Summary:

In this exercise, you'll create a Pressure Disk from a blank part file. The geometry of this part can be created from three features, a Revolve, an Extruded Cut in the shape of a hole, and a Circular Pattern.
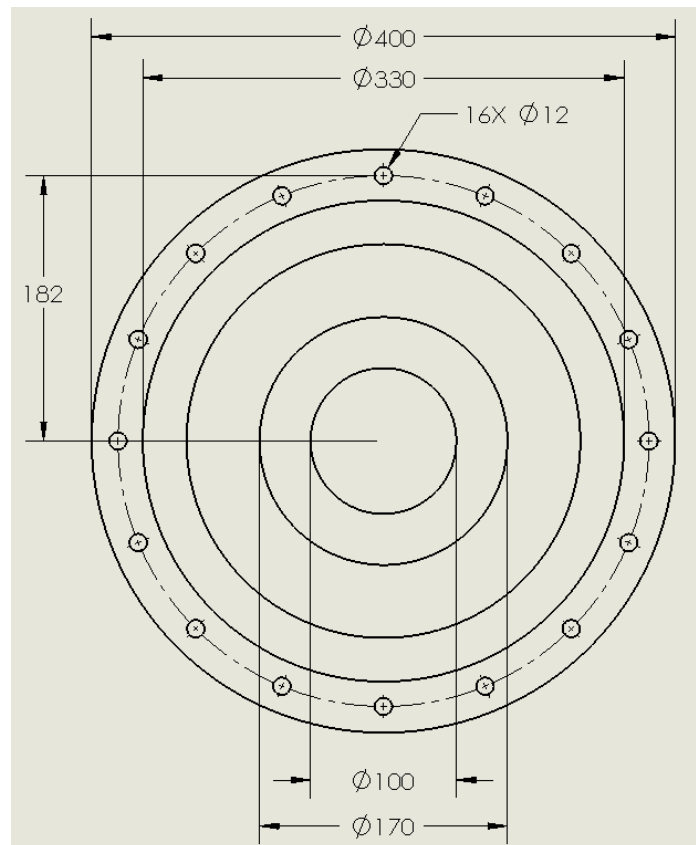
Reference Lessons:

- Revolving the Wheel Base
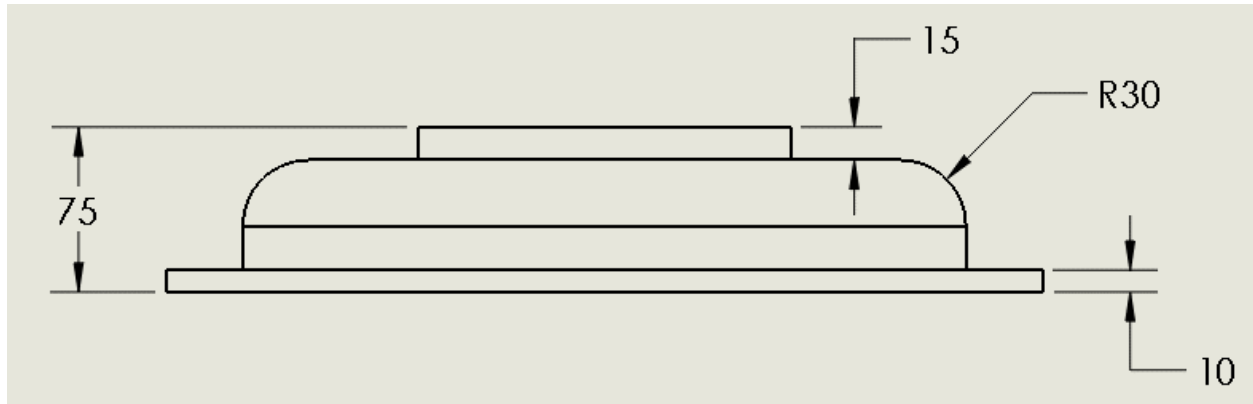- Editing the Revolved Wheel's profile
- Cutout and Circular Pattern

Instructions:

1. Open a new part file in SOLIDWORKS using millimeter units.
2. Create the geometry of the part in the images shown below.

Section view cutting through the center of the part, displaying the wall thickness: