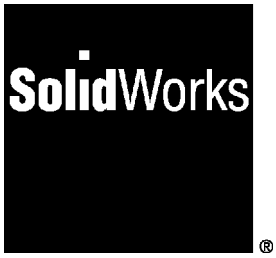


SolidWorks 2001 Student Workbook

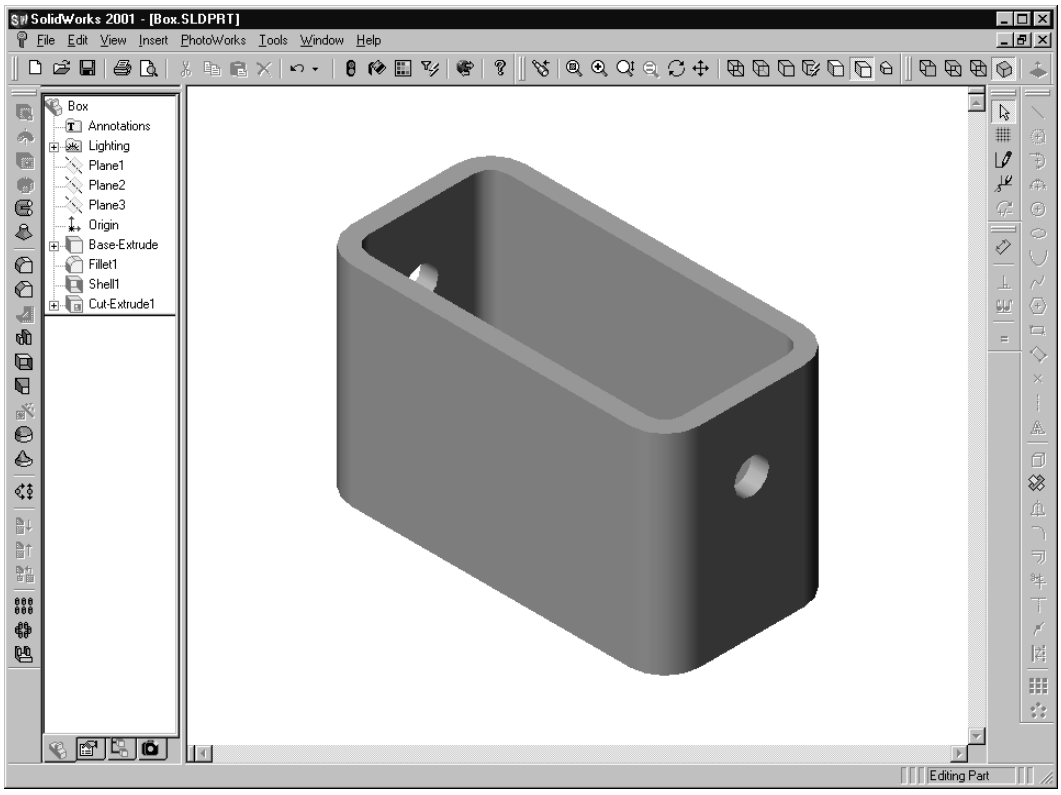
SolidWorks Corporation
300 Baker Avenue
Concord, Massachusetts 01742

Contents

1

Lesson 1: Basic Functionality

Upon successful completion of this lesson, you will be able to understand the basic functionality of SolidWorks and create the following part:

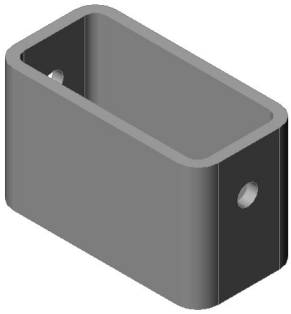


This lesson plan corresponds to *SolidWorks 2001 Getting Started*, Chapter 2, and the section *More about Basic Functionality*, at the end of Chapter 6.

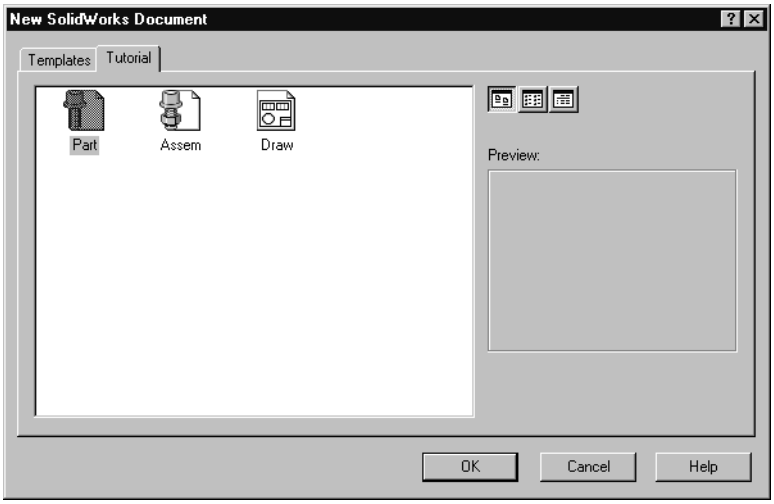
Active Learning Exercise

Use SolidWorks to create the box shown at the right.

The step-by-step instructions are given below.



Create a New Part Document



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

   The **New SolidWorks Document** dialog box appears.
2. Click the **Tutorial** tab.
3. Select the **Part** icon.
4. Click **OK**.

   A new part document window appears.

Base Feature

The Base feature requires:

- Sketch plane – Plane1 (default plane)
- Sketch profile – 2D Rectangle
- Feature type – Base-Extruded feature

Open a Sketch

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

   The sketch opens on the `Front` plane**.** `Front` is the default plane listed in the FeatureManager design tree.

Confirmation Corner

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

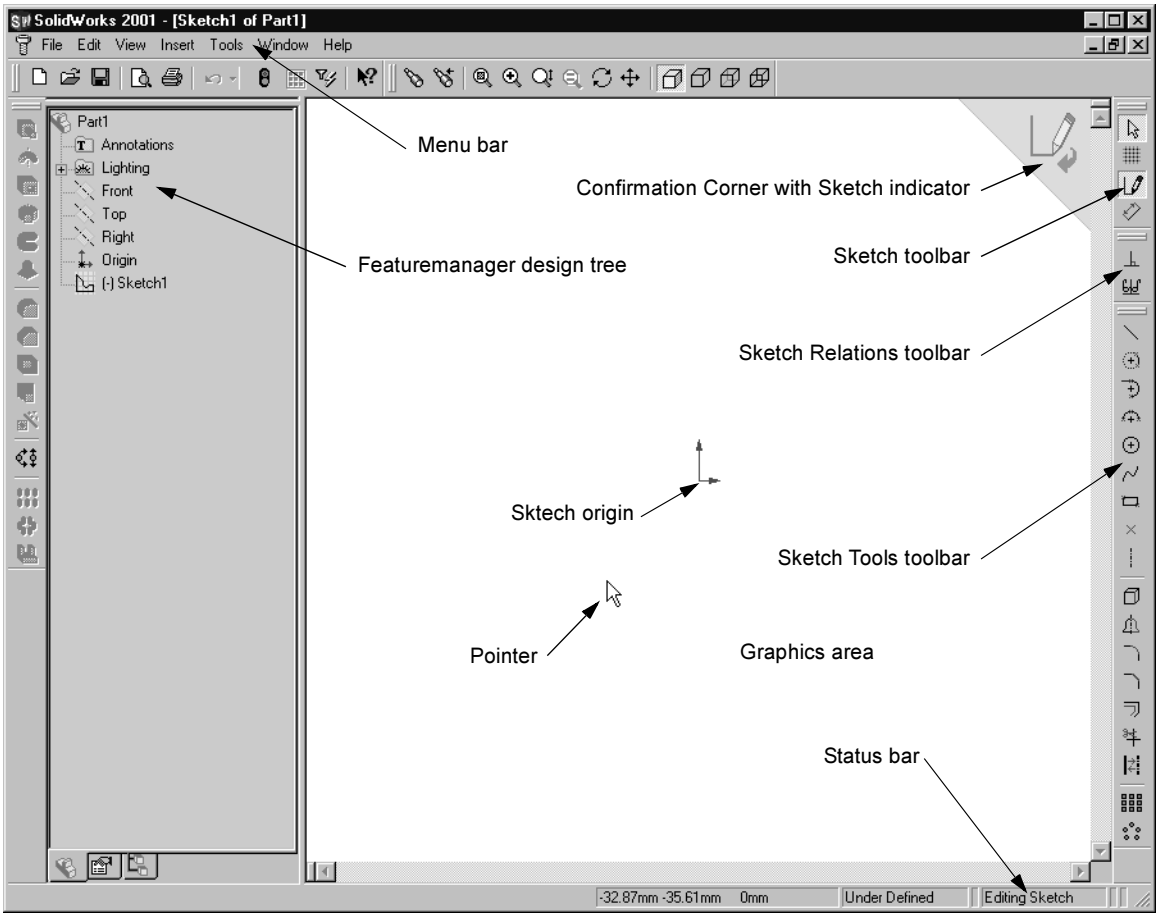
Sketch Indicator

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

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

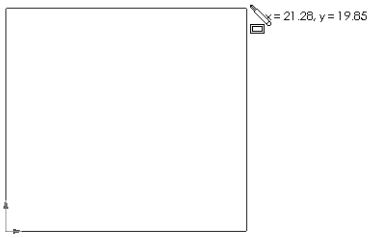
Overview of the SolidWorks Window:

- A sketch origin appears in the center of the graphics area.
- 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, in relation to the sketch origin.



Sketch a Rectangle

6. Click **Rectangle** on the Sketch Tools toolbar.
7. Click the sketch origin to start the rectangle.
8. Move the pointer up and to the right, to create a rectangle.
9. Click the mouse button again to complete the rectangle.



Add Dimensions

10 Click **Dimension** on the Sketch Relations toolbar.

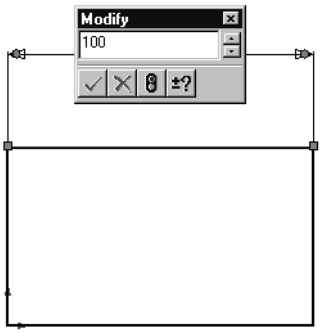
The pointer shape changes to .

11 Click the top line of the rectangle.

12 Click the dimension text location above the top line.

The **Modify** dialog box is displayed.

13 Enter **100**. Click or press **Enter**.

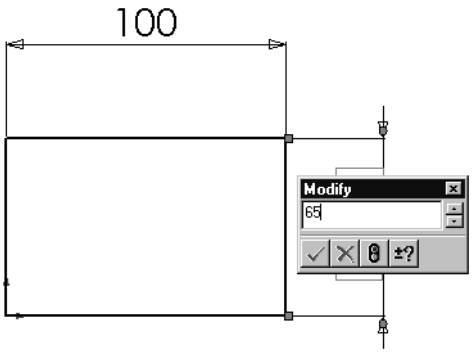


14 Click the right edge of the rectangle.

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

Click .

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



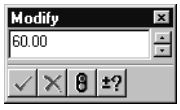
Changing the Dimension Values

The new dimensions for the box are 100mm x 60mm. Change the dimensions. Use the **Select** tool.

16 Click **Select** on the Sketch toolbar.

17 Double-click **65**.

The **Modify** dialog box appears.



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

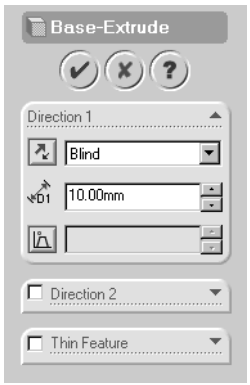
19 Click .

Extrude the Base Feature.

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

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

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



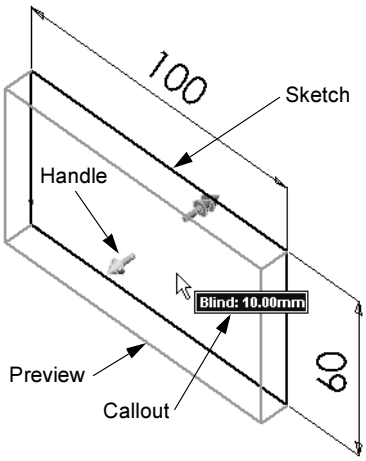
21 Preview graphics.

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

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

Click on the screen to set the preview into **Shaded** mode.

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



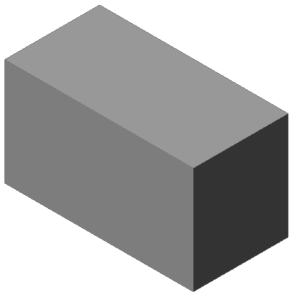
22 Extrude Feature settings.

Change the settings as shown.

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

23 Create the extrusion. Click **OK** .

The new feature, `Base-Extrude`, is displayed in the FeatureManager design tree.

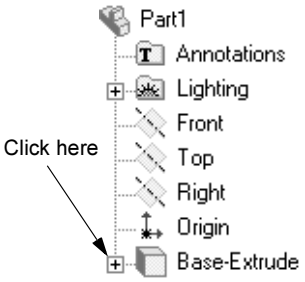


Tip

- The **OK** button on the PropertyManager is just one way to complete the command.
- A second method is the set of **OK/Cancel** buttons in the confirmation corner of the graphics area.

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

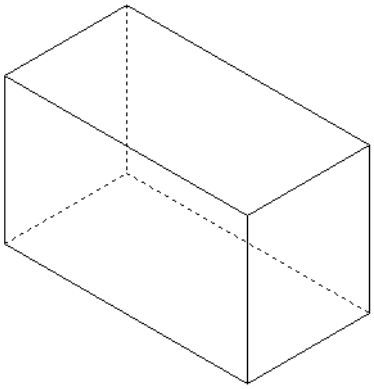
24 Click the plus sign ⊞ beside `Base-Extrude` in the FeatureManager design tree. Notice that `Sketch1`, which you used to extrude the feature, is now listed under the feature.



View Display

25 Change the display mode. Click **Hidden In Gray** on the View toolbar.

Hidden In Gray allows you to select hidden back edges of the `box`.



Save the Part

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

The **Save As** dialog box appears.

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

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

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

Round the Corners of the Part

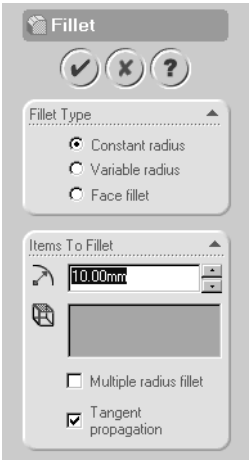
Round the four corner edges of the `box`. All rounds have the same radius (10mm). Create them as a single feature.

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

The **Fillet** PropertyManager appears.

29 Enter **10** for the **Radius.**

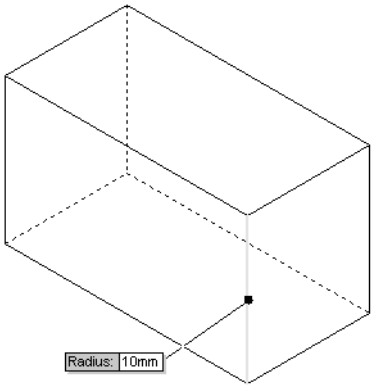
Leave the remaining settings at their default values.



30 Click the first corner edge.

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

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

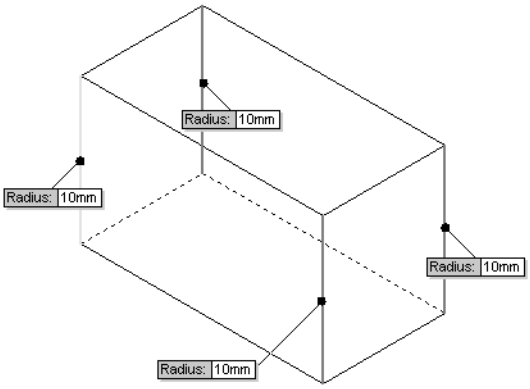


31 Identify selectable objects. Notice how the pointer changes shapes:

Edge: Face: Vertex:

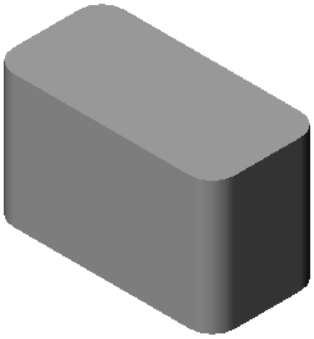
32 Click the second, third and fourth corner edges.

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



33 Click **OK** .

`Fillet1` appears in the FeatureManager design tree.



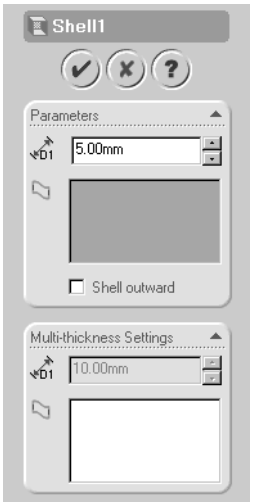
Hollow Out the Part

Remove the top face using the Shell feature.

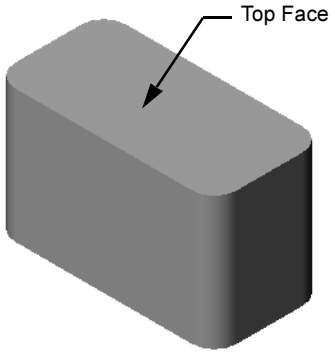
34 Click **Shell** on the Features toolbar.

The **Shell Feature** PropertyManager appears.

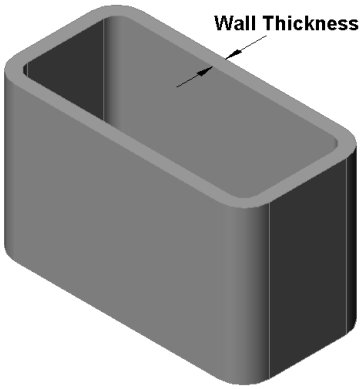
35 Enter **5** for **Thickness**.



36 Click the top face.



37 Click **OK** .



Extruded Cut Feature

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

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

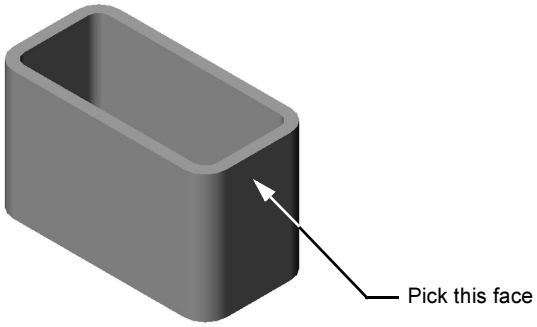
Open a Sketch

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

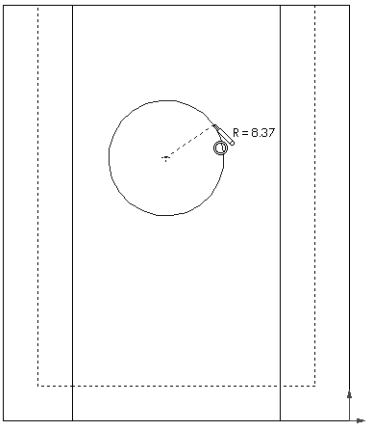
39 Click **Normal To** on the Standard Views toolbar.

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

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



Sketch the Circle



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

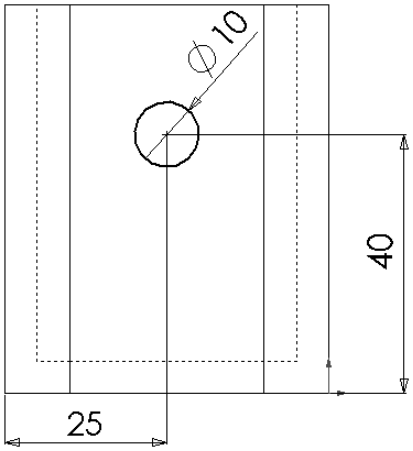
42 Position the pointer where you want the center of the circle. Click the left mouse button.

43 Drag the pointer to sketch a circle.

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

Dimension the Circle

Dimension the circle to determine its size and location.



45 Click **Dimension** on the Sketch Relations toolbar.

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

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

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

Extrude the Sketch

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

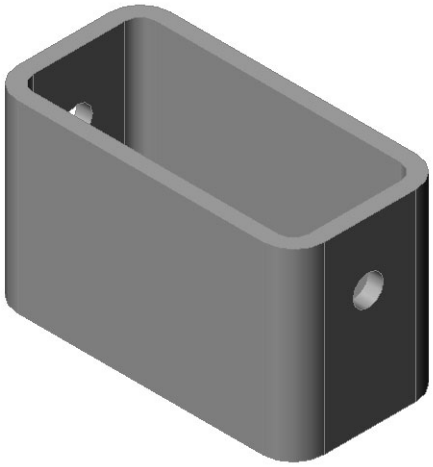
The **Extrude Cut Feature** PropertyManager appears.

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

51 Click **OK** .

52 Results.

The cut feature is displayed.



Rotate the View

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

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

54 Display the Isometric view. Click **Isometric** on the Standard Views toolbar.

Save the Part and Exit SolidWorks

55 Click **Save** on the Standard toolbar

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

5 Minute Assessment

1 How do you start a SolidWorks session?

_____

_____

2 Why do you create and use Document Templates?

_____

3 How do you start a new Part Document?

_____

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

_____

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

6 A SolidWorks 3D model consists of _____

_____

7 How do you open a sketch? _____

_____

8 What does the Fillet feature do?

_____

9 What does the Shell feature do?

_____

10 What does the Cut-Extrude feature do?

_____

11 How do you change a dimension value?

_____

Project

Switch plates are required for safety. They cover live electrical wires and protect people from electric shock. Switch plates are found in every home and school. They incorporate simple and complex designs.

⚠ Caution: Do not use metal rulers near switch plates attached to a live wall outlet.

Tasks

1 Measure a single light plate switch cover in millimeters.

2 Using paper and pencil, manually sketch the light plate switch cover.

3 Label the dimensions.

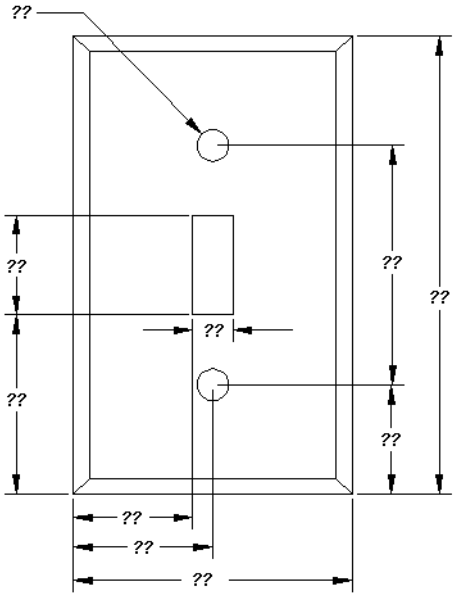
4 What is the base feature for the light plate switch cover?

Answer: _____

_____

_____

_____



5 Create a simple single light switch cover using SolidWorks. The filename for the part is `switchplate`.

6 What features are used to develop the `switchplate`?

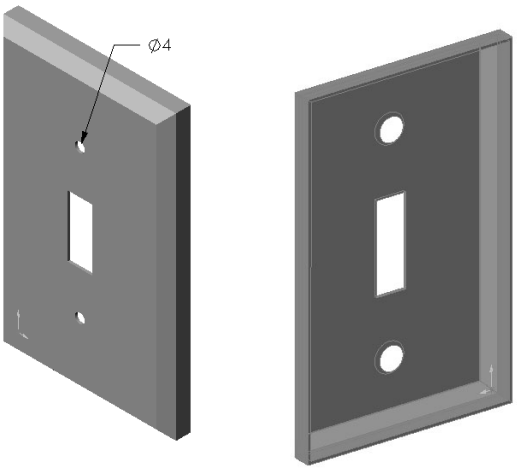
Answer: _____

_____

_____

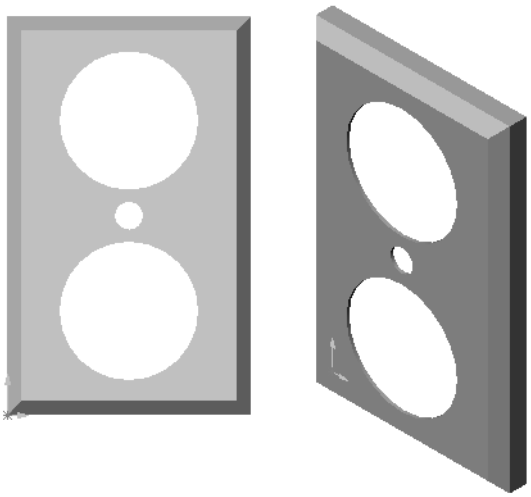
_____

_____



7 Create a simplified duplex outlet cover plate. The filename for the part is `outletplate`.

8 Save the parts. They will be used in later lessons.



Lesson 1 Vocabulary Worksheet

Name: _____Class: _____ Date:_____

Fill in the blanks with the words that are defined by the clues. Then find the words in the puzzle and circle them. The words may be vertical, horizontal, or diagonal. They may be spelled forward or backward.

1 The corner or point where edges meet: _____

2 The intersection of the three default reference planes:_____

3 A feature used to round off sharp corners: _____

4 The three types of documents that make up a SolidWorks model: (3 words) _____

_____

5 A feature used to hollow out a part: _____

6 Controls the units, grid, text, and other settings of the document:_____

7 Forms the basis of all extruded features: _____

8 Two lines that are at right angles (90°) to each other are: _____

9 The first feature in a part is called the _____ feature.

10 The outside surface or skin of a part: _____

11 A mechanical design automation software application:_____

12 The boundary of a face: _____

13 Two straight lines that are always the same distance apart are: _____

14 Two circles or arcs that share the same center are:_____

15 The shapes and operations that are the building blocks of a part: _____

16 A feature that adds material to a part: _____

17 A feature that removes material from a part: _____

18 An implied centerline that runs through the center of every cylindrical feature:_____

M H D N K R V V V R Y R A L U C I D N E P R E P L

Z H R I N C K B A S E M K Q S L I C T L H C P T N

L T A G E X G T R S R N S S E B F R U K D R E S Y

E D W I P G X C T T J I O D I M R N T T B M C O J

L V I R G Z D R L Y X B Y M L T K S N N P K R L L

L J N O Z V A E J A V M J Z B L E P K L E F K I N

A N G V T P Y M P B K H R L M C N L A E T C K D N

R X S M H T R T L Q P K M F E X T T L R T R N W H

A S E R U T A E F J H F G Q S B E Q T I J C M O L

P D K J G X H M T Y A M D J S R L X C N F X H R C

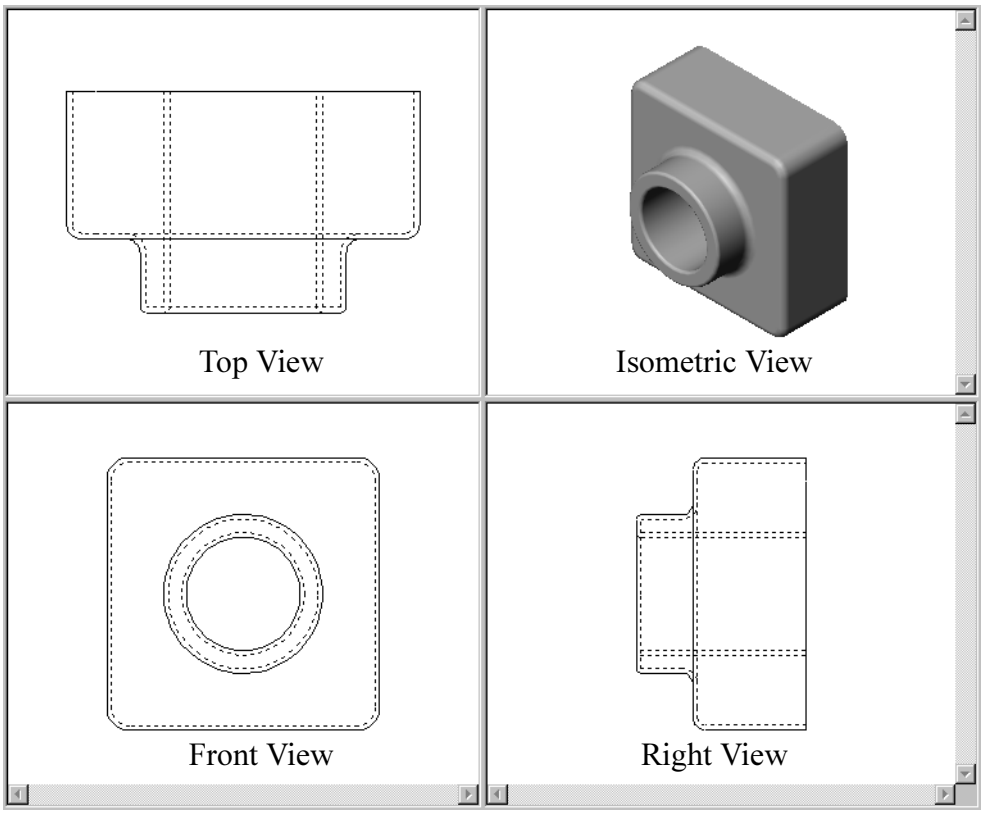
Q T M H J T Z B V C V Y X R A T M R G F D Q D K L

M K S H E L L T E J N H L X E T R E V P R Y T S M

2

Lesson 2: The 40-Minute Running Start

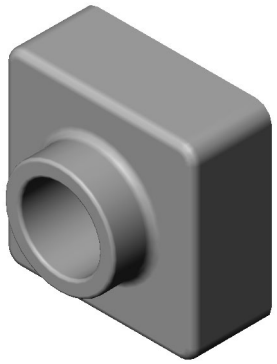
Upon successful completion of this lesson, you will be able to create and modify the following part:



This lesson plan corresponds to *SolidWorks 2001 Getting Started* pages 3-1 through 3-16.

Active Learning Exercise

Follow the instructions in *SolidWorks 2001 Getting Started* pages 3-1 through 3-16. In this lesson you will create the part shown at the right. The part name is `Tutor1.sldprt`.



5 Minute Assessment

1 What features did you use to create `tutor1`? _____

_____

2 What does the Fillet feature do?_____

_____

3 What does the Shell feature do? _____

_____

4 Name three view commands in SolidWorks.

_____

5 Where are the display buttons located? _____

_____

6 Name the three SolidWorks default planes. _____

_____

7 The SolidWorks default planes correspond to what principle drawing views? _____

_____

8 True or False. In a fully defined sketch, geometry is displayed in black. _____

_____

9 True or False. It is possible to make a feature using an over defined sketch. _____

_____

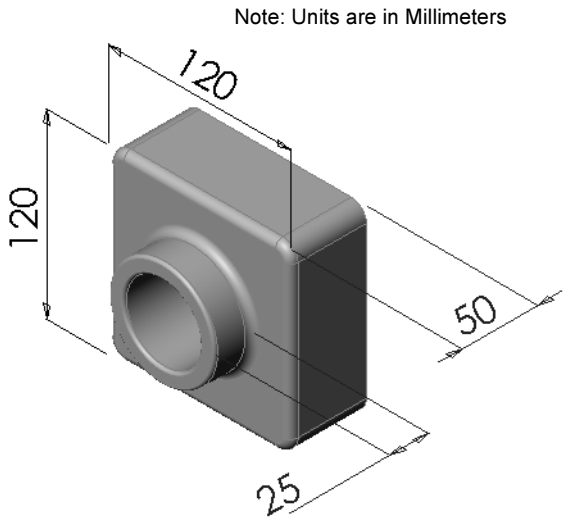
10 Name the primary drawing views used to display a model._____

_____

Additional Exercises

Task 1

The design for `Tutor1` was created in Europe. `Tutor1` will be manufactured in the US. Convert the overall dimensions of `Tutor1` from millimeters to inches.



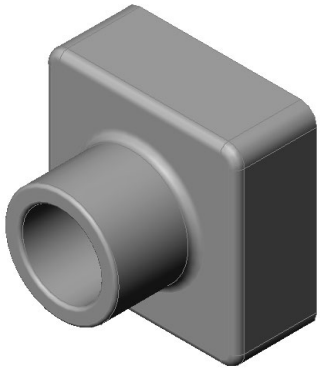
Given:

- Conversion: 25.4 mm = 1 inch
- Base-Extrude width = 120 mm
- Base-Extrude height = 120 mm
- Base-Extrude depth = 50 mm
- Boss-Extrude depth = 25 mm

Answer:

Task 2

The current overall depth of `Tutor1` is 75 mm. Your customer requires a design change. The new required overall depth is 100 mm. The Base-Extrude depth must remain fixed at 50 mm. Calculate the new Boss-Extrude depth.



Given:

- New overall depth = 100 mm
- Base-Extrude depth = 50 mm

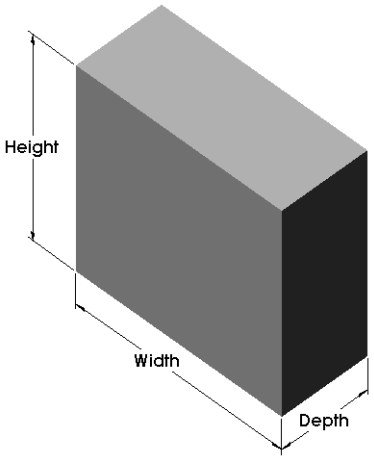
Answer:

Task 3

Using SolidWorks, modify `tutor1` to meet the customer's requirements. Change the depth of the Boss-Extrude feature such that the overall depth of the part equals 100mm. Save the modified part under a different name.

Task 4

Material volume is an important calculation for designing and manufacturing parts. Calculate the volume of the Base-Extrude feature in mm^3 for `tutor1`.



Answer:

Task 5

Calculate the volume of the Base-Extrude feature in cm^3.

Given:

- 1cm = 10mm

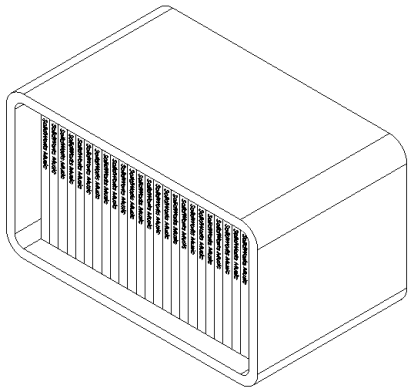
Answer:

Project – CD Storage Box

Description

You are part of a design team. The project manager has provided the following design criteria for a CD storage box:

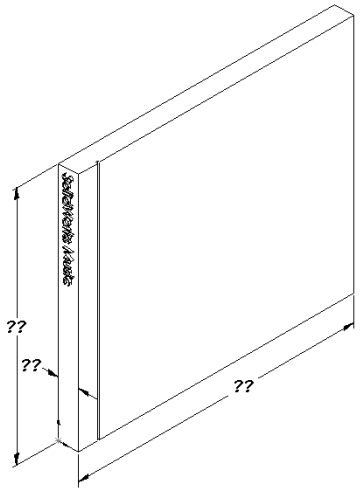
- The CD storage box is constructed of a polymer (plastic) material.
- The storage box must hold 25 CD jewel cases.
- The title of the CD must be visible when the jewel case is positioned in the storage box.
- The wall thickness of the storage box is 1cm.
- On each side of the storage box, there must be 1cm clearance between the jewel case and the inside of the box.
- There must be 2cm clearance between the top of the CD cases and the inside of the storage box.



- There must be 2cm clearance between the jewel cases and the front of the storage box.

Task 1

Measure the width, height, and depth of one CD jewel case. What are the measurements in centimeters?



Answer:

Width: _____

Height: _____

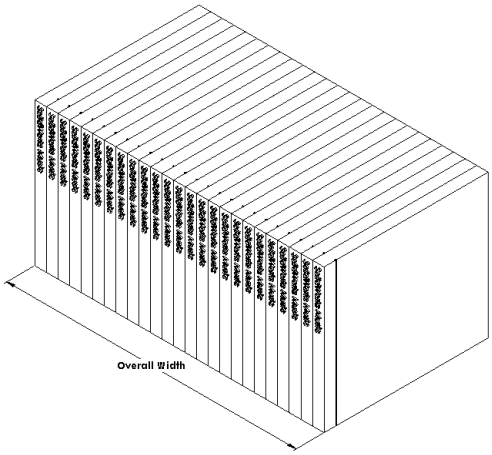
Depth: _____

Task 2

Using paper and pencil, manually sketch the CD jewel case. Label the dimensions.

Task 3

Calculate the overall size of 25 stacked CD jewel cases. Record the overall width, height and depth.



Answer:

Overall width: _____

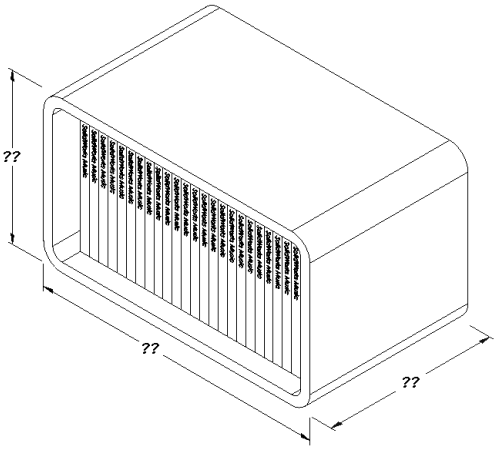
Overall height: _____

Overall depth: _____

Task 4

Calculate the overall *outside* measurements of the CD storage box. The box requires a clearance to insert and position the CD jewel cases. Add a 2cm clearance to the overall width (1cm on each side) and 2cm to the height. The wall thickness is equal to 1cm.

Answer:



Task 5

Create two parts using SolidWorks.

- Model a CD jewel case. You should use the dimensions you obtained in **Task 1**. Name the part `CD case`.

  Note: A real CD jewel case is an assembly of several parts. For this exercise, you will make a simplified representation of a jewel case. It will be a single part that represents the overall outside dimensions of the jewel case.
- Design a storage box to hold 25 CD jewel cases.
- Save both parts. You will use them to make an assembly at the end of the next lesson.

Project – Modeling

Description

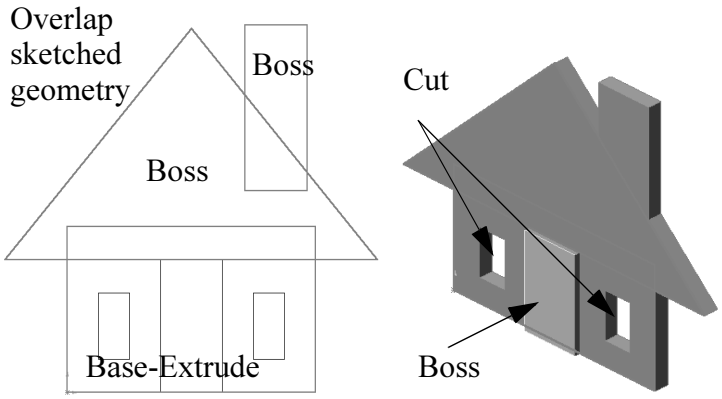
Look at the following examples. There are at least 3 features in each example. Identify the 2D Sketch tools used to create the shapes. You should:

- Consider how the part should be broken down into individual features.
- Focus on creating sketches that represent the desired shape. You do not need to use dimensions. Concentrate on the shape.
- Also, experiment and create your own designs.

Note: Each new sketch must overlap an existing feature.

Task 1

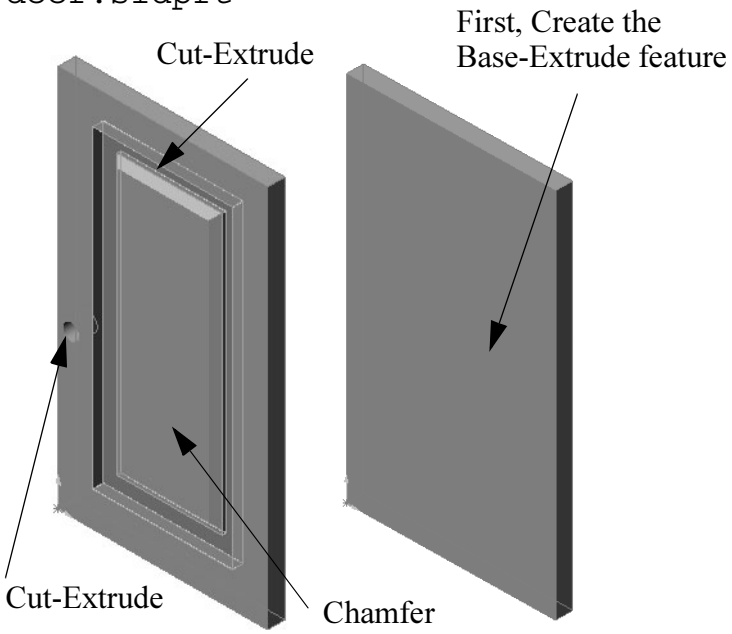
`house.sldprt`



Task 2

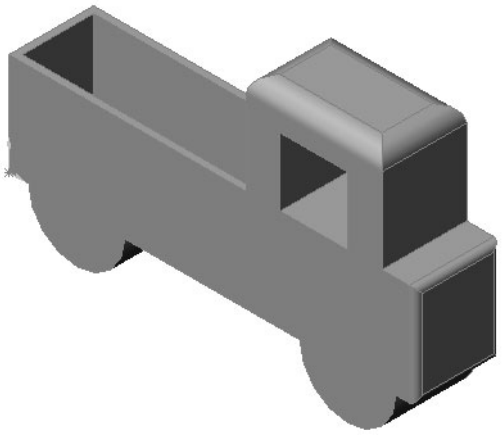
Note: The Chamfer feature is a new feature. The chamfer feature removes material along an edge. It works very similarly to a fillet except the result is a beveled edge rather than a rounded edge.

`door.sldprt`



Task 3

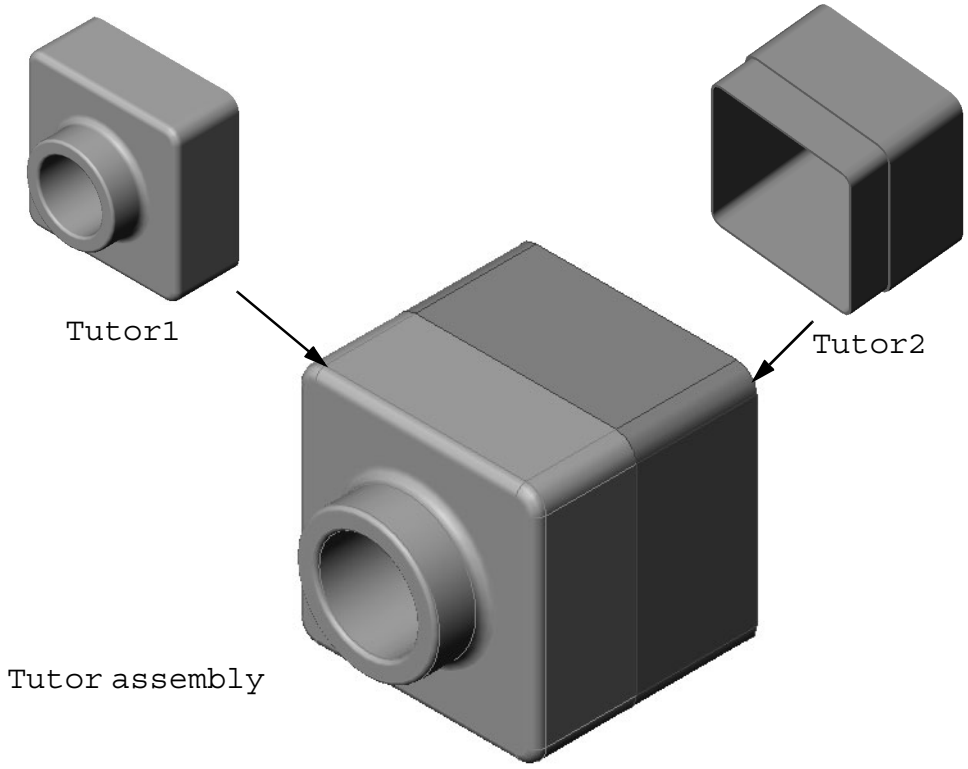
`truck.sldprt`



3

Lesson 3: Assembly Basics

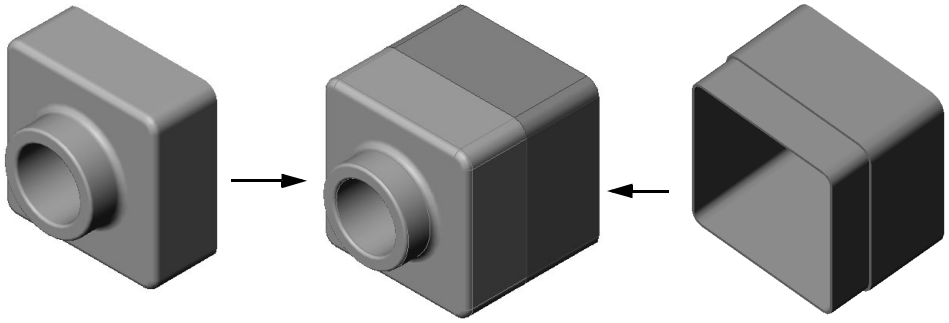
Upon successful completion of this lesson, you will be able to create and modify the part named Tutor2 and create the `Tutor` assembly.



This lesson plan corresponds to *SolidWorks 2001 Getting Started* pages 4-1 through 4-8.

Active Learning Exercises

Follow the instructions in *SolidWorks 2001 Getting Started* pages 4-1 through 4-8. In this lesson you will first create `Tutor2`. Then create you will create an assembly.



`Tutor1` `Tutor assembly` `Tutor2`

5 Minute Assessment

1 What features did you use to create `Tutor2`? _____

_____

2 What two sketch tools did you use to create the Cut-Extrude feature? _____

_____

3 What does the **Convert Entities** sketch tool do?_____

_____

4 What does the **Offset Entities** sketch tool do? _____

_____

5 In an assembly, parts are referred to as _____

_____

6 True or False. A fixed component is free to move. _____

7 True or False. Mates are relationships that align and fit components together in an assembly. _____

8 How many components does an assembly contain?_____

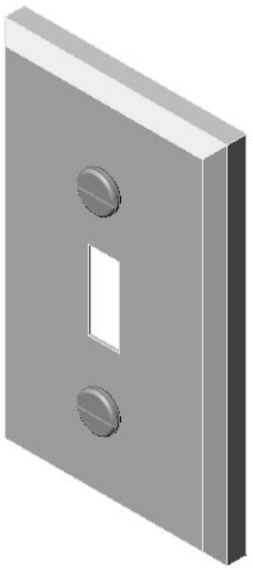
_____

9 What mates are required for the `Tutor` assembly?_____

_____

Exercises and Projects

Task 1



The `switchplate` created in Chapter 1 requires two fasteners to complete the assembly.

Question:

How do you determine the size of the holes in the `switchplate`?

Answer:

_____

_____

_____

_____

_____

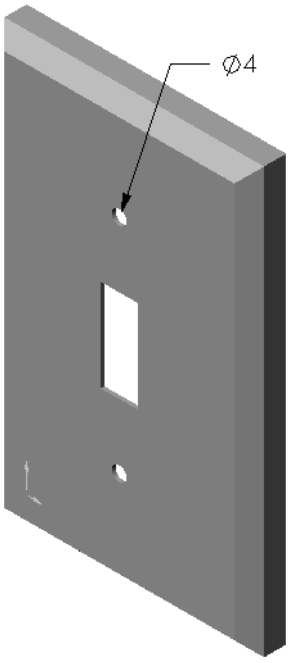
_____

Given:

- The diameter of the fastener is **3.5mm**.
- The switchplate is **10mm** deep.

Procedure:

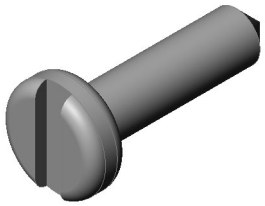
1. **Open** the `switchplate`.
2. Modify the diameter of the two holes to **4mm**.
3. **Save** the changes.



Exercises and Projects

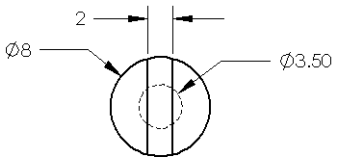
Task 2

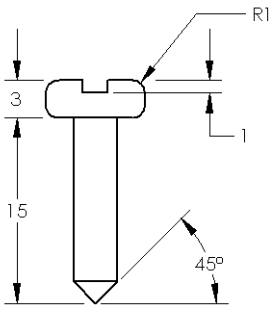
Design and model a fastener that is appropriate for the `switchplate`. Your fastener may (or may not) look like the one shown at the right.



Design Criteria:

- The fastener must be longer than the thickness of the switchplate.
- The `switchplate` is **10mm** thick.
- The fastener must be **3.5mm** in diameter.
- The head of the fastener must be larger than the hole in the `switchplate`.





Good Modeling Practice

Fasteners are always modeled in a simplified form. That is, although a real machine screw has threads on it, these are not included in the model.

Exercises and Projects

Task 3

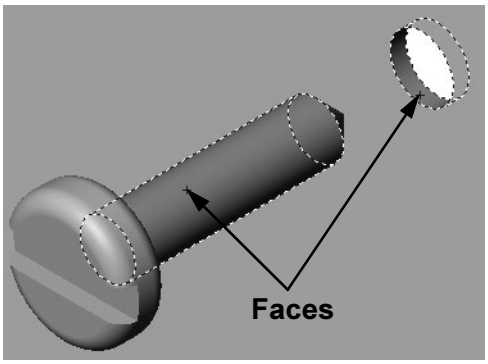
Create the `switchplate-fastener` assembly.

Procedure:

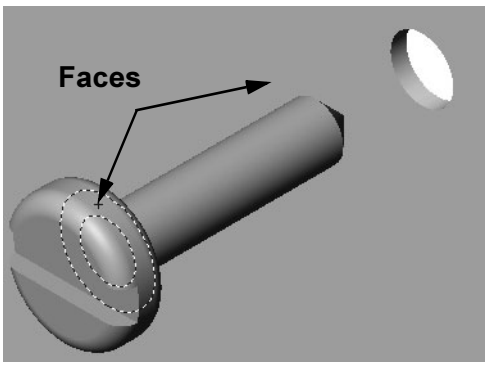
1. Create a new assembly.
   The fixed component is the `switchplate`.
2. Drag the `switchplate` into the assembly window.
3. Drag the `fastener` into the assembly window.
4. Use **Move Component** to position the `fastener` in front of the first hole.

The `switchplate-fastener` requires three mates to fully define the assembly.

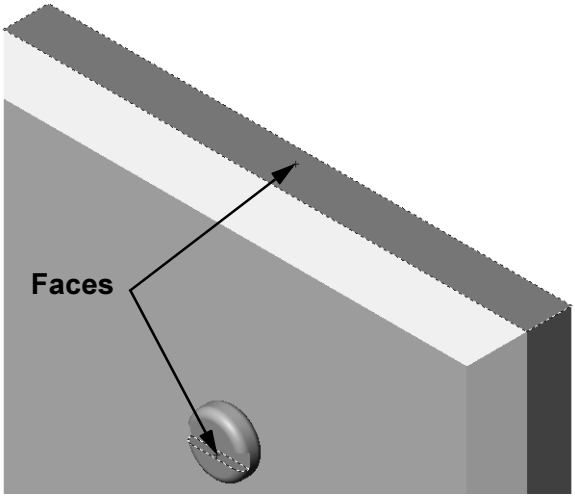
5. Create a **Concentric** mate between the cylindrical face of the `fastener` and the cylindrical face of the hole in the `switchplate`.



6. Create a **Coincident** mate between the back flat face of the `fastener` and the flat front face of the `switchplate`.



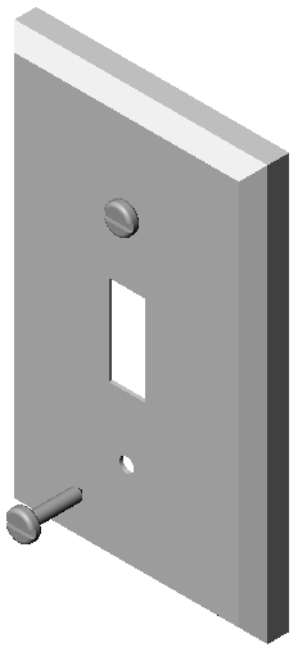
7. Create a **Parallel** mate between one of the flat faces on the slot of the `fastener` and the flat top face of the `switchplate`.
   Note: If the necessary faces do not exist in the `fastener` or the `switchplate`, create the parallel mate using the appropriate reference planes in each component.



8 Add a second instance of the `fastener` to the assembly.

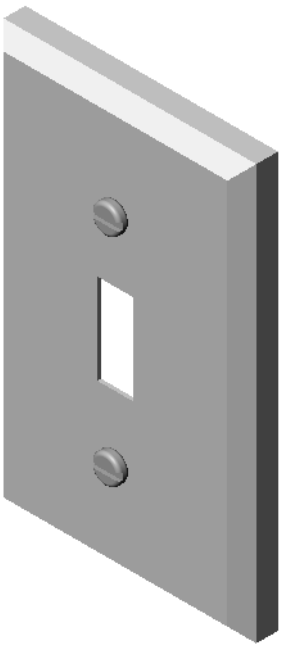
You can add components to an assembly by dragging and dropping:

- Hold the **Ctrl** key, and then drag the component either from the FeatureManager design tree, or from the graphics area.
- The pointer changes to .
- Drop the component in the graphics area by releasing the left mouse button and the **Ctrl** key.



9 Add three **mates** to fully define the second `fastener` to the `switchplate-fastener` assembly.

10 Save the `switchplate-fastener` assembly.



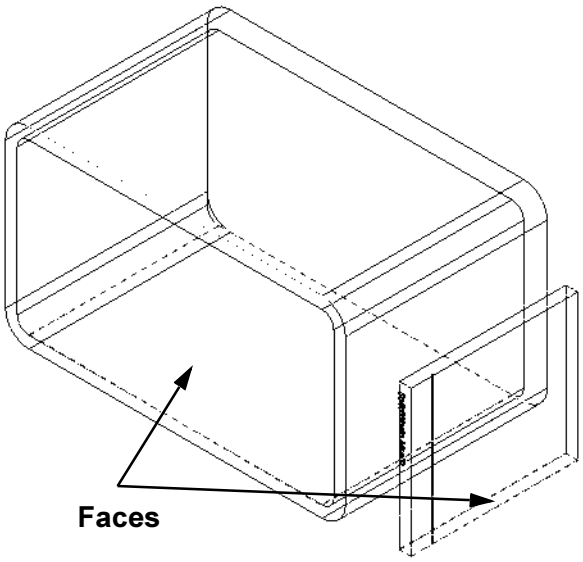
Exercises and Projects:

Task 4

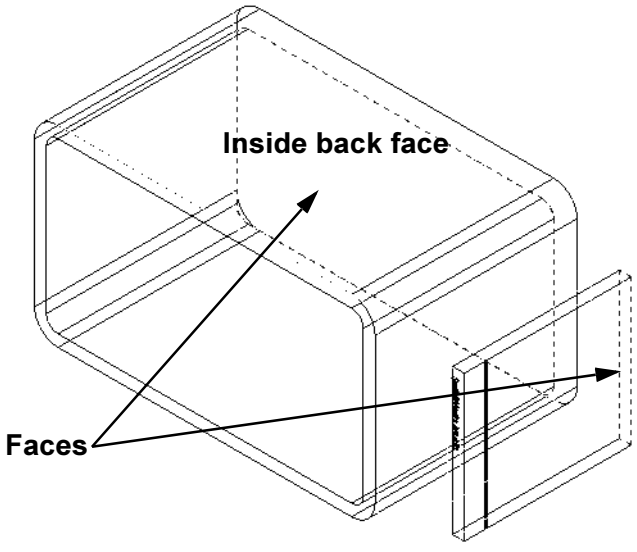
Assemble the `cdcase` and `storagebox` that you created in Chapter 2.

Procedure:

1. Create a new assembly.
   The fixed component is the `storagebox`.
2. Drag the `storagebox` into the assembly window.
   Locate the storagebox at the assembly origin using inferencing.
3. Drag the `cdcase` into the assembly window to the right of the `storagebox`.
4. Create a **Coincident** mate between the bottom face of the `cdcase` and the inside bottom face of the `storagebox`.



5. Create a **Coincident** mate between the back face of the `cdcase` and the inside back face of the `storagebox`.

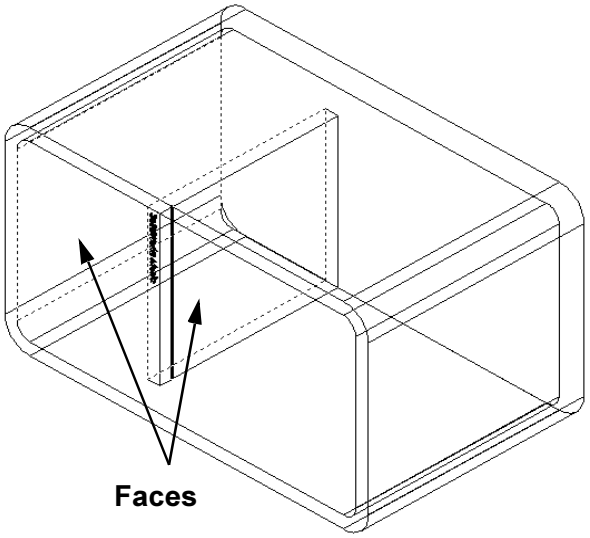


6 Create a **Distance** mate between the *left* face of the `cdcase` and the inside left face of the `storagebox`.

Enter **1cm** for **Distance**.

7 Save the assembly.

Enter `cdcase-storagebox` for the filename.



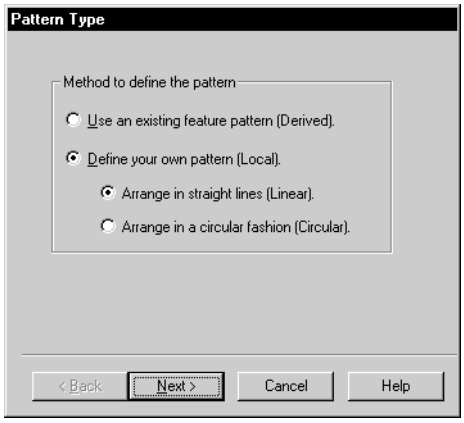
Component Patterns

Create a linear pattern of the `cdcase` component in the assembly.

The `cdcase` is the seed component. The seed component is what gets copied in the pattern.

8 Click **Insert**, **Component Pattern.**

The **Pattern Type** dialog is displayed.



9 Click **Define your own pattern (Local)**.

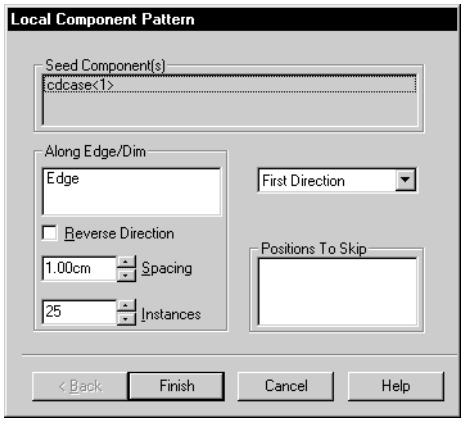
Make sure the option **Arrange in straight lines (Linear)** is selected.

10 Click **Next.**

The **Local Component Pattern** dialog box is displayed.

11 Select the component to be patterned.

Make sure the **Seed Component(s)** field is active, and then select the `cdcase` component from the FeatureManager design tree or the graphics area.



12 Define the direction for the pattern.

Click inside the **Along Edge/Dim** text box to make it active.

Click the top horizontal front edge of the `storagebox`.

13 Observe the direction arrow.

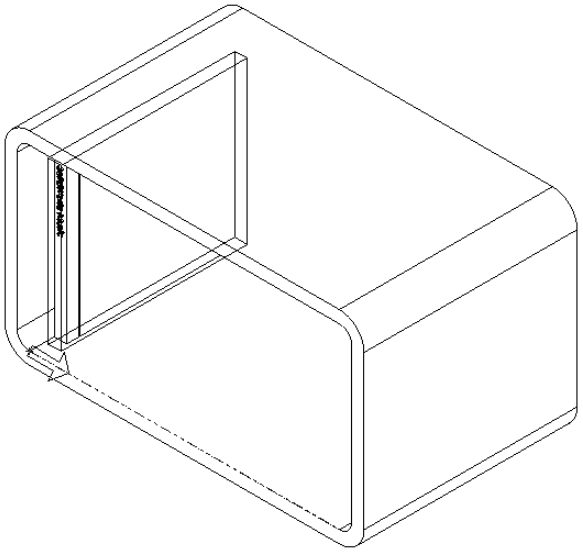
The preview arrow should point to the right. If it does not, click the **Reverse Direction** check box.

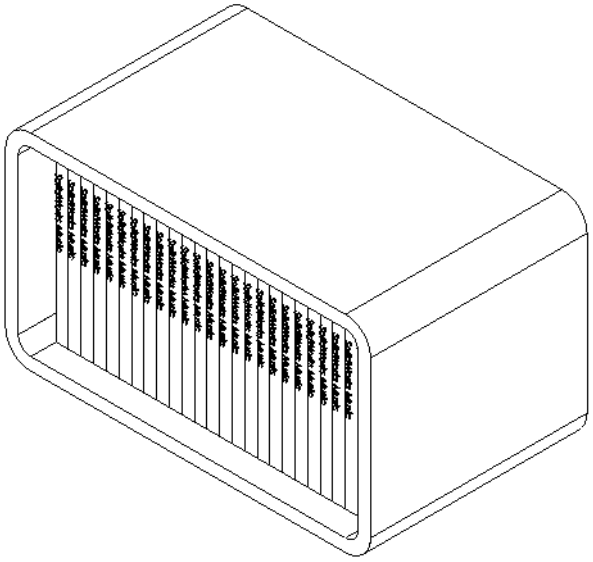
14 Enter **2** for **Spacing**. Enter **25** for **Instances**. Click **Finish**.

The Local Component Pattern Feature is added to the FeatureManager design tree.

15 Save the assembly.

Click **Save**. Use the name `cdcase-storagebox`.





Lesson 3 Vocabulary Worksheet

Name: _____Class: _____ Date:_____

Fill in the blanks with the words that are defined by the clues. Then find the words in the puzzle and circle them. The words may be vertical, horizontal, or diagonal. They may be spelled forward or backward.

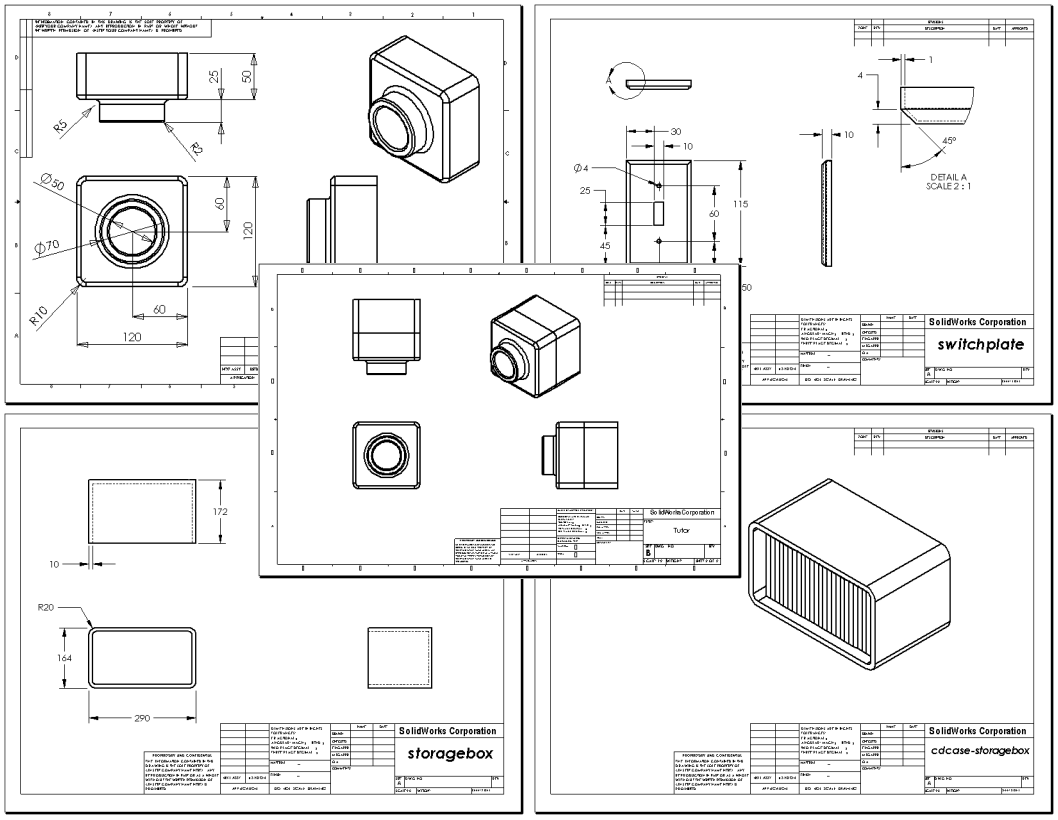
1. _____ Entities copies one or more curves into the active sketch by projecting them onto the sketch plane.
2. In an assembly, parts are referred to as: _____
3. Relationships that align and fit components together in an assembly:_____
4. The symbol (f) in the FeatureManager design tree indicates a component is:_____
5. The symbol (-) indicates a component is:_____
6. When you make a component pattern, the component you are copying is called the _____ component.
7. A SolidWorks document that contains two or more parts:_____
8. You cannot move or rotate a fixed component unless you _____ it first.

M	A	T	E	S	L	F	M	R	S	D	U
Z	G	C	T	Q	P	I	J	T	K	N	D
K	L	M	Y	R	R	X	N	R	D	F	D
B	V	L	L	J	T	E	G	E	Z	K	M
T	B	A	R	B	N	D	R	V	Q	L	F
G	A	T	S	O	G	D	N	N	K	H	K
W	N	O	P	S	E	L	Q	O	M	Q	K
J	Q	M	L	F	E	T	R	C	Y	F	T
J	O	P	I	F	D	M	W	W	L	N	K
C	R	N	Z	B	D	K	B	D	E	E	S
P	E	T	N	P	B	R	Q	L	N	T	P
D	B	M	V	Y	G	R	Z	T	Y	C	P

4

Lesson 4: Drawing Basics

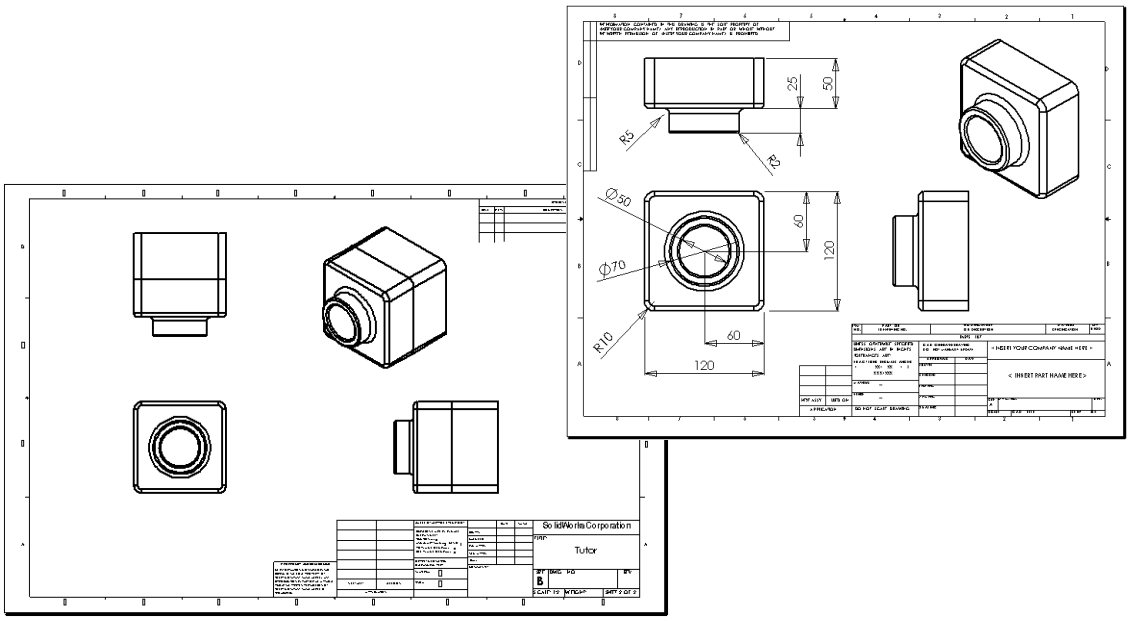
Upon successful completion of this lesson, you will be able to create detailed drawings of parts and assemblies:



This lesson corresponds to *SolidWorks 2001 Getting Started* pages 5-1 through 5-10.

Active Learning Exercises

Follow the instructions in *SolidWorks 2001 Getting Started* pages 5-1 through 5-10. In this lesson you will create two drawings. First, you will create the drawing for the part named `Tutor1` which you built in a previous lesson. Then you will create an assembly drawing of the `Tutor` assembly.



5 Minute Assessment

1 How do you open a drawing template? _____

_____

2 What is the difference between **Edit Sheet Format** and **Edit Sheet**?_____

_____

_____

_____

_____

3 A title block contains information about the part and/or assembly. Name five pieces of information that can be contained in a title block.

_____

_____

_____

_____

_____

4 True or False. Right-click **Edit Sheet Format** to modify title block information.

_____

5 What three views are inserted into a drawing when you click **Standard 3 View**?

_____

6 How do you move a drawing view? _____

_____

7 What command is used to import part dimensions into the drawing?

_____

8 True or False. Dimensions must be clearly positioned on the drawing._____

9 Give four rules for good dimensioning practice. _____

_____

_____

_____

_____

_____

_____

Exercises and Projects

Task 1

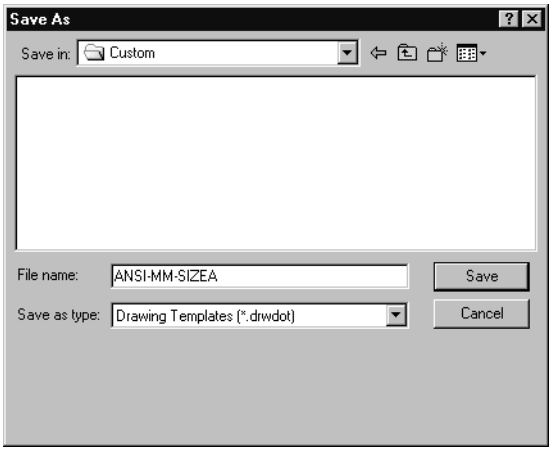
Create a new A-size ANSI standard drawing template.

Use millimeters for **Units**.

Name the template `ANSI-MM-SIZEA`.

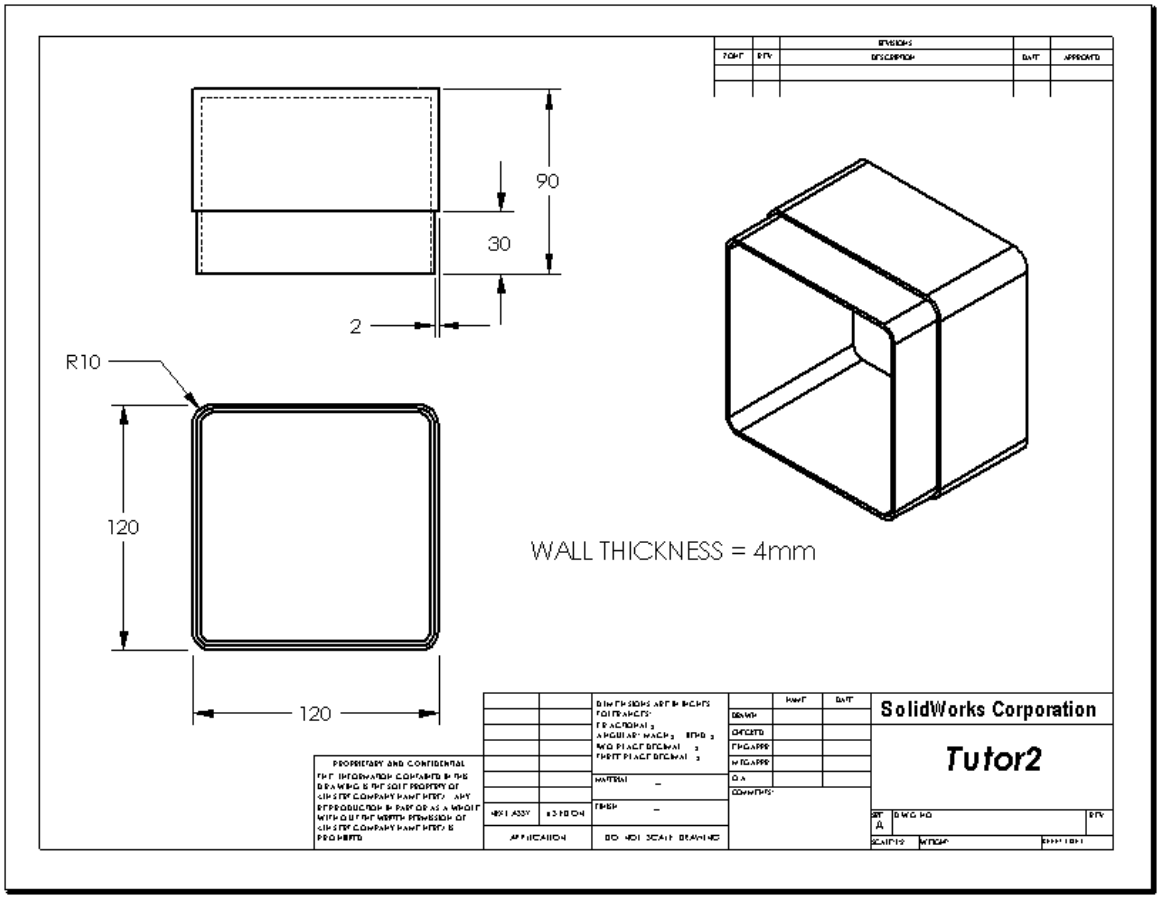
Procedure:

1. Create a new drawing using the Tutorial drawing template.
   This is an A-size sheet that uses the ISO dimensioning standard.
2. Click **Tools, Options** and then click the **Document Properties** tab.
3. Click **Detailing** and set the **Dimensioning standard** to **ANSI**.
4. Make any other desired changes to the document properties, such as the dimension text font and size.
5. Click **Units** and verify that the units are set to millimeters.
6. Click **OK** to apply the changes and close the dialog.
7. Click **File, Save As...**
8. From the **Save as type:** list, click **Drawing Template**.
   The system automatically jumps to the directory where the templates are installed.
9. Click  to create a new folder.
10. Name the new folder `Custom`.
11. Browse to the `Custom` folder.
12. Enter `ANSI-MM-SIZEA` for the name.
13. Click **Save**.
    Drawing templates have the suffix `*.drwdot`



Task 2

1. Create a drawing for `Tutor2`. Use the drawing template you created in Task 1. Review the guidelines for determining which views are necessary. Since `Tutor2` is square, the top and right views communicate the same information. Only two views are necessary to fully describe the shape of `Tutor2`.
2. Create Front and Top views. Add an Isometric view.
3. Import the dimensions from the part.
4. Create a note on the drawing to label the wall thickness.
   Right-click **Drawing Annotations, Note**. Enter **WALL THICKNESS = 4MM**.



More to Explore

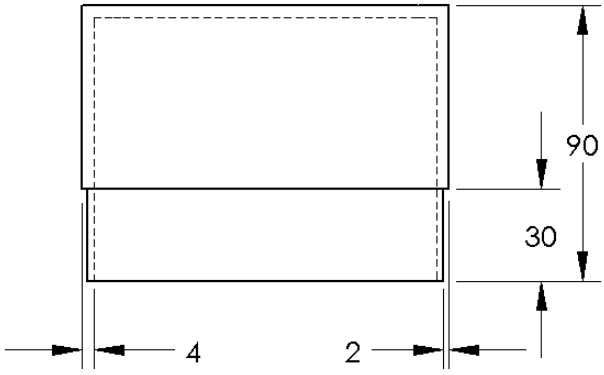
Investigate the on-line documentation to learn how to create a *parametric* note. In a parametric note, text, such as the numeric value of the wall thickness, is replaced with a dimension. This causes the note to update whenever the thickness of the shell is changed.

Once a dimension is linked to a parametric note, the dimension should *not* be deleted. That would break the link. However, the dimension can be hidden by right-clicking the dimension, and selecting **Hide** from the shortcut menu.

Procedure:

1 Import the model dimensions into the drawing.

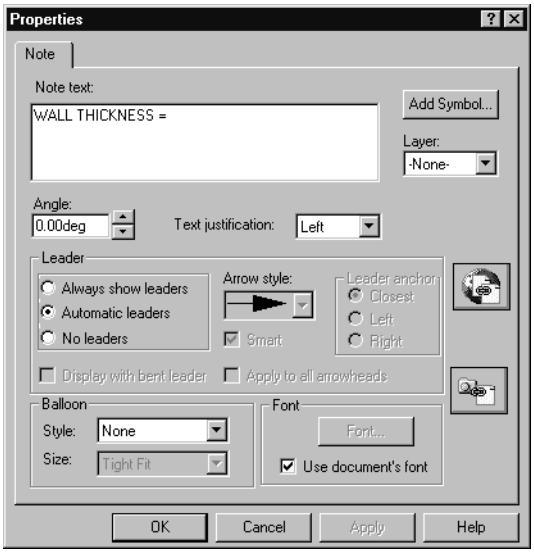
When you import the dimensions from the model, the 4mm thickness dimension of the Shell feature will also be imported. This dimension is needed for the parametric note.



2 Click A or **Insert, Annotations, Note**.

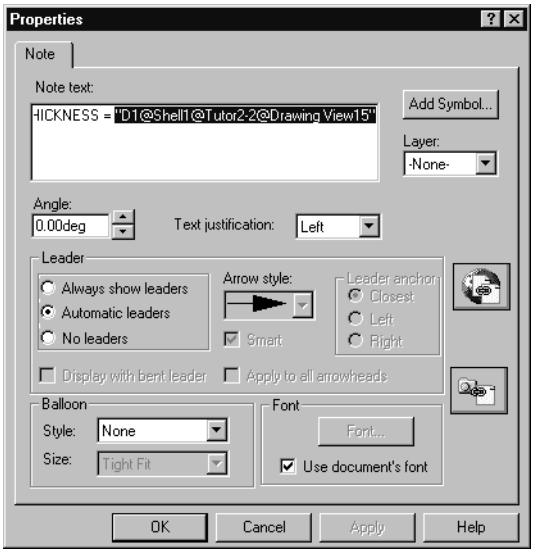
In the Properties dialog, enter the note text. For example: **WALL THICKNESS =**

Tip: To insert a note, you can also right-click in the graphics area, and select **Annotations, Note** from the shortcut menu.



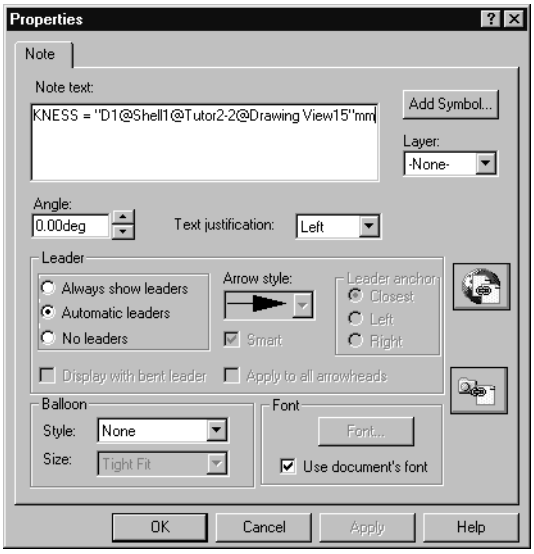
3 Click the dimension.

Instead of typing the value, click the dimension. The system will enter the name of the dimension into the text note.



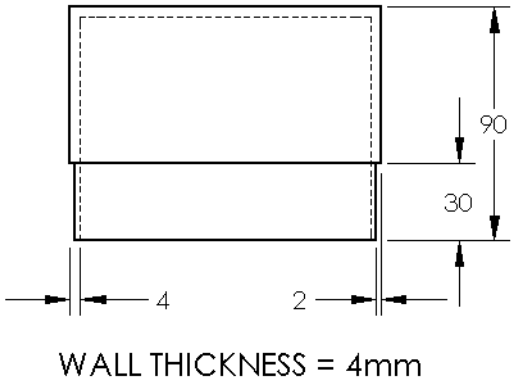
4 Type the rest of the note.

Use the arrow key to move the text insertion cursor to the end of the text string and type **mm**.



5 Click to place the note on the drawing.

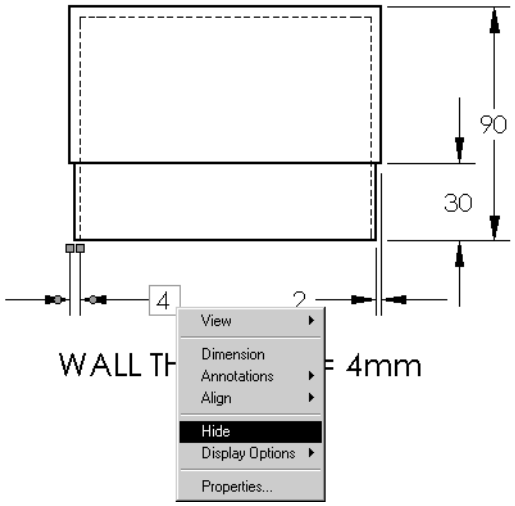
Position the note on the drawing. Then click **OK** to close the **Properties** dialog.



6 Hide the dimension.

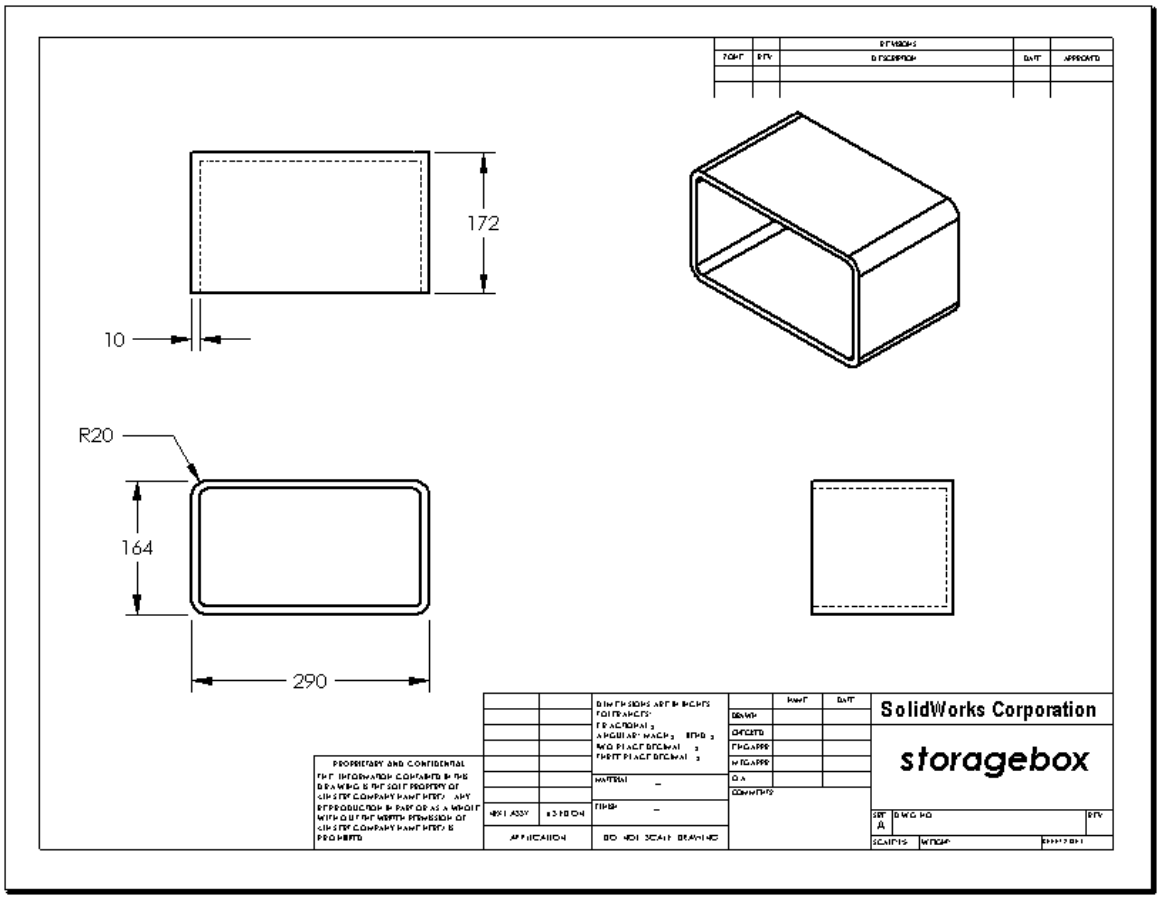
Right click the dimension, and select **Hide** from the shortcut menu.

You should *not* delete the dimension that was referenced in the parametric note. If you do, a change made to that dimension in the model will not propagate to the note. Instead you should hide the dimension.



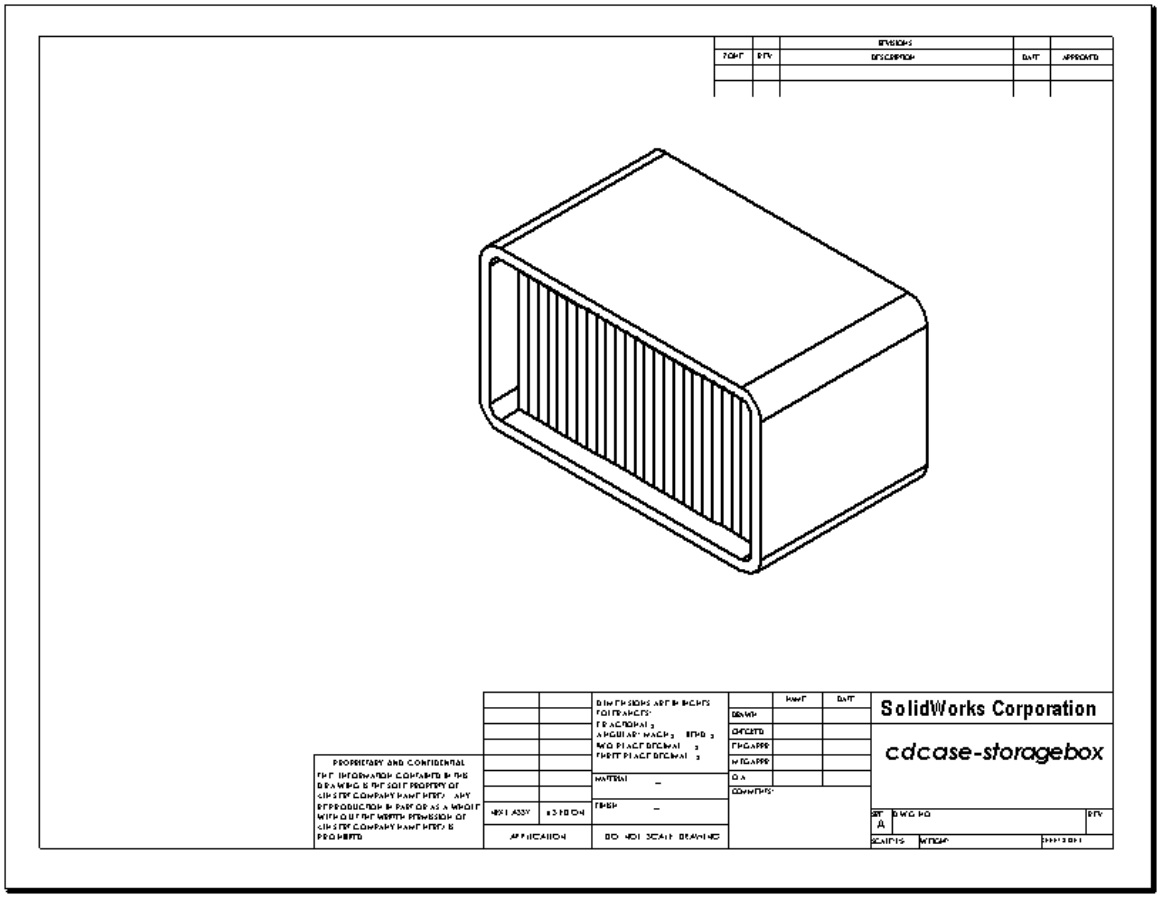
Task 3

1. Add a new sheet to the existing drawing you created in Task 2. Use the drawing template you created in Task 1.
2. Create a three standard views for the `storagebox`.
3. Import the dimensions from the model.
4. Create an Isometric view in a drawing for the `storagebox`.



Task 4

1. Add a new sheet to the existing drawing you created in Task 2. Use the drawing template you created in Task 1.
2. Create an Isometric view in a drawing for the `cdcase-storagebox` assembly.



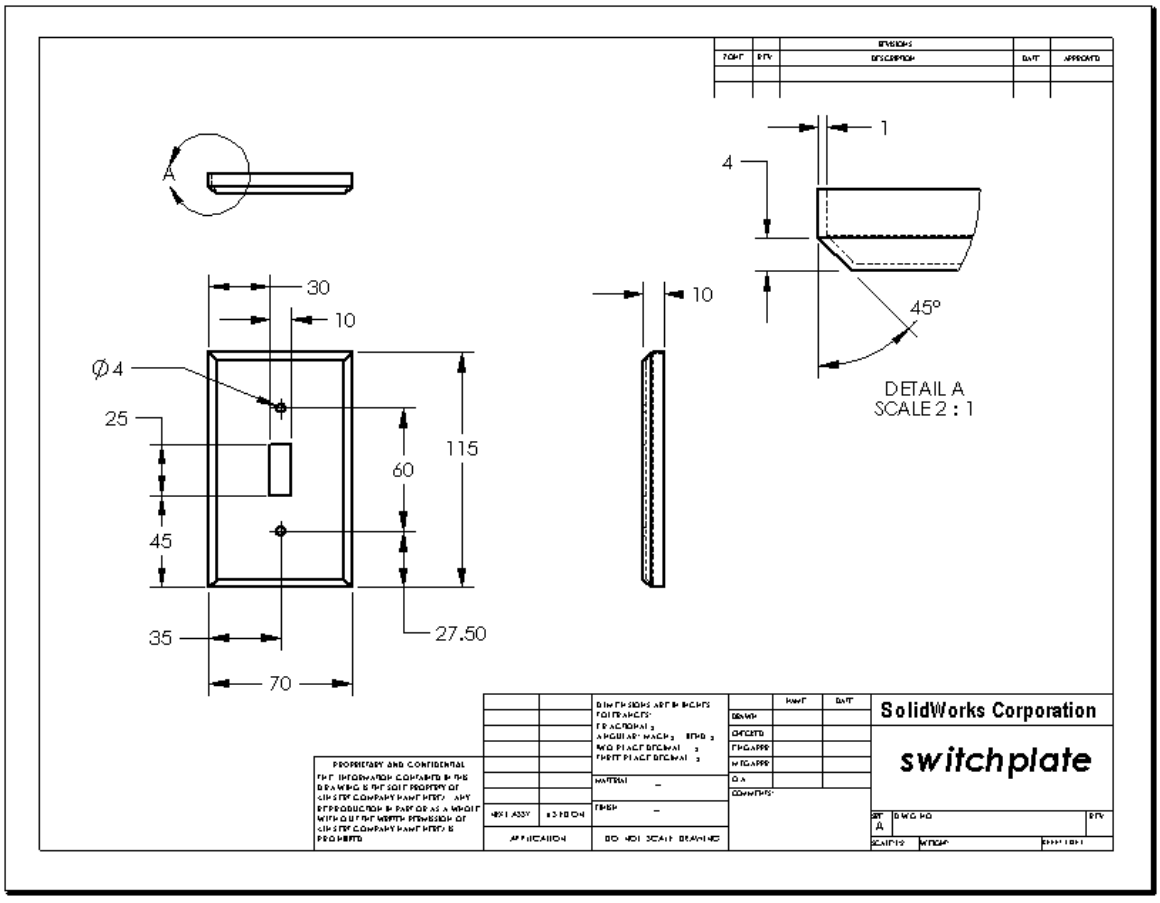
Task 5

1. Add a new sheet to the existing drawing you created in Task 2. Use the drawing template you created in Task 1.
2. Create a drawing of the `switchplate`.

The chamfer is too small to be clearly seen and dimensioned in either the Top or Right views. A detail view is required. Detail views are views that usually show only a portion of the model, at a larger scale. To make a detail view:

3. Select the view from which the detail view will be derived.
4. Click  and sketch a circle around the area you want to show.
5. While the circle is still selected (it is highlighted green), click  , or **Insert, Drawing View, Detail**.
6. Position the detail view on the drawing sheet.

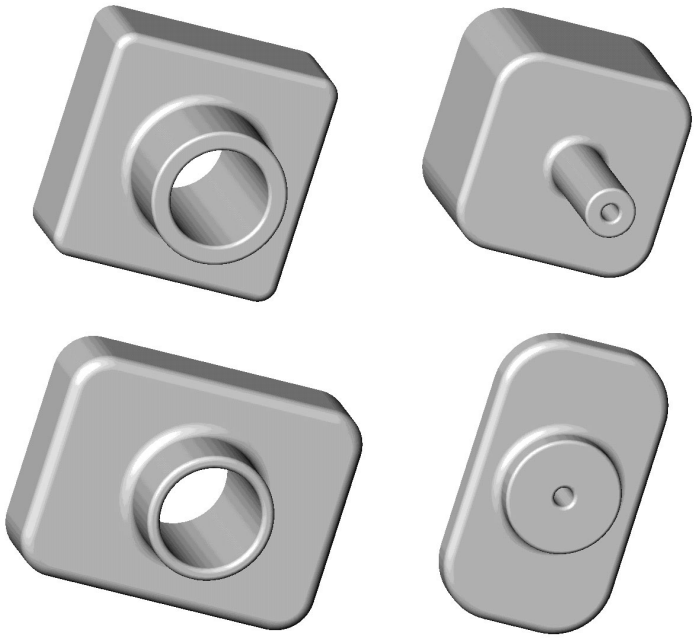
   The system automatically adds a label to the detail circle and the view itself. To change the scale of the detail view, edit the label's text.
7. You can import dimensions directly into a detail view, or drag them from other views.



5

Lesson 5: Design Tables

Upon successful completion of this lesson, you will be able to create a design table that will generate the following configurations of `Tutor1`:

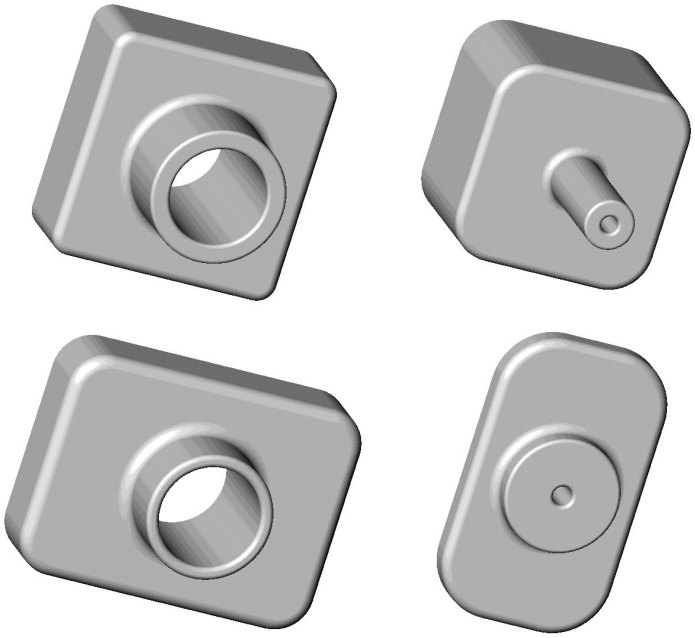


	A	B	C	D	E	F	G
1	Design Table for: Tutor3						
2		box_width@ Sketch1	box_height@ Sketch1	knob_dia@ Sketch2	hole_dia@ Sketch3	fillet_radius@ Outside_corners	Depth@ Knob
3	blk1	120	120	70	50	10	50
4	blk2	120	90	50	40	15	30
5	blk3	90	150	60	10	30	15
6	blk4	120	120	30	10	25	90

This lesson corresponds to *SolidWorks 2001 Getting Started* pages 6-1 through 6-10.

Active Learning Exercises

Create the design table for `Tuor1`. Follow the instructions in *SolidWorks 2001 Getting Started* pages 6-1 through 6-10.



	A	B	C	D	E	F	G
1	Design Table for: Tutor3						
2		box_width@ Sketch1	box_height@ Sketch1	knob_dia@ Sketch2	hole_dia@ Sketch3	fillet_radius@ Outside_corners	Depth@ Knob
3	blk1	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

5 Minute Assessment

1 What is a configuration?_____

_____

_____

2 What is a design table? _____

_____

_____

3 What additional Microsoft software application is required to create design tables in SolidWorks? _____

_____

4 What are three key elements of a design table? _____

_____

_____

_____

5 True of False. **Link Values** equates a dimension value to a shared variable name.

_____

6 Describe the advantage of using geometric relations versus linear dimensions to position the `Knob` feature on the `Box` feature. _____

_____

_____

_____

7 What is the advantage of creating a design table? _____

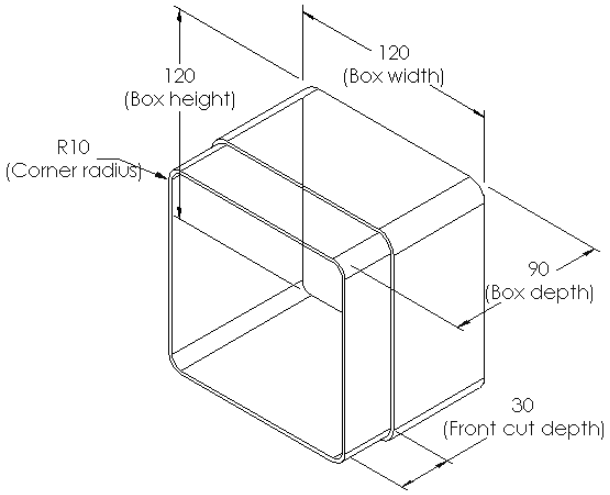
_____

_____

Exercises and Projects

Task 1

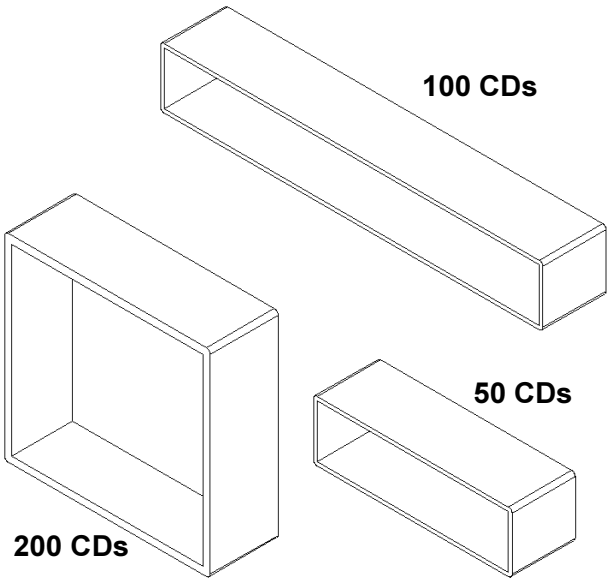
Create a design table for `Tutor2` that corresponds to the four configurations of `Tutor1`. Rename the feature names and the dimension names. Save the part as `Tutor4`.



Task 2

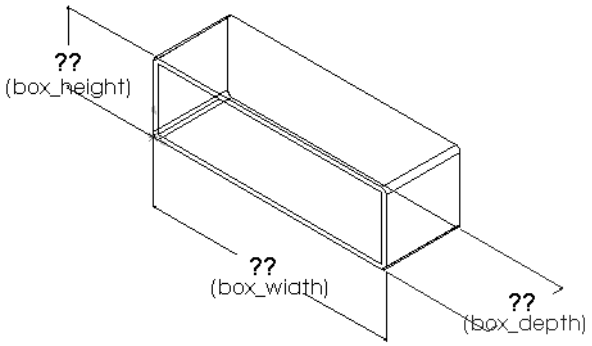
Create three configurations of the `CD storagebox` to contain 50, 100 and 200 CDs. The maximum width dimension is 120cm.

Some examples are shown at the right.



Task 3

Convert the overall dimensions of the 50 CD `storagebox` from centimeters to inches. The design for the CD `storagebox` was created overseas. The CD `storagebox` will be manufactured in the US.



Given:

- Conversion: 2.54cm = 1 inch
- `Box_width` = 54.0cm
- `Box_height` = 16.4cm
- `Box_depth` = 17.2cm

Answer:

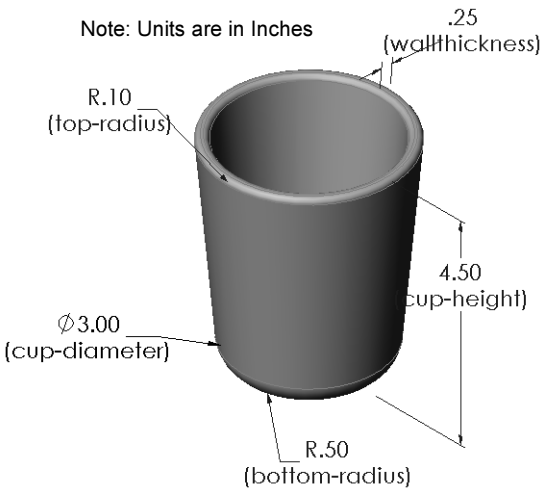
- Overall dimensions = `box_width` **x** `box_` height **x** `box_depth`
- `Box_width` = _____
- `Box_height` = _____
- `Box_depth` = _____

Task 4

What CD `storagebox` configurations are feasible for use in your classroom?

Task 5

Create a cup. In the **Extrude Feature** dialog box, use a **5° Draft Angle**. Create four configurations using a design table. Experiment with different dimensions.



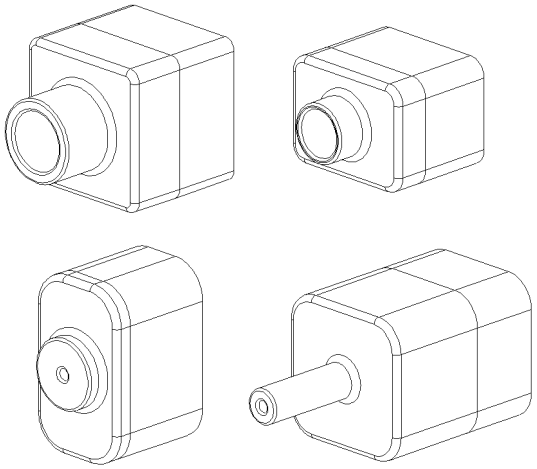
Task 6

Bring in some examples of products that lend themselves to design tables. You can bring in the actual objects or illustrations from magazines or catalogs.

More to Explore

Configurations, Assemblies, and Design Tables

When each component in an assembly has multiple configurations, it make sense that the assembly should have multiple configurations as well. There are two ways to accomplish this:

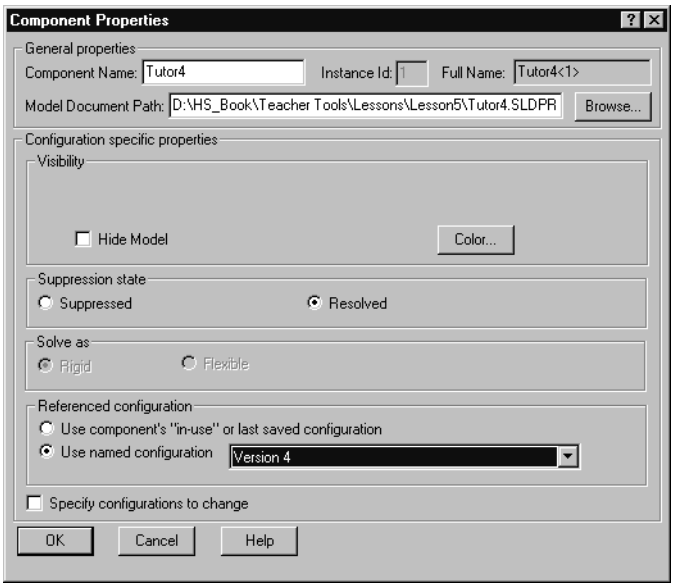


- Manually change the configuration being used by each component in the assembly.
- Create an *assembly* design table that specifies which configuration of each component is to be used for each version of the assembly.

Changing the Configuration of a Component in an Assembly

To manually change the displayed configuration of a component in an assembly:

1. Right-click the component, either in the FeatureManager design tree or in the graphics area, and select **Component Properties**.
2. In the **Component Properties** dialog, select the desired configuration from the list in the **Referenced configuration** area. Click **OK**.
3. Repeat this procedure for each component in the assembly.



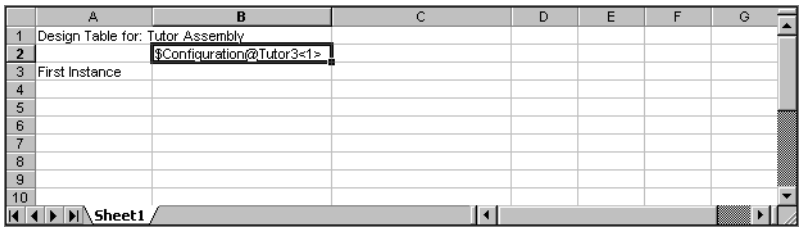
Assembly Design Tables

While manually changing the configuration of each component in an assembly works, it is neither efficient nor very flexible. Switching from one version of an assembly to another would be tedious. A better approach would be to create an assembly design table.

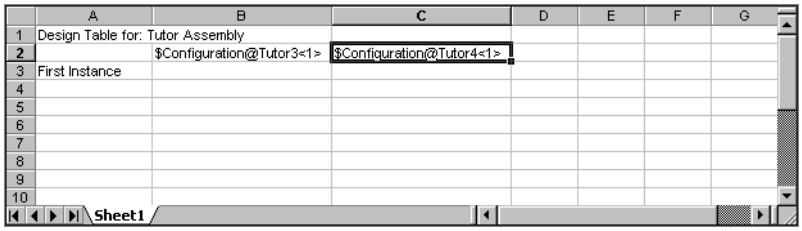
The procedure for creating an assembly design table is very similar to the procedure for creating a design table in an individual part. The most significant difference is the choice of different keywords for the column headers. The keyword we will explore here is `$CONFIGURATION@component<instance>`.

Procedure

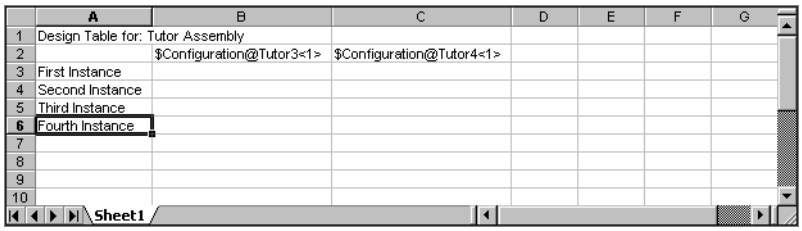
1. Click **Insert, New Design Table**.
2. In cell B2, enter the keyword `$Configuration@` followed by the name of the component and its instance number. In this example, the component is `Tutor3` and the instance is `<1>`.



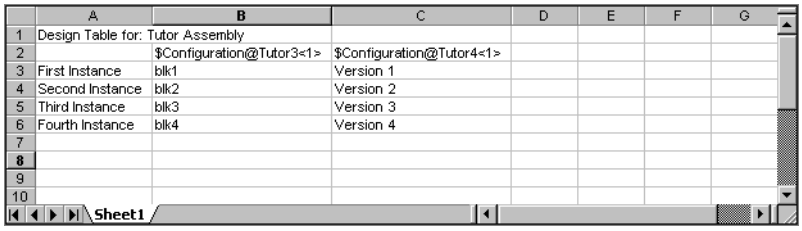
3. In cell C2, enter the keyword `$Configuration@Tutor4<1>`.



4. Add the configuration names in column A.



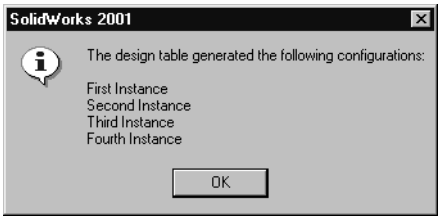
5. Fill in the cells of columns B and C with the appropriate configurations for the two components.



6 Finish inserting the design table.

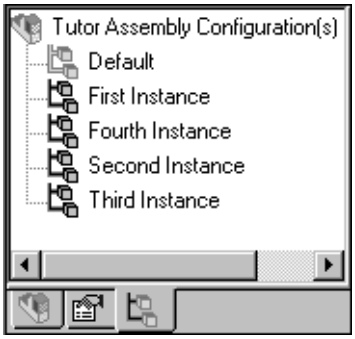
Click in the graphics area. The system reads the design table and generates the configurations.

Click **OK** to close the message dialog.



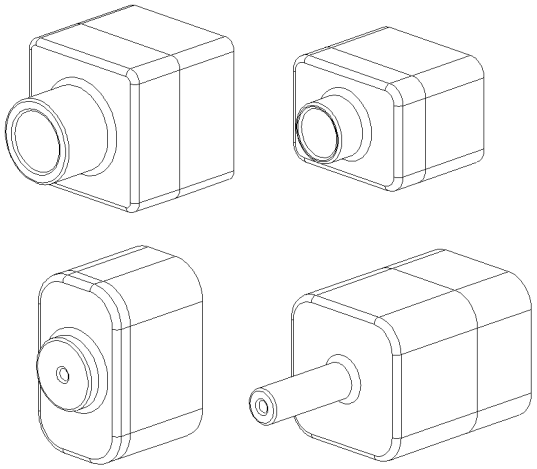
7 Switch to the ConfigurationManager.

Each of the configurations specified in the design table should be listed.



8 Test the configurations.

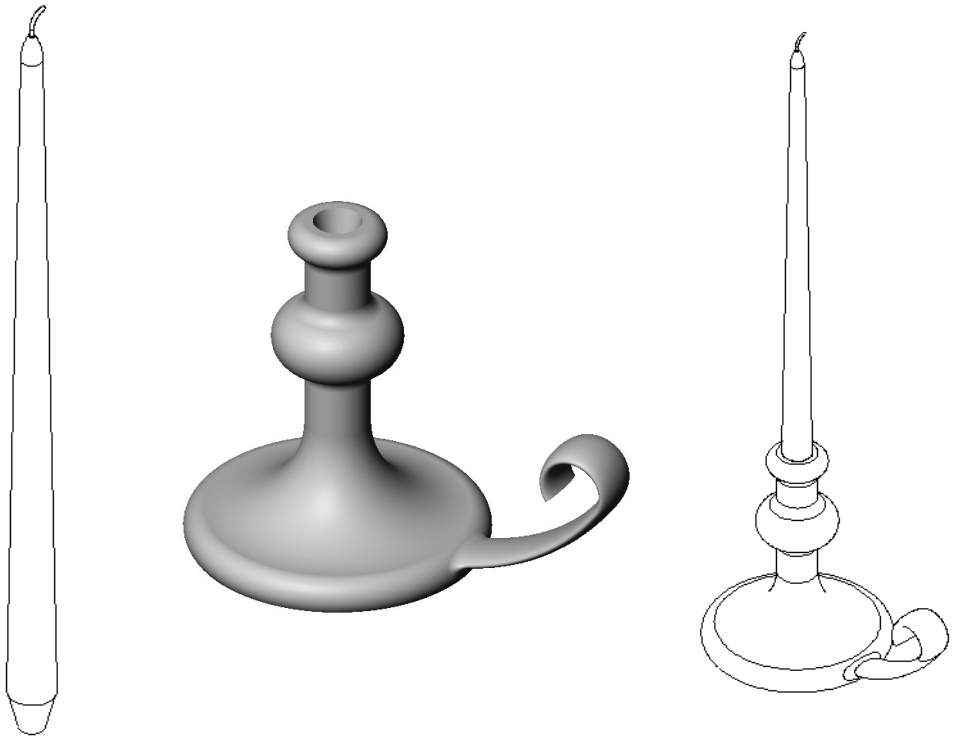
Double-click on each configuration to verify that they display correctly.



6

Lesson 6: Revolve and Sweep Features

Upon successful completion of this lesson, you will be able to create and modify the following parts and assembly:

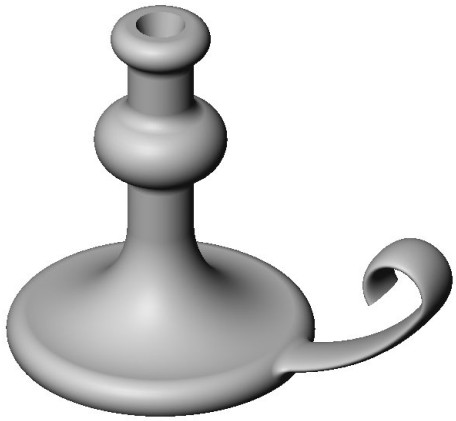


This lesson plan corresponds to *SolidWorks 2001 Getting Started* pages 7-1 through 7-10.

Active Learning Exercises

Create the candlestick. Follow the instructions in *SolidWorks 2001 Getting Started* pages 7-1 through 7-10.

The part name is `Cstick.sldprt`. However, throughout this lesson, we will refer to it as "candlestick" because that makes more sense.



5 Minute Assessment

1 What features did you use to create the candlestick?

_____

_____

_____

2 What special piece of sketch geometry is required for a revolve feature?

_____

_____

3 Unlike an extruded feature, a swept feature requires a minimum of two sketches. What are these two sketches?

_____

_____

4 What information does the pointer provide while sketching an arc?

_____

_____

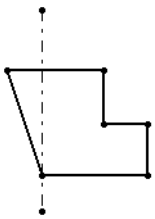
5 Examine the three illustrations at the right. Which one is not a valid sketch for a revolve feature?

_____

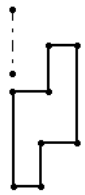
Why?_____

_____

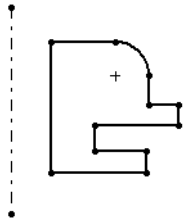
_____



A



B



C

_____

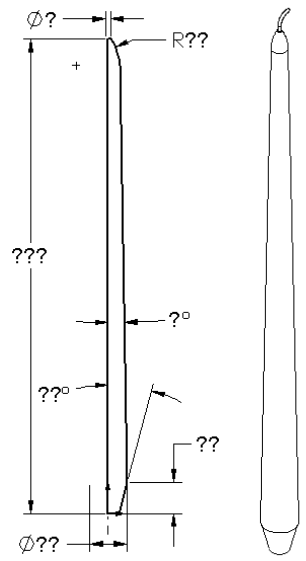
_____

Exercises and Projects

Task 1

Design a candle to fit the candlestick.

- Use a revolve feature as the base feature.
- Taper the bottom of the candle to fit into the candlestick.
- Use a sweep feature for the wick.



Question:

What other features could you use to create the candle? Use a sketch to illustrate your answer if necessary.

Answer:

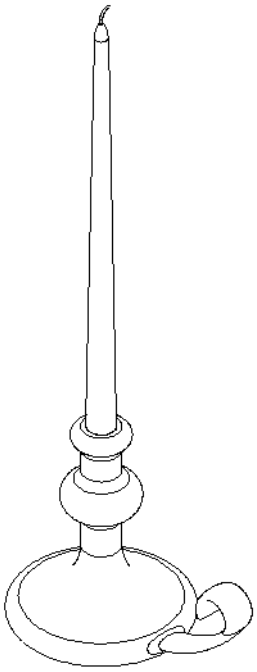
Question:

Would there be any benefit to using a design table to create the candle?

Answer:

Task 2

Create a candlestick assembly.



Task 3

You work for a candle manufacturer. Use a design table to create 15 inch, 12 inch, 10 inch and 7 inch candles.

	A	B
1	Design Table for: candle	
2		Length@Sketch1
3	15 inch candle	15
4	12 inch candle	12
5	10 inch candle	10
6	7 inch candle	7

Sheet1

Task 4

Design and model a mug. This is a rather open-ended assignment. You have an opportunity to express your creativity and ingenuity. The design of a mug can vary from the simple to the complex. A couple of examples are shown at the right.

Simple design

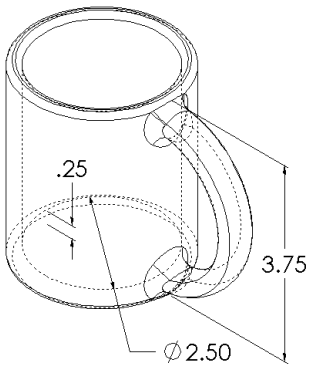
There are two specific requirements:

- Use a revolve feature for the body of the mug.
- Use a swept feature or the handle.

More complex design – a commuter's spill-proof travel mug

Task 5

How much coffee does the mug shown at the right hold?



Given:

- Inside Diameter = 2.50”
- Overall height of the mug = 3.75”
- Thickness of the bottom = 0.25”
- Coffee cups are not filled to the brim. Allow 0.5” space at the top.

Answer:

Conversion:

A cup of coffee in the US is sold by the fluid ounce, not by the cubic inch. How many ounces does the mug hold?

Given:

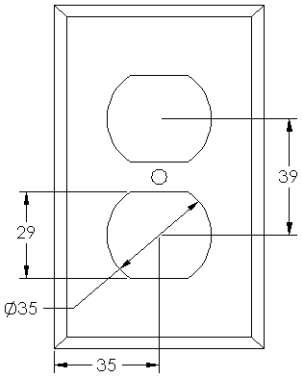
1 gallon = 231 in^3

128 ounces = 1 gallon

Answer:

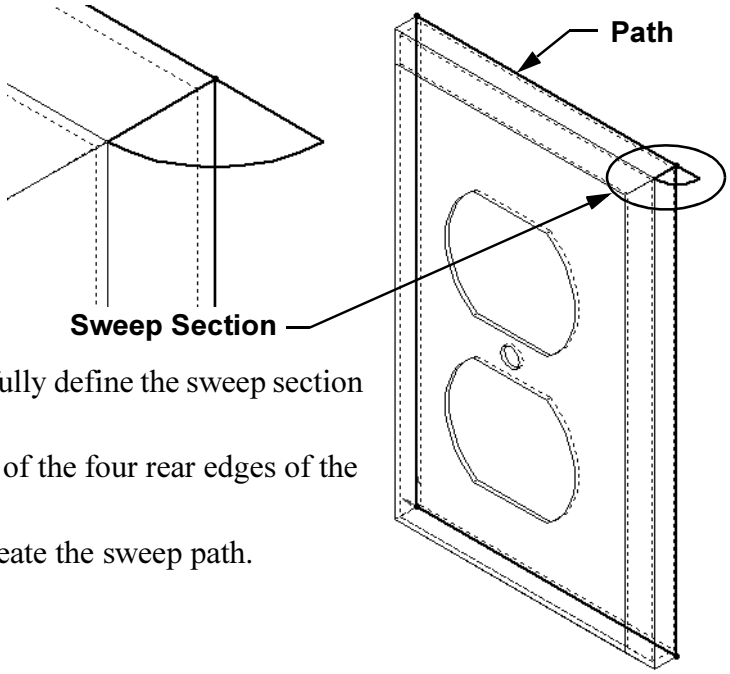
Task 6

Modify the `outletplate` that you created in Lesson 1.



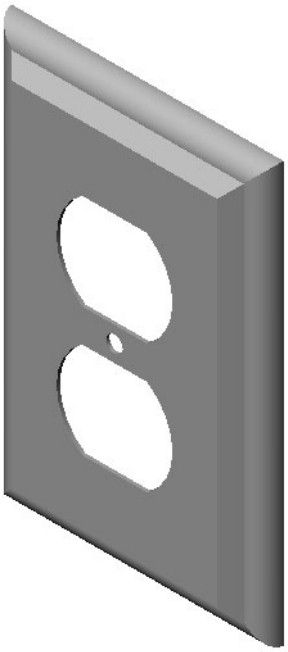
- Edit the sketch for the circular cuts that form the openings for the outlet. Create new cuts using the sketch tools. Apply what you have learned about **Link Values** and geometric relations to properly dimension and constrain the sketch.

- Add a swept boss feature to the back edge.
  - The sweep section is a 90° arc.
  - The radius of the arc is equal to the length of the model edge as shown in the accompanying illustration.



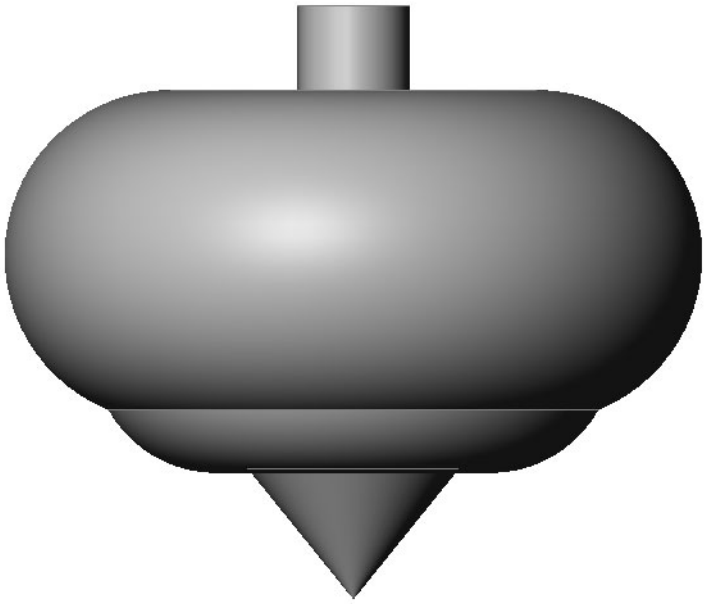
  - Use geometric relations to fully define the sweep section sketch.
  - The sweep path is made up of the four rear edges of the part.
  - Use **Convert Entities** to create the sweep path.

- The desired result is shown in the illustration at the right.



Task 7

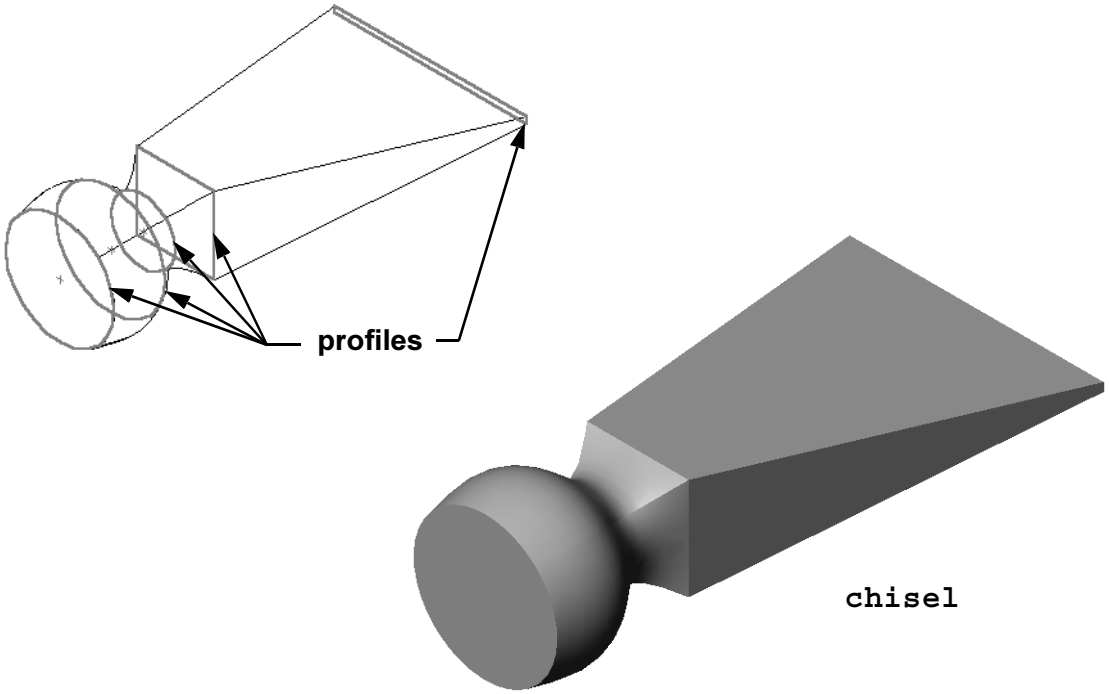
Use a revolve feature to create a toy top of your own design.



7

Lesson 7: Loft Features

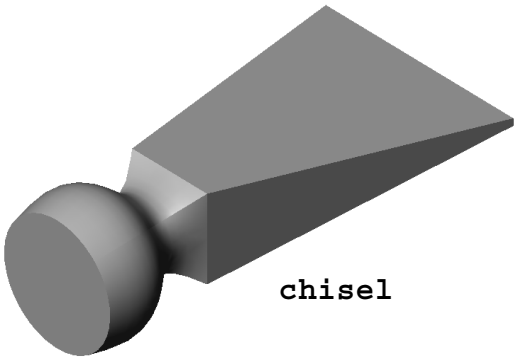
Upon successful completion of this lesson, you will be able to create the following part:



This lesson plan corresponds to *SolidWorks 2001 Getting Started* pages 8-1 through 8-6.

Active Learning Exercises

Create the `chisel`. Follow the instructions in *SolidWorks 2001 Getting Started* pages 8-1 through 8-6.



chisel

5 Minute Assessment – Answer Key

1 What features were used to create the `chisel`?

_____

_____

2 Describe the steps required to create the Base-Loft feature for the `chisel`.

_____

_____

_____

_____

_____

_____

_____

3 What is the minimum number of profiles required for a Loft feature?

_____

4 Describe the steps to copy a Sketch onto another plane.

_____

_____

_____

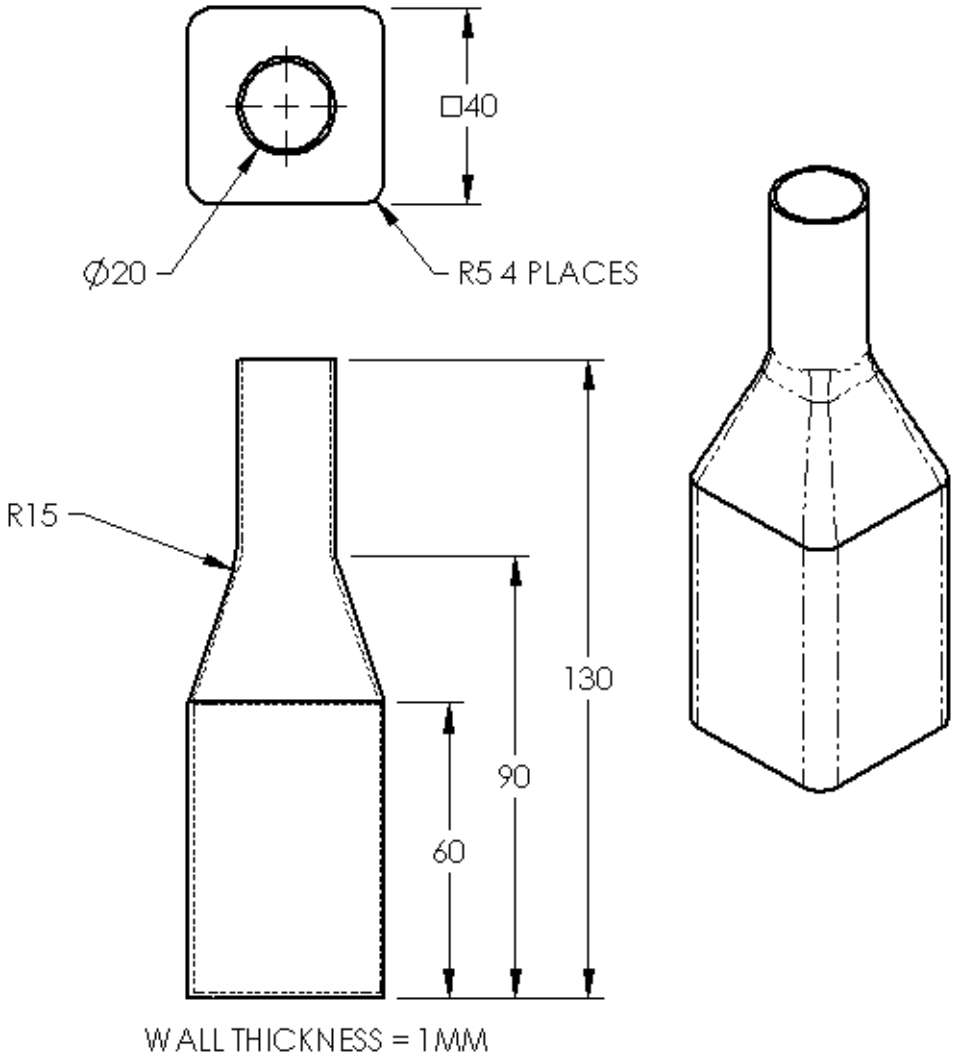
_____

_____

Exercises and Projects

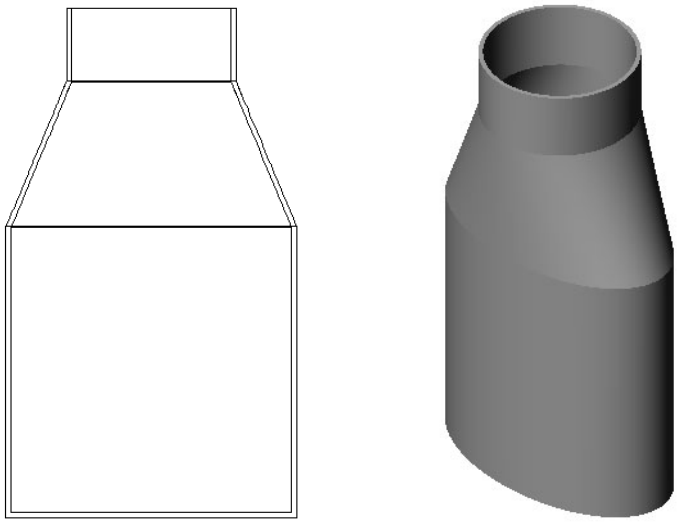
Task 1

Create the `bottle` as shown in the drawing. **Note:** all dimensions are in millimeters.



Task 2

Create `bottle2` with and elliptical Extruded-Base feature. The top of the bottle is circular. Design `bottle2` with your own dimensions.

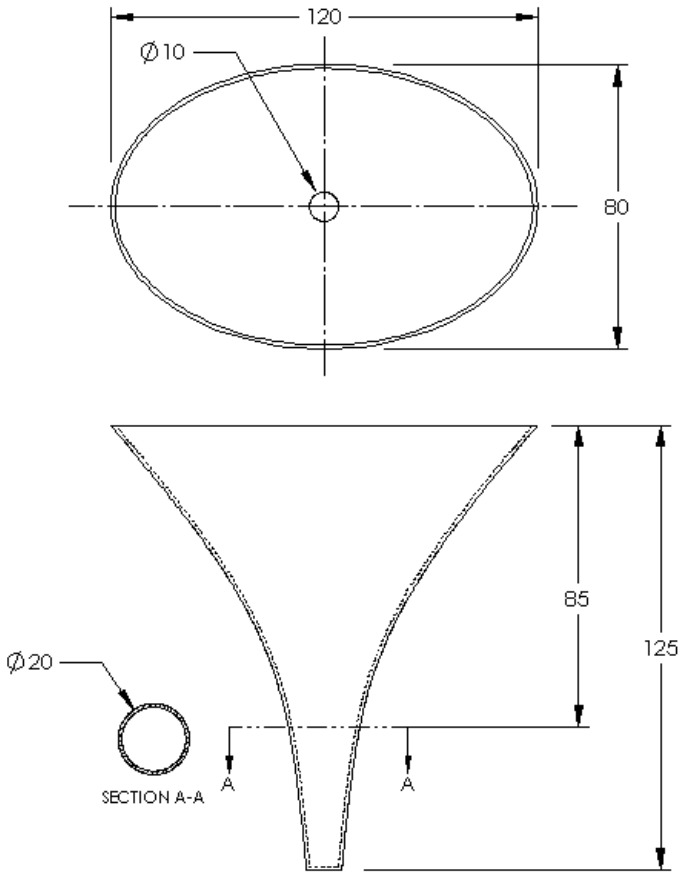


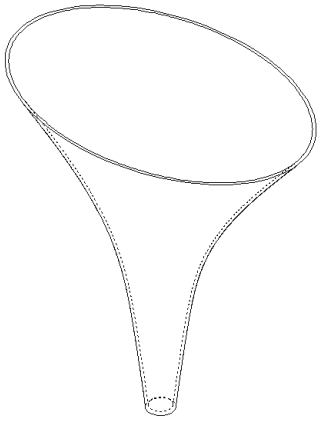
`bottle2`

Task 3

Create the `funnel` as shown in the drawing below.

- Use **1mm** for the wall thickness.

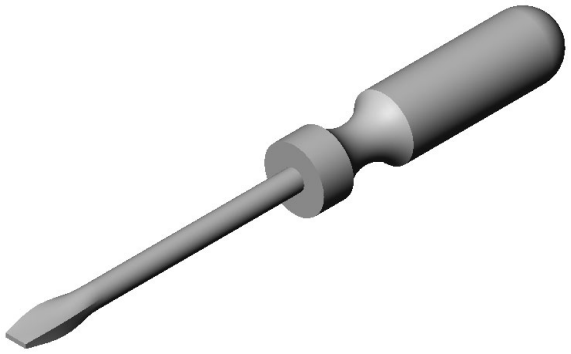




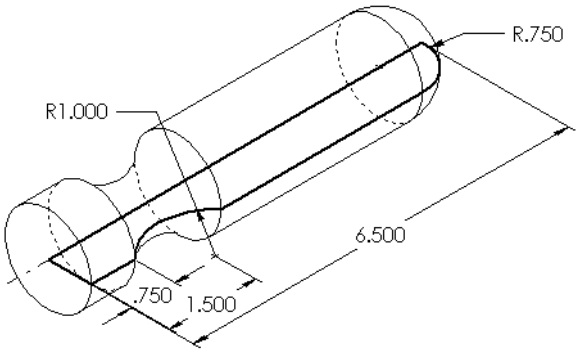
Task 4

Create the `screwdriver`.

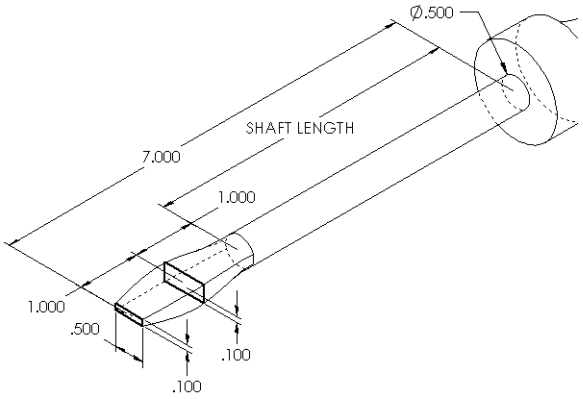
- Use **inches** for the database units.



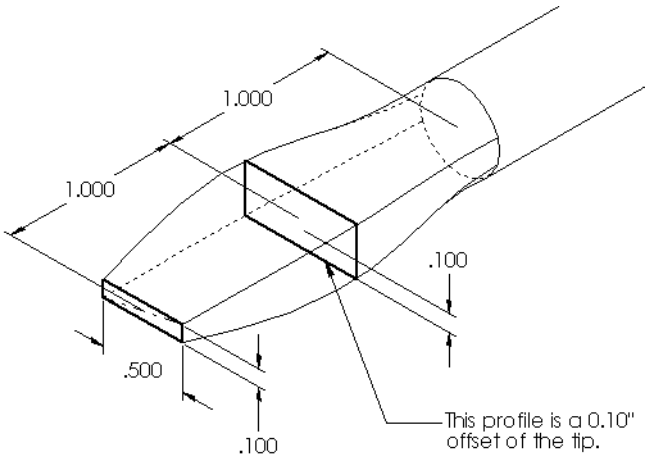
- Create the handle as the first feature. Use a revolved feature.



- Create the shaft as the second feature. Use an extruded feature.
- The overall length of the blade (shaft and tip together) is **7 inches**. The tip is **2 inches** long. Compute the length of the shaft.



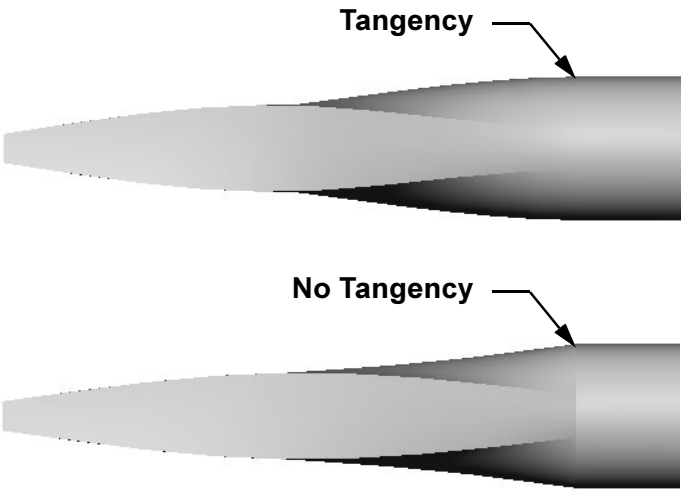
- Create the tip as the third feature. Use a loft feature.
- Create the sketch for the end of the tip first. This is a rectangle **0.50"** by **0.10"**.
- The middle, or second profile is sketched using a **0.10"** offset (to the outside) of the tip.
- The third profile is the circular face on the end of the shaft.



Matching Tangency

When you want to blend a loft feature into an existing feature such as the shaft, it is desirable to have the face blend smoothly.

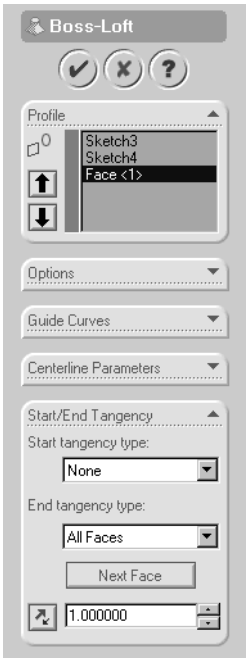
Look at the illustrations at the right. In the upper one, the tip was lofted with tangency matching to the shaft. The lower example was not.

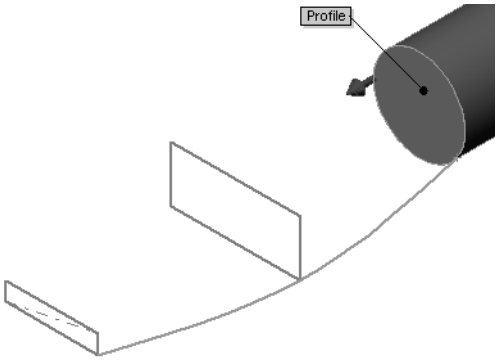


In the **Start/End Tangency** box of the PropertyManager, there are some tangency options. **End tangency** applies to the last profile, which in this case, is the face on the end of the shaft.

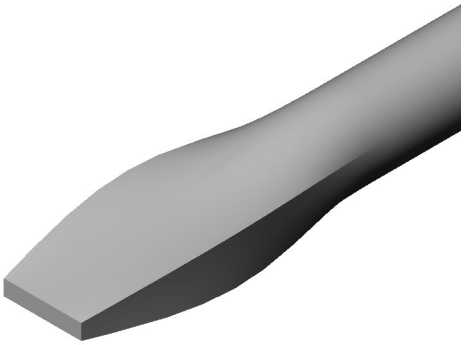
Note: If you picked the face of the shaft as the *first* profile, you would use the **Start tangency** option.

The option **All faces** will make the lofted feature tangent to the sides of the shaft.



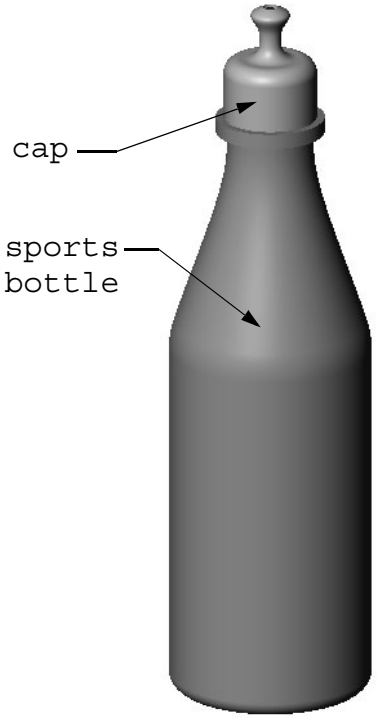


The result is shown at the right.



Task 5

- Design a 16 ounce `sportsbottle`.
- Create a `cap` for the `sportsbottle`.
- Create a `sportsbottle` assembly.



`sportsbottle` assembly

Question

How many liters are contained in the `sportsbottle`?

Conversion

- 1 fluid ounce = 29.57ml

Answer:

Task 6

A designer for your company receives the following cost information:

- Sports Drink = $0.32 per gallon based on 10,000 gallons
- 16 ounce sport bottle = $0.11 each based on 50,000 units

Question

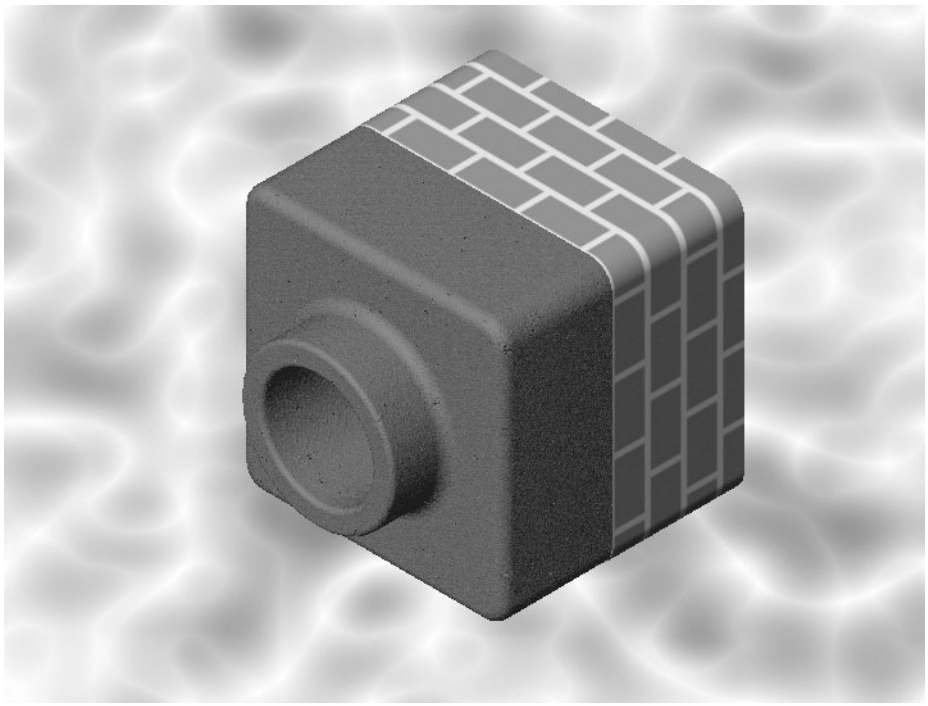
How much does it cost to produce a filled 16 oz. sportsbottle to the nearest cent?

Answer:

8

Lesson 8: Visualization

Upon successful completion of this lesson, you will create an image with PhotoWorks and an animation using SolidWorks Animator.



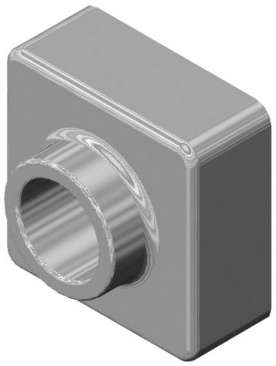
Note:

The material about the PhotoWorks and Animator applications presented in this lesson is very basic and introductory in nature. It barely scratches the surface of what these software applications can do. *SolidWorks 2001 Getting Started* contains two entire chapters about PhotoWorks – Chapter 19, and Animator – Chapter 20. If you have a keen interest in computer graphics, you might want to talk to your teacher about exploring Chapters 19 and 20 as independent enrichment activities.

Sample Files

Chapters 19 and 20 in the *SolidWorks 2001 Getting Started* tutorial reference sample files that are included on the SolidWorks software CD. The option to install the sample files must be selected when the SolidWorks software is installed.

Active Learning Exercises – PhotoWorks

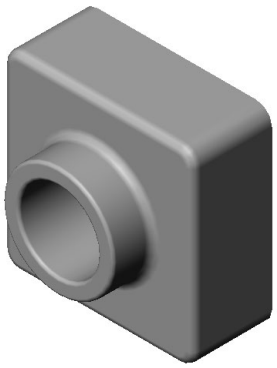


Follow the instructions in *SolidWorks 2001 Getting Started*, pages 19-1 through 19-28. Then create a PhotoWorks rendering of `Tutor1` which you built in a previous lesson.

- Apply **Chrome** material.
- Set the **Background Style** to **Graduated**.
- Save the `Tutor1.bmp` image.

The step-by-step instructions are as follows:

Getting Started



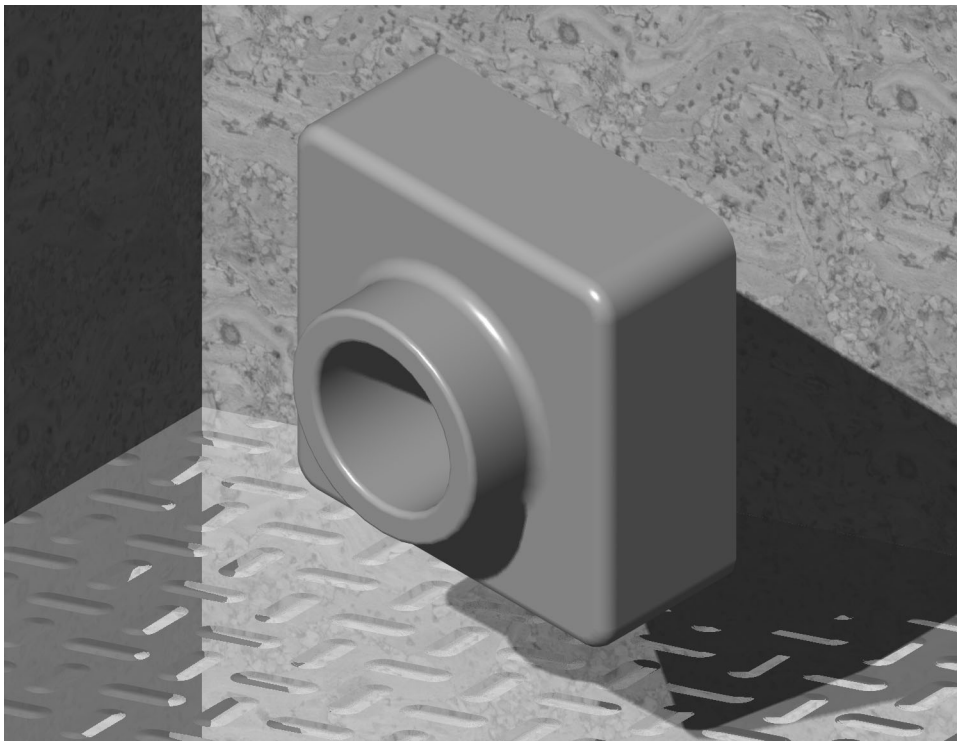
1. If **PhotoWorks** does not appear on the SolidWorks main menu bar, click **Tools, Add-Ins**, select **PhotoWorks**, and click **OK**.
2. Click **Open** on the Standard toolbar, and open the part `Tutor1` which you built in Lesson 2.
3. Set the view orientation to **Isometric** and select **Shaded** view mode from the View toolbar. Your part should look like the illustration at the right.

Shaded Rendering

Shaded rendering is the basis for all photo-realistic rendering in PhotoWorks.

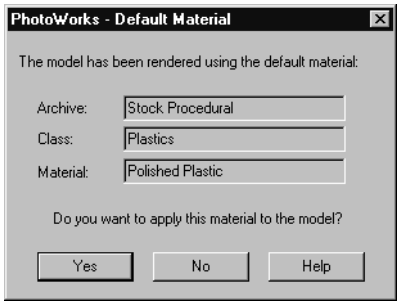
1. Click **Render** on the PhotoWorks toolbar.

   The PhotoWorks software produces a smooth-shaded rendering of the part using a default material and scene.



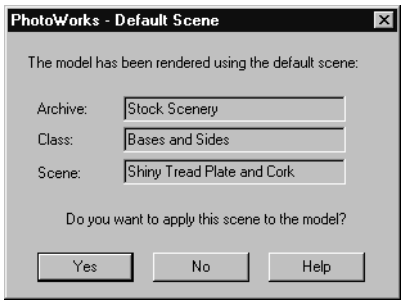
2 The **PhotoWorks - Default Material** dialog box is displayed indicating that the part has been rendered with the default material, **Polished Plastic**.

The PhotoWorks software asks whether you wish to apply this material to the model. Click **No**.



3 The **PhotoWorks - Default Scene** dialog box is displayed indicating that the part has been rendered with the default scene **Shiny Tread Plate and Cork**.

The PhotoWorks software asks whether you wish to apply this scene to the model. Click **No**.



Applying a Material

1 Click **Materials** on the PhotoWorks toolbar.

2 Double-click the **Stock Procedural** archive.

3 Click **Metals** class.

The material selection area displays a rendered image of a sphere for each material in the class.

4 Use the scroll bar to locate the **Chrome** material. Click the **Chrome** material.

The **Preview** window, to the right of the material editor, is updated to display how the part will appear when it is rendered.

Tip: You can select and apply a material in one operation by double-clicking the material in the material selection area.

5 Click **Apply**.

6 Click **Close**.

7 Click **Render**.

The part is rendered with a chrome surface.

Set the Background Style to Graduated.

1 Click **Scene** on the PhotoWorks toolbar.

The **PhotoWorks - Scene Editor** dialog box is displayed.

2 Click the **Background** tab.

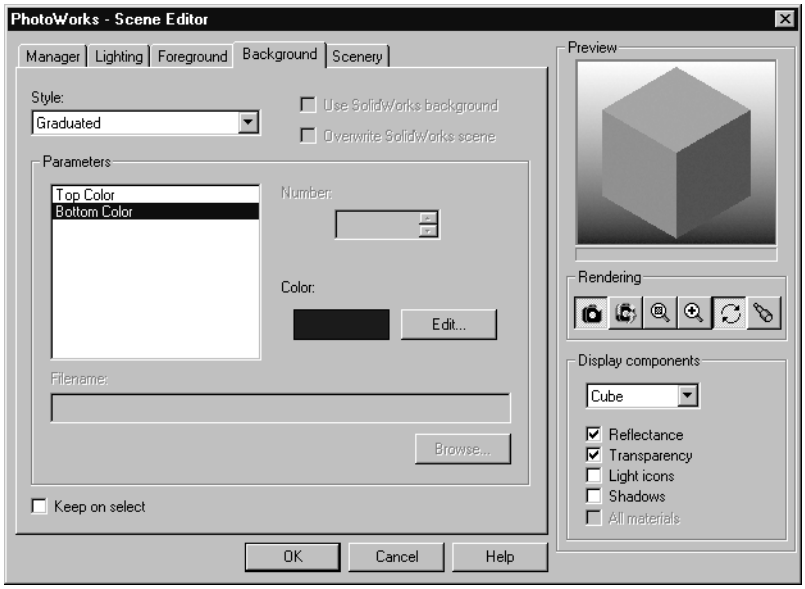
3 Click **Graduated** for **Style**.

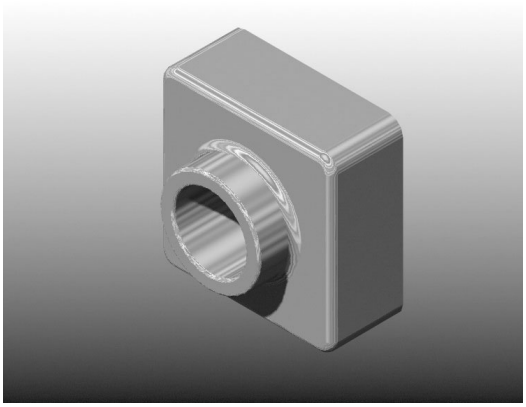
4 Under **Parameters**, click **Bottom Color**.

5 Click **Edit** and change the bottom color to **Blue**.

6 Click **OK**.

7 Click **Render**.





Saving the Image

You can save a PhotoWorks image to a file for design proposals, technical documentation and product presentations.

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`).

To Save the Image:

1 Click **Options**.

2 Click the **Image Output** tab.

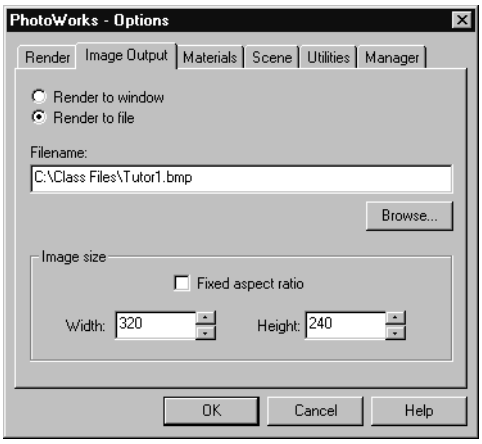
3 Click **Render to file**.

The PhotoWorks software provides a default image file name based on the name of the part. Save the file in the directory as instructed by your teacher.

4 Optionally, you may set the **Width**, and **Height**.

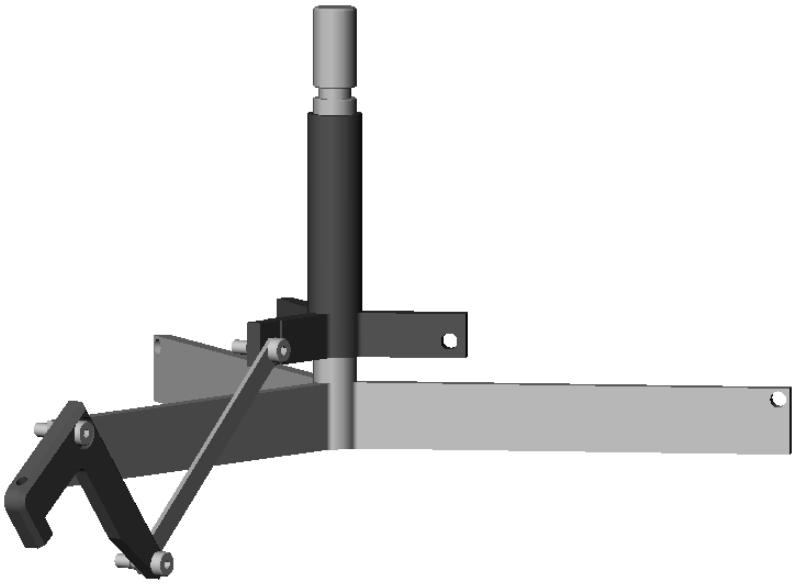
Note: If you change the **Image Size**, you should click **Fixed aspect ratio** to prevent distorting the image.

5 Click **OK**, and then click **Render**.



Active Learning Exercises – Animator

Create an animation of the `Claw-Mechanism` assembly. Follow the instructions in *SolidWorks 2001 Getting Started* pages 20-1 through 20-8.

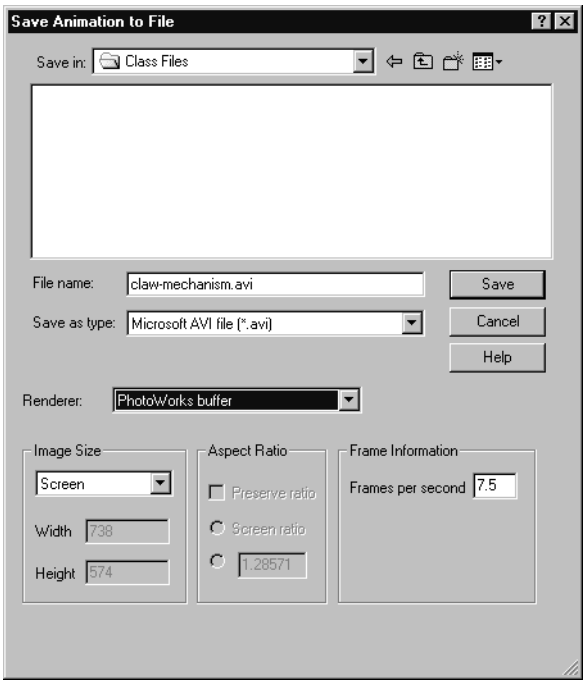


Using PhotoWorks and Animator Together

When you record an animation, the default rendering engine that is used is the SolidWorks shaded image software. This means the shaded images that make up the animation will look just like the shaded images you see in SolidWorks.

Earlier in this lesson you learned how to make photo-realistic images using the PhotoWorks application. You can record animations that are rendered using the PhotoWorks software. Since PhotoWorks rendering is much slower than SolidWorks shading, recording an animation this way takes much more time.

To use the PhotoWorks rendering software select **PhotoWorks buffer** from the **Renderer:** list on the **Save Animation to File** dialog box.

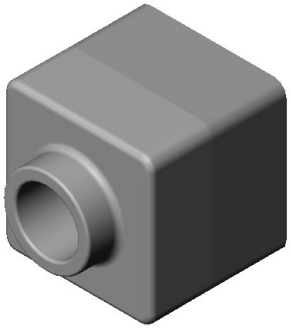


Note: The file types `*.bmp` and `*.avi` increase in file size as more materials and advanced rendering effects are applied. The larger the image size the more time is required to create the image and animation files.

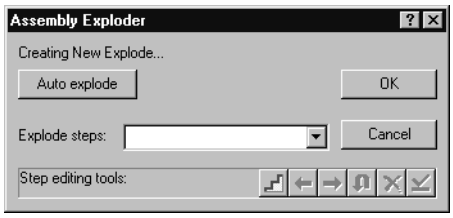
Creating an Exploded View of an Assembly

The `Claw-Mechanism` which you used in Chapter 20 of *Solidworks 2001 Getting Started* already had an exploded view. To add an exploded view to an assembly, the `Tutor` assembly for example, follow this procedure:

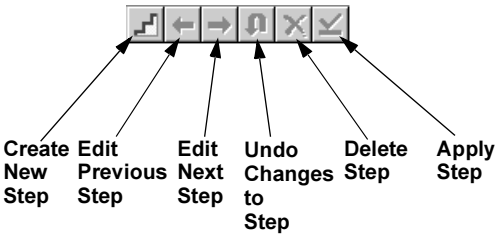
1 Click **Open** on the Standard toolbar, and open the assembly, `Tutor`, which you built in Lesson 3.



2 Click **Insert, Exploded View...**
The **Assembly Exploder** dialog box appears.

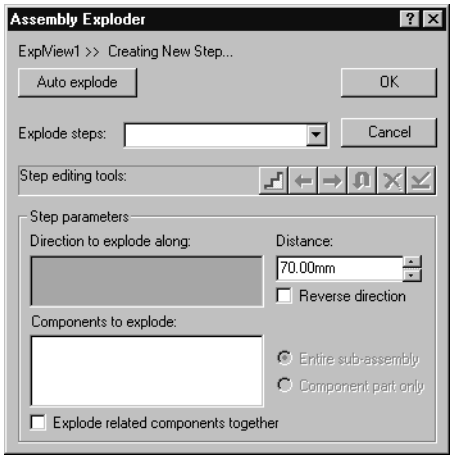


3 The Step Editing toolbar is used to create, edit, navigate through, delete, and apply explode steps. Each movement of a component in a single direction is considered a step.



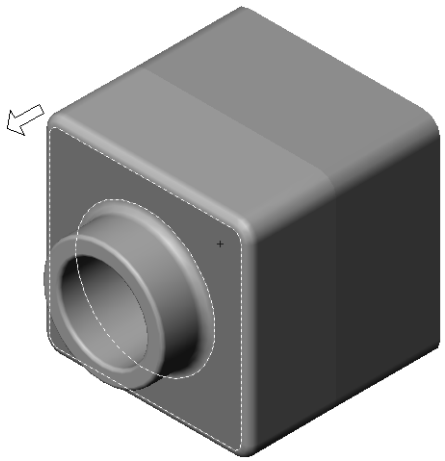
4 Click **New** on the Step Editing toolbar to begin a new explode step.
The dialog box expands to show selection lists for:

- **Direction to explode along**
- **Components to explode**
- **Distance**



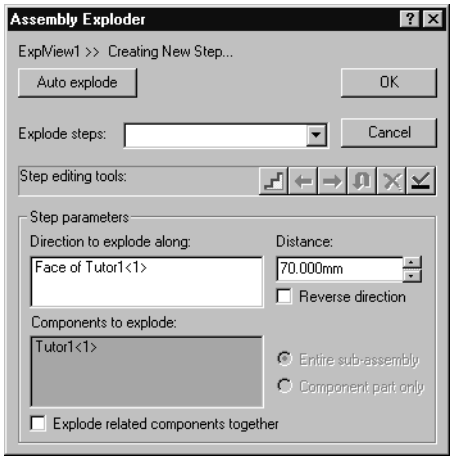
5 Click the flat face on the front of the `Tutor1` component.

An arrow appears that is perpendicular to the selected face and the name `Face of Tutor1<1>` appears in the **Direction to explode along** list.



6 Select the `Tutor1` component, either by clicking it in the FeatureManager design tree, or the graphics area.

The component name appears in the **Components to Explode** list.

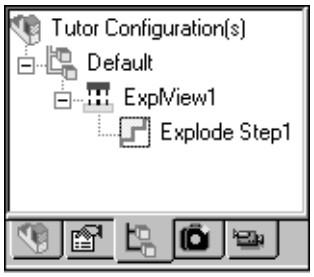


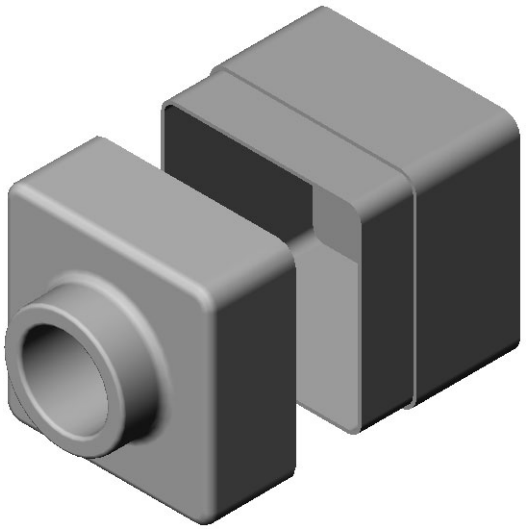
7 Set the **Distance** to **70mm** and click **Apply** on the Step Editing toolbar.

8 Since there is only one component to explode, this completes making the exploded view. Click **OK** to close the **Assembly Exploder** dialog box.

9 Results.

Note: Exploded views are related to and stored in configurations. You can only have one exploded view per configuration.





10 To collapse an exploded view, right-click in the FeatureManager design tree, and select **Collapse** from the shortcut menu.

11 To explode an existing exploded view, switch to the ConfigurationManager, and expand the configuration that contains the exploded view. Right-click the exploded view, and select **Explode** from the shortcut menu.

5 Minute Assessment

1 What is PhotoWorks?

_____

2 List the rendering effects that are used in PhotoWorks?

_____

3 The PhotoWorks_____ _____ allows you to specify and preview materials.

_____

4 Where do you set the scene background?

_____

5 What is SolidWorks Animator?

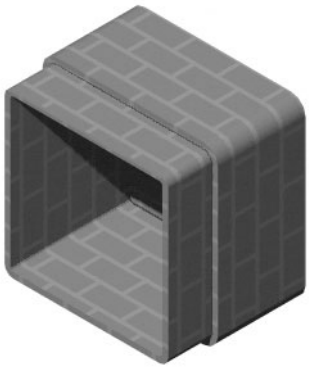
_____

Exercises and Projects

Task 1

Create a PhotoWorks rendering of `Tutor2`. Use the following settings:

- Use **Brick** material. **Pattern Scale** to **0.5**.
- Set **Background Style** to **None**.
- Save the image.

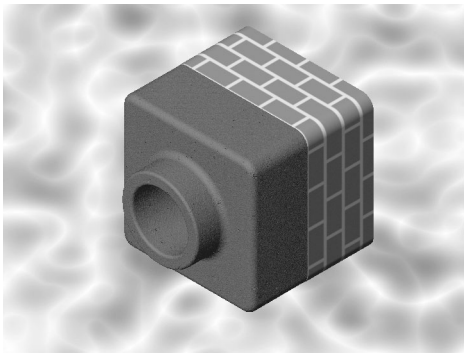


Task 2

Modify the PhotoWorks rendering of `Tutor1` that you created in the preceding Active Learning Exercise. Use the following settings:

- Change the material to **Concrete** from the **Stone** class.
- Change the **Background Style** to **None**.
- Save the image.

Task 3



Create a PhotoWorks rendering of the `Tutor` assembly. Use the following settings:

- Set the **Background Style** to **Clouds**.
- Set the **Scale** to **2**.
- Save the image.

Task 4

Create PhotoWorks renderings of any of the parts and assemblies you built during class. For example, you might render the candlestick you built is Lesson 6, or the sports bottle you made in Lesson 7. Experiment with different materials and scenes. You can try to create as realistic an image as possible, or you can create some unusual visual effects. Use your imagination. Be creative. Have fun.

Task 5

Create an animation using the `Tutor` assembly you built in Lesson 3. The animation should include the following:

- Explode the assembly for a duration of 10 seconds.
- Rotate the assembly around the Y axis for a duration of 10 seconds.
- Collapse the assembly for a duration of 10 seconds.
- Record the animation. **Optional:** Record the animation using the PhotoWorks renderer.