SolidWorks

SolidWorks® 99
Tutorial

Contents

1

Getting Started

This tutorial introduces you to some of the most commonly used features of the SolidWorks® 99 mechanical design automation system.

SolidWorks 99 is supported under the Microsoft® Windows® graphical user interface. This tutorial assumes that you have used Windows before and know basic Windows skills, such as how to run programs, resize windows, and so on.
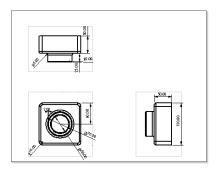
Before you begin this tutorial, you should read Chapter 1 of the *SolidWorks 99 User's Guide*, to familiarize yourself with some of the fundamentals, including:

❑ Terminology

❑ Window features, such as toolbars, menus, and views

❑ Basic graphic operations, such as selecting and moving objects

❑ The FeatureManager™ design tree

Designing with SolidWorks 99

As you do the examples in this tutorial, you will discover that the methods you use to design parts and assemblies, and to create drawings, represent a unique approach to the design process.

❑ **With SolidWorks 99, you create 3D parts, not just 2D drawings.** You can use these 3D parts to create 2D drawings and 3D assemblies.
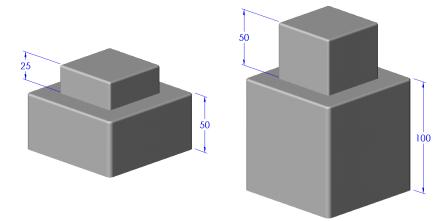
CAD: 2D drawings, made up of individual lines
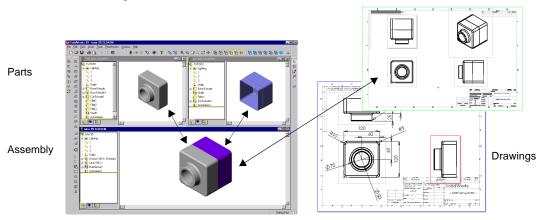
SolidWorks 99: 3D parts
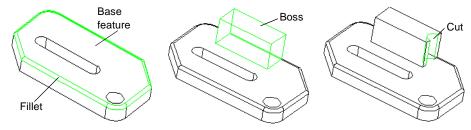
❑ **SolidWorks 99 is a dimension-driven system.** You can specify dimensions and geometric relationships between elements. Changing dimensions changes the size and shape of the part, while preserving your design intent. For example, in this part, the boss is always half as high as the base.
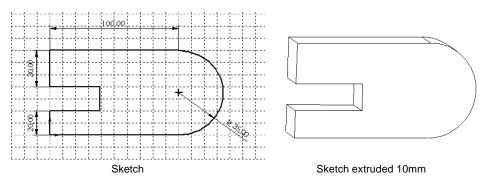
❑ **A SolidWorks 3D model consists of parts, assemblies, and drawings.** Parts, assemblies, and drawings display the same model in different documents. Any changes you make to the model in one document are propagated to the other documents containing the model.

Parts

Assembly

Drawings

❑ **You build parts from features.** Features are the shapes (bosses, cuts, holes) and operations (fillets, chamfers, shells, and so on) that you combine to build parts.

Base
feature

Boss

Cut

Fillet
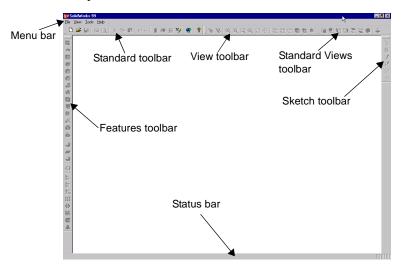
❑ **You build most features from sketches.** A sketch is a 2D profile or cross section. Sketches can be extruded, revolved, lofted, or swept along a path to create features.

Sketch

Sketch extruded 10mm

Starting SolidWorks 99

1   Click the **Start** button on the Windows taskbar.

2   Click **Programs**.

3   Click **SolidWorks 99**.

4   Click **SolidWorks 99** again.

Notice these important features of the SolidWorks window.

The toolbars may be arranged differently on your screen. You can rearrange the toolbars to suit your preferences. You can dock them at the edges of the graphics area, or you can pull them into the graphics area and allow them to float.

In this window, you can do the following:

❑   Click **File** to open a new or existing part, assembly, or drawing.

❑   Click **View, Toolbars,** or press the right mouse button (called right-click) in the toolbar region, to select which toolbars to display. The **View** menu also lets you hide or display the status bar.

❑   Click **Tools** to set SolidWorks options, or to record a macro.

❑   Click the **Maximize** icon in the upper-right corner to expand the window to full-screen size.

> **NOTE:** If a dialog box appears reminding you to register your copy of SolidWorks 99, click **OK**.

Getting Help

If you have questions while you are using the SolidWorks software, you can find answers in several ways:

❑ For **Online help**, click **Help, SolidWorks 99 Help Topics** in the menu bar. The online help also includes a special section about *New Functionality in SolidWorks 99,* a summary of the enhancements in SolidWorks 99.

❑ For helpful hints, click **Help, Tip of the Day**. To see a tip each time you start SolidWorks 99, click **Show Tips at Startup** in the **Tip of the Day** dialog box.

❑ For **Dialog box** help that describes the active dialog box, and provides access to the full online help system, click the **Help** button in the dialog box or press the **F1** key.

❑ For **Tooltips** that identify buttons on a toolbar, point at the button, and a moment later, the tooltip pops up.

❑ As you point at toolbar buttons or click menu items, the **Status Bar** at the bottom of the SolidWorks window provides a brief description of the function.

❑ The *SolidWorks 99 User's Guide* provides detailed information about installing, using, and getting the most out of the SolidWorks software.

❑ For more information and the latest news about the SolidWorks software and company, visit the SolidWorks web site, http://www.solidworks.com, or click **Help, About SolidWorks 99, Connect**.

2

The 40-Minute Running Start

This chapter guides you through the creation of your first SolidWorks model. You create this simple part:

This chapter includes:

❑ Creating a *base* feature

❑ Adding a *boss* feature

❑ Adding a *cut* feature

❑ Modifying features (adding fillets, changing dimensions)

❑ Displaying a section view of a part

❑ Displaying multiple views of a part

You should be able to complete this chapter in about 40 minutes.

NOTE:	Some of the illustrations in this tutorial have been modified for clarity. For that reason, what you see on your screen may look different from the illustrations.

Creating a New Part Document

1   To create a new part, click the **New** button ▣ on the Standard toolbar, or click **File, New** on the menu bar.

The **New** dialog box appears.

2   **Part** is the default selection, so click **OK**.

A new part window appears.

Displaying the Toolbars

The toolbars give you quick access to some of the most commonly used SolidWorks functions and features.

•   On the **View** menu, click **Toolbars.** You should see that the **Standard, View, Features, Sketch,** and **Standard Views** toolbars are selected. If you want to display additional toolbars, you can select them on this menu. However, the appropriate toolbars display automatically when you open different document types (part, assembly, or drawing) or open a sketch.

•   To display a list of available toolbars, right-click on any SolidWorks window border. A shortcut menu appears that lists the toolbars and that lets you customize the toolbars and the display of the tooltips.
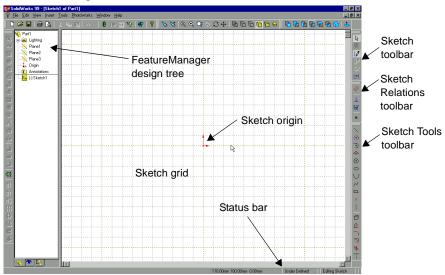
Opening a Sketch

1   To open a sketch, click the **Sketch** button 🖉 on the Sketch toolbar, or click **Insert, Sketch** on the menu bar.

This opens a sketch on **Plane1** (one of the three default planes listed in the FeatureManager design tree).

2   Notice that:

•   A sketch grid and an origin appear.

•   The Sketch Tools and Sketch Relations toolbars are displayed.

•   "Editing Sketch" 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, with regard to the sketch origin.

Before you begin sketching, make sure that your SolidWorks settings match the settings used in this tutorial.

3 Click the **Grid** ▦ button on the Sketch toolbar.
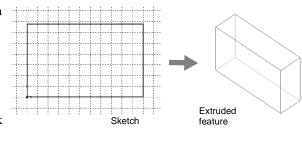
The **Options** dialog box appears.

4 On the **Grid/Units** tab:

- Make sure that **Length Unit** is set to **Millimeters** and that **Decimal places** is set to 2.
- In the **Grid, Properties** section, make sure that the **Display grid** check box is selected.

5 Click **OK**.

Sketching the Rectangle

The first feature in your part is a box extruded from a sketched rectangular profile. You begin by sketching the rectangle.

Sketch

Extruded feature
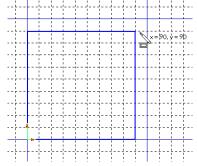
1   Click **Rectangle** on the Sketch Tools toolbar, or click **Tools, Sketch Entity, Rectangle**.

2   Move the pointer to the sketch origin, and hold down the left mouse button. Drag the pointer to create a rectangle. Release the mouse button to complete the rectangle.

As you drag, notice that the pointer displays the dimensions of the rectangle. Also, the rectangle snaps to the grid points.

If you prefer to work with snap behavior turned off, click **Grid**, click to clear the **Snap to points** check box, and click **OK**.

3   Click the **Select** button on the Sketch toolbar, or click **Tools, Select** on the menu bar, or press **Esc**.

The two sides of the rectangle that touch the origin are black. Because you began sketching at the origin, the vertex of these two sides is automatically *related* to the origin. (The vertex is not free to move.)

The other two sides (and three vertices) are blue. This indicates that they are free to move.

4   Click one of the blue sides, and drag the side or the drag handle at the vertex to resize the rectangle.

Adding Dimensions

In this section you specify the size of the sketched rectangle by adding dimensions. The SolidWorks software does *not* require that you dimension sketches before you use them to create features. However, for this example, you should add dimensions now to fully define the sketch.
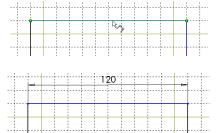
As you add dimensions, note the state of the sketch displayed in the status bar. Any SolidWorks sketch is in one of three states–each state is indicated by a different color:
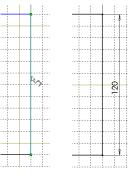
❑ In a *fully defined* sketch, the positions of all the entities are fully described by dimensions or relations or both. In a fully defined sketch, all the entities are *black*.

❑ In an *under defined* sketch, additional dimensions or relations or both are needed to completely specify the geometry. In this state, you can drag under defined sketch entities to modify the sketch. An under defined sketch entity is *blue*.

❑ In an *over defined* sketch, an object has conflicting dimensions or relations or both. An over defined sketch entity is *red*.

1 Click **Dimension** 🖉 on the Sketch Relations toolbar, or click **Tools, Dimensions, Parallel**.

The pointer shape changes to ↖.

2 Click the top edge of the rectangle, then click where you want to place the dimension.

Notice that the vertical line at the right (and the lower-right vertex) changed from blue to black. By dimensioning the length of the top of the rectangle, you defined the position of the rightmost segment. You can still drag the top segment up and down. Its blue color indicates that it is not fully defined; therefore, it can move.

3 Click the right edge of the rectangle, then click to place its dimension.

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

Changing the Dimension Values

The dimensions for the block are 120mm x 120mm. To change the dimensions, you use the **Select** tool.

1 Use one of these methods to access the **Select** tool:

- Click the **Select** button ⬚ on the Sketch toolbar.

- Click **Tools**, **Select** on the menu bar.

- Right-click in the graphics area to display the shortcut menu, then click **Select**.

TIP: Taking advantage of the shortcut menus helps you work more efficiently.

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2 Double-click one of the dimensions.

The **Modify** dialog box appears.

3 To change the dimension to 120mm, type a new value or click the arrows, then click ⬚ or press **Enter**.

4 Double-click the other dimension and change its value to 120mm.

5 To display the entire rectangle at full size and to center it in the graphics area, use one of the following methods:

- Click **Zoom to Fit** ⬚ on the View toolbar.

- Click **View, Modify, Zoom to Fit**.

- Press the **f** key.

You can edit dimension values as you create them by enabling the **Input dimension value** option. Each time you add a new dimension, the **Modify** dialog box is displayed immediately, ready for you to enter the value.

1 Click **Tools**, **Options**.

2 On the **General** tab, in the **Model** section, select the **Input dimension value** check box.

3 Click **OK**.

Extruding the Base Feature

The first feature in any part is called the *base feature*. You create this feature by extruding the sketched rectangle.

1 Click **Extruded Boss/Base** on the Features toolbar, or click **Insert**, **Base**, **Extrude**.

The **Extrude Feature** dialog box appears, and the view of the sketch changes to isometric.
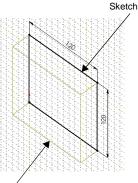
2 Specify the type and depth of the extrusion:

- Make sure that **Type** is set to **Blind**.

- Set **Depth** to 30mm. Either use the arrows to increment the value, or type the value.

  When you click the arrows, a preview of the result is displayed in the graphics area.
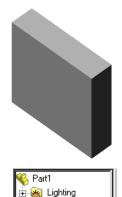
3 To see how the model would look if you extruded the sketch in the opposite direction, select the **Reverse Direction** check box. Then click to clear the **Reverse Direction** check box to extrude the sketch as shown.

4 Make sure that **Extrude as** is set to **Solid Feature**.

5 Click **OK** to create the extrusion.

Notice the new feature, **Base-Extrude,** in the FeatureManager design tree.

6 Click the plus sign beside **Base-Extrude** in the FeatureManager design tree. Notice that **Sketch1**, which you used to extrude the feature, is now listed under the feature.

Sketch

Preview of the extrusion

Click here

Changing View Mode and Display Mode

To magnify a model in the graphics area, you can use the zoom tools on the View toolbar.

Click **Zoom to Fit** to display the part full size in the current window.

Click **Zoom to Area**, then drag the pointer to create a rectangle. The area in the rectangle zooms to fill the window.

Click **Zoom In/Out**, then drag the pointer. Dragging up zooms in; dragging down zooms out.
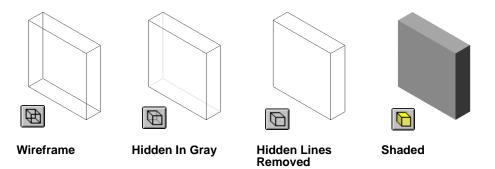
Click a vertex, an edge, or a feature, then click **Zoom to Selection.** The selected item zooms to fill the window.

Here are some other ways to zoom:

- Select a zoom mode from the **View, Modify** menu.
- Right-click a blank area, and select a zoom mode; right-click on the model, select **View**, then choose a mode.
- To zoom in steps, press the **z** key to zoom out or the **Z** key to zoom in.

To display the part in different modes, click the buttons in the View toolbar. You can also change the display mode by selecting from the **View, Display** menu.

Wireframe   **Hidden In Gray**   **Hidden Lines Removed**   **Shaded**

The default display mode for parts and assemblies is **Shaded**. You may change the display mode whenever you want.

Sketching a Boss

To create additional features on the part (such as bosses or cuts), you sketch on the model faces or planes, then extrude the sketches.

> **NOTE:** You sketch on one face or plane at a time, then create a feature based on one or more sketches.

- To open a new sketch, click a plane or face on which to sketch, then click the **Sketch** tool 🖊.
- To close a sketch, click the **Sketch** tool again, or select **Exit Sketch** from the shortcut menu.
- To edit a sketch you worked on previously, right-click the feature created from the sketch, or the sketch name, in the FeatureManager design tree, then select **Edit Sketch** from the shortcut menu.

1 Click **Hidden Lines Removed** 🖼 on the View toolbar, or click **View, Display, Hidden Lines Removed**.

2 Click **Select** 🖽 on the Sketch toolbar, if it is not already selected.

3 Click the front face of the part to select it.

The edges of the face become dotted lines to show that it is selected.

> **TIP:** The pointer changes to 🖑 to show that you are selecting the face.
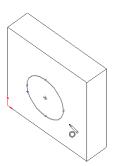
4 Click **Sketch** 🖊 on the Sketch toolbar.

– or –

Right-click anywhere in the graphics area and select **Insert Sketch**.

A grid appears on the selected face to show that it is now the active sketching plane. If you prefer to work with the grid turned off, click **Grid** 🖽, click to clear the **Display grid** check box, and click **OK**.

5 Click **Circle** ⊕ on the Sketch Tools toolbar, or click **Tools, Sketch Entity, Circle**.

6 Click near the center of the face and drag to sketch a circle.

Dimensioning and Extruding the Boss

To establish the location and size of the circle, add the necessary dimensions.

1   Click **Dimension** ✎ on the Sketch Relations toolbar, or right-click anywhere in the graphics area and select **Dimension** from the shortcut menu.

2   Click the top edge of the face, click the circle, then click a location for the dimension.

Notice the dimension preview as you click each entity. The preview shows you where the witness lines are attached, and helps you see that you have selected the correct entities for the dimension. When you add a locating dimension to a circle, the witness line is attached to the centerpoint by default.

3   Set the dimension value to 60mm. If you enabled the **Input dimension value** option (see page 2-6), the **Modify** dialog box appears, and you can enter the new value now. Otherwise, double-click the dimension, then enter the new value in the **Modify** dialog box.

4   Repeat the process to dimension the circle to the side edge of the face. Set this value to 60mm also.

5   Still using the **Dimension** tool ✎, click the circle to dimension its diameter. Move the pointer around to see the preview for the dimension.
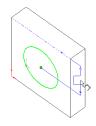
When the dimension is aligned horizontally or vertically, it appears as a linear dimension; if it is at an angle, it appears as a diameter dimension.
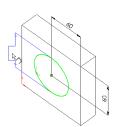
6   Click a location for the diameter dimension. Set the diameter to 70mm.
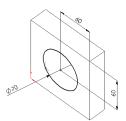
Now the circle turns black, and the status bar indicates that the sketch is fully defined.
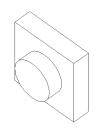
7   Click **Extruded Boss/Base** 🗔 on the Features toolbar, or click **Insert**, **Boss**, **Extrude.**

8   In the **Extrude Feature** dialog box, set the **Depth** of the extrusion to 25mm, leave the other items at the defaults, and click **OK** to extrude the boss feature.

Notice that **Boss-Extrude1** appears in the FeatureManager design tree.

Changing View Orientation

You can use the buttons on the Standard Views toolbar to set the view orientation of the sketch, part, or assembly.

	Front		**Top**
	Back		**Bottom**
	Left		**Isometric**
	Right		**Normal To** (the selected plane or planar face)

The default planes of the part correspond to the standard views as follows:

- **Plane1 - Front** or **Back**
- **Plane2 - Top** or **Bottom**
- **Plane3 - Right** or **Left**
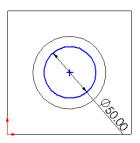
Creating the Cut

Next, create a cut concentric with the boss.

1 Click the front face of the circular boss to select it.

2 Click **Normal To** ![icon] on the Standard Views toolbar.

The part is turned so that the selected model face is now facing you.

3 Open a new sketch, and sketch a circle near the center of the boss as shown.

4 Click **Dimension** ![icon], and dimension the diameter of the circle to 50mm.
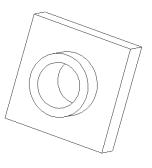
5 On the Sketch Relations toolbar, click **Add Relation** ⊥, or click **Tools**, **Relations**, **Add** on the menu bar.

The **Add Geometric Relations** dialog box appears.

6 Select the sketched circle (the inner circle) and the edge of the boss (the outer circle).

Notice the contents of the **Selected Entities** box. Only those relations that are appropriate for the selected entities are available. The most likely relation is automatically selected.

7 Make sure that **Concentric** is selected, click **Apply**, and click **Close**.

8 Click **Extruded Cut** 🔳 on the Features toolbar, or click **Insert**, **Cut**, **Extrude**.

9 In the **Extrude Cut Feature** dialog box, select **Through All** in the **Type** list, and click **OK**.

Saving the Part

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

The **Save As** dialog box appears.

2 Type **Tutor1** and click **Save.**

The extension **.sldprt** is added to the filename, and the file is saved to the current directory. If you want, you can navigate to a different directory using the Windows browse buttons, then save the file.

> **NOTE:** File names are not case sensitive. That is, files named **TUTOR1.sldprt**, **Tutor1.sldprt**, and **tutor1.sldprt** are all the same part.

Rotating and Moving the Part

To view the model from different angles, and to more easily select faces, edges, and so on, you can rotate and move the model in the graphics area.

To rotate the part, use one of the following methods:

- To rotate the part in steps, use the arrow keys. The increment of the steps is defined by the value in the **Arrow keys** box in the **View Rotation** dialog box in the **FeatureManager Design Tree** section on the **General** tab on the **Tools, Options** dialog box.

- To rotate the part in 90° increments, hold down the **Shift** key and use the arrow keys.

- To rotate the part to any angle, click **Rotate View** ⟳ on the View toolbar, or click **View, Modify, Rotate**, then drag.

- To rotate the part clockwise and counterclockwise around the center of the graphics area, using the increment value, hold down the **Alt** key and use the arrow keys.

- To rotate the part around an edge or vertex, click **Rotate View** ⟳, click the edge or vertex, then drag.

To move the part view, use one of the following methods:

- Click **Pan** ✛ on the View toolbar, or click **View, Modify, Pan**, then drag the part to move it around in the graphics area.

- Hold down the **Ctrl** key and use the arrow keys to move the view up, down, left, or right.

- Use the scroll bars to pan to a different area of the window.

Rounding the Corners of the Part
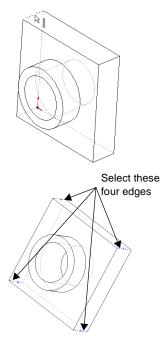
In this section you round the four corner edges of the part. Because the rounds all have the same radius (10mm), you can create them as a single feature.

1   Click **Hidden In Gray** 🔲. This makes it easy to select the hidden edges.

2   Click the first corner edge to select it.

Notice how the faces, edges, and vertices highlight as you move the pointer over them, identifying selectable objects. Also, notice the changing pointer shape:

     ↖ | edge   ↖ ⬭ face   ↖ □ vertex

3   Rotate the part approximately as shown. Use any of the methods discussed in the previous section.

4   Hold down the **Ctrl** key and click the second, third, and fourth corner edges.

Select these
four edges

5   Click **Fillet** 🔘 on the Features toolbar, or click **Insert, Features, Fillet/Round**.
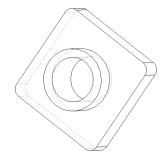
The **Fillet Feature** dialog box appears.

Notice that the **Edge fillet items** box indicates four selected edges.

6   Change the **Radius** to 10mm. Leave the remaining items at the default values.

7   Click **OK**.

The **Fillet1** feature appears in the FeatureManager design tree.

Adding Fillets

Now add fillets and rounds to other sharp edges of the part. You can select faces and edges either before or after opening the **Fillet Feature** dialog box.

1 Click **Hidden Lines Removed** 🔲.

2 Click **Fillet** 🔴 or **Insert, Features, Fillet/Round**.

3 Click the front face of the base to select it.

   Both the outside and inside edges (around the boss) are highlighted when you select the face.

   Notice that the **Edge fillet items** list shows that one face is selected.

4 Change the **Radius** to 5mm, and click **OK**.

   The inside edge is filleted and the outside edge is rounded in a single step.

5 Click **Fillet** 🔴 again.

6 Click the front face of the circular boss.

7 Change the **Radius** to 2mm, and click **OK**.

Feature names include the name of the feature type and a number that increments by one each time you add another feature of the same type.

For example, the fillet you created in the previous section is named **Fillet1** in the FeatureManager design tree. The fillets you created in this section are named **Fillet2** and **Fillet3**. If you delete **Fillet3**, the next fillet you create is named **Fillet4**; the numbers are not reused.

Note that features are listed in the FeatureManager design tree in the order in which they are created.

Shelling the Part

Next, you shell the part. Shelling hollows out the part by removing material from the selected face, leaving a thin-walled part.

1 Click **Back** 🔳 on the Standard Views toolbar.

The back of the part is now facing towards you.
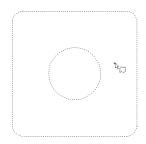
2 Click **Shell** 🔳 on the Features toolbar, or click **Insert, Features, Shell.**

The **Shell Feature** dialog box appears.
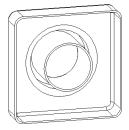
3 Click the back face to select it.

4 Change the **Thickness** to 2mm and click **OK.**

The shell operation removes the selected face.

5 To see the results, use the arrow keys on the keyboard to rotate the part approximately as shown.

Creating a Named View

You can use the **Orientation** dialog box to:

- Create your own named views.

- Switch to any of the standard views (see page 2-11) and to two additional views, ***Trimetric** and ***Dimetric**.

- Change the orientation of all the standard views.

- Restore all of the standard views to their default settings.

For more information about the **Orientation** dialog box, see Chapter 1 of the *SolidWorks 99 User's Guide*.

Now create a named view.

1 Click **View Orientation** ▨ on the View toolbar, or click **View, Orientation**, or press the **Spacebar**, to display the **Orientation** dialog box.

2 In the **Orientation** dialog box, click **New View** ▨ .

3 Type **Shell Back** in the **Named View** dialog box.

4 Click **OK**.

The new view name, **Shell Back**, is added to the **Orientation** dialog box, and you can select it at any time. To switch to a different view, double-click a different view name in the **Orientation** dialog box.

5 Click **Save** ▨ to save the part.

Changing a Dimension

This section illustrates a way to change the dimension of an extruded feature using feature handles. You can also change the dimension using the **Modify** dialog box method as discussed earlier (see page 2-6).

1   Examine the FeatureManager design tree. It shows the features of the part in the order in which you created them.

2   Double-click **Base-Extrude** in the FeatureManager design tree.

Notice that in the FeatureManager design tree, the **Base-Extrude** feature is expanded to show the sketch it was based on.

3   Click **Move/size features** 🔀 on the Features toolbar.
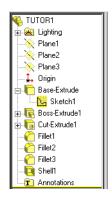
The feature handles for the extruded feature are displayed. Feature handles allow you to move, rotate, and resize some types of features.

4   Drag the **Resize** ⇐●⇒ handle to increase the depth of the extrusion from 30mm to 50mm.
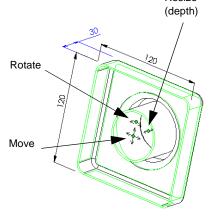
Watch the pointer for feedback about the dimension you are changing. When you release the pointer, the part rebuilds using the new dimension.
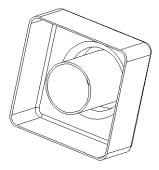
5   Click **Move/size features** 🔀 to turn off the features handle display.

6   To hide the dimensions, click anywhere outside the part in the graphics area.

7   Click **Save** 💾 to save the part.

For more information about feature handles, see Chapter 5, "Working with Parts," in the *SolidWorks 99 User's Guide* and online help.

Resize
(depth)

Rotate

Move

Displaying a Section View

You can display a 3D section view of the model at any time. You use model faces or planes to specify the section cutting planes. In this example, you use **Plane3** to cut the model view.

1 Click **Isometric** 🔵, then click **Shaded** 🔲 view mode.

2 Click **Plane3** in the FeatureManager design tree.

3 Click **Section View** 🔲 on the View toolbar, or click **View, Display, Section View**.

The **Section View** dialog box appears.

4 Specify **Section Position** of 60mm.
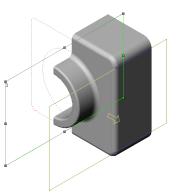
This is the offset distance from the selected plane to the section cut.

5 Click **Preview**.

When this option is selected, the view is updated each time you change a value in the dialog box.

Notice the arrow direction.

6 Click **Flip the Side to View** to cut the section in the opposite direction.
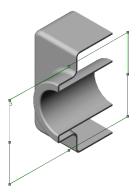
7 Click **OK**.

The section view of the part is displayed. Only the display of the part is cut, not the model itself. The section display is maintained if you change the view mode, orientation, or zoom.

8 To return to a display of the complete part, click **View, Display**, and click to clear the **Section View** check box.

– or –

Click **Section View** 🔲 again.
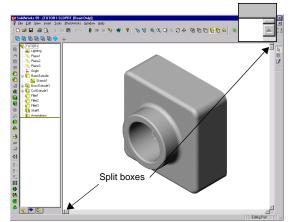
Displaying Multiple Views

You can display as many as four different views of the part in a single window, including section views and named views. This is useful when you want to select features on opposite sides of the part, or when you want to see the effect of an operation from different sides of the model simultaneously.
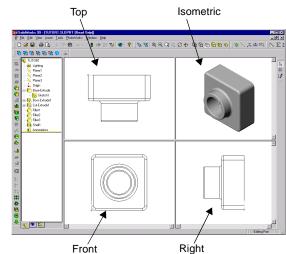
When you select a feature in one view, it is selected in all the views.

1   Drag one or both of the split boxes at the corners of the window to create panes.

Split boxes

2   Drag the split bars as needed to adjust the size of the panes. The pointer changes to ⊹⊹ when it is on a split bar.

3   Click in a pane, and change the view mode, zoom, or orientation of the view in that pane.

4   Repeat for each pane.

5   To return to a single view, drag the split bars to the side, leaving the desired view visible.
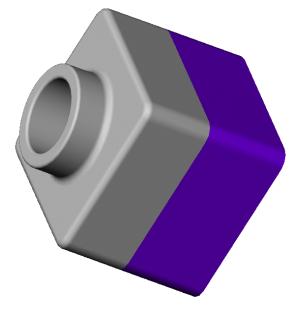
   You can adjust the width of the FeatureManager design tree pane in the same way. Place the pointer on the vertical split bar, and drag as needed.

Top   Isometric

Front   Right

Creating an Assembly

In this chapter, you define a simple *assembly*. The steps include:

❑ Building another part

❑ Adding parts to the assembly (the new part, and the part from Chapter 2)

❑ Specifying the assembly mating relations that make the parts fit together

Creating the Base Feature

You can use the same methods you learned in Chapter 2 to create the base for a new part.

1   Click **New** ⬜ or **File, New** and create a new part document.

2   Click **Sketch** 🖊, and sketch a rectangle beginning at the origin.

3   Click **Dimension** ✏️, and dimension the rectangle to 120mm x 120mm.

4   Click **Extruded Boss/Base** 🔲, and extrude the rectangle as a **Solid Feature,** with a **Type** of **Blind,** to a **Depth** of 90mm.

5   Click **Fillet** 🔲, and fillet the four edges shown with a radius of 10mm.

6   Click **Shell** 🔲. Select the *front* face of the model as the face to remove, and set the **Thickness** to 4mm.

7   Save the part as **Tutor2**. (The **.sldprt** extension is added to the file name.)

Using the Selection Filter

The **Selection Filter** allows you to more easily select the item you want in the graphics area. To show or hide the Selection Filter toolbar, click **Toggle Selection Filter Toolbar** 🔳 on the Standard toolbar, or press **F5**.

The first three buttons on the Selection Filter toolbar behave as follows:

🔽	Turns the **Selection Filter** on or off.
🔽	Clears all of the selected filters.
🔽	Selects all of the filters.

The rest of the buttons are filters. Select the filters that match the items you want to select in the graphics area.

> **TIPS:**   While the **Selection Filter** is active, the pointer changes to 🔽.
>
> After using the **Selection Filter**, click **Clear All Filters** 🔽 so that you will not be limited to the selected filters the next time you want to select items.

For more information about the **Selection Filter**, see Chapter 1 of the *SolidWorks 99 User's Guide* and online help.

Creating a Lip on the Part

In this section, you use the **Convert Entities** and **Offset Entities** tools to create sketch geometry. Then a cut creates a lip to mate with the part from Chapter 2.

> **TIP:** Use the **Selection Filter** to make selecting the faces in this section easier.

1 Zoom in on a corner of the part, select the thin wall on the front face of the part, and click **Sketch** [icon] to open a sketch.

The edges of the part face are highlighted.

2 Click **Convert Entities** [icon] on the Sketch Tools toolbar, or **Tools, Sketch Tools, Convert Entities**.

The outer edges of the selected face are projected (copied) onto the sketch plane as lines and arcs.

3 Click the front face again.

4 Click **Offset Entities** [icon] on the Sketch Tools toolbar or **Tools, Sketch Tools, Offset Entities.**

The **Offset Entities** dialog box appears.

5 Set the **Offset** distance to 2.00mm.
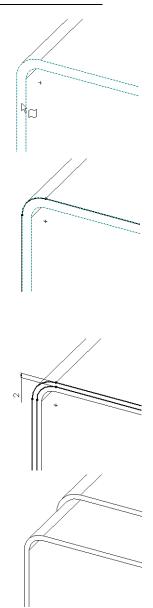
The preview shows the offset extending outward.

6 Click **Reverse** to change the offset direction.

7 Click **Apply**, then click **Close**.

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

8 Click **Extruded Cut** [icon] or **Insert**, **Cut, Extrude**.

9 In the **Extrude Cut Feature** dialog box, set the **Depth** to 30mm, and click **OK.**

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

Changing the Color of a Part

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

1   Click the **Tutor2** icon at the top of the FeatureManager design tree.

2   Click **Edit Color** ![icon] on the Standard toolbar.

The **Edit Color** dialog box appears.

3   Click the desired color on the palette, then click **OK**.

In **Shaded** mode ![icon], the part is displayed in the new color.

4   Save the part.

Creating the Assembly

Now create an assembly using the two parts.
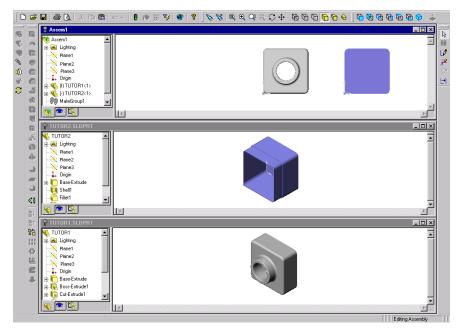
1   If **Tutor1.sldprt** (from Chapter 2) is not open, click **Open** ![icon] on the Standard toolbar and open it.

2   Click **New** ![icon] on the Standard toolbar, then select **Assembly** and click **OK**.

3   Click **Window, Tile Horizontally** to display all three windows. Close any extra windows.

4   Drag the **Tutor1** icon from the top of the FeatureManager design tree for **Tutor1.sldprt**, and drop it *in the FeatureManager design tree* of the assembly window (**Assem1**).

Notice that as you move the pointer into the FeatureManager design tree, the pointer changes to ![icon] .

Adding a part to an assembly this way results in the part automatically inferencing the assembly origin. When a part inferences the assembly origin:

- the part's origin is coincident with the assembly origin.
- the planes of the part and the assembly are aligned.

5   Drag the **Tutor2** icon from **Tutor2.sldprt,** and drop it *in the graphics area* of the assembly window, beside the **Tutor1** part.

Notice that as you move the pointer into the graphics area, the pointer changes to ![icon] .

6 Save the assembly as **Tutor.** (The **.sldasm** extension is added to the file name.) If you see a message about saving referenced documents, click **Yes**.

7 Drag a corner of the assembly window to enlarge it, or click **Maximize** ▢ in the upper-right corner to make the window full size. You no longer need to have the **Tutor1.sldprt** and **Tutor2.sldprt** windows in view.

8 Click **Zoom to Fit** ▣.

9 If the dimensions are displayed, right-click the **Annotations** folder 🇹 in the FeatureManager design tree, and *deselect* **Show Feature Dimensions**.

Manipulating the Components

When you add a part to an assembly, the part is referred to as a *component* of the assembly. You can move or rotate the components individually or together using the tools on the Assembly toolbar.

The first component you add to an assembly is *fixed* in place by default. A fixed component has the prefix **(f)** in the FeatureManager design tree. You cannot move or rotate a fixed component unless you *float* (unfix) it first.

❑ To float a fixed component, right-click the component in either the FeatureManager design tree or in the graphics area, then select **Float** from the shortcut menu. The prefix changes to **(-)**, indicating that the component's position is under defined.

❑ To move and rotate a component in the assembly, you can use the following tools on the Assembly toolbar.

Click **Move Component**, click the component's name in the FeatureManager design tree or click one of the component's faces, then move the component.

Click **Rotate Component Around Centerpoint**, click the component's name in the FeatureManager design tree or click one of the component's faces, then rotate the component.

Both the **Move Component** and **Rotate Component Around Centerpoint** tools remain active so that you can move other non-fixed components in succession.

Hold down **Ctrl** and click both the component and an axis, linear edge, or sketched line. Then click **Rotate Component Around Axis**, and rotate the component.

If the axes are not currently displayed, click **View, Axes** (for user-defined axes) or **View, Temporary Axes** (for axes defined implicitly by the software.)

❑ To exit from move or rotate mode, you can:

- Click the tool again.

- Click another tool.

- Click **Tools, Select**.

- Click **Select** from the shortcut menu or the toolbar.

- Press **Esc**.

❑ To change the orientation of the entire assembly in the graphics area, use the tools on the Standard Views toolbar.

❑ To scroll or rotate the entire assembly in the graphics area, use the **Pan** ⊞ and **Rotate View** ⟳ buttons on the View toolbar.

Mating the Components

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

1 Click **Isometric** 🔷 on the Standard Views toolbar.

2 Click **Mate** 🖉 on the Assembly toolbar, or click **Insert, Mate**.

The **Assembly Mating** dialog box appears.

3 Click the top edge of **Tutor1**, then click the outside edge of the lip on the top of **Tutor2**.

The edges are listed in the **Items Selected** list.

4 Select **Coincident** under **Mate Types,** and **Closest** under **Alignment Condition**.

5 Click **Preview** to preview the mate.

The selected edges of the two components are made coincident.

6 Click **Apply**.

Select these edges

The position of the **Tutor2** component in the assembly is not fully defined, as shown by the **(-)** prefix in the FeatureManager design tree. **Tutor2** still has some degrees of freedom to move in directions that are not yet constrained by mating relations.

1 Click **Move Component** 🔄, then click the **Tutor2** component.
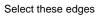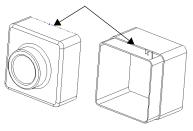
Notice the pointer shape ✛.

2 Drag the component from side to side, then use one of the methods discussed in the previous section to exit move mode.

3 Select **Tutor2**, hold down **Ctrl**, select the mated edge, and click **Rotate Component Around Axis** 🔁.

Notice the pointer shape 🔁 .

4 Drag to rotate the component around the mated edge, then exit rotate mode.

Adding More Mates
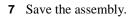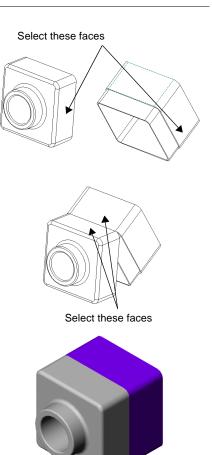
1 Select the rightmost face of one component, then hold down **Ctrl,** and select the corresponding face on the other component.

2 Click **Mate** 🖾 or **Insert, Mate**.

3 In the **Assembly Mating** dialog box, select **Coincident** and **Closest** again.

4 Click **Preview** to preview the mate.

5 Click **Apply**.

6 Repeat Steps 1 through 5, selecting the top faces of both components, to add another **Coincident** mate.

Select these faces

Select these faces

7 Save the assembly.

4

Drawing Basics

In this chapter, you create a multi-sheet drawing of the parts and assembly from the previous chapters. This chapter includes:

❑ Opening and editing a *drawing template*
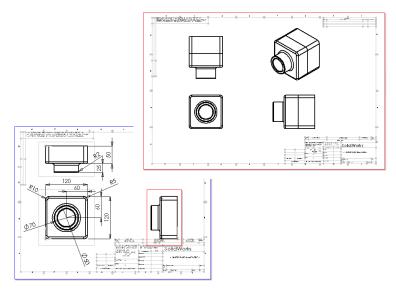
❑ Inserting *standard views* of a part model

❑ Adding *model and reference annotations*

❑ Adding another *drawing sheet*

❑ Inserting a *named view*

❑ Inserting, moving, editing, and saving a *bill of materials*

Opening a Drawing Template

First you prepare the drawing template for one of the parts you created.

1   Click **New** 🗋 on the Standard toolbar.

2   Select **Drawing** and click **OK**.

The **Template to Use** dialog box appears.

3   Under **Standard Template**, select **A-Landscape**.

4   Click **OK**.

A new drawing window appears, with note text informing you that you can create your own template, or modify this one, and to see online help for more information about modifying templates.

The Drawing toolbar is also displayed.

5   Right-click anywhere in the drawing, and select **Edit Template** from the shortcut menu.

6   Click the note text to select it, and press the **Delete** key. Click **Yes** to confirm the delete.

7   Zoom in on the title block, then double-click the text **< INSERT YOUR COMPANY NAME HERE >**.

The **Properties** dialog box appears.

8   Change the **Note text** to the name of your company.

9   Click **Font**. In the **Choose Font** dialog box, choose a different font, style, or size, then click **OK**.

10   Click **OK** to close the **Properties** dialog box.

11   To save this as the standard **A-Landscape** template, click **File, Save Template**, and click **OK.** The default extension for a drawing template is **.slddrt**.

Click **Yes** to confirm that you want to overwrite the existing template. The next time you choose this template, you will not need to perform these edits again.

> **NOTE:**   If you want to save the template with a new name (*not* to overwrite the standard template), click **File, Save Template, Custom Template**. Click **Browse** and navigate to the directory where you want to save the template. Type a name and click **Save**. Click **OK** to close the **Save Template** dialog box.

Setting the Detailing Options

Next, set the default dimension font, and the style of dimensions, arrows, and so forth. For this tutorial, use the settings described below. Later, you can set the Detailing options to match your company's standards.

1 Click **Tools**, **Options**.

2 Click the **Detailing** tab.

3 In the **Dimensioning Standard** section, in the **Trailing Zeroes** box, select **Show**.

4 In the **Dimensions** section, click **Dim Font**.

The **Choose Font** dialog box appears.

5 Click **Points**, and type or select 16.

6 Click **OK**.

7 Click the **Arrows** button, and review the default styles and sizes.

Notice the different attachment styles for edges, faces, and unattached items.

8 Click **OK.**

9 Click **OK** again to close the **Options** dialog box.

For more information about these options, see Chapter 9, "Drawings," and Chapter 10, "Detailing," of the *SolidWorks 99 User's Guide* and online help.

Creating a Drawing of a Part

1   If **Tutor1.sldprt** is not still open, open the part document now. Then return to the drawing window.

2   Right-click anywhere in the drawing, and select **Edit Sheet**.

3   Click **Standard 3 View** 🖼 in the Drawing toolbar, or click **Insert**, **Drawing View, Standard 3 View**.

Notice the pointer ⬚, and the message in the status bar, "Select the model to display in the drawing."

4   From the **Window** menu, select **Tutor1.sldprt**.

The **Tutor1.sldprt** window comes forward.
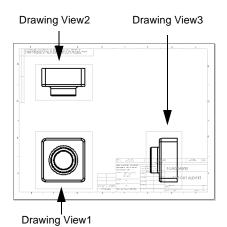
5   Click in the graphics area of the part window.
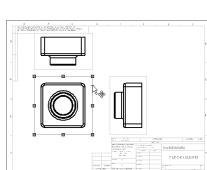
The drawing window returns to the front with three views of the selected part.

To move a view, click inside its boundary, then drag it by its green border. The pointer changes to ⬚ when it is at the border of a selected view.

Drawing View2 and **Drawing View3** are aligned to **Drawing View1,** and only move in one direction to preserve the alignment.

- To move **Drawing View2** vertically, drag up and down.

- To move **Drawing View3** horizontally, drag sideways.

- To move all the views together, click **Drawing View1** and drag in any direction.

6   Move the views on the drawing sheet.

Drawing View2   Drawing View3

Drawing View1

Adding Dimensions to a Drawing

Drawings contain 2D views of models. You can choose to display dimensions that are already specified in the model in all of the drawing views.
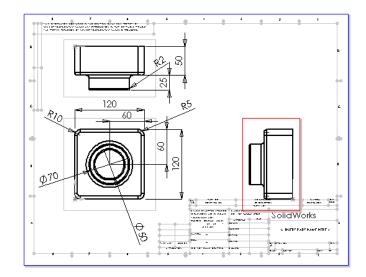
1 With nothing selected, click **Insert**, **Model Items**.

The **Insert Model Items** dialog box appears. You can select which types of dimensions, annotations, and reference geometry to import from the model.

2 Make sure that **Dimensions** and **Import Items into All Views** are selected, and click **OK**.

Dimensions are imported into the view where the feature they describe is most visible. Only one copy of each dimension is imported.

3 Drag the dimensions to position them.

> **TIP:** Select a drawing view, then click **Zoom To Selection** 🔍 to zoom the view to fill the screen. Click **Zoom to Fit** 🔍 to see the entire drawing sheet.

4 Click **Save** 💾, and save the drawing document as **Tutor1**. The default extension is **.slddrw**.
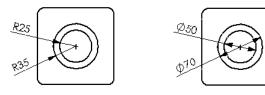
Dimensioning Tips for Drawings
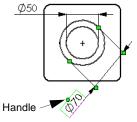
❑ To *remove* an unwanted dimension, select it and press the **Delete** key.

❑ To *hide* a dimension, click **View, Hide/Show Dimensions**, then click the dimensions you want to hide. You can toggle a dimension's visibility by clicking it again with the hide or show pointer ◐.

❑ To *move* a dimension to another view, click the dimension, hold down **Shift**, and drag the dimension to the desired location within the destination view boundaries. (Do not drag by the handles when doing this.)

❑ To *copy* a dimension to another view, click the dimension, hold down **Ctrl**, and drag the dimension to the desired location within the destination view boundaries. (Do not drag by the handles when doing this.)

❑ To *center* the dimension text between the witness lines, right-click the dimension, and select **Center text**.

❑ For dimensions on circular features, you have these options:

  • To change a *radius* dimension to a *diameter* dimension, right-click the dimension, and select **Display As Diameter**.

  • To display a *diameter* dimension as a *linear* dimension, right-click the dimension, and select **Display As Linear**.

  • If the linear dimension is not placed at the angle you want, select the dimension, and drag the green handle on the dimension value. The angle of the witness lines snaps in 15° increments.

Display As Radius (default)   **Display As Diameter**   **Display As Linear**

❑ To modify the appearance of leaders, text, arrows, and so on, right-click the dimension, and select **Properties**. Edit the available options, and click **OK**.

❑ To add reference dimensions in the drawing:

  • Click **Tools, Dimensions**, then choose a dimension type.

    – or –

  • Click **Dimension** and choose a dimension type from the shortcut menu.

  Reference dimensions appear in parentheses by default.

□ To add annotations in the drawing:

- Click **Insert, Annotations**, then choose the type of annotation to add.

  – or –

- Choose a tool from the Annotations toolbar.

For more information about adding and aligning dimensions and annotations in drawings, refer to Chapter 10, "Detailing," of the *SolidWorks 99 User's Guide* and online help.

Modifying Dimensions

When you change a model dimension in the drawing view, the model is automatically updated to reflect the change, and vice versa.

1 In **Drawing View2**, double-click the dimension for the depth of the boss extrusion.

2 In the **Modify** dialog box, change the value from 25mm to 40mm, and press **Enter**.

3 On the Standard toolbar, click **Rebuild** .

The part rebuilds using the modified dimension. Both the drawing and the part model are updated.

Double-click this dimension
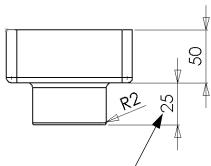
4 Click **Window**, and select the **Tutor1.sldprt** window.

5 Double-click **Boss-Extrude1** in the FeatureManager design tree to display the dimensions of the feature.
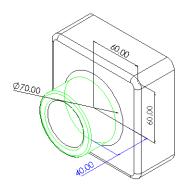
Notice that the depth dimension is 40mm.

6 Return to the drawing window, and save the drawing.

The system notifies you that the model referenced in the drawing has been modified, and asks if you want to save it.

7 Click **Yes** to save both the drawing and the updated model.

Now rebuild the assembly that contains the modified part.

1 Click **Window**. If **Tutor.sldasm** is not still open, open it now. Otherwise, switch to the **Tutor.sldasm** window.

If a message appears asking you if you want to rebuild the assembly, click **Yes**.
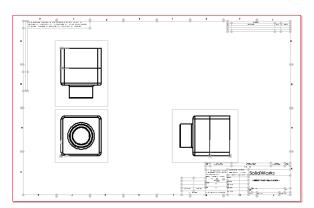
2 Return to the drawing window.

Adding Another Drawing Sheet

Now you create an additional drawing sheet for the assembly, including the standard three views, and an isometric view.

1 Click **Insert, Sheet**, or right-click the sheet tab at the bottom of the window, and select **Add**.

2 In the **Sheet Setup** dialog box, under both **Paper size** and **Template**, select **B-Landscape**, and click **OK**. Edit the template as described on page 4-2.

3 To bring the assembly into the drawing sheet, use one of the following methods:

- Click **Standard 3 View** 🖼, right-click in the graphics area, and select **Insert From File**. Then navigate to **Tutor.sldasm** in the **Insert Component** dialog box, and click **Open**.

   – or –

- Cascade or tile the windows, then drag the **Tutor** assembly icon from the top of the FeatureManager design tree of the assembly window into the drawing window. (By default, the standard three views are added when you use drag-and-drop.)

4 Reposition the views on the sheet if needed.

If the drawing sheet is too small, you can choose a different size.

1 Right-click in a blank area of the drawing window (not inside the boundaries of a view) and select **Properties**.

2 Select a different **Paper size** or **Template**.

3 Click **OK**.

Inserting a Named View

You can add named views to drawings, showing the model in different orientations. You can use:

- a standard view (**Front**, **Top**, **Isometric**, and so on)
- a named view orientation that you defined in the part or assembly
- the current view in the part or assembly document

Zoom levels are ignored, however, and the entire model is always displayed in the selected orientation.

In this section you add an isometric view of the assembly.

1 Click **Named View** 🔲 or **Insert, Drawing View, Named View**.

The pointer 🔍◇ indicates that you may select a model to display in the drawing.

2 To select the model to display, right-click in the graphics area, and select **Insert From File**. Then navigate to **Tutor.sldasm** in the **Insert Component** dialog box, and click **Open**.

The **Drawing View - Named View** dialog box appears. Note its similarity to the **Orientation** dialog box.
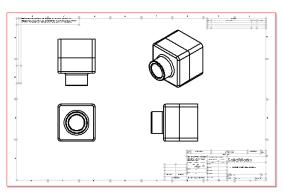
3 Select ***Isometric** from the list, then click **OK**. If you are in the assembly window, return to the drawing window.

The pointer ✛ indicates that you may select a location in the drawing to place the named view.

4 Click where you want to place the view.

If a message appears asking you if you want to switch the view to use isometric (true) dimensions, click **Yes.**

5 If any origins appear in the drawing, click **View, Origins** to turn them off.
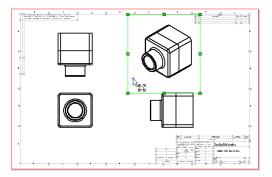
Inserting a Bill of Materials

You can insert a bill of materials (BOM) into the drawing of an assembly.

> **NOTE:** You must have the Microsoft® Excel 97 spreadsheet program installed on your computer to insert a bill of materials into a drawing.

Because a drawing can contain views of different parts and assemblies, you must pre-select the view for which you want to create a bill of materials.

1 With **Sheet2** still active, select one of the views.

2 Click **Insert, Bill of Materials**.
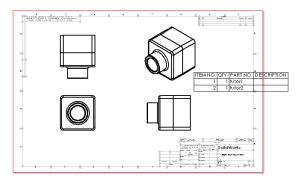
The **Select BOM Template** dialog box is displayed.

3 Click **Open** to use the bill of materials template file, **Bomtemp.xls**.

The **Bill of Materials Properties** dialog box is displayed.

4 Make sure that the **Use the document's note font when creating the table** check box is selected, click to clear the **Use table anchor point** check box, and click **OK**.

A bill of materials is displayed that lists the parts in your assembly.

Moving a Bill of Materials

You can move the bill of materials to a new location on the drawing to match your company's standards.

1 Click the bill of materials.

The pointer changes to the move shape ⬚⬚ .

2 Drag the worksheet to a new location.

For information about attaching a bill of materials to an anchor point, see Chapter 10, "Detailing," in the *SolidWorks 99 User's Guide* and online help.
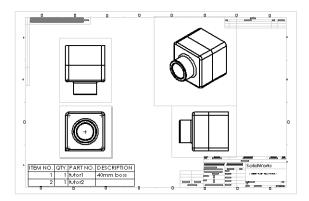
Editing a Bill of Materials

Next, enter a description for **Tutor1**.

1 Right-click the bill of materials and select **View BOM Table**.

While the bill of materials is active, it is displayed with shaded borders and row and column headers. Excel toolbars replace the SolidWorks toolbars.

2 Drag the lower-right corner of the border to resize the worksheet to see all the rows.

3 Click in cell **D2**, type a description (such as **40mm boss**), then press **Enter.**

4 Click outside the drawing sheet to close it and to return to editing the drawing sheet.

Saving a Bill of Materials

You can save the bill of materials as an Excel file for use with other applications.

1   Click the bill of materials.

2   Click **File, Save As**. The **Save Bill of Materials Table** dialog box is displayed. Notice that the **Save as type** is set to **Excel Files (*.xls)** by default.

3   Type **Tutor1_BOM** in **File name** and click **Save**.

The extension **.xls** is added to the filename, and the file is saved to the current directory. If you wish, you can navigate to a different directory, then save the file.

> **NOTE:**   The Excel file is not linked to the bill of materials in the drawing. If assembly components change, the bill of materials automatically updates, but the Excel file does not.

For more information about adding a bill of materials, see Chapter 10, "Detailing," of the *SolidWorks 99 User's Guide* and online help.

Printing the Drawing
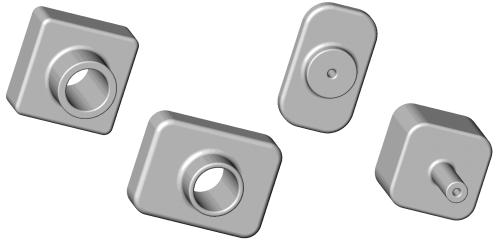
1   Click **File, Print**. The **Print** dialog box appears.

2   Set **Print range** to **All**, and make sure that the **Scale to Fit** check box is selected.

3   Click **OK** to close the **Print** dialog box and print the drawing.

4   Click **Save** , then close the drawing.

5

Using a Design Table

In this chapter you use a design table to create several variations of the part you designed in Chapter 2, "The 40-Minute Running Start." To use a design table, you must have Microsoft Excel 97 on your system.

This exercise demonstrates the following:

❑ *Renaming* features and dimensions

❑ *Displaying* feature dimensions

❑ *Linking values* of model dimensions

❑ *Verifying* geometric relations

❑ Creating a *design table*

❑ Displaying part *configurations*

Renaming Features

It is a good practice to give meaningful names to the features in your parts, especially when you plan to use a design table. This can save confusion in complex parts, and it is helpful to other people who use the parts later.

1   Open the part called **Tutor1.sldprt** that you created in Chapter 2.

2   Change the generic name **Base-Extrude** to something more meaningful.

> **NOTE:** Feature names cannot contain the @ character.

   a)   Click *two times* on **Base-Extrude** in the FeatureManager design tree (do not double-click; you must pause slightly between clicks).

   b)   When **Base-Extrude** is highlighted in a box, type the new name, **Box**, and press **Enter**.

3   Rename these other features:
   - **Boss-Extrude1 => Knob**
   - **Cut-Extrude1 => Hole_in_knob**
   - **Fillet1 => Outside_corners**

4   Save the part as **Tutor3.sldprt**.

> **TIP:**   To give descriptive names to features as you create them, click **Tools, Options**, and select the **General** tab. Select the **Name feature on creation** check box in the **FeatureManager Design Tree** section. Each time you create a new feature, the name of the new feature in the FeatureManager design tree is automatically highlighted, and ready for you to enter a new name.

Displaying Dimensions

You can display or hide all the dimensions for all the features of the part. Then you can turn the display of dimensions on and off, either individually, or on a feature-by-feature basis.

1   To display all the dimensions for the part, right-click the **Annotations** 🅣 folder in the FeatureManager design tree, and select **Show Feature Dimensions**. Notice that the dimensions that are part of a feature's definition (such as the depth of an extruded feature) are blue.

2   To hide the dimensions for the **Fillet2**, **Fillet3**, and **Shell1** features, right-click each feature in the FeatureManager design tree or in the model, and select **Hide All Dimensions**.

3 To display the dimension names along with the values in the model, click **Tools, Options,** and select the **General** tab. In the **Model** section, select the **Show dimension names** check box, and click **OK**.

Linking Values

There are several ways of defining equality between model dimensions, including relations, equations, or linked values.

❑ **A geometric relation**. You can add an **Equal** geometric relation between sketch entities, or between a sketch entity and a model edge.

❑ **An equation**. In any equation, the right side drives the left side (*driven = driving*); only the driving dimension may be modified.

❑ **Linked values**. This is a way to control values that are not part of a sketch, such as the depth of two extruded features. You cannot use a geometric relation for these values.

In a sketch or otherwise, for any type of dimension, linking values works better than an equation for simple equality. You can change either value; you do not have to remember which dimension is driving.

You link dimensions by assigning them the same variable name. Then you can modify the value of *any* of the linked dimensions, and all of the other dimensions with the same variable name change accordingly. You can unlink any of the dimensions without affecting the ones that you want to remain linked.

For this example, you set the extrusion depth of the **Box** and the **Knob** to be equal:

1 Right-click the dimension for the extruded depth (50.00mm) of the **Box**, and select **Link Values**. In the **Shared Values** dialog box, type **depth** in the **Name** box, and click **OK**.

2 Right-click the dimension for the depth (40.00mm) of the **Knob**, and select **Link Values**. Click the arrow beside the **Name** box, select **depth** from the list, and click **OK**. (Each time you define a new **Name** variable, it is added to this list.)

Notice that the two dimensions now have the same name, **depth**.
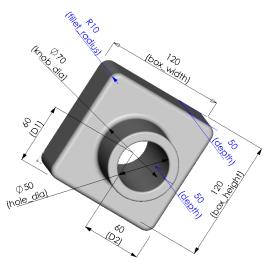
3 Click **Rebuild** �'8'🔲 to rebuild the part.

Renaming Dimensions

You can change individual dimension names. Renaming dimensions is a good practice, and it is especially useful when you plan to use a design table. You use the dimension names to identify the elements you plan to change, and as headings in the design table worksheet.

1 Change the name of the knob diameter dimension:

a) Right-click the **Knob** diameter dimension (70.00mm), and select **Properties**.

b) In the **Dimension Properties** dialog box, select the text in the **Name** box and type in a new name, **knob_dia**. Notice that the **Full name** box is updated also.

c) Click **OK**.

2 Rename the height of the box (120.00mm) to **box_height**.

3 Rename the width of the box (120.00mm) to **box_width**.

4 Rename the diameter of the hole in the knob (50.00mm) to **hole_dia**.

5 Rename the radius of the outside corners (10.00mm) to **fillet_radius**.
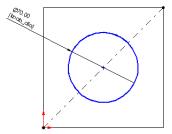

6 Save the part.

Verifying Relations

Before you proceed, you should define some geometric relations that ensure that the knob is positioned correctly with respect to the center of the box, regardless of the size. Relations add to the integrity of the design, and they are often the most effective way to convey the design intent accurately.

1 In the FeatureManager design tree or the model, right-click the **Knob** feature, and select **Edit Sketch**.

2 Click **Hidden Lines Removed** 🗋, and click **Normal To** 🔱.

3 Delete the dimensions (60.00mm) between the circle and the sides of the box.

4 Click the centerpoint of the circle, and drag the circle to one side temporarily.

5 Click **Centerline** ⫶, and sketch a diagonal centerline as shown.

6 Add a midpoint relation between the centerline and the circle:

    a) Click **Add Relation** ⊥ or **Tools, Relations, Add.**

    b) Click the centerpoint of the circle and the centerline.

    c) Click **Midpoint**, and click **Apply.**

    d) Click **Close**.

Now verify the relations in this sketch:

1 Click **Display/Delete Relations** 🔛 or **Tools, Relations, Display/Delete.**

2 Click **Next** or **Previous** ⬅➡ in the **Display/Delete Relations** dialog box to review all the relations in the sketch. As you display each relation, the entities are highlighted in the graphics area. Click the **Entities** tab for more information about the highlighted entities.

3 Click **Close** to close the **Display/Delete Relations** dialog box.

> **NOTE:** If a sketch entity is selected when you click **Display/Delete Relations**, only the relations on the selected entity are listed. Click a different entity to display its relations. You can change the **Criteria** in the **Display relations by** box to specify the types of relations (**All, Dangling**, and so on) that are displayed.

4 Click **Sketch** 🖊 to close the sketch.

5 Save the part.

Inserting a New Design Table

If you have Microsoft Excel 97 on your computer, you can use it to embed a new design table directly in the part document. A design table allows you to build several different *configurations* of a part by applying the values in the table to the dimensions of the part.

1 Click **Tools, Options**, **General**, make sure that the **Edit Design Tables in separate Window** check box is not selected, and click **OK**.

2 Click **Isometric** , and make sure that you can see all of the part's dimensions in the graphics area. After resizing and repositioning the part, click **Select** to deselect any active View tool.

3 Click **Insert, New Design Table**.

An Excel worksheet appears in the part document window. Excel toolbars replace the SolidWorks toolbars. By default, the first row (cell A3) is named **First Instance**, and column header cell B2 is active.

4 Double-click the **box_width** dimension *value* (120) in the graphics area.

Notice that the pointer changes to when it is over a dimension value.

The dimension name and value are inserted in column B. The adjacent column header cell, C2, is activated automatically.

TIP: To uncover dimensions hidden by the design table, point at the Excel worksheet's shaded border and drag the worksheet to another location in the graphics area. To resize the worksheet, drag the handles at the corners or sides.

---

5 To insert the rest of the dimension names and values shown in the following illustration, double-click each dimension value in the graphics area.

NOTE: If you see **$STATE@** followed by a feature name in a column header cell, you selected a face instead of a dimension value in the graphics area. To replace a feature name with a dimension name, click the cell in the worksheet, then double-click the correct dimension value in the graphics area.

---

6 Name the rows (cells A4:A6) **blk2** through **blk4**. These are the names of the configurations that the design table produces.

	A	B	C	D	E	F	G
2		box_width@Sketch1	box_height@Sketch1	knob_dia@Sketch2	hole_dia@Sketch3	fillet_radius@Outside_corners	depth@Knob
3	First Instance	120	120	70	50	10	50
4	blk2						
5	blk3						
6	blk4						

Sheet1

7 Type the following dimension values into the worksheet:

	A	B	C	D	E	F	G
2		box_width@Sketch1	box_height@Sketch1	knob_dia@Sketch2	hole_dia@Sketch3	fillet_radius@Outside_corners	depth@Knob
3	First Instance	120	120	70	50	10	50
4	blk2	120	90	50	40	15	30
5	blk3	90	150	60	10	30	15
6	blk4	120	120	30	10	25	90

Sheet1

8 To close the worksheet and create the configurations, click anywhere outside the worksheet in the graphics area.

An informational dialog box appears, listing the new configurations created by the design table. Click **OK** to close the dialog box.

The design table is *embedded* and saved in the part document.

9 Save the part.

Viewing the Configurations

Now look at each of the configurations generated by the table.

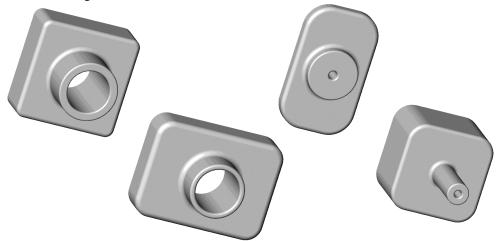**1** Click the Configuration tab at the bottom of the FeatureManager design tree.

The list of configurations is displayed.

2 Double-click the name of a configuration.

In the **Confirm Show Configuration** dialog box, select **Don't ask for confirmation again in this session**, and click **OK**.

As you display each of the configurations, the part rebuilds using the dimensions for the selected configuration.

Editing the Design Table

To make changes to the design table:

1 Click **Edit, Design Table**.

2 Make the desired changes.

3 To close the design table, click anywhere in the graphics area outside the design table.

The configurations update as needed to reflect the changes.

> **TIP:** When using this or any other OLE object, you may need to click
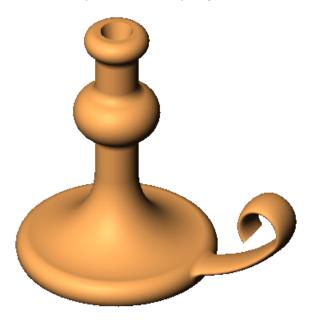> **Zoom to Fit** when returning to the SolidWorks window.

Deleting the Design Table

To delete the design table, click **Edit, Delete Design Table**. Deleting a design table does *not* delete the configurations associated with it.

6

Revolve and Sweep Features

In this chapter, you create a candlestick by performing the following:

❑ Creating a *revolved* feature

❑ Sketching and dimensioning *arcs* and an *ellipse*

❑ Creating a *sweep* feature

❑ Using *relations*
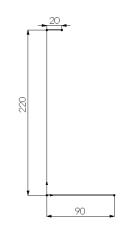
❑ Creating an *extruded cut feature with a draft angle*
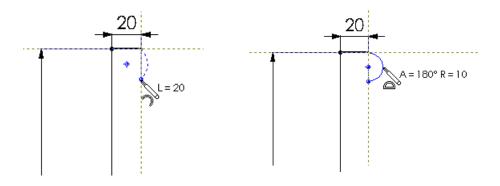
Sketching a Revolve Profile

You create the base feature of the candlestick by
revolving a profile around a centerline.

1 Open a new part document.

2 Click **Sketch** ✐ to open a sketch on **Plane1**.

3 Click **Line** ◣ or **Tools, Sketch Entity, Line**.
Sketch a vertical line through the origin, and sketch
the two horizontal lines as shown.

4 Click **Dimension** ✐ or right-click and select
Dimension from the shortcut menu. Dimension the
lines as shown.

Now sketch and dimension the arcs and lines needed to
complete the profile.

1 Click **3 Pt Arc** ⬠ or **Tools, Sketch Entity, 3 Point Arc**, and point at the endpoint of
the top horizontal line. Drag an arc downward for a length of 20mm (L=20), and
release the pointer. Then drag the highlighted point to adjust the angle of the arc to
180° (A=180°) and the radius to 10mm (R=10). Notice that the centerpoint of the arc
snaps to the vertical inferencing line. Release the pointer.

TIP:	Watch the pointer for feedback and for *inferencing*. As you sketch, inferencing pointers and lines help you align the pointer with existing sketch entities and model geometry. For more information about inferencing, see Chapter 2 of the *SolidWorks 99 User's Guide* and online help.

2 Click **Line** ⬚ or right-click and select **Line**, then sketch a vertical line starting at the lower endpoint of the arc.
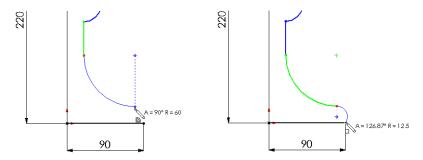
Do not dimension the line at this time.

3 Click **3 Pt Arc** ⬚ or right-click and select **3 Point Arc**, and sketch an arc with the following measurements: length of 40mm, angle of 180°, and radius of 20mm.
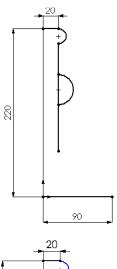
Sketch the arc so that the arc endpoints are coincident with the line.
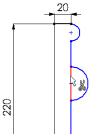
4 Click **Trim** ⬚ or **Tools, Sketch Tools, Trim**, and point at the sketch segment between the endpoints of the arc.

The sketch segment is highlighted in red. Click the highlighted segment to delete it.

5 Right-click and select **Dimension** from the shortcut menu. Dimension the upper vertical line to 40mm.

6 Click **Add Relation** ⬚ or **Tools, Relations, Add**. The **Add Geometric Relations** dialog box appears.

   a) Click the vertical lines on each side of the arc.

   b) Make sure that **Equal** is selected in the **Add Geometric Relations** dialog box.

   c) Click **Apply**, then click **Close**.

7 Click **Tangent Arc** ⬚ or **Tools, Sketch Entity, Tangent Arc**, and point at the endpoint of the lower vertical line. Drag the arc until the angle is 90° and the radius is 60mm. Release the pointer.

8 Sketch another tangent arc. Drag the arc until the endpoint is coincident with the endpoint of the bottom horizontal line.
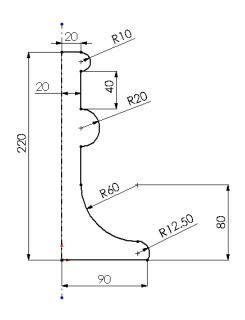
9 Dimension the rest of the sketch as shown.

When you are done dimensioning, the sketch is fully defined. (All lines and endpoints are black.)

10 Click **Centerline** ⬚ or **Tools, Sketch Entity, Centerline**, and sketch a vertical centerline through the origin.

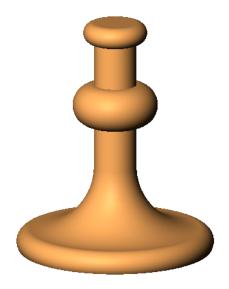This centerline is the axis around which the profile revolves.

Creating the Revolve Feature

1 Click **Revolved Boss/Base** 🔩 on the Features toolbar, or **Insert, Base, Revolve**.

The **Revolve Feature** dialog box appears.

2 Leave the default values of **Type** as **One-Direction**, **Angle** at 360°, and **Revolve as** at **Solid Feature**.

3 Click **OK.**

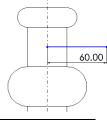**4** Save the part as **Cstick.sldprt**.

Sketching the Sweep Path

A sweep is a base, boss, or cut created by moving a *section* along a *path*. In this example, you create the candlestick handle by using a sweep.

First, you sketch the sweep path. The path can be an open curve, or a closed, non-intersecting curve. Neither the path nor the resulting sweep may self-intersect.

1 Click **Plane1**, then click **Sketch** to open a new sketch.

2 Click **Front** on the Standard Views toolbar, and click **Hidden Lines Removed** on the View toolbar.

3 Click **View, Temporary Axes**. Notice that the temporary axis of the revolved base appears.

4 Right-click and select **Line**. Point at the temporary axis.

The pointer changes to indicating that the pointer is exactly on the temporary axis.
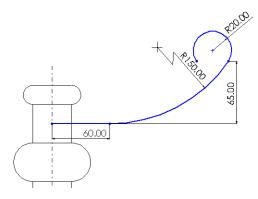
5 Sketch a horizontal line as shown, and dimension the line to 60mm.

6 Select **Tangent Arc** from the shortcut menu, and sketch an arc. Dimension the arc to a radius of 150mm.

TIP: If the centerpoint of a radial dimension is out of view, right-click the dimension, and select **Properties**. Select the **Foreshortened radius** check box, then click **OK**.

---

7 Select the endpoints of the tangent arc, and set the vertical dimension to 65mm.

TIPS: As you move the pointer, the dimension snaps to the closest orientation. When the preview indicates the dimension type and location you want, right-click to lock the dimension type. Click to place the dimension.

---

8   Select **Tangent Arc** from the shortcut menu, and sketch another arc as shown. Dimension it to a radius of 20mm.

9   Click **Add Relation** ⊥ or **Tools, Relations, Add**. The **Add Geometric Relations** dialog box appears.

   a)   Click the endpoints of the tangent arc you just sketched.

   b)   Make sure that **Horizontal** is selected in the **Add Geometric Relations** dialog box.

   c)   Click **Apply**, then click **Close**.

   The dimensions and relations prevent the sweep path from changing size and shape when moved.

10   Click **Display/Delete Relations** 📇 or **Tools, Relations, Display/Delete**.
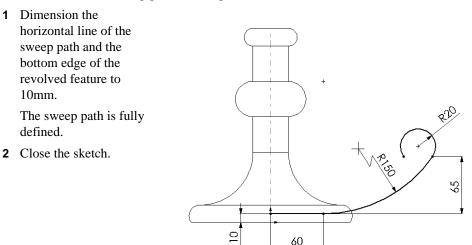
   The **Display/Delete Relations** dialog box appears. It lists all of the relations in the current sketch, including relations that are added automatically as you sketch and relations that you add manually.

11   In the **Display relations by** box, make sure that **Criteria** is selected, and that **All** is selected in the **Criteria** box.

12   Using the **Next** ➡ or **Previous** ⬅ buttons, view each relation.

13   When **Type** is **Coincident,** click the **Entities** tab, then click each item listed beneath **Entity**.

   The coincident relation was added automatically between the sweep path and the revolved base. The line entity is related to an entity outside the current sketch. The **External information** section lists the external entity to which the line entity has a relation. The point entity exists in the current sketch.

14   Click **Close**.

Next, dimension the sweep path with respect to the revolved base.

1 Dimension the horizontal line of the sweep path and the bottom edge of the revolved feature to 10mm.

The sweep path is fully defined.

2 Close the sketch.

Sketching the Sweep Section

1 Select **Plane3** from the FeatureManager design tree, then click **Sketch** 📝 to open a new sketch.

2 Click **Normal To** 🔷 on the Standard Views Toolbar.

3 Click **Ellipse** ⬯ or **Tools, Sketch Entity, Ellipse,** and sketch an ellipse anywhere.
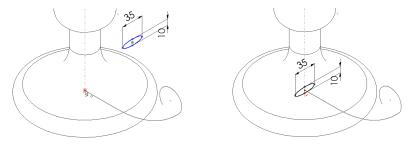
> **TIP:** To sketch an ellipse, drag horizontally from the centerpoint of the ellipse to set the width of the ellipse, release the pointer, then drag vertically to set the height.

4 Dimension the ellipse as shown.

5 Click **Add Relation** ⊥ or **Tools, Relations, Add**.

6 Click both side points of the ellipse and add a **Horizontal** relation. This relation ensures that the ellipse is not slanted.

7 Click **Isometric** 🔷.

8   Click the center point of the ellipse and the endpoint of the horizontal line of the sweep path. Click **Coincident**, click **Apply**, and click **Close**.

This coincident relation ensures that the center point of the sweep section lies on the plane of the sweep path.

9   Click **View, Temporary Axes** to hide the temporary axis.

10 Close the sketch.

11 If the dimensions are displayed in the graphics area, right-click the **Annotations** [T] folder, and deselect **Show Feature Dimensions**.

Creating the Sweep

Now you combine the two sketches to create the sweep.
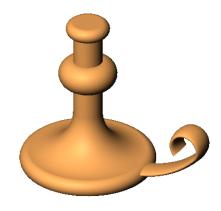
1   Click **Insert**, **Boss**, **Sweep**.

The **Sweep** dialog box appears.

2   Click the **Sweep section** box, then click **Sketch3** in the FeatureManager design tree (or click the ellipse in the graphics area).

3   Click the **Sweep path** box, then click **Sketch2** in the FeatureManager design tree (or click the sweep path in the graphics area).

4   Make sure the **Orientation/Twist control** is set to **Follow path**.

5   Click **OK** to create the sweep.
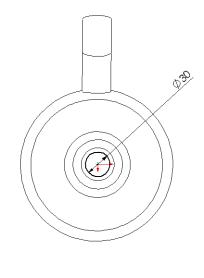
The candlestick's handle is complete.

6   Save the part.

Creating the Cut

Create a cut to hold a candle.

1 Click the top face of the revolved base feature, then click **Sketch** 🖊.

2 Click **Normal To** 🔷.

3 Click **Circle** ⊕ or **Tools, Sketch Entity, Circle**, and point at the sketch origin. Sketch and dimension a circle as shown.

4 Click **Extruded Cut** 🔲 or **Insert, Cut, Extrude**.

- Set **Type** to **Blind**.
- Set **Depth** to 25mm.
- Select **Draft While Extruding**, and specify an **Angle** of 15°.

5 Click **OK**.

6 To see the angled cut, click **Hidden In Gray** 🔲, and rotate the part using the arrow keys.

Adding the Fillets

Add fillets to smooth some of the edges on the part.
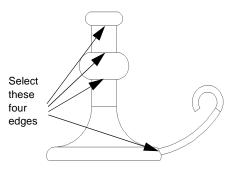
> **TIP:** Use the **Selection Filter** to make selecting the edges in this section easier.

1   Click **Front** , and click **Hidden Lines Removed** .

2   Click **Fillet** or **Insert, Features, Fillet/Round**.

3   In the **Fillet Feature** dialog box, specify a **Radius** of 10mm.
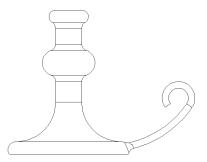
4   Click the four edges indicated.

Select these four edges

Notice the list of edges in the **Edge fillet items** box. If you click the wrong edge accidentally, click the edge in the graphics area again to deselect it, or select the name of the edge in the **Edge fillet items** box and press **Delete**.
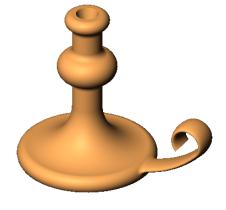
5   Click **OK**.

Fillets are added to each of the selected edges.

6   Click **View Orientation** , and double-click ***Trimetric** in the **Orientation** dialog box.

7   Click **Shaded** .

8   Save the part.

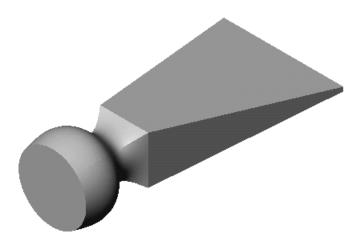# Creating a Loft

In this chapter, you create this chisel using *loft* features.

A loft is a base, boss, or cut created by connecting multiple cross sections, or *profiles*.

The steps for creating this loft include:

❑ Creating planes

❑ Sketching, copying, and pasting the profiles

❑ Creating a solid by connecting the profiles (*lofting*)

Setting Up the Planes

To create a loft, you begin by sketching the profiles on faces or planes. You can use existing faces and planes, or create new planes. For this example, you use one existing plane and create several new planes.

1   Open a new part document.

By default, the planes in a SolidWorks model are not visible. However, you can display them. For this example, displaying **Plane1** is helpful.

2   Click **View,** make sure **Planes** is selected, then right-click **Plane1** in the FeatureManager design tree. Select **Show** from the shortcut menu. (To make it easier to see the planes as you add them, click **View Orientation** , and double-click ***Trimetric**.)

3   With **Plane1** still selected, click **Plane** on the Reference Geometry toolbar, or click **Insert, Reference Geometry, Plane**.

4   Select **Offset** and click **Next**.

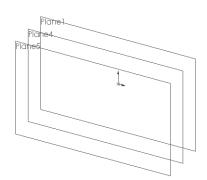**5**   Set the **Distance** to 25mm, and click **Finish**.

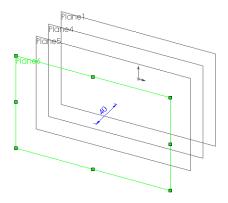A new plane, **Plane4**, is created in front of **Plane1**.

The planes used in a loft do not have to be parallel, but for this example they are.

6   With **Plane4** still selected, click **Plane** again, and add another offset plane at a distance of 25mm (this is **Plane5**).

7   Another way to create an offset plane is to copy an existing plane. Select **Plane5** in the graphics area, hold down **Ctrl**, and drag to a location in front of **Plane5**. Drag the *edge* or the *label*, not the handles. (Dragging the *handles* changes the size of the plane display.)

Another offset plane, **Plane6**, is created.

8   To set the offset distance for the new plane, double-click **Plane6**, change the dimension value to 40mm, and click **Rebuild** .
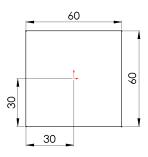
Sketching the Profiles

You create the chisel handle by lofting between simple profile sketches.

1 Click **Plane1** either in the FeatureManager design tree or the graphics area, and click **Sketch** [icon]. Change the view orientation to **Front** [icon].

2 Sketch and dimension a 60mm square as shown.

> **TIP:** To center the dimension text between the witness lines, right-click the dimension, and select **Center text**. If you move the dimension, the text remains centered (unless you drag the text outside the witness lines).

3 Exit the sketch.

For the next profile, you can sketch with grid snapping turned off.
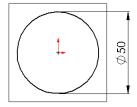
4 Click **Grid** [icon] on the Sketch toolbar.

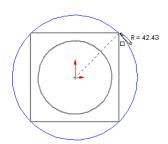The **Options** dialog box appears, with the **Grid/Units** tab displayed.

5 Click to clear the **Snap to points** check box, and click **OK**.

6 Open a sketch on **Plane4**, and sketch a circle, centered on the origin.

It appears as though you are sketching on top of the first sketch. However, the first sketch is on **Plane1**, and it is not affected by sketching on **Plane4**, a parallel plane in front of it.

7 Dimension the circle to 50mm in diameter.

8 Exit the sketch.

9 Open a sketch on **Plane5**, and sketch a circle, centered on the origin. As you drag, make the diameter of the circle coincident with the vertex of the square. (Watch for the [icon] pointer.)
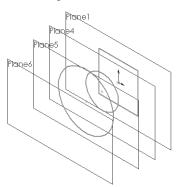
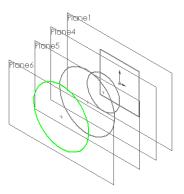**10** Exit the sketch.

Copying a Sketch

You can copy a sketch from one plane to another to create another profile.

1 Click **Isometric** to see how the sketches line up.

> **TIP:** If a sketch is on the wrong plane, you can change the plane. Right-click the sketch, select **Edit Sketch Plane**, then click the new plane for the sketch in the FeatureManager design tree.

2 Click **Sketch3** (the larger circle) in the FeatureManager design tree or the graphics area.

3 Click **Copy** on the Standard toolbar, or click **Edit, Copy.**

4 Click **Plane6** in the FeatureManager design tree or the graphics area.

5 Click **Paste** on the Standard toolbar, or click **Edit, Paste**.

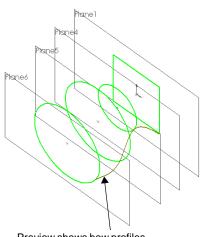When you paste a sketch on a plane, a new sketch is created automatically on that plane.

Create the Loft

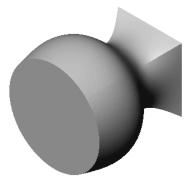Now use the **Loft** command to create a solid feature based on the profiles.

1 Click **Insert, Base, Loft**.

2 In the graphics area, select each sketch. Click near the same place on each profile (the lower-right side, for example), and select the sketches in the order you want to connect them.

A preview shows you how the profiles will be connected; the system connects the points or vertices on the profile closest to where you click.

3 Examine the preview.

- If the sketches appear to be connected in the wrong order, you can use the **Up** or **Down** buttons in the **Loft** dialog box to rearrange the order.

- If the preview indicates that the wrong points will be connected, right-click in the graphics area, select **Clear Selections**, and select the profiles again.
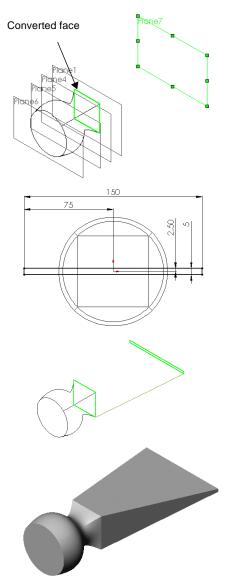
4 Click **OK** to create a solid base feature.

Preview shows how profiles will be connected

Creating a Boss Loft

For the pointed end of the chisel, you create another loft, a boss this time. One of the profiles is the square from the base feature. However, you cannot use the same sketch in two features; you need to make another sketch for use in the boss feature.

1 Click the square face of the base, open a new sketch, then click **Convert Entities** 🗗.

This way, if the square profile of the base changes, this profile will change also.

2 Exit the sketch.

3 Hold down **Ctrl**, and drag **Plane1** to create an offset plane *behind* **Plane1**.

4 Right-click the new plane, **Plane7**, and select **Edit Definition**. In the **Offset Plane** dialog box, set the **Distance** to 200mm, make sure that **Reverse Direction** is selected, and click **Finish**.

5 Open a sketch on **Plane7**. Sketch and dimension a narrow rectangle as shown.

6 Exit the sketch.

7 Click **Insert, Boss, Loft**.

8 Click near the lower-right corner of the square and the rectangular sketches. Examine the preview to verify that the correct vertices will be connected.

9 Click **OK**.

Working with Patterns

In this chapter, you learn how to create a *linear pattern* and a *circular pattern.* A linear pattern is a one- or two-dimensional array of features. A circular pattern is a circular array of features.
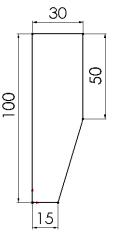
The steps include:

❑ Creating a revolved base feature

❑ Using mirroring to create a feature

❑ Creating a linear pattern

❑ Deleting and restoring an instance of the linear pattern

❑ Creating a circular pattern

❑ Using an equation to drive the circular pattern

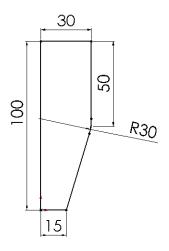## Creating the Revolved Base Feature

In this example you create a housing for a microphone. Because the housing is cylindrical, you can create the housing as a revolved feature.

1   Open a new part, and open a sketch on the default plane, **Plane1**.

2   Click **Grid** ⊞ and make sure that **Length Unit** is set to **Millimeters**, set **Decimal places** to 0, and click to clear **Snap to points**. Click **OK**.

3   Sketch and dimension the profile as shown.

4   Click the **Fillet** ▱ tool on the Sketch Tools toolbar.

    a)   Set **Radius** to 30mm.

    b)   Leave **Keep constrained corners** selected so that the corner dimensions and relations are retained to a virtual intersection point.

    c)   Select the endpoint of the 50mm vertical line that is coincident with the endpoint of the diagonal line.
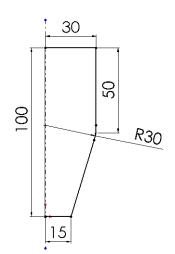
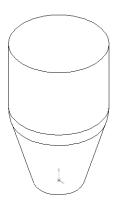   **d)**   Click **Close**.

The corner is filleted away.

5 Sketch a vertical **Centerline** ▯ through the origin.

The centerline is the axis around which the profile revolves.

6 Click **Revolved Boss/Base** 🔧 on the Features toolbar, or click **Insert, Base, Revolve**.

7 Leave the default values of **Type** as **One-Direction**, **Angle** at 360°, and **Revolve as** at **Solid Feature**. Click **OK** to create the revolved base.

8 Click **Hidden Lines Removed** 🗐.

9 Click **Save** 🖫, and save the part as **Mhousing.sldprt**.

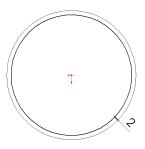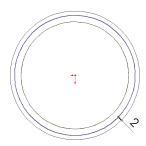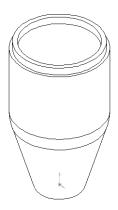# Extruding a Thin Feature

Now, create a thin-walled extrusion for the microphone capsule.

1   Select the top face and open a sketch.

2   Click **Top** 🔲 to change the view orientation.

3   Click **Offset Entities** 🔲.

   a)   Set **Offset** to 2mm.

   b)   Click **Reverse** to offset the edge to the inside.

   c)   Click **Apply**, then click **Close** to exit the **Offset Entities** dialog box.

4   Click **Extruded Boss/Base** 🔲 or **Insert, Boss, Extrude**.

   a)   Leave **Type** as **Blind**.

   b)   Specify a **Depth** of 5mm.

   c)   Set **Extrude as** to **Thin Feature**.

   d)   Click the **Thin Feature** tab.

   • Leave **Type** as **One-Direction**.

   • Set **Wall Thickness** to 3mm.

   • Click **Reverse** to extrude the wall to the inside.

   e)   Click **OK** to create the thin-walled extrusion.

5   Click **Isometric** 🔲 for a better view of the thin-walled extrusion.
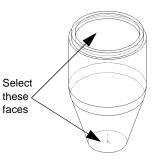
6   Save the part.

Shelling the Part

Hollow out the part by removing the top and bottom faces.

1 Click **Hidden In Gray** 🔲.

2 Click **Shell** 🔲 or **Insert, Features, Shell**.

The **Shell Feature** dialog box appears.

3 Set **Thickness** to 3mm.

4 Click the **Faces to remove** box, then click the top and bottom faces as shown.

Select these faces

TIP: To select an edge or face that is behind the near surface (a hidden edge or face), right-click and choose **Select Other** from the shortcut menu.

The **Yes/No** pointer appears. When you point and right-click **(N)**, you cycle through the edges or faces under the pointer, highlighting each of them in turn.

When the edge or face that you want is highlighted, click **(Y)**.

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5 Click **OK**.

6 To see the shelled part better, click **Shaded** 🔲 and rotate the part.

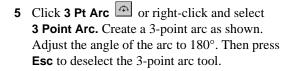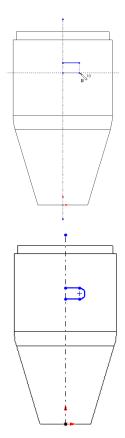## Creating an Oblong Cut

Next you create a profile of an oblong on a reference plane. Use mirroring to take advantage of symmetry and to decrease the number of relations needed to fully define the sketch.

1   Click **Hidden Lines Removed** 🔲.

2   Open a sketch on **Plane1**, and click **Normal to** 🔷.

3   Click **Centerline** 🔳, and sketch a vertical centerline through the origin.

4   Click **Line** 🔲, and sketch two horizontal lines of equal length, beginning at the centerline.

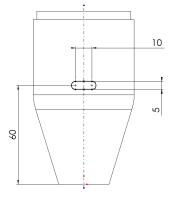   Watch for the on-curve pointer 🖋 that indicates when you are exactly on the centerline.

5   Click **3 Pt Arc** 🔲 or right-click and select **3 Point Arc.** Create a 3-point arc as shown. Adjust the angle of the arc to 180°. Then press **Esc** to deselect the 3-point arc tool.

6 Mirror the sketch entities.

   a) Hold down **Ctrl**, and select the centerline, both horizontal lines, and the 3-point arc.

   b) Click **Mirror** on the Sketch Tools toolbar, or click **Tools, Sketch Tools, Mirror**.

   The sketch is mirrored on the other side of the centerline.

7 Dimension the oblong as shown.

   Now that the sketch is fully defined, create the cut.

8 Click **Isometric** .

9 Click **Extruded Cut** or **Insert, Cut, Extrude**.

   • Select **Type** as **Through All**.

   • Click **Reverse Direction**.

   • Leave **Extrude as** at **Solid Feature**.

10 Click **OK** to create the cut.

Creating the Linear Pattern

Next create a linear pattern of the oblong cut. You use a vertical dimension to specify the direction in which to create the linear pattern.
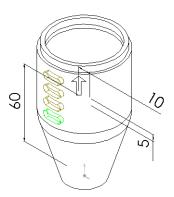
1   Double-click **Cut-Extrude1** in the FeatureManager design tree.

   The dimensions of the **Cut-Extrude1** feature appear in the graphics area.
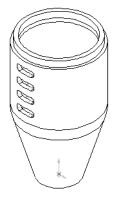
2   Click **Linear Pattern** 🔲 on the Features toolbar, or click **Insert, Pattern/Mirror, Linear Pattern**.

   • Leave at **First Direction**.

   • Click the **Direction selected** box, then click the 60mm dimension in the graphics area.

     An arrow appears in the preview indicating the direction of the pattern. If the arrow is not pointing up, click **Reverse direction**.

   • Set **Spacing** to 10mm. This value is the distance from a point on one instance of the patterned feature to the corresponding point on the next instance.

   • Set **Total instances** to 4. This value includes the original cut-extrude feature.

   • Make sure that **Cut-Extrude1** is listed in the **Items to copy** box.

3   Click **OK** to create the linear pattern.

4   Save the part.

Deleting and Restoring an Instance of a Pattern

You can delete an instance of a pattern if necessary.

1 Click **Zoom To Area** 🔍, then drag the pointer to create a rectangle around the linear pattern.

2 Click **Select** ⬚, and select a face on the top pattern instance.

3 Press the **Delete** key.

The **Pattern Deletion** dialog box appears.

4 Make sure that **Delete Pattern Instances** is selected and that the location of the pattern instance to be deleted in the **Instances Deleted** box is **(4, 1)**.

5 Click **OK** to close the dialog box.

The selected instance of the pattern is deleted.

6 Click **Zoom To Fit** 🔍 to view the entire part.

Now restore the deleted instance of the pattern.

1 Right-click **LPattern1** in the FeatureManager design tree, then select **Edit Definition**.

The **Linear Pattern** dialog box appears.

2 In the **Instances deleted** box, click the deleted instance **(4, 1)**, then press the **Delete** key.

The pattern instance is removed from the **Instances deleted** box, and restored in the preview.

3 Click **OK**.

Creating a Circular Pattern of a Linear Pattern

Now create a circular pattern of the linear pattern, using a temporary axis as the axis of revolution.
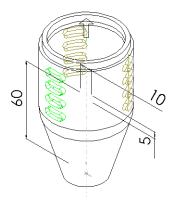
1   Click **View, Temporary Axes**.

2   Click **Circular Pattern** 🌼 on the Features toolbar, or click **Insert, Pattern/Mirror, Circular Pattern**.

- Click the **Direction selected** box, then click the temporary axis that passes through the center of the revolved feature.

  An arrow appears in the preview indicating the direction of the pattern. If the arrow is not pointing up, click **Reverse direction**.

- Set **Spacing** to 120°.

- Set **Total instances** to 3.

- Make sure that **LPattern1** is listed in the **Items to copy** box.

3   Click **OK** to create the circular pattern.

A circular pattern of the linear pattern is created around the part's axis of revolution.

4   Click **View, Temporary Axes** to turn off the display of axes, then click **Shaded** 🔲.

NOTE: If you need to use a circular pattern in a part that does not have a temporary axis in the desired place, you can create an axis, or you can use a linear edge as an axis. For more information about creating an axis, see Chapter 3, "Reference Geometry," in the *SolidWorks 99 User's Guide* and online help.

Using an Equation in the Pattern

You can use an equation to drive the circular pattern. In this example, the equation calculates the spacing angle by dividing 360° by the number of instances desired. This creates a full circle of equally spaced patterns.

1 In the FeatureManager design tree, double-click **CirPattern1.**

Two values appear on the part: 3 (total instances) and 120° (spacing angle).

2 Click **Equations** Σ on the Tools toolbar, or click **Tools, Equations**.

3 Click **Add** in the **Equations** dialog box.

4 Click the spacing angle value (120) on the part. (You may have to move the dialog boxes to uncover the dimension.)

The name of the value, **D2@CirPattern1** (the second dimension in the circular pattern), is entered the **New Equation** dialog box.

5 Using the calculator buttons in the **New Equation** box, enter **= 360 /** (or type **=360/**).

6 Click the total instances value (3). **D1@CirPattern1** is added to the equation.

The equation should look as follows:

"D2@CirPattern1" = 360 / "D1@CirPattern1"

7 Click **OK** to complete the equation, and click **OK** again to close the **Equations** dialog box.

An **Equations** folder $\boxed{\Sigma}$ is added to the FeatureManager design tree. To add, delete, or edit an equation, right-click the folder, and select the desired operation.

Now test the equation.

1 Increase the total instances of the circular pattern from three to four.

    a) Double-click the total instances value (3).

    b) Set the value in the **Modify** dialog box to 4.

2 Click $\boxed{8}$ in the **Modify** dialog box to rebuild the model, then click $\boxed{\checkmark}$ to save the current value and to close the **Modify** dialog box.

– or –

Press **Enter**, then click **Rebuild** $\boxed{8}$ on the Standard toolbar, or click **Edit, Rebuild.**

3 Save the part.

9

Creating Fillets

This chapter describes how to use different types of fillets. In this example, you create a knob by:

❑ Using *relations* in your sketches

❑ Adding *draft* angles to extruded features

❑ Adding *face blend*, *constant radius*, and *variable radius fillets*
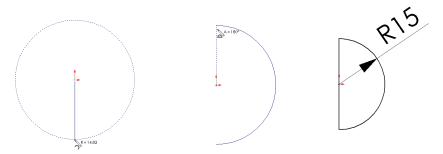
❑ Using *mirroring* to assure symmetry

Creating the Base

You can capture the symmetry of the knob in the design intent of the part. You build one half of the part, then mirror the model to create the other half. Any changes you make to the original half are reflected in the other half.

When you relate features to the origin and the planes, you need fewer dimensions and construction entities. You can more easily modify the part when you build it this way.

1 Open a new part document, and open a sketch on **Plane1**.

2 Click **Grid** ▦ . Make sure that **Length Unit** is set to **Millimeters**, set **Decimal places** to 2, and click to clear the **Snap to points** check box. Click **OK**.

3 Sketch a centerpoint arc.

   a) Click **Centerpoint Arc** 🔘 on the Sketch Tools toolbar, or click **Tools, Sketch Entity, Centerpoint Arc**.

   b) Drag downward from the origin. A circumference guideline is displayed.

   c) Drag an arc 180° counterclockwise around the origin.

TIP: The pointer changes to ◿ when a 180° arc exists.

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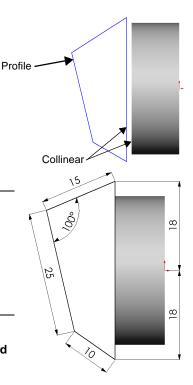
4 Connect the arc endpoints with a vertical line.

5 Dimension the arc radius to 15.00mm.

6 Select the line, hold down **Ctrl**, click the origin, click **Add Relation** ⊥ , and add a **Midpoint** relation.

7 Click **Extruded Boss/Base** 🗔 or **Insert, Base, Extrude**, then extrude the profile with **Type** of **Blind**, and **Depth** of 10.00mm.
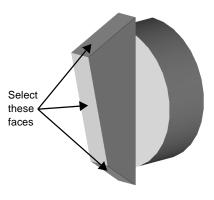
Creating the Grip

Now, create the grip of the knob.

1 Change the view orientation to **Right** .

2 Click **Plane3**, and open a sketch.

3 Sketch four lines as shown to create the profile. Do not create any inferenced perpendicular relations between lines.

4 Add a **Collinear** relation between the vertical sketch line and the model edge.

5 Dimension as shown.

> **TIP:** If the dimension font is too large for the model and the sketch entities, you can change the display scale of the dimensions. Right-click the **Annotations** folder in the FeatureManager design tree, and select **Details**. In the **Annotation Properties** dialog box, select the **Always display text at the same size** check box, and click **OK**.

6 Click **Extruded Boss/Base** or **Insert, Boss, Extrude**, then extrude the profile with **Type** of **Blind** and **Depth** of 5.00mm.

Adding Draft to the Grip
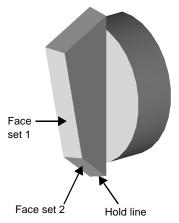
1 Change the view orientation to ***Dimetric.**

2 Click **Draft** on the Features toolbar, or click **Insert, Features, Draft**.

- Leave **Type of draft** as **Neutral Plane.**
- Set **Draft angle** to 10.00°.
- Select **Plane3** as **Neutral plane**.
- Click **Faces to draft**, and select the three faces shown.

3 Click **OK** to create the drafts and to close the dialog box.

Creating a Face Blend Fillet

Next, blend some of the faces using a face blend fillet with a hold line. This type of fillet removes the faces that share an edge with the hold line. The distance between the hold line and the selected edges determines the radius of the fillet.

1   Click **Fillet** 🔘 or **Insert, Features, Fillet/Round**.

2   Set **Fillet type** to **Face Blend**.

3   Click **Face set 1**, and select the face labeled **Face set 1**.

4   Click **Face set 2**, and select the face labeled **Face set 2**.

5   Click the **Advanced Face Fillet** tab.

6   Select the **Set fillet boundary** check box.

7   Click **Hold lines**, and select the edge labeled **Hold line**.

Face set 1

Face set 2   Hold line
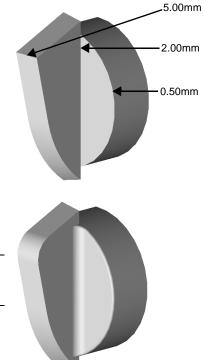
8   Click **OK**.

9   Click **Save** 💾 or **File, Save**, and save the part as **Knob.sldprt**.

Creating Constant Radius Fillets

Now, round some of the edges using a series of constant radius fillets.
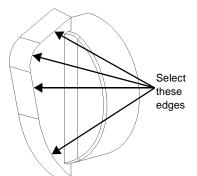
1 Click **Fillet** 🔲 or **Insert, Features, Fillet/Round**.

- Select the edge of the grip labeled **5.00mm**.

- Leave **Fillet type** as **Constant Radius**.

- Set **Radius** to 5.00mm.

2 Click **OK**.

3 Repeat Steps 1 and 2 to add fillets to the edges labeled **2.00mm** and **0.50mm**. Change the radius values to match the values of the labels.

> **TIP:** When filleted edges intersect, it is good practice to add the larger fillet first.

Creating a Variable Radius Fillet

1   Click **Fillet** 🗔 or **Insert, Features, Fillet/Round**.

2   Set **Fillet type** to **Variable Radius**.

3   Select the four edges shown here.

Select these edges

4   Set the radius values for the five vertices as shown in the illustration.

    a)   Click **Vertex1** in **Vertex List**.

        The value for **Vertex1** appears on the part.

    b)   Change the value in the **Radius** box to match the value of the label.

    c)   Click each vertex in **Vertex List**, and change the value to match the label.

R1.00

R1.50

R1.50

R1.50

R1.00

5   Click **OK** to close the **Fillet Feature** dialog box.

> **TIP:**   To verify the radius values, double-click **VarFillet1** in the FeatureManager design tree.

6   Save the part.
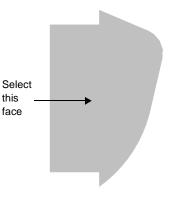
Mirror the Model

To take advantage of the part's symmetry and to finish the part, mirror the part about the planar face that is coincident with **Plane3**.

1 Change the view orientation to **Left** 🔲.

2 Click **Insert, Pattern/Mirror, Mirror All**.

3 Select the planar face shown.

4 Click **OK**.

A mirror image of the original part is joined to the part at the selected face to make a complete, symmetrical part.

Select this face →
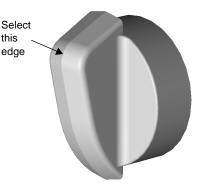
Fillet the Parting Line
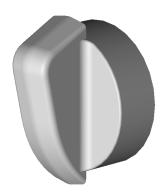
When you mirrored the drafted grip, it created a parting line along the top of the grip. Smooth the parting line by adding a constant radius fillet.

1 Change the view orientation to ***Dimetric.**

2 Click **Fillet** 🔲 or **Insert, Features, Fillet/Round.**

3 Select the edge shown, and leave **Fillet type** as **Constant Radius**.

4 Set **Radius** to 5.00mm.

Select this edge

5 Make sure **Propagate to tangent faces** is selected, then click **OK**.

The fillet extends along all of the segments of the edge.
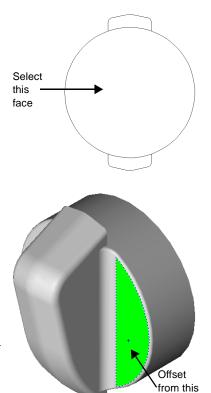
Creating a Thin-Walled Body

Now remove material from the round base of the knob to create a thin-walled body.

1 Change the view orientation to **Back** 🔲.

2 Select the back face of the knob, and open a sketch.

3 With the back face still selected, click **Offset Entities** 🔲 or **Tools, Sketch Tools, Offset Entities**.

4 Set **Offset** to 1.00mm, and select **Reverse** to offset the edge to the inside.

5 Click **Apply**, then click **Close**.

Select this face

6 Change the view orientation to **Isometric** 🔲.

7 Click **Extruded Cut** 🔲 or **Insert, Cut, Extrude**.

8 Set **Type** to **Offset From Surface**, and set **Offset** to 1.00mm.

9 Click **Selected Items**, and select the face shown.

10 Click **OK**.

> **TIP:** Using **Offset Entities** and **Offset From Surface** ensure that the wall thickness remains 1.00mm, even if you change the base diameter or base depth.

Offset from this face

11 To examine the part, click **Rotate View** 🔲 and rotate the part.

12 Save the part.

Mating Parts in an Assembly
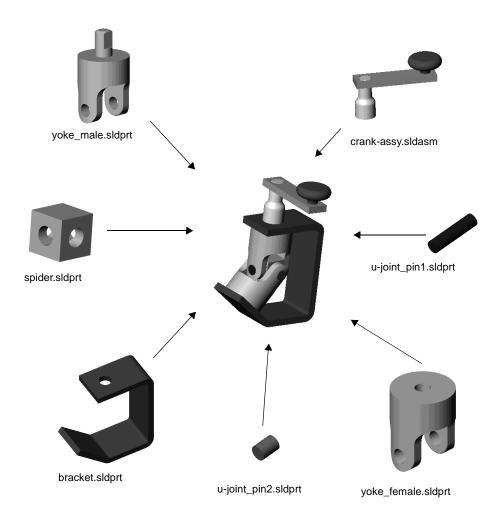
This chapter guides you through the creation of the universal joint assembly shown here, and demonstrates the following:

❑ Bringing parts into an assembly

❑ Using these *assembly mating* relations:

- Coincident
- Concentric
- Parallel
- Tangent

❑ Using *automatic mating*

❑ *Testing* mating relations

❑ *Exploding* and *collapsing* the assembly

Introduction

This assembly uses the following parts and assembly, located in the directory
*install_dir***samples\tutorial\universal_joint**.

yoke_male.sldprt

crank-assy.sldasm

spider.sldprt

u-joint_pin1.sldprt

bracket.sldprt

u-joint_pin2.sldprt

yoke_female.sldprt

Setting the Assembly Load Option

You can load an assembly with its active components *fully resolved* or *lightweight*.

- **Fully resolved**. All model information is loaded in memory.

- **Lightweight**. A subset of model information is loaded in memory. The remaining model information is loaded if the component is selected or if the component is affected by changes that you make in the current editing session.

You can improve the performance of large assemblies significantly by using lightweight components.

NOTE: You can only set the option to load an assembly with lightweight parts when no assemblies or drawings of assemblies are open.

The assembly you build in this chapter includes a sub-assembly whose parts could be loaded lightweight. However, there are no significant benefits in using lightweight parts, for these reasons:

- The sub-assembly is small, consisting of only three simple components.

- You select two of the three components as you build the assembly, thereby resolving them anyway.

1 Before you open the assembly document, click **Tools, Options, Performance**.

2 Under **Assemblies**, click to clear the **Automatically load parts lightweight** check box.

3 Click **OK**

For more information about lightweight parts, see Chapter 6, "Working with Assemblies," in the *SolidWorks 99 User's Guide* and online help.

Inserting the First Part into the Assembly

This section describes how to insert a part into the assembly.

1   Click **File, Open**, and open **bracket.sldprt** found in the directory
*install_dir***samples\\tutorial\\universal_joint**.

2   Click **File**, **New**, **Assembly**. If the assembly origin is not displayed, click **View,
Origins**.

3   Tile the windows so that you can see both the part window and the assembly window.
(Click **Window**, **Tile Vertically** or **Tile Horizontally.**)

4   Click the part name, **bracket**, at the top of the FeatureManager design
tree in the **bracket.sldprt** window. Drag **bracket** into the **Assem1**
window, and drop it on the assembly origin. As you drag, watch for the
pointer shown here. This pointer indicates an inference to the assembly
origin.

When you place a component this way, the *component origin* is located
coincident with the *assembly origin*, and the planes of the part and the
assembly are aligned. This procedure, while not required, helps you
establish an initial orientation for the assembly.

> **NOTE:**   You can create this type of inference with any component as you add
> it to the assembly. You can also create the inference to the assembly
> origin by dropping the component in the FeatureManager design tree
> of the assembly window.

5   Close the **bracket.sldprt** window, and maximize the **Assem1**
window.

Notice that the FeatureManager design tree contains the feature
(f)bracket<1>. Because this is the first component inserted into
the assembly, **bracket** is fixed **(f)**. It cannot be moved or rotated
unless you float (unfix) it. The **<1>** means that this is the *first
instance* of **bracket** in the assembly.

The assembly also contains an empty **MateGroup1** feature.
This feature is a placeholder for the mates that you add later.

6   Click **Isometric** 🔷, and click **Hidden Lines Removed** 🗗.

Bringing More Components into the Assembly

Another way to add components to the assembly is to drag them in from the Microsoft Windows Explorer.

1 Start Windows Explorer (if it is not already running).

2 Navigate to the directory *install_dir***samples\tutorial\universal_joint**.

3 Click each of the items listed below, and drag it into **Assem1**. Place them approximately as shown.

- **yoke_male.sldprt**

- **yoke_female.sldprt**

- **spider.sldprt**

4 Examine the FeatureManager design tree, and expand each item to see the features used to make the components.

Notice that each of the new components has the prefix **(-)** before its name, indicating that its location is under defined. You can move and rotate these components.

5 To collapse the entire FeatureManager design tree in one step, right-click **Assem1** in the FeatureManager design tree and select **Collapse Items**.

6 Practice moving and rotating the individual components using the following tools on the Assembly toolbar:

Click **Move Component**, click the component's name in the FeatureManager design tree or click one of the component's faces, then move the component.

Click **Rotate Component Around Centerpoint**, click the component's name in the FeatureManager design tree or click one of the component's faces, then rotate the component.

Both the **Move Component** and **Rotate Component Around Centerpoint** tools remain active so that you can move other non-fixed components in succession.

Hold down **Ctrl** and click both the component and an axis, linear edge, or sketched line. Then click **Rotate Component Around Axis**, and rotate the component.

If the axes are not currently displayed, click **View, Axes** (for user-defined axes) or **View, Temporary Axes** (for axes defined implicitly by the software.)

7 Save the assembly as **U-joint.sldasm**.

Mating the Bracket with the Male Yoke

The following pages describe how to add various types of assembly mating relations.

First, mate the bracket and the male yoke.

1   Click **Mate** 🖉 or **Insert**, **Mate**.
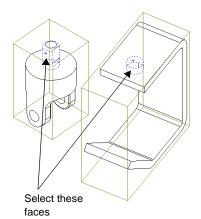
The **Assembly Mating** dialog box appears.
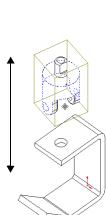
2   Click the cylindrical face of the boss on the male yoke and the cylindrical inside face of the top hole in the bracket.

> **NOTE:**   You can also select the items to mate before opening the **Assembly Mating** dialog box. Hold down **Ctrl** as you select the items.

3   Select **Concentric,** click **Preview** to check the mate, and click **Apply**.

The boss of the male yoke and the bracket hole are now concentrically mated.

Select these faces

4   To test the mate, click **Move Component** 🖉, and drag the male yoke. You should only be able to drag up and down, following the axis of the concentric mate. (The yoke may spin as it moves.)

5   Click **Mate** 🖉 or **Insert**, **Mate** again.

6   Click the pushpin 🖼 in the **Assembly Mating** dialog box, and move the dialog box to a convenient location.

The **Assembly Mating** dialog box stays open and on top of the other windows as you continue to add mates. When you return to select mode (either by clicking **Select** 🖱 or **Tools, Select**), the **Assembly Mating** dialog box closes.
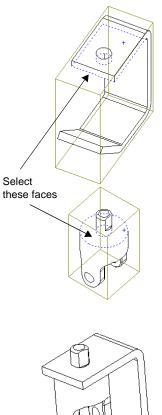
7 Click the top inside face of the bracket and the top face of the male yoke.

TIP: To select the top inside face of the bracket without rotating the bracket, right-click the top of the bracket, and click **Select Other**. Click **N** until the correct face is highlighted, then click **Y**.

Select
these faces

8 Select **Coincident** in the **Assembly Mating** dialog box, click **Preview**, and click **Apply**.

The top of the yoke is now inserted into the bracket hole.

Mating the Male Yoke with the Spider
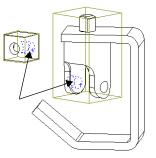
1 Select the inside faces of one pin hole on the male
yoke and one spider pin hole.

2 Click **Concentric**, click **Preview**, and click **Apply**.

The spider and the male yoke are now concentrically
mated.

3 Select the flat spider face that contains the hole you
selected in Step 1 and the inside face of the male yoke.
Use **Select Other** or rotate the assembly if necessary.
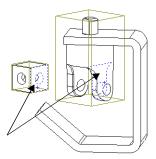
> **NOTE:** To move and rotate components while
> the **Assembly Mating** dialog box is
> open, use the **Pan** ⊕ and **Rotate
> View** ↻ tools on the View toolbar.
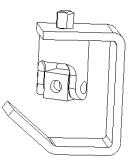
4 Click **Coincident**, then click **Preview**.

The spider should be placed inside the male yoke as
shown.

- If the mate looks correct, click **Apply**.

- If the mate looks wrong, click **Undo**, select the
correct faces, and click **Apply**.

5 Close the **Assembly Mating** dialog box.

Mating the Female Yoke and the Spider
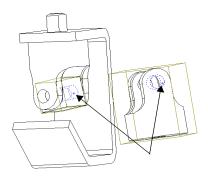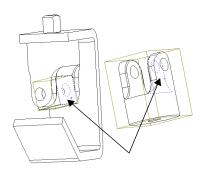
1 Using the tools on the Assembly toolbar (see page 10-5), move and rotate the female yoke to approximately the position shown here.

2 Click **Mate** 🖉 or **Insert, Mate**, then click the pushpin 📌 in the **Assembly Mating** dialog box.

3 Select the inside face of the pin hole of the female yoke and one of the visible spider pin holes.

4 Click **Concentric**, click **Preview**, and click **Apply**.

The spider and the female yoke are concentrically mated.
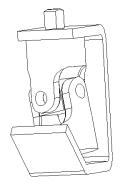
5 Select the flat spider face that contains the hole you used in Step 3, and the inside face of the female yoke.

6 Click **Coincident**, click **Preview**, and click **Apply**.

The female yoke should be positioned as shown. The rotation may be different in your assembly because it is based on the initial position of the two components before mating.
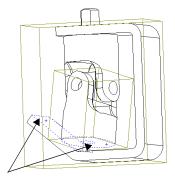
Mating the Female Yoke with the Bottom of the Bracket
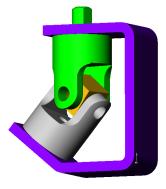
1   Select the bottom face of the female yoke and the top slanted face of the bracket.

2   Click **Parallel**, and click **Preview**.

The female yoke is aligned to the bracket.

3   If the female yoke is upside down, change the
Alignment Condition, and click **Preview** again.

- **Anti-aligned** means that the normal vectors for
  the selected faces point in *opposite* directions.

- **Aligned** means that the normal vectors for the
  selected faces point in the *same* direction.

- **Closest** means that the selected faces may be
  either aligned or anti-aligned, depending on the
  positions they occupy when selected.

4   Click **Apply**, then close the **Assembly Mating**
dialog box.

5   Save the assembly.
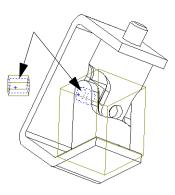
Mating the Small Pins to the Female Yoke

Another way to add components to an assembly is to use the **Insert** menu.

1 Click **Insert, Component, From File**, then navigate to
install_dir**samples\tutorial\universal_joint**.

2 Select **u-joint_pin2.sldprt**, then click **Open**.

3 Click the 👆 pointer in the graphics area where you want to place the component.

The **u-joint_pin2<1>** component is added to the assembly.

4 Click **Mate** 📎 or **Insert, Mate,** then click the pushpin 📌 in the **Assembly Mating** dialog box.

5 Select the cylindrical face of the pin and an inside face of a pin hole on the female yoke.

6 Add a **Concentric** mate.

7 Select the end face of the pin and the outside face of the female yoke.

8 Add a **Tangent** mate.

You use **Tangent** (instead of **Coincident**) for this mate because one face is flat and the other face is cylindrical.

9 Close the **Assembly Mating** dialog box.

10 Hold down **Ctrl**, then drag the **u-joint_pin2<1>** icon from the FeatureManager design tree into the graphics area.

A copy of the component is added to the assembly, **u-joint_pin2<2>**.The **<2>** notation indicates the *second instance* of this part in the assembly.

11 Repeat Steps 4 through 9 to mate the second instance of the pin to the other hole in the female yoke.

12 Save the assembly.

Using Automatic Mating to Mate the Large Pin

For some mates, you can create mating relationships automatically. You can inference the geometry of existing components as you drag and drop new components into the assembly.

In this section, you create a concentric mate automatically. For more information about automatic mating, see Chapter 6, "Working with Assemblies," in the *SolidWorks 99 User's Guide* and online help.

1   Click **File, Open**, and open **u-joint_pin1.sldprt** found in the directory
*install_dir***samples\tutorial\universal_joint**.

2   Tile the windows so that you can see the part and the assembly windows.

3   Change the view orientation of the part to **Isometric** , if necessary.

4   Change the view mode in the assembly window to **Shaded** , and change the view orientation to **Isometric** . Zoom in on the pin hole in the male yoke.

Shaded mode allows you to see the preview of the automatic mate better.

5   Select the cylindrical face of the pin, and drag the pin into the assembly. Point at an inside face of the pin hole on the male yoke in the assembly window. (The pin may disappear behind the assembly.)

Preview of pin
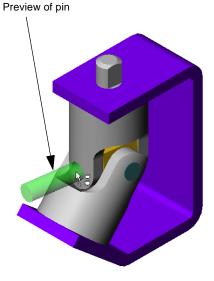
When the pointer is over the pin hole, the pointer changes to . This pointer indicates that a concentric mate will result if the pin is dropped at this location. A preview of the pin snaps into place.

If the preview indicates that you need to flip the alignment condition, press the **Tab** key to toggle the alignment (aligned/anti-aligned).
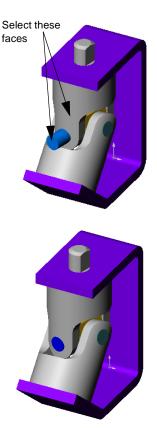
6   Drop the pin.

A concentric mate is added automatically.

7   Close the **u-joint_pin1.sldprt** window, and maximize the assembly window.
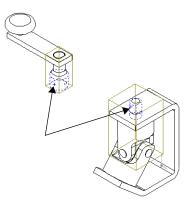
8 Click **Mate** 🖉 or **Insert, Mate**, then select the end face of the pin and the outside face of the male yoke as shown.

Select these faces
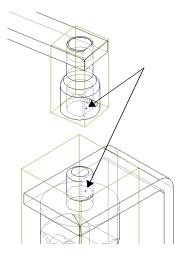
9 Add a **Tangent** mate.

10 Save the assembly.
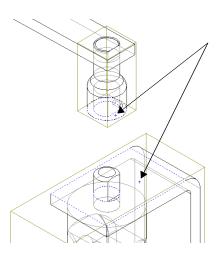
Mating the Handle to the Assembly

1 Click **Hidden Lines Removed** 🔲 .

2 Drag *install_dir***samples\tutorial\universal_joint\crank-assy.sldasm** from Windows Explorer and drop it into the assembly window.

3 Click **Mate** 🔗 or **Insert, Mate**.

4 Select the outside face of the crankshaft and the *cylindrical* face of the male yoke boss (*not* the flat face on the boss).

5 Add a **Concentric** mate.

6 Click **Move Component** 🔄, and drag the crankshaft above the male yoke boss.

7 Click **Mate** 🔗 or **Insert, Mate**, and click the pushpin 📌 in the **Assembly Mating** dialog box.

8 Click **Hidden In Gray** 🔲, then click **Zoom to Area** 🔍 and zoom in on the crankshaft and male yoke boss.

9 Select the *flat* face of the male yoke boss and the flat face on the inside of the crankshaft. Use **Select Other** to more easily select any hidden faces.

10 Add a **Parallel** mate.
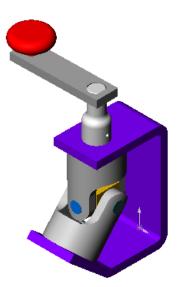
11 Select the bottom face of the crankshaft and the top face of the bracket. Add a **Coincident** mate.

12 Close the **Assembly Mating** dialog box, and save the assembly.

13 Click **Isometric** [icon], then click **Shaded** [icon].

The completed assembly should look as shown.

14 Click the ⊞ beside **MateGroup1** of the assembly (not the **crank-assy** sub-assembly) to see the mates.

> **NOTE:** If you have added or deleted mates, the names of the mates in your assembly may differ from those shown here.

Each mate is identified by the type and a number, and the names of the components involved are shown.

As you click each mate, the faces involved are highlighted.

You can rename the mates in the same way that you rename the features of a part, if desired.
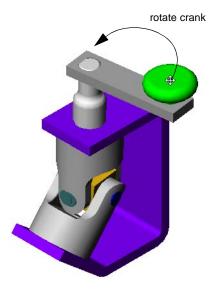
⊟ 🗍🗍 MateGroup1
  ⬡ Concentric1 (bracket<1>,yoke_male<1>)
  ⬡ Coincident1 (bracket<1>,yoke_male<1>)
  ⬡ Concentric2 (yoke_male<1>,spider<1>)
  ⬡ Coincident2 (yoke_male<1>,spider<1>)
  ⬡ Concentric3 (yoke_female<1>,spider<1>)
  ⬡ Coincident3 (yoke_female<1>,spider<1>)
  ⬡ Parallel1 (bracket<1>,yoke_female<1>)
  ⬡ Concentric4 (yoke_female<1>,u-joint_pin2<1>)
  ⬡ Tangent1 (yoke_female<1>,u-joint_pin2<1>)
  ⬡ Concentric5 (yoke_female<1>,u-joint_pin2<2>)
  ⬡ Tangent2 (yoke_female<1>,u-joint_pin2<2>)
  ⬡ Concentric6 (yoke_male<1>,u-joint_pin1<2>)
  ⬡ Tangent3 (yoke_male<1>,u-joint_pin1<2>)
  ⬡ Concentric7 (yoke_male<1>,crank-assy<1>)
  ⬡ Parallel2 (yoke_male<1>,crank-assy<1>)
  ⬡ Coincident4 (bracket<1>,crank-assy<1>)

Rotating the Crank Handle

You can turn the crank of the assembly by selecting the sub-assembly, and moving the handle.

1 Click **Move Component** 🔄.

2 Click **crank-knob<1>** in the FeatureManager design tree, or click a face on one of the components of the sub-assembly.

3 Drag the pointer in a circular motion in the graphics area.

    The crank turns and rotates the male and female yokes. All of the mating relationships are maintained.

rotate crank

Exploding the Assembly

You can create an exploded view of the assembly. An *exploded view* consists of one or more *explode steps*. In this section, you define the first step in an exploded view.
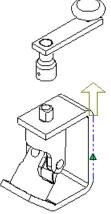
1 Click **Insert, Exploded View**.

2 In the **Assembly Exploder** dialog box, in the **Step Editing Tools** box, click **New** .

The **Assembly Exploder** dialog box expands.

3 Click a vertical edge on the bracket to set the **Direction to explode along.**

If the preview arrow is pointing down, select the **Reverse direction** check box.

4 Click the **Components to explode** box. Then click a face of a component of the crank assembly in the graphics area, or click the **crank-assy** component in the FeatureManager design tree.

5 Examine the contents of the boxes under **Step Parameters**. Make sure that the **Entire sub-assembly** option is selected. If you need to make any other changes:

- Select and delete the contents of the **Components to explode** box.

  – or –

- Click the **Components to explode** box, right-click in the graphics area, select **Clear Selections**, and select again.

6 Click **Apply** .

Notice the green arrow-shaped handle in the graphics area.
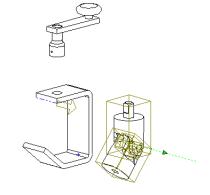
7 Drag the green handle up and down until the crank assembly is positioned at a reasonable distance from the bracket. (You can specify the position by using the **Distance** box if you prefer.)

8 Click **Apply** again to confirm the new distance value in the step.

Do *not* click **OK** yet. Leave the **Assembly Exploder** open, so you can continue adding steps to the exploded view. You click **OK** only when all the steps in the view are completed.

Adding Explode Steps

Now add explode steps for other components.

1   Click **New** to create the next explode step.

2   Click a horizontal edge on the bracket.

3   Click the male yoke, the female yoke, the spider and the pins (either in the graphics area or the FeatureManager design tree).

4   Verify the **Step Parameters**, and click **Apply**.

5   Adjust the distance as desired.

6   Click **Apply**.

7   Click **OK** to save the exploded view with its two steps.

8   Click a blank area in the graphics area to deselect all the selected items.

9   To collapse the assembly, restoring it to its previous condition, right-click anywhere in the graphics area and select **Collapse**.

Editing the Exploded View

You can edit the explode steps, or add new ones if needed. You access the exploded view from the Configuration Manager.

1 Click the Configuration tab ![tab] at the lower-left corner of the FeatureManager design tree to change to the configuration view.

2 Double-click **Default**, or click the ⊞ to expand the view.

If you are asked to confirm showing the configuration, click **OK**.

3 Double-click **ExplView1** to explode the assembly again (or right-click **ExplView1,** and select **Explode**).

4 Right-click **ExplView1**, and select **Edit Definition**.

5 Using the **Previous Step** and **Next Step** buttons ![buttons], or the **Explode steps** list, review each of the steps in the exploded view. Edit any step as desired, then click **Apply** ![apply] before editing or adding another step.

6 Click **New** ![new] to create a new explode step, then practice exploding more of the assembly. Remember to click **Apply** ![apply] each time you complete a step.

7 When you are satisfied with the entire exploded view, click **OK**.

8 To collapse the entire assembly, right-click the assembly name at the top of the FeatureManager design tree, and select **Collapse**.

9 Save the assembly.

Advanced Design Techniques
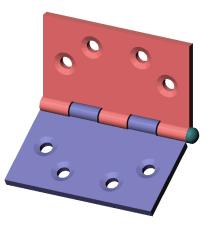
Suppose that you want to design a simple, basic hinge assembly that you can modify easily to make similar assemblies. You need an efficient way to create two matching hinge pieces and a pin for a variety of hinge assembly sizes.

Some analysis and planning can help you develop a design that is flexible, efficient, and well defined. You can then adjust the size as needed, and the hinge assembly will still satisfy the design intent.

This chapter discusses:

❑ Analyzing the assembly to determine the best approach

❑ Using a *layout sketch*

❑ Suppressing features to create *part configurations*

❑ Creating a new part *in the context* of the assembly

This chapter assumes that you know how to perform basic assembly operations, such as moving and rotating components, and adding mates. (These topics are covered in Chapters 3 and 10 of this tutorial.)

Analyzing the Assembly

Successful customers tell us that the key to using the SolidWorks software effectively is planning. By performing a careful analysis, you can design better, more flexible, functional models. Before you begin, analyze the assembly with the following considerations in mind:

❑ Consider dependencies between the components of an assembly. This will help you decide on the best approach:

  • Using *bottom-up* design, you build the parts independently, then insert them into the assembly.

  • Using *top-down* design, you may begin with some ready-made parts. Then you create other components *in the context of the assembly.* You reference the features of some components of the assembly to drive the dimensions of the other components.

❑ Identify the features that make up each individual part. Understand the dependencies between the features of each part. Look for patterns, and take advantage of symmetry whenever possible.

❑ Consider the order in which the features are created, and keep in mind the manufacturing processes that will be used to make the parts.

Dependencies in the Assembly

The hinge pieces

The two pieces of the hinge are alike: the size and thickness of the body, the barrel that receives the pin, and the placement of the screw holes. The only differences between the two pieces are the cuts and tabs on the barrel, where they fit together.

There are several ways to approach this problem:

❑ **Copy**. You could make one piece, make a *copy* of it, then modify the copy as needed for the second piece. However, if you wanted to make another assembly in a different size, you would need to edit both pieces. This is not the best approach; it leaves room for error, because the pieces are independent of each other.

❑ **Derive**. You could create a base part consisting of only the common elements, then *derive* the two pieces from it (using **Insert, Base Part** or **Insert, Mirror Part**). To make changes to the common dimensions, you edit the original, and the derived parts are updated automatically. This behavior is useful in some circumstances, but it has drawbacks for this application. You do not have access to the driving dimensions of the original part when editing a derived part, so you cannot reference those dimensions when creating the features that differ.

❑ **Configure**. The method that you use for this example is to make two different *configurations* of the same part. This is the best way to ensure that you always have matching pieces, because a single part document is used to create the two pieces. The part document contains all the possible features to be used. Then you create configurations by *suppressing* selected features, removing them from the active configuration.

The pin

You need to know the dimensions of the barrel to create a pin that is exactly the right size for the assembly. By creating the pin *in the context of the assembly,* you can accomplish this for any size hinge.

Conclusion

For this assembly, it makes sense to use a combination of design methodologies. First, design the hinge pieces, including the necessary configurations, and insert them in an assembly (bottom-up design). Then design the pin in the context of the assembly (top-down design), referencing the model geometry of the hinge pieces as necessary.

Analysis of the Individual Parts

Now that you understand the dependencies between components, take a look at the parts individually.

The common features of the hinge pieces

The base feature is a flat rectangle, with a round barrel along one edge. The diameter of the barrel is dependent on the thickness of the base. Each piece has four countersunk holes. The position of the holes is symmetric with respect to the midpoint of the long edge. As the size of the hinge changes, you want the holes to remain properly spaced along the length and width.

The different features of the hinge pieces

The cuts (and corresponding tabs) along the barrel are the features that distinguish the two pieces. One piece has three cuts, and the other has two cuts. The placement is symmetric with respect to the midpoint of the long edge. Each cut should be slightly larger than the corresponding tab, so the hinge will not bind when assembled.

The pin

The pin is dependent on the hinge pieces for its length and diameter dimensions. The domed head of the pin should match the outer diameter of the barrel.

Feature Order

Now, outline the features you will use and decide on the order to create them.
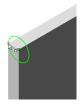
1   *Base feature* – extrude as a thin feature. Because the part has symmetric features, use a mid-plane extrusion. Then you can use the mid-plane as a plane of symmetry for mirroring other features.

2   *Barrel* – sweep a circular profile along the long model edge. Then extrude a cut, concentric with the boss.

3   *Countersunk holes* – use the **Hole Wizard** to create a complex hole profile, then use equations and mirroring to position several copies.

4   *Cuts for tabs* – create a layout sketch, referencing the dimensions of the base. Use the sketch to extrude two different cut features, one with three tabs, one with two tabs.

5   *Configurations* – define the two configurations used in the assembly by *suppressing* one cut feature in each configuration.

6   *Assembly* – insert and mate the hinge pieces (one of each configuration).

7   *Pin* – insert a new part while in the assembly. Reference the geometry of the hinge piece to sketch a profile and a path. Then use a sweep to create the base feature.

8   *Pin head* – convert the barrel profile to create a sketch, then extrude it. Finally, add a dome to the flat surface of the head.
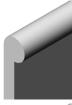
A Final Word
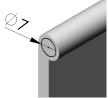
This may seem like a great deal of planning to develop a simple assembly. However, it is a worthwhile exercise if it helps you discover the best approach to building the parts *before* you start designing them. By thoroughly analyzing the issues before you begin, you can create a flexible, fully parametric model. As you change any of its parameters, the others update accordingly.

Creating the Basic Hinge Piece

1 Open a part and open a sketch on **Plane1**. Sketch a vertical line and dimension it to 60mm in length.

2 Click **Extruded Boss/Base** 🔲 or **Insert, Base, Extrude** to extrude the sketch:

    a) On the **End Condition** tab, set **Type** to **Mid Plane**, and **Depth** to 120mm.

    b) On the **Thin Feature** tab, set **Type** to **One-Direction**, **Wall Thickness** to 5mm, and select the **Reverse** check box.

    c) Click **OK**.

3 Open a sketch on the narrow vertical face. Sketch a circle at the upper edge, with its center at the front vertex.

4 Add a coincident relation between the edge of the circle and the back vertex to fully define the sketch, then close the sketch.

5 Click **Insert, Boss, Sweep**. Click the **Sweep section** box, then click the circle sketch (if it is not already listed). Click the **Sweep path** box, then click one of the long model edges. Click **OK**.

6 Cut a hole through the barrel:

    a) Open a sketch on the narrow face.

    b) Sketch and dimension a small circle as shown, and add a concentric relation to the outside edge of the barrel.

    c) Click **Extruded Cut** 🔲 or **Insert, Cut, Extrude**. Select **Type** of **Through All**, and click **OK**.

7 Save the part as **Hinge.sldprt**.

Adding the Screw Holes

In this section, you add holes for screws. To position each hole, one dimension is fixed, and the other is driven by an equation.

1 Click the large model face, then click **Hole Wizard** on the Features toolbar, or click **Insert, Features, Hole, Wizard**.

2 In the **Hole Definition** dialog box, set **Hole type** to **Countersunk,** and **End condition** to **Through All**.

3 To specify the dimensions, double-click a number in the **Value** column, and enter a new value. Set **Diameter** to 8mm, **C-Sink Angle** to 82°, and **C-Sink Diameter** to 15mm.

4 Click **Next**. Drag the point at the center of the hole to the approximate location on the face shown here. Click **Finish**.

Expand the **Hole1** feature in the FeatureManager design tree. A hole created with the hole wizard contains two sketches, one with a point to locate the center of the hole, and one for the contour of the hole.

5 Hold down **Ctrl**, then drag the **Hole1** feature from the graphics area or the FeatureManager design tree, and drop **Hole1** at another location on the same face to make a copy.

6 Right-click the under defined sketch containing the point for **Hole1**, and select **Edit Sketch**. Dimension the point to both edges of the hinge as shown. Do not close the sketch.

7 Add an equation to control the vertical dimension for the point:

  a) Click **Equations** Σ or **Tools, Equation**, then click **Add**.

  b) Double-click the base to expose its dimensions.

  Click the appropriate dimensions to create the following equation.
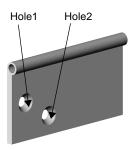
  "D2@Sketch5" = "D1@Sketch1" / 2
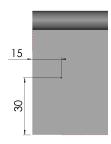
  D2@Sketch5 is the 30mm dimension in the sketch. **D1@Sketch1** is the 60mm dimension of the base.

NOTE: If you dimensioned the 30mm dimension before the 15mm dimension, then the 30mm dimension is **D1@Sketch5**.

This sets the distance between the point and the *bottom* edge to one-half the height (60mm) of the hinge.

8 Click **OK** to close the **New Equation** dialog box, then click **OK** to close the **Equations** dialog box. Exit the sketch.

9 Edit the under defined sketch containing the point for **Hole2**. Dimension the point as shown. Do not close the sketch.

10 Right-click the **Equations** folder ⬛ in the FeatureManager design tree, and select **Add Equation**.

11 Double-click the base to expose its dimensions.

12 Add the following equation:

"D1@Sketch6" = "D1@Base-Extrude-Thin" / 3

D1@Sketch6 is the 40mm dimension in the sketch.
D1@Base-Extrude-Thin is the 120mm dimension of the base.

NOTE: If you dimensioned the 15mm dimension before the 40mm dimension, then the 40mm dimension is **D2@Sketch6**.

The distance between the point and the *side* edge equals one-third of the length (120mm) of the hinge.

13 Click **OK** to close the **New Equations** dialog box. In the **Equations** dialog box, notice the values in the **Evaluates To** column.

14 Click **OK** to close the **Equations** dialog box, then exit the sketch.

15 Mirror the holes:

a) Click **Mirror Feature** ⬛ on the Features toolbar, or click **Insert, Pattern/Mirror, Mirror Feature**.

b) Click **Plane1** in the FeatureManager design tree.

Plane1 appears in the **Mirror plane** box.

c) Click each hole in either the FeatureManager design tree or in the graphics area.
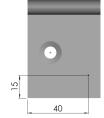
Hole1 and **Hole2** appear in the **Features to mirror** box.

d) Click **OK**.
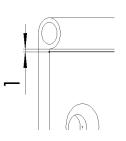
Creating a Layout Sketch for the Cuts

The layout sketch you create in this section divides the length of the hinge into five equal parts. Using equations and mirroring ensures that the five parts remain equal when you change the overall length of the hinge. You use this layout as a guide for making the cuts in the sections that follow.

1   Open a sketch on the large model face, and name it **layout for cuts**.

2   Click the lower edge of the sweep feature and click **Offset Entities** ⬚. Set **Offset** to 1mm, click **Reverse** if necessary to offset *below* the selected edge, make sure that **Select chain** is not selected, click **Apply**, and click **Close**.

3   Hold down **Ctrl** and click the edges shown, then click **Convert Entities** ⬚.

Click these edges
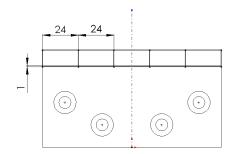
4   Click **Extend** ⬚ in the Sketch Tools toolbar, or click **Tools, Sketch Tools, Extend**, then click the converted edges. Each vertical line is extended to meet the nearest sketch entity, in this case, the offset horizontal line.

5   Sketch a horizontal line to connect the two converted edges across the top.

6   Sketch two vertical lines as shown, and dimension them. As you sketch the lines, be sure that you do not inference the geometry of the holes. Also, because the dimensions will be driven by an equation, the values of the dimensions do not matter at this time.

24   24

7   Add the equations:

    a)   Right-click the **Equations** folder ⬚, and select **Add equation**.

    b)   Add equations that set each dimension to one-fifth of the dimension of the overall length.
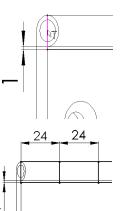
          "D2@layout for cuts" = "D1@Base-Extrude-Thin" / 5
          "D3@layout for cuts" = "D1@Base-Extrude-Thin" / 5

8   Sketch a vertical centerline across the midpoint of the part. Hold down **Ctrl**, click the two vertical lines, and click **Mirror** ⬚.

The sketch is complete and should be fully defined.

24   24

9   Exit the sketch.
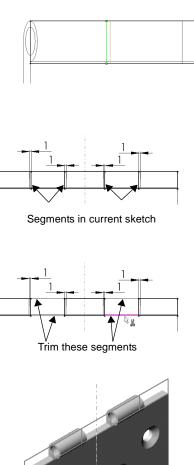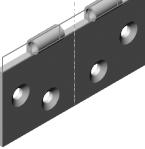
Cutting the Hinge (3Cuts)

Now you can reference the **layout for cuts** sketch to create the first set of cuts. Because you want each cut to be slightly wider than the corresponding tab on the other half of the hinge, you use offsets from the layout sketch entities.

1 Open a sketch on the large model face.

2 Click the bottom line in the layout sketch, and click **Convert Entities** 🔲. In the **Resolve Ambiguity** box, click **closed contour**, and click **OK**. This copies the entire outside contour into the current sketch.

3 Click one of the vertical lines near the edge of the part, and click **Offset Entities** 🔲. Set **Offset** to 1mm, click **Reverse** if necessary to offset the line towards the *middle* of the part, make sure that **Select chain** is not selected, and click **Apply**. Repeat for the vertical line near the opposite edge of the part.

4 Click one of the vertical lines near the center of the part, and offset the line by 1mm toward the *outside* of the part (making the center cut wider). Repeat for the remaining vertical line.

Segments in current sketch

5 Click **Close** to exit the **Offset Entities** dialog box.

6 Click **Trim** 🔲, then trim the horizontal lines as indicated, leaving three closed rectangles.

Trim these segments

7 Click **Extruded Cut** 🔲 or **Insert, Cut, Extrude**. Click **Both Directions**, and select **Through All** as **Type** for *both* **Direction 1** and **Direction 2**.

8 Click **OK**.

9 Rename the cut feature **3Cuts**.

10 Save the part.
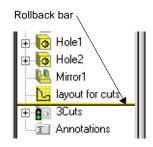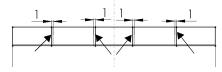
Cutting the Hinge (2Cuts)
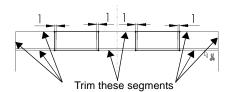
Now you use the same methods to create the cuts for the other half of the hinge.

1 Roll back the design to the **3Cuts** feature by dragging the rollback bar to just below the **layout for cuts** sketch.

2 Repeat Steps 1 and 2 from the previous section.

3 Click one of the vertical lines near the edge of the part, and click **Offset Entities**. Set the **Offset** to 1mm, offset it towards the *outside* of the part, make sure that **Select chain** is not selected, and click **Apply**. Repeat for the vertical line near the opposite edge of the part.

Segments in current sketch

4 Click one of the vertical lines near the center of the part, and offset it by 1mm toward the *middle* of the part. Repeat for the remaining vertical line.

5 Click **Close** to exit the **Offset Entities** dialog box.

6 Click **Trim**. Trim the three segments at each end and the two segments in the middle, leaving two closed rectangles.

Trim these segments

7 Extrude the cut as described in the previous section.

8 Rename this cut feature **2Cuts**.

9 Right-click the **layout for cuts** sketch, and select **Hide**.

Creating the Part Configurations

Roll the design forward by dragging the rollback bar all the way to the bottom of the FeatureManager design tree.

The part now has the entire barrel removed by the two cut features. This is the default configuration, which includes all the features. In this section, you make two more configurations of the part by suppressing selected features.

The OuterCuts configuration

1   Click the Configuration tab ⬛ at the bottom of the window to change to the Configuration Manager view.

2   Right-click the part name at the top of the FeatureManager design tree, and select **Add Configuration**.

3   Enter a **Configuration Name**, such as **OuterCuts**, in the box. Enter **Comments** if desired, and click **OK**.

4   Click the FeatureManager tab at the bottom of the window to switch back to the FeatureManager view. Notice the configuration name beside the part name at the top of the tree: **hinge (OuterCuts)**.

5   Click the **2Cuts** feature, then click **Suppress** ⬛ on the Features toolbar, or click **Edit, Suppress**.

The **2Cuts** feature is grayed out in the FeatureManager design tree, and is inactive in the current configuration.

The InnerCuts configuration

1   Repeat Steps 1 and 2 from the previous section.

2   Enter a **Configuration Name**, such as **InnerCuts**, in the box, then click **OK**.

3   Switch back to the FeatureManager view. Notice the configuration name: **hinge (InnerCuts)**.

4   Click the **3Cuts** feature, then click **Suppress** ⬛. (Now both cuts are suppressed.)

5   Click the **2Cuts** feature, then click **Unsuppress** ⬛ on the Features toolbar, or click **Edit, Unsuppress**.

The **3Cuts** feature is grayed out in the FeatureManager design tree, and the **2Cuts** feature is active in the current configuration.

6   Save the part.
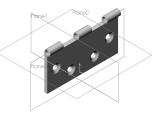
Inserting and Mating the Parts in an Assembly

Now you can begin creating the assembly.

1 Open a new assembly document.

2 Tile the windows, and drag the hinge from the top of the FeatureManager design tree of the open part window into the assembly window. Inference the assembly origin as you place the component to align the planes of the assembly and the component.

3 Maximize the assembly window.

4 Right-click the component, and select **Component Properties**. Under **Referenced configuration**, notice that **Use named configuration** and **InnerCuts** are selected by default. **InnerCuts** is the active configuration name of the part added in Step 1. Click **OK** to close the dialog box.

5 Hold down **Ctrl**, then drag the **hinge** from either the graphics area or the FeatureManager design tree, and drop it beside the first one to create another instance.
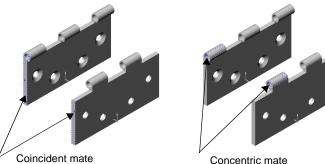
Use **Move Component** 🔄 and **Rotate Component Around Axis** 🔀 to turn the second **hinge** so that it faces the first one.

6 To change the named configuration, edit the component properties of the second **hinge**. Click **Use named configuration**, select **OuterCuts** from the list, and click **OK**.

7 Create a **Coincident** mate between the narrow front faces of the components. Create a **Concentric** mate between the inside faces of the barrels.

Coincident mate

Concentric mate

You should be able to open and close the hinge assembly using **Move Component** 📄.

8 Save the assembly as **Hinge.sldasm**.

Creating a New Part in the Assembly
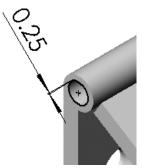
Now you add the pin. The pin references the inner diameter of the barrel and the overall length of the hinge pieces.

1 Click **Insert, Component, New**. Enter a name for the new component, such as **Pin.sldprt**, and click **Save**.

2 Click the narrow model face on the front of the assembly. The new part will be positioned on this face, with its location fully defined by an **InPlace** mating relation.

A sketch is opened automatically on the selected face. Notice that the **Edit Part** 🔴 tool in the Assembly toolbar is selected, and that the pin component is displayed in pink in the FeatureManager design tree.

3 Click the inner circular edge of the barrel, then offset it to the inside by 0.25mm.

4 Exit the sketch.
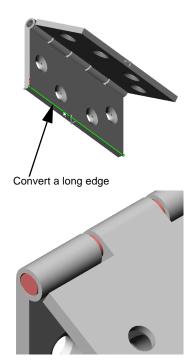
5   In the FeatureManager design tree, expand the **pin** component, click **Plane3**, and open a sketch. Click one of the long edges of the model, then click **Convert Entities** 🔲 .

6   Exit the sketch.

Convert a long edge

7   Click **Insert, Base, Sweep**. Use the two sketches as the section and path, and click **OK** to create the base feature of the pin.
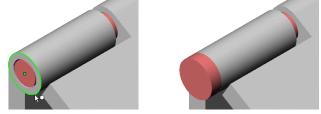
Notice that the part you are editing is pink, and the status bar in the lower-right corner indicates that you are still editing the part.
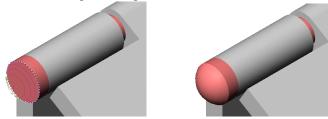
Adding a Head to the Pin

Now reference the barrel of the hinge to create the head of the pin.

1 Open a sketch on the flat end of the pin, and sketch a circle anywhere.

2 Select the circle and the outer circular edge of the barrel, and add a **Coradial** relation.

3 Click **Extruded Boss/Base** . Set **Type** to **Blind**, set **Depth** to 3mm, and click **OK**.

4 To add a dome to the head of the pin, click **Dome** on the Features toolbar, or click **Insert, Features, Dome**.

5 Click the flat face of the pin, set **Height** to 3mm. Observe the preview of the dome. Click **OK**. This completes the pin.
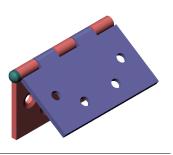
6 Right-click in the graphics area, and select **Edit Assembly: Hinge**. Alternatively, you can click **Edit Part** on the Assembly toolbar to return to editing the assembly.

7 Save the assembly.

Changing the Color of a Component

For easier viewing, you can change the color of assembly components.

1   Click one of the assembly components in either the
FeatureManager design tree or in the graphics area,
then click **Edit Color** 🔳.

2   Choose a color from the palette, then click **OK**.
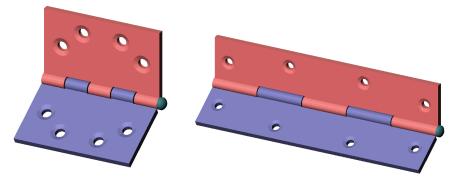
Editing the Hinge Components

Now you can make this same hinge assembly in a different size.

1   In the FeatureManager design tree, expand the hinge component that uses the
InnerCuts configuration. Double-click the **Base-Extrude-Thin** feature to display its
dimensions.

2   Double-click any of the dimensions. The **Modify** dialog box appears.

3   Change the dimension value, and make sure that **All Configurations** is selected.

4   Click ✅ to close the **Modify** dialog box.

If desired, repeat Steps 2 through 4 to change another value.

5   Click **Rebuild** 🚦 or **Edit, Rebuild**. All of the components in the assembly update
automatically. (If you see a message indicating that the pin has rebuild errors, click
Rebuild 🚦 again.)
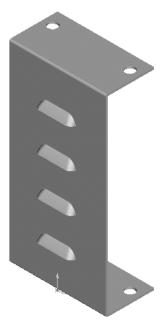
Creating a Sheet Metal Part

In this chapter, you create the sheet metal part shown here. This chapter demonstrates:

❑ Extruding a *thin feature*

❑ Inserting *bends*

❑ *Rolling back* a design

❑ Using the *Feature Palette™* window

❑ Applying a *forming tool*

❑ Creating, positioning, and patterning a *form feature*

For more information about SolidWorks sheet metal functions, see Chapter 12 of the *SolidWorks 99 User's Guide* and online help.
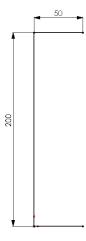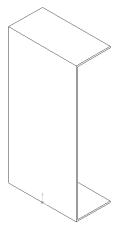
Extruding a Thin Feature

When developing a sheet metal part, it is generally a good idea to design the part in the *bent-up* state. This allows you to capture the design intent and the dimensions of the finished part.

Sheet metal parts must have a uniform thickness. One way to achieve this is to extrude a *thin feature* base from an open profile sketch.

1 Open a new part document, open a sketch on **Plane3**, and click **Normal To** ![icon].

2 Click **Grid** ![icon] on the Sketch toolbar, click to clear the **Display grid** and **Snap to points** check boxes, and click **OK**.

3 Starting at the origin, sketch a vertical line upward, and dimension the line to 200mm.

4 Sketch two horizontal lines as shown. Dimension the upper horizontal line to 50mm.

5 Click **Add Relation** ![icon] or **Tools, Relations, Add**, and add an **Equal** relation between the two horizontal lines.

6 Click **Extruded Boss/Base** ![icon] or **Insert, Base, Extrude.**

The **Extrude Thin Feature** dialog box appears.

7 On the **End Condition** tab:

- Set **Type** to **Mid Plane**.
- Set **Depth** to 100mm.

8 On the **Thin Feature** tab:

- Set **Type** to **One-Direction**.
- Set **Wall Thickness** to 2mm (the thickness of the part).
- Select **Reverse** to extrude the wall thickness inside, if necessary.

9 Click **OK**.

Inserting Sheet Metal Bends

Now you convert the thin feature part to a sheet metal part. To create the bends, you must specify the following:

❑ Fixed face – the face that remains fixed when the software unfolds (flattens) the sheet metal part.

❑ Default bend radius – the default inside bend radius used when creating a bend or adding a wall.

❑ Bend allowance – use one of the following methods:

- **Bend table**. A material-specific table that you create, containing bend allowances derived from calculations based on thickness and bend radius.

- **K-factor**. A ratio that represents the location of the neutral sheet to the thickness of the sheet metal part.

- **Bend allowance** value. An explicit value that you enter based on your experience and shop practices.

1   Click **Insert Bends** 🔧 on the Features toolbar, or click **Insert, Features, Bends**.

   The **Flatten-Bends** dialog box appears.

2   Select the front face of the thin feature base to be the fixed face.

3   Set the **Default bend radius** to 2mm.

4   Under **Bend allowance**, make sure that **Use k-factor** is selected. For this example, use the default value of 0.5.

5   Make sure that **Use auto relief** is selected. This allows the software to add relief cuts wherever necessary to make the bends.

   For this example, leave the relief type as **Rectangular** and leave **Relief ratio** at the default value of 0.5. The relief ratio is the distance by which the relief cut extends past a bend region.

6   Click **OK**.

7   Save the part as **Cover.sldprt**.
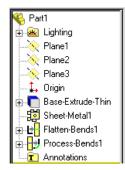
Rolling Back the Design

Examine the FeatureManager design tree. Three new features exist that represent the steps in the process of creating a sheet metal part.

❑ **Sheet-Metal1**. The **Sheet-Metal** feature marks the beginning of the process. It contains the default bend parameters.

❑ **Flatten-Bends1**. The **Flatten-Bends** feature adds the necessary bends with the bend allowance and flattens the part into a flat sheet with bend lines at the appropriate places.

❑ **Process-Bends1**. The **Process-Bends** feature folds (processes) the flattened part, returning it to its bent-up state.
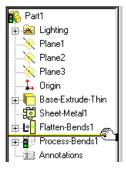
Now flatten the sheet metal part to insert holes in the flanges. You could have inserted the holes before inserting the bends. However, in this example, you insert the holes in the same order as the manufacturing process: the flat shape of the part is cut, the holes are punched, then the part is folded.
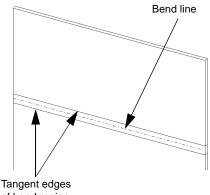
To flatten the bent-up part, you roll back to the flattened state, and insert the new feature just before the **Process-Bends** feature. Inserting the new feature before the **Process-Bends** feature ensures that the new feature is visible when the part is flat.

1 Click **Hidden Lines Removed** 🗔.

2 Roll back the design to the flattened state using one of the following methods:

- Click **Flattened** 🔲 on the Features toolbar.

- Click **Process-Bends1** in the FeatureManager design tree, then click **Edit, Rollback**.

- Click the rollback bar at the bottom of the FeatureManager design tree, then drag the bar up until it is above **Process-Bends1**. The pointer changes to a hand, and the bar changes from yellow to blue when selected.

Whichever method you use, the part is flattened, revealing the tangent edges of the bend regions. The overall developed length of the flat sheet is calculated, compensating for the bend radius and bend allowance.

3 To see the actual bend lines, right-click the **Sharp-Sketch** feature under **Flatten-Bends**, and select **Show**.

4 To hide the bend lines, right-click the **Sharp-Sketch** feature again, and select **Hide**.

Bend line

Tangent edges of bend region

Inserting the Holes

Now that the part is flat, insert the holes.
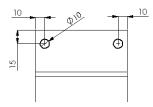
1 Open a sketch on the front face, or on either of the flange faces.

2 Click **Centerline** [ ], and sketch a horizontal centerline across the midpoints ▷ of the front face as shown.

3 With the centerline still selected, click **Mirror** [ ] or click **Tools, Sketch Tools, Mirror**.

4 Sketch two circles on the upper flattened flange.

The two circles are mirrored on the bottom flattened flange.

5 Dimension the upper-left circle to a diameter of 10mm.

6 Click **Add Relation** [ ] or **Tools, Relations, Add**.

   a) Add an **Equal** relation between the two upper circles.

   b) Add a **Horizontal** relation between the centerpoints of the two upper circles.

   c) Close the **Add Geometric Relations** dialog box.

7 Finish dimensioning the upper circles as shown.

All four circles are now fully defined.
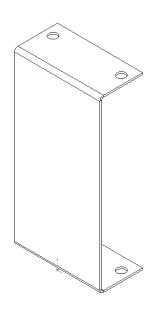
8 Click **Extruded Cut** 🔲 or **Insert, Cut, Extrude**, set **Type** to **Through All**, and click **OK**.

9 To restore the part to the folded state, click **Flattened** 🔲 or drag the rollback bar to the bottom of the FeatureManager design tree.

Examine the FeatureManager design tree. Notice that the **Cut-Extrude** feature is between the **Flatten-Bends** and **Process-Bends** features.

10 Save the part.

Using Forming Tools and the Feature Palette Window

Sheet metal forming tools are special SolidWorks parts that act as dies that bend, stretch, or otherwise form sheet metal. You apply forming tools to sheet metal parts through the Feature Palette window to create louvers, lances, ribs, and so on.

The SolidWorks software includes some sample forming tools to get you started. You use one of these forming tools in this example.

For more information about forming tools and the Feature Palette, see Chapter 11 of the *SolidWorks 99 User's Guide* and online help.

Applying the Forming Tool

1 Click **Tools, Feature Palette** to display the Feature Palette window.

By default, the Feature Palette window opens at the top level folder, or **Palette Home**. The Feature Palette window stays on top of the SolidWorks window while you work.

2 Double-click the forming tools folder 🖼 to open it, then double-click the folder labeled **Louvers**.

Palette items are displayed as thumbnail graphics, making it easy to locate, select, and insert them into SolidWorks part and assembly documents.

3 To apply the louver to the sheet metal part, drag the louver from the Feature Palette window to the front face of the sheet metal part. Do *not* drop the forming tool yet.

By default, forming tools travel downward through the selected face.

Preview indicating downward

4 To switch the direction of travel upward, press the **Tab** key.
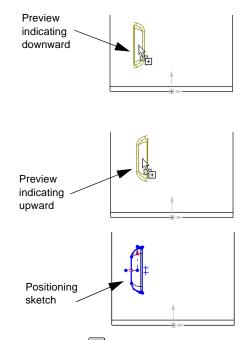
The preview updates automatically.

Preview indicating upward

5 Drop the forming tool.

The **Position form feature** dialog box is displayed.
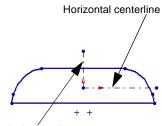
Leaving the **Position form feature** dialog box open, locate the louver on the face using the positioning sketch.

Positioning sketch

6 To rotate the positioning sketch 90°, click **Modify Sketch** 🖼 on the Sketch toolbar, or click **Tools, Sketch Tools, Modify**.

7 Type 90 in the **Rotate** box in the **Modify Sketch** dialog box, and press **Enter**.

8 Click **Close**.
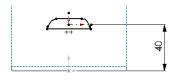
9 Click **Dimension** ✐, click **Plane2** in the
FeatureManager design tree, and click the
horizontal centerline of the positioning sketch.
Set the dimension value to 40mm.

Horizontal centerline

Vertical centerline
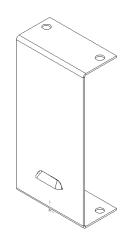
10 To center the louver on the face and to fully
define the positioning sketch, add a **Collinear**
geometric relation between **Plane3** and the
vertical centerline of the positioning sketch.

40

11 Click **Finish** to exit the **Position form feature**
dialog box.

12 Click the ⊠ button in the Feature Palette window
to close it.

Examine the FeatureManager design tree. Notice
that the form feature, **louver1**, appears after the
Process-Bends1 feature.
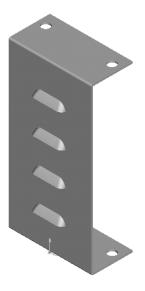
Patterning the Form Feature

Now create a linear pattern of the louver.

1 Click **Linear Pattern** ▦ or **Insert, Pattern/Mirror, Linear Pattern**.

2 Click the **Direction selected** box, then click a vertical edge of the front face.

An arrow appears in the preview indicating the direction of the pattern.

3 Select **Reverse direction** to point the arrow upward, if necessary.

4 Set **Spacing** to 40 and **Total instances** to 4.

5 Make sure that **louver1** is listed in the **Items to copy** box.

6 Select **Geometry pattern**.

The **Geometry pattern** option speeds up the creation and rebuilding of the pattern. Individual instances of the feature are copied, but not solved.

7 Click **OK**.

8 Save the part.

13
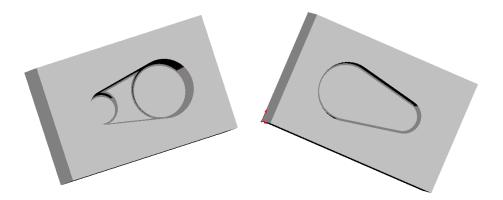
Creating a Mold

In this chapter, you create a design part, then you develop a mold from which the part can be formed. This chapter discusses the following topics:

❑ *Linking* dimension values

❑ Creating an *interim assembly* from a design part and a mold base part

❑ *Editing in context* by inserting a *cavity*

❑ *Deriving* component parts

❑ Understanding *external references*

Creating the Design Part
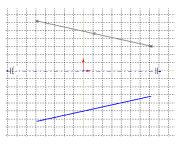
The first step is to create the part for which you want to make a mold. You create it as a solid model, just as you do any other part.
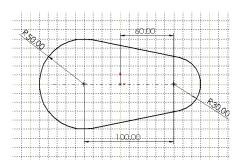
1   Open a new part document and open a sketch.

2   Sketch a horizontal centerline through the origin.

3   Click **Mirror** ⬛ or **Tools**, **Sketch Tools**, **Mirror**.

4   Sketch a sloping line on one side of the centerline as shown.

5   Click **Mirror** again to turn mirroring off.

6   Click **Tangent Arc** ⬛ or **Tools, Sketch Entity, Tangent Arc.**

7   Sketch and dimension the two arcs as shown. To dimension the distance between the arcs, select anywhere on the arcs.

   For more information about dimensioning arcs, see Chapter 2 in the *SolidWorks 99 User's Guide* and online help.

8   Click **Extruded Boss/Base** ⬛ or **Insert**, **Base**, **Extrude**.

9   In the **Extrude Feature** dialog box:

   • Set **Type** to **Mid Plane** and **Depth** to 60mm.

   • Select the **Draft While Extruding** check box, and set **Angle** to 10°.

   • Click to clear the **Draft Outward** check box, if necessary.

10 Click **OK**.
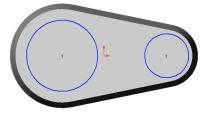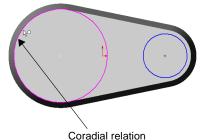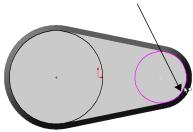
Adding Bosses
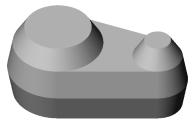
1 Open a new sketch on the front face of the part, and click **Normal To** ⬙.

2 Sketch two circles approximately as shown.

3 Add a coradial relation to align the center points of the large circle and the large arc, and to make them the same size:

   a) Click **Add Relation** ⊥ or **Tools**, **Relations**, **Add.**

   b) Select the circle and the inside edge of the larger arc (the drafted edge).

   c) Select **Coradial**.

   d) Click **Apply**.

4 Add a coradial relation between the smaller circle and arc, then close the **Add Geometric Relations** dialog box.

Coradial relation

5 Click **Extruded Boss/Base** ⬚, and extrude the bosses with the following settings:

   • **Type** of **Blind**

   • **Depth** of 20mm

   • **Draft While Extruding** check box selected

   • **Angle** of 30°

   • **Draft Outward** check box not selected

6 Click **OK**.
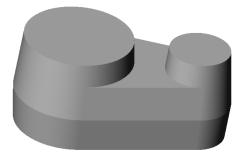
Linking Dimension Values
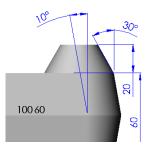
You can make the draft angles of the boss and the base equal by linking the dimension values. Then, if you change the value of either draft angle, the other draft angle updates accordingly.
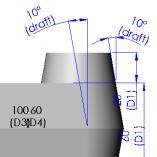
1 In the FeatureManager design tree, right-click the **Annotations** folder 🇹, and select **Show Feature Dimensions**.

2 Right-click the dimension of the draft angle of the base (10°), and select **Link Values**.

3 Type **draft** in the **Name** box, then click **OK**.

4 Right-click the dimension of the draft angle of the boss (30°), and select **Link Values**.

5 Click the arrow beside the **Name** box, select **draft** from the list, and click **OK**.

Each time you create a new **Name** variable, it is added to this list.

6 Click **Tools, Options**. On the **General** tab, under **Model**, select **Show dimension names,** then click **OK**. Notice that the draft angles have the same name.

7 Click **Rebuild** 🔴 or **Edit, Rebuild**. The part rebuilds with the boss extrusion at the same draft angle as the base.

8 Double-click the draft angle of either the base or boss, and change it to 5°.

9 Click **Rebuild** 🔴. The draft angle changes on both the base and the boss.

10 To turn off the visibility of the dimensions, right-click the **Annotations** folder 🇹, and deselect **Show Feature Dimensions**.

11 Save this part as **Widget.sldprt.**
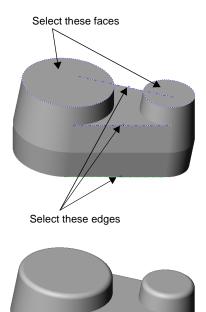
Rounding the Edges

1 Click **Fillet** 🔲 or **Insert, Features, Fillet/Round**.

2 Select the two faces and three edges shown.

Select these faces

Select these edges
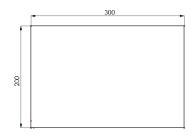
3 Set the **Radius** to 5mm.

4 Click **OK**.

5 Save the part.

Creating the Mold Base

The next step is to create the mold base part, a solid block large enough to accommodate the design part (the part to be molded).

1 Open a new part document and open a sketch. Sketch a rectangle starting at the origin and dimension it to 300mm by 200mm.

2 Click **Extruded Boss/Base** 🔲 or **Insert**, **Base**, **Extrude**. Extrude the rectangle with **Type** as **Blind** and **Depth** of 200mm.

300

200

3 Save the part as **Box.sldprt**.
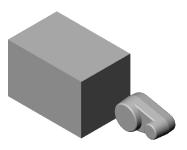
Creating an Interim Assembly

This section describes how to create an interim assembly, bringing together the design part and the mold base.

1 Click **File, New, Assembly**. If the origin is not displayed, click **View, Origins**.

2 Tile the windows. (Click **Window, Tile Horizontally** or **Tile Vertically**.)

There should be three windows open: **Widget.sldprt**, **Box.sldprt**, and **Assem1**. (Close any other windows.)

3 In the **Box.sldprt** window, click on the part name **Box** in the FeatureManager design tree, drag it into the **Assem1** window, and drop it on the origin in the FeatureManager design tree. Watch for the pointer. The planes of the box are aligned to the planes of the assembly, and the component is fixed in place.

4 Drag the widget from the graphics area of the **Widget.sldprt** window, and drop it in the assembly window beside the box in the graphics area.

5 Make the assembly window full size, and change to isometric view orientation.

6 In the FeatureManager design tree, click the ⊞ beside each component to expand the view of the features.

Centering the Design Part in the Mold Base

Now you need to position the design part to center it within the mold base. You can place the widget roughly where you want it by dragging, then more precisely by using *distance mates* between the planes of the components.
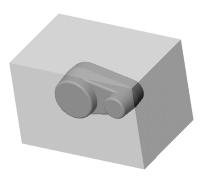
To see the widget inside the box, you could use **Hidden In Gray** or **Wireframe** display mode. Or, you can make the box transparent to see the widget inside, even in **Shaded** mode.

1 Right-click the **Box** component in the FeatureManager design tree, and select **Component Properties**. Click the **Color** button, then click **Advanced**.

2 In the **Material Properties** dialog box, drag the slider for **Transparency** to the right, a little less than halfway. Click **OK** to close each of the dialog boxes.

3 Click **Move Component** , and click the
widget component in the graphics area. Drag
the widget into the box. Notice how you can
see through the box. Change the view
orientation, and continue to move the widget
until it is roughly in the center of the box.

4 Click **Mate** or **Insert**, **Mate**.

The **Assembly Mating** dialog box appears.

5 In the FeatureManager design tree, click
Plane1 of the **Box** and **Plane1** of the **Widget**.
Click **Distance**, specify 100mm, and click
Preview.

6 Click **Rotate View** , and rotate the assembly to check the position of the widget. If
necessary, click to clear the **Flip Dimension To Other Side** check box, and click
Preview again.

7 Click the pushpin in the **Assembly Mating** dialog box now to keep it in place for
the next few steps.

8 Click **Apply**.

9 Add another distance mate, this time between **Plane2** of the **Box** and **Plane2** of the
Widget. Specify a distance of 100mm, click **Preview**, and click to clear the **Flip
Dimension to Other Side** check box if necessary.

10 Repeat for **Plane3** of the components, with a distance of 150mm.

The widget should be centered in the box.

11 Close the **Assembly Mating** dialog box.

12 Save the assembly as **Mold.sldasm.**

Creating the Cavity

In this section, you edit the mold base component **Box** in the context of the assembly. You
change the box from a solid block to a block with a *cavity* in the middle, shaped like the
design component **Widget**.

1 Click **Hidden in Gray** .

2 Click the **Box** component in the FeatureManager design tree or the graphics area, and
click **Edit Part** on the Assembly toolbar.

The **Box** component changes to pink in the graphics area and in the FeatureManager
design tree. The status bar in the lower-right corner reads "Editing Part."

> **NOTE:** It is important to be aware that you are editing the *part*, not the *assembly*,
> because the changes you are about to make will be reflected in the original
> part document, **Box.sldprt**. See Chapter 7 of the *SolidWorks 99 User's
> Guide* for more information.

3   Click **Insert, Features, Cavity**.

The **Cavity** dialog box appears.

4   Select **Widget** in the FeatureManager design tree.

Its name appears in the **Design Component** box.
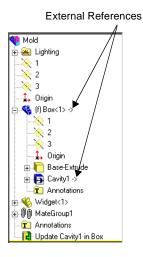
5   Set **Scaling Type** to **About Component Centroids** and **Scaling Factor in %** to 2.

These settings control how the cavity is enlarged to compensate for material shrinkage.

6   Click **OK** to create a cavity in the shape of the **Widget** part.

7   Return to assembly editing mode either by clicking **Edit Part** 🔲 again, or by selecting **Edit Assembly: Mold** from the shortcut menu.

8   Save the assembly.

Listing External References

Examine the FeatureManager design tree. The **(f)Box<1> ->** component contains a **Cavity1 ->** feature. The **->** arrow indicates an *external reference*. This occurs when you reference one part (or feature) to create a feature in another part. The new feature is dependent on the referenced feature of the other part.

A cavity has an external reference to the design part on which it is based. Therefore, if you modify **Widget**, the **Cavity1** feature of **Box** updates to reflect that change. Notice the **Update Cavity1 in Box** feature at the bottom of the design tree.

To list the external references, right-click the part or feature with the arrow, and select **List External Refs**.

External References

NOTE: External references update automatically only if all of the documents involved are open when a change is made. Otherwise, the references are considered to be out-of-context. To update out-of-context references, you must open and rebuild the document where the reference was created (in this example, the mold assembly). See Chapter 7 of the *SolidWorks 99 User's Guide* for more information.

Cutting the Mold
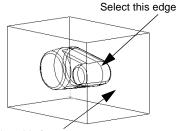
The last step is to cut the box in half to make the pieces of the mold. You *derive* the parts of the mold from the edited **Box** component.

1 Select the **Box** component, either in the model or the FeatureManager design tree, and click **File**, **Derive Component Part**.

A part window appears for the derived part. A derived part always has another part as its first feature. The first feature has an arrow **->** after the name, because it has an external reference to the part from which you derived it. You can list the external references as described in the previous section.

2 Click **Isometric** , then click either **Hidden in Gray** or **Wireframe** to see the cavity inside the box.

3 Select the narrow face of the box closest to you, and open a new sketch.

4 Select the edge of the cavity closest to the end of the box.
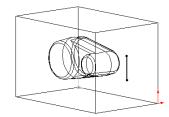
This edge is on the plane where you want to separate the mold.

Select this edge
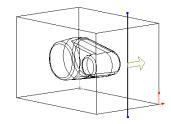
Select this face

5 Click **Convert Entities** or **Tools**, **Sketch Tools**, **Convert Entities** to project the edge onto the sketch plane.

6 Click the line and drag each endpoint so that the line is wider than the box.

7 Click **Extruded Cut** or **Insert**, **Cut**, **Extrude**. In the **Extrude Cut Feature** dialog box:

- Set **Type** to **Through All**.

- Leave the **Flip Side to Cut** check box not selected.
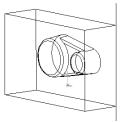
Notice the direction of the arrow in the graphics area. It points to the side where the material will be *removed*.
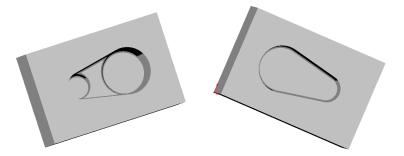
Click **OK.**

8 Click **Shaded** , and rotate the part to see the cavity.

9 Save this half of the mold as **Top_mold.sldprt**.

10 To create the other half of the mold, return to the **Mold** assembly window and repeat Steps 1 through 7.

Reverse the direction of the cut by selecting the **Flip Side to Cut** check box in the **Extrude Cut Feature** dialog box.

11 Save this half of the mold as **Bottom_mold.sldprt**.
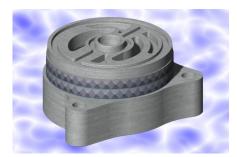
Learning to Use PhotoWorks

This chapter will guide you step-by-step through your first rendering session with PhotoWorks® software for SolidWorks 99. You are going to use the PhotoWorks software to create photo-realistic images of a SolidWorks model.

Start with a part like this...

...then use the PhotoWorks software to add rendering effects such as materials, lights, shadows, and backgrounds, to create images like these...

PhotoWorks Fundamentals

Before you begin, there are a few things you need to know about the PhotoWorks software.

❑ **PhotoWorks software creates realistic images directly from SolidWorks models.** The PhotoWorks software interacts with the 3D geometry that you create with SolidWorks software. All changes to SolidWorks models are accurately represented in PhotoWorks images.

❑ **PhotoWorks software is for use with 3D SolidWorks parts and assemblies.** It cannot be used with SolidWorks drawings.

❑ **PhotoWorks software is fully integrated with SolidWorks.** The PhotoWorks software is supplied as a SolidWorks dynamic link library (**.dll**) add-in. You access all the controls for the PhotoWorks rendering interface from the *PhotoWorks* item on the main SolidWorks menu bar, or from the PhotoWorks toolbar. This menu bar is displayed whenever a SolidWorks part or assembly document is open.

❑ **PhotoWorks materials give you control over the appearance of SolidWorks models.** *Materials* are used in the PhotoWorks software to specify model surface properties such as color, texture, reflectance, and transparency. The PhotoWorks software is supplied with an extensible archive of pre-defined materials, (metals, plastics, woods, stones, and so on), which can be attached to, and stored with, individual SolidWorks parts and faces. *Texture mapping* is also supported, enabling you to attach 2D textures such as scanned images and logos, to the surfaces of your models. *Material archives* help you to organize and manage your own collections of materials and textures.

❑ **PhotoWorks scenes add photo-realism to your designs.** Each SolidWorks model is associated with a PhotoWorks *scene*, for which you can specify properties such as lighting, shadows, and backgrounds. Once you are happy with the look of your scene, you can save it to an image file. You can then incorporate the image in design proposals, technical documentation, product presentations, and so on. *Scene archives* help you to organize and manage your own scene templates.

Getting Started
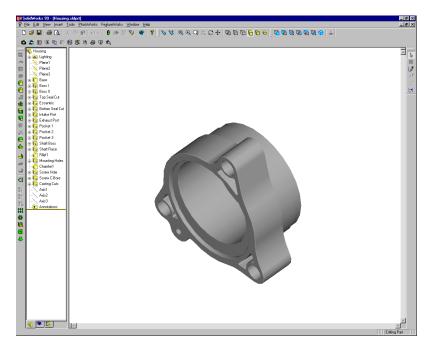
This section describes getting started with the PhotoWorks software.

1   If **PhotoWorks** does not appear on the SolidWorks main menu bar, click **Tools, Add-Ins**, select **PhotoWorks**, and click **OK**.

2   Click **Open** 🖼 on the Standard toolbar, and open the file:

  *install_dir***samples\tutorial\photowks\Housing.sldprt**

Notice that there is a **PhotoWorks Help** item available on the main **Help** menu, and that a PhotoWorks toolbar has been added to the SolidWorks window, beneath the Standard toolbar. Context-sensitive, online help is also available for most PhotoWorks features by clicking the **Help** button in the dialog box or by pressing the **F1** key.

3   Set view orientation to ***Isometric**, then select the **Shaded** view mode from the **View** toolbar. Your screen should look like this:

Checking the Options Settings

Before you begin, make sure that your SolidWorks settings match the ones used in this
example so that your results will be the same.

1   Click **Tools, Options**, and select the **Grid/Units** tab. Make sure that **Length Unit** is set
to **Millimeters** and **Decimal places** is set to 2.

2   Click **Tools, Options**, and select the **Performance** tab. Make sure that **Fine** is selected
in the **Shaded** section.

3   Click **OK**.

Now set PhotoWorks options.

1   Click **Options** 🔍 on the PhotoWorks toolbar, or click **PhotoWorks**, **Options**.

2   On the **Render** tab, the PhotoWorks software provides options for trading image
quality with rendering performance. Select these options if desired:

- **Anti-aliasing** eliminates jagged silhouette edges. Rendering is slower, but images
  are smoother. For final image rendering, select this option.

- **Overlay image** prevents the current image from being cleared before the next image
  is rendered. This option does not affect rendering speed.

3   On the **Materials** tab, the PhotoWorks software provides options for controlling the
transfer of material properties between the SolidWorks and PhotoWorks software. By
default, material properties such as color and reflectance are maintained separately in
SolidWorks and PhotoWorks software. The options are:

- **Overwrite SolidWorks properties on select/edit** updates SolidWorks material
  properties automatically when selecting or editing materials within PhotoWorks.

- **Apply SolidWorks properties for render** causes the PhotoWorks software to use
  SolidWorks material properties during rendering.

For the purpose of this example, leave both boxes clear.

4   Click **OK**.

Shaded Rendering

Shaded rendering is the basis for all photo-realistic rendering in PhotoWorks.

1 Click **Render** 📷 on the PhotoWorks toolbar, or click **PhotoWorks**, **Render**.

The PhotoWorks software produces a solid, smooth-shaded rendering of the part against a graduated background.

The **PhotoWorks - Default Material** dialog is displayed, indicating that the part has been rendered with the default material, **Polished Plastic**.

The default material can be applied to the model automatically for you, if you do not wish to create and apply a material yourself. However, in this example, you will learn how to create and apply your own materials. (You can also set up your own default material.)

The PhotoWorks software asks whether you wish to apply this material to the model.

2 Click **No**.

3 Use the arrow keys, **Orientation** dialog box, zoom, or rotate tools to change the part view.

The view returns to the normal, SolidWorks, shaded view.

4 Click **Render** 📷 or **PhotoWorks**, **Render** again.

Each time you change the view, you need to render the image again.

To abort a rendering, click **Stop** in the **PhotoWorks - Render** dialog box.

Selecting a Procedural Material

Next, you can add more realism to the part by selecting a PhotoWorks *material* for it. A material defines how the surface of a part reacts to light. Each material consists of properties that determine various aspects of its appearance, such as surface color, reflectance, roughness, transparency, and pattern.

The PhotoWorks software supports both *procedurally defined* (solid) and *texture-mapped* (wrapped) materials.

In PhotoWorks, all material selection operations are performed from the **Material** tab on the **PhotoWorks - Material Editor** dialog box.

1 Click **Materials** 🔳 on the PhotoWorks toolbar, or click **PhotoWorks, Materials**.

The material **Manager** tab has two display panels:

- A *Material Archive* tree, which lists all the material archives currently available
- A *material selection* area, in which to view and select materials

2 Double-click the **Stock Procedural** archive (or click the **+** beside its name) to display the *material classes* it contains.

3 Click the **Metals** class to display the materials it contains.

The material selection area shows a rendered image of a sphere for each material in the class.

4 Use the scroll bar to locate the **Chrome** material, then select it.

The **Preview** window, to the right of the material editor, is updated to show how the part will appear when it is rendered.

5 Click **Apply**, then click **Close**.

NOTE: You can also select and apply a material in one operation by double-clicking the image in the material selection area.

---

6 Click **Render** 📷 or **PhotoWorks, Render**.
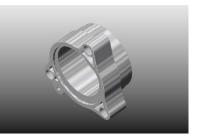
The part is rendered with a chrome surface.

7 Rotate the part, then render again.

Notice how the reflections change on the curved surfaces of the part.

8 Click **Materials** 🔳 or **PhotoWorks, Materials** again.
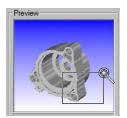
Notice that the icon representing the material currently associated with the part is highlighted in the material selection area when you re-open the material editor.

Now examine the **Preview** window of the material editor.

In the **Rendering** section, you have the following options for rendering the preview:

- In **Automatic** mode 📷, each time you change a material property, the preview is rendered again.
- In **Manual** mode 📷, you can change as many properties as you want, then render the preview once to incorporate all the changes. To render the preview in **Manual** mode, click **Automatic** mode 📷. Click again to return to **Manual** mode 📷.
- In **Full** mode 📷, the PhotoWorks software uses photo-realistic rendering for the preview.
- In **Interactive** mode 📷, the PhotoWorks software uses OpenGL rendering for the preview.

NOTE: You can also use PhotoWorks OpenGL rendering in the main SolidWorks window, by selecting **Interactive Rendering** from the main **PhotoWorks** menu, or by clicking **Interactive Rendering** 📷 on the PhotoWorks toolbar.

- In the **Display components** section, you can choose to display the **Model**, or you can choose a simpler geometric shape. Preview rendering is faster with a simpler shape that approximates that of the model, such as a **Cylinder**. For certain types of change, you may need to see the details on the model.
- Click **Zoom to Fit** 🔍 to display the part full size in the **Preview** window.
- Click **Zoom to Area** 🔍 to zoom in on a particular area of the **Preview** window by positioning the pointer over it, then clicking and dragging a bounding box to enclose the selected area.

- Click **Rotate View** 🔄 to rotate the part by clicking and dragging in the **Preview** window.
- You can also choose to disable various material properties temporarily, such as reflectance and transparency, to further accelerate preview rendering.

NOTE: The material editor is a *modeless* dialog. You can keep the **PhotoWorks - Materials Editor** dialog box open while selecting other SolidWorks geometry and reference objects.

Adding Color

Some Photoworks **Stock Procedural** materials (such as those in the **Metals** class) have specific colors associated with them. Other materials have colors that you can edit.

1 Under **Stock Procedural**, in the **Plastics** class, double-click the **Polished Plastic** material to select and apply it.

2 Click **Render**   or **PhotoWorks, Render**.

The part is rendered in gray plastic.

Now change the color.

1 Click **Materials**   or **PhotoWorks, Materials**, and click the **Color** tab.

2 In the **Colors** section, click **Edit**.

3 Select a color from the palette, then click **OK**.

The **Preview** window shows how the part will appear when it is rendered.

4 Click **Apply**, then click **Close**.
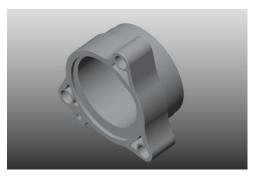
5 Click **Render**   or **PhotoWorks, Render**.

Some materials have both primary and secondary colors.

1 Click **Materials**   or **PhotoWorks, Materials**.

2 Under **Stock Procedural**, in the **Stones** class, select the **Brick** material.

3 Click the **Color** tab.

4 Edit the **Primary color** and **Secondary color** to your liking, observing the effect in the **Preview** window. (The **Secondary color** determines the color of the mortar between bricks.)

5 Change the **Pattern scale** value to adjust the size of the bricks. (The PhotoWorks software calculates the initial scale value automatically. You can reset the **Pattern scale** to this initial value by clicking **Auto scale** at any time.)

6 Click **Apply**, then click **Close**.

7 Click **Render**   or **PhotoWorks, Render**.

Selecting a Texture-Mapped Material

Texture mapping enables you to wrap 2D textures, such as bitmaps of scanned images, onto SolidWorks models.

The PhotoWorks software is supplied with several archives of texture-mapped materials to get you started. You can create and manage material archives of your own using the PhotoWorks material editor.

1   Click **Materials** ⊞ or **PhotoWorks**, **Materials**.

2   Double-click the **Wood Textures** archive (or click the **+** beside its name) to display the material classes it contains.

3   Click the **Wood** class to display the materials it contains.

The material selection area shows a thumbnail texture map for each material in the class.

4   Select the **Pine** material.

TIP:	To view a texture map at its full resolution, right-click the thumbnail in the material selection area.

5   Click **Close**.

The PhotoWorks software notifies you that the material has changed, and asks if you wish to apply the change.

6   Click **Yes**.

7   Set view orientation to ***Isometric**.

8   Click **Render** 📷 or **PhotoWorks**, **Render**.

The part is rendered with a pine veneer finish. Notice how the woodgrain wraps around the faces of the part. The mapping required to wrap a 2D texture around a 3D SolidWorks model is determined by the shape of the SolidWorks geometry.

The PhotoWorks software supports several *texture spaces,* and will examine the SolidWorks geometry to select the best mapping in each case. By editing the properties of the texture space, you can change the appearance of the texture as it is applied to the part.

Changing the Texture and Reflectance

Next, you can change the texture to brushed metal.

1 Click **Materials** 🔳 or **PhotoWorks**, **Materials**.

2 Open the **Metal Textures** archive.

3 Click the **Brushed** class, then click the **Brushed 1** material.

The preview is rendered with a brushed metal finish.

4 Click the **Texture Space** tab.

The PhotoWorks software has selected the **Automatic** texture space. This texture space selects one of the three world-coordinate axes (either *x*, *y*, or *z*) whose plane is most closely aligned with that of the geometry at each point on the surface of the model.

Other texture spaces available within the PhotoWorks software are **Planar, Cylindrical,** and **Spherical**.

5 Now click the **Reflectance** tab.

Notice that the **Style** is set to **Plastic**. The PhotoWorks software supports several reflectance styles.
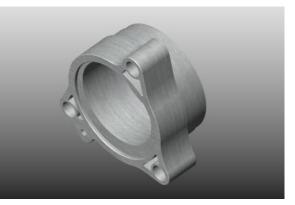
6 Change the **Style** to **Metal**.

The preview is rendered with a specular metallic appearance.

7 Change the **Style** to **Glass**.

The preview is rendered with a realistic approximation of glass reflectance, including transparency, reflection, and refraction.

8 Change the **Style** back to **Plastic**.

9 Click **Apply**, then click **Close**.

10 Click **Render** 📷 or **PhotoWorks**, **Render**.

Adding a Displacement

Now add a displacement to the material to give the part an irregular finish. A displacement adds small perturbations to the surface texture, to give an otherwise smooth material an irregular, indented, or undulating appearance. Displacements are useful for representing surface types such as rough metal castings and pressed sheet metal.

1 Click **Materials** 🔳 or **PhotoWorks**, **Materials**.

2 Click the **Displacement** tab.

3 Change the **Style** from **None** (the default) to **Rough**.

4 Set **Scale** to 0.01.

The **Scale** parameter controls the overall size of the displacement. Increasing the **Scale** makes the surface perturbations appear larger.

5 Set **Amplitude** to 0.1.

The **Amplitude** parameter controls the magnitude and orientation of the perturbations relative to the surface. A positive value causes the perturbations to appear as protrusions from the surface. A negative value causes them to appear as indentations in it.

6 Set **Detail** to 2.
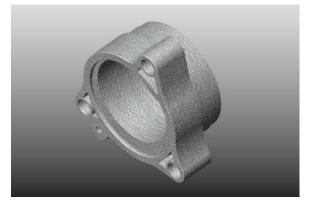
The **Detail** parameter controls the complexity of the surface texture. A low value produces a simple texture. A higher value produces a more complex texture.

7 Set **Sharpness** to 2.

The **Sharpness** parameter controls the boundaries between the perturbations. A low value produces abrupt changes between the peaks and troughs of the displacements. A higher value produces smoother transitions.

8 Click **Apply**, then click **Close**.

9 Click **Render** 📷 or **PhotoWorks**, **Render**.

The part is rendered with a rough, cast-metal finish.
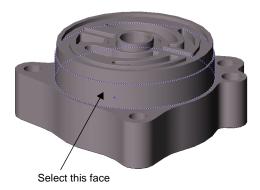
Applying Texture to Individual Faces

You can also apply texture-mapped materials to individual faces. For example, you may wish to use a texture to draw attention to a particular face of a model.

1 Set view orientation to ***Top**.

2 Select a face of a particular feature, in this case, **Boss I**.

3 Rotate the part to approximately the orientation shown.

4 Click **Materials** 🔲 or **PhotoWorks**, **Materials**.

> **NOTE:** When you edit the material on a selected face, the **Preview** window displays the selected face only, rather than the whole part or assembly.

Select this face

5 Select the **Knurl Large** material in the **Metal Textures**, **Machined** class.

Now change the appearance of the texture to a more appropriate scale for the boss.

1 Click the **Texture Space** tab.
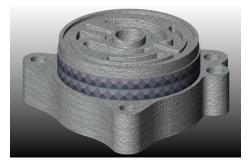
Notice that the PhotoWorks software has selected the **Cylindrical** texture space for the mapping onto the boss. You need to scale the texture both around and along the axis of the boss.

2 Drag the **Scaling**, **Around axis** slider to a position between the third and fourth notches on the scale, and observe the effect in the **Preview** window.

3 Set **Scaling**, **Along axis** to 22.00mm.

4 Set **Orientation**, **Offset along axis** to 3.75mm.

This shifts the texture along the axis of the boss so that the two rows of knurls are positioned correctly.

5 Click **Apply**, then click **Close**.

6 Click **Render** 📷 or **PhotoWorks**, **Render**.

The PhotoWorks software adds a knurled metal finish to the boss.

Adding a Bump Map
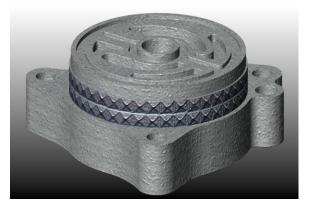
You can apply a special form of displacement called a *bump map* to enhance the 3D appearance of a material.

1   With **Boss I** still selected, click **Materials** or **PhotoWorks**, **Materials**.

2   Click the **Displacement** tab.

3   Set the **Style** to **Bump Map**.

Notice that by default the PhotoWorks software selects the same texture map file used for the material as the basis for the bump map. You can specify a different file for the bump map by clicking **Filename**, **Browse**.

4   Click **Apply**, then click **Close**.

5   Click **Render** or **PhotoWorks**, **Render**.

The bump map gives a more pronounced, 3D appearance to the knurled finish on the boss.

Adding a Decal to a Face
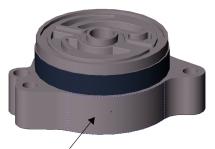
You can use the PhotoWorks *decal editor* to attach custom labels, such as company logos or part numbers, to SolidWorks models.

The PhotoWorks software includes a wizard to take you through the steps involved in creating and adding a decal to a SolidWorks model.

1   Set view orientation to ***Top**, and rotate the part to approximately the orientation shown.

2   Select the large curved face on the **Base**.

3   Click **Decals** 🔳 on the PhotoWorks toolbar, or click **PhotoWorks, Decals**.

The **PhotoWorks - Decal Editor** dialog box appears, which includes:

- A *Decal Manager* tree, which lists all decals attached to the current part, feature, or face.

- A *display* area, in which to view the components of individual decals.

Notice that **Create new decal with wizard** is selected.

4   Click **Create New Decal** 🔲 on the decal editor toolbar.

The **PhotoWorks - Decal Wizard** is displayed.

5   After reading the welcome note, click **Next** to select a decal image.

6   Click **Browse**, then locate and open this file:
*install_dir***samples\tutorial\photowks\decals\pw_image.bmp**.

The image file contains the decal artwork – in this case, a simple part number.

7   Click **Next** to create a decal mask.

8   Click **From file**, then click **Next**.

9   Click **Browse**, then locate and open this file:
*install_dir***samples\tutorial\photowks\decals\pw_mask.bmp**.

10   Click **Next** to view the complete decal, consisting of the image combined with the mask.

11   Click **Next** through to the **Finished!** page of the wizard, then click **Finish**.

The PhotoWorks software adds the new decal to the Decal Manager tree, giving it the name **Decal1**. The PhotoWorks software displays the components of the decal in the display area on the decal **Manager** tab. Also, **Image**, **Mask**, and **Mapping** tabs are added to the **PhotoWorks - Decal Editor** dialog box.

Select this face

Adjusting the Decal

Now use the decal editor to fine-tune the scale and orientation of the decal on the face.

1 With **Decal1** still selected in the Decal Manager tree, click the **Mapping** tab.

Notice that the PhotoWorks software has created a **Cylindrical** mapping for the decal, with reference to the **Selected face**. However, the scale and orientation of the decal require some adjustment to position it correctly.

2 Drag the **Scaling**, **Around axis** slider to a position halfway between **Small** and **Large**.

3 Set **Scaling**, **Along axis** to 14.00mm.

4 Set **Orientation**, **Rotation about axis** to 85°.

The **Preview** window shows the decal correctly sized and centered on the face.

5 Click **Close**.

The PhotoWorks software notifies you that the decal has changed, and asks if you wish to apply the change.

6 Click **Yes**.

7 Click **Render** 📷 or **PhotoWorks**, **Render**.

The PhotoWorks software scales the decal and offsets it to the specified position on the face.

Editing Decals

You can change the image, mask, or mapping properties of a decal at any time by first selecting it in the Decal Manager tree, then clicking the appropriate tab. You can also rename any decal by selecting and editing its name in the Decal Manager tree. (To access the Decal Manager tree, click **Decals** 🖼 or **PhotoWorks, Decals**.)

You can use the tools on the Decal Manager toolbar to manipulate decals:

❑ Click **Copy Decal** 🖼 and **Paste Decal** 🖼 to duplicate any selected decal in the Decal Manager tree, for positioning elsewhere on the part.

❑ Click **Toggle Decal Display** 🖼 to turn on or off the display of any selected decal. This is useful for temporarily suppressing one or more decals when positioning multiple overlapping decals on a part.

❑ Click **Cut Decal** ✂ to delete any selected decal from the Decal Manager tree.

❑ Click **Move Decal Up** 📄, **Move Decal Down** 📄, or **Reverse Order of Decals** 📄 to re-order overlapping decals in the Decal Manager tree. These actions change the order in which the decals are displayed on the part when it is rendered. You can also re-order decals by dragging and dropping them within the Decal Manager tree.

Composing a Scene
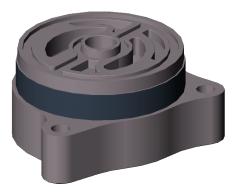
With the PhotoWorks software you can add advanced rendering effects such as shadows and reflections, as well as composing backgrounds against which to display SolidWorks parts and assemblies.

Composing a scene can improve visual realism by giving your model a more solid, 3D appearance. Rather than leaving the model floating in space, shadows can be employed to anchor it against a simple geometric backdrop. You can apply PhotoWorks materials to the backdrop for added realism. A complex model may also produce *self-shadowing*, where one part of the model blocks some of the light falling on another part.

Scene composition is performed using the PhotoWorks *scene editor*.

1 Set view orientation to ***Top**, and rotate the part to approximately the orientation shown.

2 Click **Scene** ⚓ on the PhotoWorks toolbar, or click **PhotoWorks, Scene**.

The **PhotoWorks - Scene Editor** dialog box is displayed, and includes a scene **Manager** tab, from which to access scene archives, plus separate pages for specific scene properties such as foregrounds, backgrounds, and scenery.

The scene **Manager** tab has two display panels:

- A *Scene Archive* tree, which lists all the scene archives currently available
- A *scene selection* area, in which to view and select scene templates

Notice that the icon representing the scene currently associated with the part is highlighted in the scene selection area when you open the scene editor. In this example, the **Default** scene, in the **Basic** class of the **Stock Combinations** archive, has already been selected.

3 Click the **Lighting** tab.

4 Click **Display shadows**.

The PhotoWorks software generates shadows for all SolidWorks directional lights, point lights, and spotlights in the scene.

5 In the **Preview** window of the scene editor, under the **Display components** section, select the **Shadows** check box.

Notice how the **Preview** window shows the raised boss casting a shadow onto the base of the housing. Internal self-shadowing of the part is also visible.

NOTE: You can also specify shadow properties for individual SolidWorks lights, using the **PhotoWorks properties** on the appropriate SolidWorks **Light properties** dialog boxes.

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6   Click the **Background** tab.

7   Change the **Style** from **Graduated** (the default) to **Clouds**.

8   In the **Parameters** section, make sure that **Scale** is selected, then set **Number** to 2.

9   Modify the **Sky Color**, **Cloud Color**, and **Detail** parameters, if desired, observing the effect in the **Preview** window.

Other background options include scaled or tiled images, or plain colors.

The scene editor also includes a **Foreground** tab, from which you can select various styles of attenuation, to simulate atmospheric phenomena, such as fog and depth-cueing.

10   Click **OK**.

11   Click **Render** 📷 or **PhotoWorks**, **Render**.

Creating Background Scenery

The visual effectiveness of your presentation can be improved still further by setting the part against a geometric backdrop, rather than simply leaving it suspended in space.

With PhotoWorks, you can create simple background scenery consisting of a horizontal *base* plane and vertical *sides* surrounding the part. You can control the size and position of the scenery relative to the part, and select PhotoWorks materials to associate with the base and sides.

The scenery dimensions are calculated from the bounding box of the SolidWorks model. The scenery will never obscure the part. Only those planes visible behind the part will be displayed. Any reflective materials attached to the part will pick up and reflect color and texture from the background scenery.

1. Click **Scene** ⚜ or **PhotoWorks**, **Scene**, then click the **Scenery** tab.

2. In the **Base** section:

   a) Click **Display**.

   Notice that the default material, **Polished Plastic**, has been selected for the base.

   b) Click **Edit**.

   The **PhotoWorks - Material Editor** dialog box appears.

   c) Double-click **Stone Textures**, click **Stone**, then select the **Pink Marble** material.

   d) Click the **Texture Space** tab. In the **Scaling** section, set both **Width** and **Height** to 65.00mm.

   e) Click **OK**.

3. In the **Sides** section:

   a) Click **Display**.

   Notice that the default material, **Polished Plastic**, has been selected for the sides.

   b) Click **Edit**.
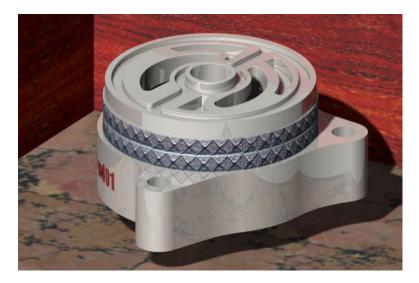
   The **PhotoWorks - Material Editor** dialog box appears.

   c) Double-click **Wood Textures**, click **Wood**, then select the **Mahogany** material.

   d) Click the **Texture Space** tab. In the **Scaling** section, set both **Width** and **Height** to 50.00mm.

   e) Click **OK**.

4. In the **Base size** section, set both **Base width** and **Base height** to 125.00mm, to reduce the size of the base relative to the model.

5. Set **Base offset** to -25.00mm, to move the base closer to the model.

6. Set **Sides height** to 75.00mm.

7. Click **OK**.

8 Now change the part material one more time:

   a) Click on the background in the SolidWorks window to select the entire part.

   b) Click **Materials** 🔳 or **PhotoWorks**, **Materials**.

   c) Double-click **Stock Procedural**, click **Metals**, then select the **Silver Plate** material.

   d) Click **Apply**, then click **Close**.

9 Click **Render** 📷 or **PhotoWorks**, **Render**.

   Notice how the base of the part reflects the knurled boss and the background scenery.

Saving an Image File

You can save a PhotoWorks image to a file for use in design proposals, technical documentation, product presentations, and so on. The PhotoWorks software supports Bitmap (**.bmp**), TIFF (**.tif**), Targa (**.tga**), and JPEG (**.jpg**) formats, as well as PostScript (**.ps**) and the PhotoWorks image format (**.lwi**).

1 Click **Options** 🔍 or **PhotoWorks**, **Options**.

2 Click the **Image Output** tab.

3 Click **Render to file**.

   The PhotoWorks software suggests an image file name based on the name of the part, along with the extension appropriate to the selected image format. By default, the image will be stored in the same directory as the part.

4 If you wish to select a different file name, file type, or storage location for the image:

   a) Click **Browse**.

   b) Locate the directory in which you wish to store the image.

   c) Enter a name for the file.

   d) Select the desired format from the **Save as type** list.

   e) Click **Save**.

5 Optionally, you may set **Image Size**, **Width,** and **Height**.

6 Click **OK**.

7 Click **Render** 📷 or **PhotoWorks**, **Render**.

   The PhotoWorks software notifies you that the image will be saved as *install_dir***samples\tutorial\photowks\Housing.bmp**, and requests confirmation of the image file details.

8 Click **Yes**.

   The PhotoWorks software renders the image to a file, and notifies you when processing is complete.

9 Click **OK**.

Viewing an Image File

You can view previously saved images using the PhotoWorks image viewer. All the image formats available in the PhotoWorks software (except PostScript) are supported by this utility.

1   Click **View Image File** 🖳 on the PhotoWorks toolbar, or click **PhotoWorks**, **View Image File**.

2   Locate an image file (*install_dir***samples\tutorial\photowks\Housing.bmp**, for example), then click **Open**.

> **NOTE:** Click **Preview** on the file browser if you wish to preview the image file before opening. This is useful if you have several image files from which to choose.

The PhotoWorks software loads the image file and displays it in a separate window. The SolidWorks menu bar is disabled temporarily while viewing an image file.

3   Close the **PhotoWorks - Image Viewer** window.

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