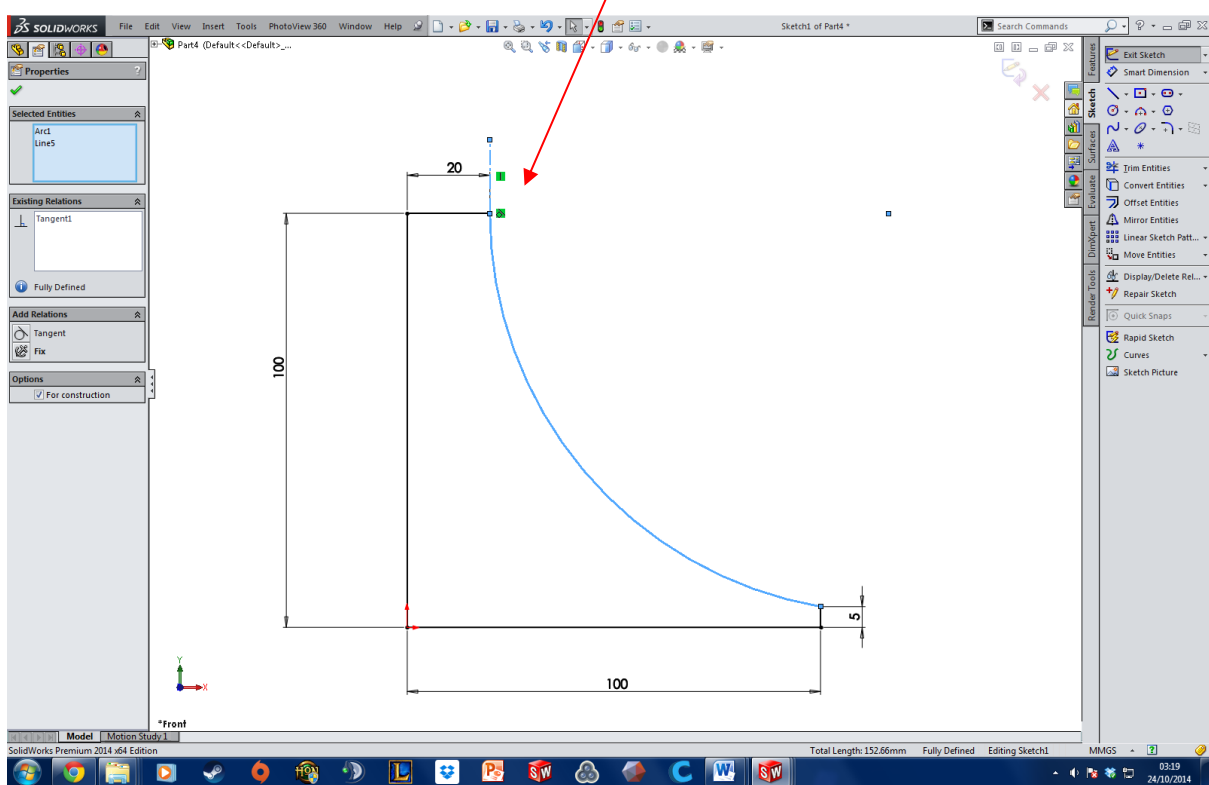


Turbine Blade

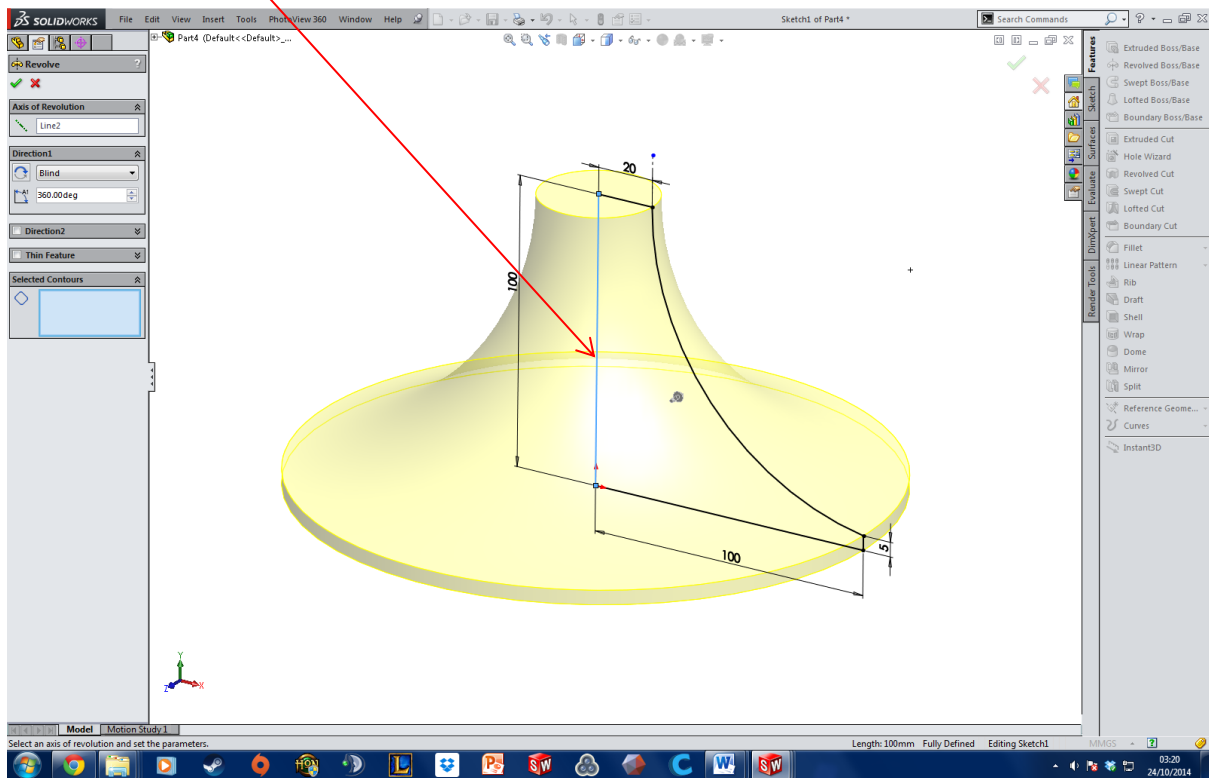
Create a New Part and Save it with the name **D02 L02 Turbine**



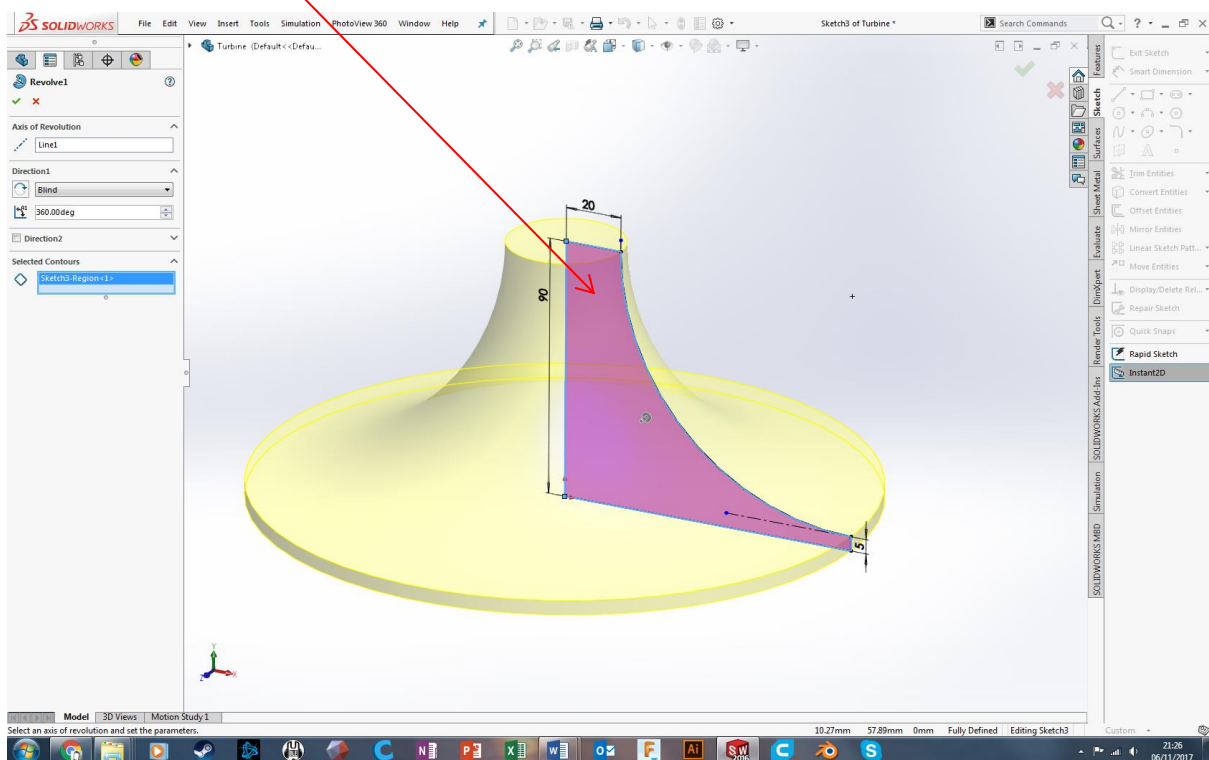
- Create a new sketch on the **Front Plane**
- Draw a sketch as shown below; make the **top centreline tangent to the curve**



- Use the **Revolve Boss/Bass** found in **Features**
- The **Axis of Revolution** is the middle line;

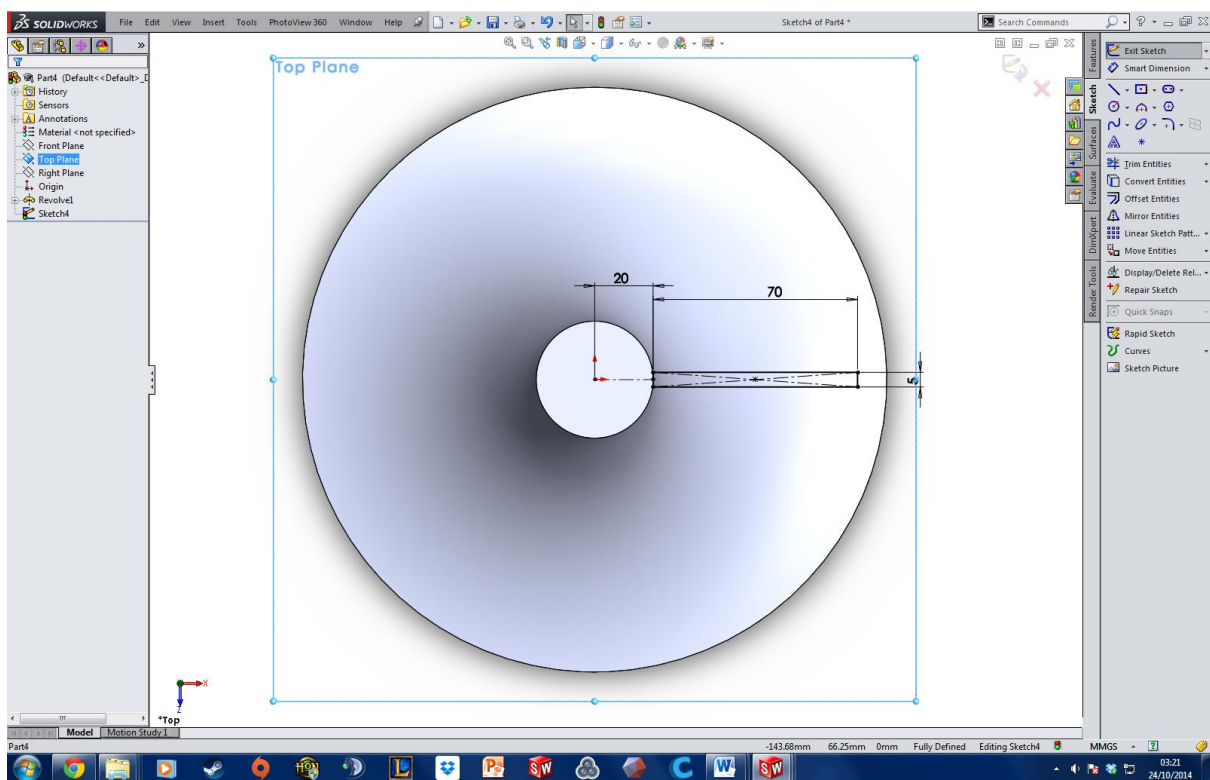
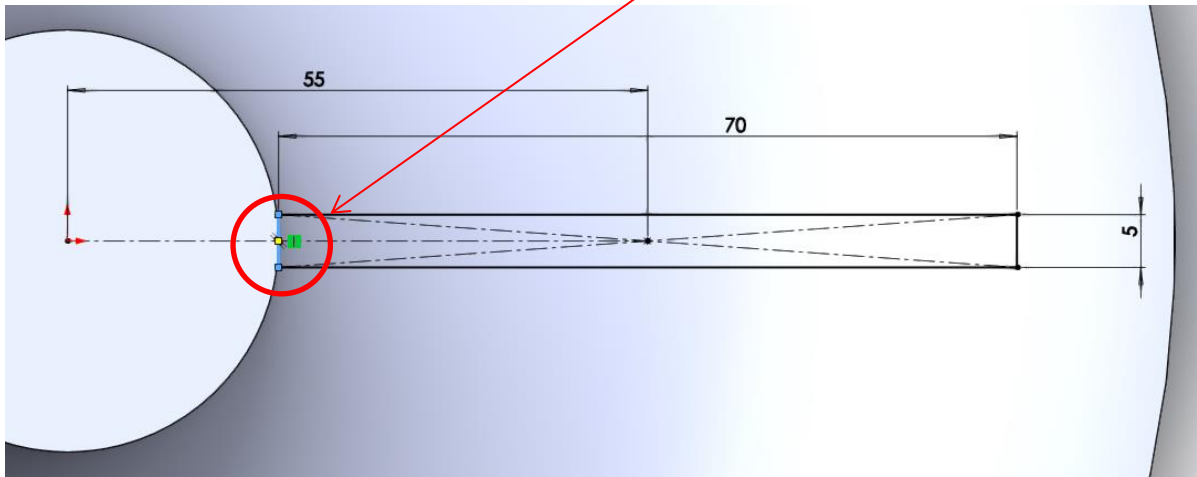


- **Selected Contours** are shown below

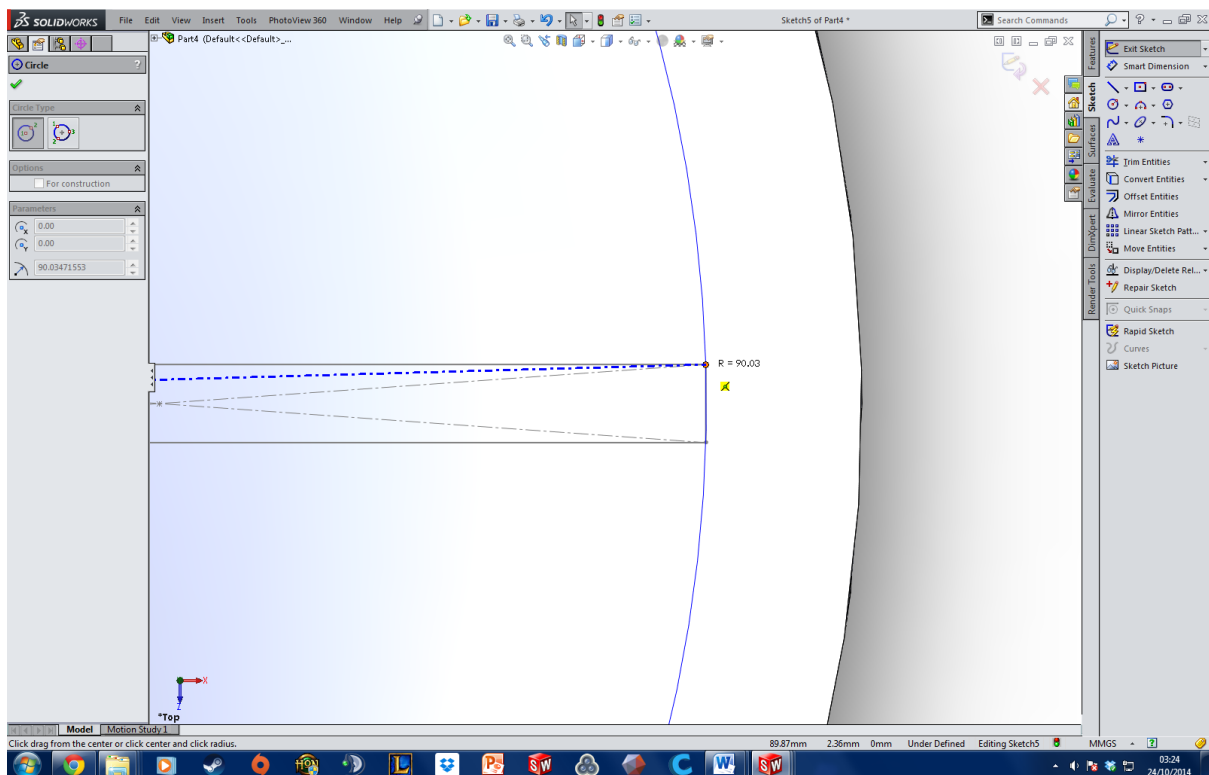
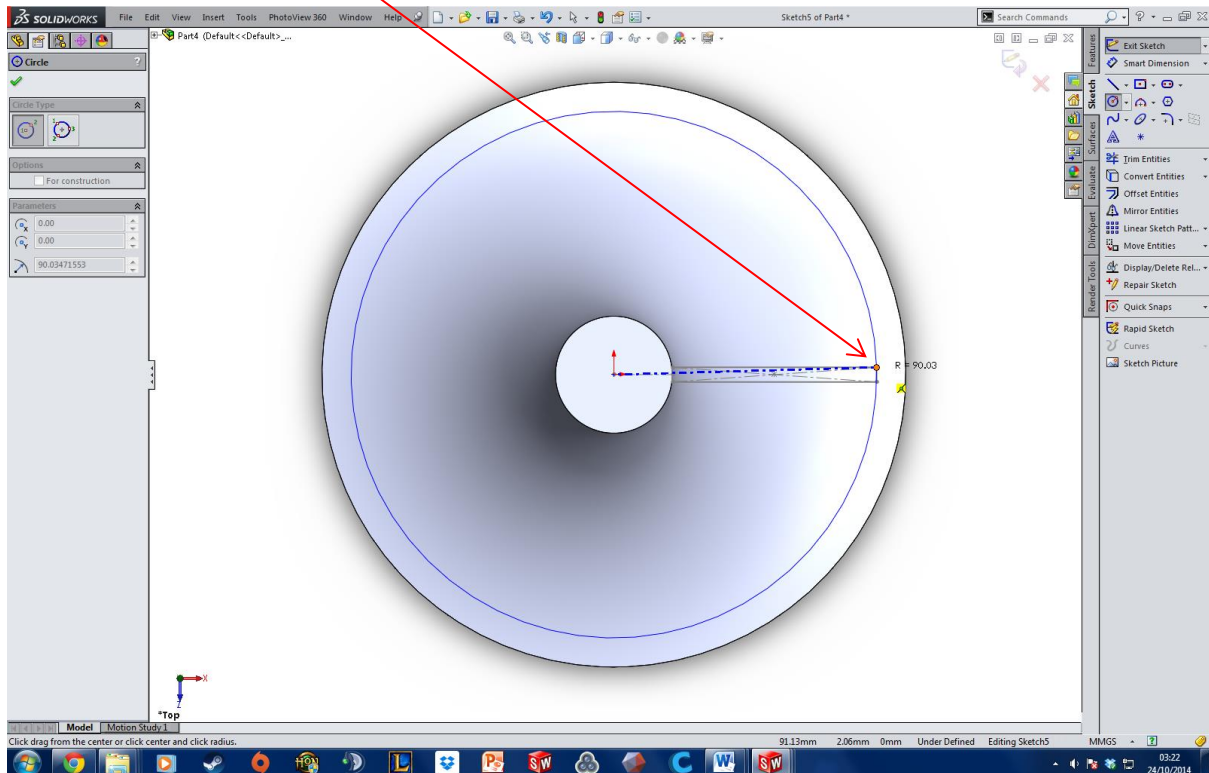


- Click the **Green Tick** and **Save File**

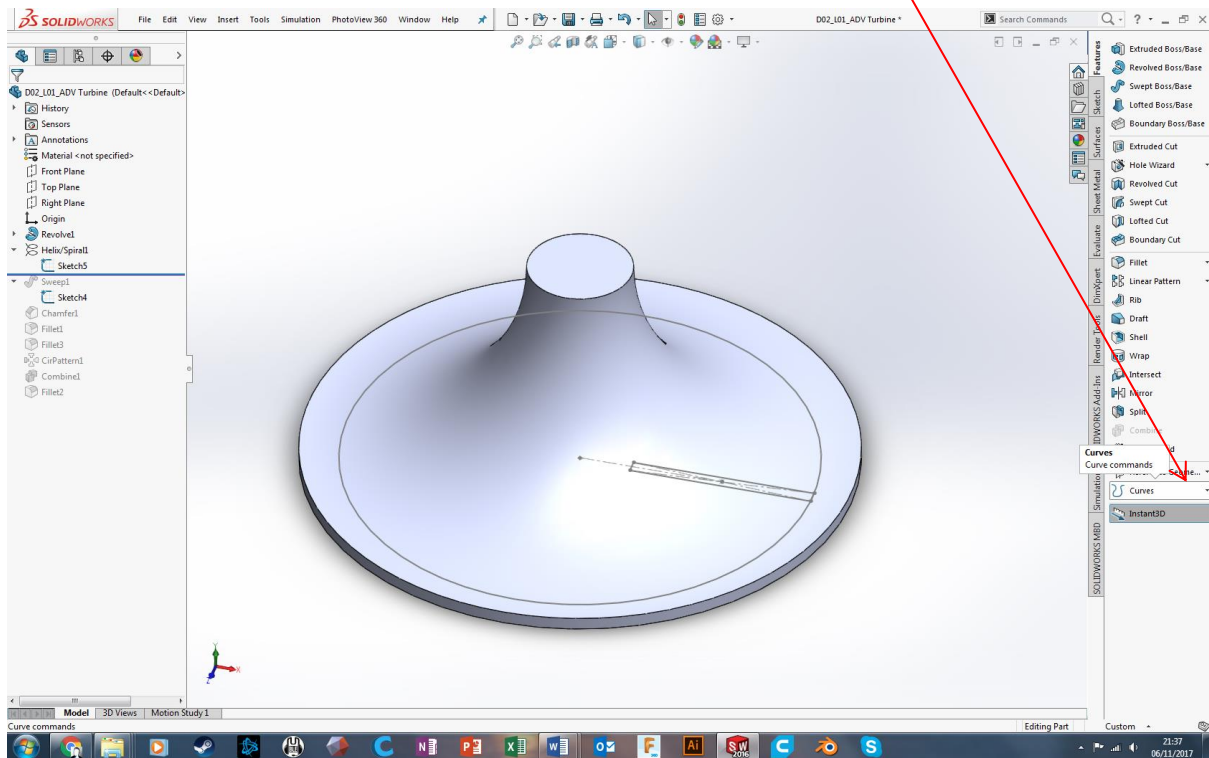
- Create a new sketch on the **Top Plane**
- Create a **Center Rectangle** with the left line connected to the **yellow point** (shown below)
- **Click accept**



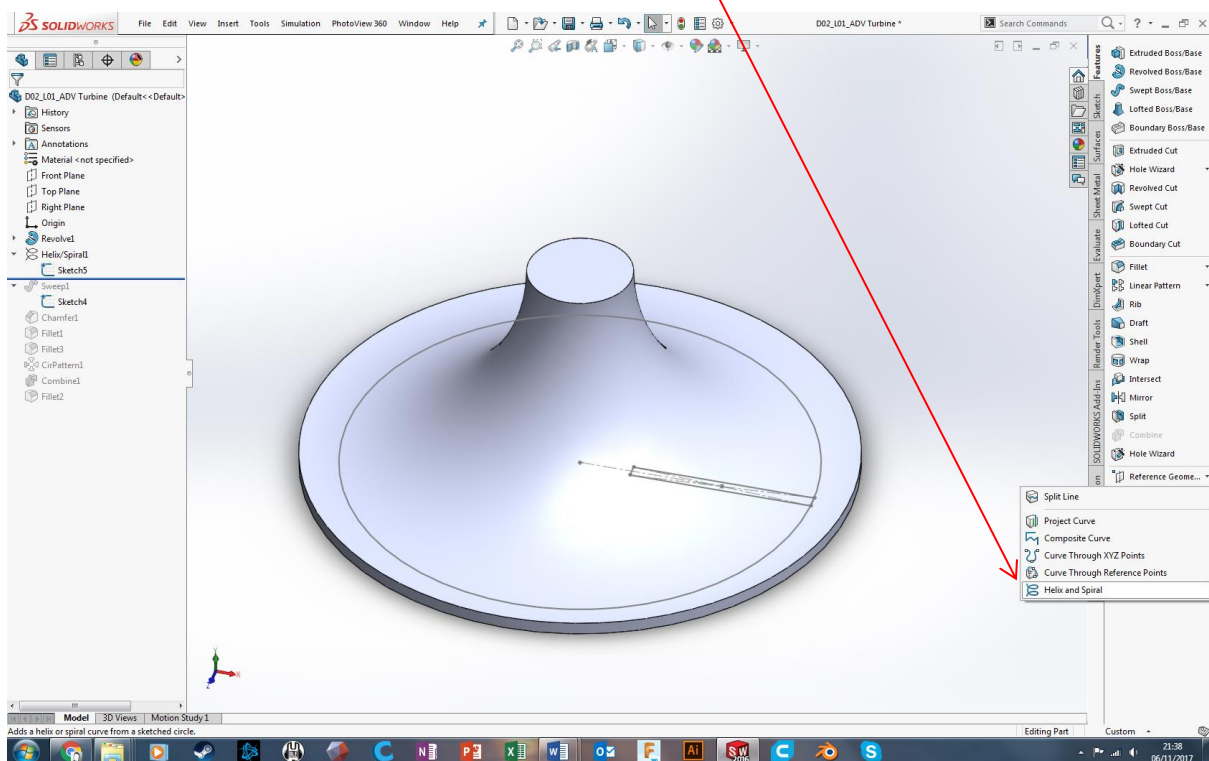
- Create a new sketch on the **Top Plane**
- Create a **circle** with **centrepnt** at the **point of origin** and the **outer diameter** should be **coincident** with the **top point of the square** (shown below)
- **Click Accept**



- Under the **Features Tools** click on the **Curves** drop down menu

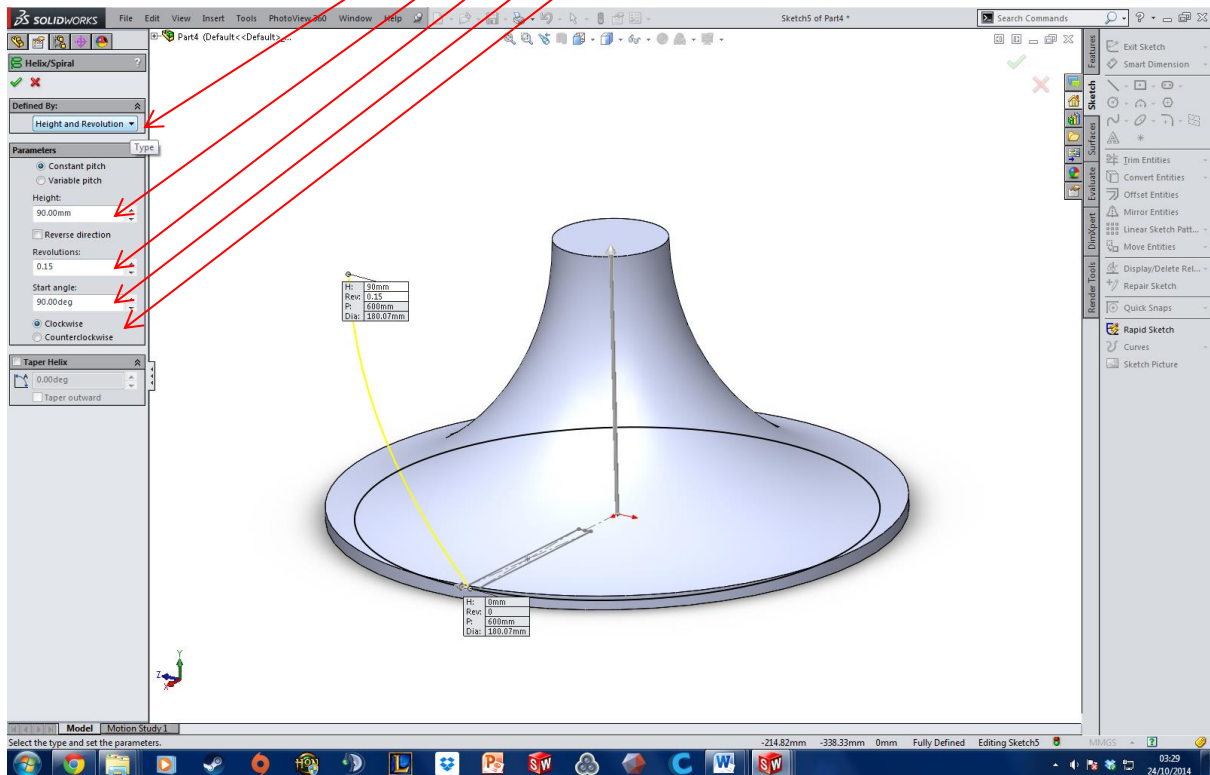


- From the **Curves** drop down menu select **Helix and Spiral**

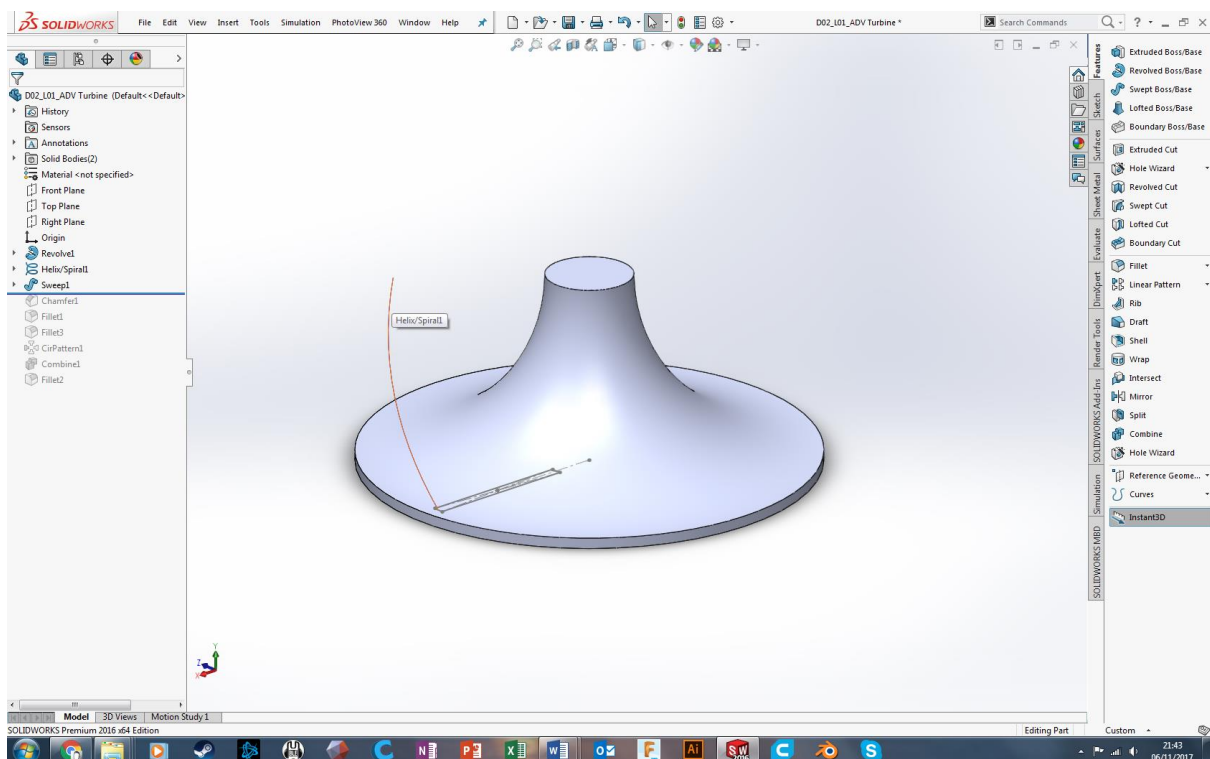


- Under the Helix/Spiral menu on the left, change the input data to the following information

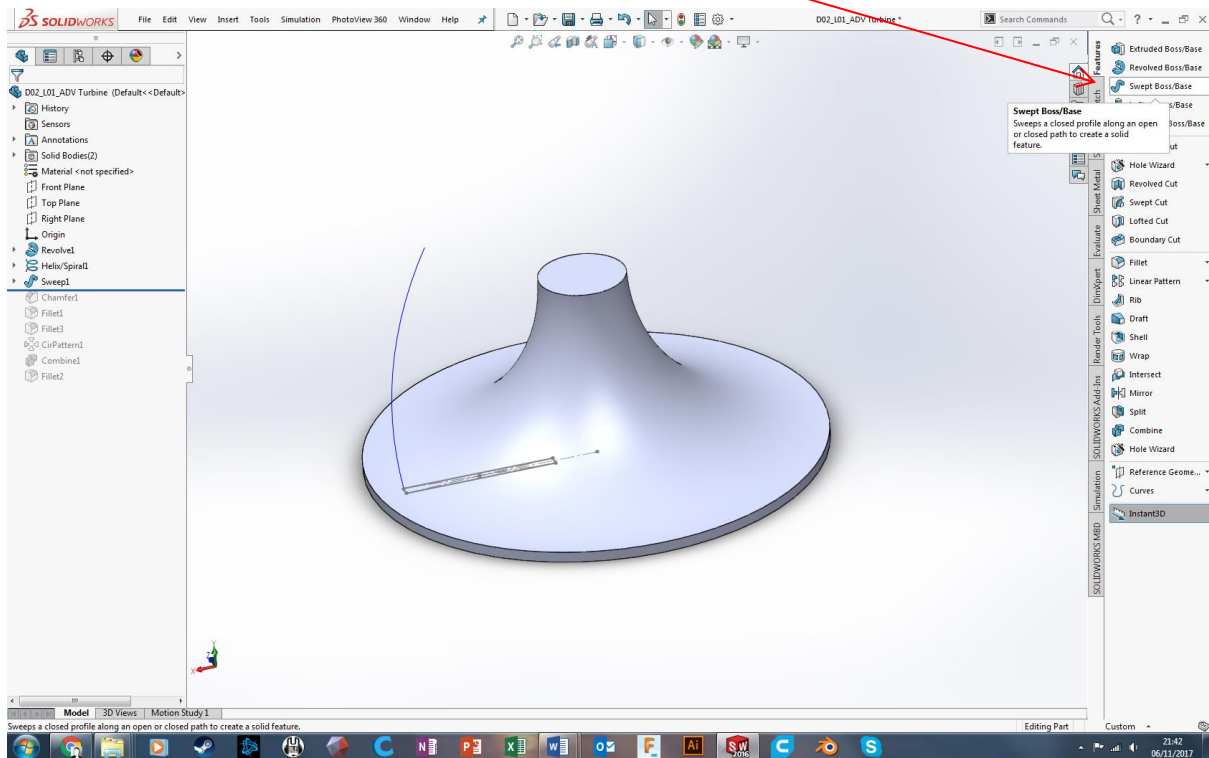
Defined By	Height and Revolution
Height	90.00mm
Revolution	0.15
Start Angle	90.00 deg
	Clockwise



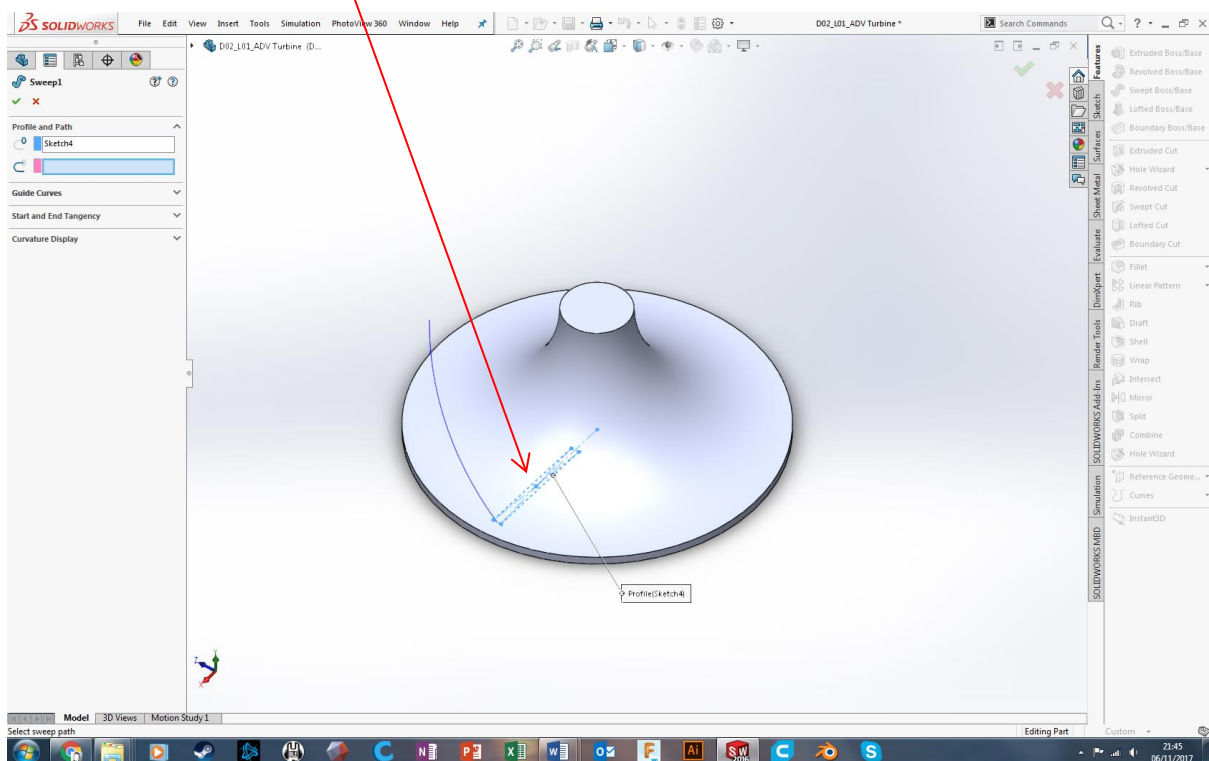
- The Helix should look as follows:



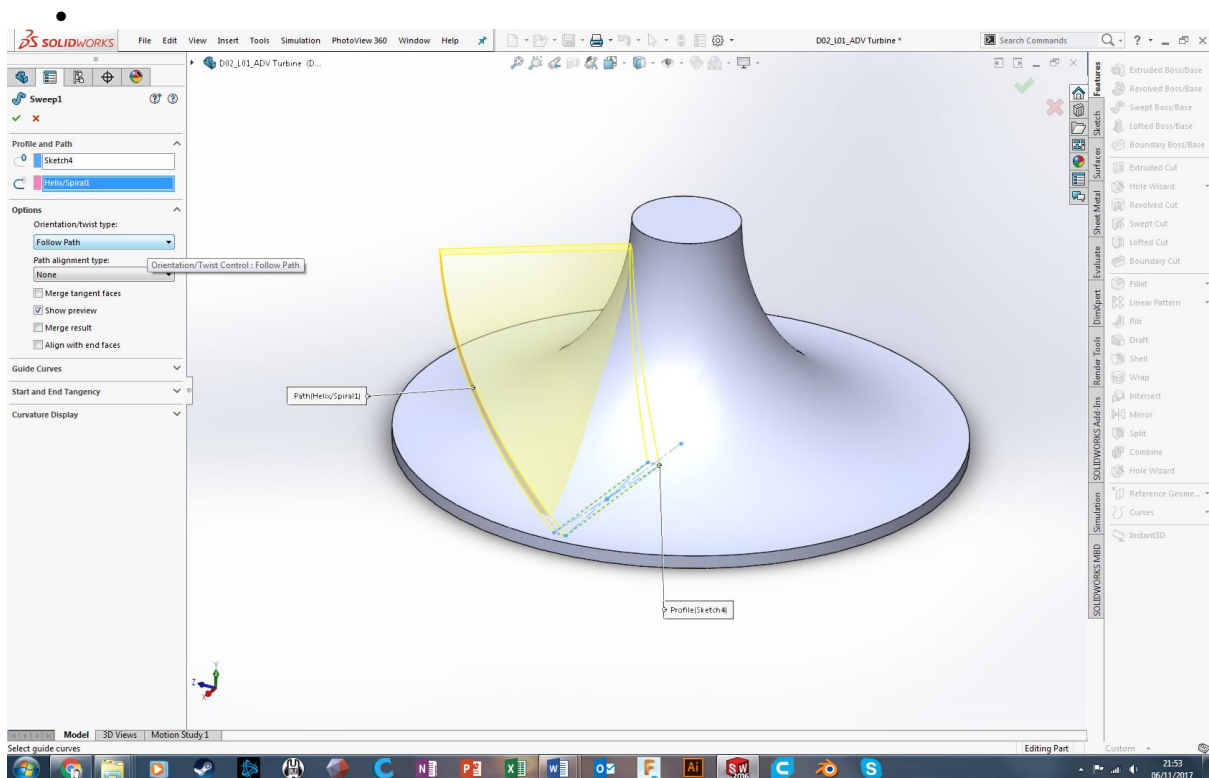
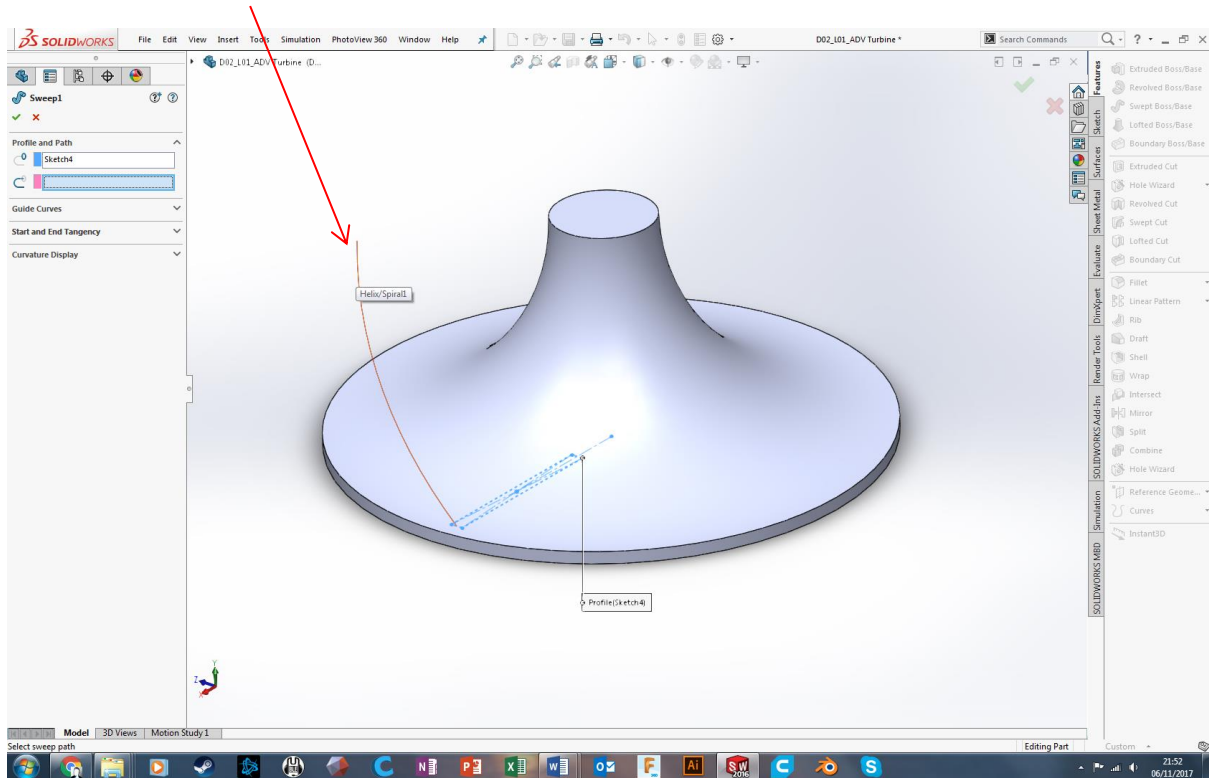
- Select the **Swept Boss/Base** found in **Features**



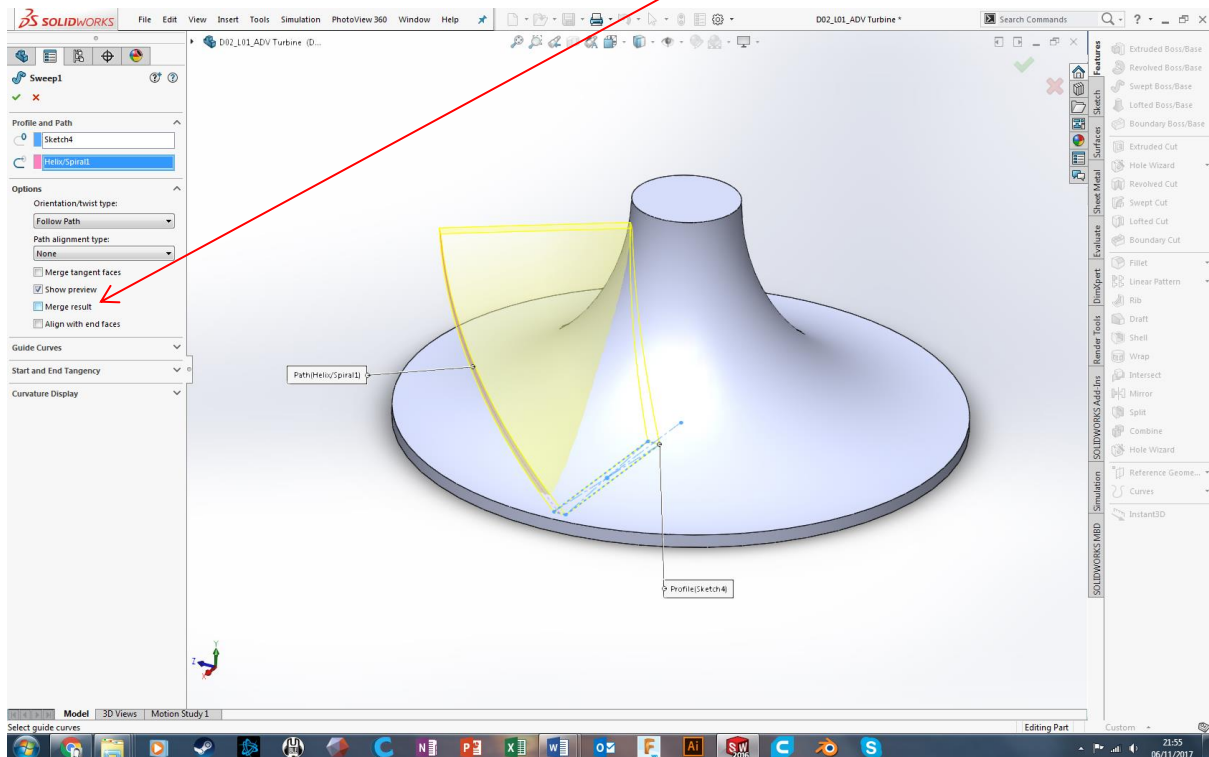
- Select the **Centre Rectangle** as the **Profile**



- Select the **Helix** as the **Path**

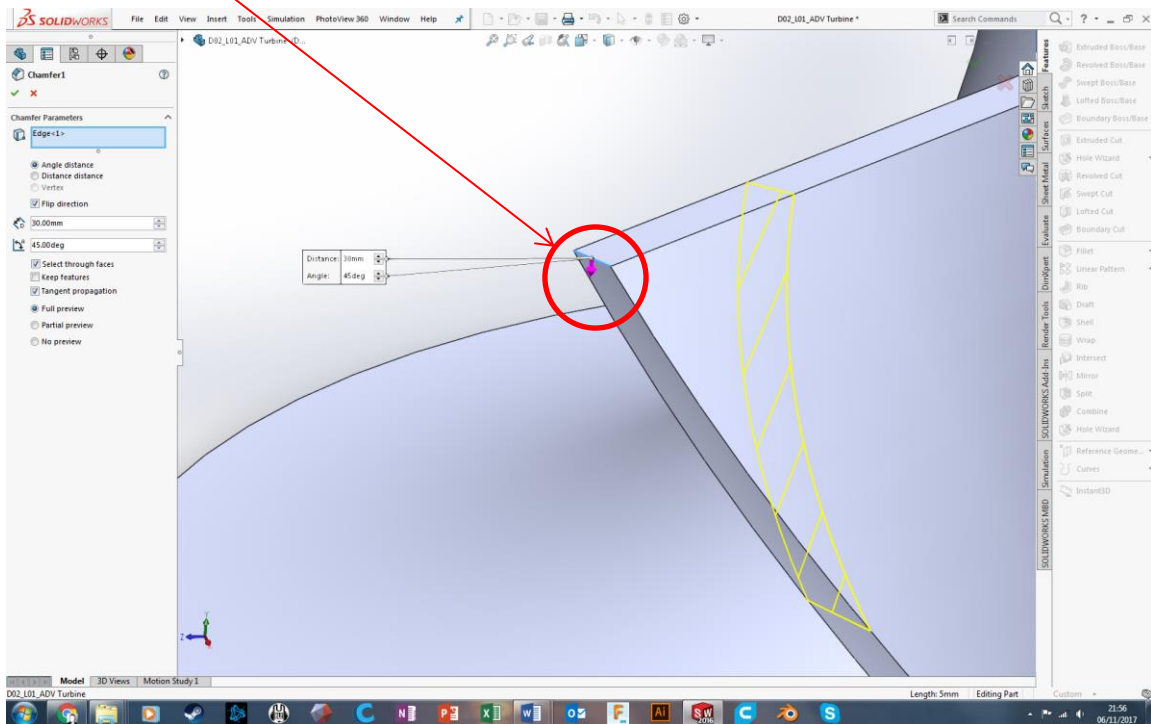


- Under the Sweep **Options** make sure the **merge results** option is not ticked

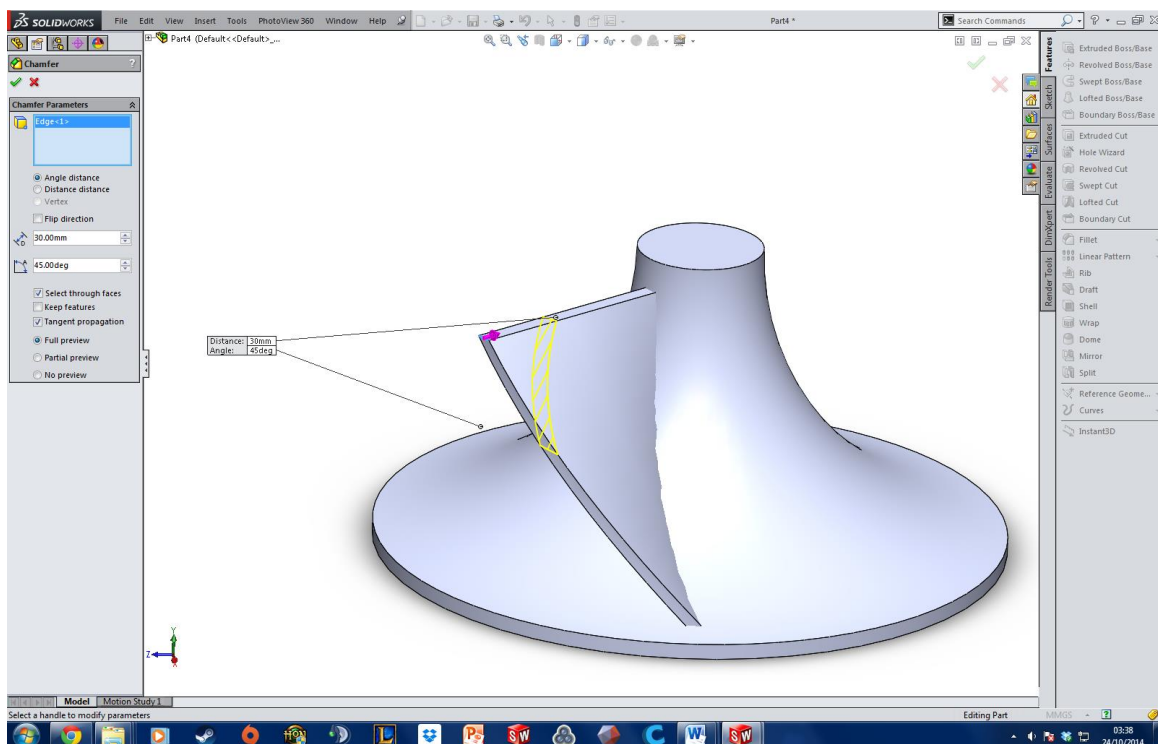


- Click Accept

- Use the **Chamfer Tool** found under **Features**
- **Select the edge of the Sweep**

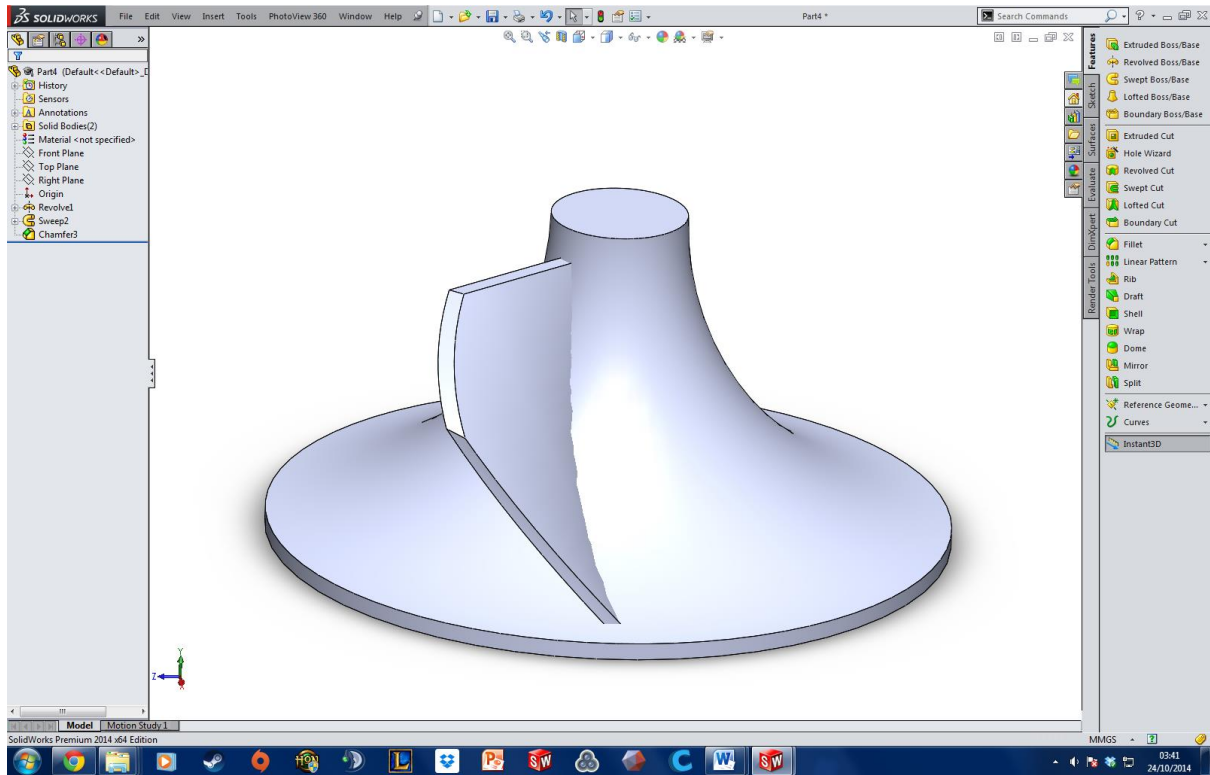


- **Distance is 30.00mm**
- **Angle is 45.00 deg**



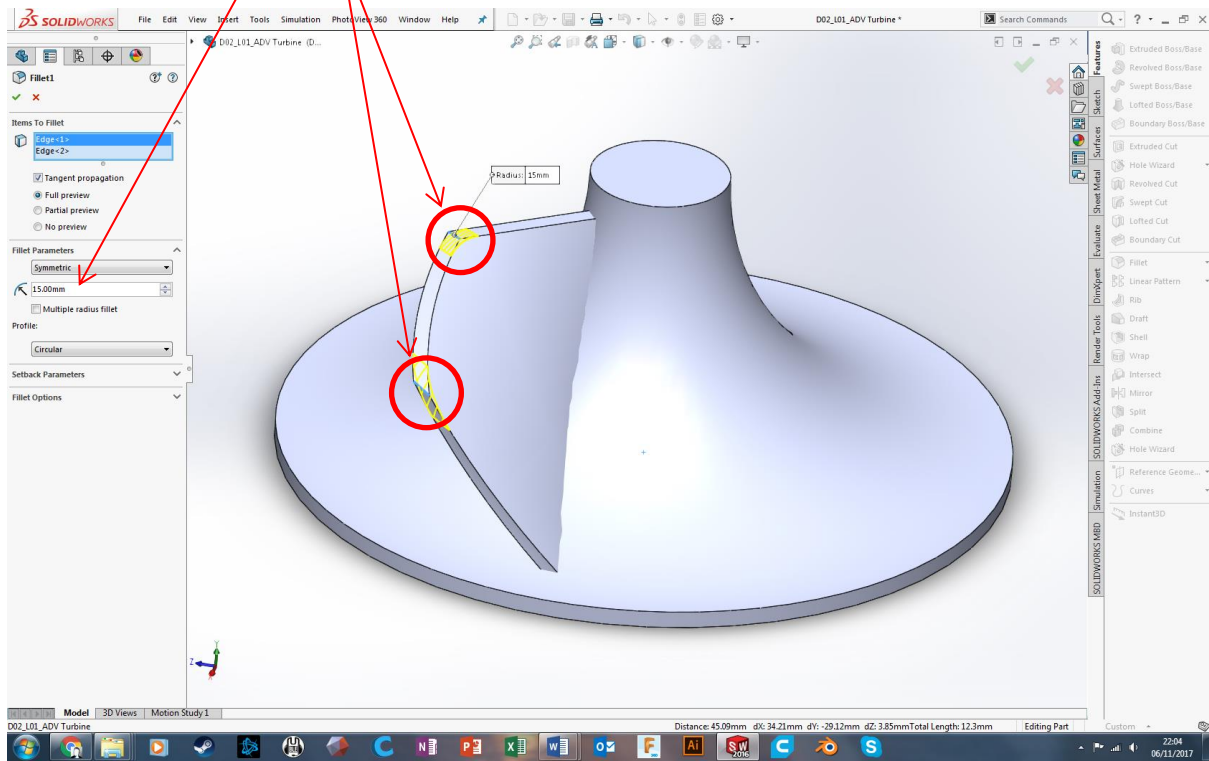
- **Click Accept**

- The part should look as follows:



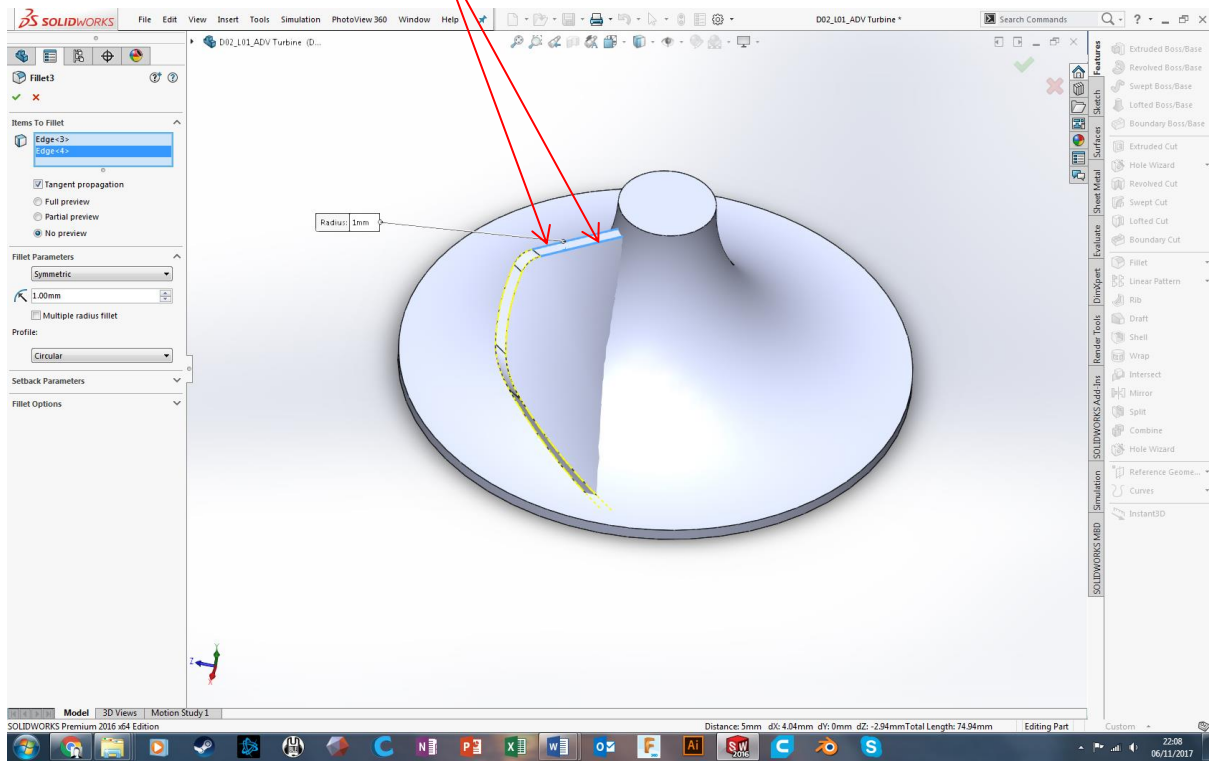
- Save the part

- Use the **Fillet Tool** found under **Features**
- **Select the two edges of the Sweep (shown below)**
- **Radius 15.00mm**

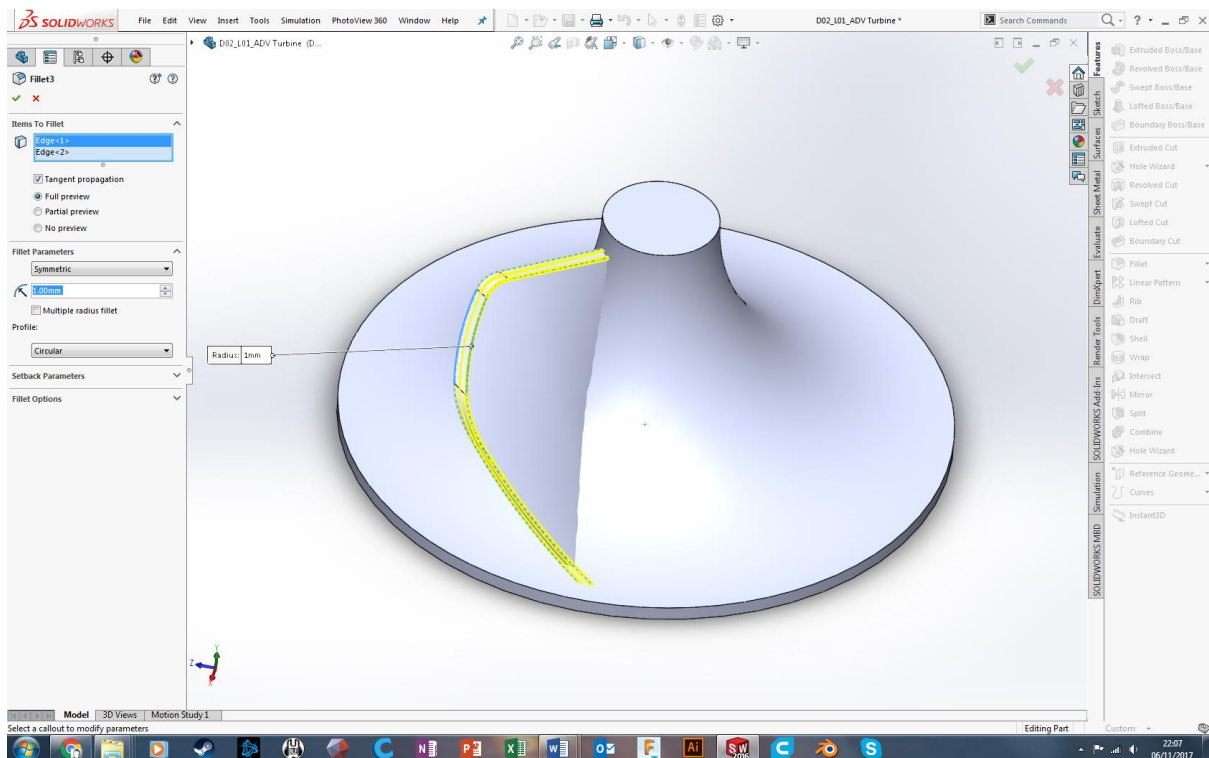


- **Click Accept**

- Use the **Fillet Tool** found under **Features**
- **Select the two outer edges of the Swept (shown below)**
- Radius 1.00mm

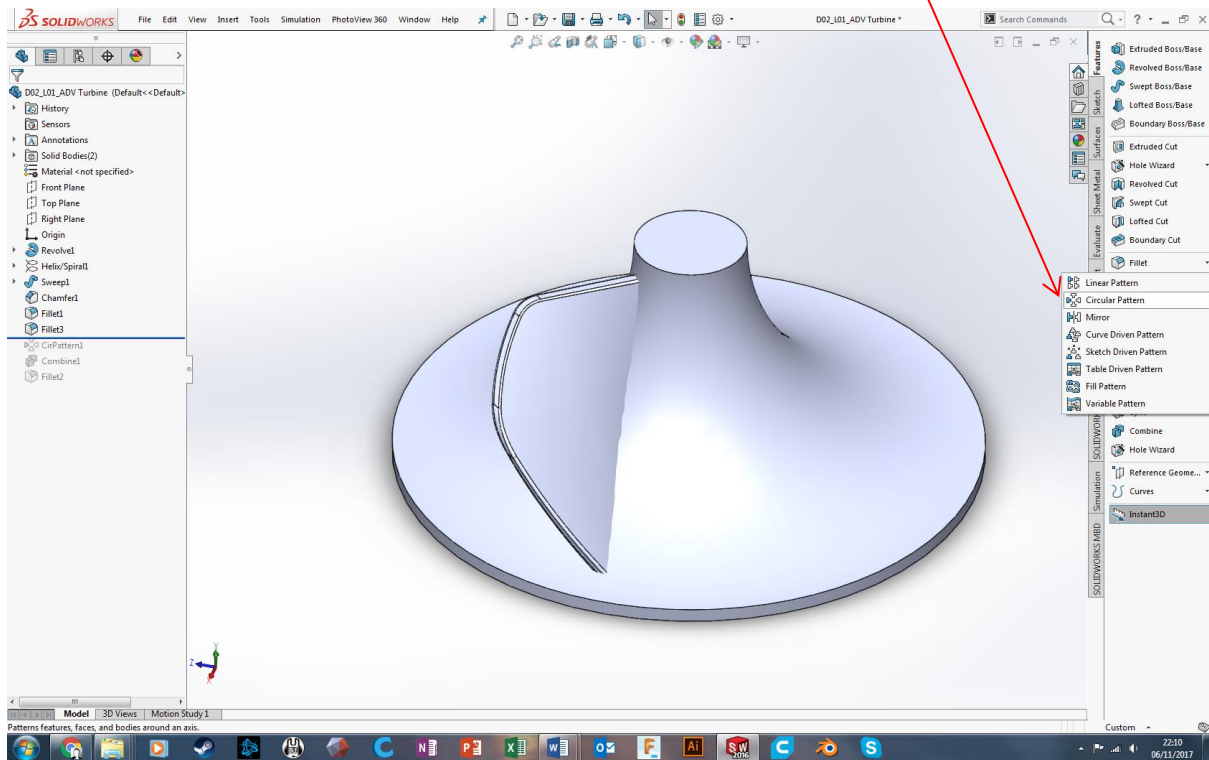


- The part should look as follows:

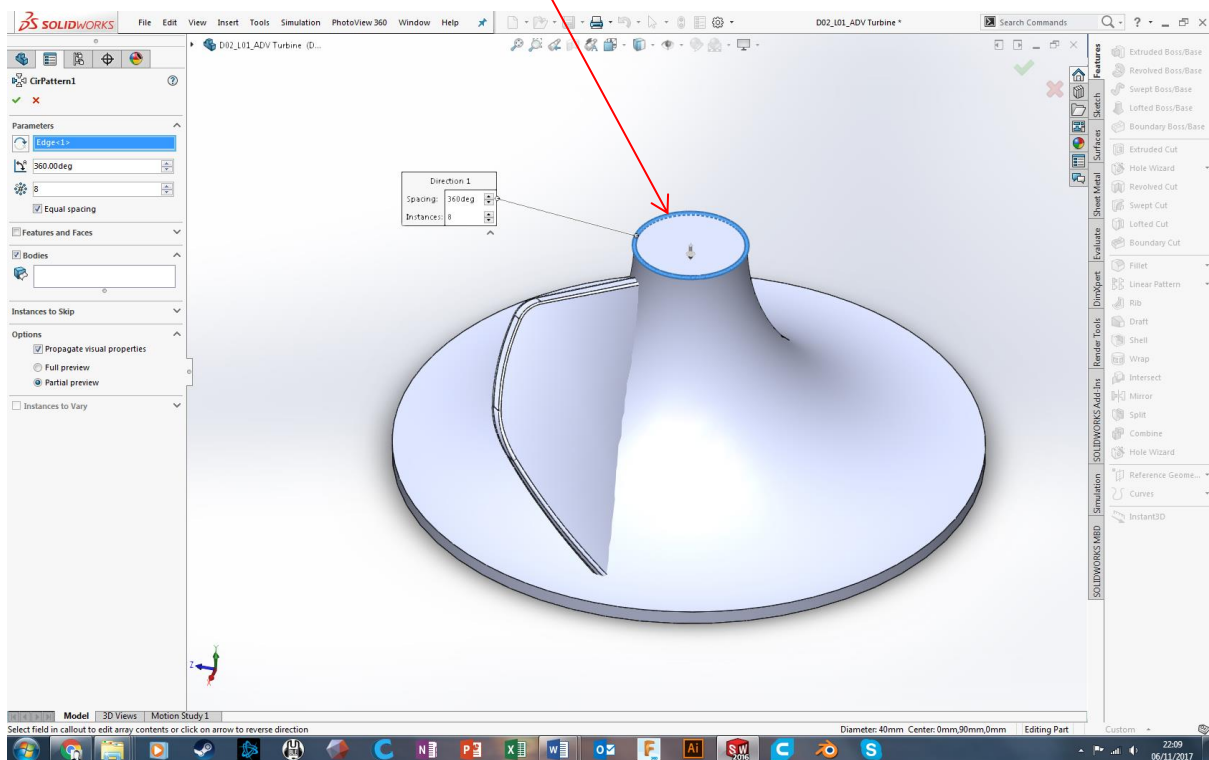


- Click **Accept**

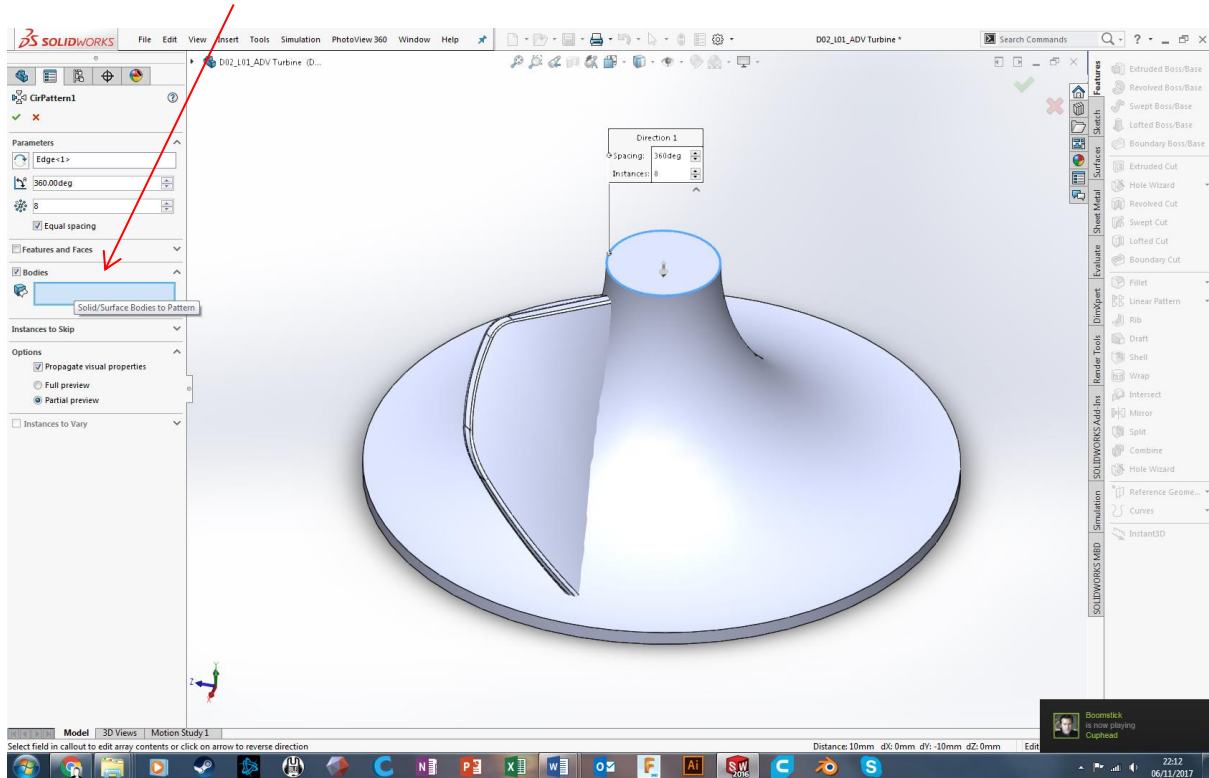
- Create a **Circular Pattern** found under **Features > Linear Patterns**



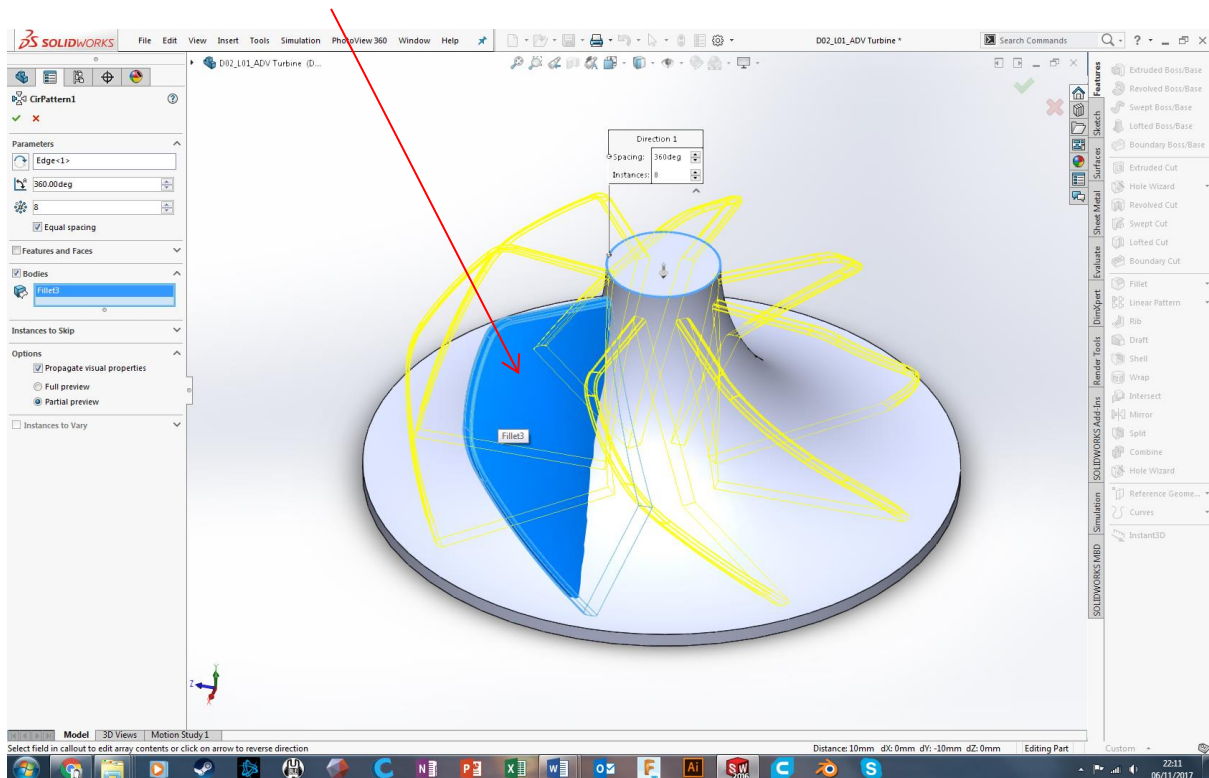
- Under the parameters, select the circle (below) as the pattern axis



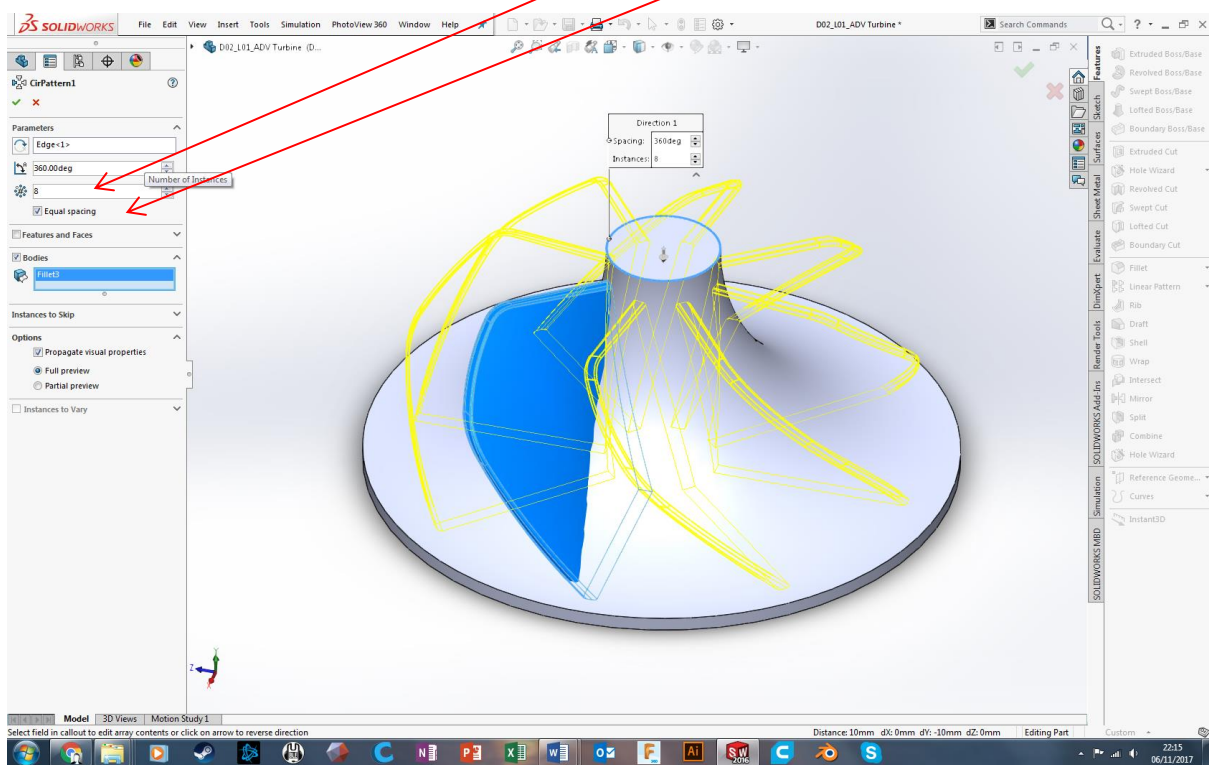
- Click on Bodies (In the menu panel)



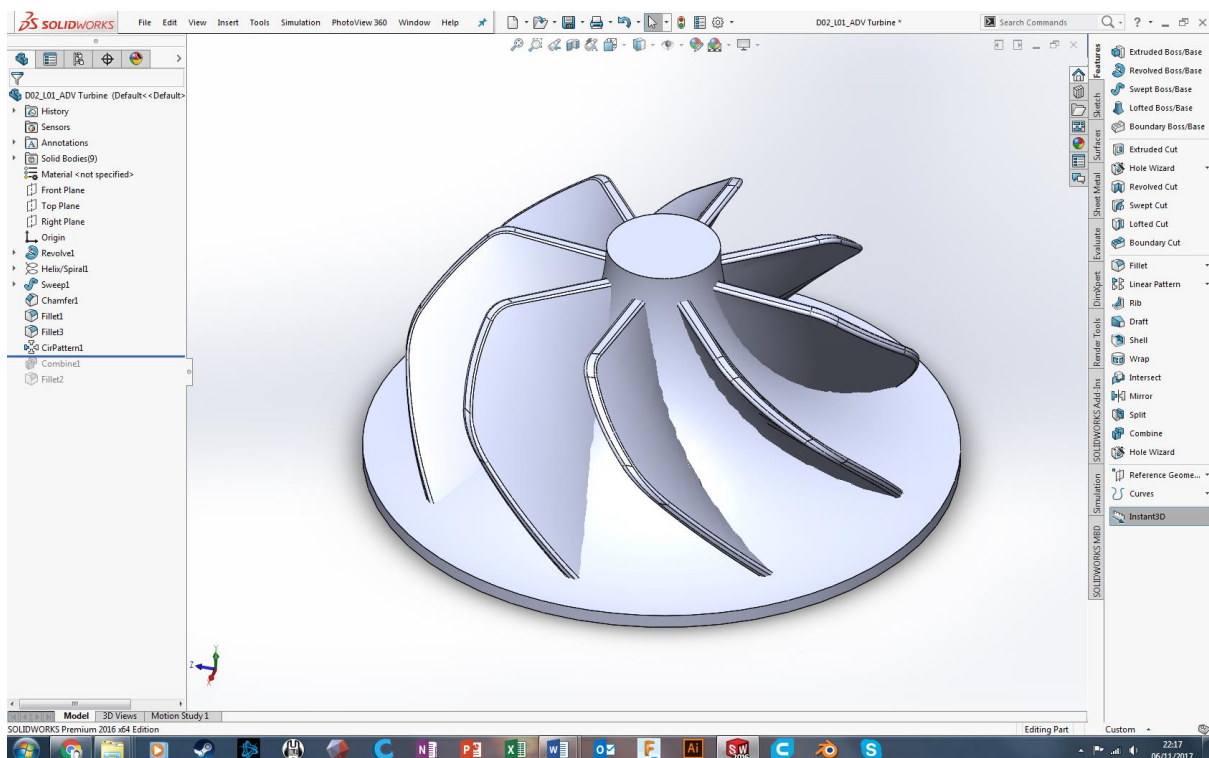
- Select the Sweep Profile as the Body to Pattern



- The number of **instances** should be **8** and select **Equal Spacing**

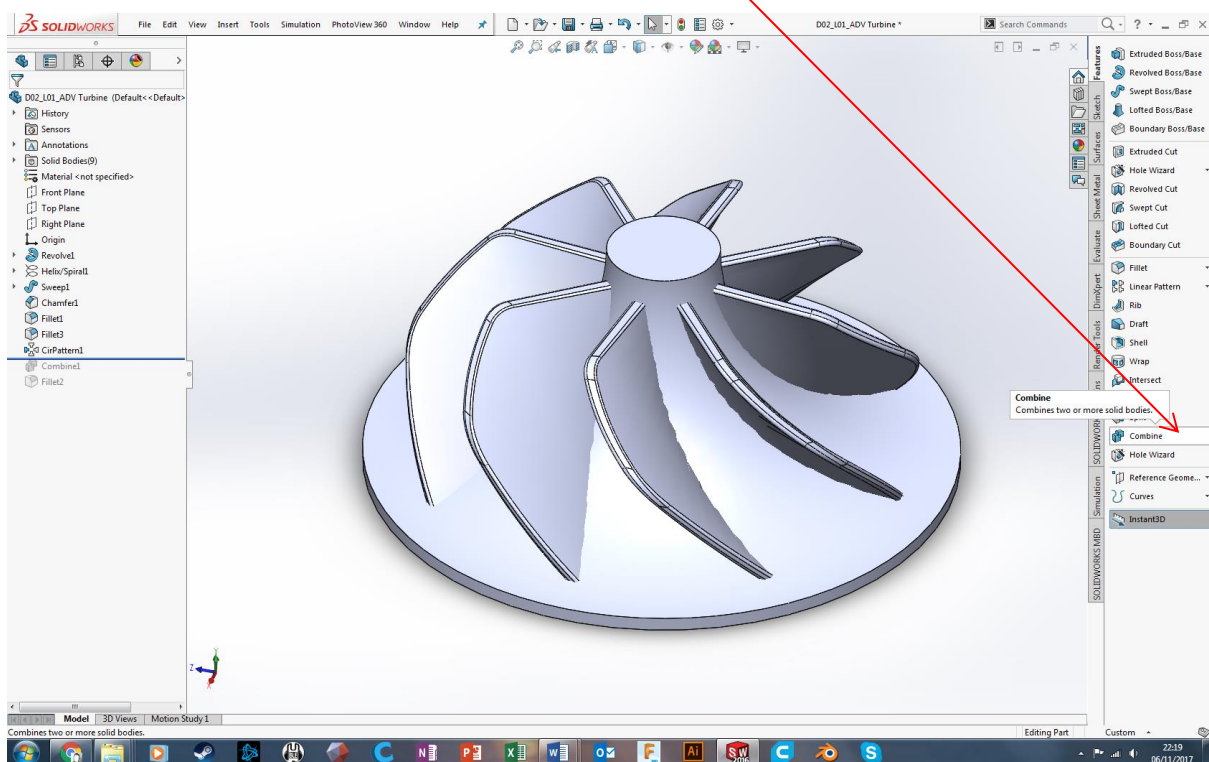


- The part should look as follows:

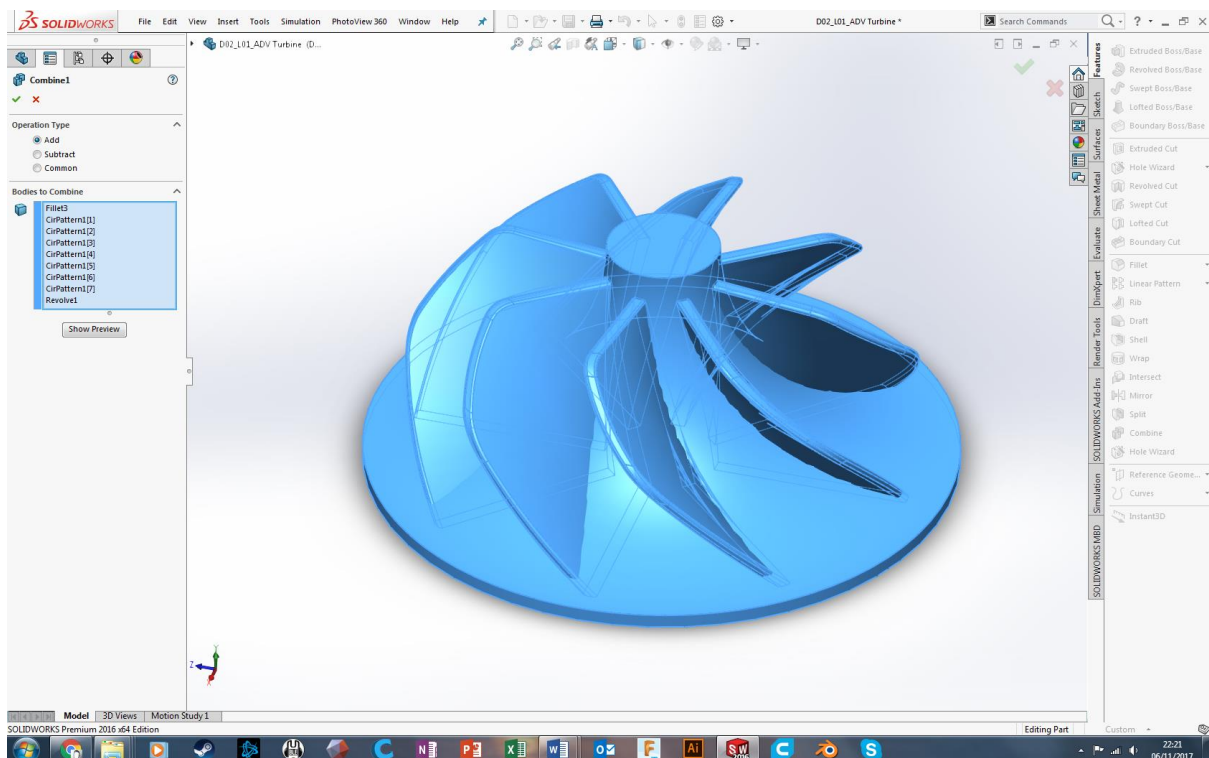


- Click **Accept** and **Save**

- Select the **Combine** found under **Features > Combine Tool** (shown below)

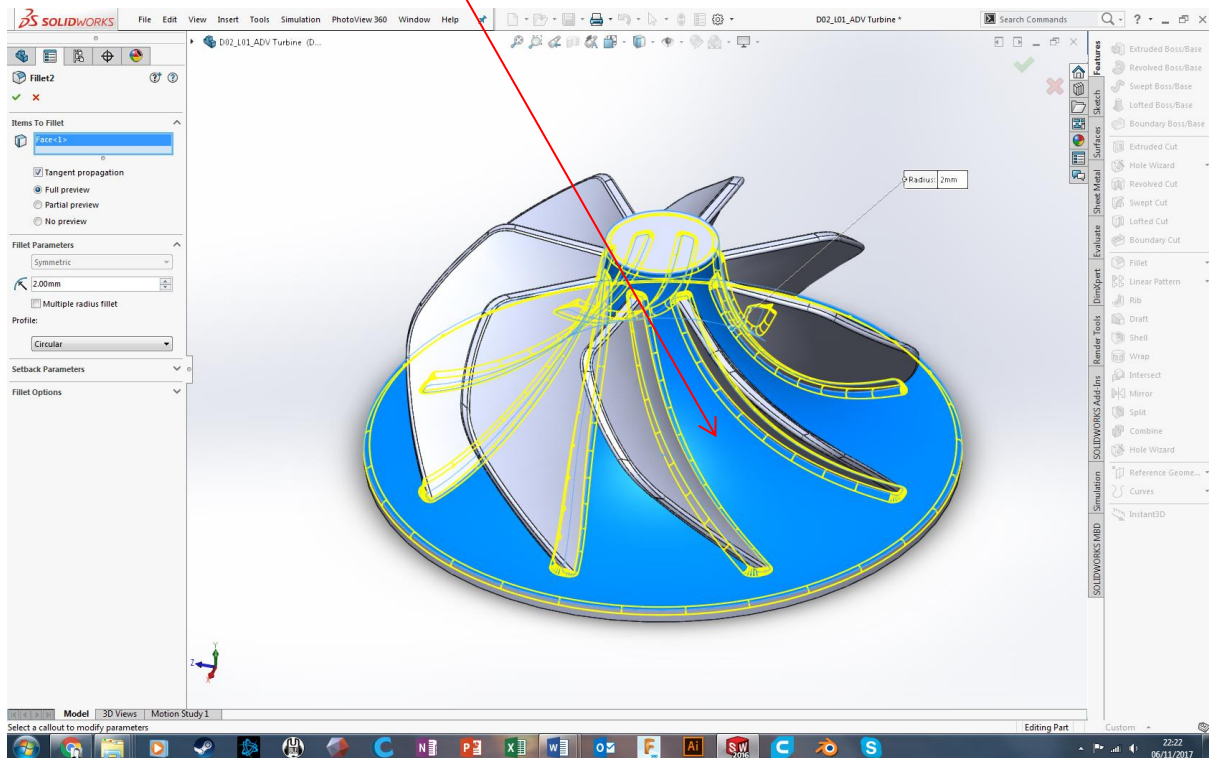


- **Select Add**
- Select all of the **Features** as the **Bodies to Combine** (as below)

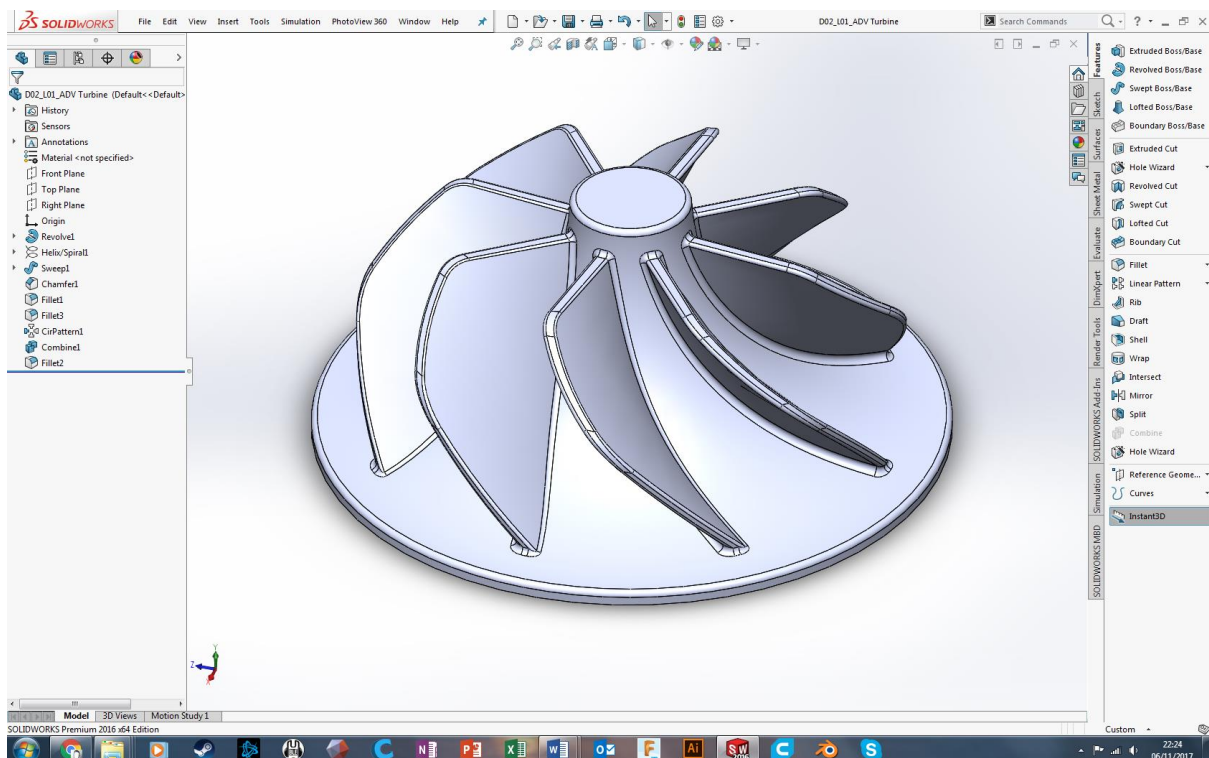


- **Click Accept**

- Use the **Fillet Tool** found under **Features**
- **Select the Surface (shown below)**
- Radius 2.00mm



- The part should look as follows:



- Click **Accept** and **Save**