

Exercise 22: Wheel

Create this part using the dimensions provided. Use relations and equations where applicable to maintain the design intent.

This lab uses the following skills:

- Revolved features.
- Optional: Text in a sketch.

Units: **millimeters**



Design Intent

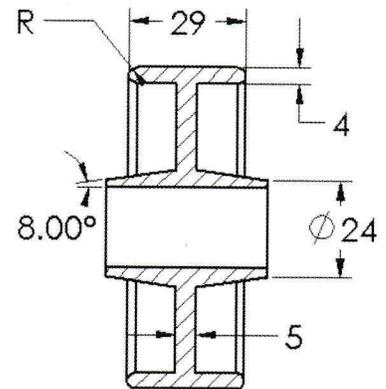
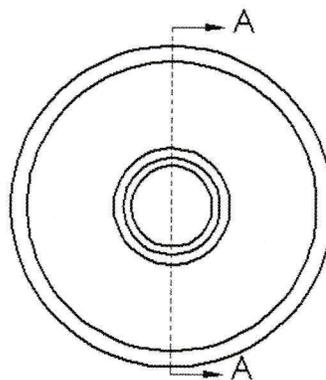
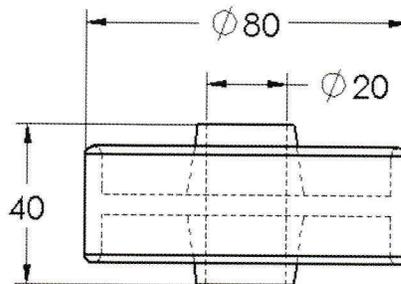
The design intent for this part is as follows:

1. Part is symmetrical about the axis of the hub.
2. Hub has draft.

Dimensioned Views

Use the following graphics with the description of the design intent to create the part.

Front and Top views, and Section A-A from Front view.



SECTION A-A

Optional: Text in a Sketch

Text can be added to a sketch and extruded to form a cut or a boss. The text can be positioned freely, located using dimensions or geometric relations, or made to follow sketch geometry or model edges.

Introducing: Text Tool

The text tool allows you to insert text into a sketch and use it to create an extruded boss or cut feature. Since SolidWorks software is a true Windows application, it supports whatever fonts you have installed on your system.

Where to Find It

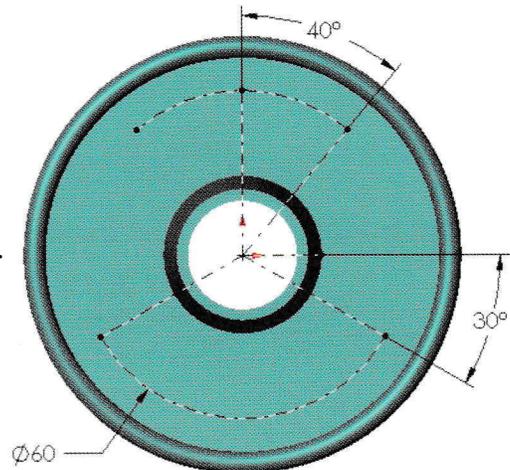
- Click **Tools, Sketch Entity, Text**.
- Or click the  tool on the Sketch Tools toolbar.

1 Construction geometry.

Sketch on the front face and add construction lines and arcs as shown.

Tip

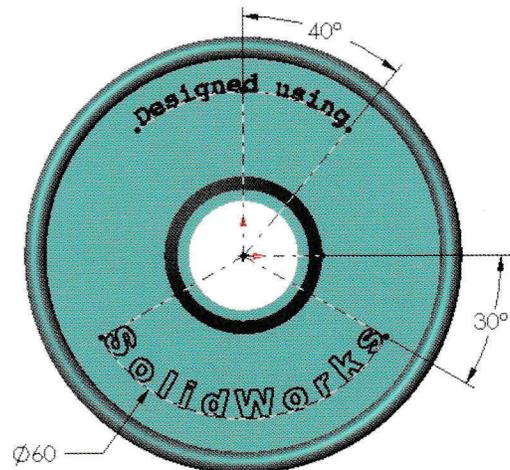
Use **Symmetric** relationships between the endpoints of the arcs and the vertical centerline.

**2 Text on a curve.**

Create two pieces of text, one attached to each arc. They have the following properties:

- **Text:** Designed using
- **Font:** Courier New 14pt
- **Alignment:** Center Align
- **Width Factor:** 100%
- **Spacing:** 100%

- **Text:** SolidWorks
- **Font:** Arial Black 20pt.
- **Alignment:** Full Justify
- **Width Factor:** 100%
- **Spacing:** not applicable when using Full Justify

**3 Extrude.**

Extrude a boss with a **Depth** of 1mm and **Draft** of 1°.

4 Save the part and close it.