SolidWorks®

SolidWorks Teacher Guide
and Student Courseware

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U.S. Patents 5,815,154; 6,219,049; 6,219,055

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Document Number: PME0213-ENG

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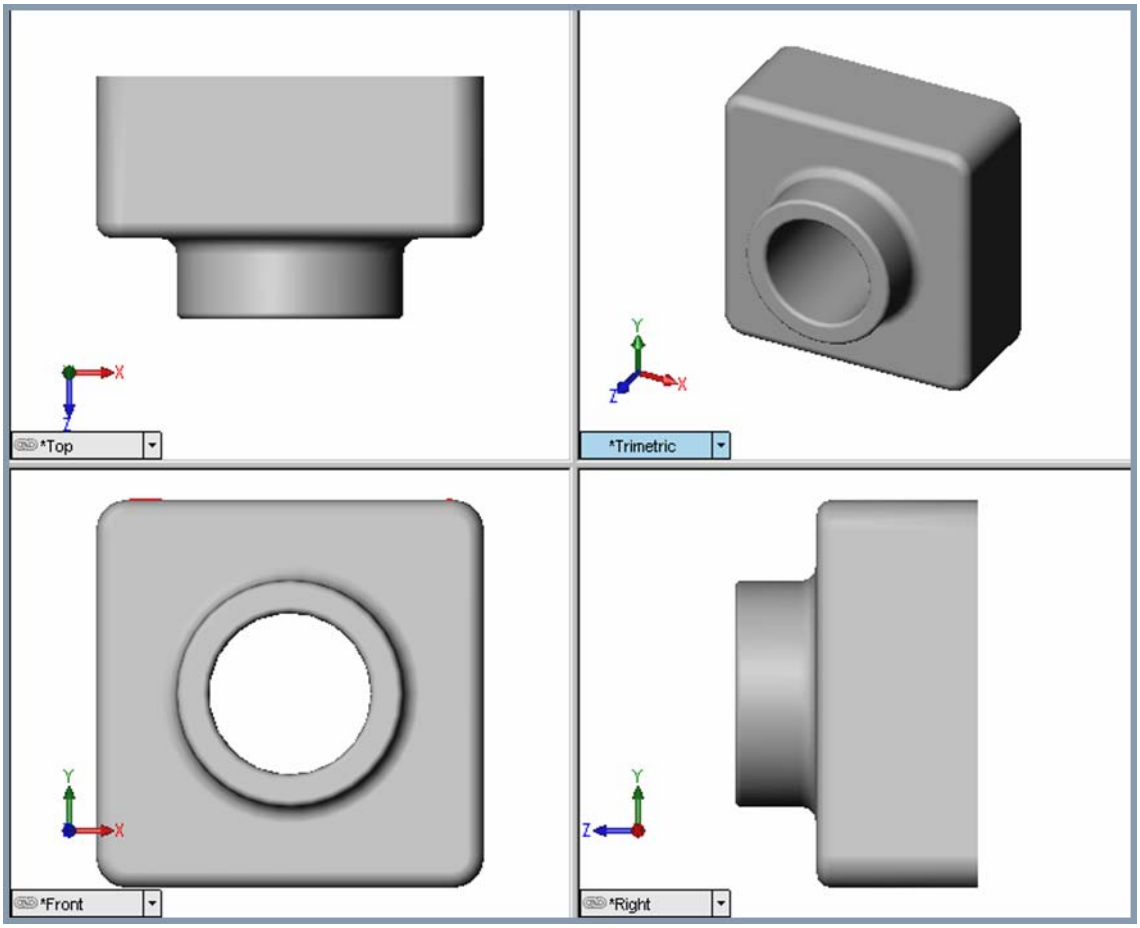
Lesson 3: The 40-Minute Running Start

Goals of This Lesson

- Students will be able to create and modify the following part:



Before Beginning This Lesson

- Complete the previous lesson — Basic Functionality.

Resources for This Lesson

This lesson plan corresponds to *Lesson 1 – Parts* in the SolidWorks Online Tutorials. For more information about the Online Tutorials, See “Online Tutorials” on page v.

Review of Lesson 2: Basic Functionality

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.

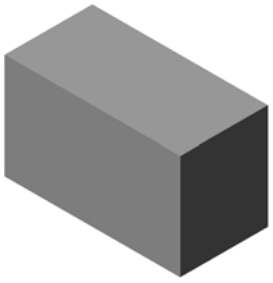
3. Name the features that are used to create the `box` in Lesson 1.

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

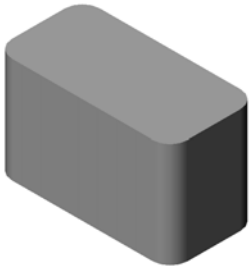
4. What is the base feature of the box?

   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.

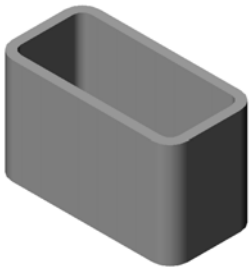
   - Use a cardboard box to illustrate the Base feature.



1. Base Feature



2. Fillet Feature



3. Shell Feature

4. Cut Feature

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.

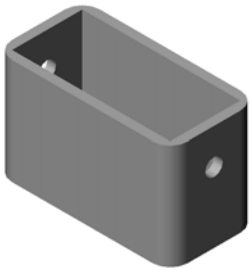
7. How do you create the Base feature?

   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.



Outline of Lesson 3

- ❑ In Class Discussion — Base Features
- ❑ Active Learning Exercise — Create a Part
- ❑ Exercises and Projects — Modifying a 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

In Class Discussion — Base Features

- 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.

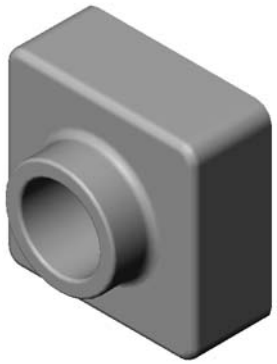
> **Note:** You would probably not want to use the Fillet feature for a new piece of chalk.

Board Eraser:

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

Active Learning Exercise — Create a Part

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



5 Minute Assessment – Answer Key

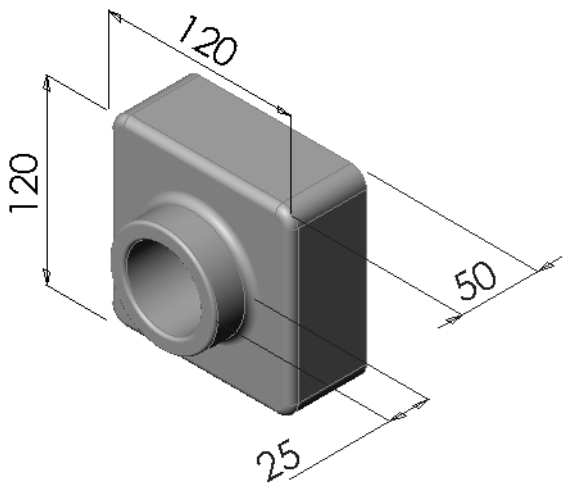
1. What features did you use to create `Tutor1`?
   Answer: Extruded Boss, Fillet, Shell and Extruded Cut.
2. What does the Fillet feature do?
   Answer: The Fillet feature rounds sharp edges and faces
3. What does the Shell feature do?
   Answer: The Shell feature removes material from the selected 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 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.

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.

Note: Units are in Millimeters



Given:

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

Answer:

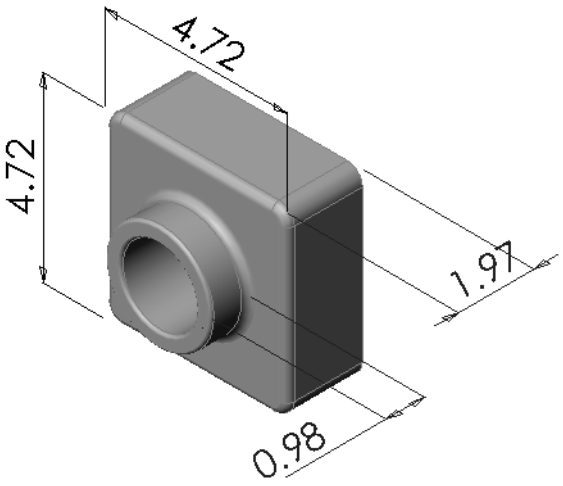
- Overall depth = Base depth + Boss depth
  Overall depth = 1.97” + 0.98” = 2.95”
- Overall dimensions = Base width **x** Base height **x** Depth
  Overall dimensions = 4.72” **x** 4.72” **x** 2.95”

In Class Demonstration:

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

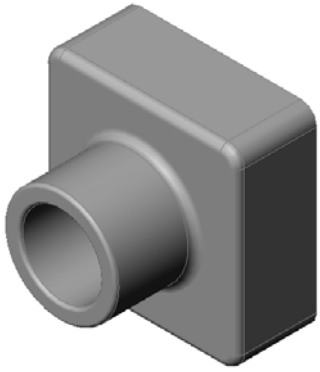
1. Click **Tools, Options**.
2. Click the **Document Properties** tab.
3. Click **Units**.
4. Click **Inches** from the **Linear units** list. Click **OK**.
5. Double-click the `Tutor1` features to display the dimensions.
   - Base width = 4.72”
   - Base height = 4.72”
   - Base depth = 1.97”
   - Boss depth = 0.98”

Note: Units are in Inches



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:

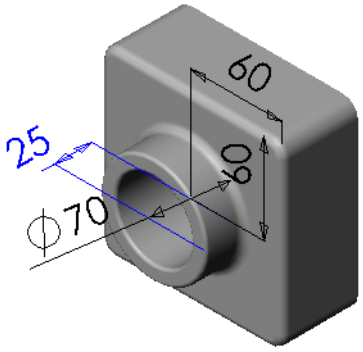
- Overall depth = Base depth + Boss depth
  Boss depth = Overall depth - Base depth
  Boss depth = 100mm - 50 mm
  Boss depth = 50 mm

Task 3— Modifying the Part

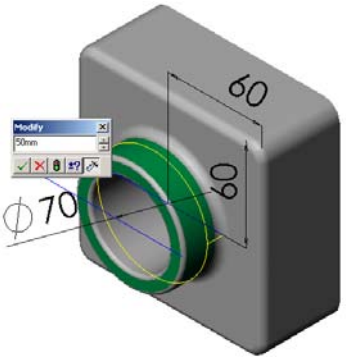
Using SolidWorks, modify `Tutor1` to meet the customer's requirements. Change the depth of the Boss feature such that the overall depth of the part equals 100 mm. Save the modified part under a different name.

Answer:

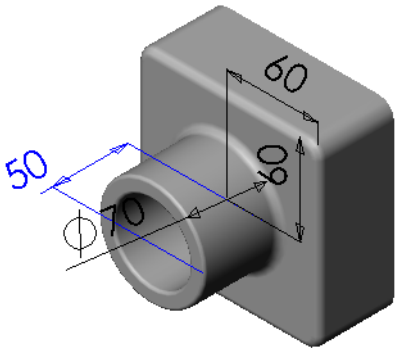
1. Double-click on the `Extrude2` feature.



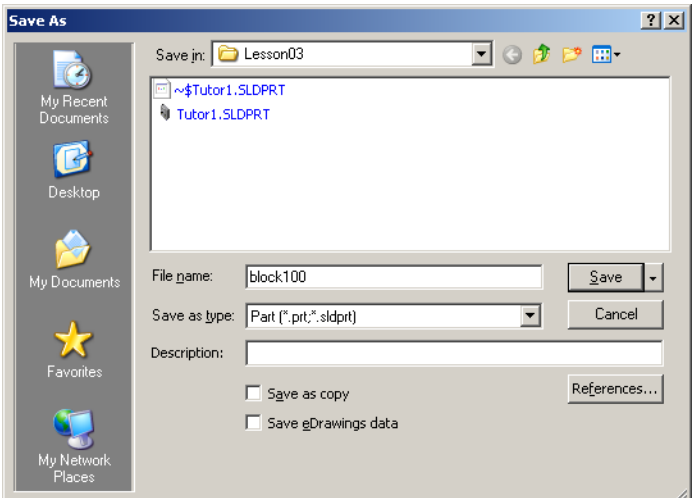
2. Double-click on the **25 mm** depth dimension.
3. In the **Modify** dialog, enter the value **50mm**.
4. Press **Enter**.



5 Click **Rebuild**.



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

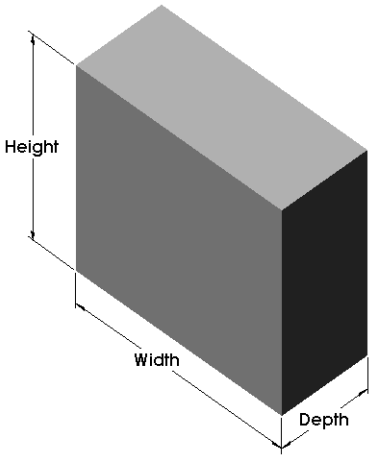


When you use **File, Save As**, you save a copy of the document with a new name or path. You can create a new folder in the **Save As** dialog box if needed. After you use **File, Save As**, you are working in the *new* document. The original document is closed without saving.

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 mm^3 for `Tutor1`.



Answer:

- Volume = Width **x** Height **x** Depth
  Volume = 120mm **x** 120mm **x** 50mm = 720,000 mm^3

Task 5— Calculating the Volume of the Base feature

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

Given:

- 1cm = 10mm

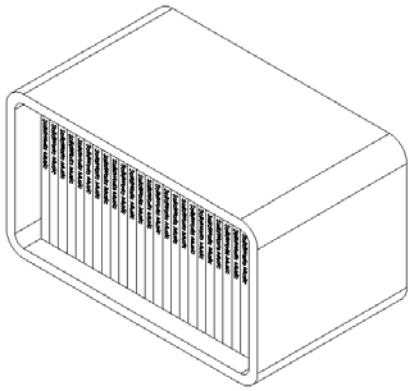
Answer:

- Volume = Width **x** Height **x** Depth
  Volume = 12cm **x** 12cm **x** 5cm = $720cm^3$

Exercises and Projects — Creating a CD Jewel Case and Storage Box

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.

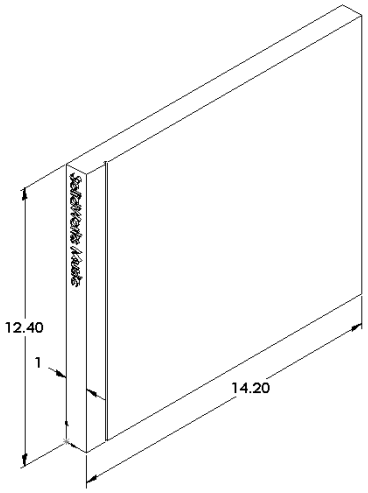


Task 1 — Measuring the CD Jewel Case

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

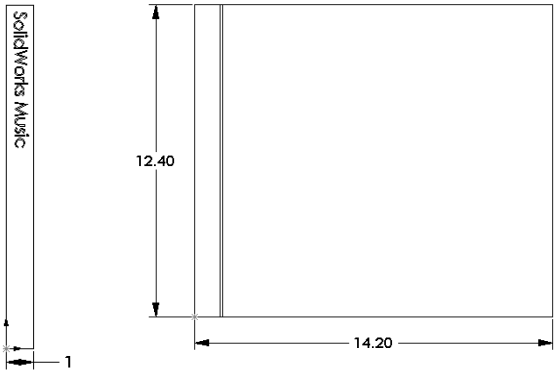
Answer:

Approximately 14.2cm **x** 12.4cm **x** 1cm



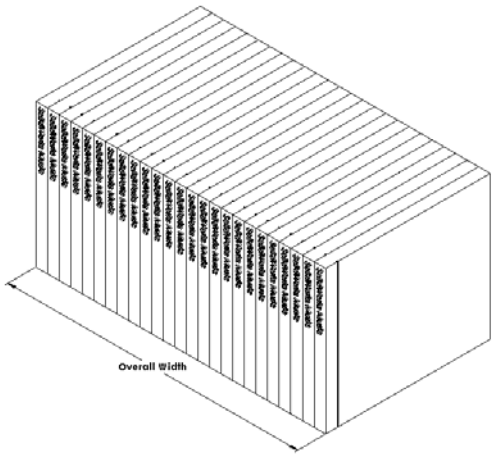
Task 2— Rough Sketch of the Jewel Case

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



Task 3 — Calculate the Overall Case Capacity

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



Given:

- CD jewel case width = 1cm
- CD jewel case height = 12.4cm
- CD jewel case depth = 14.2cm

Answer:

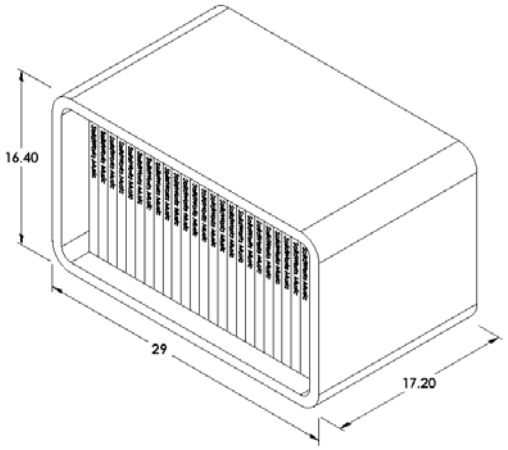
- Overall width of 25 CD jewel cases = 25 **x** 1cm = 25cm
- Overall size for 25 CD jewel cases = Overall width **x** CD case height **x** CD case depth
  Overall size for 25 CD jewel cases = 25cm **x** 12.4cm **x** 14.2cm

Task 4— Calculate the Outside Measurements of the CD Storage Box

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:

- Clearance = 2cm
- Wall thickness = 1cm
- Wall thickness is applied to both sides of the width and height dimensions. Wall thickness is applied to one side of the depth dimension.
- CD storage box width = Overall width of 25 CD jewel cases + Clearance + Wall thickness + Wall thickness
  CD storage box width = 25cm + 2cm + 1cm + 1cm = 29cm



- CD storage box height = CD case height + Clearance + Wall thickness + Wall thickness
  CD storage box height = 12.4cm + 2cm + 1cm + 1cm = 16.4cm
- CD storage box depth = CD case depth + Clearance + Wall thickness
  CD storage box depth = 14.2cm + 2cm + 1cm = 17.2cm
- Overall size CD storage box = Storage box width **x** Storage box height **x** Storage box depth
  Overall size CD storage box = 29cm **x** 16.4cm **x** 17.2cm

Task 5— Creating the CD Jewel Case and Storage Box

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.

More to Explore — Modeling More Parts

Description

Look at the following examples. There are at least three 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.

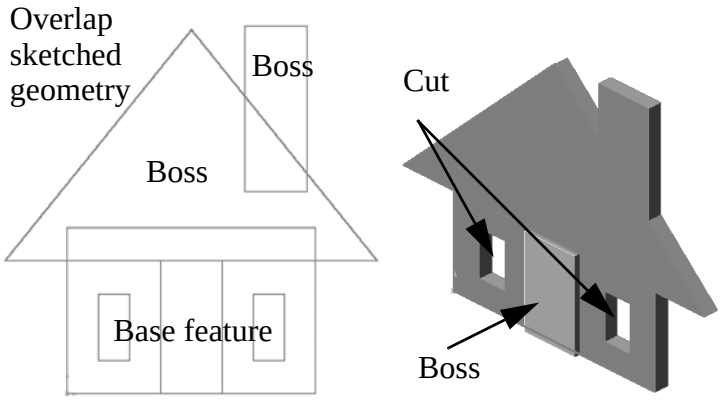
> **Note:** Each new sketch should overlap an existing feature.

Task 1

`house.sldprt`

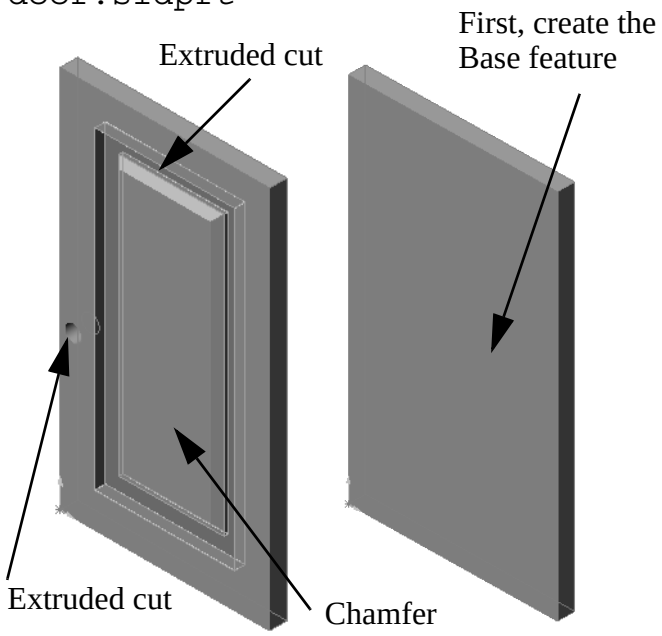
Answer:

- ❑ The features used to create the house are:
  - Base feature - Sketch a rectangle to create the 1st floor.
  - Extrude cut - Sketch a rectangle to create the windows.
  - Extruded boss - Sketch a triangle to create the roof.
  - Extruded boss - Sketch a rectangle to create the chimney.
  - Extruded boss - Sketch a rectangle to create the front door.



Task 2

door.sldprt



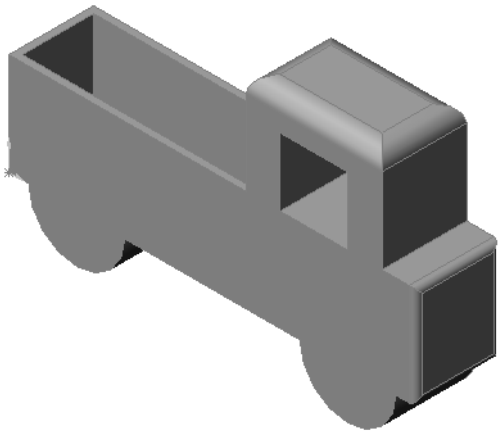
Answer:

- The features used to create the door are:
  - Base feature - Sketch a rectangle to create the door.
  - Extruded cut - Sketch a circle to create the door hole.
  - Extruded cut - Sketch a rectangle to create the panel.
  - Chamfer - Select the middle face. Click **Chamfer** on the Features toolbar. Enter a value for **Distance**. Click **OK.**

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.

Task 3

truck.sldprt



Answer:

- The features used to create the `truck` are:
  - Base feature - Sketch a rectangle to create the body.
  - Extruded boss - Sketch two circles to create the wheel.
  - Extruded boss - Sketch the rectangle to create the cab.
  - Extruded cut - Sketch the rectangle to create the window.
  - Shell - Select the back top face to create the truck bed.
  - Fillet - Select the top face and the front face to round the cab roof and truck front.

Lesson 3 Quiz

Name: _____Class: _____ Date:_____

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

1 How do you begin a new part document? _____

_____

2 How do you open a sketch? _____

_____

3 What is the Base feature? _____

_____

4 What color is the geometry of a fully defined sketch?_____

_____

5 How can you change a dimension value?_____

_____

6 What is the difference between an extruded boss feature and an extruded cut feature?

_____

7 What is a fillet feature?_____

_____

8 What is a shell feature?_____

_____

9 Name four types of geometric relations you can add to a sketch? _____

_____

10 What is a section view? _____

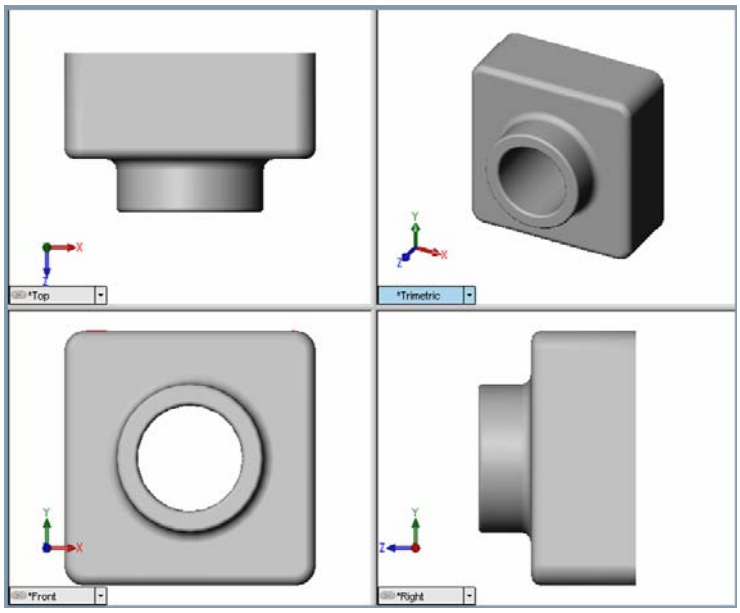
_____

11 How do you create multiple views of a part?_____

_____

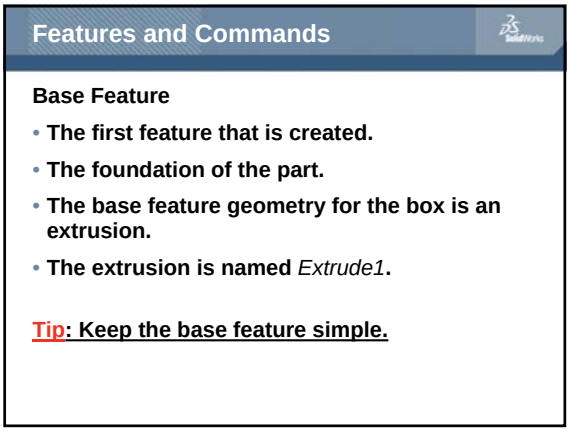
Lesson Summary

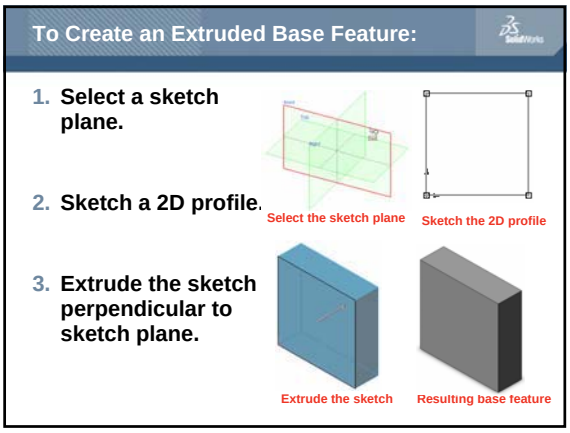
- Base Feature is the first feature that is created — the foundation of the part.
- The Base Feature is the workpiece to which everything else is attached.
- You can create an Extruded Base Feature by selecting a sketch plane and extruding the sketch perpendicular to sketch plane.
- Shell Feature creates a hollow block from a solid block.
- The views most commonly used to describe a part are:
  Top View
  Front View
  Right View
  Isometric View

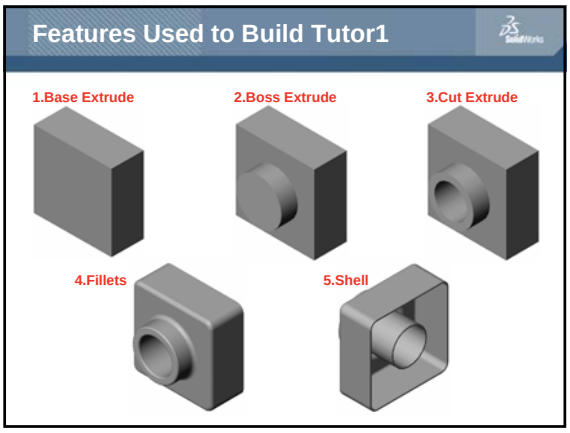


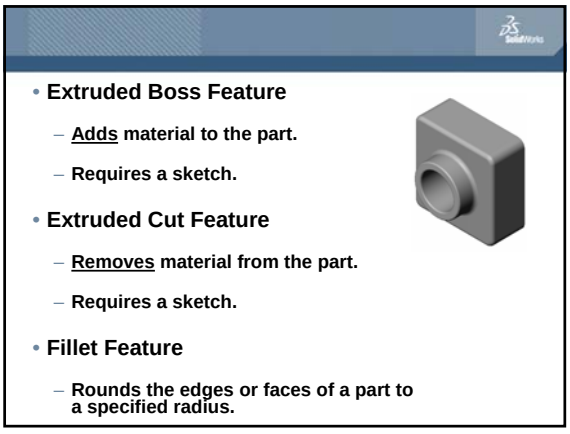
Thumbnail Images of PowerPoint® Slides

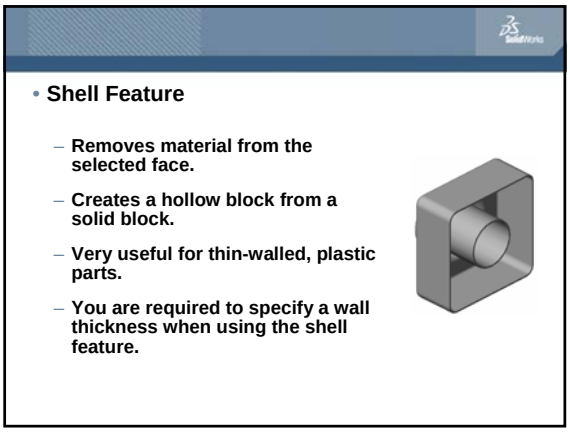
The following thumbnail images, arranged left to right, show the PowerPoint slides provided with this lesson.











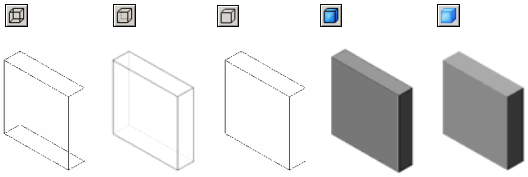
View Control

Magnify or reduce the view of a model in the graphics area.

- Zoom to Fit – displays the part so that it fills the current window.
- Zoom to Area – zooms in on a portion of the view that you select by dragging a bounding box.
- Zoom In/Out – drag the pointer upward to zoom in. Drag the pointer downward to zoom out.
- Zoom to Selection – the view zooms so that the selected object fills the window.

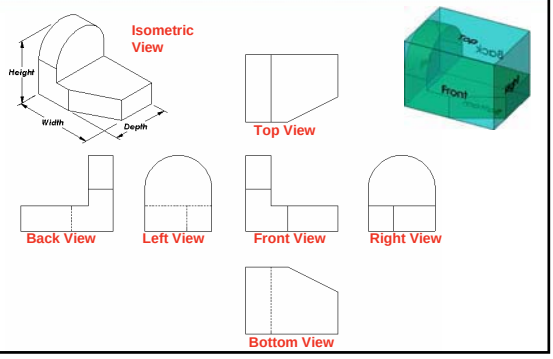
Display Modes

- **Illustrate the part in various display modes.**



Wireframe | Hidden lines Visible | Hidden Lines Removed | Shaded With Edges | Shaded

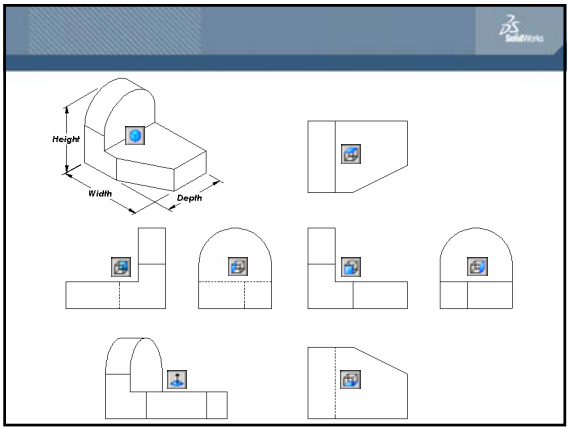
Standard Views



View Orientation

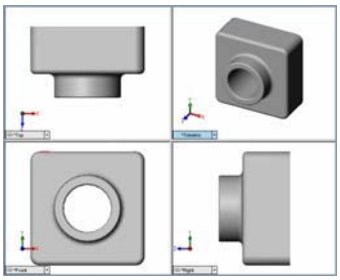
Changes the view display to correspond to one of the standard view orientations.

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



View Orientation

- **The views most commonly used to describe a part are:**
  - Top View
  - Front View
  - Right View
  - Isometric View

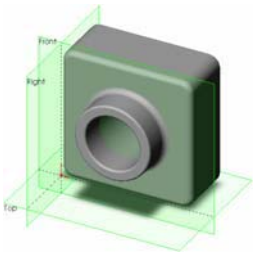


Default Planes

- **Default Planes**
  - Front, Top, **and** Right

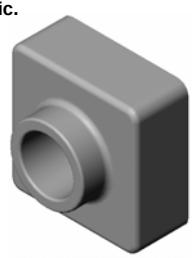
Correspond to the standard principle drawing views:

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



Isometric View

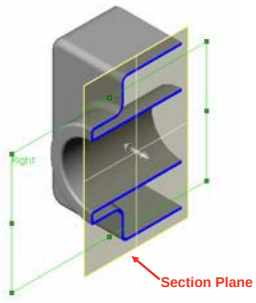
- **Displays the part with height, width, and depth equally foreshortened.**
  - **Pictorial rather than orthographic.**
  - **Shows all three dimensions – height, width, and depth.**
  - **Easier to visualize than orthographic views.**



Section View

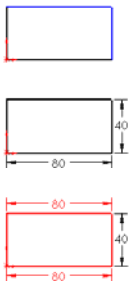
- **Displays the internal structure of a model.**
- **Requires a section cutting plane.**


Mouse over



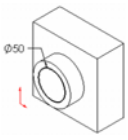
The Status of a Sketch

- **Under defined**
  - **Additional dimensions or relations are required.**
  - **Under defined sketch entities are *blue* (by default).**
- **Fully defined**
  - **No additional dimensions or relationships are required.**
  - **Fully defined sketch entities are *black* (by default).**
- **Over defined**
  - **Contains conflicting dimensions or relations, or both.**
  - **Over defined sketch entities are *red* (by default).**



Geometric Relations

- **Geometric relations are the rules that control the behavior of sketch geometry.**
- **Geometric relations help capture design intent.**
- **Example: The sketched circle is concentric with the circular edge of the extruded boss feature.**
- **In a concentric relation, selected entities have the same center point.**



Geometric Relations

- **The SolidWorks default name for circular geometry is an Arc#.**
- **SolidWorks treats circles as 360° arcs.**