

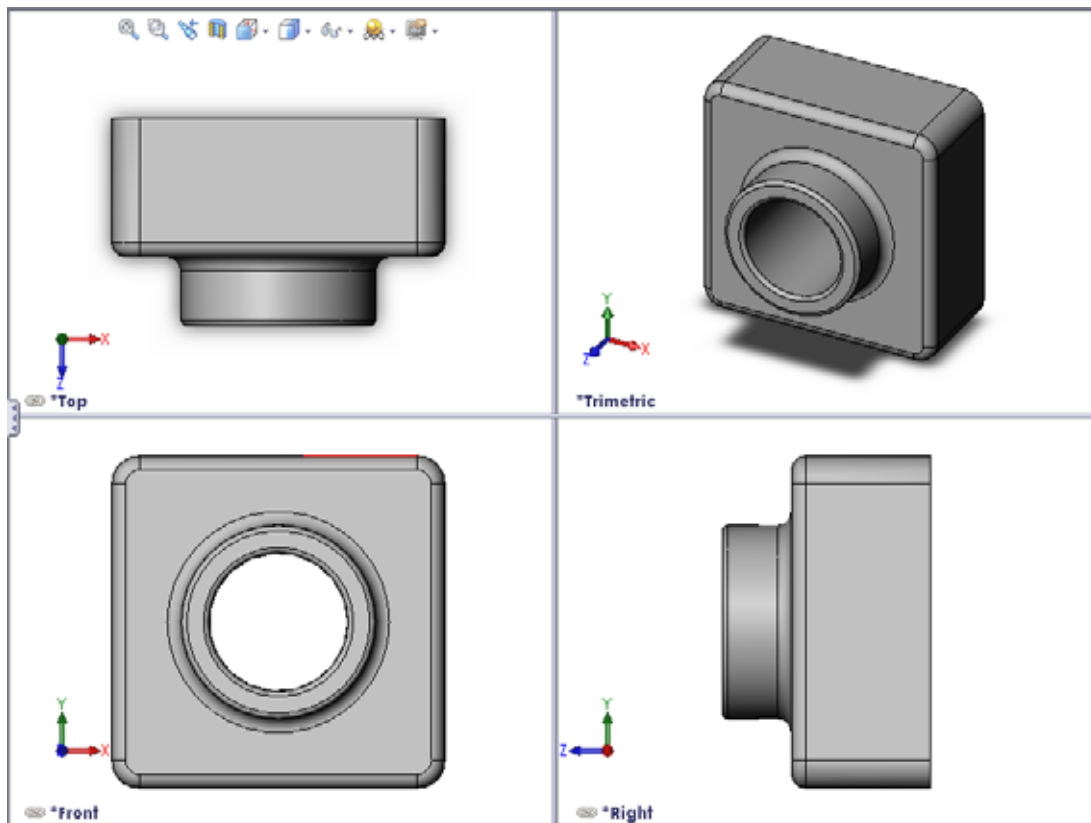
## Lesson 3: The 40-Minute Running Start

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### Goals of This Lesson

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Create and modify the following part:



### Before Beginning This Lesson

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Complete Lesson 2: Basic Functionality.

### Resources for This Lesson

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This lesson plan corresponds to *Getting Started: Lesson 1 – Parts* in the SolidWorks Tutorials. For more information, see “SolidWorks Tutorials” on page v.



SolidWorks education suite contains 80 tutorials in engineering design, sustainability, simulation and analysis.

## Review of Lesson 2: Basic Functionality

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### Questions for Discussion

- 1 A SolidWorks 3D model consists of three documents. Name the three documents.

**Answer:** Part, Assembly and Drawing.

- 2 Parts are built from features. What are features?

**Answer:** Features are the shapes (bosses, cuts and holes) and the operations (fillets, chamfers and shells) that you use to build a part.

- 3 Name the features that are used to create the box in Lesson 1.

**Answer:** Extruded Boss, Fillet, Shell, and Extruded Cut.

- 4 What is the base feature of the box?

**Answer:** The base feature is the first feature of the box. The base feature is the foundation of the part. The base feature geometry for the box is an extrusion. The extrusion is named *Extrude1*. The base feature represents the general shape of the box.

- 5 Why did you use the Fillet feature?

**Answer:** The fillet feature rounds the sharp edges and faces. The result of using the fillet feature created the rounded edges of the box.

- 6 Why did you use the Shell feature?

**Answer:** The shell feature removes material. The result of using the shell feature created a hollow block from a solid block.

- 7 How do you create the Base feature?

**Answer:** To create a solid Base feature:

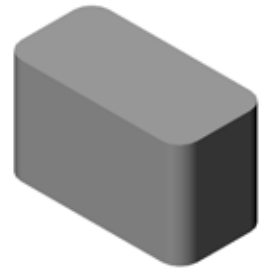
- Sketch a rectangular profile on a flat 2D plane.
- Extrude the profile perpendicular to the sketch plane.

- 8 What would have happened if the Shell feature was created before the Fillet feature?

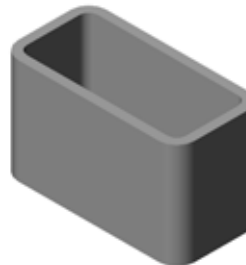
**Answer:** The inside corners of the box would be sharp instead of rounded.



1. Base Feature



2. Fillet Feature



3. Shell Feature



4. Cut Feature



## Outline of Lesson 3

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- ❑ In Class Discussion — Base Features
- ❑ Active Learning Exercise — Create a Part
- ❑ Exercises and Projects — Modifying the Part
  - Converting Dimensions
  - Calculating the Modification
  - Modifying the Part
  - Calculating Material Volume
  - Calculating the Volume of the Base Feature
- ❑ Exercises and Projects — Creating a CD Jewel Case and Storage Box
  - Measuring the CD Jewel Case
  - Rough Sketch of the Jewel Case
  - Calculate the Overall Case Capacity
  - Calculate the Outside Measurements of the CD Storage Box
  - Creating the CD Jewel Case and Storage Box
- ❑ More to Explore — Modeling More Parts
- ❑ Lesson Summary

## Competencies for Lesson 3

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Students develop the following competencies in this lesson:

- ❑ **Engineering:** Utilize 3D features to create a 3D part. Create a pencil sketch of a profile for chalk and an eraser.
- ❑ **Technology:** Work with a common music/software case and determine the size of a CD container.
- ❑ **Math:** Apply concentric relations (same center) between circles. Understand conversion from millimeters to inches in an applied project. Apply width, height, and depth to a right prism (box).
- ❑ **Science:** Calculate volume of a right prism (box).

## In Class Discussion — Base Features

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- ❑ Select a simple object in the classroom, a piece of chalk or board eraser.
- ❑ Ask the students to describe the Base feature of these objects.
- ❑ How would you create the additional features for these objects?

### **Answer**

#### **Chalk:**

- ❑ Sketch a circular 2D profile.
- ❑ Extrude the 2D profile. The extruded 2D profile creates the Base feature. The Base feature is named `Extrude1`.
- ❑ Select the circular edge on the Base feature. Create a Fillet feature. The Fillet feature removes sharp edges.

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**Note:** You would probably not want to use the Fillet feature for a new piece of chalk.

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#### **Board Eraser:**

- ❑ Sketch a rectangular 2D profile.
- ❑ Extrude the 2D profile. The extruded 2D profile creates the Base feature.
- ❑ Select the 4 corners on the Base feature. Create a Fillet feature to remove the sharp edges.

## Active Learning Exercises — Create a Part

Follow the instructions in *Getting Started: Lesson 1 – Parts* of the SolidWorks Tutorial. In this lesson you will create the part shown at the right. The part name is Tutor1.sldprt.



## Lesson 3 — 5 Minute Assessment — Answer Key

Name: \_\_\_\_\_ Class: \_\_\_\_\_ Date: \_\_\_\_\_

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

1 What features did you use to create Tutor1?

**Answer:** Extruded Boss, Fillet, Shell and Extruded Cut.

2 What does the Fillet feature do?

**Answer:** The Fillet feature rounds sharp edges and faces.

3 What does the Shell feature do?

**Answer:** The Shell feature removes material from the selected face.

4 Name three view commands in SolidWorks.

**Answer:** Zoom to Fit, Rotate View, and Pan.

5 Where are the display buttons located?

**Answer:** The display buttons are located on the Heads-up View toolbar.

6 Name the three SolidWorks default planes.

**Answer:** Front, Top, and Right.

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

**Answer:**

- Front = Front or Back view
- Top = Top or Bottom view
- Right = Right or Left view

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

**Answer:** True.

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

**Answer:** False.

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

**Answer:** Top, Front, Right and Isometric views.

## Lesson 3 — 5 Minute Assessment

## REPRODUCIBLE

Name: \_\_\_\_\_ Class: \_\_\_\_\_ Date: \_\_\_\_\_

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

- 1 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.  
\_\_\_\_\_

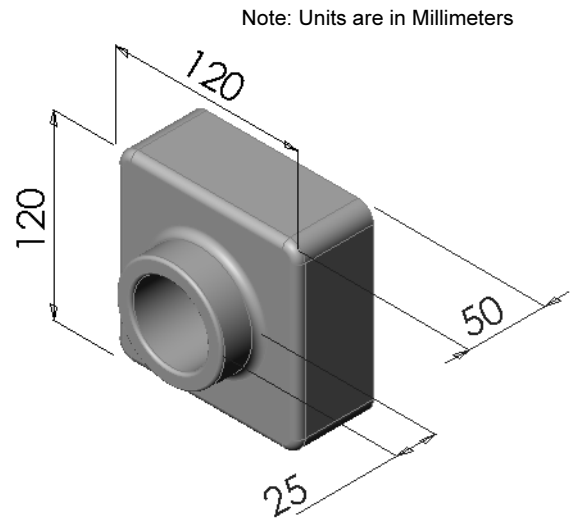
## Exercises and Projects — Modifying the Part

### Task 1 — Converting Dimensions

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 width = 120 mm
- ❑ Base height = 120 mm
- ❑ Base depth = 50 mm
- ❑ Boss depth = 25 mm



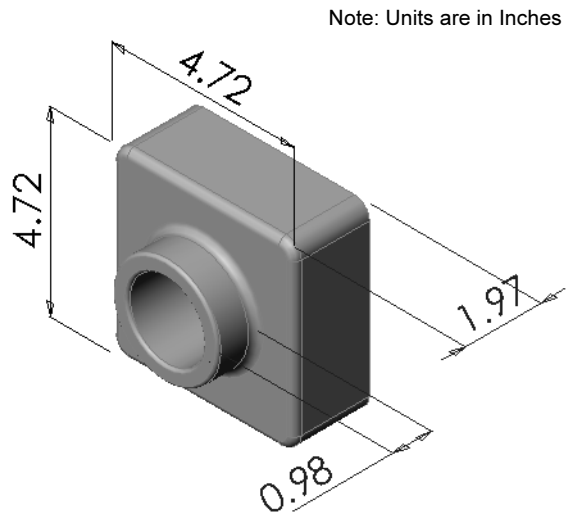
**Answer:**

- ❑ Overall depth = Base depth + Boss depth  
Overall depth = 1.97" + 0.98" = 2.95"
- ❑ Overall dimensions = Base width x Base height x Depth  
Overall dimensions = 4.72" x 4.72" x 2.95"

### In Class Demonstration:

SolidWorks supports both metric and English units. Demonstrate the software conversion from metric to English units.

- 1 Click **Tools, Options**.
- 2 Click the **Document Properties** tab.
- 3 Click **Units**.
- 4 Change **Unit system** to **Custom** and select **inches** for **Length**. Click **OK**.
- 5 Double-click the Tutor1 features to display the dimensions.
  - Base width = 4.72"
  - Base height = 4.72"
  - Base depth = 1.97"
  - Boss depth = 0.98"
- 6 Change the **Length** of the part back to **Millimeters** for the next task.



## Task 2 — Calculating the Modification

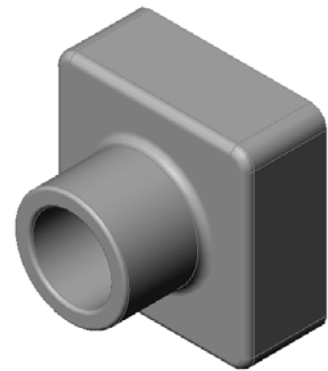
The current overall depth of Tutor1 is 75 mm. Your customer requires a design change. The new required overall depth is 100 mm. The Base depth must remain fixed at 50 mm. Calculate the new Boss depth.

### Given:

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

### Answer:

- Overall depth = Base depth + Boss depth
- Boss depth = Overall depth - Base depth
- Boss depth = 100mm - 50 mm
- Boss depth = 50 mm



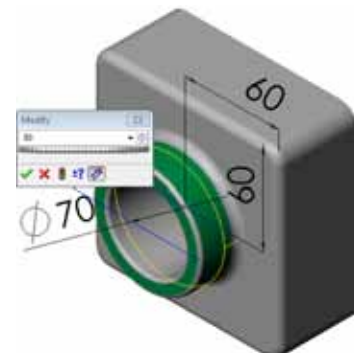
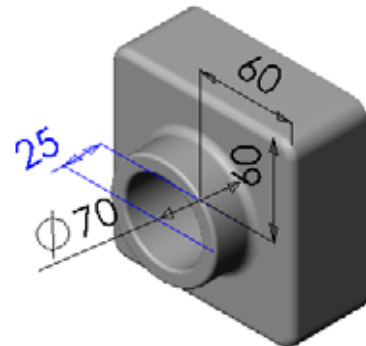
## Task 3 — Modifying the Part

Using SolidWorks, modify Tutor1 to meet the customer's requirements. Change the depth of the Boss feature such that the overall depth of the part equals 100 mm.

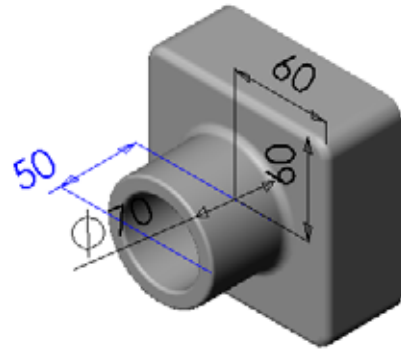
Save the modified part under a different name.

### Answer:

- 1 Double-click on the Extrude2 feature.
- 2 Double-click on the **25 mm** depth dimension.
- 3 In the **Modify** dialog, enter the value **50mm**.
- 4 Press **Enter**.



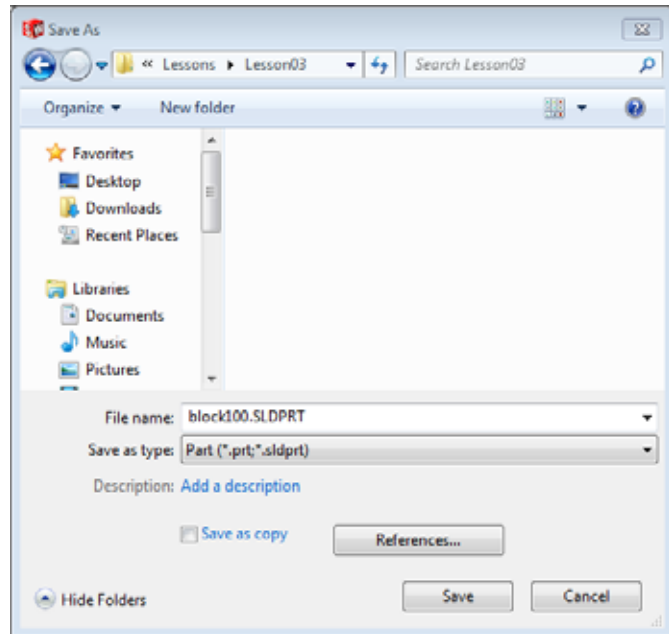
5 Click **Rebuild**.



6 Click **File, Save As** to create block100.

When you use **File, Save As**, you save a copy of the document with a new name or path. You can create a new folder in the **Save As** dialog box if needed. After you use **File, Save As**, you are working in the *new* document. The original document is closed without saving.

If you click the **Save as copy** check box you will save a copy of the document, with a new name, *without* replacing the active document. You continue to work in the original document.



#### Task 4 — Calculating Material Volume

Material volume is an important calculation for designing and manufacturing parts. Calculate the volume of the Base feature in  $\text{mm}^3$  for Tutor1.

**Answer:**

- Volume = Width x Height x Depth
- Volume =  $120\text{mm} \times 120\text{mm} \times 50\text{mm} = 720,000 \text{ mm}^3$

#### Task 5 — Calculating the Volume of the Base feature

Calculate the volume of the Base feature in  $\text{cm}^3$ .

**Given:**

- $1\text{cm} = 10\text{mm}$

**Answer:**

- Volume = Width x Height x Depth
- Volume =  $12\text{cm} \times 12\text{cm} \times 5\text{cm} = 720\text{cm}^3$

