Active Learning Exercises — Creating a Basic Part

Use SolidWorks to create the box shown at the right.

The step-by-step instructions are given below.

Create a New Part Document

1 Create a new part. Click **New** on the Standard toolbar.

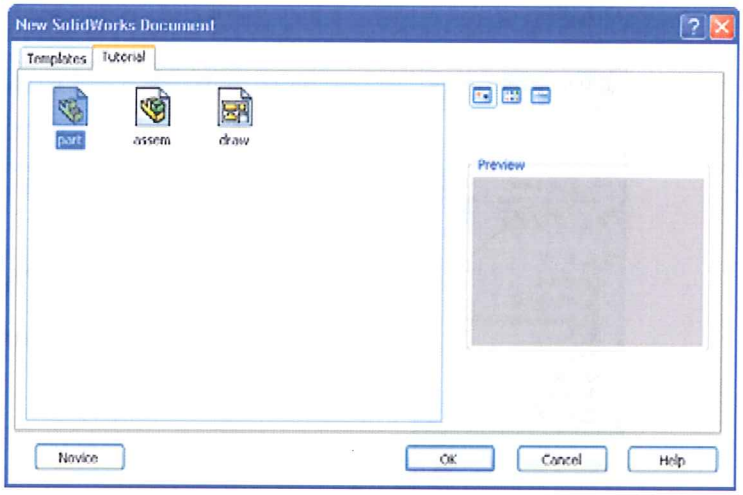
The **New SolidWorks Document** dialog box appears.

2 Click the **Tutorial** tab.

3 Select the **Part** icon.

4 Click **OK**.

A new part document window appears.



Base Feature

The Base feature requires:

- ❑ Sketch plane – `Front` (default plane)
- ❑ Sketch profile – 2D Rectangle
- ❑ Feature type – Extruded boss feature

Open a Sketch

1 Click to select the `Front` plane in the FeatureManager design tree.

2 Open a 2D sketch. Click **Sketch** on the Sketch toolbar.

Confirmation Corner

When many SolidWorks commands are active, a symbol or a set of symbols appears in the upper right corner of the graphics area. This area is called the **Confirmation Corner**.

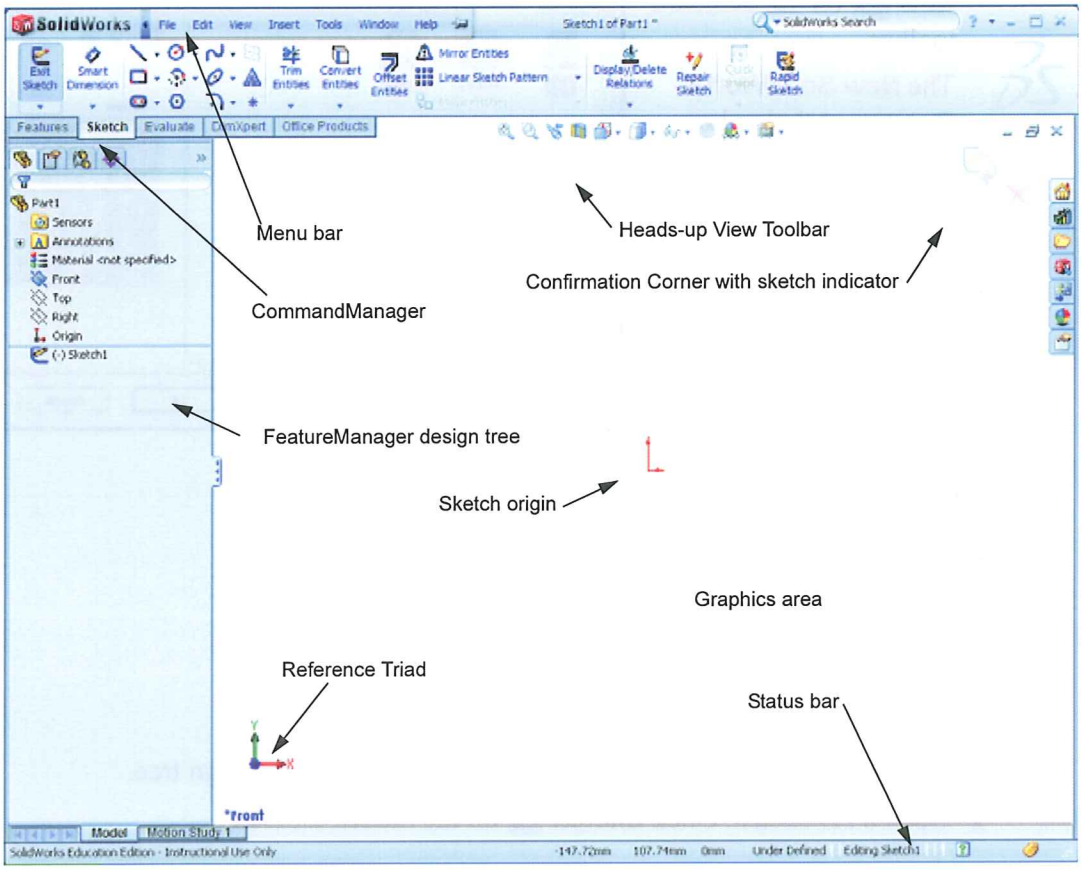
Sketch Indicator

When a sketch is active, or open, a symbol appears in the confirmation corner that looks like the **Sketch** tool. It provides a visual reminder that you are active in a sketch. Clicking this symbol exits the sketch saving your changes. Clicking the red X exits the sketch discarding your changes.

When other commands are active, the confirmation corner displays two symbols: a check mark and an X. The check mark executes the current command. The X cancels the command.

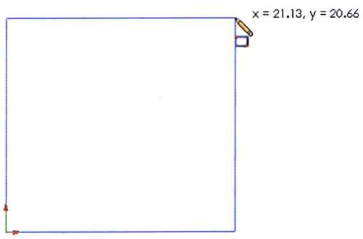
Overview of the SolidWorks Window

- A sketch origin appears in the center of the graphics area.
- **Editing Sketch1** appears in the status bar at the bottom of the screen.
- `Sketch1` appears in the FeatureManager design tree.
- The status bar shows the position of the pointer, or sketch tool, in relation to the sketch origin.

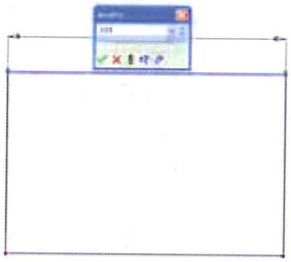


Sketch a Rectangle

1. Click **Corner Rectangle** on the Sketch toolbar.
2. Click the sketch origin to start the rectangle.
3. Move the pointer up and to the right, to create a rectangle.
4. Click the mouse button again to complete the rectangle.



Add Dimensions



1 Click **Smart Dimension** on the Dimensions/Relations toolbar.

The pointer shape changes to .

2 Click the top line of the rectangle.

3 Click the dimension text location above the top line.

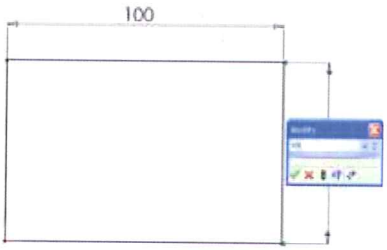
The **Modify** dialog box is displayed.

4 Enter **100**. Click or press **Enter**.

5 Click the right edge of the rectangle.

6 Click the dimension text location. Enter **65**. Click .

The top segment and the remaining vertices are displayed in black. The status bar in the lower-right corner of the window indicates that the sketch is fully defined.



Changing the Dimension Values

The new dimensions for the box are 100mm x 60mm. Change the dimensions.

1 Double-click **65**.

The **Modify** dialog box appears.

2 Enter **60** in the **Modify** dialog box.

3 Click .

Extrude the Base Feature.

The first feature in any part is called the *Base Feature*. In this exercise, the base feature is created by extruding the sketched rectangle.

1 Click **Extruded Boss/Base** on the Features toolbar.

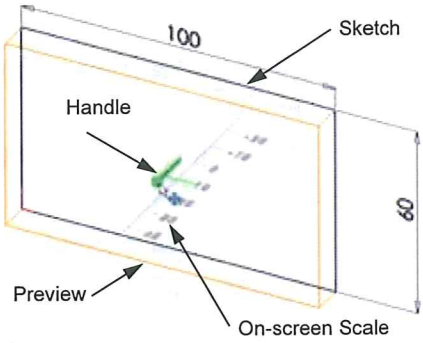
TIP: If the Features toolbar is not visible (active), you may also access the feature commands from the CommandManager.

The **Extrude** PropertyManager appears. The view of the sketch changes to trimetric.

2 Preview graphics.

A preview of the feature is shown at the default depth.

Handles appear that can be used to drag the preview to the desired depth. The handles are colored magenta for the active direction and gray for inactive direction. A callout shows the current depth value.

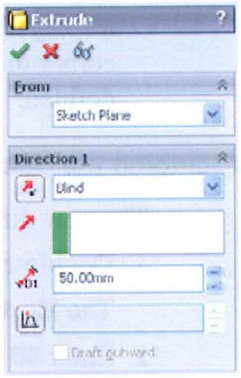


The cursor changes to . If you want to create the feature now, click the right mouse button. Otherwise, you can make additional changes to the settings. For example, the depth of extrusion can be changed by dragging the dynamic handle with the mouse or by setting a value in the PropertyManager.

3 Extrude feature settings.

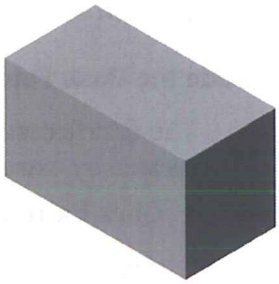
Change the settings as shown.

- End Condition = **Blind**
- (Depth) = **50**



4 Create the extrusion. Click **OK** .

The new feature, `Extrude1`, is displayed in the FeatureManager design tree.

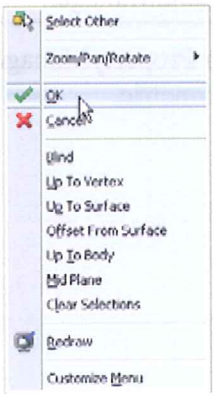


TIP:

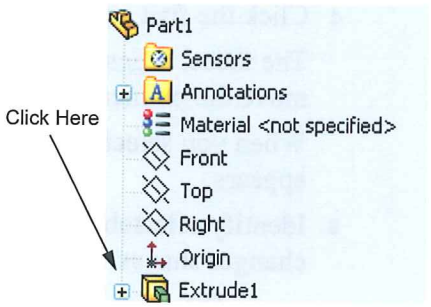
The **OK** button on the PropertyManager is just one way to complete the command.

A second method is the set of **OK/Cancel** buttons in the confirmation corner of the graphics area.

A third method is the right-mouse shortcut menu that includes **OK**, among other options.



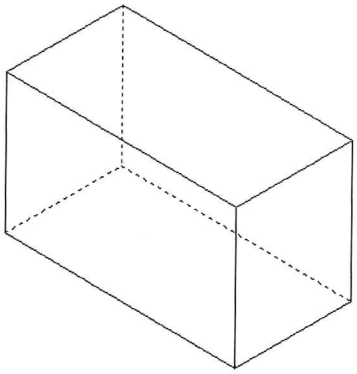
5 Click the plus sign ⊞ beside `Extrude1` in the FeatureManager design tree. Notice that `Sketch1` — which you used to extrude the feature — is now listed under the feature.



View Display

Change the display mode. Click **Hidden Lines Visible** on the View toolbar.

Hidden Lines Visible enables you to select hidden back edges of the box.



Save the Part

1 Click **Save** on the Standard toolbar, or click **File, Save.**

The **Save As** dialog box appears.

2 Type `box` for the filename. Click **Save**.

The `.sldprt` extension is added to the filename.

The file is saved to the current directory. You can use the Windows browse button to change to a different directory.

Round the Corners of the Part

Round the four corner edges of the `box`. All rounds have the same radius (10mm). Create them as a single feature.

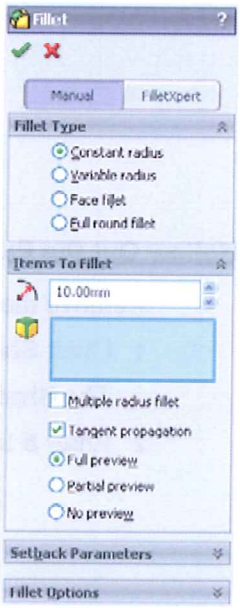
1 Click **Fillet** on the Features toolbar.

The **Fillet** PropertyManager appears.

2 Enter **10** for the **Radius.**

3 Select **Full preview.**

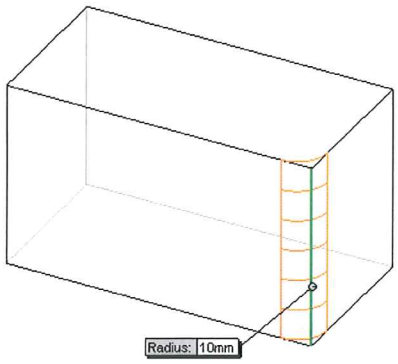
Leave the remaining settings at their default values.



4 Click the first corner edge.

The faces, edges, and vertices are highlighted as you move the pointer over them.

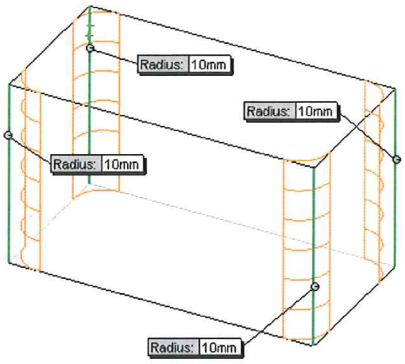
When you select the edge, a callout Radius: 10mm appears.



5 Identify selectable objects. Notice how the pointer changes shapes:

Edge: Face: Vertex:

6 Click the second, third and fourth corner edges.

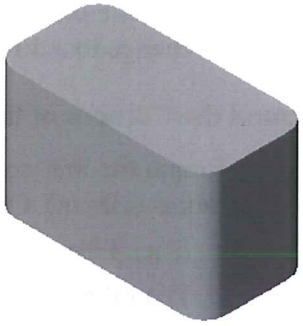


Note: Normally, a callout only appears on the *first* edge you select. This illustration has been modified to show callouts on each of the four selected edges. This was done simply to better illustrate which edges you are supposed to select.

7 Click **OK** .

`Fillet1` appears in the FeatureManager design tree.

8 Click **Shaded** on the View toolbar



Hollow Out the Part

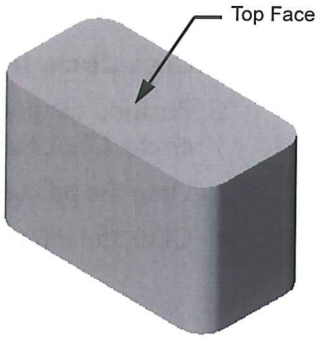
Remove the top face using the Shell feature.

1 Click **Shell** on the Features toolbar.

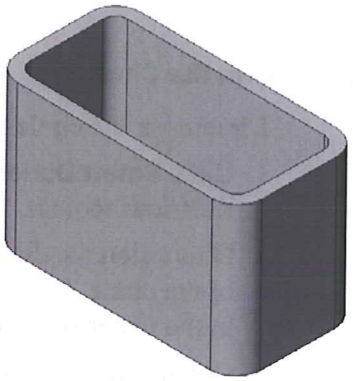
The **Shell** PropertyManager appears.

2 Enter **5** for **Thickness**.

3 Click the top face.



4 Click ✔.



Extruded Cut Feature

The Extruded Cut feature removes material. To make an extruded cut requires a:

- Sketch plane – In this exercise, the face on the right-hand side of the part.
- Sketch profile – 2D circle

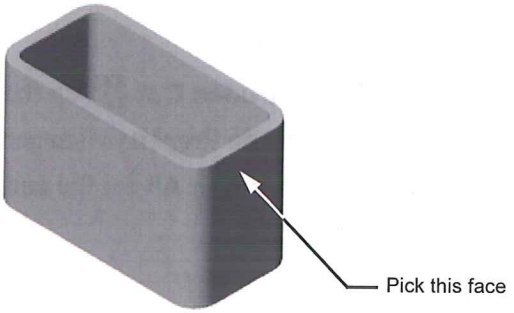
Open a Sketch

1 To select the sketch plane, click the right-hand face of the box.

2 Click **Right** on the Standard Views toolbar.

The view of the box turns. The selected model face is facing you.

3 Open a 2D sketch. Click **Sketch** on the Sketch toolbar.



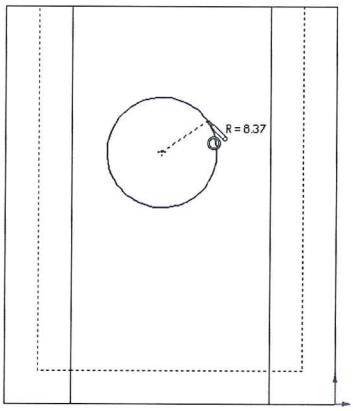
Sketch the Circle

1 Click **Circle** on the Sketch Tools toolbar.

2 Position the pointer where you want the center of the circle. Click the left mouse button.

3 Drag the pointer to sketch a circle.

4 Click the left mouse button again to complete the circle.



Dimension the Circle

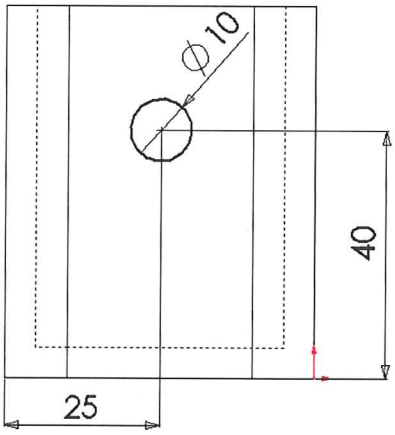
Dimension the circle to determine its size and location.

1 Click **Smart Dimension** on the Dimensions/ Relations toolbar.

2 Dimension the diameter. Click on the circumference of the circle. Click a location for the dimension text in the upper right corner. Enter **10**.

3 Create a horizontal dimension. Click the circumference of the circle. Click the left most vertical edge. Click a location for the dimension text below the bottom horizontal line. Enter **25**.

4 Create a vertical dimension. Click the circumference of the circle. Click the bottom most horizontal edge. Click a location for the dimension text to the right of the sketch. Enter **40**.



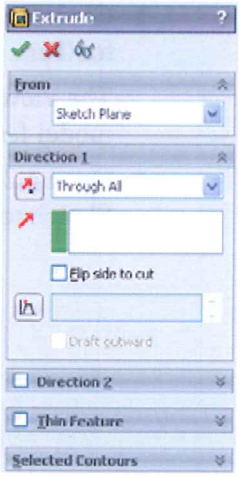
Extrude the Sketch

1 Click **Extruded Cut** on the Features toolbar.

The **Extrude** PropertyManager appears.

2 Select **Through All** for the end condition.

3 Click .



4 Results.

The cut feature is displayed.

Rotate the View

Rotate the view in the graphics area to display the model from different angles.

1 Rotate the part in the graphics area. Press and hold the middle mouse button. Drag the pointer up/down or left/right. The view rotates dynamically.

2 Click **Isometric** on the Standard Views toolbar.

Save the Part

1 Click **Save** on the Standard toolbar.

2 Click **File, Exit** on the Main menu.

Lesson 2 — 5 Minute Assessment

Name: _____ Class: _____ Date: _____

Directions: Answer each question by writing the correct answer or answers in the space provided or circle the answer as directed.

1 How do you start a SolidWorks session?

_____

_____

2 Why do you create and use Document Templates?

_____

_____

3 How do you start a new Part Document?

_____

4 What features did you use to create the `box`?

_____

5 True or False. SolidWorks is used by designers and engineers.

_____

6 A SolidWorks 3D model consists of _____ _____ _____.

_____

7 How do you open a sketch?

_____

8 What does the Fillet feature do?

_____

9 What does the Shell feature do?

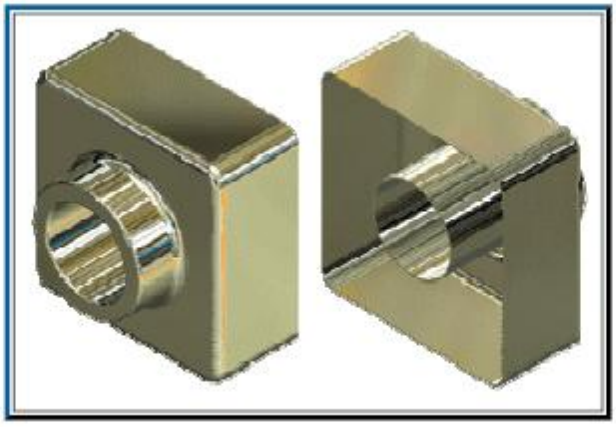
_____

10 What does the Cut-Extrude feature do?

_____

11 How do you change a dimension value?

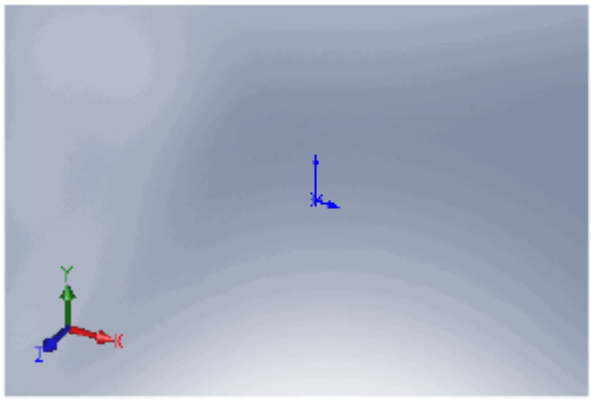
_____



Setting Up a New Part Document

Task

Open a new part document and save it as **Tutor1**.



1. Click **New** (Standard toolbar).
2. In the **New SolidWorks Document** dialog box, double-click **Part**

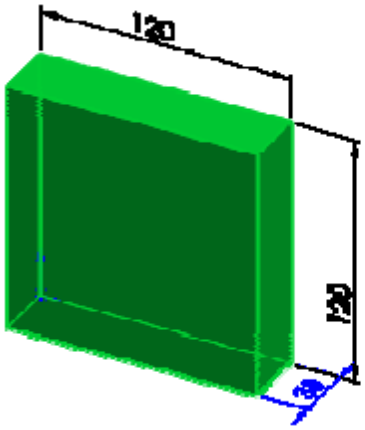
   For a description of the SolidWorks user interface, see the *User In SolidWorks Help*.

3. Click **Save** (Standard toolbar).
4. In the dialog box, type **Tutor1** for **File name**.
5. Click **Save**.

Creating the Base

Task

Extrude a rectangle with one corner on the origin and dimensioned as shown.



Sketching the Base

1. Click **Extruded Boss/Base** (Features toolbar).

   The Front, Top, and Right planes appear and the pointer changes to . As you move the pointer over a plane, the border of the plane is highlighted.

2. Select the Front plane.

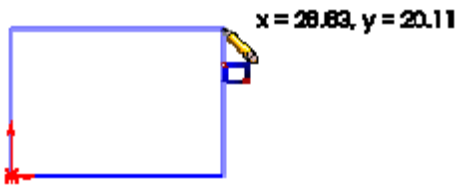
   - The display changes so the Front plane faces you.
   - The Sketch toolbar commands appear in the CommandManager.
   - A sketch opens on the Front plane.

3. Click **Rectangle** (Sketch toolbar).

4. Move the pointer to the sketch origin .

   The pointer is on the origin when it changes to .

5. Click the origin and drag the pointer up and to the right. Notice that it displays the current dimensions of the rectangle.

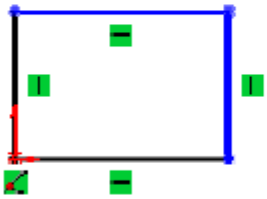


You do not have to be exact with the dimensions.

6. Release the **Rectangle** tool.

Dimensioning the Base

1. Click **Select** on the Standard toolbar.



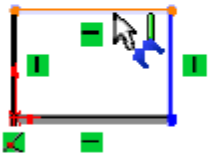
What are the colored squares with symbols?

The sides of the rectangle that touch the origin are black. Because you started sketching at the origin, the vertex of these two sides is automatically coincident with the origin, as shown by the symbol. This relationship constrains the sketch.

2. Drag one of the blue sides or drag the vertex to resize the rectangle.

3. Click **Smart Dimension** (Sketch CommandManager).

4. Select the top edge of the rectangle.



3. Click above the line to place the dimension.

The Modify dialog box appears.

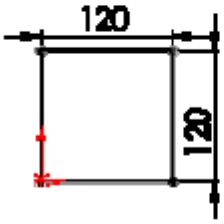
4. Set the value to **120**.

5. Click ✔.

   The sketch resizes to reflect the 120mm dimension.

6. Click **Zoom to Fit** (View toolbar) to display the entire rectangle and center it in the graphics area.

7. Repeat steps 2-6, with a vertical line, setting the height of the rectangle to 120mm.

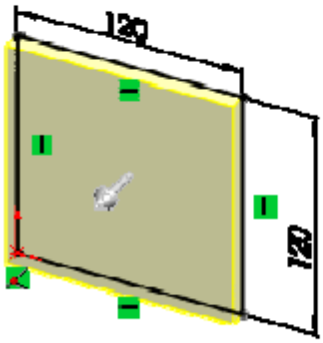


   The sketch is now fully defined, as shown in the status bar at the bottom of the SolidWorks window.

Extruding the Base

1. Click **Exit Sketch** (Sketch toolbar).

   The **Boss-Extrude** PropertyManager appears in the left pane, the view of the sketch changes to **Trimetric**, and a preview of the extrusion appears in the graphics area.



2. In the PropertyManager, under **Direction 1**:

   - Select **Blind** in **End Condition**.
   - Set **Depth** to 30.

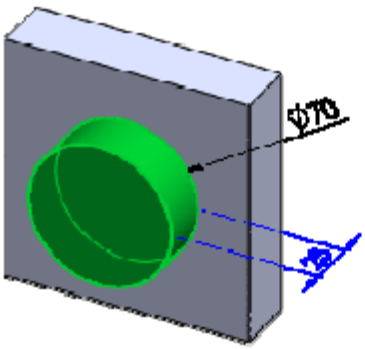
3. Click ✔.

   The new feature, **Boss-Extrude1**, appears in the FeatureManager design tree and in the graphics area.

Adding the Boss

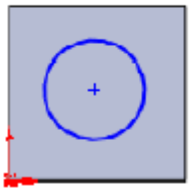
Task

Extrude a boss, centered on the model and dimensioned as shown.

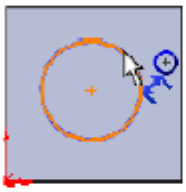


Sketching and Dimensioning the Boss

1. Click the front face of the model to preselect the sketch plane for the next feature.
2. Click **Extruded Boss/Base** (Features toolbar).
3. Click **Normal To** (Standard Views toolbar).
4. Click **Circle** (Sketch toolbar).
5. Click near the center of the face and move the pointer to sketch a circle.
6. Release the circle tool.



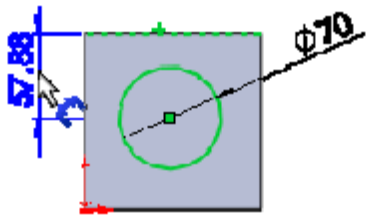
7. Click **Smart Dimension** (Dimensions/Relations toolbar).
8. Select the circle.



9. Move the pointer outside the model to see the current dimension.
10. Click to place the dimension.
11. In the Modify dialog box:
    a. Set the value to 70.
    b. Click ✔.

Constraining the Boss

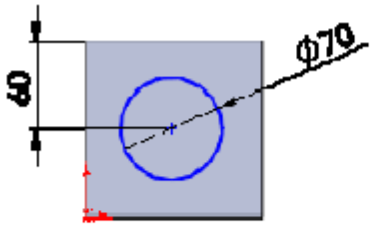
1. Still using **Smart Dimension** , select the top edge of the face, select the circle, and click to place the dimension.



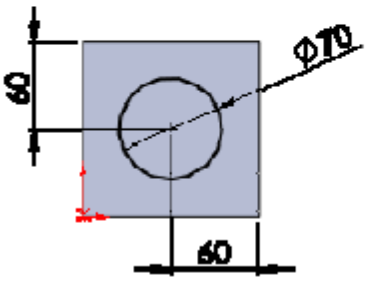
2. In the **Modify** dialog box:

   a. Set the value to **60**.

   b. Click ✔.



3. Repeat steps 1 and 2, selecting the right edge of the face and the circle.

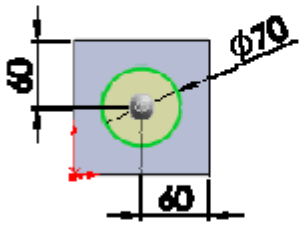


The circle turns black, and the status bar indicates that the sketch is fully defined.

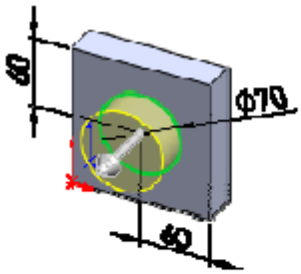
Extruding the Boss

1. Click **Exit Sketch** (Sketch toolbar).

   The Boss-Extrude PropertyManager appears in the left pane, and a preview of the extrusion appears in the graphics area.



2. Click **Trimetric** (Standard Views toolbar).



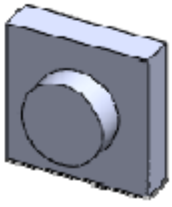
3. In the PropertyManager, under **Direction 1**:

   a. Select **Blind** in **End Condition**.

   b. Set **Depth** to **25**.

4. Click .

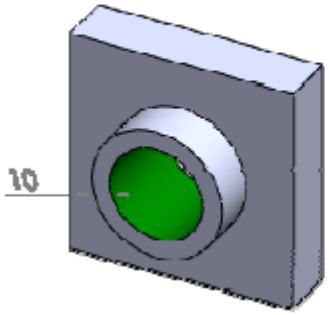
   Boss-Extrude2 appears in the FeatureManager design tree.



Cutting the Hole

Task

Cut a hole through the entire part with a radius 10mm less than the boss.

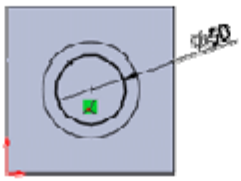


Extruding the Hole

1. Click **Extruded Cut** (Features toolbar).
2. Select the front face of the circular boss.
3. Click **Normal To** (Standard Views toolbar).
4. Click **Circle** (Sketch toolbar).
5. Move the pointer to the center of the boss.

The pointer changes to indicate that the center of the circle is coincident with the center of the boss.

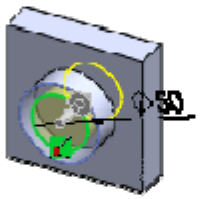
6. Drag to create the circle and release the tool.
7. Click **Smart Dimension** (S) and set the diameter of the hole to **50**.



7. Click **Exit Sketch** (Sketch toolbar).

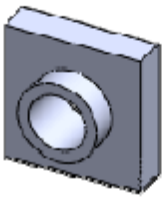
   The sketch closes and the Cut-Extrude PropertyManager appears.

8. Click **Trimetric** (Standard Views toolbar).



9. Under **Direction 1**, set **End Condition** to **Through All**.

10. Click ✔.

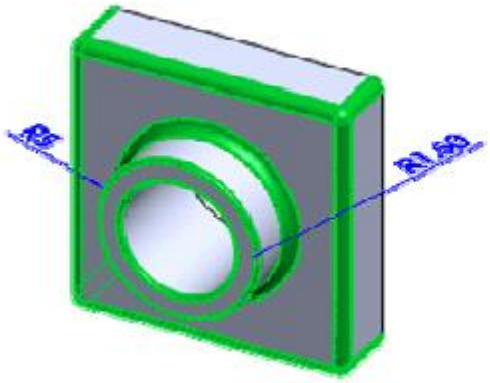


Task

Create fillets:

- 5mm radius for:
  - The edge between the base and boss
  - The corners of the base
  - The front perimeter of the base

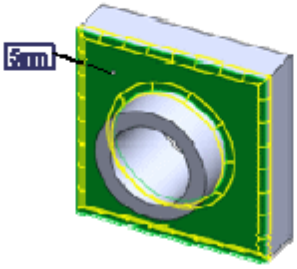
- 1.5mm radius for the top edges of the boss and hole



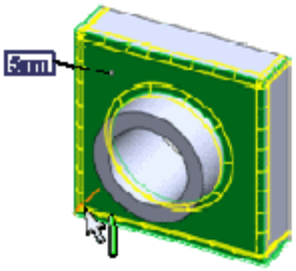
Creating the Base Fillets

1. Click **Fillet** (Features toolbar).
2. Under **Fillet Type**, select **Constant radius**.
3. Select the front face of the base.
4. Under **Items To Fillet:**
   a. Set **Radius** to **5**.
   b. Select **Full Preview**.

   The face is highlighted and a preview of the filleted face is displayed.



5. Select the four edges at the corners of the base.

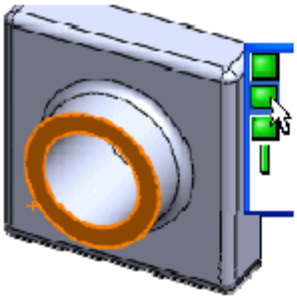


   As you select each edge, its name is added to **Edges, Faces, Features and Loops** and the preview is updated.

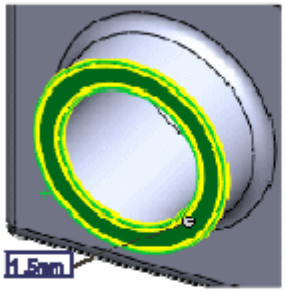
6. Click ✔.

Creating the Boss Fillets

1. Click **Fillet** (Features toolbar).
2. Under **Items to Fillet**, set **Radius** to **1.5**.
3. Right-click on either the inner or outer edge of the boss face and click **Select Other**.
4. Select the face of the boss from the pop-up list.



5. Click **Zoom to Selection** (View toolbar).



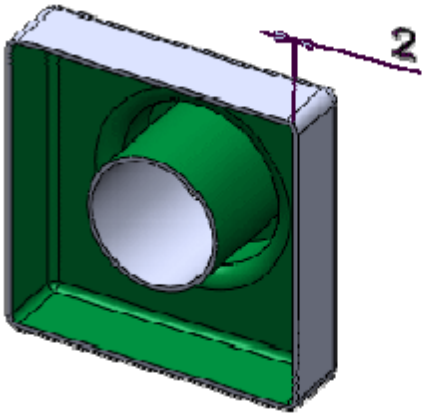
6. Click ✔.

Shelling the Part

Task

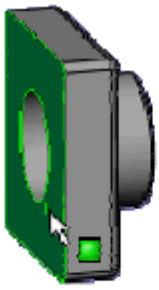
Hollow out the part by:

- Removing the back face
- Leaving a shell thickness of 2 mm



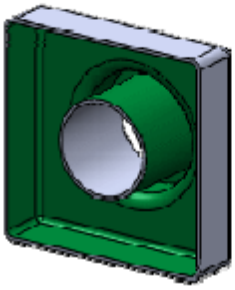
Creating the Shell

1. Click **Rotate View** (View toolbar).
2. Drag the pointer to rotate the part until you can see the back.
3. Release the tool. (How?)
4. Select the back face.



5. Click **Shell** (Features toolbar).
6. Under **Parameters**, set **Thickness** to **2**.
7. Click .

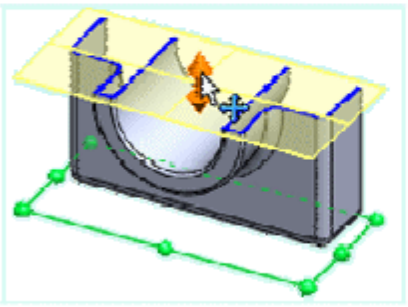
   The shell operation removes the selected face and leaves a thin-walled part.



Creating a Section View of the Shell

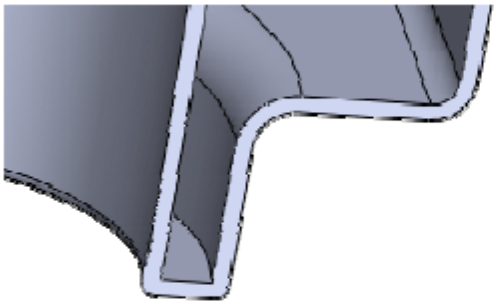
1. Click **Trimetric** (Standard Views toolbar).

2. Click **Section View** (View toolbar).
3. In the PropertyManager, under **Section 1**, click **Top Plane** .
4. Drag the handle up to show the section view.



5. Click .

   You can rotate and zoom the section view. Only the display of the part is cut, not the model itself.



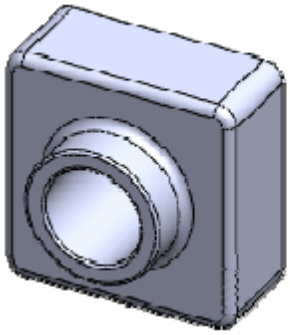
5. Click **Section View** (View toolbar) to clear the section view.
6. Save the part.

Editing Features

Task

Edit the base feature to:

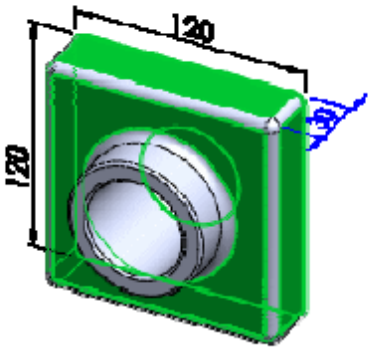
- Increase the depth of the model
- Modify the edge fillets
- Recreate the face fillets



Editing the Base Feature

1. Click **Trimetric** on the Standard Views toolbar.
2. Double-click **Boss-Extrude1** in the FeatureManager design tree.

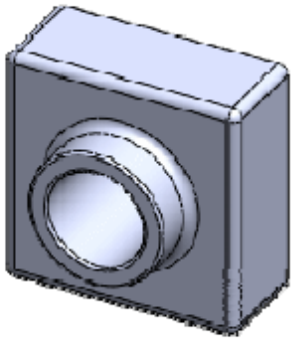
   The feature dimensions appear in the graphics area.



3. Double-click **30.**

4. In the **Modify** dialog box, set the value to **50** and click ✔.

5. Click **Rebuild** (Standard toolbar) to regenerate the model with the new dimension.



6. Click **Save** to save the part.

Modifying the Base Fillets

The base fillets were created in a single feature, **Fillet1**. You want to change the radius of just the edge fillets. To do this, you remove the fillets on the front face of the base and add them back as a separate fillet feature.

1. In the FeatureManager design tree, right-click **Fillet1** and select **Edit Feature**.

Recreating the Face Fillets

To recreate the face fillets you removed, you add a Fillet feature before the **Shell1** feature. If you add it after the Shell feature, the filleted area is not shelled.

1. In the FeatureManager design tree, place the pointer over the rollback bar below the **Shell1** feature.

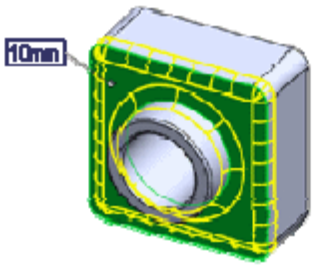
   The pointer changes to a hand:

2. Drag the rollback bar above the **Shell1** feature.

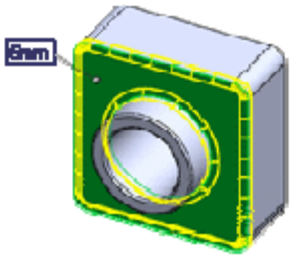
3. Click **Fillet** (Features toolbar).

4. Select the front face of the base.

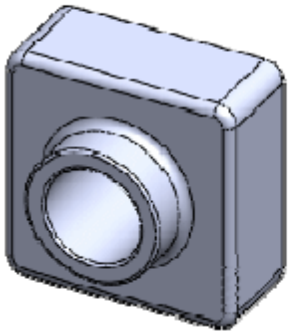
   The model shows the last radius used, 10mm.



5. Under **Items To Fillet**, change the **Radius** to **5**.



6. Click ✔.
7. Drag the rollback bar below the **Shell1** feature.

Finish

Congratulations! You have completed this tutorial.