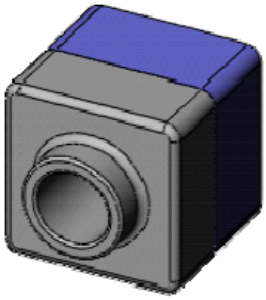
An assembly is a combination of two or more parts, also called components, within one SolidWorks document. You position and orient components using mates that form relations between components.

In this lesson, you build a simple assembly based on the part you created in Lesson 1.

This lesson discusses the following:

- Adding parts to an assembly
- Moving and rotating components in an assembly
- Creating display states in an assembly



Creating the Base Feature

You can use the same methods you learned in Lesson 1 to create the base for a new part.

1. Click **New**  (Standard toolbar) and open a new part.
2. Click **Extruded Boss/Base**  (Features toolbar) and select the **Front** plane.

   A sketch opens on the **Front** plane.
3. Sketch a rectangle beginning at the origin.
4. Click **Smart Dimension**  (Dimensions/Relations toolbar) and dimension the rectangle to 120mm x 120mm.

5. Click **Sketch** (Sketch toolbar) to exit the sketch.

   The **Extrude** PropertyManager and a preview of the extrusion appear.

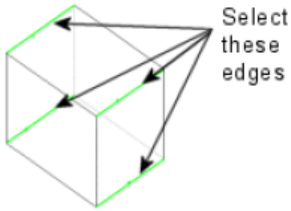
6. Under **Direction1**:

   - Set **End Condition** to **Blind**.
   - Set **Depth** to **90**.

7. Click to create the extrusion.

8. Click **Hidden Lines Visible** (View toolbar).

9. Click **Fillet** (Features toolbar) and select the four edges shown.



10. In the PropertyManager, under **Items to Fillet**, set **Radius** to **5**.

11. Click to fillet the selected edges.

Creating the Base Feature (Continued)

Next, you shell the part.

1. Click **Hidden Lines Removed** on the View toolbar.

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

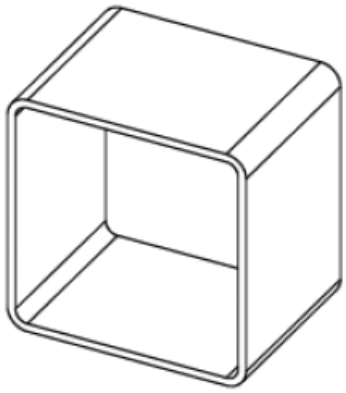
   The **Shell** PropertyManager appears.

3. Select the front face of the model.

   The face is listed in **Faces to Remove** in the PropertyManager.

4. Under **Parameters**, set **Thickness** to **4**.

5. Click **OK** .



6. Save the part as **Tutor2**.

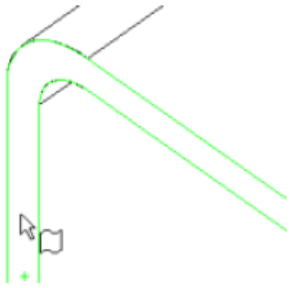
Creating a Lip on the Part

In this section, you use the **Convert Entities** and **Offset Entities** tools to create sketch geometry. Then you create a cut to make a lip to mate with the part from Lesson 1.

1. Click **Zoom to Area** (View toolbar) and drag-select to a corner of the part, as shown. Click **Zoom to Area** again to turn off the tool.

2. Select the front face of the thin wall.

   The edges of the face are highlighted.



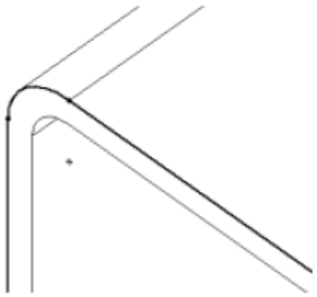
3. Click **Extruded Cut** (Features toolbar).

A sketch opens on the selected face.

4. Click **Convert Entities** (Sketch toolbar).

   The outer edges of the selected face are projected (copied) onto the sketch plane as lines and arcs. The **Convert Entities** PropertyManager appears. Because you do not want to convert more entities, you close the PropertyManager.

5. In the PropertyManager, click ✔.



Creating a Lip on the Part (Continued)

5. Click the front face again.

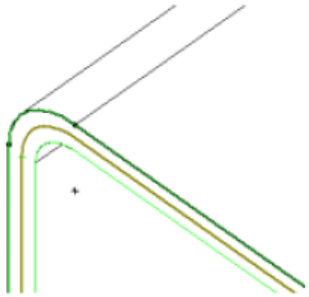
6. Click **Offset Entities** on the Sketch toolbar.

   The **Offset Entities** PropertyManager appears.

7. Under **Parameters**, set **Offset Distance** to **2**.

   The preview shows the offset extending outward.

8. Select **Reverse** to change the offset direction.



9. Click **OK** ✔.

   A set of lines is added to the sketch, offset from the outside edge of the selected face by 2mm. This relation is maintained if the original edges change.

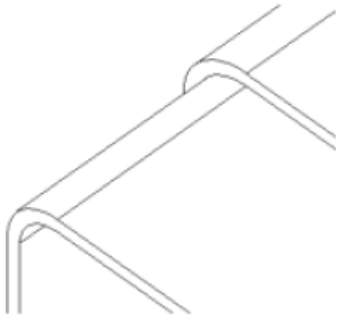
Creating a Lip on the Part (Continued)

10. Click **Sketch** (Sketch toolbar) to exit the sketch.

    The **Extrude** PropertyManager appears.

11. Under **Direction 1**, set **Depth** to **20**, then click ✔.

    The material between the two lines is cut, creating the lip.



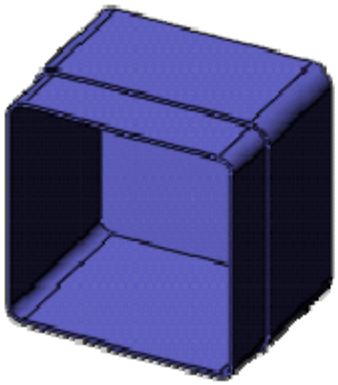
12. Click **Zoom to Fit** (View toolbar).

Changing the Color of a Part

You can change the color and appearance of a part or its features.

1. Click **Shaded With Edges** on the View toolbar.
2. Right-click the **Tutor2** icon at the top of the FeatureManager design tree.
3. On the shortcut menu, select **Appearances**, and then select **tutor2.**
4. In the PropertyManager, under **Color**, select the desired color on the color palette, then click **OK** ✔.

5. Save the part.



Creating the Assembly

Now create an assembly using the two parts.

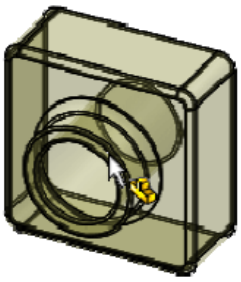
1. If **Tutor1.sldprt** is not open, click **Open** (Standard toolbar) and open the part you created, or browse to <*install_dir*>**samples****tutorial****lesson2****Tutor1.sldprt**.

2. Click **New** on the Standard toolbar, click **Assembly**, then click **OK**.

   The **Begin Assembly** PropertyManager appears.

3. Under **Part/Assembly to Insert**, select **Tutor1**.

   A preview of **Tutor1** appears in the graphics area, and the pointer changes to .

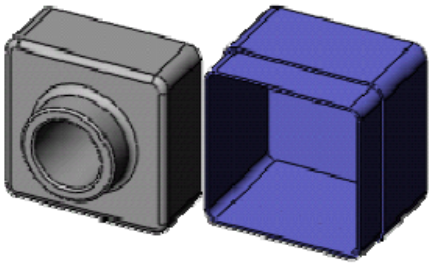


4. Click **Keep Visible** in the PropertyManager, so you can insert more than one component without having to re-open the PropertyManager.

5. Click anywhere in the graphics area to place **Tutor1**.

6. In the PropertyManager under **Part/Assembly to Insert**, select **Tutor2**.

7. Click in the graphics area to place **Tutor2** beside **Tutor1**.



8. Click ✔.

9. Click **Zoom to Fit**  .

10. Save the assembly as **Tutor**. (The **.sldasm** extension is added to the file name.) If you see messages about rebuilding the assembly and saving referenced documents, click **Yes**.

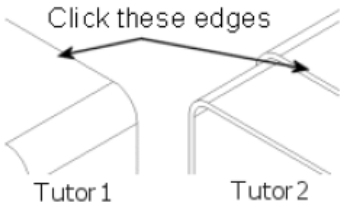
Mating the Components

In this topic, you define assembly mating relations between the components, making them align and fit together.

1. Click **Mate**  (Assembly toolbar).

   The **Mate** PropertyManager appears.

2. In the graphics area, select the top edge of **Tutor1**, then select the outside edge of the lip on the top of **Tutor2**.



   The **Mate** pop-up toolbar appears, and the components move into place, previewing the mate. The edges are listed in the **Entities to Mate**  box under **Mate Selections** in the PropertyManager.

3. On the **Mate** pop-up toolbar:

a. Click **Coincident** as the mate type.

b. Click **Add/Finish Mate**.

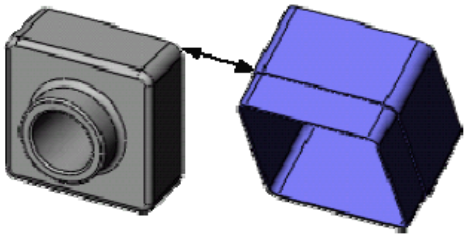
A coincident mate appears under **Mates** in the PropertyManager.

Mating the Components (Continued)

The position of **Tutor2** is not fully defined yet. It still has some degrees of freedom to move in directions that are not yet constrained by mates.

Test degrees of freedom by moving the components.

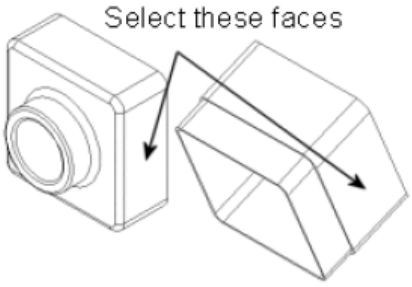
1. In the graphics area, select the **Tutor2** component and hold down the left mouse button.



2. Drag the component from side to side to observe the available degrees of freedom.

Adding More Mates

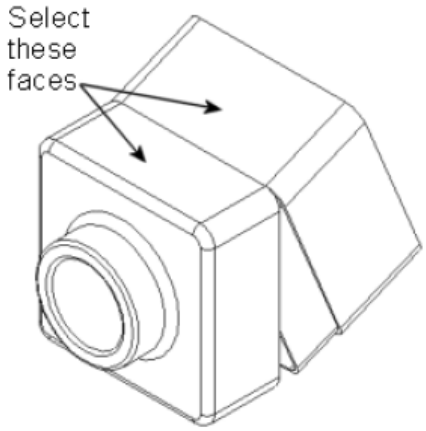
1. Select the rightmost face of one component, then select the corresponding face on the other component.



2. On the **Mate** pop-up toolbar, click **Coincident**, then click **Add/Finish Mate**.

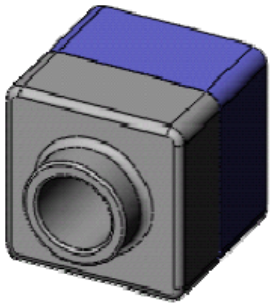
   Another coincident mate appears under **Mates** in the PropertyManager.

3. Repeat steps 1 and 2, but select the top faces of both components, to add another **Coincident** mate.



4. Click **OK** .

5. Save the assembly.



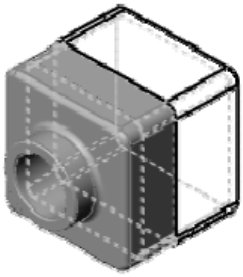
Using Display States

You can change the display settings of the components and save the settings in a display state.

1. At the top of the FeatureManager design tree, to the right of the tabs, click **Show Display Pane** .

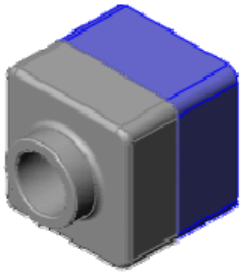
The Display Pane shows the different display settings (appearances, transparency, etc.) of each component.

2. Right-click anywhere in the Display Pane and select **Add Display State**.
3. Type a name and press **Enter**.
4. Move the pointer over **Tutor2** in the FeatureManager design tree, then:
   a. Move the pointer into the **Display Mode** column.
   b. When the pointer changes to , click, then select **Hidden Lines Visible** .



5. Click **Hide Display Pane** .
6. Right-click and select **Default_Display State-1**.

   The assembly returns to its original display state.



Congratulations! You have completed this lesson!