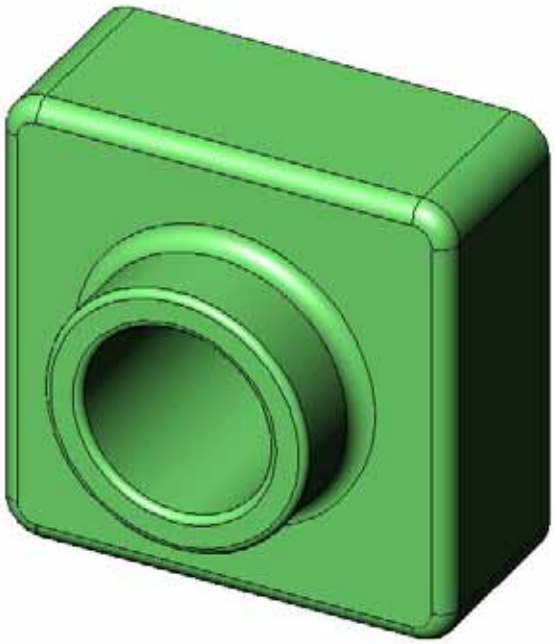
SolidWorks Education
Learn. Create. Succeed.

Engineering Design and Technology Series

Instructor's Guide to Teaching SolidWorks® Software



Dassault Systèmes - SolidWorks Corporation
300 Baker Avenue
Concord, Massachusetts 01742 USA
Phone: +1-800-693-9000

Outside the U.S.: +1-978-371-5011
Fax: +1-978-371-7303
Email: info@solidworks.com
Web: http://www.solidworks.com/education

© 1995-2010, Dassault Systèmes SolidWorks Corporation, a Dassault Systèmes S.A. company, 300 Baker Avenue, Concord, Mass. 01742 USA. All Rights Reserved.

The information and the software discussed in this document are subject to change without notice and are not commitments by Dassault Systèmes SolidWorks Corporation (DS SolidWorks).

No material may be reproduced or transmitted in any form or by any means, electronically or manually, for any purpose without the express written permission of DS SolidWorks.

The software discussed in this document is furnished under a license and may be used or copied only in accordance with the terms of the license. All warranties given by DS SolidWorks as to the software and documentation are set forth in the license agreement, and nothing stated in, or implied by, this document or its contents shall be considered or deemed a modification or amendment of any terms, including warranties, in the license agreement.

Patent Notices

SolidWorks® 3D mechanical CAD software is protected by U.S. Patents 5,815,154; 6,219,049; 6,219,055; 6,611,725; 6,844,877; 6,898,560; 6,906,712; 7,079,990; 7,477,262; 7,558,705; 7,571,079; 7,590,497; 7,643,027; 7,672,822; 7,688,318; 7,694,238; 7,853,940 ; and foreign patents, (e.g., EP 1,116,190 and JP 3,517,643).

eDrawings® software is protected by U.S. Patent 7,184,044; U.S. Patent 7,502,027; and Canadian Patent 2,318,706.

U.S. and foreign patents pending.

Trademarks and Product Names for SolidWorks Products and Services

SolidWorks, 3D PartStream.NET, 3D ContentCentral, eDrawings, and the eDrawings logo are registered trademarks and FeatureManager is a jointly owned registered trademark of DS SolidWorks.

CircuitWorks, Feature Palette, FloXpress, PhotoWorks, TolAnalyst, and XchangeWorks are trademarks of DS SolidWorks.

FeatureWorks is a registered trademark of Geometric Software Solutions Ltd.

SolidWorks 2011, SolidWorks Enterprise PDM, SolidWorks Simulation, SolidWorks Flow Simulation, and eDrawings Professional are product names of DS SolidWorks.

Other brand or product names are trademarks or registered trademarks of their respective holders.

COMMERCIAL COMPUTER SOFTWARE - PROPRIETARY

U.S. Government Restricted Rights. Use, duplication, or disclosure by the government is subject to restrictions as set forth in FAR 52.227-19 (Commercial Computer Software - Restricted Rights), DFARS 227.7202 (Commercial Computer Software and Commercial Computer Software Documentation), and in the license agreement, as applicable.

Contractor/Manufacturer:

Dassault Systèmes SolidWorks Corporation, 300 Baker Avenue, Concord, Massachusetts 01742 USA

Copyright Notices for SolidWorks Standard, Premium, Professional, and Education Products

Portions of this software © 1986-2010 Siemens Product Lifecycle Management Software Inc. All rights reserved.

Portions of this software © 1986-2010 Siemens Industry Software Limited. All rights reserved.

Portions of this software © 1998-2010 Geometric Ltd.

Portions of this software © 1996-2010 Microsoft Corporation. All rights reserved.

Portions of this software incorporate PhysX™ by NVIDIA 2006-2010.

Portions of this software © 2001 - 2010 Luxology, Inc. All rights reserved, Patents Pending.

Portions of this software © 2007 - 2010 DriveWorks Ltd.

Copyright 1984-2010 Adobe Systems Inc. and its licensors. All rights reserved. Protected by U.S. Patents 5,929,866; 5,943,063; 6,289,364; 6,563,502; 6,639,593; 6,754,382; Patents Pending.

Adobe, the Adobe logo, Acrobat, the Adobe PDF logo, Distiller and Reader are registered trademarks or trademarks of Adobe Systems Inc. in the U.S. and other countries.

For more copyright information, in SolidWorks see Help > About SolidWorks.

Copyright Notices for SolidWorks Simulation Products

Portions of this software © 2008 Solversoft Corporation.

PCGLSS © 1992-2007 Computational Applications and System Integration, Inc. All rights reserved.

Copyright Notices for Enterprise PDM Product

Outside In® Viewer Technology, © Copyright 1992-2010, Oracle

© Copyright 1995-2010, Oracle. All rights reserved.

Portions of this software © 1996-2010 Microsoft Corporation. All rights reserved.

Copyright Notices for eDrawings Products

Portions of this software © 2000-2010 Tech Soft 3D.

Portions of this software © 1995-1998 Jean-Loup Gailly and Mark Adler.

Portions of this software © 1998-2001 3Dconnexion.

Portions of this software © 1998-2010 Open Design Alliance. All rights reserved.

Portions of this software © 1995-2009 Spatial Corporation.

This software is based in part on the work of the Independent JPEG Group.

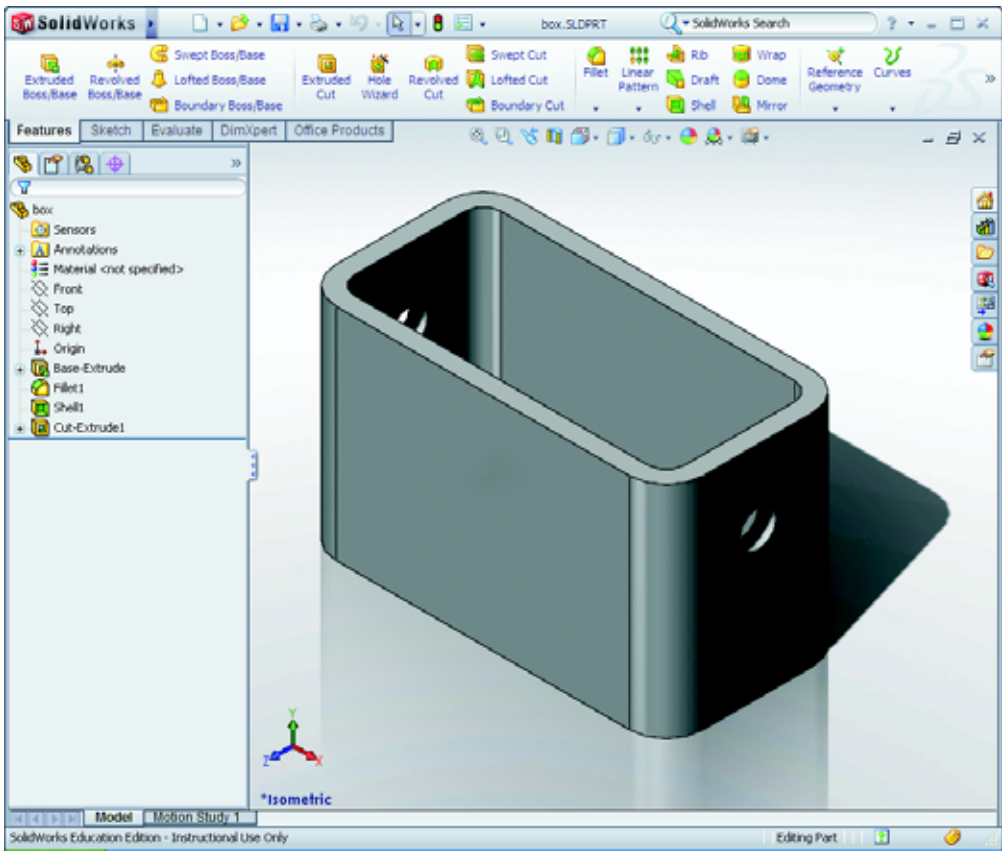
Document Number: PME0119-ENG

2

Lesson 2: Basic Functionality

Goals of This Lesson

- Understand the basic functionality of the SolidWorks software.
- Create the following part:



Before Beginning This Lesson

Complete Lesson 1: Using the Interface.

Access a wide range of free, informative resources - full video tutorials, PDF guides, project files, and demo clips - designed to help you become a top SolidWorks user. Visit http://www.solidworks.com/tutorials.

Review of Lesson 1: Using the Interface

The interface is how *you* interact with the computer in the following ways:

- Use windows to view files.
- Use the mouse to select buttons, menus, and model elements.
- Run programs — like SolidWorks mechanical design software.
- Find, open, and work with files.
- Create, save, and copy files.
- SolidWorks runs on the Microsoft Windows graphical user interface.
- The mouse lets you move around the interface.
- The quickest way to open a file is to double-click on it.
- Saving a file preserves the changes that you have made to it.
- SolidWorks windows display graphic and non-graphic model data.
- Toolbars display frequently used commands.

Outline of Lesson 2

- In Class Discussion — The SolidWorks Model
- Active Learning Exercise — Creating a Basic Part
  - Create a New Part Document
  - Overview of the SolidWorks Window
  - Sketch a Rectangle
  - Add Dimensions
  - Changing the Dimension Values
  - Extrude the Base Feature
  - View Display
  - Save the Part
  - Round the Corners of the Part
  - Hollow Out the Part
  - Extruded Cut Feature
  - Open a Sketch
  - Sketch the Circle
  - Dimension the Circle
  - Extrude the Sketch
  - Rotate the View
  - Save the Part
- In Class Discussion — Describing the Base Feature
- Exercises and Projects — Designing a Switch Plate
- More to Explore — Modifying a Part
- Lesson Summary

Competencies for Lesson 2

Students develop the following competencies in this lesson:

- **Engineering**: Develop a 3D part based on a selected plane, dimensions, and features. Apply the design process to develop the box or switch plate out of cardboard or other material. Develop manual sketching techniques by drawing the switch plate.
- **Technology**: Apply a windows based graphical user interface.
- **Math**: Understand units of measurement, adding and subtracting material, perpendicularity, and the x-y-z coordinate system.

In Class Discussion — The SolidWorks Model

SolidWorks is design automation software. In SolidWorks, you sketch ideas and experiment with different designs to create 3D models. SolidWorks is used by students, designers, engineers, and other professionals to produce simple and complex parts, assemblies, and drawings.

The SolidWorks model is made up of:

- Parts
- Assemblies
- Drawings

A part is a single 3D object made up of features. A part can become a component in an assembly, and it can be represented in 2D in a drawing. Examples of parts are bolt, pin, plate, and so on. The extension for a SolidWorks part file name is .SLDPRT. Features are the *shapes* and *operations* that construct the part. The Base feature is the first feature that is created.The Base feature is the foundation of the part.

An assembly is a document in which parts, features, and other assemblies (sub-assemblies) are mated together. The parts and sub-assemblies exist in documents separate from the assembly. For example, in an assembly, a piston can be mated to other parts, such as a connecting rod or cylinder. This new assembly can then be used as a sub-assembly in an assembly of an engine. The extension for a SolidWorks assembly file name is .SLDASM.

A drawing is a 2D representation of a 3D part or assembly. The extension for a SolidWorks drawing file name is .SLDDRW.

Active Learning Exercises — Creating a Basic Part

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.

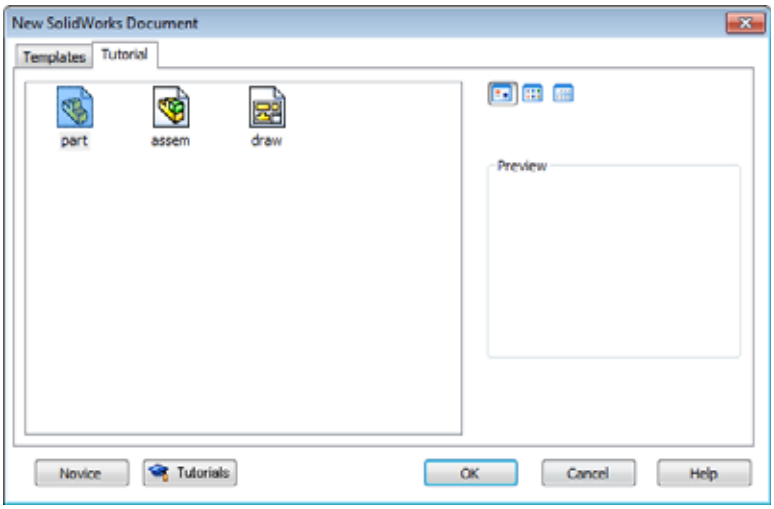
The **New SolidWorks Document** dialog box appears.

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 – `Front` (default plane)
- Sketch profile – 2D Rectangle
- Feature type – Extruded boss feature

Open a Sketch

1 Click to select the `Front` plane in the FeatureManager design tree.

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

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**.

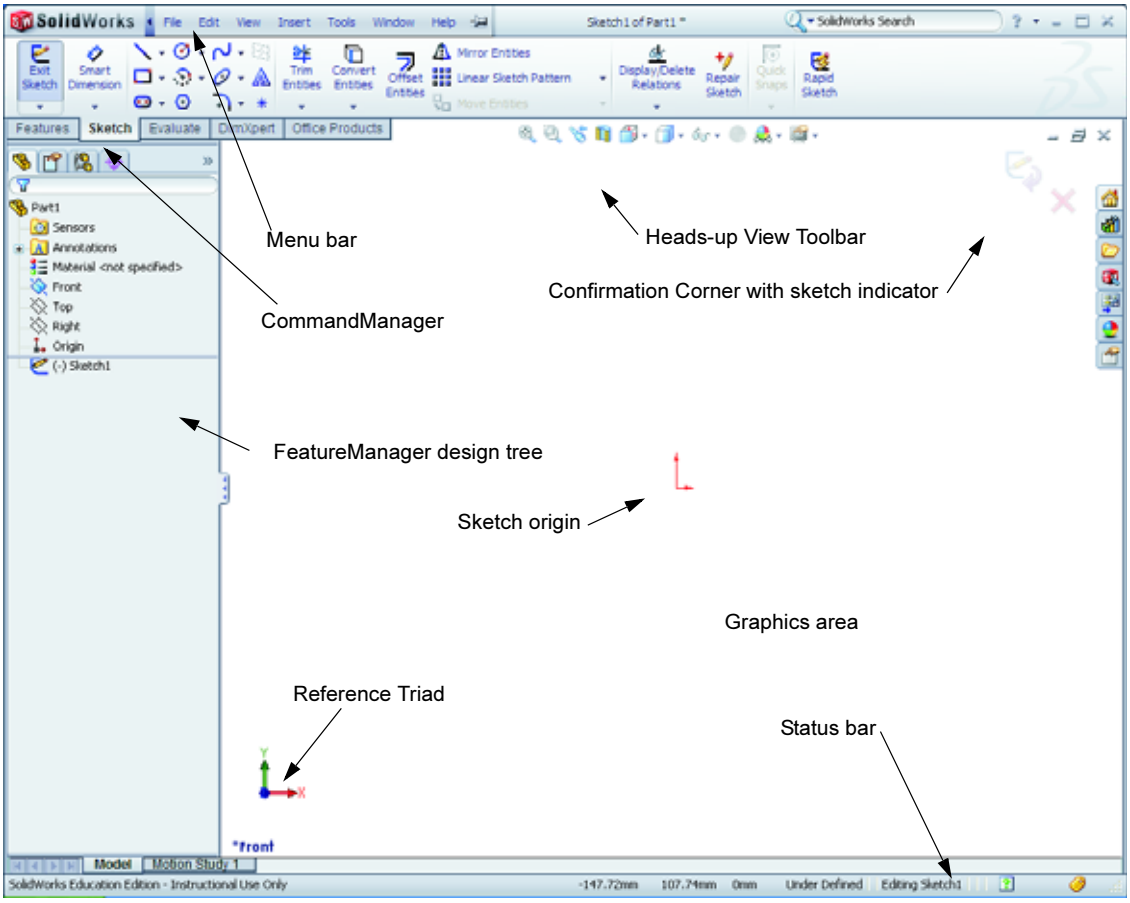
Sketch Indicator

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

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.

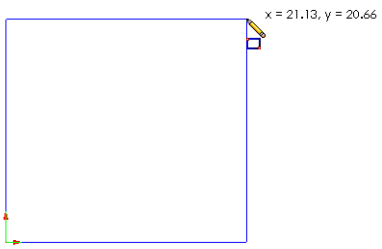
Overview of the SolidWorks Window

- A sketch origin appears in the center of the graphics area.
- **Editing Sketch1** 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

1. Click **Corner Rectangle** on the Sketch toolbar.
2. Click the sketch origin to start the rectangle.
3. Move the pointer up and to the right, to create a rectangle.
4. Click the mouse button again to complete the rectangle.



Add Dimensions

1 Click **Smart Dimension** on the Dimensions/Relations toolbar.

The pointer shape changes to .

2 Click the top line of the rectangle.

3 Click the dimension text location above the top line.

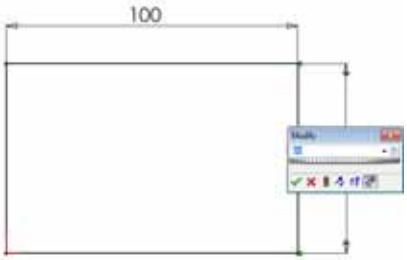
The **Modify** dialog box is displayed.

4 Enter **100**. Click or press **Enter**.

5 Click the right edge of the rectangle.

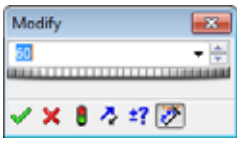
6 Click the dimension text location. Enter **65**. Click .

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.



1 Double-click **65**.

The **Modify** dialog box appears.

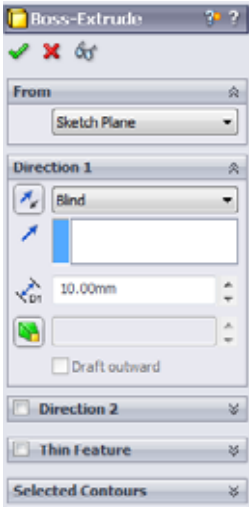
2 Enter **60** in the **Modify** dialog box.

3 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.

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



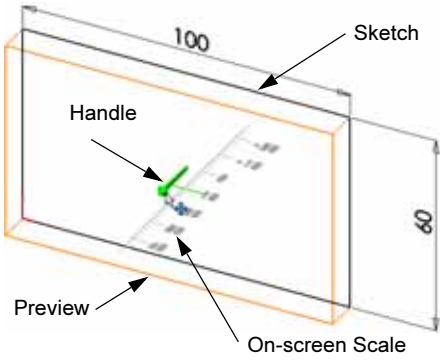
TIP: If the Features toolbar is not visible (active), you may also access the feature commands from the CommandManager.

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

2 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 magenta for the active direction and gray for inactive direction. A callout shows the current depth value.

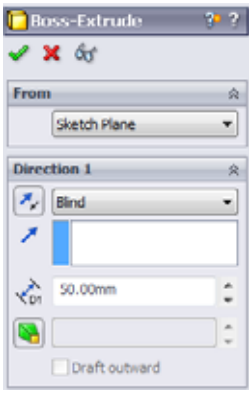


The cursor changes to . If you want to create the feature now, click the right mouse button. Otherwise, you can make additional changes to the settings. For example, the depth of extrusion can be changed by dragging the dynamic handle with the mouse or by setting a value in the PropertyManager.

3 Extrude feature settings.

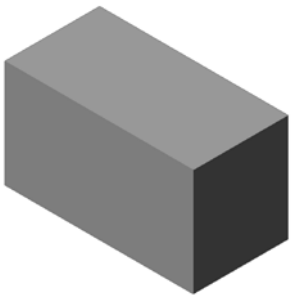
Change the settings as shown.

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



4 Create the extrusion. Click **OK** .

The new feature, `Boss-Extrude1`, is displayed in the FeatureManager design tree.

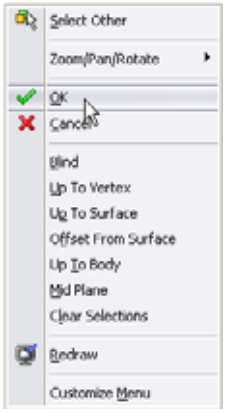


TIP:

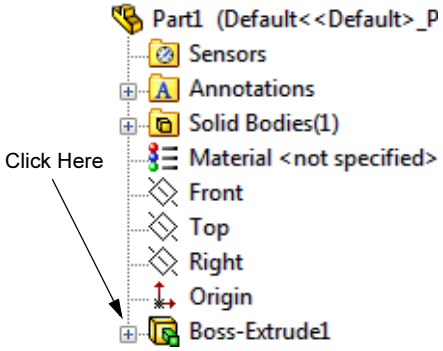
The **OK** button on the PropertyManager is just one way to complete the command.

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.



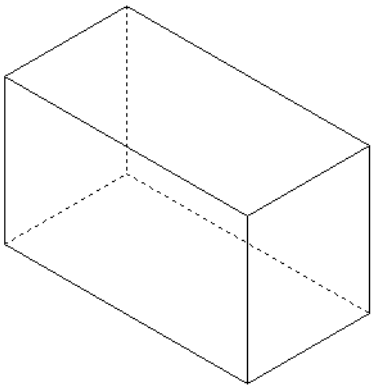
5 Click the plus sign ⊞ beside `Extrude1` in the FeatureManager design tree. Notice that `Sketch1` — which you used to extrude the feature — is now listed under the feature.



View Display

Change the display mode. Click **Hidden Lines Visible** on the View toolbar.

Hidden Lines Visible enables you to select hidden back edges of the box.



Save the Part

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

The **Save As** dialog box appears.

2 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.

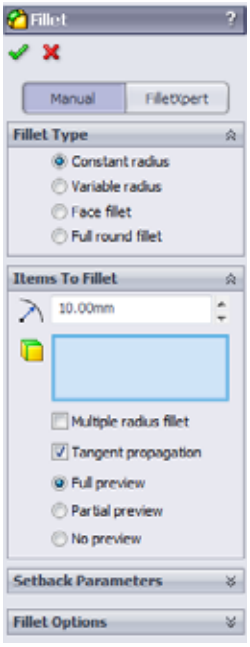
1 Click **Fillet** on the Features toolbar.

The **Fillet** PropertyManager appears.

2 Enter **10** for the **Radius.**

3 Select **Full preview**.

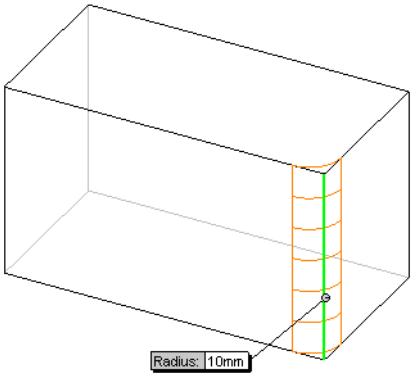
Leave the remaining settings at their default values.



4 Click the first corner edge.

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

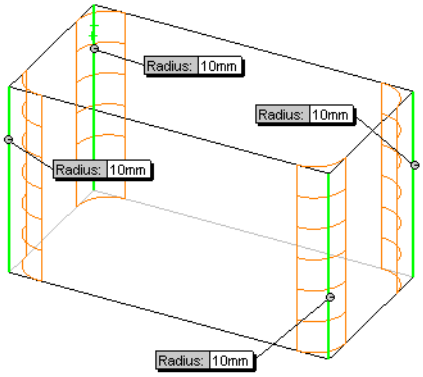
When you select the edge, a callout Radius: 10mm appears.



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

Edge: Face: Vertex:

6 Click the second, third and fourth corner edges.

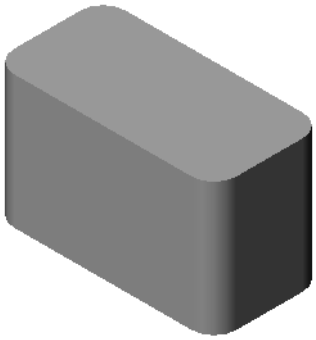


Note: Normally, a callout only appears on the *first* edge you select. This illustration has been modified to show callouts on each of the four selected edges. This was done simply to better illustrate which edges you are supposed to select.

7 Click **OK** .

`Fillet1` appears in the FeatureManager design tree.

8 Click **Shaded** on the View toolbar



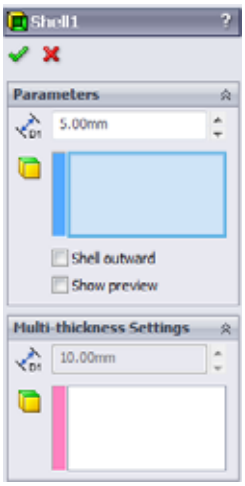
Hollow Out the Part

Remove the top face using the Shell feature.

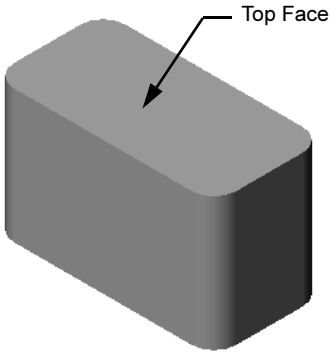
1 Click **Shell** on the Features toolbar.

The **Shell** PropertyManager appears.

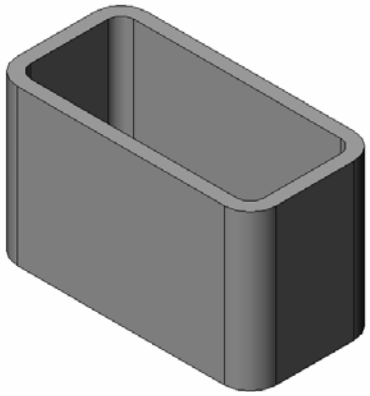
2 Enter **5** for **Thickness**.



3 Click the top face.



4 Click ✔.



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

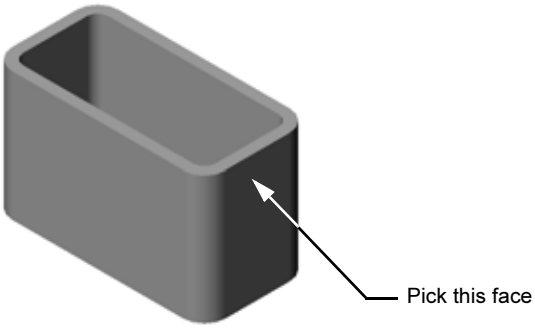
Open a Sketch

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

2 Click **Right** on the Standard Views toolbar.

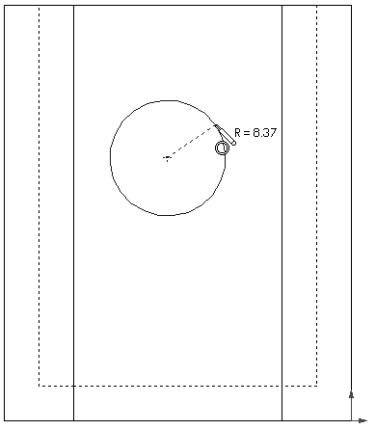
The view of the `box` turns. The selected model face is facing you.

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



Sketch the Circle

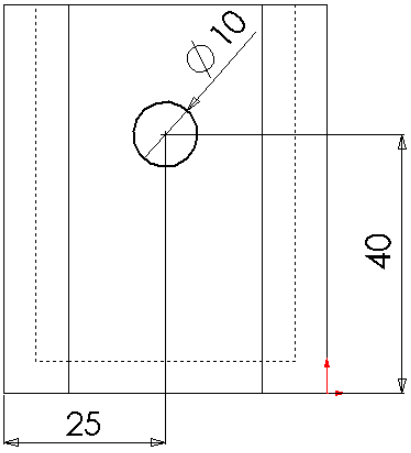
1. Click **Circle** on the Sketch Tools toolbar.
2. Position the pointer where you want the center of the circle. Click the left mouse button.
3. Drag the pointer to sketch a circle.
4. Click the left mouse button again to complete the circle.



Dimension the Circle

Dimension the circle to determine its size and location.

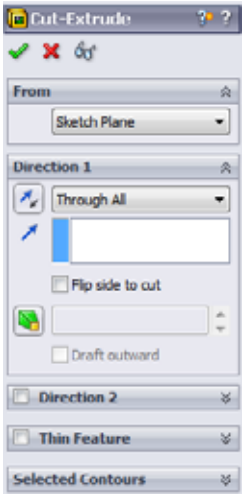
1. Click **Smart Dimension** on the Dimensions/ Relations toolbar.
2. Dimension the diameter. Click on the circumference of the circle. Click a location for the dimension text in the upper right corner. Enter **10**.
3. Create a horizontal dimension. Click the circumference of the circle. Click the left most vertical edge. Click a location for the dimension text below the bottom horizontal line. Enter **25**.
4. 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

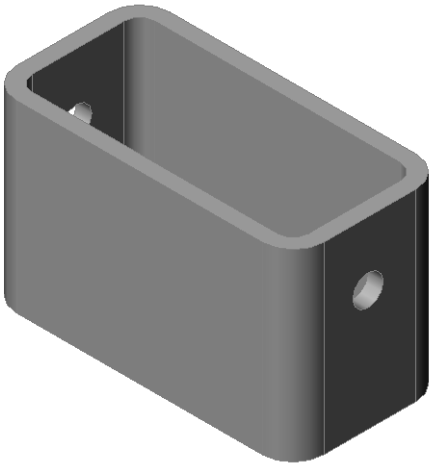
1. Click **Extruded Cut** on the Features toolbar.

   The **Extrude** PropertyManager appears.
2. Select **Through All** for the end condition.
3. Click .



4 Results.

The cut feature is displayed.



Rotate the View

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

1 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.

2 Click **Isometric** on the Standard Views toolbar.

Save the Part

1 Click **Save** on the Standard toolbar.

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

Lesson 2 — 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 How do you start a SolidWorks session?

Answer: Click . Click All Programs. Click the SolidWorks folder. Click the SolidWorks application.

2 Why do you create and use Document Templates?

Answer: Document Templates contain the units, grid and text settings for the model. You can create Metric and English templates each with different settings.

3 How do you start a new Part Document?

Answer: Click the **New** icon. Select a part template.

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

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

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

Answer: True.

6 A SolidWorks 3D model consists of _____ _____ _____.

Answer: Parts, assemblies and drawings.

7 How do you open a sketch?

Answer: Click the Sketch icon on the Sketch toolbar.

8 What does the Fillet feature do?

Answer: The Fillet feature rounds sharp edges.

9 What does the Shell feature do?

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

10 What does the Cut-Extrude feature do?

Answer: The Cut-Extrude feature removes material.

11 How do you change a dimension value?

Answer: Double-click on the dimension. Enter the new value in the **Modify** dialog box.

Lesson 2 — 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 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?

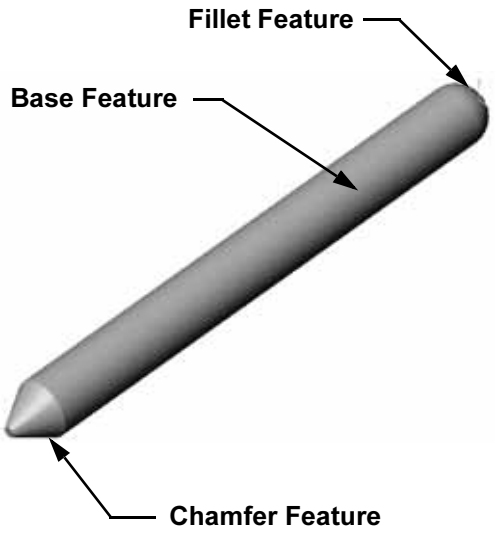
_____

In Class Discussion — Describing the Base Feature

Pick up a pencil. Ask the students to describe the base feature of the pencil. How would you create the additional features for the pencil?

Answer

- ❑ Sketch a circular 2D profile.
- ❑ Extrude the 2D sketch. This creates the base feature which is named `Extrude1`.
- ❑ Select one circular edge on the base feature. Create a fillet feature. The fillet feature removes sharp edges. The fillet feature creates the eraser for the pencil.
- ❑ Select the other circular edge on the base feature. Create a chamfer feature. The chamfer feature creates the point for the pencil.



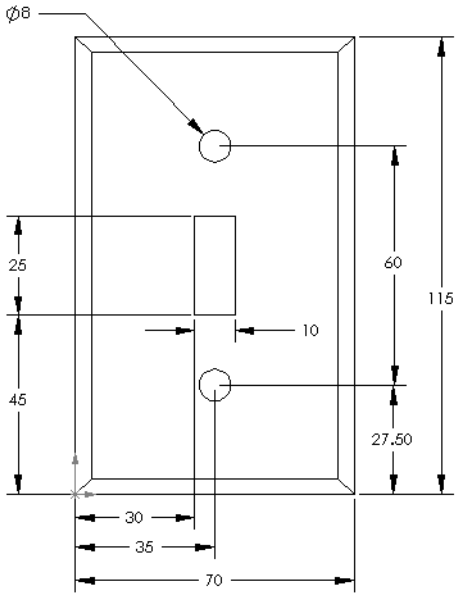
Exercises and Projects — Designing a Switch Plate

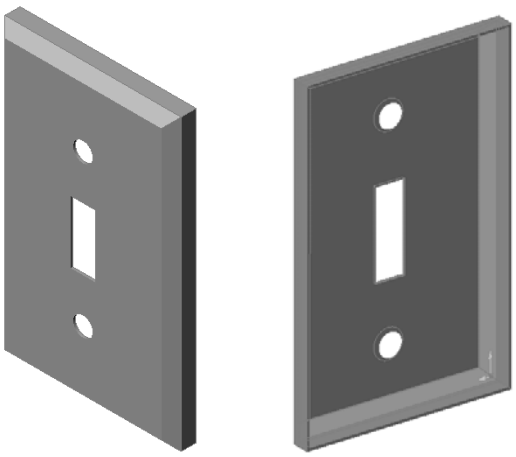
Switch plates are required for safety. They cover live electrical wires and protect people from electric shock. Switch plates are found in every home and school.

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

Tasks

1. Measure a single light plate switch cover.
   Answer: Overall a single switch plate is approximately 70mm x 115mm x 10mm. The switch cut-out is approximately 10mm x 25mm.
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: It is an extruded boss feature.





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: The extruded boss, chamfer, shell and extruded cut features are used to create the `switchplate`.

- The order in which the features are created is important.

  First – create the base feature.

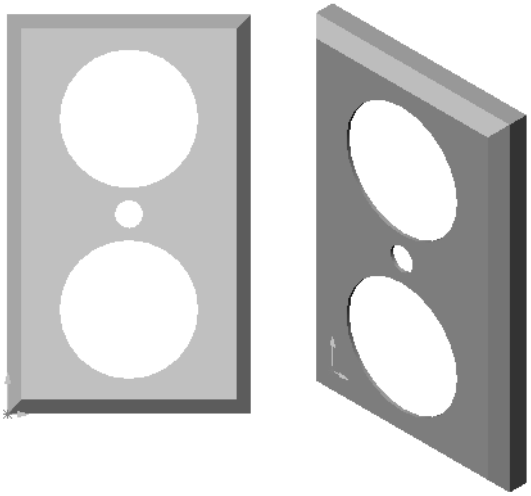
  Second – create the chamfer feature.

  Third – create the shell feature.

  Fourth – create the cut feature for the switch hole.

  Fifth – create the cut feature for the screw holes.

- The file `switchplate.sldprt` is found in `Lessons\Lesson2` in the `SolidWorks Teacher Tools` folder.



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

Answer: The `outletplate.sldprt` file is found in `Lessons\Lesson2` in the `SolidWorks Teacher Tools` folder.

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

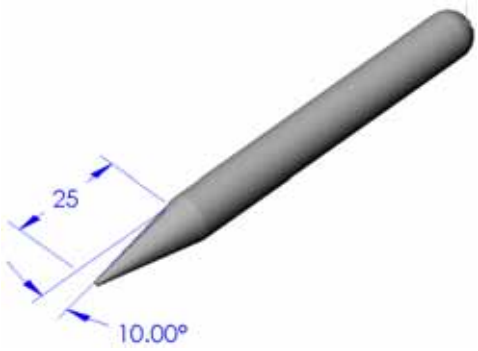
More to Explore — Modifying a Part

Many pencils have a longer, sharper point than the one shown earlier. How can this be accomplished?

Answer

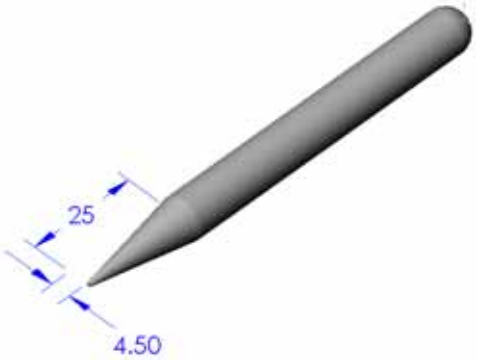
Answers will vary. One possibility is:

- Double-click chamfer feature, either in the FeatureManager design tree or the graphics area.
- Change the angle to **10°**.
- Change the distance to **25mm**.
- Click **Rebuild** on the Standard toolbar to rebuild the part.



Another possibility is:

- Edit the definition of the chamfer feature.
- Change the **Type** option to **Distance-Distance**.
- Set the **Distance1** value to **25mm**.
- Set the **Distance2** value to **4.5mm**.
- Click **OK** to rebuild the chamfer feature.



Lesson 2 Vocabulary Worksheet — Answer Key

Name: _____Class: _____ Date:_____

Fill in the blanks with the words that are defined by the clues.

1. The corner or point where edges meet: **vertex**
2. The intersection of the three default reference planes: **origin**
3. A feature used to round off sharp corners: **fillet**
4. The three types of documents that make up a SolidWorks model: **parts, assemblies, drawings**
5. A feature used to hollow out a part: **shell**
6. Controls the units, grid, text, and other settings of the document: **template**
7. Forms the basis of all extruded features: **sketch**
8. Two lines that are at right angles (90°) to each other are: **perpendicular**
9. The first feature in a part is called the **base** feature.
10. The outside surface or skin of a part: **face**
11. A mechanical design automation software application: **SolidWorks**
12. The boundary of a face: **edge**
13. Two straight lines that are always the same distance apart are: **parallel**
14. Two circles or arcs that share the same center are: **concentric**
15. The shapes and operations that are the building blocks of a part: **features**
16. A feature that adds material to a part: **boss**
17. A feature that removes material from a part: **cut**
18. An implied centerline that runs through the center of every cylindrical feature: **axis**

Lesson 2 Vocabulary Worksheet REPRODUCIBLE

Name: _____Class: _____ Date:_____

Fill in the blanks with the words that are defined by the clues.

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: _____

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:_____

Lesson 2 Quiz — 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. You build parts from features. What are features?
   Answer: Features are the shapes (bosses, cuts and holes) and the operations (fillets, chamfers and shells) that are use to build a part.
2. Name the features that are used to create the `box` in Lesson 2.
   Answer: Extruded Boss, Fillet, Shell and Extruded Cut.
3. How do you begin a new part document?
   Answer: Click the **New** tool or click **File, New**. Select a part template.
4. Give two examples of shape features that require a sketched profile.
   Answer: Shape features are Extruded Boss, Extruded Cut, and Hole.
5. Give two examples of operation features that require a selected edge or face.
   Answer: Operation features are Fillet, Chamfer and Shell.
6. Name the three documents that make up a SolidWorks model.
   Answer: Parts, assemblies and drawings
7. What is the default sketch plane?
   Answer: The default sketch plane is `Front`.
8. What is a plane?
   Answer: A plane is a flat 2D surface.
9. How do you create an extruded boss feature?
   Answer: Select a sketch plane. Open a new sketch. Sketch the profile. Extrude the profile perpendicular to the sketch plane.
10. Why do you create and use document templates?
    Answer: Document templates contain the units, grid and text settings for the model. You can create Metric and English templates, each with different settings.

Lesson 2 Quiz 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 You build parts from features. What are features? _____

_____

2 Name the features that are used to create the `box` in Lesson 2._____

_____

3 How do you begin a new part document? _____

_____

4 Give two examples of shape features that require a sketched profile. _____

_____

5 Give two examples of operation features that require a selected edge or face. _____

_____

6 Name the three documents that make up a SolidWorks model. _____

_____

7 What is the default sketch plane? _____

_____

8 What is a plane? _____

_____

9 How do you create an extruded boss feature? _____

_____

10 Why do you create and use document templates? _____

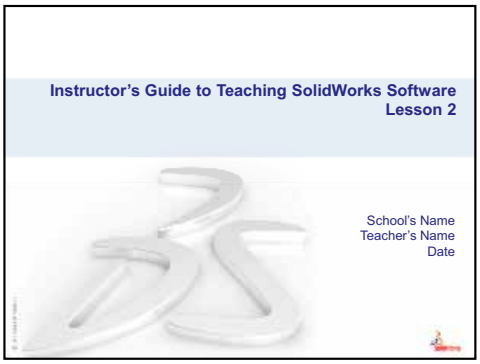
_____

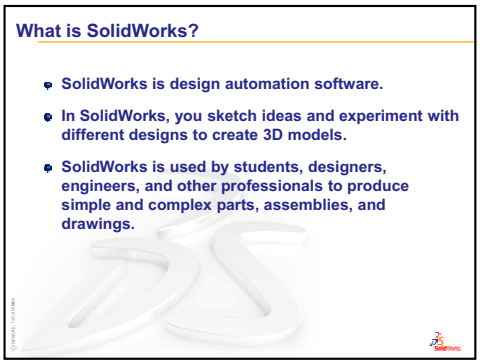
Lesson Summary

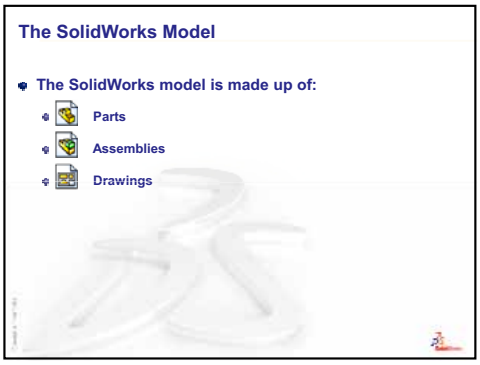
- SolidWorks is design automation software.
- The SolidWorks model is made up of:
  Parts
  Assemblies
  Drawings
- Features are the building blocks of a part.

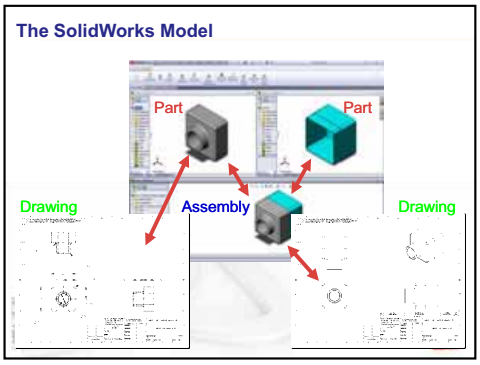
Thumbnail Images of PowerPoint Slides

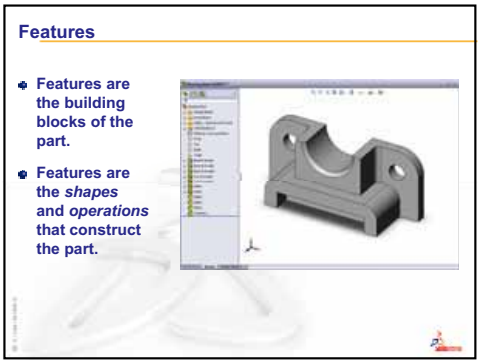
The following thumbnail images, arranged left to right, show the PowerPoint slides provided with this lesson.

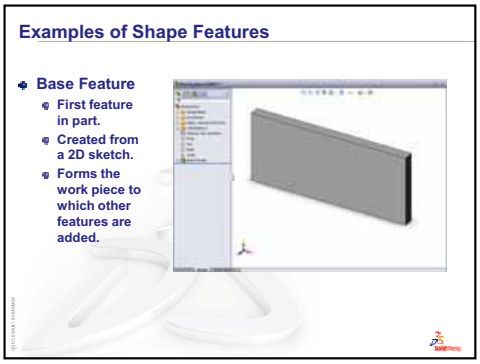






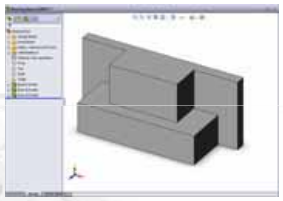






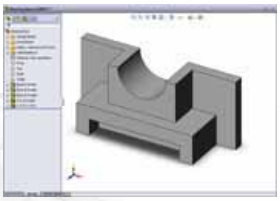
Examples of Shape Features

- Boss feature
  - Adds material to part.
  - Created from 2D sketch.



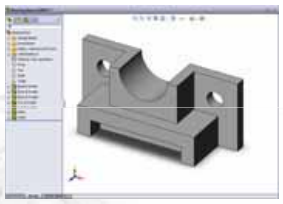
Examples of Shape Features

- Cut feature
  - Removes material from part.
  - Created from 2D sketch.



Examples of Shape Features

- Hole feature
  - Removes material.
  - Works like more intelligent cut feature.
  - Corresponds to process such as counter-sink, thread, counter-bore.

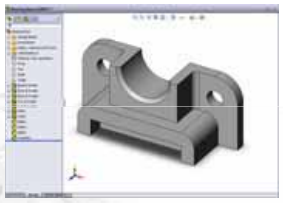


Examples of Shape Features

- Fillet feature
  - Used to round off sharp edges.
  - Can remove or add material.
    - Outside edge (convex fillet) removes material.
    - Inside edge (concave fillet) adds material.

Examples of Shape Features

- Chamfer feature
  - Similar to a fillet.
  - Bevels an edge rather than rounding it.
  - Can remove or add material.

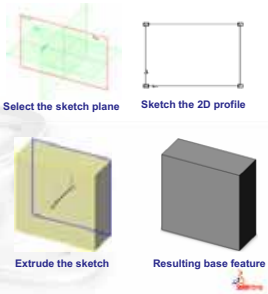


Sketched Features & Operation Features

- Sketched Features
  - Shape features have sketches.
  - Sketched features are built from 2D profiles.
- Operation Features
  - Operation features do not have sketches.
  - Applied directly to the work piece by selecting edges or faces.

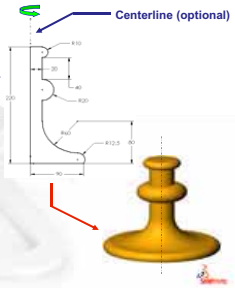
To Create an Extruded Base Feature:

1. Select a sketch plane.
2. Sketch a 2D profile.
3. Extrude the sketch perpendicular to sketch plane.



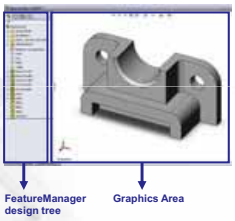
To Create a Revolved Base Feature:

1. Select a sketch plane.
2. Sketch a 2D profile.
3. Sketch a centerline (optional).
4. Revolve the sketch around a sketch line or centerline.

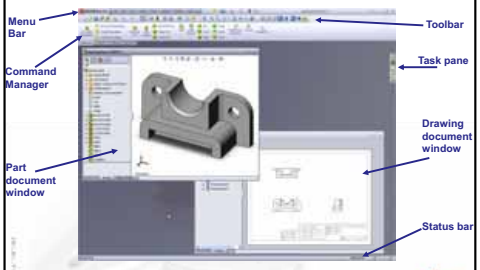


Terminology: Document Window

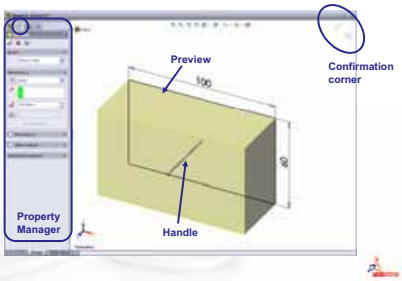
- Divided into two panels:
  - Left panel contains the FeatureManager® design tree.
    - Lists the structure of the part, assembly or drawing.
  - Right panel contains the Graphics Area.
    - Location to display, create, and modify a part, assembly or drawing.



Terminology: User Interface

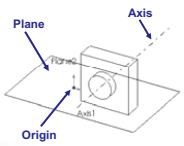


Terminology: PropertyManager



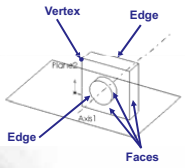
Terminology: Basic Geometry

- Axis - An implied centerline that runs through every cylindrical feature.
- Plane - A flat 2D surface.
- Origin - The point where the three default reference planes intersect. The coordinates of the origin are:
  (x = 0, y = 0, z = 0).



Terminology: Basic Geometry

- Face – The surface or "skin" of a part. Faces can be flat or curved.
- Edge – The boundary of a face. Edges can be straight or curved.
- Vertex – The corner where edges meet.



Features and Commands

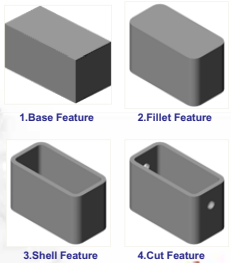
Base feature

- The Base feature is the first feature that is created.
- The Base feature is the foundation of the part.
- The Base feature geometry for the box is an extrusion.
- The extrusion is named Extrude1.

Features and Commands

Features used to build the *box* are:

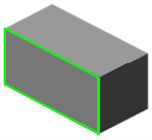
- Extruded Base feature
- Fillet feature
- Shell feature
- Extruded Cut feature



Features and Commands

To create the extruded base feature for the *box*:

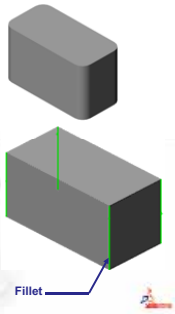
- Sketch a rectangular profile on a 2D plane.
- Extrude the sketch.
- By default extrusions are perpendicular to the sketch plane.



Features and Commands

Fillet feature

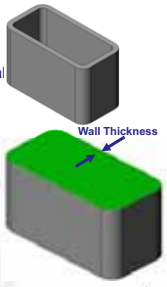
- The fillet feature rounds the edges or faces of a part.
- Select the edges to be rounded. Selecting a face rounds all the edges of that face.
- Specify the fillet radius.



Features and Commands

Shell feature

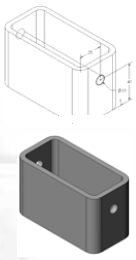
- The shell feature removes material from the selected face.
- Using the shell feature creates a hollow box from a solid box.
- Specify the wall thickness for the shell feature.



Features and Commands

To create the extruded cut feature for the *box*:

- Sketch the 2D circular profile.
- Extrude the 2D Sketch profile perpendicular to the sketch plane.
- Enter Through All for the end condition.
- The cut penetrates through the entire part.

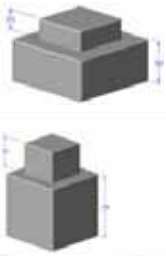


Dimensions and Geometric Relationships

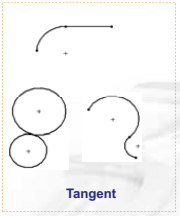
- Specify dimensions and geometric relationships between features and sketches.
- Dimensions change the size and shape of the part.
- Mathematical relationships between dimensions can be controlled by equations.
- Geometric relationships are the rules that control the behavior of sketch geometry.
- Geometric relationships help capture design intent.

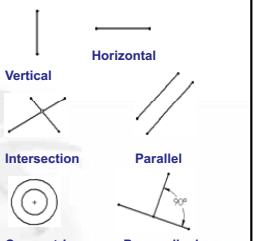
Dimensions

- Dimensions
  - Base depth = 50 mm
  - Boss depth = 25 mm
- Mathematical relationship
  - Boss depth = Base depth ÷ 2

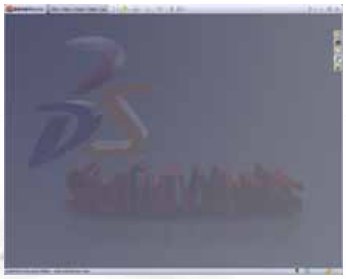


Geometric Relationships



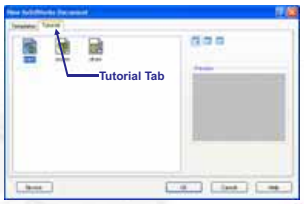


The SolidWorks Window



Creating New Files Using Templates

- Click New on the Standard toolbar.
- Select a document template:
  - Part
  - Assembly
  - Drawing



Document Templates

- Document Templates control the units, grid, text, and other settings for the model.
- The Tutorial document templates are required to complete the exercises in the *Online Tutorials*.
- The templates are located in the Tutorial tab on the New SolidWorks Document dialog box.
- Document properties are saved in templates.

Document Properties

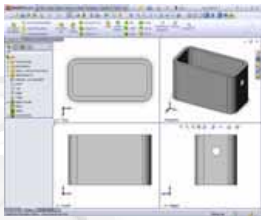
- Accessed through the Tools, Options menu.
- Control settings like:
  - Units: English (inches) or Metric (millimeters)
  - Grid/Snap Settings
  - Colors, Material Properties and Image Quality

System Options

- Accessed through the Tools, Options menu.
- Allow you to customize your work environment.
- System options control:
  - File locations
  - Performance
  - Spin box increments

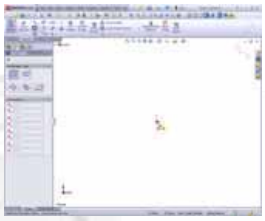
Multiple Views of a Document

- Click the view pop-up menu.
- Select an icon. The viewport icons include:
  - Single View
  - Two View (horizontal and vertical)
  - Four View



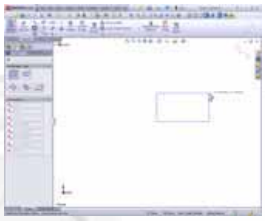
Creating a 2D Sketch

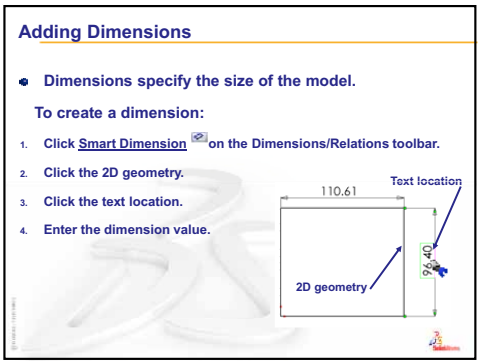
1. Click Sketch on the Sketch toolbar.
2. Select the Front plane as a sketch plane.
3. Click Rectangle on the Sketch Tools toolbar.
4. Move the pointer to the Sketch Origin.



Creating a 2D Sketch

5. Click the left mouse button.
6. Drag the pointer up and to the right.
7. Click the left mouse button again.




Adding Dimensions
Dimensions specify the size of the model.
To create a dimension:
1. Click Smart Dimension on the Dimensions/Relations toolbar.
2. Click the 2D geometry.
3. Click the text location.
4. Enter the dimension value.
110.61
Text location
2D geometry
96.40