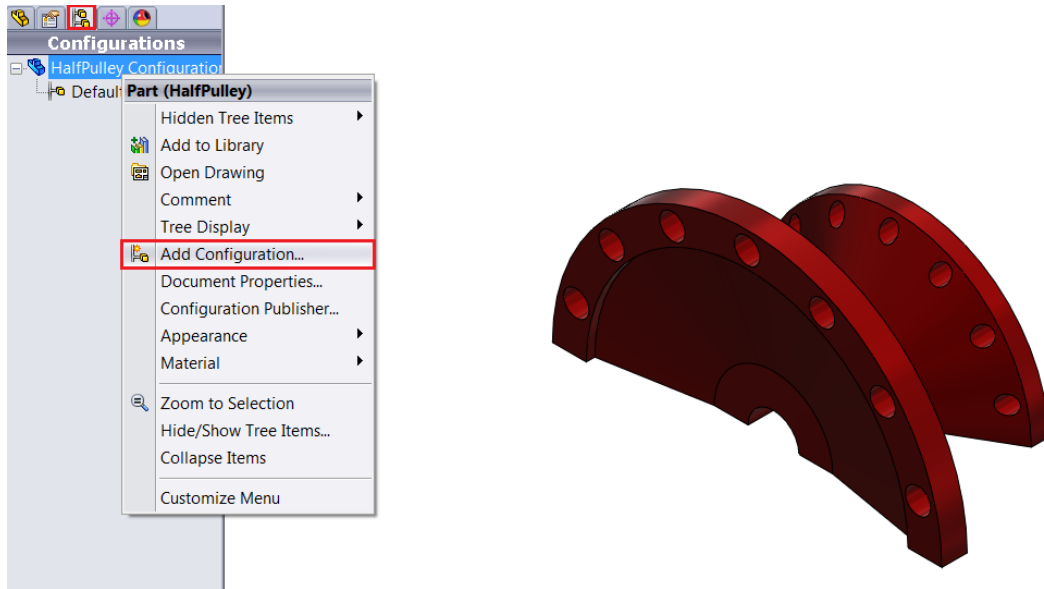# SOLIDWIZE

ONLINE SOLIDWORKS TRAINING

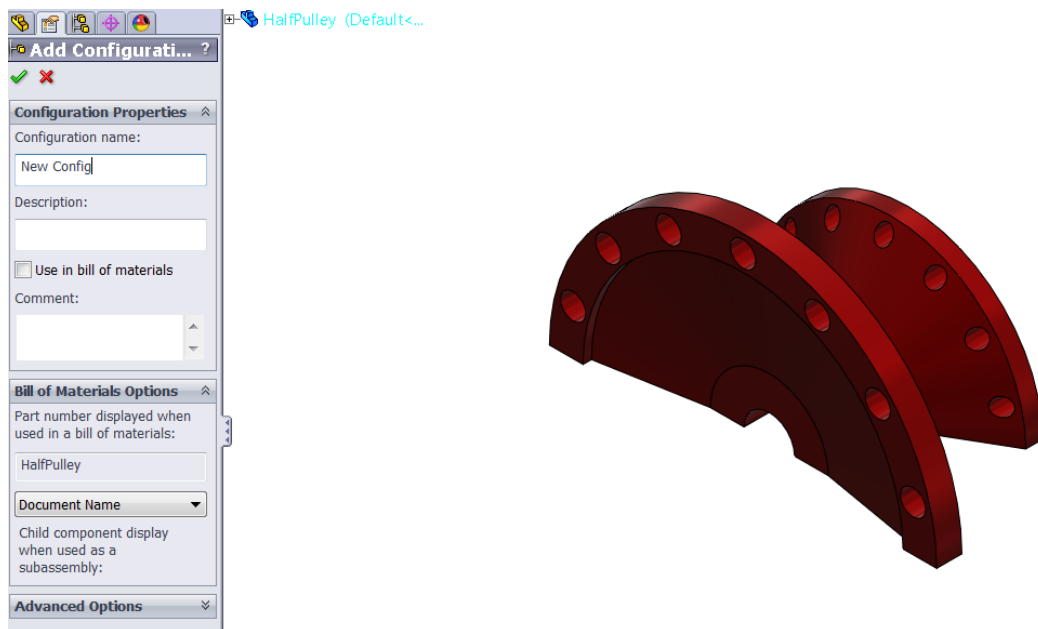Configuration: Half Pulley, Changing Dimensions

Step 1: Creating a New Configuration

Open **Revolve_HalfPulley.sldprt** which was created in the Revolve: Half Pulley exercise. The completed part can be also be downloaded accompanying this exercise.
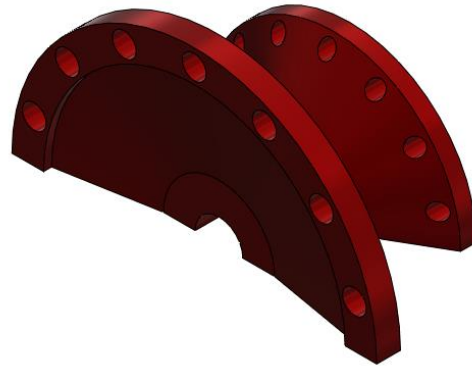
Go to the configurations tab which can be found above the property manager, as highlighted below. To add a new configuration, right click the name of the part and select **"Add Configuration."**

This will bring up the property manager for adding a new configuration. Under **Configuration Properties,** enter the configuration name **"New Config."** Select the green check mark to accept the configuration.
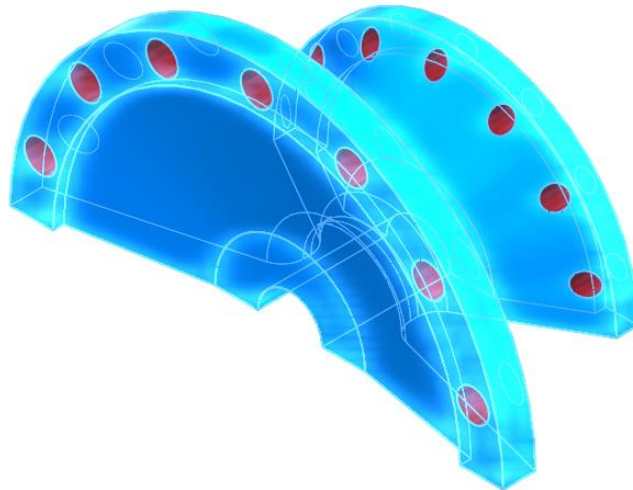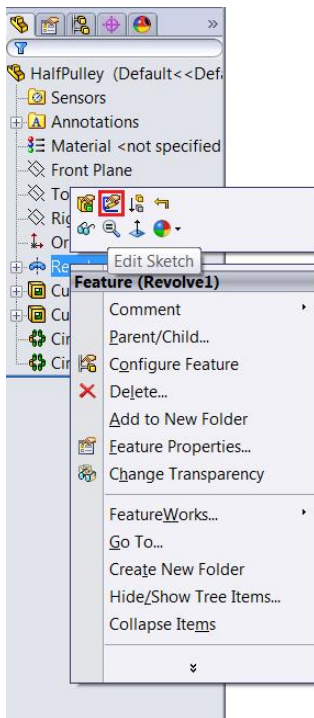
Once a new configuration has been created, it will be listed in the configuration tab under the default configuration. To toggle between configurations, double click the desired configuration.
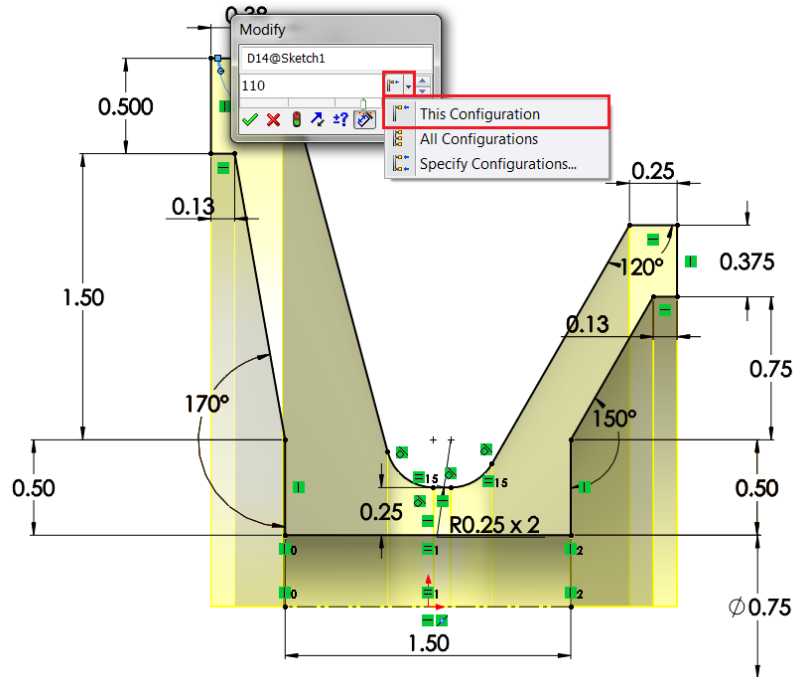

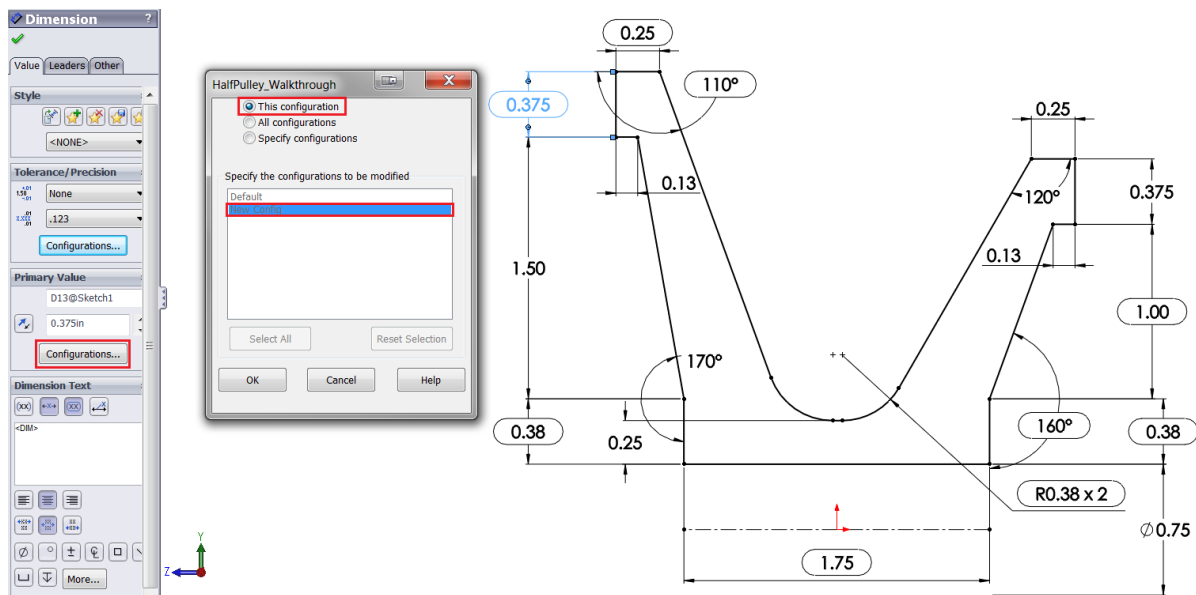**Step 2: Changing the Revolved Profile**

To open the sketch used to create the revolved profile, right click the **Revolved** feature in the feature tree and select the sketch icon shown below.
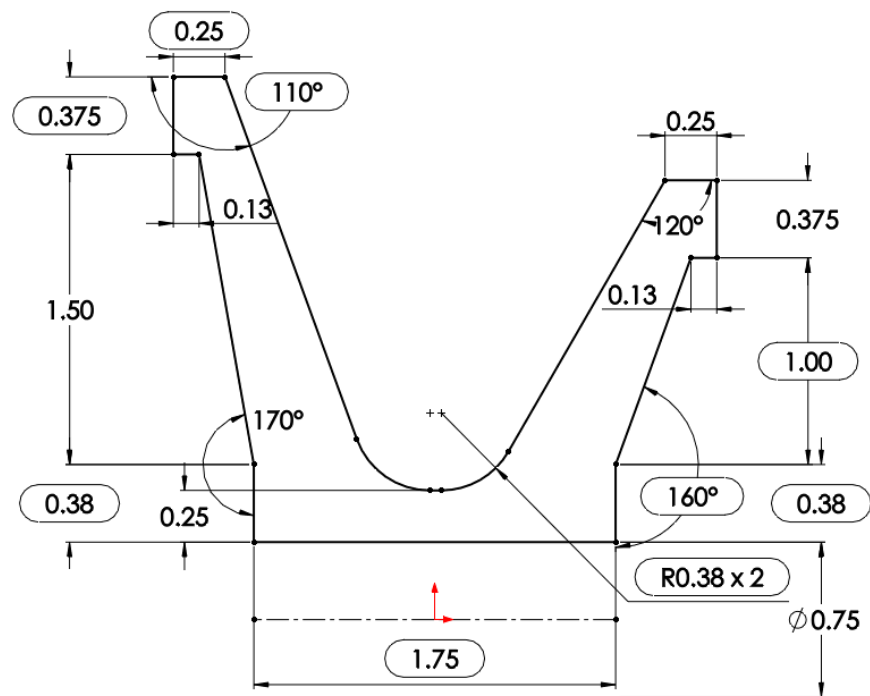
Change the original sketch dimensions to the following dimensions shown below. The dimensions that have been changed are shown with dimension bubbles. **IMPORTANT:** Before accepting each dimension, you must select which configuration the dimensions will be applied to. There are two ways to do this: 1) select the drop-down menu in the pop-up dimension box shown below:
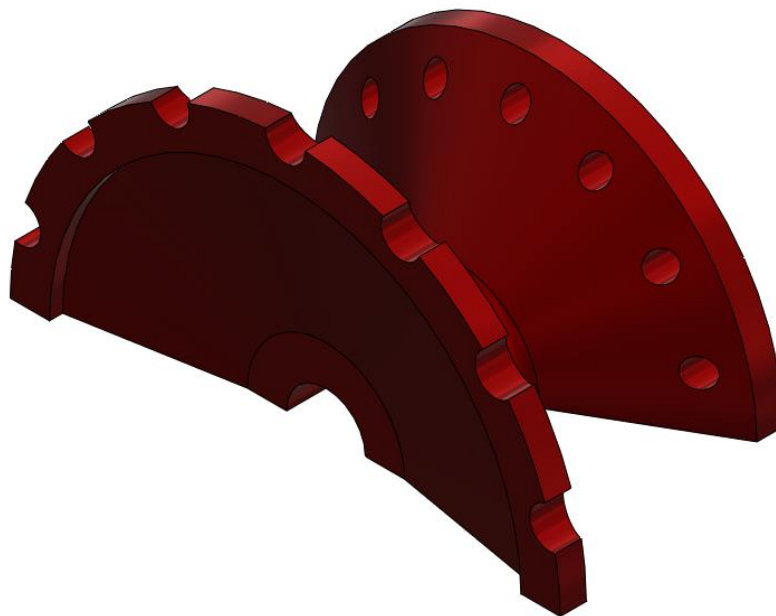

2) Select the "**Configuration**" button in the dimension property manager as shown below and select "**This Configuration**":

The following dimensions shown in bubbles are dimensions that have been changed from the original part:
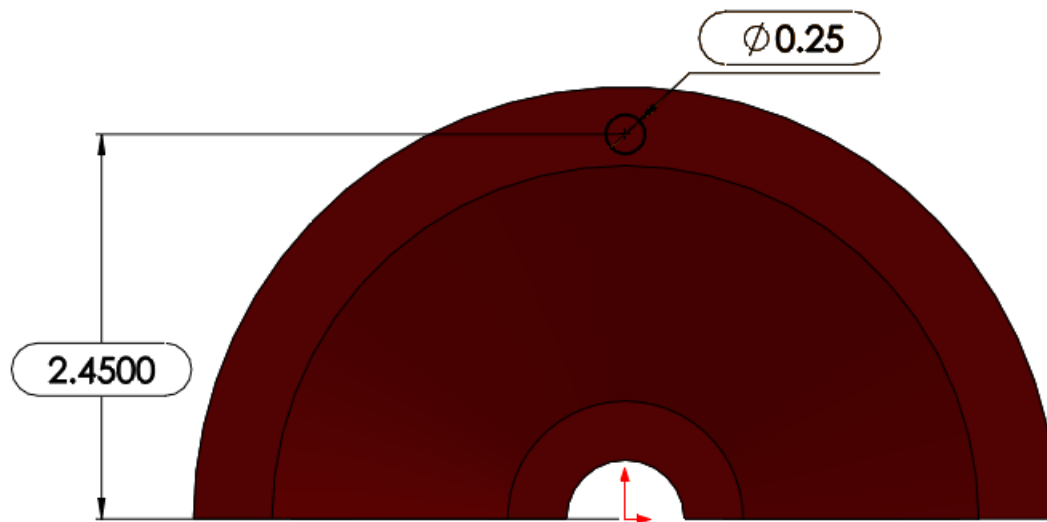

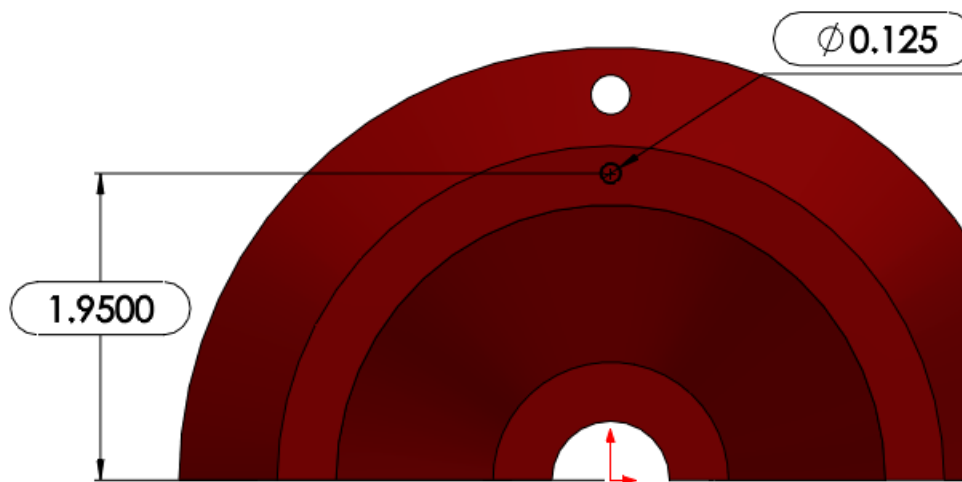Exit the sketch. The updated part will reflect the sketch changes as shown below:

Step 3: Changing the Cut Sketches

Edit the sketch for **Cut-Extrude1** and change the dimensions to the following:
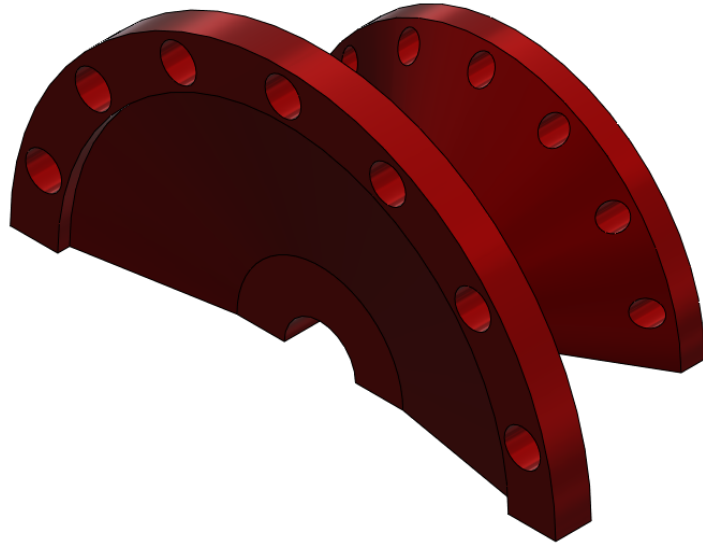

Edit the sketch for **Cut-Extrude2** and change the dimensions to the following:
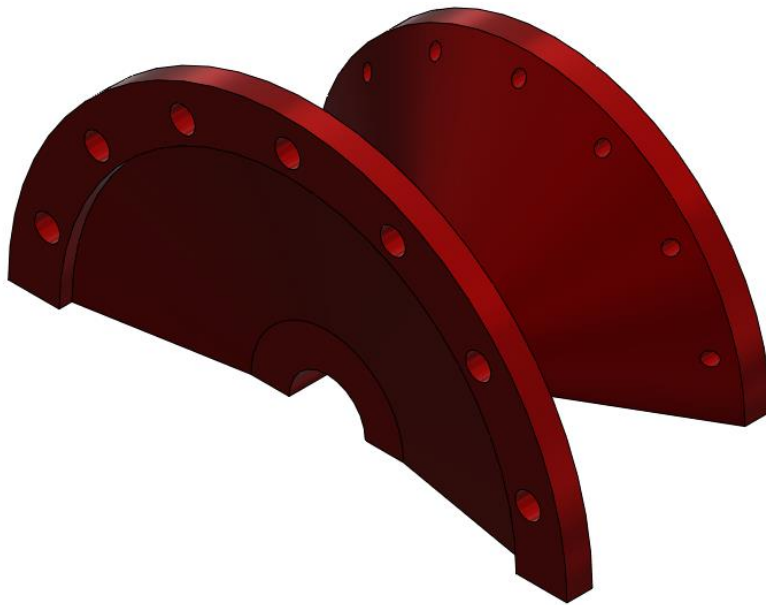
After exiting all sketches, the new configure part is shown below in comparison to the original part:
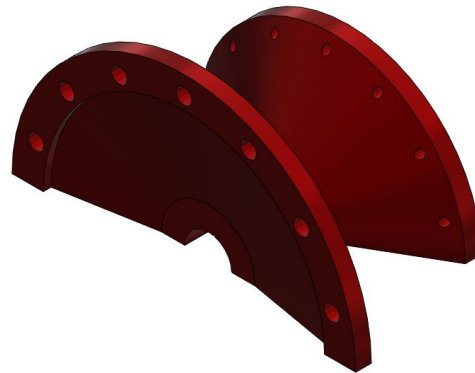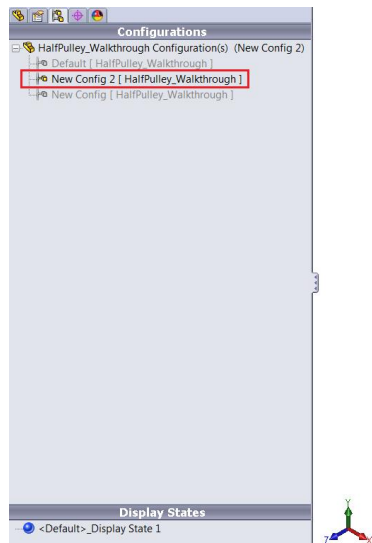
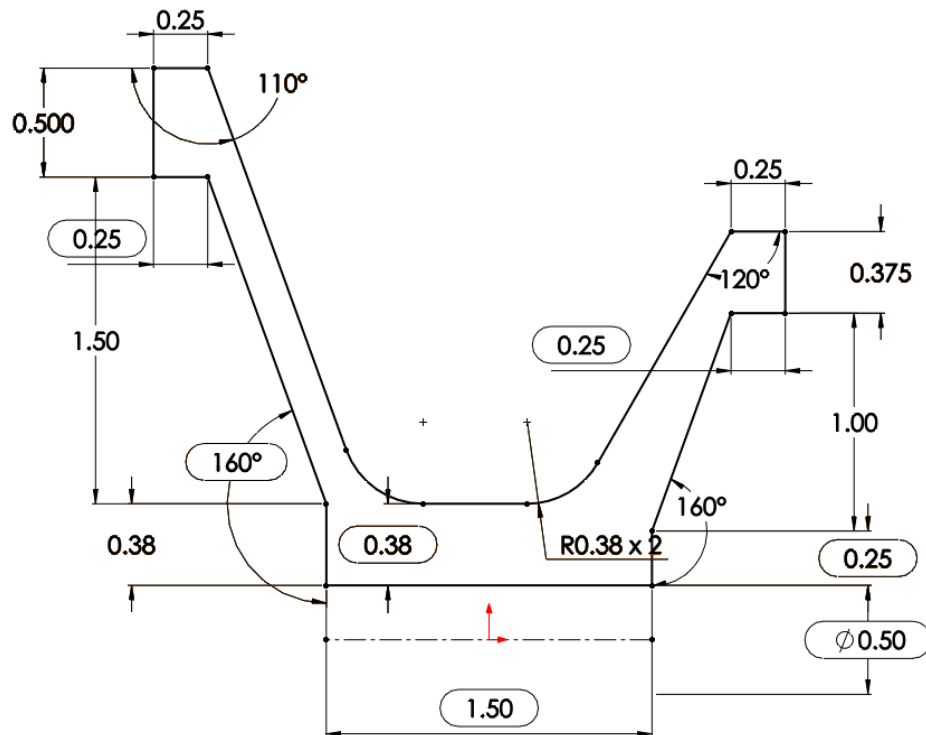**Original Part:**

New Configuration:
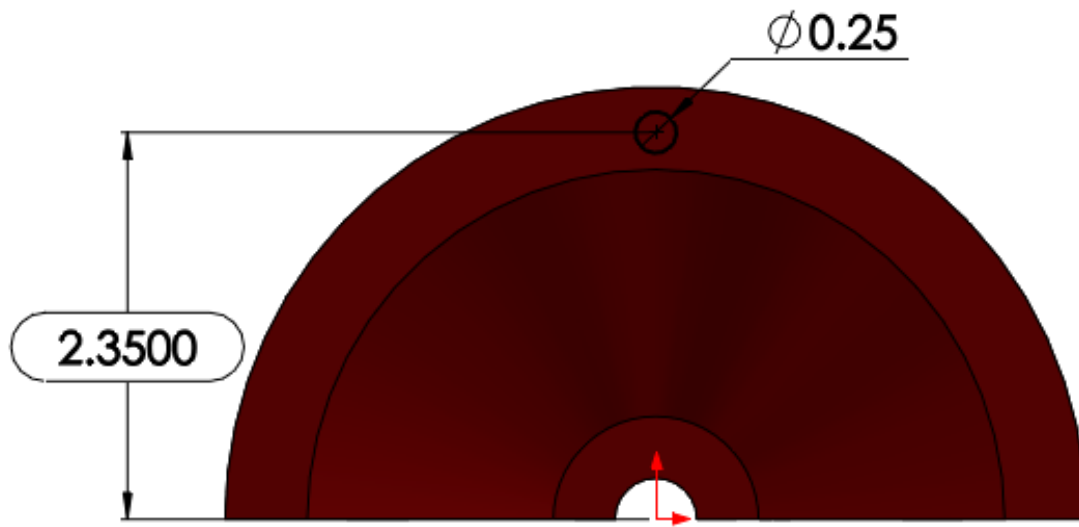
Step 4: Creating another Configuration

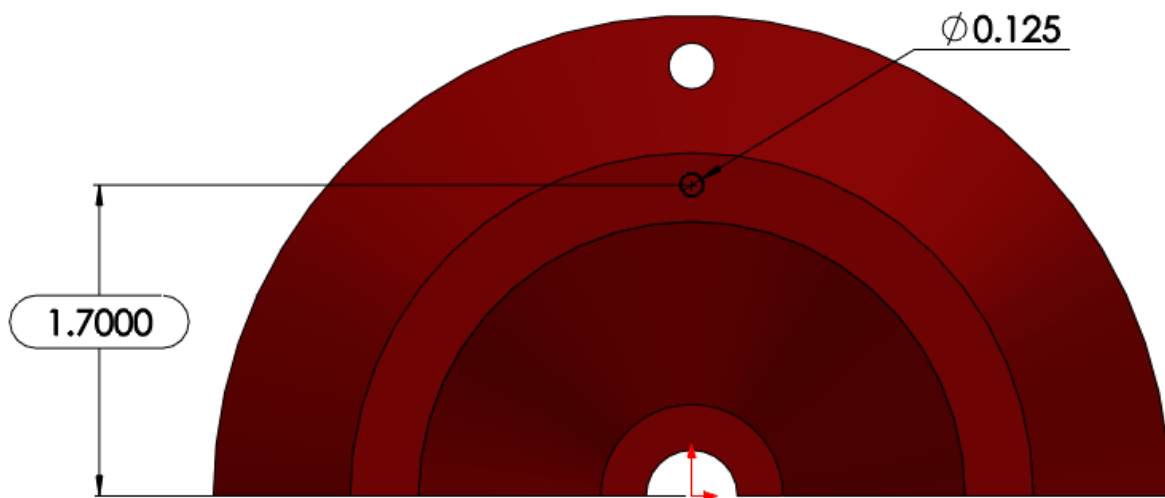Following **Step 1**, create a new configuration with the name: "**New Config 2.**"

Edit the revolved profile to the following sketch. Again, the updated dimensions are shown in dimensions bubbles. **IMPORTANT:** Make sure to select which configuration to apply the dimensions to before accepting dimensions!

Edit the sketch for **Cut-Extrude1** and change the dimensions to the following:

⌀0.25

2.3500

Edit the sketch for **Cut-Extrude2** and change the dimensions to the following:
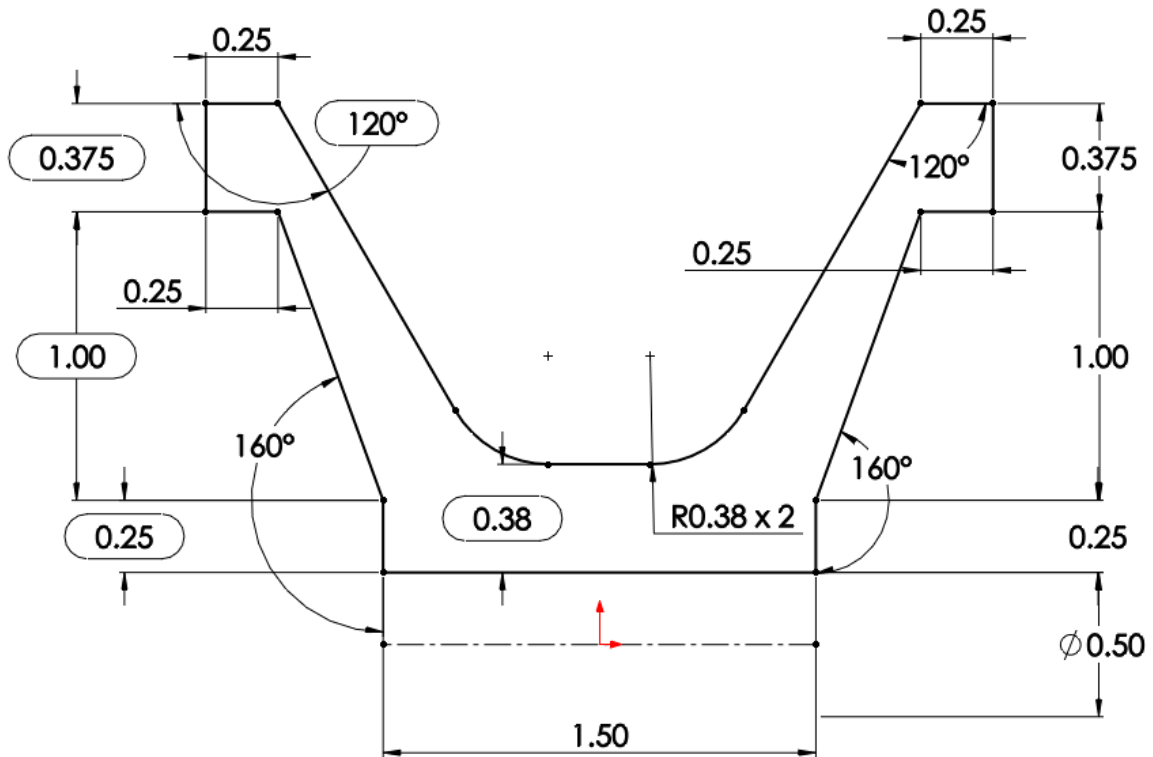
⌀0.125

1.7000

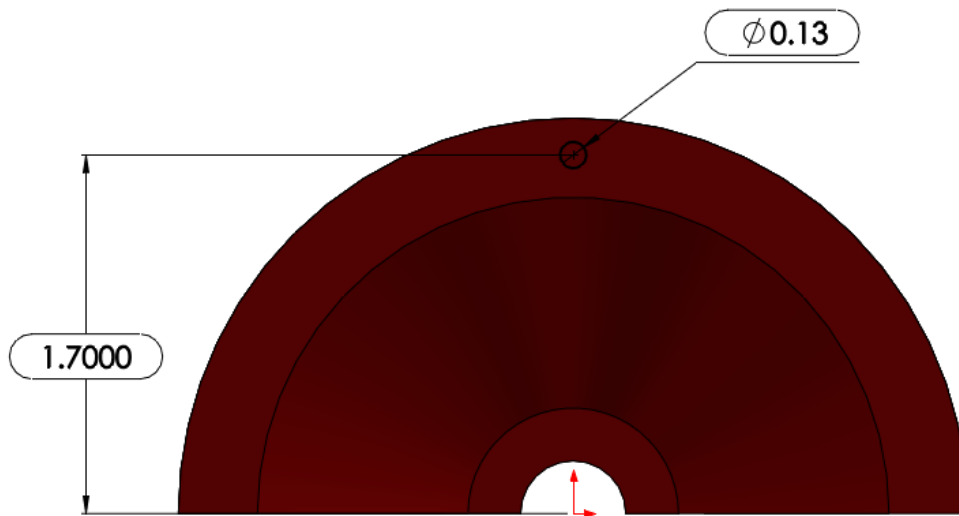The resulting configuration should look like the following:

Step 5: Creating the Last Configuration

You will create one more configuration and name it **"New Config 3."**

Edit the revolved profile sketch to the following:

Also edit the **Cut-Extrude1** sketch to the following dimensions:

The resulting configuration should look like the following:

Step 6: Save and Exit

Save the part as **Config_HalfPulley_1.sldprt** and exit the part.