

Exercise 23: Flange

Create this part using the dimensions provided. Use relations wisely to maintain the design intent.

This lab uses the following skills:

- *Revolved Features* on page 195.

Units: **millimeters**



Design Intent

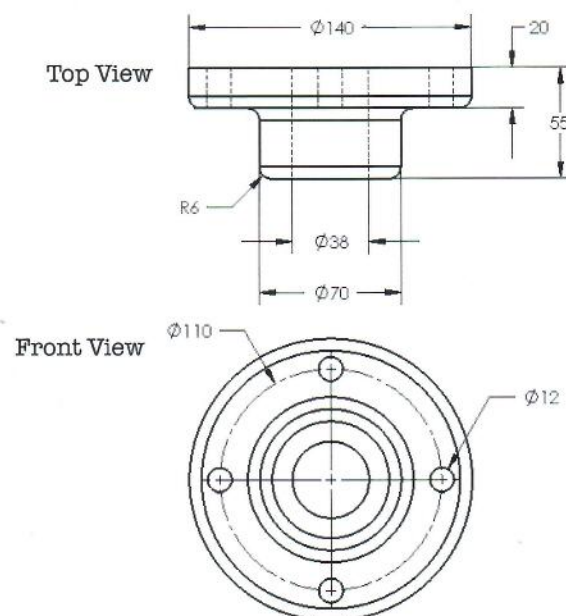
The design intent for this part is as follows:

1. Holes in the pattern are equally spaced.
2. Holes are equal diameter.
3. All fillets are equal and are **R6mm**.

Note that construction circles can be created using the **Properties** of a circle.

Dimensioned Views

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



Exercise 24:
Wheel

Create this part using the dimensions provided. Use relations wisely to maintain the design intent.

This lab uses the following skills:

- *Revolved Features* on page 195.

Units: **millimeters**

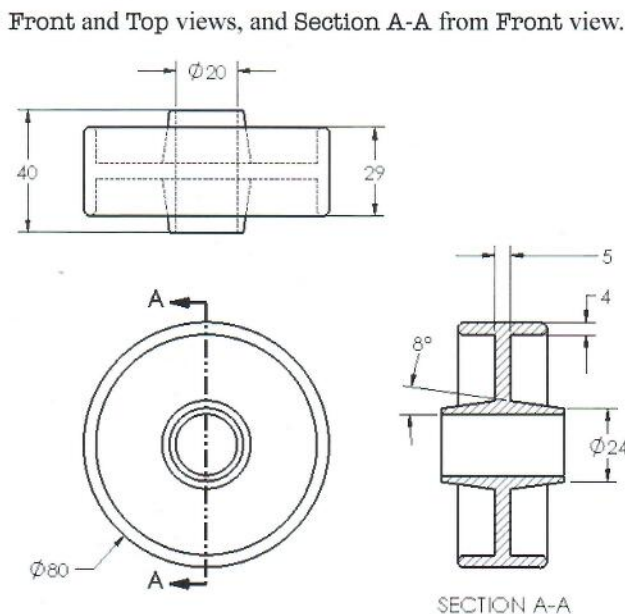
**Design Intent**

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.



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.

Where to Find It

- CommandManager: **Sketch > Text** 
- Menu: **Tools, Sketch Entities, Text**
- Shortcut Menu: Right-click in the graphics area and click **Text** 

Tip

1 Construction geometry.

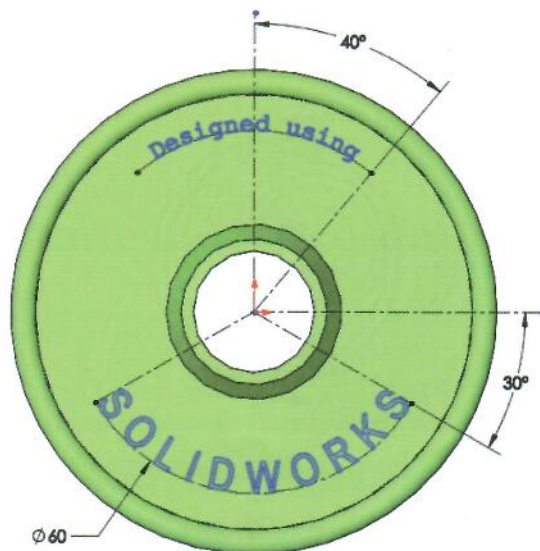
Sketch on the front face and add construction lines and arcs as shown.

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 11pt**
- Alignment: **Center Align**
- Width Factor: **100%**
- Spacing: **100%**
- Text: **SOLIDWORKS**
- Font: **Arial 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°.

**Note**

Extruding text can be time consuming.

- 4 **Save the part and close it.**
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Exercise 25: Guide

Create this part using the information and dimensions provided. This lab reinforces the following skills:

- *Introducing: Slots on page 202.*

Units: **millimeters**



Procedure

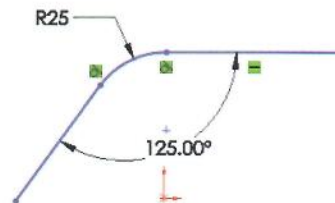
Create a new mm part and name it Guide. Create the geometry as shown in the following steps.

Note

These images show the sketch relations (**View, Sketch Relations**) for clarity.

1 Lines and fillet.

Open a sketch on the Front plane. Create the sketch lines, a sketch fillet and an angular dimension as shown.



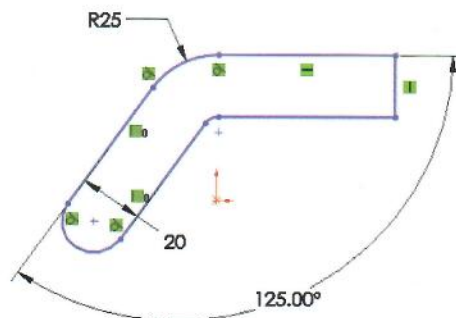
2 Offset.

Use offset entities to create the **20mm** offset as shown.



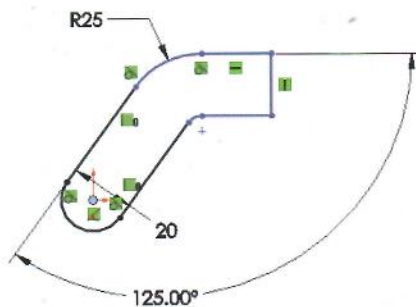
3 Close ends.

Close the ends using a tangent arc and a line as shown.



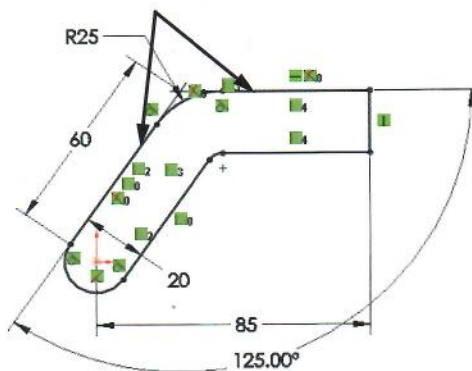
4 Drag to origin.

Drag the centerpoint of the arc to the Origin and drop it. This creates a Coincident relation.

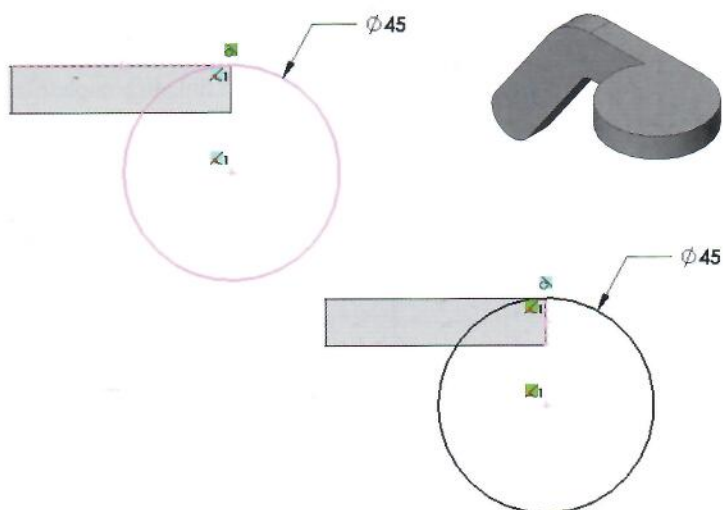
**5 Fully defined.**

Add a Virtual Sharp by selecting the two lines as shown and clicking **Point** . Complete the sketch by adding dimensions as shown.

Extrude the sketch **10mm**.

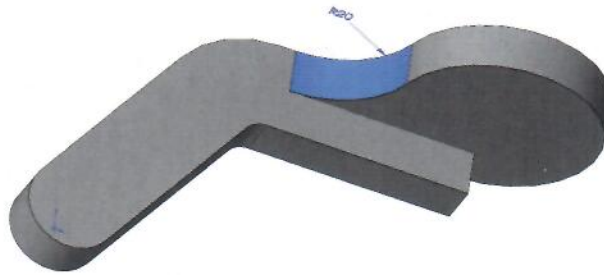
**6 Circle and boss.**

Add a circle to a new sketch on the top face of the model. Use **Tangent** and **Coincident** relations to relate the circle to the geometry. Fully define and extrude the sketch **10mm** as shown.



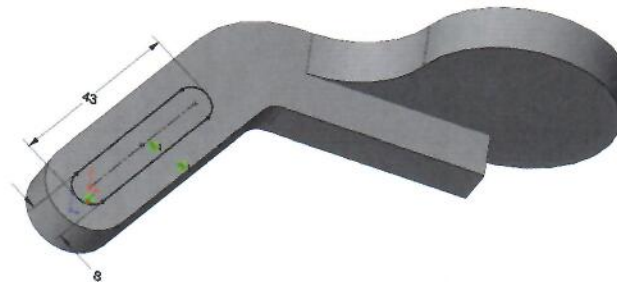
7 Fillet.

Add a fillet **R20mm** as shown.



8 Slot.

Use **Straight Slot** with the options **Overall Length** and **Add Dimensions** to create the geometry shown below. Create a through all cut with the sketch geometry.

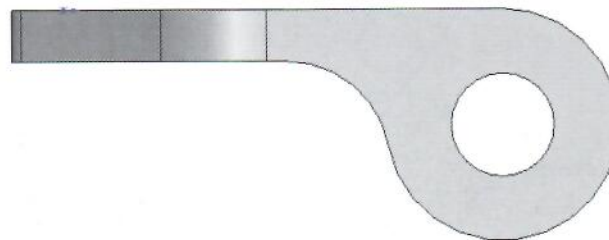


Tip

The slot sketch should be fully defined. It may require a **Parallel** relation.

9 Hole.

Add a **20mm** through all hole to complete the part.



10 Save and close.

**Exercise 26:
Tool Post**

Create this part using the dimensions provided. Use relations wisely to maintain the design intent.

This lab uses the following skills:

- *Revolved Features* on page 195.

Units: **millimeters**

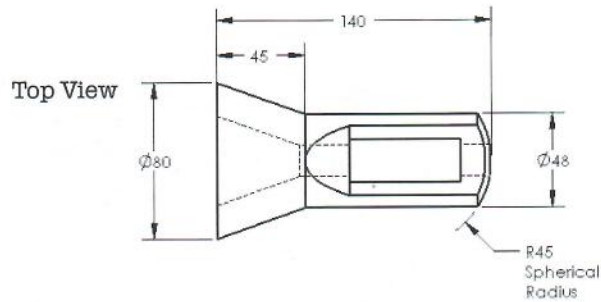
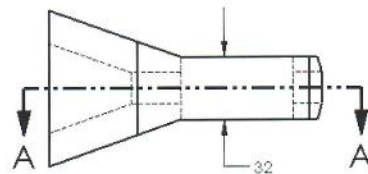
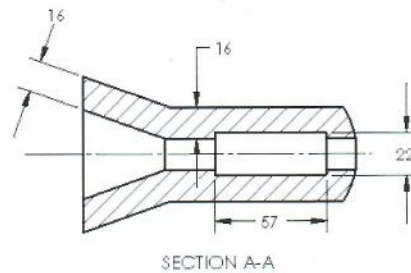
**Design Intent**

The design intent for this part is as follows:

1. Part is symmetrical.
2. Center hole is through all.

Dimensioned Views

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

**Front View****Section A-A**

Exercise 27: Ellipse

Use an ellipse to create the part.
This lab reinforces the following skills:

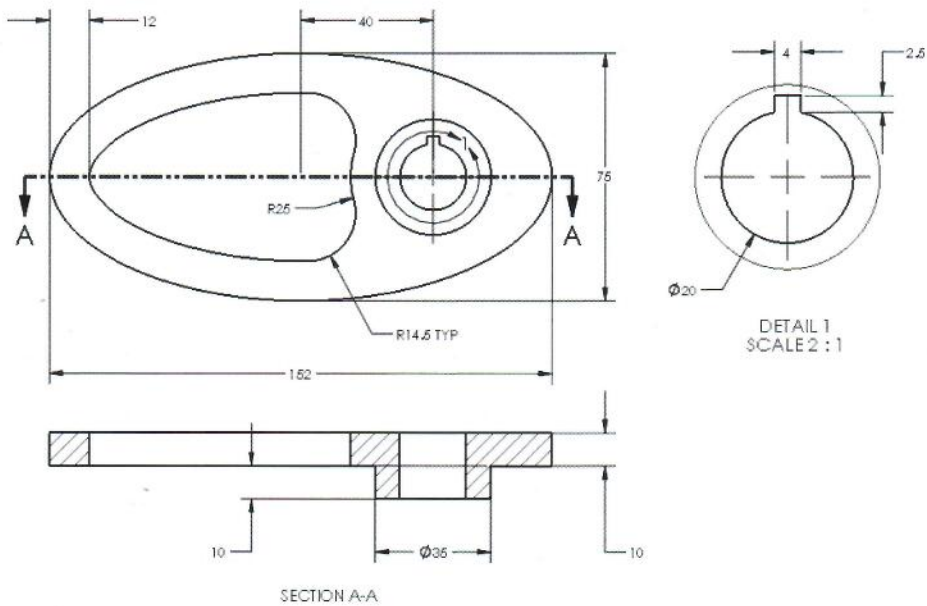
- *Introducing: Insert Ellipse on page 207.*

Units: **millimeters**



Procedure

Create a new mm part. Create the geometry as shown in the following drawing.



**Exercise 28:
Sweeps**

Create these three parts using swept features. These require a path and a section.

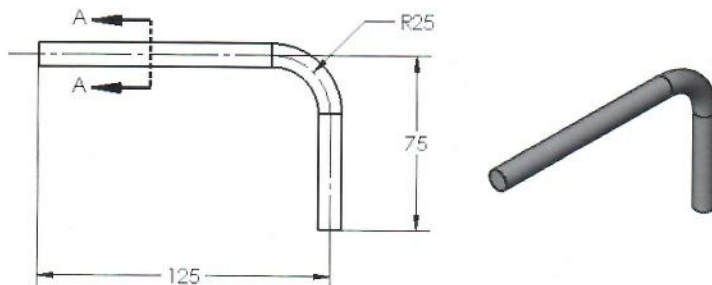
This lab uses the following skills:

- *Introducing: Sweep* on page 208.

Units: **millimeters**

Slide Stop

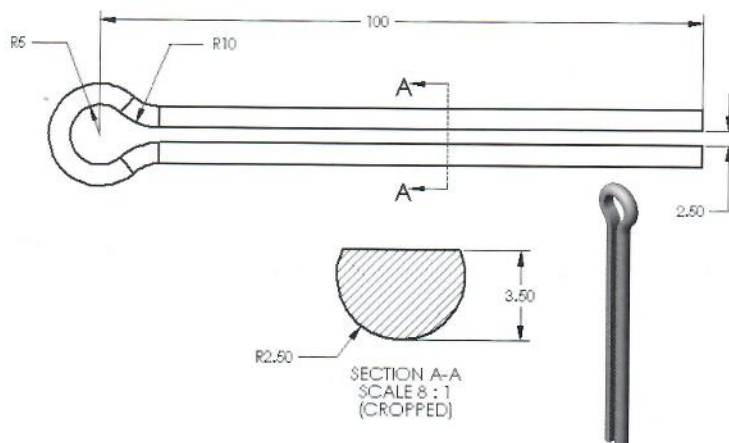
The Slide Stop uses a path that describes the centerline of the sweep.



SECTION A-A
SCALE 1 : 1

Cotter Pin

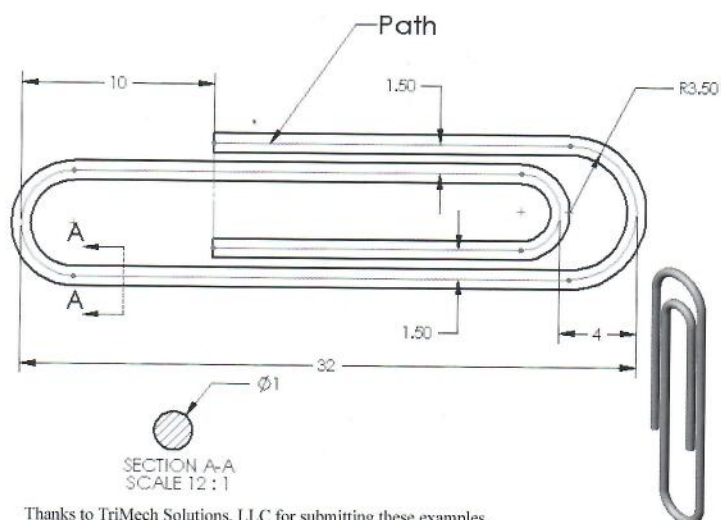
The Cotter Pin uses a path that describes the inner edge of the sweep.



SECTION A-A
SCALE 8 : 1
(CROPPED)

Paper Clip

The Paper Clip is defined by a path that describes the centerline of the sweep.



Thanks to TriMech Solutions, LLC for submitting these examples.

Mitered Sweep

The Mitered Sweep is defined by a path that describes the outer edge of the sweep.

