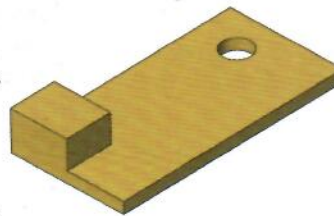


Exercise 7: Plate

Create this part using the information and dimensions provided. Sketch and extrude profiles to create the part. This lab reinforces the following skills:

- *Choosing the Best Profile* on page 62.
- *Introducing: Corner Rectangle* on page 67.
- *Sketching on a Planar Face* on page 69.
- *Boss Feature* on page 69.
- *Using the Hole Wizard* on page 74.

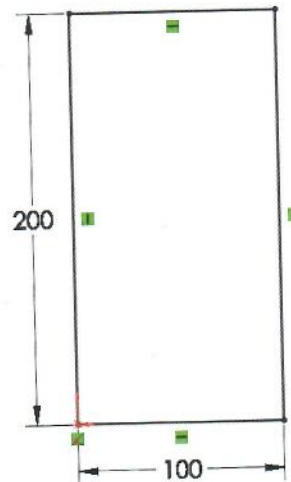


Units: **millimeters**

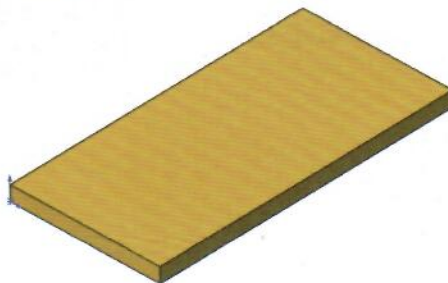
Procedure

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

- 1 **Sketch base feature.**
Create a new sketch on the Top plane. Add the geometry and dimensions as shown.



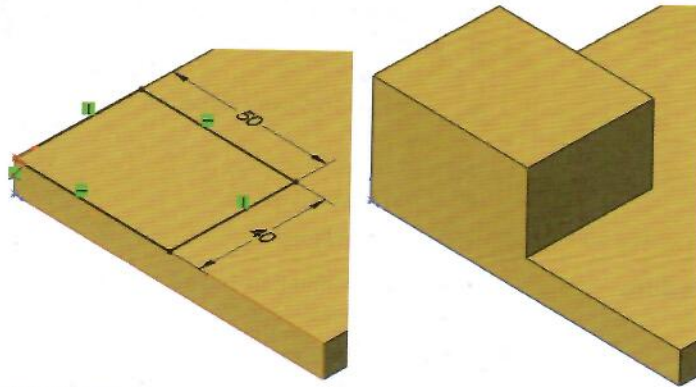
- 2 **Extrude base feature.**
Extrude the sketch **10mm** as shown.



3 Boss.

Create a new sketch on the top face of the solid. Add the geometry and dimensions as shown.

Extrude a boss **25mm**.

**4 Hole Wizard.**

Click **Hole Wizard**  and click the face shown.

Click the **Type** tab. Set the properties of the hole as follows:

Type: Hole

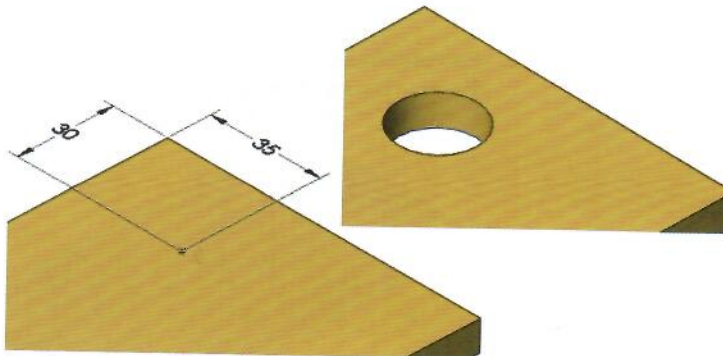
Standard: Ansi Metric

Type: Drill sizes

Size: 25mm

End Condition: Through All

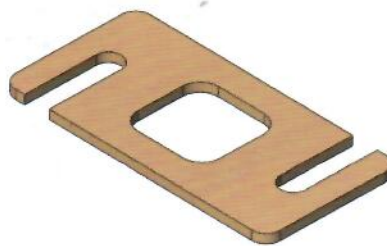
Click the **Positions** tab. Place the points as shown.

**5 Save and close the part.**

Exercise 8: Cuts

Use rectangles, tangent arcs and cut features to create the part. This lab reinforces the following skills:

- *Introducing:*
Corner Rectangle on page 67.
- *Tangent Arc Intent Zones* on page 70.
- *Cut Feature* on page 72.
- *Filleting* on page 75.



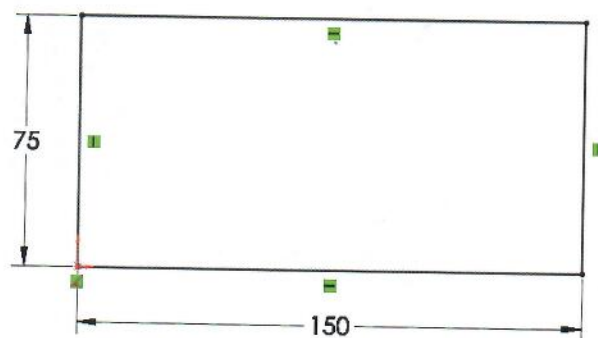
Units: **millimeters**

Procedure

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

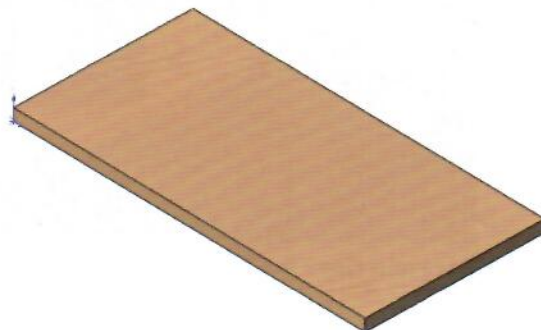
1 Sketch base feature.

Create a new sketch on the Top plane. Add the geometry and dimensions as shown.



2 Extrude base feature.

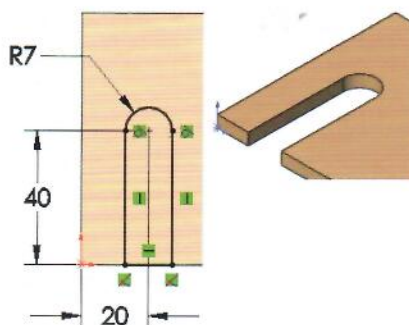
Extrude the sketch **5mm** as shown.



3 Cut slot.

Create a new sketch on the top face of the solid. Add the geometry and dimensions as shown.

Extrude a cut using **Through All**.

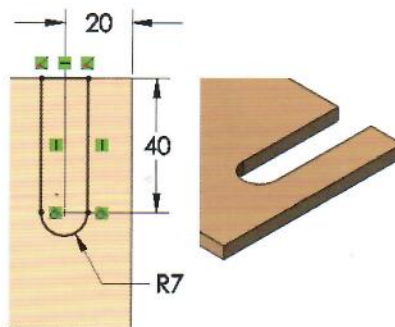
**Tip**

Remember to create a closed profile by sketching the line across the bottom.

4 Cut another slot.

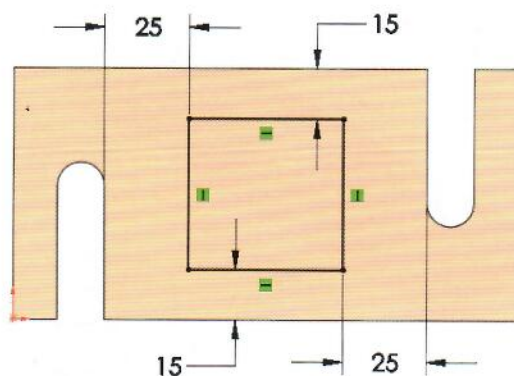
Create a new sketch using the same face. Add the geometry and dimensions as shown.

Extrude another cut using **Through All**.

**5 Cut rectangle.**

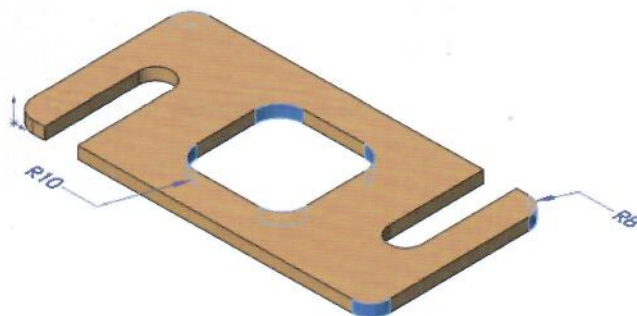
Create a new sketch using the same face. Add the geometry and dimensions as shown.

Extrude another cut using **Through All**.



6 Fillets.

Add fillets of **R10mm** and **R8mm** to the edges as shown.



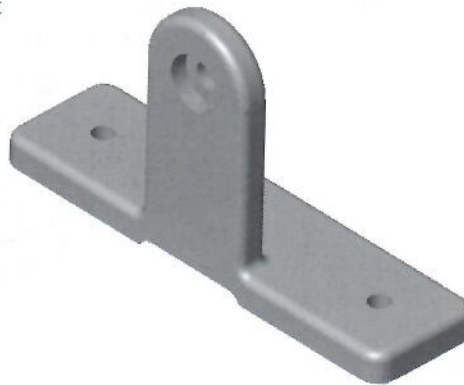
7 Save and close the part.

**Exercise 9:
Basic-Changes**

Make changes to the part created in the previous lesson.

This exercise uses the following skills:

- *Changing Parameters* on page 97.
- *Rebuilding the Model* on page 97.

**Procedure**

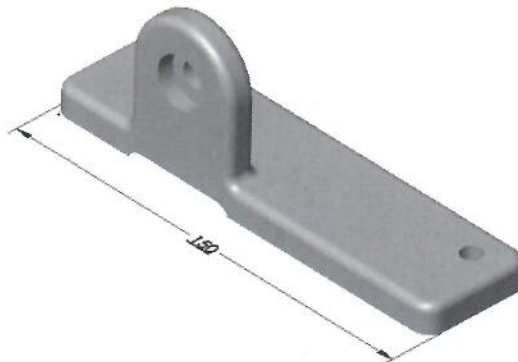
Open an existing part in the Lesson03\Exercises folder.

**1 Open the part
Basic-Changes.**

Several changes will be performed on the model to resize it and check the design intent.

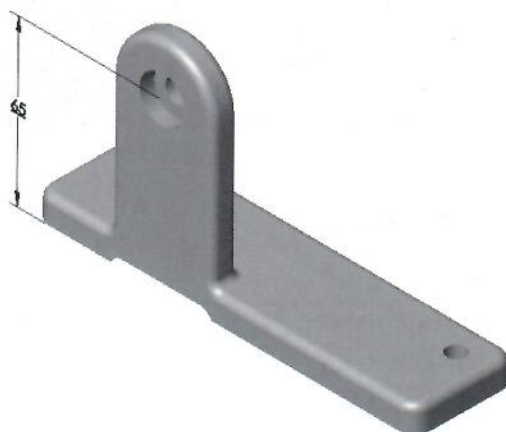
**2 Overall dimension.**

Double-click the first feature (**Base Plate**) in the FeatureManager design tree or on the screen to access the dimensions. Change the length dimension to **150mm** (shown bold and underlined below) and rebuild the model.



3 Boss.

Double-click the **Vert boss** feature and change the height dimension as shown. Rebuild the part.



4 Hole locations.

Double-click the **Ø7.0 (7) Diameter Hole1** feature and change the position dimensions to **20mm**. Rebuild the model.



5 Center the Vert Boss.

Determine the proper value and change the dimension that centers the **Vert Boss** on the base.

Tip

Optionally, you can delete the dimension and add a relations that centers the **VertBoss** relative to the base.



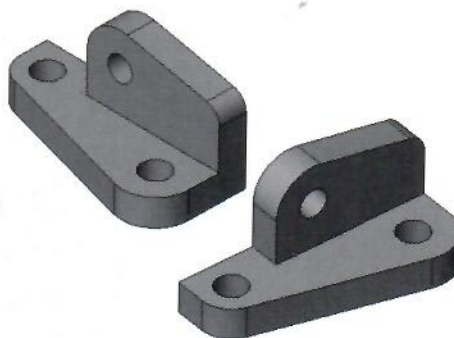
6 Save and close the part.

**Exercise 10:
Base Bracket**

This lab reinforces the following skills:

- *Choosing the Best Profile* on page 62.
- *Boss Feature* on page 69.
- *Using the Hole Wizard* on page 74.
- *Filleting* on page 75.

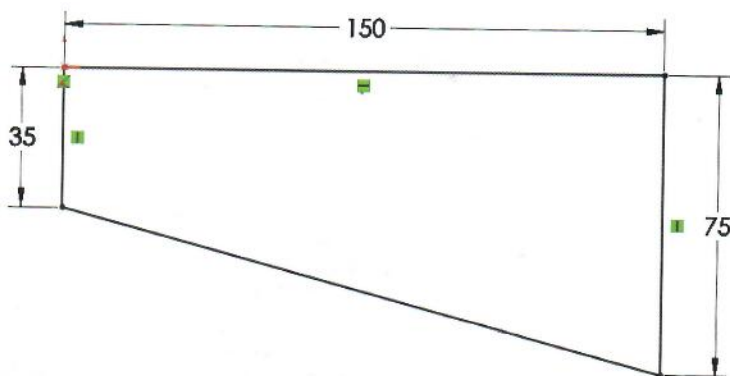
Units: **millimeters**

**Procedure**

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

1 Sketch base feature.

Create a new sketch on the Top plane. Add the geometry and dimensions as shown.

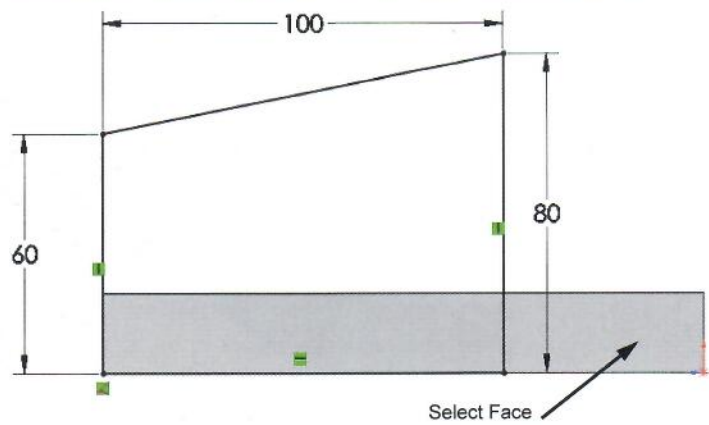
**2 Extrude base feature.**

Extrude the sketch **20mm** to create the base feature as shown.



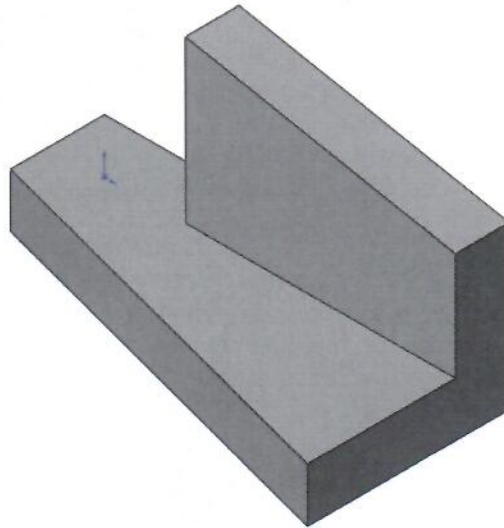
3 Sketch on rear face.

Change to the **Back** view orientation, select the face indicated and create a new sketch. Add the geometry and dimensions as shown.



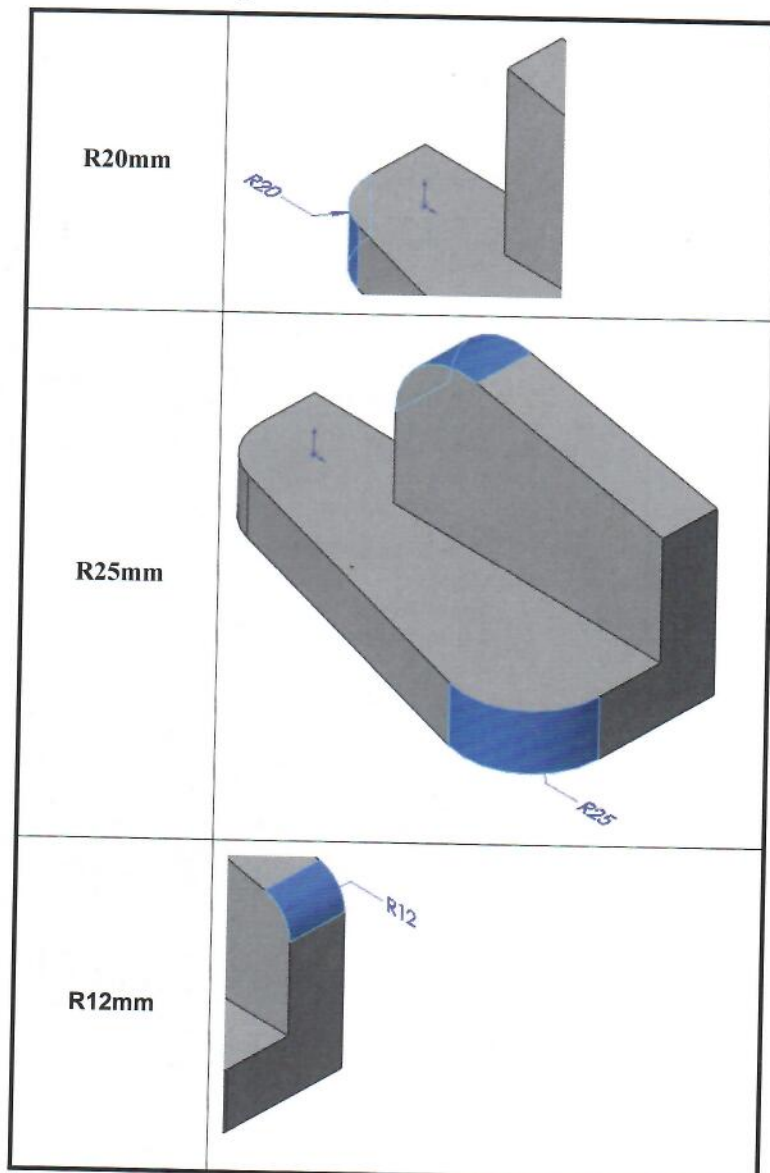
4 Extrude boss.

Extrude to the sketch **20mm** as shown.



5 Fillets.

Add fillets to the edges as shown.



6 Hole Wizard.

Click **Hole Wizard**  and click the face shown. Click the **Type** tab and set the properties of the hole as follows:

Type: Hole

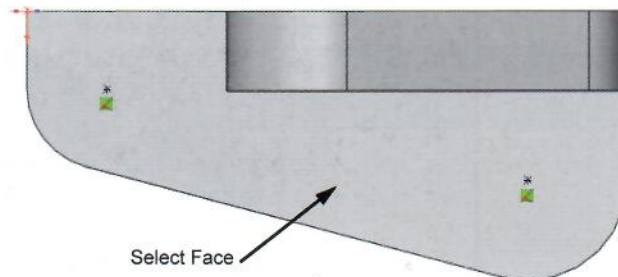
Standard: Ansi Metric

Type: Drill sizes

Size: 20mm

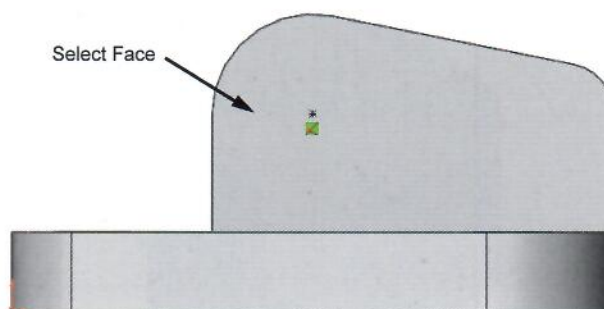
End Condition: Through All

Click the **Positions** tab and locate the holes as shown.



7 Second hole.

Repeat the procedure to create an **18mm** hole on a different face as shown.



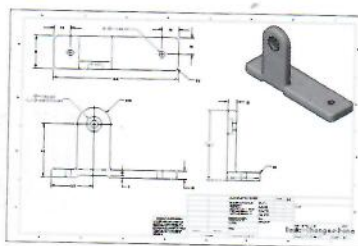
8 Save and close the part.

**Exercise 11:
Part Drawings**

Create this part drawing using the information provided.

This lab reinforces the following skills:

- *New Drawing* on page 87.
- *Drawing Views* on page 88.
- *Center Marks* on page 91.
- *Dimensioning* on page 92.

**Procedure**

Create a new drawing and add the views and dimensions shown in the following steps.

1 Open part.

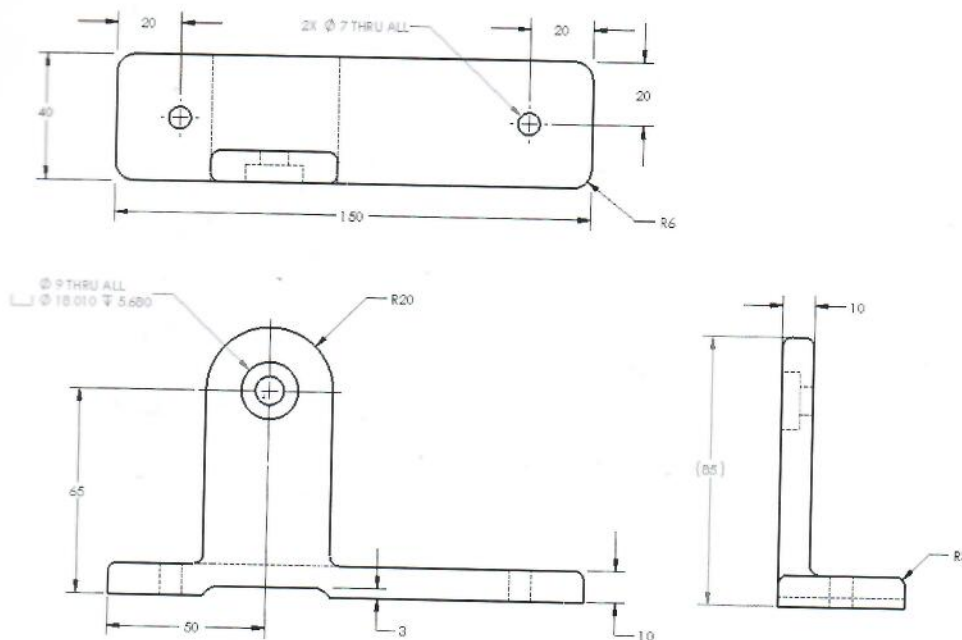
Open the part *Basic-Changes-done* from the *Lesson 3/Exercises* folder.

2 New drawing.

Use the **Make Drawing from Part** command and the *B_Size_ANSI_MM* template to create the drawing views as shown.

3 Dimensions.

Add the annotations and dimensions as shown.

**4 Save and close all files.**