SolidWorks®

Engineering Design Project
The Mountainboard

Teacher Resources

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Contents

I

Introduction

Purpose of this Document

The material included in this document is intended for the use of the SolidWorks teachers/instructors.

Information contained here includes the different methods of teaching this course, answers to the various tests within the curriculum and additional resources that are provided to support the instruction.

Course Organization

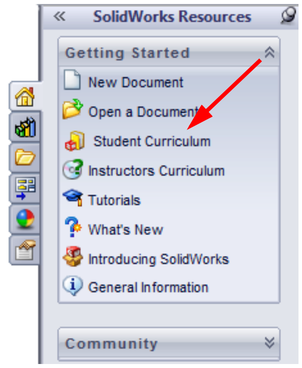
The course is based around a design project using SolidWorks. In addition to the material in the course, the online tutorials can be used to supplement the training. Depending on the total hours of class, laboratory time and homework time allotted, as well as the availability of SolidWorks to the students; the tutorials:

- can be assigned as homework.
- taught during class with the mountainboard project as the laboratory exercises.
- can be done to supplement the lessons with the mountainboard as the focus of class time.

Education Edition Curriculum and Courseware

All material for the Mountainboard course is provided by download.

Course material for the students can be downloaded from within SolidWorks. Click the SolidWorks Resources tab in the Task Pane and then select Student Curriculum.

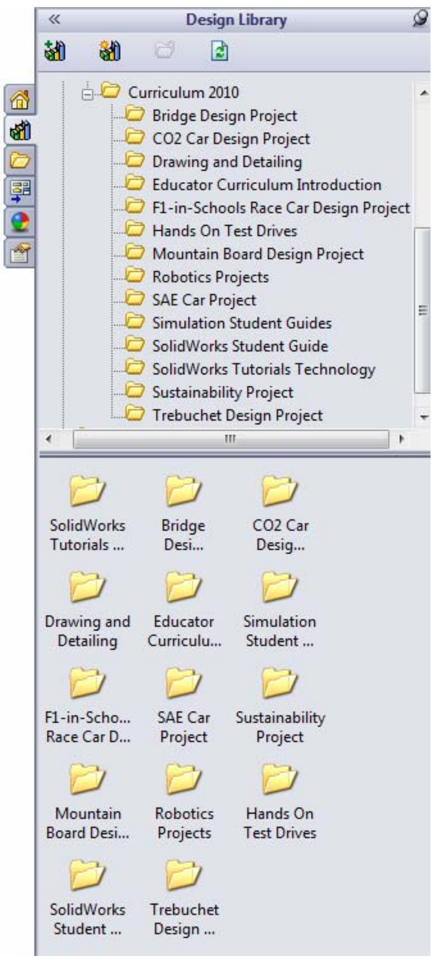


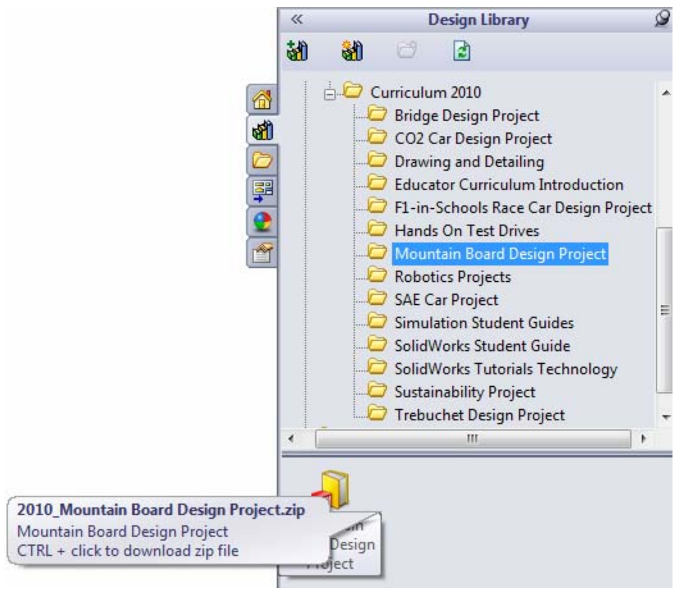
Select the curriculum for the current year.

Double-click the course you would like to download. Control-select the course to download a ZIP file.

The `Lessons` file contains the folder structure and the parts needed to complete the lessons.

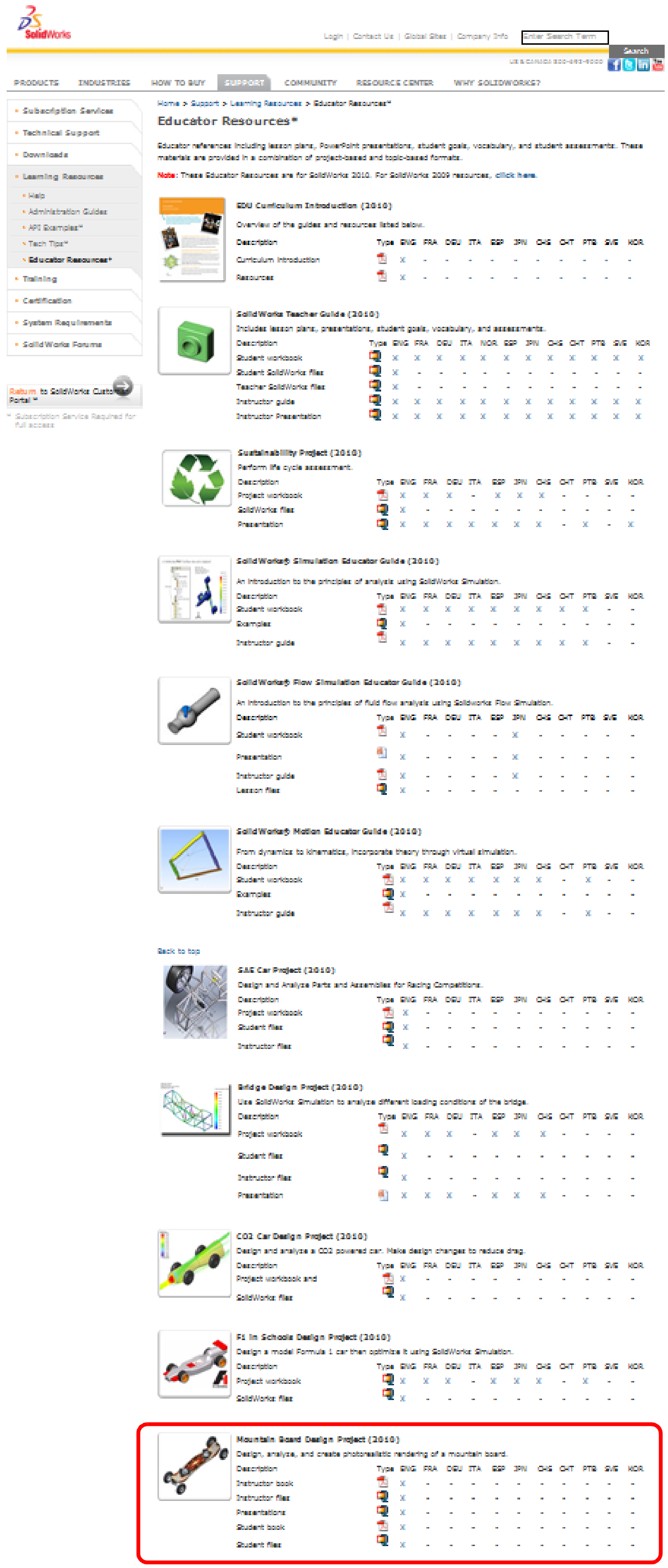
The `Student Guide` contains the PDF file of the course.



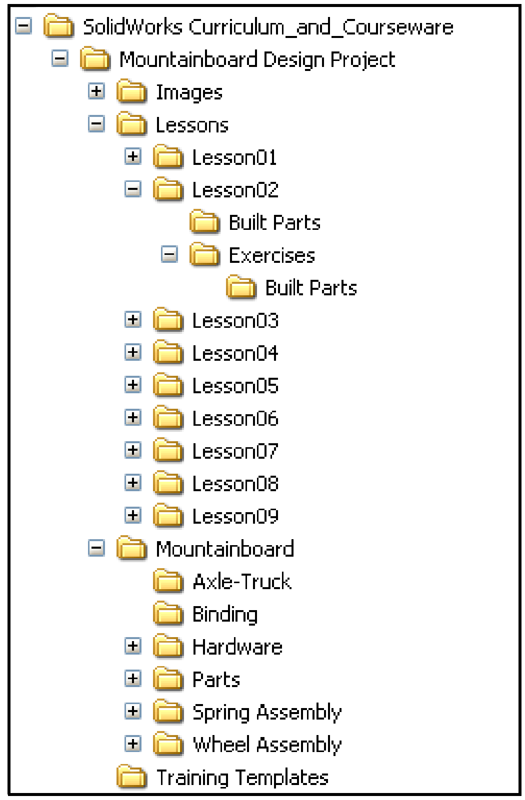


Unzipping the ZIP file creates a folder named `SolidWorks Curriculum_and_Courseware_2011`. This folder contains directories for this course.

Course material for teachers can also be downloaded from the SolidWorks web site. Click the SolidWorks Resources tab in the Task Pane and then select Instructors Curriculum. This will take you to the Educator Resources page shown at right.



The folder Mountainboard Design Project is used for this course and contains the following items:



Student Resources

- Lessons — there are folders corresponding to the lesson in the *Mountainboard Design Project* course. These folders contain copies of some of the parts referred to in the lessons, as well as the student exercises. Only parts that the students need for the assemblies, but are not required to create, have been provided. Students can save the parts, assemblies and drawings they create during the course in the appropriate lesson folder.
- Training Templates — a folder containing the part, assembly, and drawing templates students need to do the active learning exercises.
- Images — a folder containing the images used to create PhotoWorks appearances, decals and scenes.
- Mountainboard — a folder provided as a place for students to save the individual mountainboard files they will create during the lessons. There are several sub-folders for the various sub-assemblies the students will create plus a folder with some additional hardware that will be needed to complete the assemblies.

Teacher Resources

- Lessons — there are folders corresponding to the lesson in the *Mountainboard Design Project* course. These folders contain copies of the parts, assemblies, and drawings referred to in the lessons, as well as the student exercises. Review the models contained in these folders before you present the lecture. Use them to assist in your instruction.

- Each Lesson and Exercises folder contains a sub-folder for Built Parts. Files contained in these folders represent the finished product of the lesson or exercise. To distinguish these files from files created by the teacher or student, they all have the suffix "_&".
- PowerPoint Slides — this folder contains PowerPoint slides for each lesson. A summery of the PowerPoint slides is included in this document. The Microsoft PowerPoint slides supplied here have slides for both the online tutorials and the mountainboard lessons. Depending on your approach to teaching the course, you may use all, some or none of the slides.

  You can reproduce these as student handouts, and modify them to suit your needs.
- Mountainboard-complete — a folder containing all the completed files of the mountainboard. Each file in this directory ends in "_&" to show that it is a completed file.

Operating System

This course has been modified to reflect the use of Windows 7® as it is currently the operating system most widely used.

Setting Up SolidWorks

SolidWorks **Options** must be set up to find the templates installed by the DVD.

After loading the SolidWorks Curriculum_and_Courseware_2011, start SolidWorks.

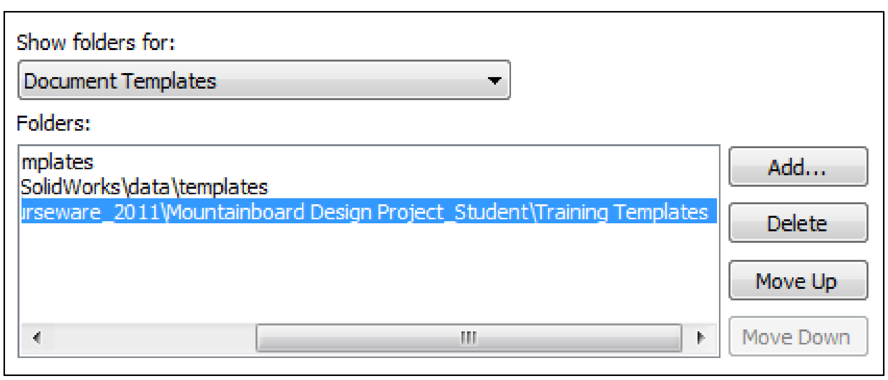
Click **Tools, Options**, then select **File Locations**.

Make sure **Document Templates** is showing in the **Show folders for** list.

Click **Add**.

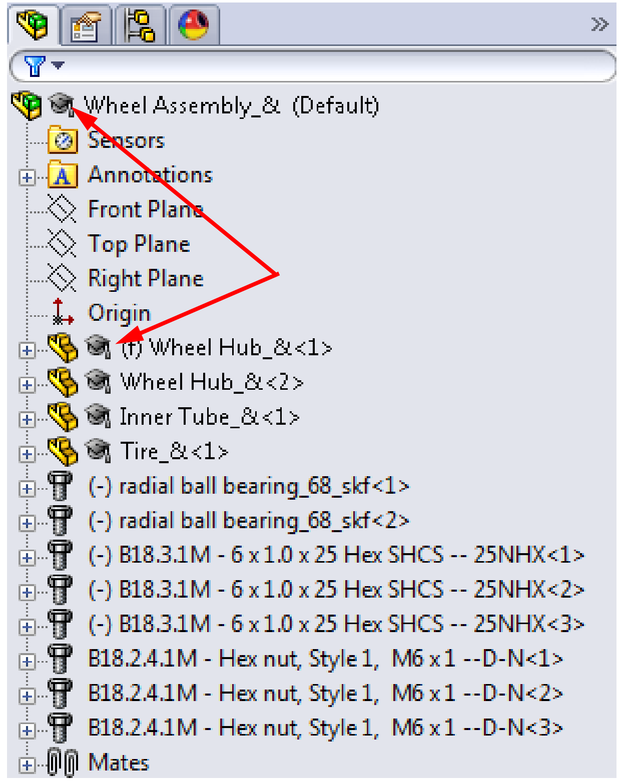
Navigate to the SolidWorks Curriculum_and_Courseware_2011\Mountainboard Design Project\Training Templates folder, then click **OK**.

Click **OK** to close the options.



Education Version Icon

Any file opened, modified and save in the EDU version of SolidWorks will show a mortarboard icon in the FeatureManager design tree.



Accessing SolidWorks Commands

There are many ways to access the different SolidWorks commands, such as:

- Individual toolbars
- Context toolbars
- CommandManager
- Main menus
- Right-click context menus
- Flyout menus
- “S” key menus
- Mouse Gestures
- Hot keys

Because of the large number of choices, many of the specific directions such as “Click **Insert, Sketch** from the menu” or “Click **Sketch** on the Sketch toolbar” have been replaced by just the command “Click **Sketch**”. This allows each teacher choose how to introduce the various methods.

1

Using the Interface

Purpose of this Document

This and following chapters provide the answers to:

- 5 Minute Assessment
- Lesson Reviews
- Lesson Quizzes
- Vocabulary Worksheet

Also included:

- Summary of PowerPoint slides available for the lesson

For some lessons, additional information is also provided with ideas on how to approach the material in the lesson.

Lesson 1

If your students are already experienced with the Microsoft Windows Graphical User Interface, you may wish to skip to the section of this lesson that familiarizes students with the SolidWorks user interface.

5 Minute Assessment – #1 Answer Key

1 Search for the SolidWorks part file `Paper Towel Base`. How did you find it?

Answer: Click , enter search criteria in the Search Box. Or, open Windows Explorer and select a drive or folder and type the file name in the Search Box.

2 What is the quickest way to search for a file?

Answer: Click , and type the name of the file in the Search Box.

3 How do you open the file from the **Search Results** window?
Answer: Double-click on the file name.

4 How do you start the SolidWorks program?

Answer: Click , **All Programs**, **SolidWorks**, **SolidWorks**.

5 What is the quickest way to start the SolidWorks program?
Answer: Double-click the SolidWorks desktop shortcut (if one exists).

Lesson 1 Vocabulary Worksheet — Answer Key

1 Shortcuts for collections of frequently used commands: **toolbars**

2 Command to create a copy of a file with a new name: **File, Save As**

3 One of the areas that a window is divided into: **panel**

4 The graphic representation of a part, assembly, or drawing: **model**

5 Character that you can use to perform wild card searches: **asterisk or ***

6 Area of the screen that displays the work of a program: **window**

7 Icon that you can double-click to start a program: **desktop shortcut**

8 Action that quickly displays menus of frequently used or detailed commands: **right-click**

9 Command that updates your file with changes that you have made to it: **Save**

10 Action that quickly opens a part or program: **double-click**

11 The program that helps you create parts, assemblies, and drawings: **SolidWorks**

12 Panel of the SolidWorks window that displays a visual representation of your parts, assemblies, and drawings: **graphics area**

13 Technique that allows you to find all files and folders that begin or end with a specified set of characters: **wild card search**

Lesson 1 Quiz – Answer Key

1 How do you start the SolidWorks application program?

Answer: Click , **All Programs**, **SolidWorks**, **SolidWorks**; or double-click on the SolidWorks desktop shortcut; or double-click on a SolidWorks file.

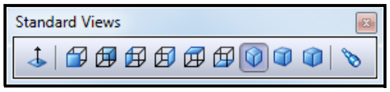
2 Which command would you use to create a copy of your file?

Answer: **File**, **Save As**

3 Where do you see a 3D representation of your model?

Answer: Graphics Area.

4 Look at the illustration (at right). What is this collection of frequently used commands called?



Answer: Toolbar

5 How would you find a file if you could not remember the whole file name?

Answer: Perform a wild card search.

6 Which command would you use to preserve changes that you have made to a file?

Answer: **File**, **Save**

7 Which character helps you perform a wild card search?

Answer: Asterisk or *

8 Circle the cursor that is used to resize a window.

Answer: 

9 Circle the cursor that is used to resize a panel.

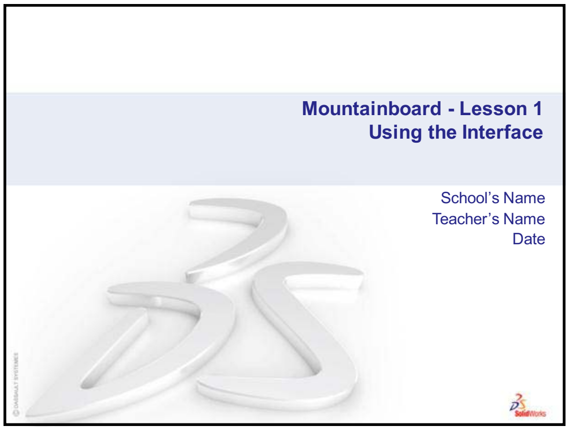
Answer: 

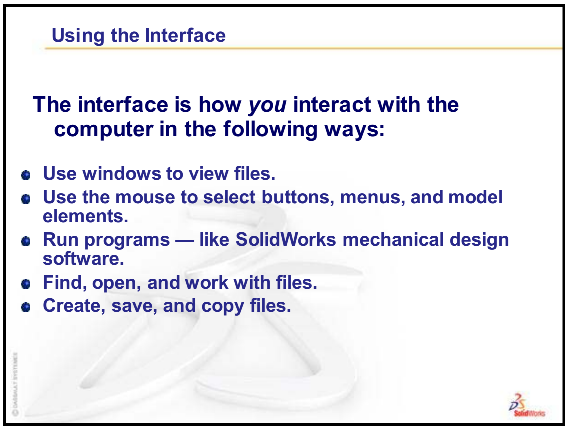
10 Circle the button that is used to get online help.

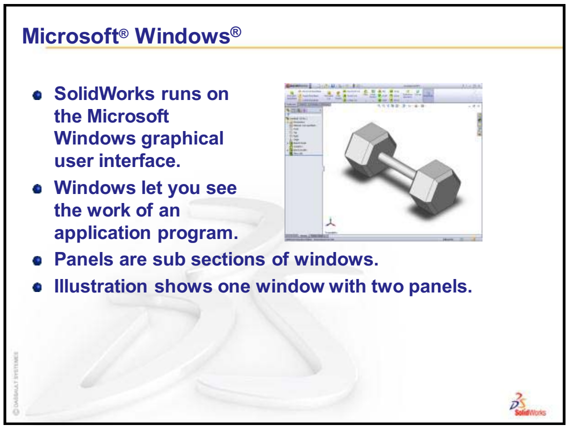
Answer: 

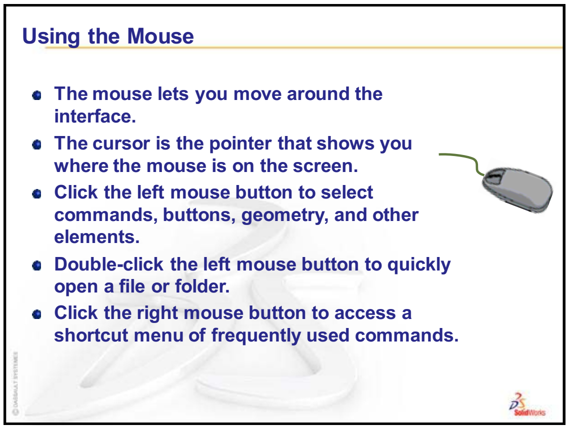
Thumbnail Images of PowerPoint Slides

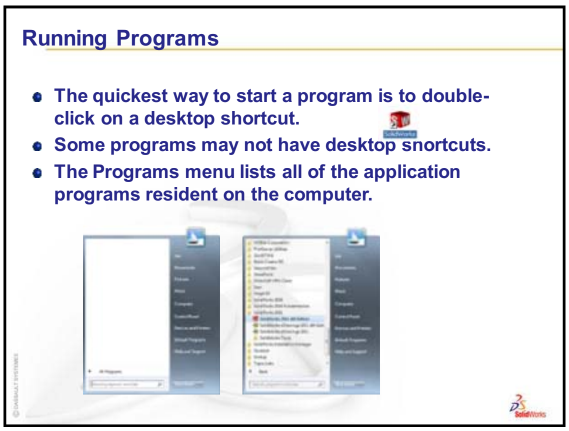
The following thumbnail images, arranged left to right, show the PowerPoint slides provided with this lesson.

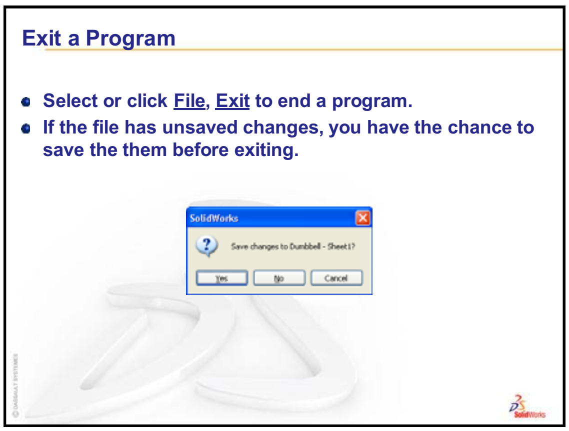












Searching for a File or Folder

- Type search criteria into search box.
- Use * to perform wild card searches.

Wild Card Searches

- Search for all files of a particular type by searching for the file type suffix.
  - Example: *.SLDPRT
- Search for all files that begin the same.
  - Example: *bearing**
- Search for all files that have common letters in the file name.
  - Example *plate*

Opening a File

- The quickest way to open a file is to double-click on it.
- The File menu displays your most recently used files.

Saving and Copying Files

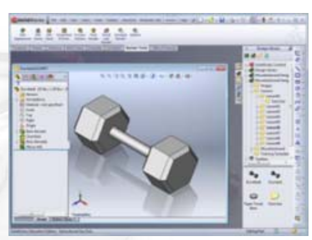
- Saving a file preserves the changes that you have made to it.
- Use File, Save As to copy a file.
- File, Save As creates an exact duplicate of the file as it existed at the moment that you copied it.

Resizing Windows

- Allows you to customize the appearance of your screen.
- View multiple files at the same time.
- Use ↔ ↕ ⤡ to change the size of a window.
- Use ⇹ ≑ to change the size of panels within a window.

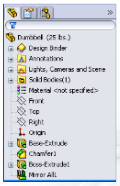
Using the SolidWorks Interface

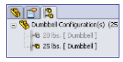
- SolidWorks windows display graphic and non-graphic model data.
- Toolbars display frequently used commands.



Left Side of SolidWorks Window

- FeatureManager design tree™
- Property Manager
- Configuration Manager

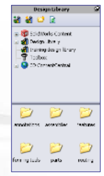




Right Side of SolidWorks Window

The Task Pane

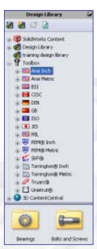
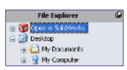
- SolidWorks Resources
- Design Library

Right Side of SolidWorks Window

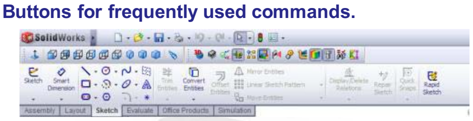
The Task Pane

- Toolbox
- File Explorer

Toolbars

Buttons for frequently used commands.

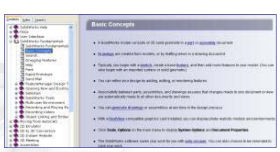


- You can select the toolbars to display.
- Toolbars are displayed at the top and sides of the window.
- You can also access the toolbars from the CommandManager.

Getting Help

To view comprehensive online help:

- Click ? .
- Select Help, SolidWorks Help.
- Help displays in a separate window.



2

Basic Functionality

5 Minute Assessment – #2 Answer Key

1 How do you start a SolidWorks session?

Answer: Click **Start**, 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 setting 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 `Binding Anchor`?

Answer: Extruded Boss, Fillet, 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: Select a plane or planar face, then click the Sketch icon on the Sketch toolbar.

8 What does the Fillet feature do?

Answer: The Fillet feature rounds sharp edges.

9 What tool calculates the volume of a part?

Answer: The Mass Properties tool.

10 What does the Cut-Extrude feature do?

Answer: The Cut-Extrude feature removes material.

11 How do you change an existing feature?

Answer: Right-click on the feature and select **Edit Feature**.

Lesson 2 Vocabulary Worksheet – Answer Key

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. Controls the units, grid, text, and other settings of the document: **template**
6. Forms the basis of all extruded features: **sketch**
7. Two lines that are at right angles (90°) to each other are: **perpendicular**
8. The first feature in a part is called the **base** feature.
9. The outside surface or skin of a part: **face**
10. A mechanical design automation software application: **SolidWorks**
11. The boundary of a face: **edge**
12. Two straight lines that are always the same distance apart are: **parallel**
13. Two circles or arcs that share the same center are: **concentric**
14. The shapes and operations that are the building blocks of a part: **features**
15. A feature that adds material to a part: **boss**
16. A feature that removes material from a part: **cut**
17. An implied centerline that runs through the center of every cylindrical feature: **temporary axis**

Lesson 2 Quiz — Answer Key

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 `Binding Anchor` in Lesson 2.

Answer: Extruded Boss, Fillet 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 an example of an operation feature that requires a selected edge or face.

Answer: Operation features are Fillet or Chamfer.

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 setting for the model. You can create Metric and English templates, each with different settings.

11 What is a section view?

Answer: A section view shows the part as if it were cut into two pieces. This displays the internal structure of the model.

Thumbnail Images of PowerPoint Slides

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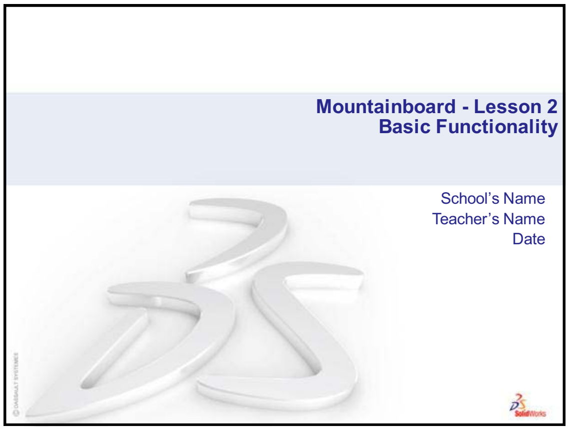
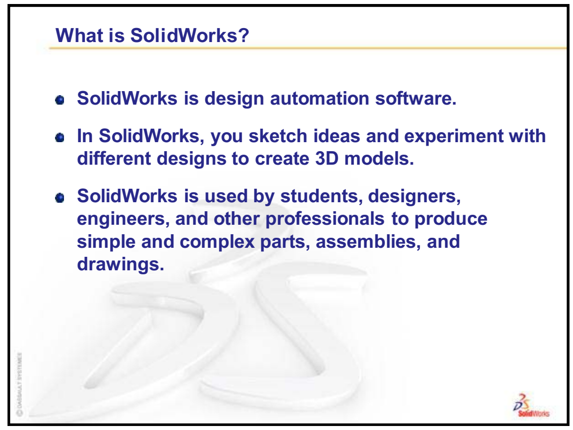
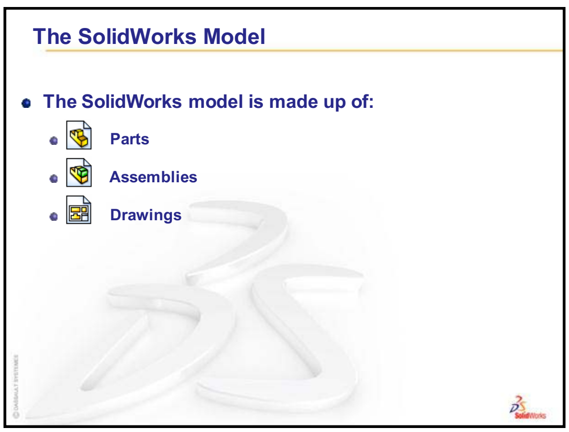
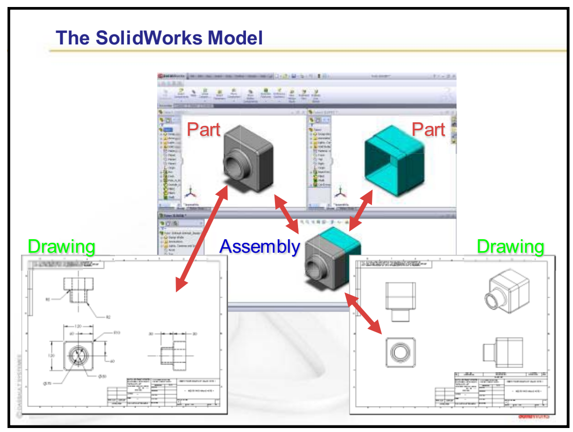
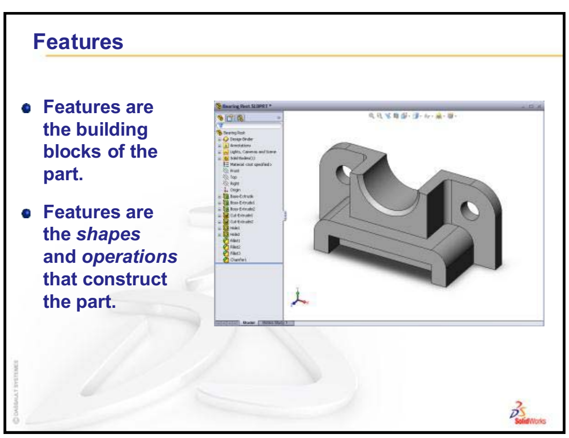
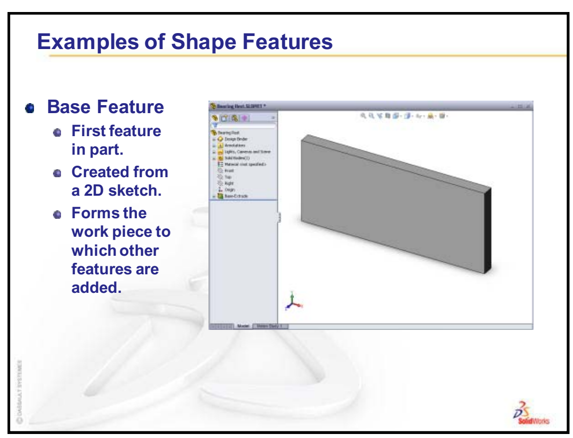






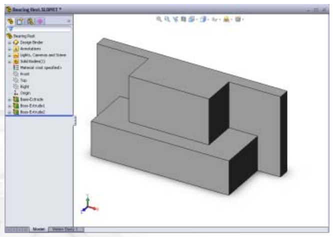






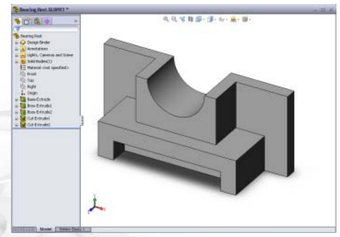
Examples of Shape Features

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



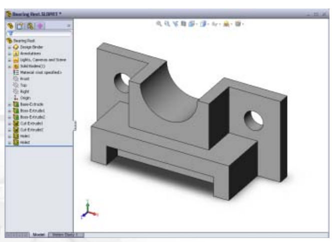
Examples of Shape Features

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



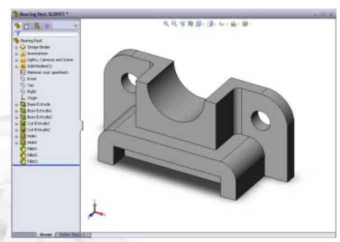
Examples of Shape Features

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



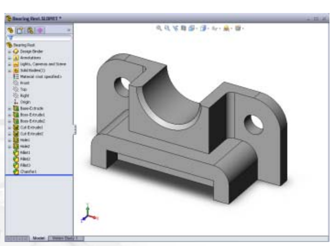
Examples of Shape Features

- 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.

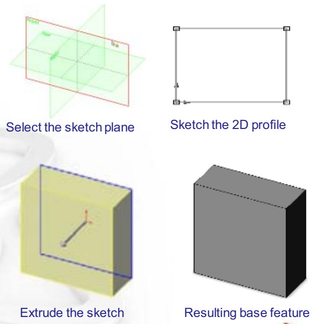


Sketched Features & Operation Features

- 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.

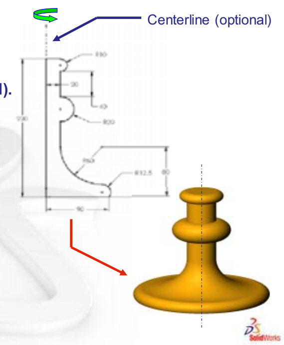
To Create an Extruded Base Feature:

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



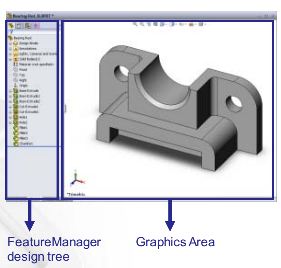
To Create a Revolved Base Feature:

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.

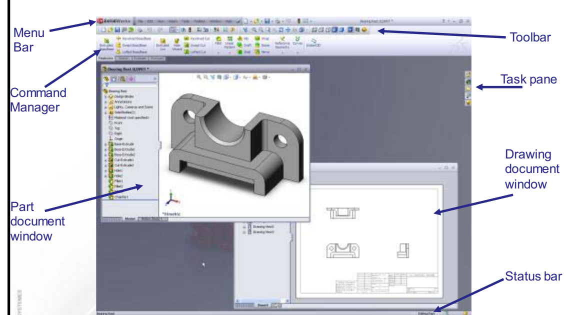


Terminology: Document Window

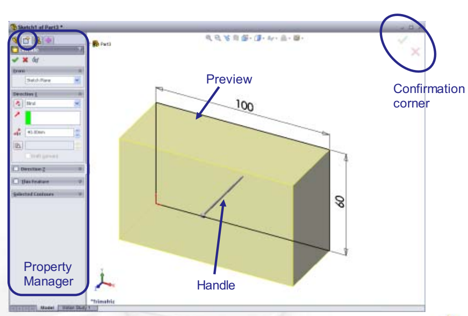
- Divided into two panels:
  - Left panel contains the FeatureManager® design tree.
    - Lists the structure of the part, assembly or drawing.
  - Right panel contains the Graphics Area.
    - Location to display, create, and modify a part, assembly or drawing.



Terminology: User Interface

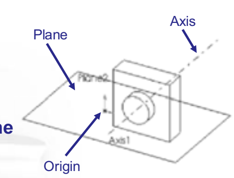


Terminology: PropertyManager



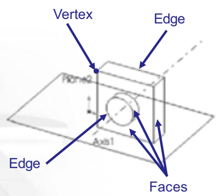
Terminology: Basic Geometry

- 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

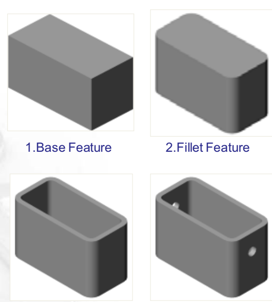
Base feature

- **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:

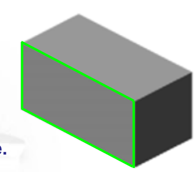
- **Extruded Base feature**
- **Fillet feature**
- **Shell feature**
- **Extruded Cut feature**


1.Base Feature
2.Fillet Feature
3.Shell Feature
4.Cut Feature

Features and Commands

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

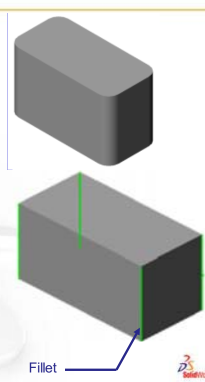
- **Sketch a rectangular profile on a 2D plane.**
- **Extrude the sketch.**
- **By default extrusions are perpendicular to the sketch plane.**



Features and Commands

Fillet feature

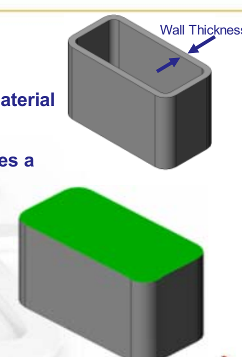
- **The fillet feature rounds the edges or faces of a part.**
- **Select the edges to be rounded. Selecting a face rounds all the edges of that face.**
- **Specify the fillet radius.**



Features and Commands

Shell feature

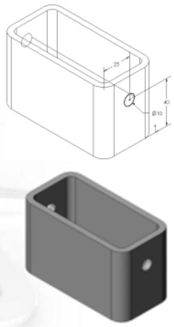
- **The shell feature removes material from the selected face.**
- **Using the shell feature creates a hollow box from a solid box.**
- **Specify the wall thickness for the shell feature.**



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.

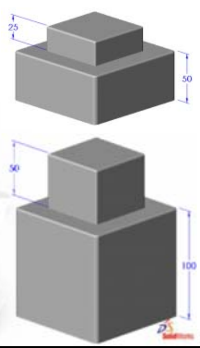


Dimensions and Geometric Relationships

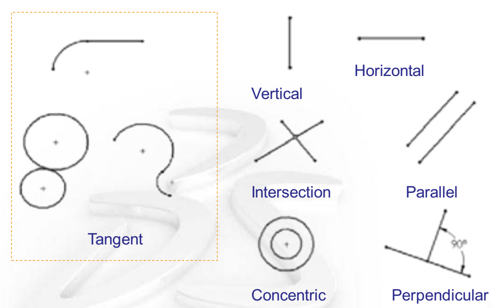
- Specify dimensions and geometric relationships between features and sketches.
- Dimensions change the size and shape of the part.
- Mathematical relationships between dimensions can be controlled by equations.
- Geometric relationships are the rules that control the behavior of sketch geometry.
- Geometric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

Dimensions

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



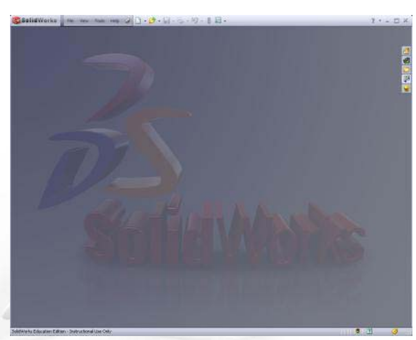
Geometric Relationships



To Start SolidWorks

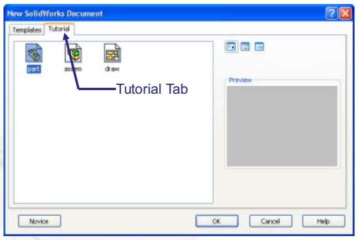
- Click the Start button on Windows task bar.
  - Click All Programs.
    - Click the SolidWorks folder.
      - Click the SolidWorks application.

The SolidWorks Window



Creating New Files Using Templates

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

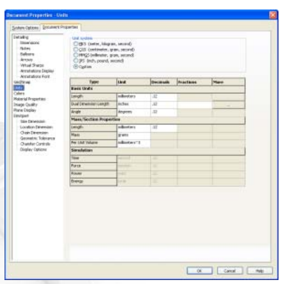


Document Templates

- **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.**

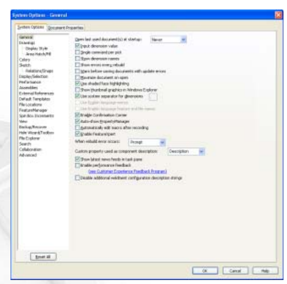
Document Properties

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



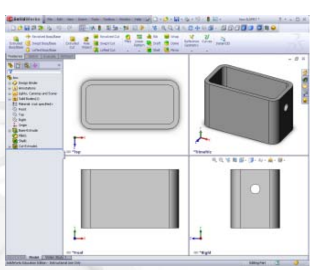
System Options

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



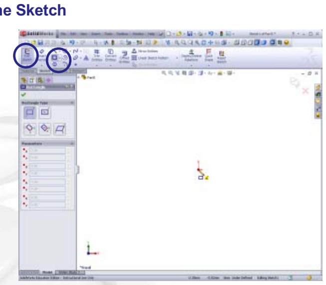
Multiple Views of a Document

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



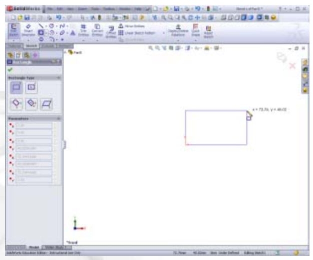
Creating a 2D Sketch

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.

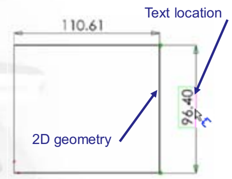


Adding Dimensions

- 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.



The Mountainboard

Design Process

- Project goals.
  - Design Intent
    - How does the model respond to change
    - Anticipated changes in the design
  - Analysis
    - Estimation of the lifecycle of the product
    - How will we insure that the parts are strong enough
  - Output
    - Engineering reports
    - Presentations
    - Marketing material

Key parts

- The Binding
  - Right and Left versions
  - Adjusts position
    - Rotation
    - Along Deck

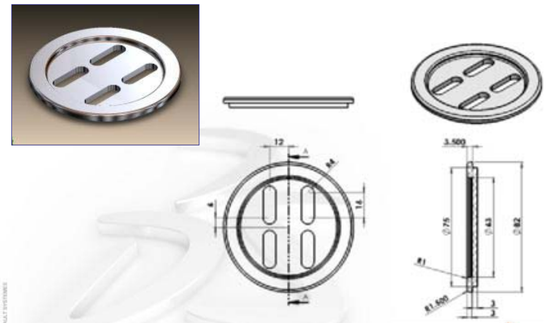
Key parts

- The Deck
  - Flexible
  - Mounting for Binding and Truck
  - Non-slip surface
  - Support a 100 kg rider
- Truck and Axle
  - Adjustable suspension
  - Mounting for optional braking system

Key parts

- Wheel Assembly
  - Easy to assemble
  - Tire and Tube are purchased parts
  - Sealed bearings
  - Mounting for optional braking system

Binding Anchor



Design Intent - Binding Anchor

- Clamps and Positions the Binding on the Deck
- Positioning
  - Along centerline
  - At angle to centerline
- No sharp edges to injure a rider

Determining the Weight

Weight = Volume X Density

- Volume can be calculated from the geometry of the model
- Density can be obtained from handbooks, data sheets or online

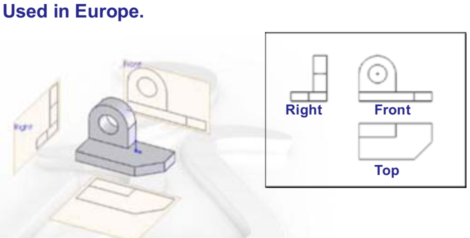
Mass Properties

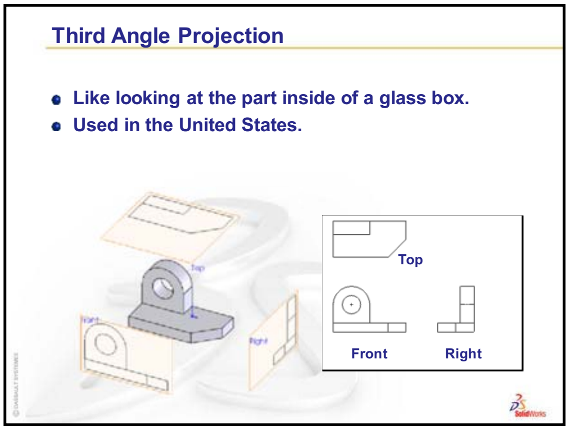
The Mass Properties tool can calculate:

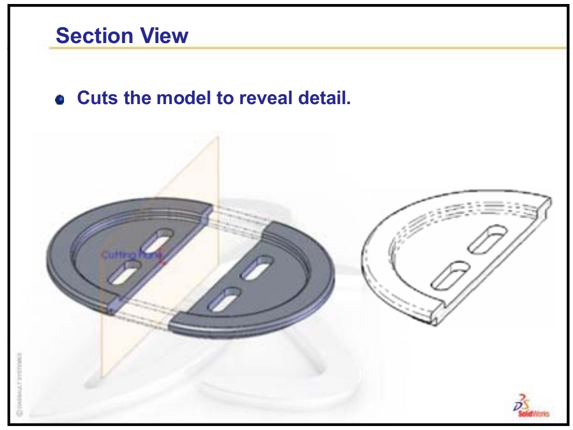
- Volume
- Mass
- Surface Area
- Center of Mass
- Moments of Inertia

First Angle Projection

- Like projecting the model on a screen behind the model.
- Used in Europe.




Third Angle Projection
Like looking at the part inside of a glass box.
Used in the United States.
Top
Front
Right


Section View
Cuts the model to reveal detail.

3

Basic Parts — The Binding

Review of Lesson 2: Basics

Questions for Discussion

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

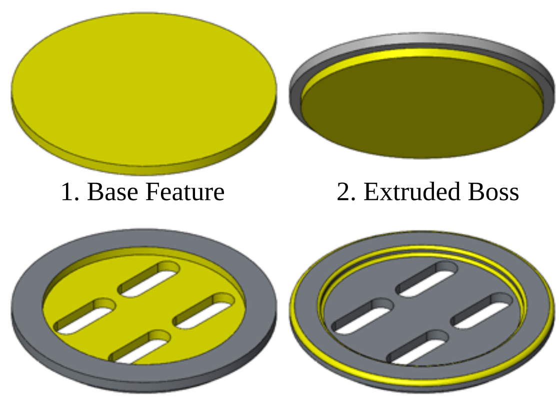
Answer: Part, Assembly and Drawing.

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 Binding Anchor in Lesson 2.

Answer: Extruded Boss, Extruded Cut, and Fillet.



1. Base Feature 2. Extruded Boss 3. Extruded Cuts 4. Fillets

4 What is the base feature of the Binding Anchor?

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

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 Binding Anchor.

6 How did you create the Base feature?

Answer: To create a solid Base feature:

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

5 Minute Assessment – #3 Answer Key

1 What features did you use to create Binding Base Plate?
Answer: Extruded Boss, Fillet, Insert Bends and Extruded Cut.

2 What does the Fillet feature do?
Answer: The Fillet feature rounds sharp edges and faces

3 Name three view commands in SolidWorks.
Answer: Zoom (to fit, to area, to selection), Rotate View, Pan and Roll.

4 Where are the display buttons located?
Answer: The display buttons are located on the View toolbar, Heads-up Standard View toolbar and Mouse Gestures.

5 Name the three SolidWorks default planes.
Answer: Front, Top, and Right.

6 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

7 True or False. In a fully defined sketch, geometry is displayed in black.
Answer: True.

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

9 Name the primary drawing views used to display a model.
Answer: Top, Front, Right and Isometric views.

Lesson 3 Quiz — Answer Key

1. How do you begin a new part document?
   Answer: Click the **New** icon. Select a part template.

2. How do you open a sketch?
   Answer: Select the desired sketch plane. Click the **Sketch** icon on the Sketch toolbar.

3. What is the Base feature?
   Answer: The base feature is the first feature of a part. It is the foundation of the part.

4. What color is the geometry of a fully defined sketch?
   Answer: Black

5. How can you change a dimension value?
   Answer: Double-click on the dimension. Enter the new value in the **Modify** dialog box.

6. What is the difference between an extruded boss feature and an extruded cut feature?
   Answer: The boss feature adds material. The cut feature removes material.

7. How do you extrude a cut so that the material outside the sketch is removed?
   Answer: Select **Flip Side to Cut**.

8. What is a fillet feature?
   Answer: The Fillet feature rounds the edges or faces of a part at a specified radius.

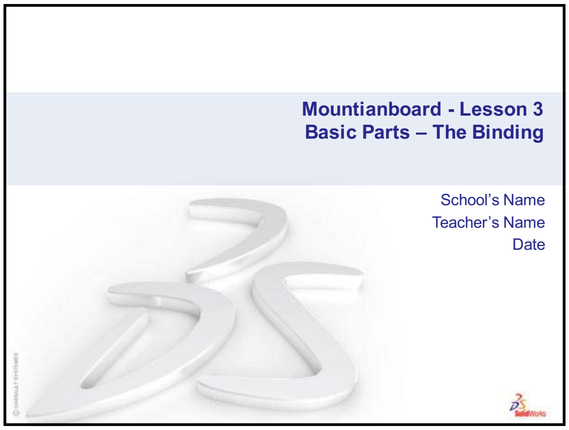
9. How do you start a new Assembly document?
   Answer: Click the **New** icon. Select an assembly template. Click **OK**.

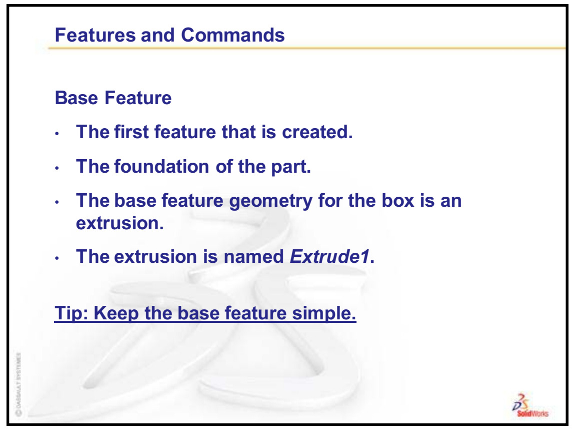
10. What are components?
    Answer: Components are parts contained in an assembly.

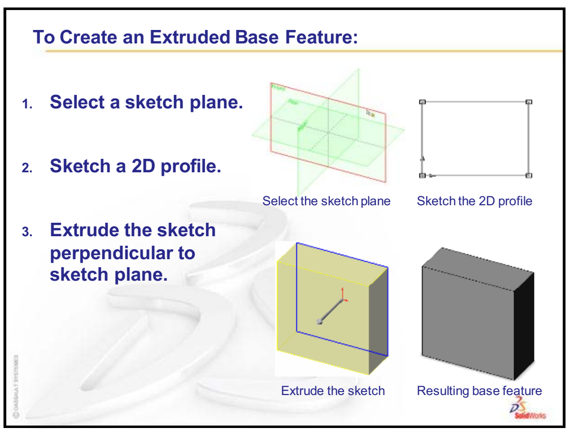
11. Name four types of geometric relations you can add to a sketch?
    Answer: The Geometric Relations you can add to a Sketch are: horizontal, vertical, collinear, coradial, perpendicular, parallel, tangent, concentric, midpoint, intersection, coincident, equal, symmetric, fix, pierce and merge points.

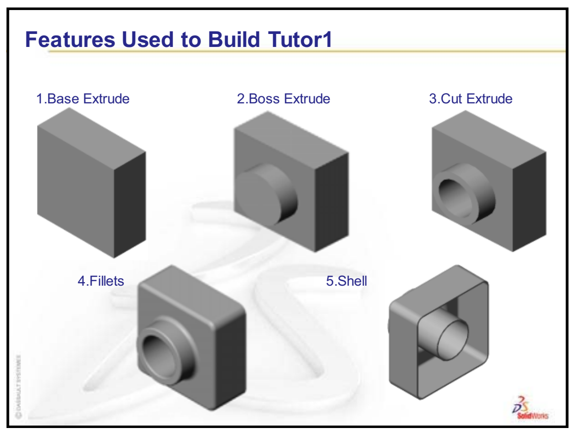
Thumbnail Images of PowerPoint Slides

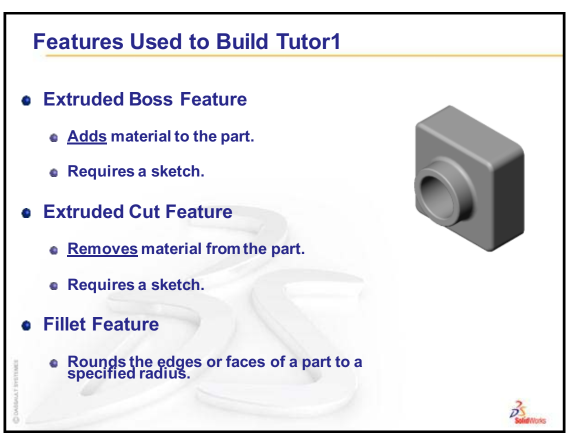
The following thumbnail images, arranged left to right, show the PowerPoint slides provided with this lesson.

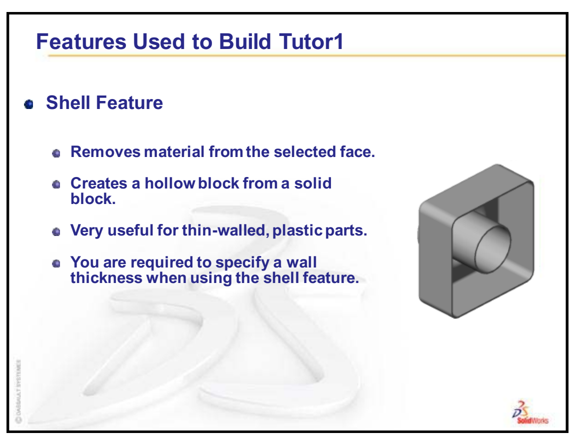












View Control

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

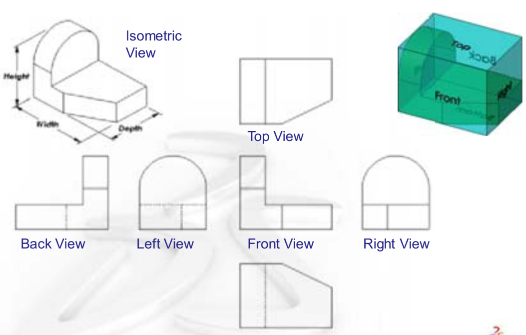
- **Zoom to Fit – displays the part so that it fills the current window.**
- **Zoom to Area – zooms in on a portion of the view that you select by dragging a bounding box.**
- **Zoom In/Out – drag the pointer upward to zoom in. Drag the pointer downward to zoom out.**
- **Zoom to Selection – the view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

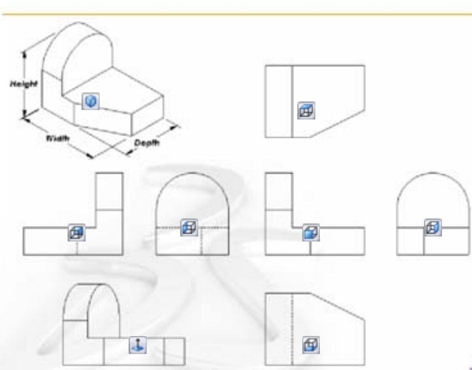
Standard Views



View Orientation

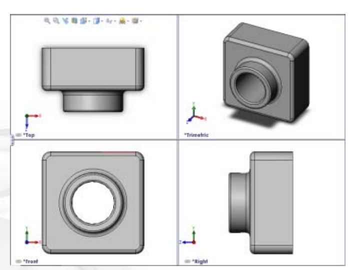
Changes the view display to correspond to one of the standard view orientations.

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



View Orientation

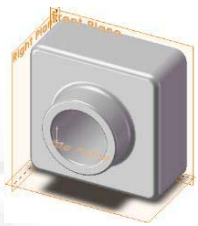
- **The views most commonly used to describe a part are:**
  - **Top View**
  - **Front View**
  - **Right View**
  - **Isometric View**



Default Planes

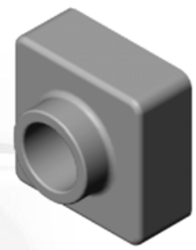
- **Default Planes**
  - Front, Top, **and** Right

Correspond to the standard principle drawing views:
  - Front **= Front or Back view**
  - Top **= Top or Bottom view**
  - Right **= Right or Left view**



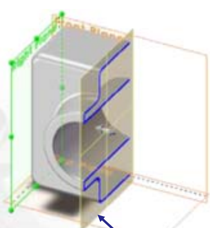
Isometric View

- **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.**



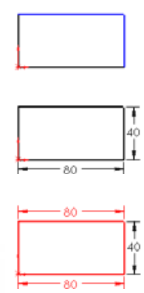
Section View

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



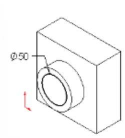
The Status of a Sketch

- **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).**



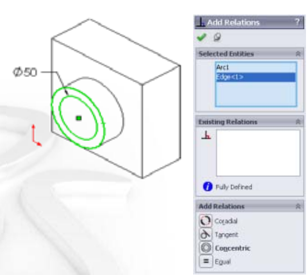
Geometric Relations

- **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.**



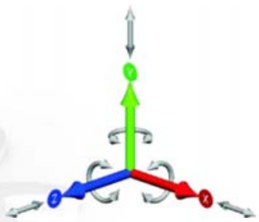
Geometric Relations

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



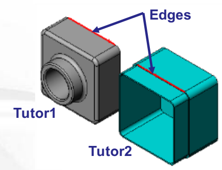
Degrees of Freedom: There are Six

- They describe how an object is free to move.
- Translation (movement) *along* X, Y, and Z axes.
- Rotation *around* X, Y, and Z axes.



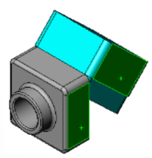
Mate Relationships

- Mates relationships align and fit together components in an assembly.
- The Tutor assembly requires three mates to fully define it.
  The three mates are:
- <u>Coincident</u> between the top back edge of Tutor1 and the edge of the lip on Tutor2.



Mate Relationships

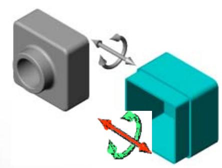
- Second Mate: <u>Coincident</u> mate between the right face of Tutor1 and the right face of Tutor2.



- Third Mate: <u>Coincident</u> mate between the top face of Tutor1 and the top face of Tutor2.

Mates and Degrees of Freedom

- The first mate removes all but two degrees of freedom.
- The remaining degrees of freedom are:
  - Movement *along* the edge.
  - Rotation *around* the edge.

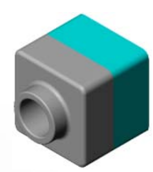


Mates and Degrees of Freedom

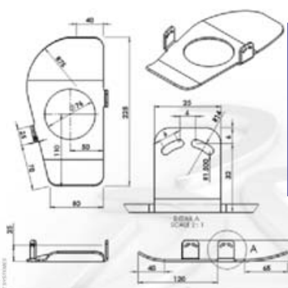
- The second mate removes one more degree of freedom.
- The remaining degree of freedom is:
  - Rotation *around* the edge.

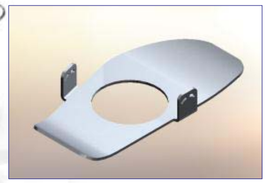
Mates and Degrees of Freedom

- The third mate removes last degree of freedom.
- No remaining degrees of freedom.
- The assembly is fully defined.



Binding Base Plate





Design Intent

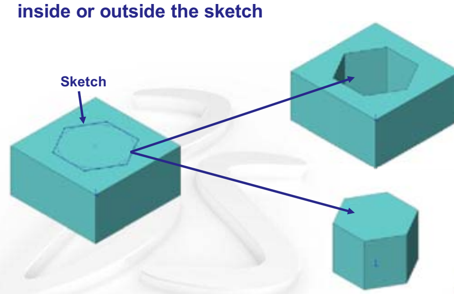
- There will be two versions: left foot and right foot
- Binding will be held in place by the Binding Anchor
- The front and back (toe and heel ends) will curve upward
- Side tabs will help to hold the foot centered
- Side tabs will have slots to attach the flexible straps that go over the foot

Manufacturing

- Part will be cut from flat piece and bent to final shape
- Use sheet metal functions to create a flat pattern

Flip Side To Cut

- An Extruded Cut can remove either the material inside or outside the sketch



4

Revolved Features — The Wheel Hub

Review of Lesson 3 — The Binding

Questions for Discussion

1 What are the two ways material can be removed with an Extruded Cut?

Answer: Material either inside the sketch or outside the sketch can be removed by either selecting or clearing **Flip Side to Cut**.

2 What is the primary requirement for a part that is to be turned into sheet metal with the command **Insert, Bends**?

Answer: The material must be uniform thickness.

3 What do mates do in an assembly?

Answer: They remove degrees of freedom.

4 When calculating Mass Properties of an assembly, how is the density of each part determined?

Answer: The material is applied to the individual parts. If no material is assigned to a part, a default value will be used.

5 Minute Assessment – #4 Answer Key

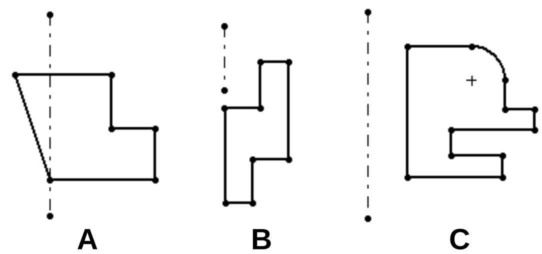
1 What special piece of sketch geometry is useful, but not required for a revolved feature?

Answer: A Centerline

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

Why? ____________________

Answer: Sketch **A** is not a valid sketch for a revolve feature because the profile crosses the centerline.



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

Answer: The **Convert Entities** sketch tool creates one or more curves in a sketch by projecting geometry onto the sketch plane.

4 In an assembly, parts are referred to as ____________________.

Answer: In an assembly, parts are referred to as components.

5 True or False. A fixed component is free to move?

Answer: False

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

Answer: True

7 How many components does an assembly contain?

Answer: An assembly contains two or more components.

8 In which window do you find ready-to-use hardware components?

Answer: The Toolbox folder in the Design Library.

9 True or False: Parts from Toolbox automatically size to the components they are being placed on.

Answer: False

10 True or False: Toolbox parts can only be added to assemblies.

Answer: True

Lesson 4 — Answer Key

1. How do you start a new Assembly document?

   Answer: Click the **New** icon. Select a assembly template. Click **OK**.

2. What are components?

   Answer: Components are parts contained in an assembly.

3. The **Convert Entities** sketch tool projects selected geometry onto the ________ plane?

   Answer: Current sketch.

4. True or False. Edges and faces can be selected items for Mates in an assembly.

   Answer: True.

5. A component in an assembly displays a (-) prefix in the FeatureManager. Is the component fully defined?

   Answer: No. A component that contains the (-) prefix is not fully fixed and still has some degrees of freedom. Additional mates are required if the component needs to be fixed.

6. What actions do you perform when an edge or face is too small to be selected by the pointer.

   Answer:

   - Use **Zoom** options from the View toolbar to increase the geometry size
   - Use **Selection Filters**
   - Right mouse click and click **Select Other**
   - Press G and use the Magnifying Glass

7. How do you establish a mate relationship between a Toolbox part and the part it is being placed on?

   Answer: The mate relationship is established when the Toolbox part snaps to the other part. You do not have to explicitly define the relationship.

8. How would you determine the correct length of a machine screw that fastens two parts using a washer, lock washer, and nut?

   Answer: Measure the thickness of both parts, the washer, the lock washer, and nut. Use a screw that is the next size longer so that the threads of the screw engage all of the threads of the nut.

9. How do you specify the location of a Toolbox part?

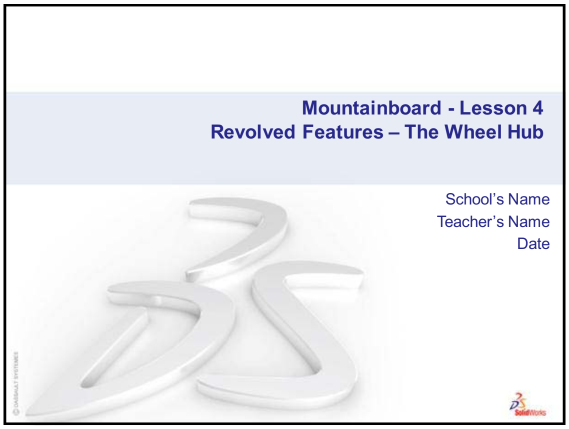
   Answer: You place Toolbox parts by dragging them and dropping them in the assembly.

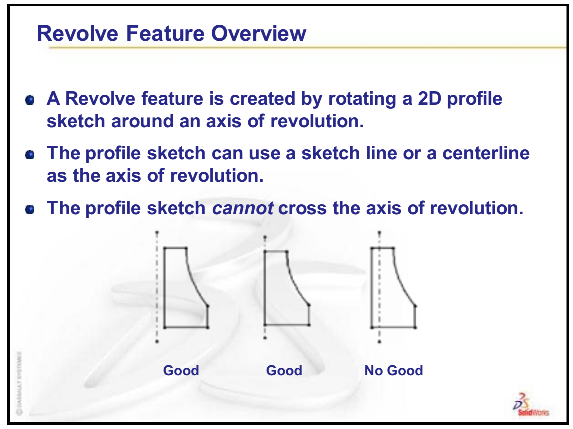
10. True or False. Screw threads are always displayed in Schematic mode — showing all details.

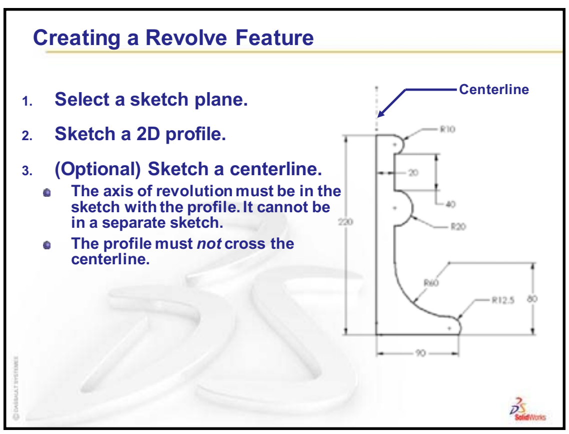
    Answer: False

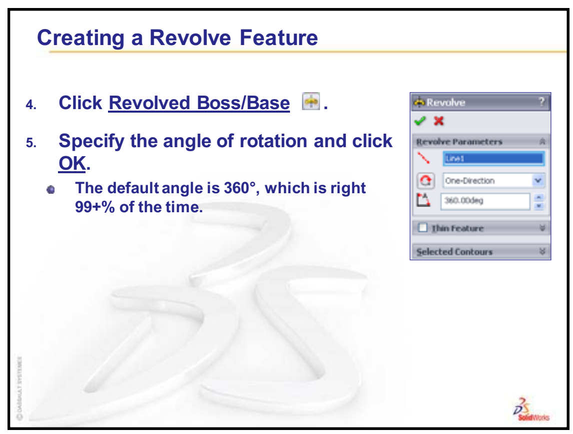
Thumbnail Images of PowerPoint Slides

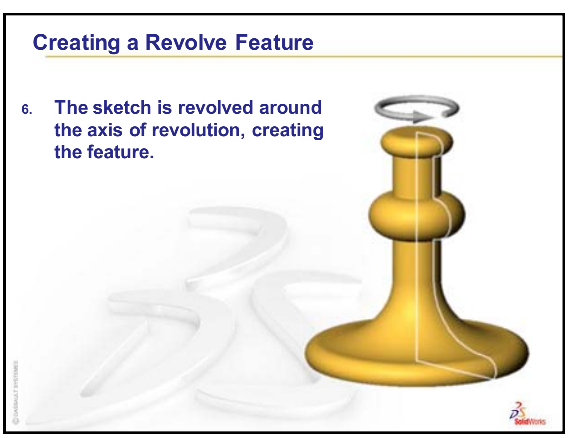
The following thumbnail images, arranged left to right, show the PowerPoint slides provided with this lesson.

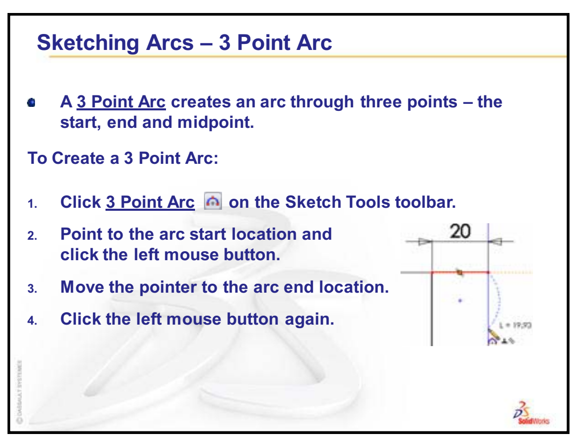






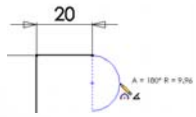






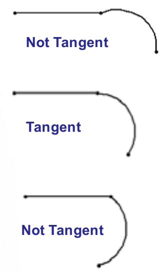
Creating a 3 Point Arc:

5. Drag the arc midpoint to establish the radius and direction (convex vs. concave).
6. Click the left mouse button a third time.



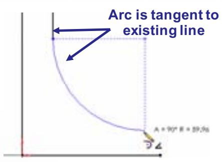
Sketching Arcs – Tangent Arc

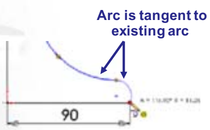
- The Tangent Arc tool creates an arc that has a smooth transition to an existing sketch entity.
- Saves the work of sketching an arc and then manually adding a geometric relation to make it tangent.
- Start point of the arc *must* connect to an existing sketch entity.



To Create a Tangent Arc:

1. Click Tangent Arc on the Sketch Tools toolbar.
2. Point to the arc start location, and click the left mouse button.
3. Drag to create the arc.
   - The arc angle and radius values are displayed on the pointer when creating arcs.
4. Click the left mouse button.





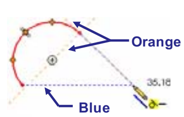
Pointer Feedback

- As you sketch, the pointer provides feedback and information about alignment to sketch entities and model geometry.

	Horizontal		Midpoint
	Vertical		Intersection
	Parallel		Endpoint, Vertex or Centerpoint
	Perpendicular		On
	Tangent		

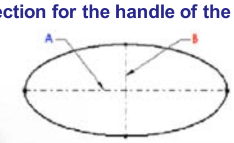
Inferencing

- Dotted lines appear when you sketch, showing alignment with other geometry.
- This alignment information is called *inferencing*.
- Inference lines are two different colors: orange and blue.
   - Orange inference lines capture and add a geometric relation such as Tangent.
   - Blue lines show alignment and serve as an aid to sketching, but do not actually capture and add a geometric relation.



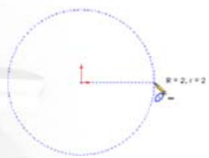
Ellipse Sketch Tool

- Used to create the sweep section for the handle of the candlestick.
- An Ellipse has two axes:
   - Major axis, labeled A at the right.
   - Minor axis labeled B at the right.
- Sketching an ellipse is a two-step operation, similar to sketching a 3 Point Arc.



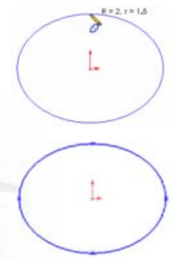
To Sketch an Ellipse:

1. Click Tools, Sketch Entity, Ellipse.
   - Tip: You can use Tools, Customize to add the Ellipse tool to the Sketch Tools toolbar.
2. Position the pointer at the center of the ellipse.
3. Click the left mouse button, and then move the pointer horizontally to define the major axis.
4. Click the left mouse button a second time.



Sketching an Ellipse:

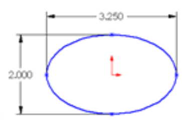
5. Move the pointer vertically to define the minor axis.
6. Click the left mouse button a third time. This completes sketching the ellipse.

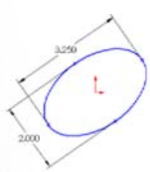


Fully Defining an Ellipse

Requires 4 pieces of information:

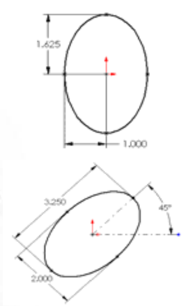
- Location of the center:
  - Either dimension the center or locate it with a geometric relation such as Coincident.
- Length of the major axis.
- Length of the minor axis.
- Orientation of the major axis.
  - Even though the ellipse at the right is dimensioned, and its center is located coincident to the origin, it is free to rotate until the orientation of the major axis is defined.





More About Ellipses

- The major axis does not have to be horizontal.
- You can dimension half the major and/ or minor axis.
  - It is like dimensioning the radius of a circle instead of the diameter.
- You do not have to use a geometric relation to orient the major axis.
  - A dimension works fine.

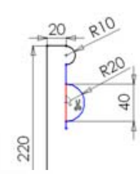


Trimming Sketch Geometry

- The Trim tool is used to delete a sketch segment.
- The segment is deleted up to its intersection with another sketch entity.
- The entire sketch segment is deleted if it does not intersect any other sketch entity.

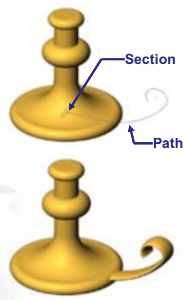
To Trim a Sketch Entity:

1. Click Trim on the Sketch Tools toolbar.
2. Position the pointer over the sketch segment.
3. The segment that will be trimmed is highlighted in red.
4. Click the left mouse button to delete the segment.



Sweep Overview

- The Sweep feature is created by moving a 2D profile along a path.
- A Sweep feature is used to create the handle on the candlestick.
- The Sweep feature requires two sketches:
  - Sweep Path
  - Sweep Section



Sweep Overview – Rules

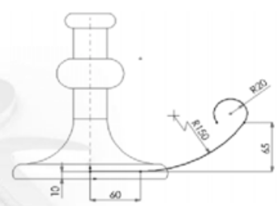
- The sweep path is a set of sketched curves contained in a sketch, a curve, or a set of model edges.
- The sweep section must be a closed contour.
- The start point of the path must lie on the plane of the sweep section.
- The section, path or the resulting solid cannot be self-intersecting.

Sweep Overview – Tips

- Make the sweep path first. Then make the section.
- Create small cross sections away from other part geometry.
- Then move the sweep section into position by adding a Coincident or Pierce relation to the end of the sweep path.

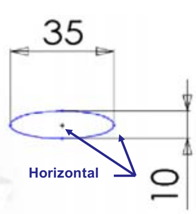
To Create the Sweep Path:

1. Open a sketch on the Front plane.
2. Sketch the Sweep path using the Line and Tangent Arc sketch tools.
3. Dimension as shown.
4. Close the sketch.



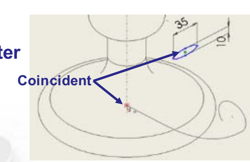
To Create the Sweep Section:

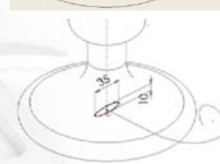
1. Open a sketch on the Right plane.
2. Sketch the Sweep section using the Ellipse sketch tool.
3. Add a Horizontal relation between the center of the ellipse and one end of the major axis.
4. Dimension the major and minor axes of the ellipse.



Creating the Sweep Section:

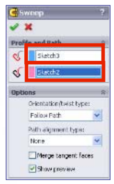
5. Add a Coincident relation between the center of the ellipse and the endpoint of the path.
6. Close the sketch.





To Sweep the Handle:

1. Click <u>Sweep</u> on the Features toolbar.
2. Select the Sweep path sketch.
3. Select the Sweep section sketch.
4. Click <u>OK</u>.



Sweeping the Handle – Results

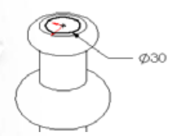
Extruded Cut with Draft Angle

- Creates the opening for a candle in the top of the candlestick.
- Same process as extruding a boss except it removes material instead of adding it.
- Draft tapers the shape.
- Draft is important in molded, cast, or forged parts.
  - Example: Ice cube tray – without draft it would be very hard to get the ice cubes out of the tray.
  - Find other examples.

To Create the Cut:

1. Open a sketch on the top face of the candlestick.

2. Sketch a circular profile <u>Concentric</u> to the circular face.
3. Dimension the circle.



Creating the Cut:

4. Click <u>Extruded Cut</u> on the Features toolbar.
5. End Conditions:
   - Type = Blind
   - Depth = 25mm
   - Draft = On
   - Angle = 15°
6. Click <u>OK</u>.

Extruding the Cut– Results

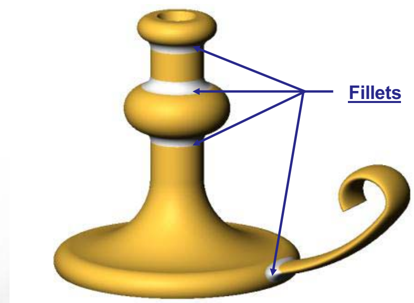
Fillet Feature

- Fillets are used to smooth the edges of the candlestick.

Selection Filters

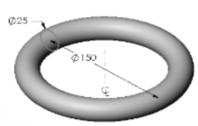
- Help in selecting the correct geometry.
- Click to turn on Selection Filter toolbar.
- Use the Edge selection filter .
- Pointer changes appearance when filter is active.

Filleting the Edges – Results

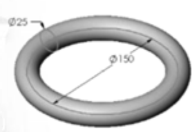


Best Practice – Keep it Simple

- Do not use a sweep feature when a revolve or extrude will work.
- Sweeping a circle along a circular path appears to give the same result as a revolve feature.
- However, the revolve feature:
  - Is mathematically less complex
  - Is easier to sketch – one sketch vs. two



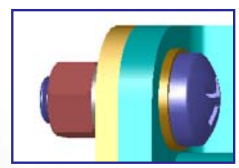
Revolve



Sweep

What is Toolbox?

- Ready-to-use standard parts such as bolts, screws, washers, lock washers, and so forth.
- Eliminates the need to model most fasteners and many other standard parts.
- Easy drag-and-drop placement.

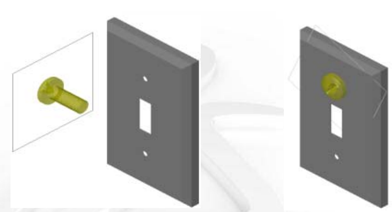


Toolbox Browser

- Browser contains libraries of ready-to-use components.
- Click Toolbox to access Toolbox Browser.
- Contains libraries of ready-to-use parts.

Easy Drag-and-Drop Placement

- Drag from Toolbox Browser
- Snap to assembly



Mate Relationships Defined at Placement

- When the Toolbox part snaps to the assembly, the mate relationship between the Toolbox part and the other part is established.
- When using washers, place them first. Subsequent hardware — like screws and nuts — mate to the washers.

Standard Hardware Matches Standard Holes

- Hardware placed on holes created using the Hole Wizard can be appropriately sized.
- Standard sized parts help to keep your designs realistic.

Specifying Toolbox Part Properties

- Change part properties to customize the hardware to your design.
  - Specify the properties as you are placing the part.
  - Ability to change the properties after the part is placed.

Thread Display

- Simplified — Represents the hardware with few details. Most common display.

- Cosmetic — Represents some details of the hardware.

- Schematic — Very detailed display which is used for unusual or custom designed hardware.

Supported Standards

- Toolbox supports international standards
  - ANSI
  - AS
  - BSI
  - CISC
  - DIN
  - GB
  - IS
  - ISO
  - JIS
  - KS
  - MIL

Libraries from Leading Manufacturers

- Toolbox contains standard parts libraries from leading manufacturers such as:
  - PEM®
  - Torrington®
  - Truarc®
  - SKF®
  - Unistrut®

Wheel Assembly

Components

- Wheel Hub Design Intent
  - The part will be molded from plastic
  - Two wheel hubs will fit together to form a single wheel
  - Index pins and holes will keep the two hubs from rotating relative to one another
  - There must be a passage for the tube stem
  - The bolt holes must capture the nuts to keep

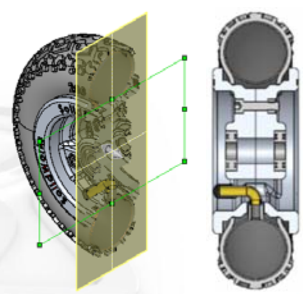
Components

- Tire and Tube
  - These will both be purchased parts
  - The models will be imported

- Bearings, Bolts and Nuts
  - These are purchased parts
  - Use Toolbox to create the parts

Wheel Assembly

- Section View

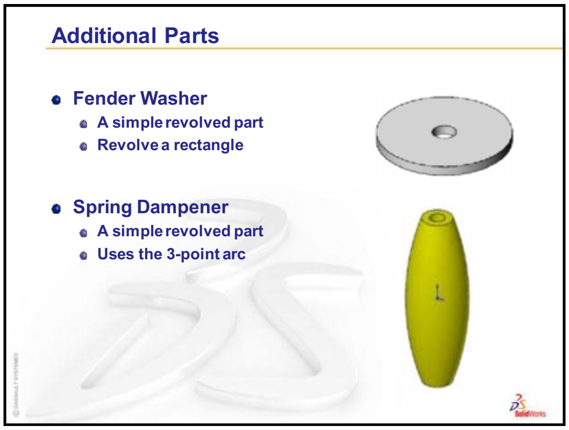


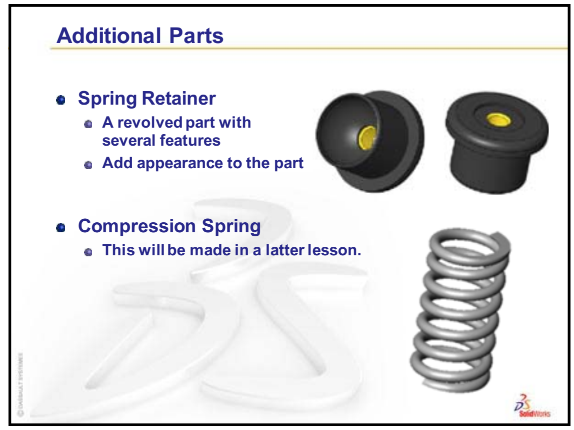
Wheel Assembly

- Exploded View

Additional Parts

- The Spring Assembly will be created later. It will need the following parts:


Additional Parts
Fender Washer
A simple revolved part
Revolve a rectangle
Spring Dampener
A simple revolved part
Uses the 3-point arc


Additional Parts
Spring Retainer
A revolved part with several features
Add appearance to the part
Compression Spring
This will be made in a latter lesson.

5

Thin Features — The Deck

Review of Lesson 4: Revolved Features

Questions for Discussion

1 Describe the steps required to create a revolved feature.

Answer: To create a revolved feature:

- Sketch a profile on a 2D plane.
- The profile sketch may optionally include a centerline as the axis of revolution. The centerline (or sketch line as axis of revolution) must not cross the profile.
- Click Revolved Boss/Base on the Features toolbar.
- Enter a rotation angle. The default angle is 360°.

2 Describe an assembly.

Answer: An assembly combines two or more parts in a single document. In an assembly or sub-assembly, parts are referred to as components.

3 What does the command Convert Entities do?

Answer: Convert Entities projects one or more curves onto the active sketch plane. Curves can be edges of faces or entities in other sketches.

4 What does a selection filter do?

Answer: A selection filter allows you to more easily select the item you want in the Graphics Area by only allowing you to select a specified type of entity.

5 What does it mean when a component in an assembly is "fixed"?

Answer: A fixed component in an assembly cannot move. It is locked in place. By default, the first component added to an assembly is automatically fixed.

6 What are mates?

Answer: Mates are the relationships that align and position components in as assembly.

7 What are degrees of freedom?

Answer: Degrees of freedom describe how an object is free to move. There are six degrees of freedom. They are translation (movement) along the X, Y, or Z axes, and rotation around the X, Y, or Z axes.

8 How are degrees of freedom related to mates?

Answer: Mates eliminate degrees of freedom.

5 Minute Assessment — #5 Answer Key

1 What is a thin feature?

Answer: A thin feature is a feature created from an open sketch.

2 How do you lock a dimension orientation so that it remains horizontal, vertical or aligned.

Answer: Move the cursor until the dimension is in the desired orientation, then right-click to lock the orientation.

3 You have selected a surface and clicked view **Normal To**  but you want to look at the reverse side of the surface, what do you do?

Answer: Click **Normal To**  again, it will toggle from the front to the back of the surface.

4 True or False: The Mirror command can only mirror a single feature at a time.

Answer: False

Lesson 5 Quiz — Answer Key

1 How is the ConfigurationManager used in SolidWorks?

Answer: The ConfigurationManager is used to switch from one configuration to another.

2 Can SimulationXpress be used to analyze parts where the sum of the forces do not add up to zero?

Answer: No, SimulationXpress can only analyze parts that are static (sum of the forces and moments must equal zero).

3 What is a Free Body Diagram?

Answer: A Free Body Diagram is used to calculate all the external forces acting on a body.

4 Name an advantage to using the Hole Wizard as compared to creating a sketch and either extruding or revolving a cut.

Answer: The Hole Wizard contains all the correct geometry and sizes for holes that follow the common engineering standards. If you create a hole by extruding or revolving a sketch you create, you have to look up the correct values for all the dimensions.

5 What does it mean when the Factor of Safety is less than one?

Answer: When the Factor of Safety is less than one, the part has exceeded its Yield Strength.

6 How is the number 345,678 expressed in Engineering Notation?

Answer: 345.678e3 (345.678e+003 or 345.678×10^3 are also valid). The powers of ten are always in increments of 3.

7 How is the number 345,678 expressed in Scientific Notation?

Answer: 3.45678e5 (3.45678e+005 or 3.45678×10^5 are also valid). There is only one digit to the left of the decimal point.

8 What is the shape of the finite elements used by SolidWorks SimulationXpress?

Answer: Tetrahedrons

9 True or False: When a feature is Suppressed, it is removed from memory and not calculated.

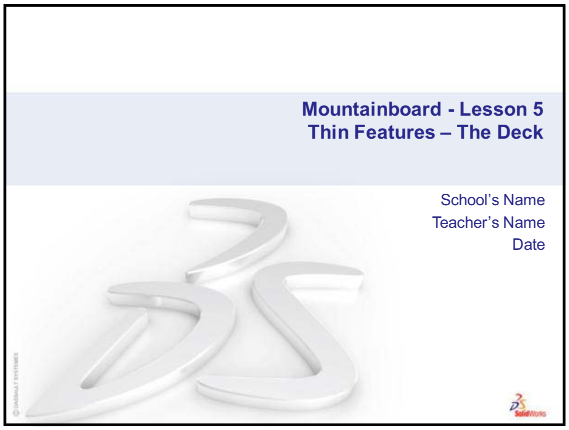
Answer: True

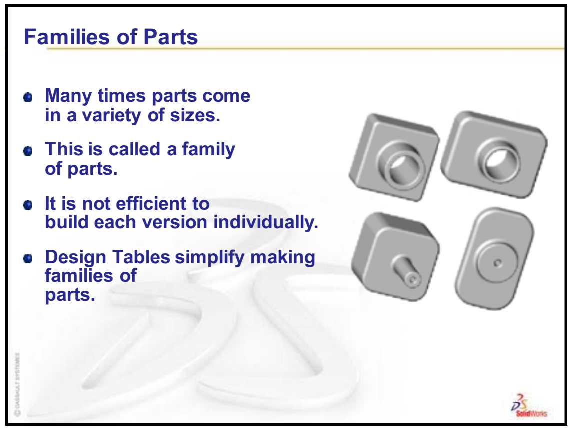
10 Name two things that can be controlled by configurations.

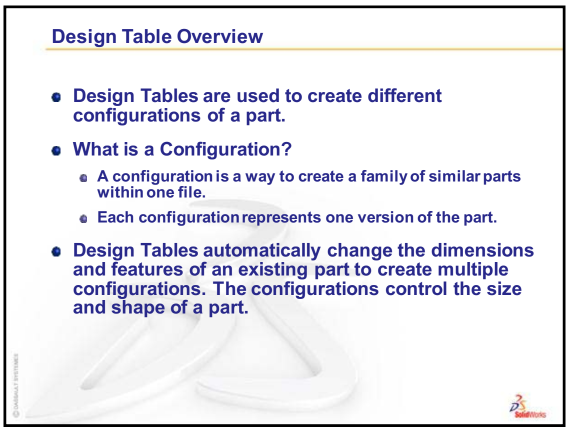
Answer: Dimensions and suppression state.

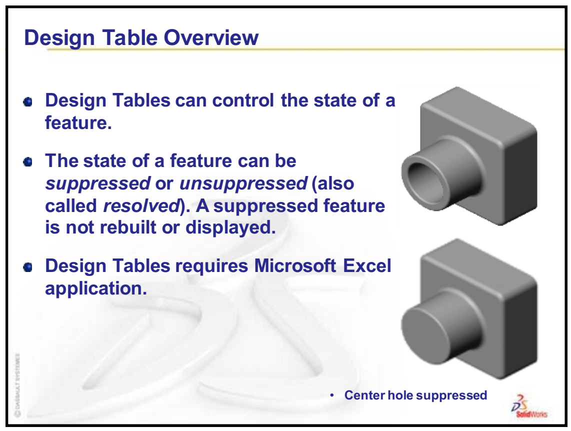
Thumbnail Images of PowerPoint Slides

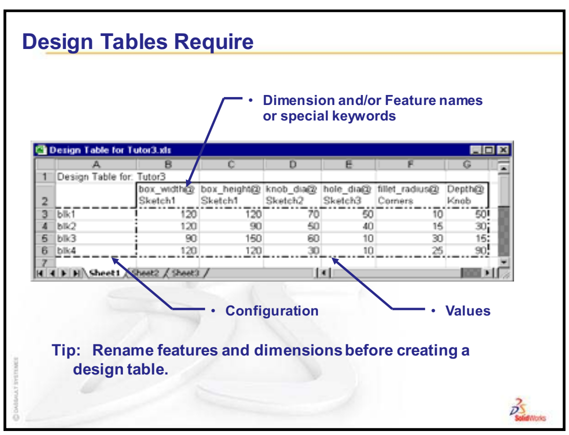
The following thumbnail images, arranged left to right, show the PowerPoint slides provided with this lesson.

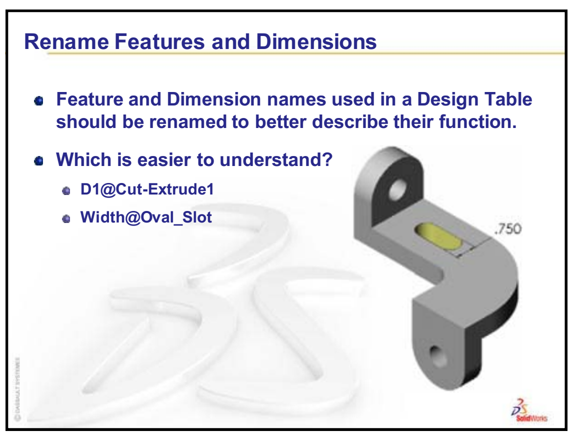










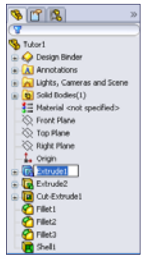


To Rename a Feature

- **Click-pause-click on *Extrude1* in the FeatureManager design tree (do not double-click).**

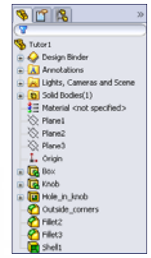
Tip: Instead of the click-pause-click technique, you can select the feature, and then press the function key F2.

- **The feature name is highlighted in blue, ready to be edited.**
- **Type the new name, *Box*, and press Enter.**



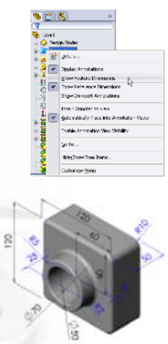
Rename the Other Features Used in the Design Table

- **Rename *Extrude2* to *Knob*.**
- **Rename *Cut-Extrude1* to *Hole_in_knob*.**
- **Rename *Fillet1* to *Outside_corners*.**



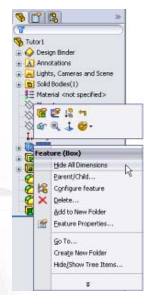
To Display Feature Dimensions

- **Right-click the *Annotations* folder, and select Show Feature Dimensions from the shortcut menu.**



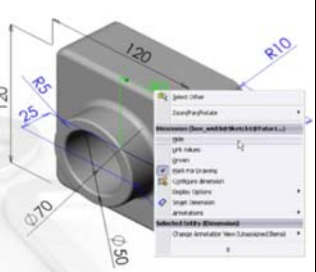
To Hide All the Feature Dimensions for a Selected Feature

- **Right-click the feature in the FeatureManager design tree, and select Hide All Dimensions from the shortcut menu.**



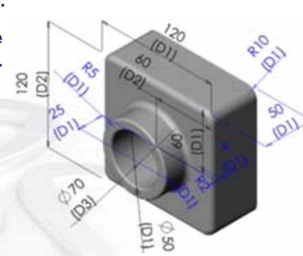
To Hide Individual Dimensions

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



To Display Dimension Names

1. **Click Tools, Options.**
2. **Click General on the System Options tab.**
3. **Click Show dimension names.**
4. **Click OK.**



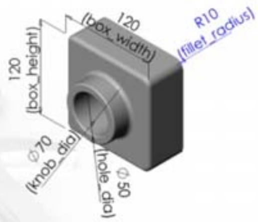
To Rename a Dimension

1. Display the dimension.
   - Either double-click the feature to display its dimensions.
   - Or, right-click the Annotations folder, and select Show Feature Dimensions.
2. Click the 70mm diameter dimension, and in the PropertyManager, rename the dimension to *knob_dia*, then click OK.

Note: *"@Sketch2" is automatically added to the dimension name.*

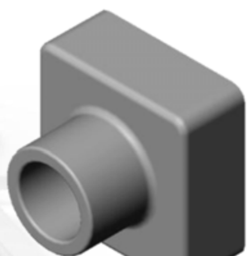
Rename these Dimensions

- Height of the box to *box_height*.
- Width of the box to *box_width*.
- Diameter of the hole in the knob to *hole_dia*.
- Radius of outside corners to *fillet_radius*.



Design Intent

- The depth of the *Knob* should always be equal to the depth of the *Box* (the base feature).
- The *Knob* should always be centered on the *Box*.
- Dimensions alone are not always the best way to capture design intent.

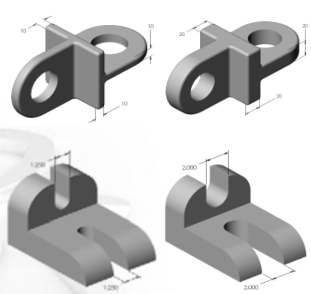


Linking Values

- The Link Values command relates dimensions to each other through shared variable names.
- If the value of one linked dimension is modified, then all of the linked dimensions are modified.
- Link Values is excellent for making feature dimensions equal to each other.
- This is an important tool for capturing design intent.

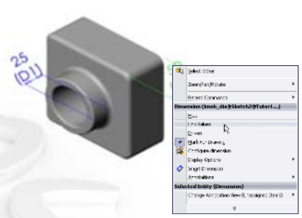
Examples of Uses for Link Values

- The thickness of the square and the two tabs is always equal.
- The width of both slots is always equal.



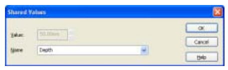
Link the Depth of the *Box* to the Depth of the *Knob*

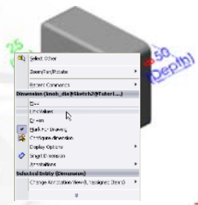
1. Display the dimensions.
2. Right-click on the depth dimension for the *Box*, and select Link Values from the shortcut menu.



Linking the *Box* to the *Knob*

3. Type *Depth* in the Name text box and then click OK.
4. Right-click on the depth dimension for the *Knob*, and select Link Values from the shortcut menu.

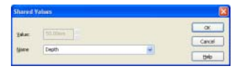


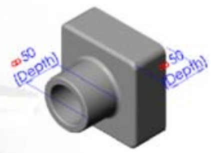


Linking the *Box* to the *Knob*

5. Select *Depth* from the list, and click OK.
6. Both dimensions have the same name and value.
7. Rebuild the part to update the geometry.

Tip: Use the CTRL key to select several dimensions at the same time and link them in one step.



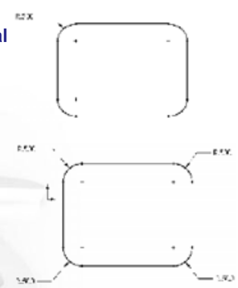


Geometric Relations

- Relate geometry through physical relationships such as:
  - Concentric
  - Coradial
  - Midpoint
  - Equal
  - Collinear
  - Coincident

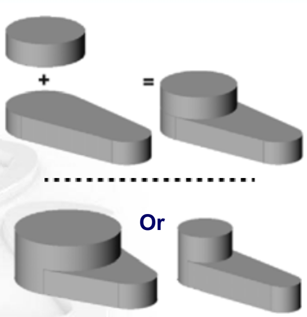
Examples of Geometric Relations

- The Sketch Fillet tool automatically creates one radial dimension and 3 Equal relations.
- Changing the dimension changes all 4 fillets.
- This technique is better than having 4 radial dimensions.



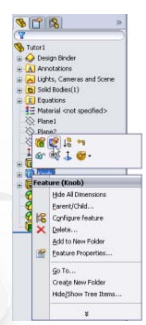
Examples of Geometric Relations

- Two features.
- Making the circle for the boss Coradial with the edge of the base ensures that the boss will always be the correct size regardless of how the base changes.



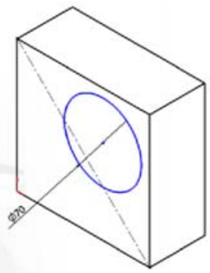
To Center the *Knob* on the *Box*

1. Right-click the *Knob* feature, and select Edit Sketch from the shortcut menu.



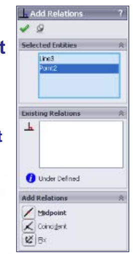
Centering the *Knob* on the *Box*

2. Delete the linear dimensions.
3. Notice the circle is blue, indicating it is under defined.
4. Drag the circle to one side. Without dimensions to locate it, it is free to move.
5. Click , and sketch a diagonal Centerline.



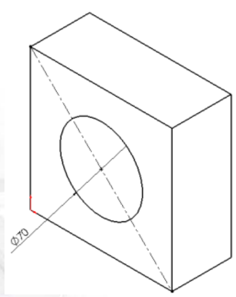
Centering the *Knob* on the *Box*

6. Click Add Relation .
7. Select the centerline and the point at the center of the circle.
   - Note: If the centerline is still highlighted when Add Relations opens, the line automatically appears in the Selected Entities list and you do not have to select it again.
   - If you select the wrong entity, right-click in the graphics area, and select Clear Selections.



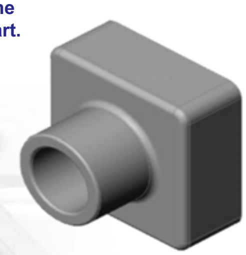
Centering the *Knob* on the *Box*

8. Click Midpoint, and then click Apply and Close.
9. The circle will now stay centered on the *Box* feature.



Centering the *Knob* on the *Box*

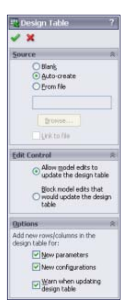
10. Click Rebuild to exit the sketch and rebuild the part.



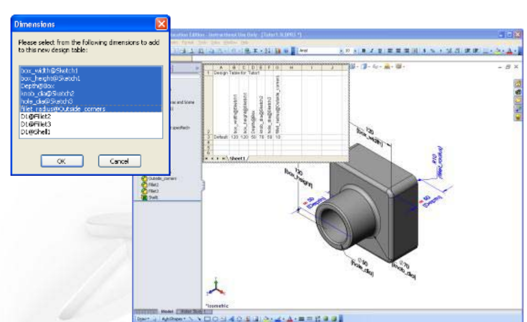
To Insert a New Design Table

1. Position the part in the lower right hand corner of the graphics area.
2. Click Insert, Design Table.

   The PropertyManager appears.
3. Select the Auto-create option to create a new design table automatically .



Inserting a New Design Table



Inserting a New Design Table

- An Excel worksheet is displayed in the part document window.
- Excel toolbars replace the SolidWorks toolbars.
- By default, the first configuration is named *Default*. You can (and should) change this to something more meaningful.

Review of a Design Table's Format

Dimension and/or Feature names or special keywords go in this row

	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

Configuration names go in this column

Values go here

Inserting a New Design Table

1. Double-click the *box_width* dimension.

   The full dimension name is inserted into cell B2. The dimension value is inserted into cell B3.
   The next cell, C2, is automatically selected.

	A	B
1	Design Table for: Tutor3	
2		box_width@Sketch1
3	Default	120
4		
5		
6		

2. Double-click the box_height dimension.

	A	B	C
1	Design Table for: Tutor3		
2		box_width@Sketch1	box_height@Sketch1
3	Default	120	120
4			
5			
6			

Inserting a New Design Table

3. Repeat this process for knob_dia, hole_dia, fillet_radius, and Depth.
   - Note: Since the depth dimensions of the Knob and the Box are linked together, you only need one of them in the design table.

	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@O utside_corners	Depth@Kn ob
3	Default	120	120	70	50	10	50
4							
5							
6							

Excel tip: Dimension names tend to be very long. Use the Excel command Format, Cells, and click Wrap Text on the Alignment tab.

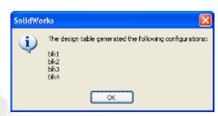
Inserting a New Design Table

1. Enter new configuration names in column A:
   - Replace Default with blk1.
   - Fill cells A4 through A6 with blk2, blk3, and blk4.
2. Fill in the dimension values as shown below.

	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_radiu s@Outside _corners	Depth@Kn ob
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

To Close the Excel Worksheet

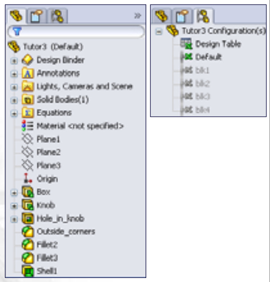
1. Click in the graphics area outside the worksheet.
2. The system builds the configurations.



3. Click OK.
   The Design Table is embedded and stored in the part document.
   The design table icon appears in the FeatureManager. Design Table
4. Save the part document.

To View Part Configurations

1. Click the Configuration Manager tab at the bottom of the FeatureManager window.
   The list of configurations is displayed.
2. Double-click each configuration.



Viewing Part Configurations

3. The part is automatically rebuilt using the dimension values from the design table.

Mechanics of Solids

- Exterior Loads
- Interior Loads
- Material properties

Exterior Loads

- Forces and loads acting on the outside of the body
- Free Body Diagrams
  - Known loads
  - Replace attachments with forces and moments

Newtons Laws

The fundamental principles of mechanics

First Law
- If the resultant forces acting on a body are zero:
  - Body will remain at rest, or
  - Body will move with constant velocity

Second Law
- If the resultant forces acting on a body are *not* zero:
  - Body will accelerate proportional to the magnitude of resultant force.

Third Law
- Forces of action and reaction between bodies in contact have same magnitude and opposite direction

Internal Forces

- Stress
  - Measure of force per unit area
  - Units
    - Pounds per square inch
    - Newtons per square meter
- Strain
  - Measure of elongation
  - Units - length/length
    - inches per inch
    - millimeters per millimeter

Internal Forces

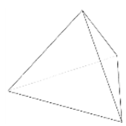
- Deformation
  - The result from totaling all the internal strain

Finite Element Analysis

- Numerical method
  - Part is broken into many small elements (meshing)
  - Matrix of equations describe the relationship between elements
  - Equations solved simultaneously
- Used to solve problems in:
  - Machine design
  - Acoustics
  - Electromagnetics
  - Solid mechanics
  - Fluid dynamics

Meshing

- During the meshing operation, the model is broken into many tetrahedral elements.
- The tetrahedral elements are <u>not</u> infinitesimal.
- SolidWorks Simulation can use other element shapes, SimulationXpress only uses tetrahedrons.



The Mountainboard

The Deck

- Design Intent
  - The Deck will be created as a laminated piece
  - Symmetric front to back and side to side
  - Mounting holes for Bindings
  - Mounting holes for Trucks

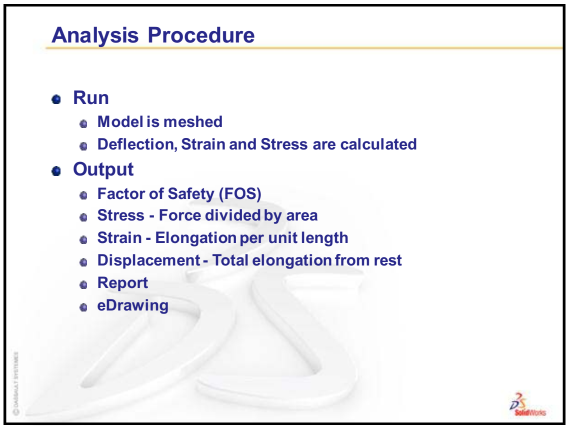
SolidWorks SimulationXpress

- Linear Static Analysis
  - Linear – The Stress-Strain curve is a straight line
  - Static - Loads are applied slowly
- Wizard driven
  - Follow the on screen instructions

Analysis Procedure

The basic steps for analysis are:

- Material - material can be added in either:
  - SolidWorks
  - SimulationXpress
- Fixtures
  - Fixed - A fixed restraint stops all movement
- Loads
  - Force - force can either be applied normal to the surface or normal to a selected plane or face.
  - Pressure - pressure is always normal to a surface


Analysis Procedure
Run
Model is meshed
Deflection, Strain and Stress are calculated
Output
Factor of Safety (FOS)
Stress - Force divided by area
Strain - Elongation per unit length
Displacement - Total elongation from rest
Report
eDrawing

6

Multibody Parts — The Axle and Truck

Review of Lesson 5 — Thin Features

Questions for Discussion

1. Different part configurations can have different______________, ________________?

   Answer: dimension values, suppressed features.

2. Thin features can be created from:

   a) Open sketch

   b) Closed sketch

   c) Either an opened or closed sketch.

   Answer: C

3. True or False: SimulationXpress can be used for linear static analysis of parts.

   Answer: True

4. How do you “lock in” a dimension orientation?

   Answer: Move the mouse until the correct orientation appears, then click the right mouse button to lock it in.

5. Where can you apply a material to a part so that it can be used in SimulationXpress?

   Answer: You can either apply the material in the part, or you can apply the material in the SimulationXpress Wizard.

6. Split lines are used to do what?

   Answer: Split lines are used to split single faces into multiple faces.

7. What is the only end condition available for a cut made with an open sketch?

   Answer: Through All

5 Minute Assessment — #6-1 Answer Key

1. What are the three Boolean operations that can be done with multi-bodies?

   Answer: Add, Subtract and Common.
2. What SolidWorks tool is used on multibody solids to do Boolean operations?

   Answer: The **Combine** tool.
3. What determines the radius of a Full Round Fillet?

   Answer: The geometry of the model.
4. What type of fillet can be used to have the fillet radius change along the length of an edge?

   Answer: **Variable Radius Fillet**.
5. What mirroring option is used to mirror half of a part to get the full part?

   Answer: **Mirror Body** (as compared to Mirror Features and Mirror Faces)

5 Minute Assessment — #6-2 Answer Key

1. What are the three steps of the FEA process?

   Answer: Pre-processing, Solution, Post-processing.
2. What happens during discretization or meshing.

   Answer: The discretization process splits the geometry into relatively small and simply-shaped entities, called finite elements.
3. The slope of the Stress-Strain curve is called _______________?

   Answer: Modulus of Elasticity or Young's Modulus.
4. What are the three conditions that must be met to use SolidWorks Simulation?

   Answer: The three conditions are:
   - The material must be linear (the slope of the stress-strain curve must be linear)
   - The deformation must be small
   - The loads must be static. The loads must be slowly applied (no impact) and must not vary over time.
5. If you apply a material in SolidWorks, do you have to apply it again in SolidWorks Simulation?

   Answer: No, the material will carry forward into SolidWorks Simulation.

5 Minute Assessment – #6-3 Answer Key

1. What are the three primary requirements to use SolidWorks Simulation?

   Answer: The loads are static, the material has a linear stress-strain curve, and deflections are small.

2. True or False: When creating a linear pattern, in two directions, the directions must be 90 degrees apart?

   Answer: False, they can be at any angle relative to one another.

3. Relative to the sketch plane, which direction can you extrude a rib?

   Answer: Ribs can be extruded either parallel to the sketch plane or normal to it.

4. What are the three Boolean operations that can be done with the **Combine** command?

   Answer: Add (Union), Subtract, Common (Intersection)

Lesson 6 Quiz — Answer Key

1 What SolidWorks tool is used on multibody solids to do Boolean operations?

Answer: The **Combine** tool.

2 When you create a Full Round Fillet, what determines its radius?

Answer: The geometry of the model.

3 What type of fillet can be used to have the fillet radius change along the length of an edge?

Answer: A **Variable Radius Fillet**.

4 What mirroring option is used to mirror half of a part to get the full part?

Answer: **Mirror Body** (as compared to Mirror Features and Mirror Faces)

5 What are the three steps of the FEA process?

Answer: Pre-processing, Solution, Post-processing.

6 What happens during discretization or meshing.

Answer: The discretization process splits the geometry into relatively small and simply-shaped entities, called finite elements.

7 The slope of the Stress-Strain curve is called _______________?

Answer: Modulus of Elasticity or Young's Modulus.

8 What are the three conditions that must be met to use SolidWorks Simulation?

Answer: The three conditions are:

- The material must be linear (the slope of the stress-strain curve must be linear)
- The deformation must be small
- The loads must be static. The loads must be slowly applied (no impact) and must not vary over time.

9 If you apply a material in SolidWorks, do you have to apply it again in SolidWorks Simulation?

Answer: No, the material will carry forward into SolidWorks Simulation.

10 True or False: When creating a linear pattern in two directions, the directions must be 90 degrees apart?

Answer: False, they can be at any angle relative to one another.

11 Relative to the sketch plane, which direction can you extrude a rib?

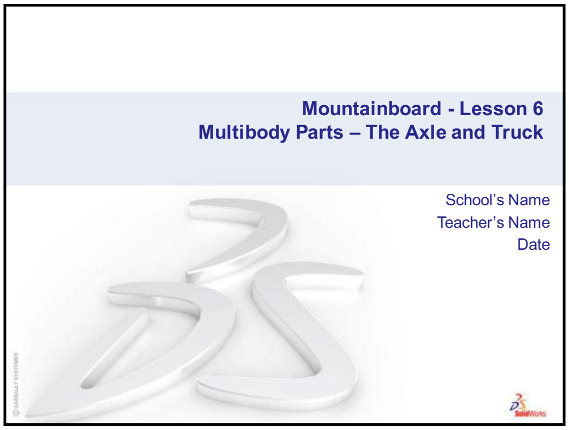
Answer: Ribs can be extruded either parallel to the sketch plane or normal to it.

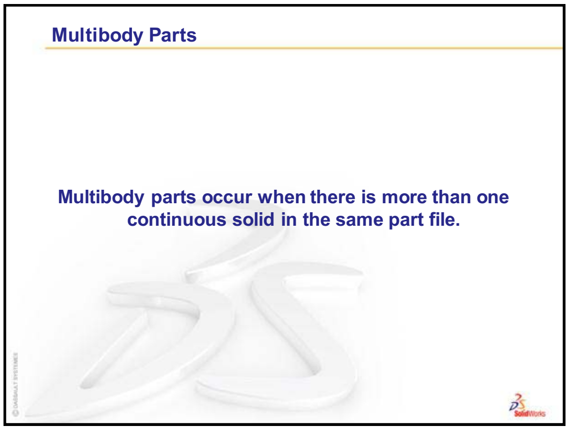
12 What are the three Boolean operations that can be done with the **Combine** command?

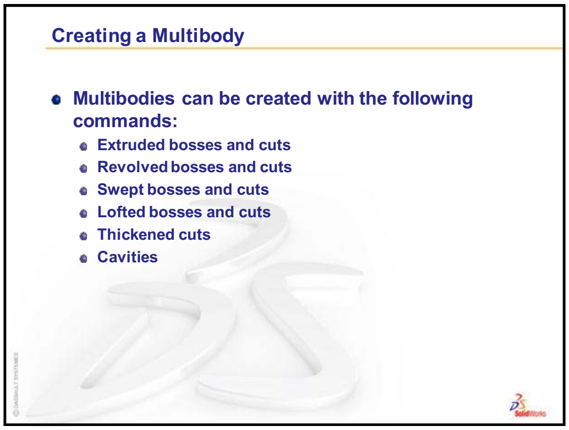
Answer: Add (Union), Subtract, Common (Intersection)

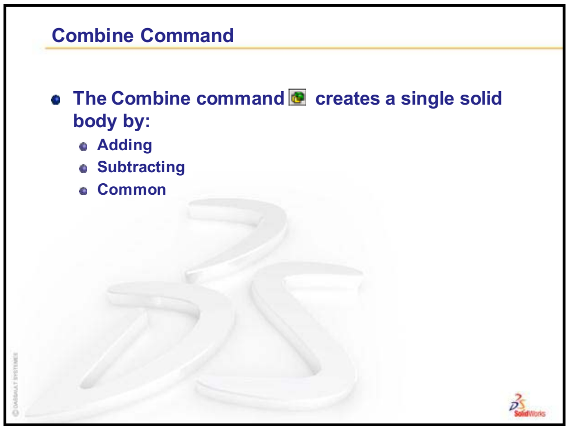
Thumbnail Images of PowerPoint Slides

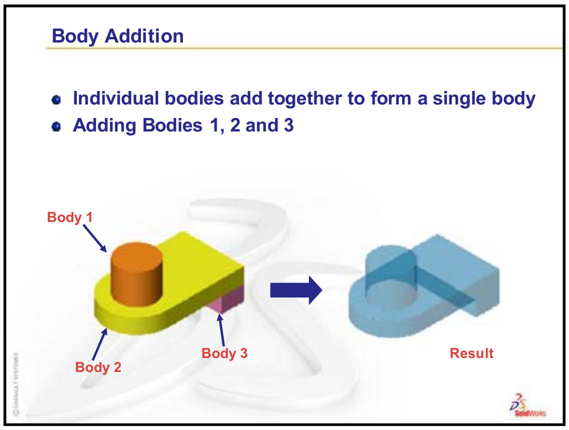
The following thumbnail images, arranged left to right, show the PowerPoint slides provided with this lesson.

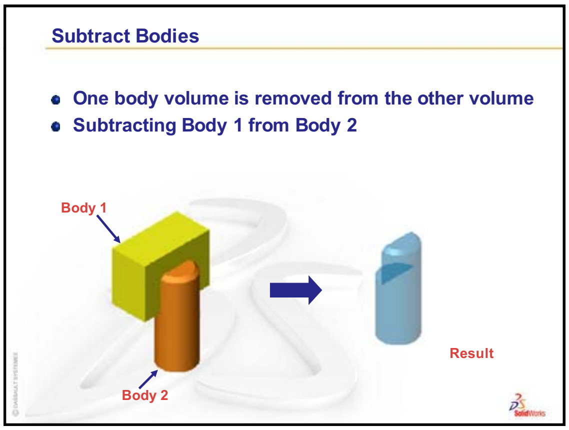






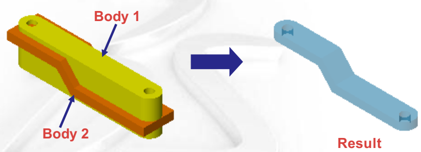




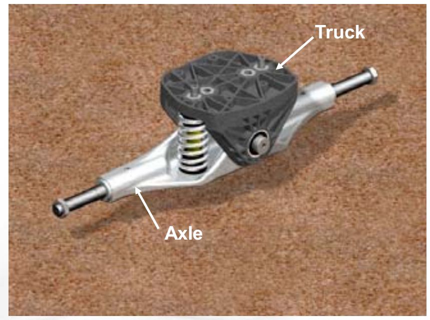


Common Body

- The resulting body is the volume common to initial volumes
- Also known as an *Intersection*
- *Combining Bodies 1 and 2 using Common*

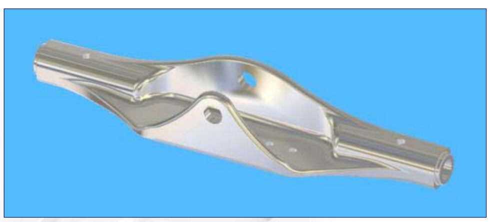


Axle and Truck



The Axle

- The Axle is the connection between the wheels and the Truck.



Design Intent

- Connects the wheels to the Truck
- Part will be machined from aluminum stock
- Part will pivot about the King Pin that goes through the Truck
- Mounting must be provided for optional brake system

SolidWorks Simulation

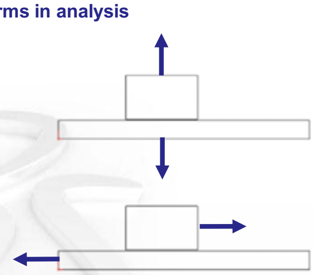
SolidWorks Simulation provides a more in depth FEA package than SolidWorks SimulationXpress

- The levels of SolidWorks Simulation software
  - SolidWorks SimulationXpress
  - SolidWorks Simulation
  - SolidWorks Simulation Professional
  - SolidWorks Simulation Premium

Terms

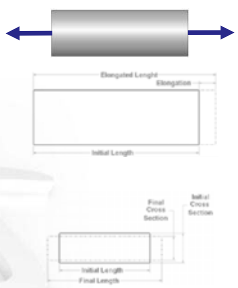
Some important terms in analysis

- Stress
  - Tensile Stress
  - Shear Stress



Terms

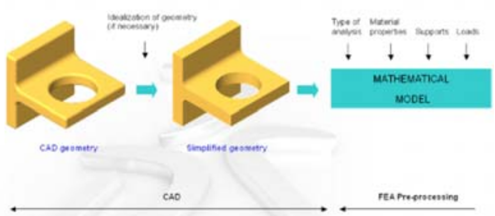
- Strain
  - Elongation divided by initial length
- Modulus of Elasticity
- Poison's Ratio
  - Reduction in cross section
  - Ratio of axial strain to cross section strain



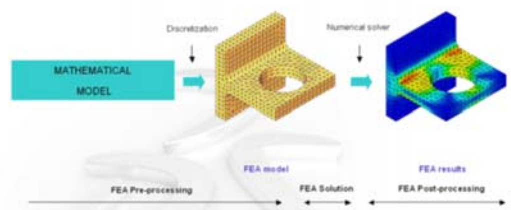
Steps in the FEA Process

- Preprocessing
- Solution
- Post-processing

Build Mathematical Model

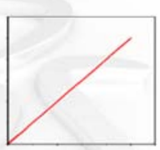


Build Finite Element Model



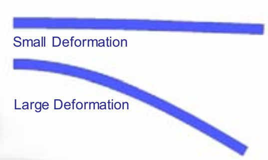
Limitations of SolidWorks Simulation

- Material is linear
  - Stress is proportional to Strain
  - Slope of curve is Young's Modulus (Modulus of Elasticity)



Limitations of SolidWorks Simulation

- Deformations are small
  - Deformation normally less than 5%
- Loads are static
  - Slowly applied
  - Non-changing



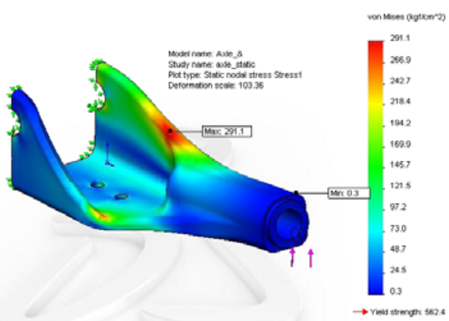
The Truck

- The Truck holds the King Pin which is the shaft around which the Axle pivots.

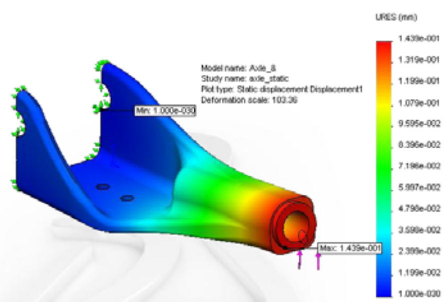
Design Intent

- One face mounts solidly to the Deck
- Bearings mounted to allow King Pin to pivot
- Hex nuts will be molded into the Truck for suspension adjustment

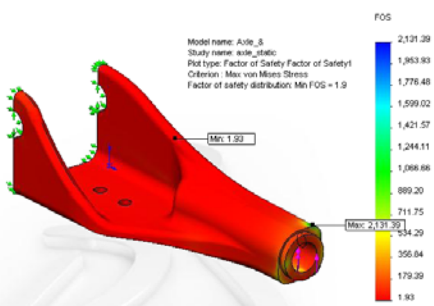
Stress Results



Displacement Results



Factor Of Safety



7

Sweeps and Lofts — Springs and Binding

Review of Lesson 6 — Multibody Parts — The Axle and Truck

Questions for Discussion

1. What is a multibody solid?

   Answer: A multibody solid is a part with more than one enclosed volumes.

2. What is a linear material?

   Answer: A linear material is one in which the stress-strain curve is a straight line from no load to the yield point.

3. List some differences between SolidWorks SimulationXpress and SolidWorks Simulation.

   Answer:

   - SolidWorks SimulationXpress is a sub-set of SolidWorks Simulation.
   - SolidWorks SimulationXpress has only two types of loads, force and pressure, while SolidWorks Simulation adds several others such as gravity, centrifugal, bearing, remote, connectors and temperature.
   - SolidWorks SimulationXpress has only one type of fixture, fixed, while SolidWorks Simulation adds several others such as symmetry, on flat face, on cylindrical face, on spherical face and immovable.
   - SolidWorks Simulation has additional controls to adjust the size and distribution of the mesh elements.
   - SolidWorks Simulation has many more reports to show the results of the analysis.

4. List some types of refinements we may apply to our models once they have all the functional features.

   Answer:

   - Reduce weight by removing material or changing the material.
   - Add features to strengthen weak areas.
   - Remove or relocate features where there is interference.

5 Minute Assessment – #7 Answer Key

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

   Answer: The sweep section (or profile) and the sweep path.

2. What information does the pointer provide while sketching an arc?

   Answer: The pointer displays: arc angle in degrees, arc radius and inferences to model or sketch geometry.

3. What does Convert Entities do?

   Answer: Convert Entities projects lines, arcs, curves and edges in our model onto the sketch plane and creates lines, arcs and curves in our current sketch.

4. How many loft profiles are required to create a loft feature?

   Answer: At least two.

5. What are the functions of Guide Curves when creating a sweep?

   Answer: Guide Curves can be used to control either the shape or twist of the profile as it sweeps along the path.

Lesson 7 Quiz – Answer Key

1 Describe the steps required to create a swept feature.

Answer: To create a swept feature:

- Sketch the Sweep path. The path must not be self-intersecting.
- Sketch the Sweep section (profile).
- Add a Geometric Relation between the sweep section and the path.
- Click Sweep on the Features toolbar.
- Select the Sweep path.
- Select the Sweep cross section.

2 Each of the following parts was created with *one* feature.

- Name the Base feature for each part.
- Describe the 2D geometry used to create the Base feature of the part.
- Name the sketch plane or planes required to create the Base feature.


Part 1


Part 2


Part 3

Answer:

- Part 1: Extrude – created with an L-shaped profile sketched on the Right plane.
- Part 2: Revolve – created with 3 tangent arcs and 3 lines and a centerline sketched on the Top plane. The angle of rotation is 270°. **Note:** The 2D profile could also be sketched on the Right plane.
- Part 3: Sweep – created with an ellipse cross section sketched on the Right plane and an S-shaped path composed of 2 lines and 2 tangent arcs sketched on the Front plane.

3 Describe the steps required to create a Loft feature.

<u>**Answer:**</u>

- Create the planes required for the profile sketches.
- Sketch a profile on the first plane.
- Sketch the remaining profiles on the corresponding planes.
- Click Loft from the Features toolbar.
- Select the profiles.
- Review the connecting curve.
- Click OK.

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

<u>**Answer:**</u> The minimum number of profiles for a Loft feature is two.

5 True or False. The location where you select each profile determines how the Loft feature is created.

<u>**Answer:**</u> True.

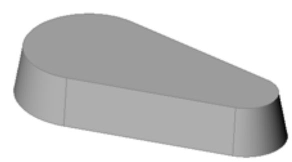
6 What two sketches are required to create a Sweep feature?

<u>**Answer:**</u> The sweep feature requires a <u>Sweep Path</u> sketch and a <u>Sweep Section (profile)</u> sketch.

7 Where can you find additional sketch tools that are not located on the Sketch Tools toolbar?

<u>**Answer:**</u> Click **Tools**, **Sketch Entities** from the main menu.

8 Multiple choice. Examine the illustration at the right. How should you create this object?



a. Use a **Revolve** feature

b. Use a **Sweep** feature

c. Use an **Extrude** feature with the option **Draft while extruding**.

<u>**Answer:**</u> c.

9 True or False. A SolidWorks part can contain more than one closed volume.

<u>**Answer:**</u> True

10 What is the name of the entity created by combining curves, sketch geometry and model edges into a single curve.

<u>**Answer:**</u> Composite curve.

11 When exploding components in an assembly, how do you reorient the direction of the Triad?

<u>**Answer:**</u> Right-click the center of the Triad and select “Align To”. Select an edge or planar face to align the triad.

12 How many exploded views can be created of an assembly?

<u>**Answer:**</u> One per configuration

Thumbnail Images of PowerPoint Slides

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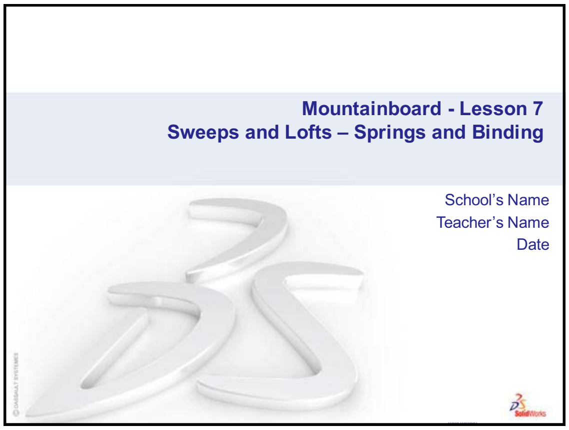
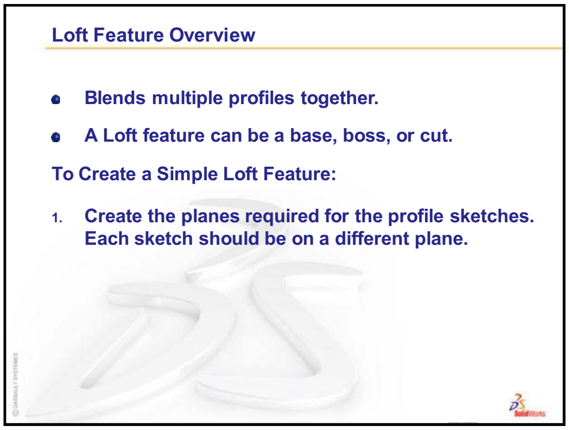
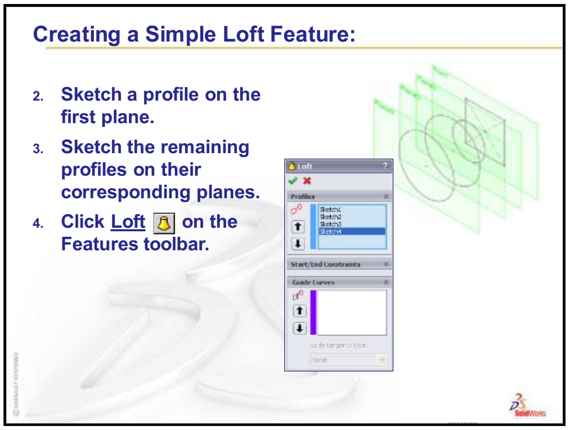
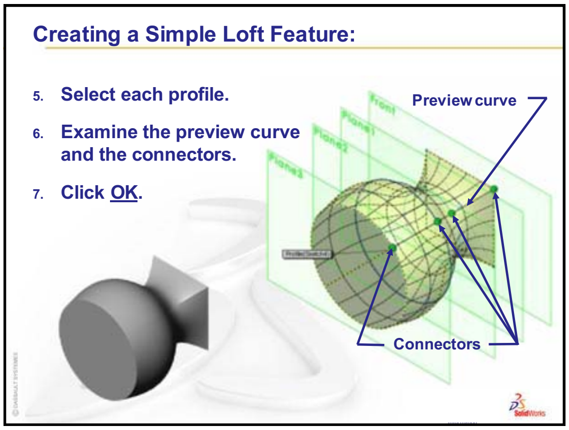
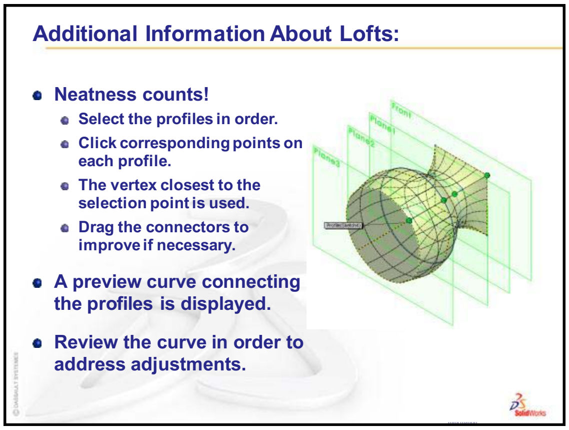
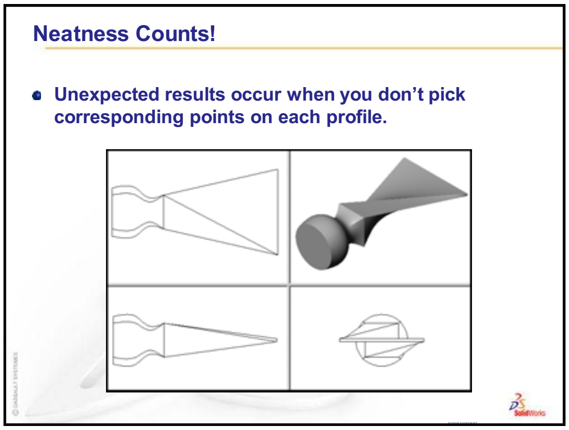






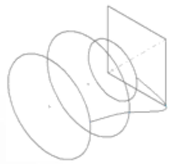






Neatness Counts!

- Rebuild errors can occur if you select the profiles in the wrong order.



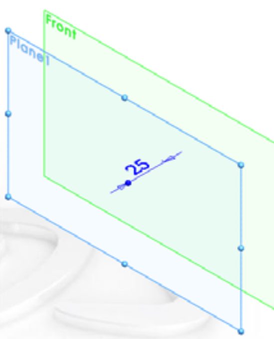
Rebuild Errors
The feature could not be created because it would produce self-intersecting geometry.

To Create an Offset Plane:

1. Hold down Ctrl and drag the *Front* plane in the direction you want the offset to go.

   NOTE: Ctrl-drag is a common Windows technique for copying objects.
2. The Plane PropertyManager appears.
3. Enter 25mm for Distance.
4. Click OK.

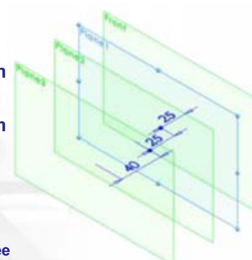
Creating an Offset Plane – Results



Setting up the Planes

Additional offset planes are required.

- *Plane2* is offset 25mm from *Plane1*.
- *Plane3* is offset 40mm from *Plane2*.
- Verify the positions of the planes.
  - Click View, Planes.
  - Double-click the planes to see their offset dimensions.

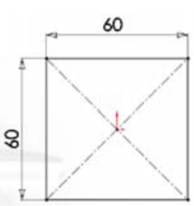


Sketch the Profiles

- The Loft feature is created with 4 profiles.
- Each profile is on a separate plane.

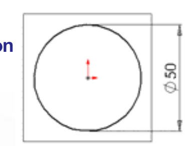
To Create the First Profile:

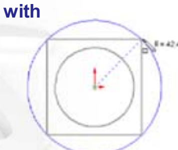
1. Open a sketch on the Front plane.
2. Sketch a square.
3. Exit the sketch.



Sketch the Remaining Profiles:

1. Open a sketch on *Plane1*.
2. Sketch a circle and dimension
3. Exit the sketch.
4. Open sketch on *Plane2*.
5. Sketch a circle whose circumference is coincident with the corners of the square.
6. Exit the sketch.

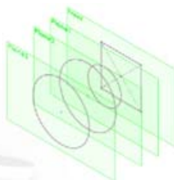




To Copy a Sketch:

1. Select *Sketch3* in the FeatureManager design tree or graphics area.
2. Click Edit, Copy or click Copy on the Standard toolbar.
3. Select *Plane3* in the FeatureManager design tree or graphics area.
4. Click Edit, Paste or click Paste on the Standard toolbar.

   A new sketch, *Sketch4*, is created on *Plane3*.

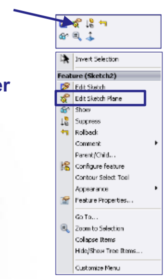


More About Copying Sketches

- External relations are deleted.
- For example, when you copied *Sketch3*, the geometric relations locating the center and defining the circumference were deleted.
- Therefore, *Sketch4* is under defined.
- To fully define *Sketch4*, add a Coradial relation between the copied circle and the original.
- If you sketch a profile on the wrong plane, move it to the correct plane using Edit Sketch Plane. Do not copy it.

To Move a Sketch to a Different Plane:

1. Right-click the sketch in the FeatureManager design tree.
2. Select Edit Sketch Plane from either the Context toolbar or the shortcut menu.
3. Select a different plane.
4. Click OK .



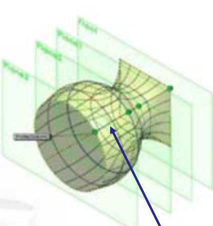
Loft Feature

- The Loft feature blends the 4 profiles to create the handle of the *chisel*.

1. Click Loft on the Features toolbar.

Creating the Loft Feature:

2. Select each profile.
   Click on each sketch in the same relative location – the right side.
3. Examine the preview curve.
   The preview curve shows how the profiles will be connected when the loft feature is created.

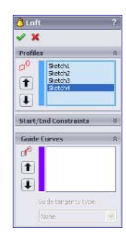


Preview curve

Creating the Loft Feature:

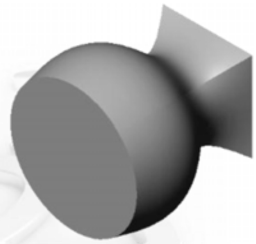
4. The sketches are listed in the Profiles box.

   The Up/Down arrows are used to rearrange the order of the profiles.



Creating the Loft Feature:

5. Click OK .

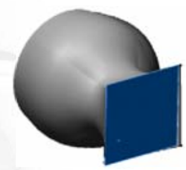


A Second Loft Feature Creates the Bit of the *Chisel*:

- The second Loft feature is composed of two profiles: Sketch5 and Sketch6.

To Create Sketch5:

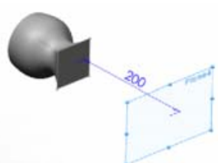
1. Select the square face.
2. Open a sketch.
3. Click Convert Entities .
4. Exit the sketch.



To Create *Sketch6*:

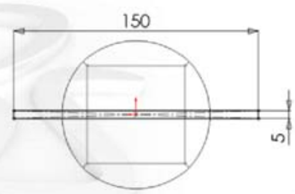
1. Offset *Plane4* ***behind*** the *Front* plane.

   Hold down Ctrl and drag the *Front* plane in the direction you want the offset to go.
2. The Plane PropertyManager appears.
3. Enter 200mm for Distance.
4. Click OK .



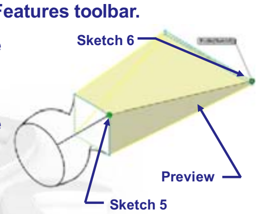
To Create *Sketch6*:

5. Open a sketch on *Plane4*.
6. Sketch a narrow rectangle centered on the origin.
7. Dimension the rectangle.
8. Exit the sketch.

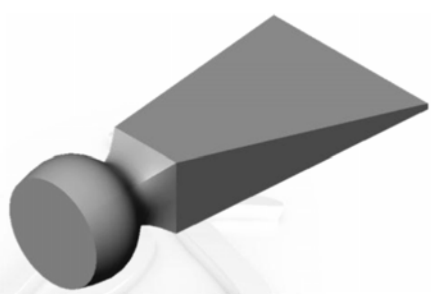


To Create the Second Loft Feature:

1. Click Loft on the Features toolbar.
2. Select *Sketch5* in the upper right corner of the square.
3. Select *Sketch6* in the upper right corner of the rectangle.
4. Examine the preview curve.
5. Click OK.

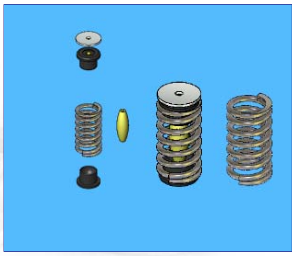


Finished *Chisel*

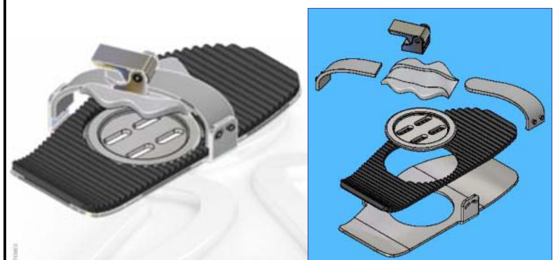


Spring Assembly

- Several parts were created in an earlier lesson
- The Spring will be created by sweeping a circle along a helix.

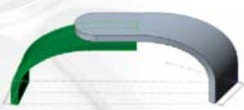


Binding Straps and Pad



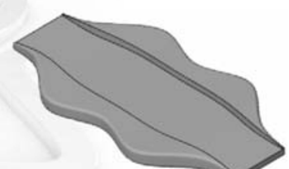
Binding Straps

- Straps created in a single part
- Solid bodies saved to separate part files
- Created as sweeps with guide curves
- Guide curves used to:
  - Control the sweep profile size and shape
  - Control the twist of the profile as it sweeps along the path



Foam Pad

- Created as a loft
- Created in the same part as the two straps
- Saved as separate part
- Flat and curved versions



Binding Pad

- Created as an In-context feature
- Pattern follows curve of Binding Base Plate
- Trimmed to shape of Binding Base Plate

8

Final Assembly

Review of Lesson 7 — Sweeps and Lofts

1 What is the primary difference between a sweep and a loft?

Answer: A sweep moves a single section or profile along a path, while a loft connects two or more profiles with the path being determined by the position of the profiles.

2 What are Composite Curves?

Answer: Composite Curves are a collection of lines, arcs and splines that act as a single curve.

3 How is the radius of a Full Round Fillet determined?

Answer: The radius is determined by the radius required for the fillet to be tangent to the three selected faces.

4 How do you hide a component in an assembly?

Answer: Either right-click the part and select **Hide**, or select the part and click **Hide/ Show Components**  on the Assembly toolbar.

5 Can you hide a feature in a part?

Answer: No, you can only hide components in an assembly.

6 How do you create an AVI recording of an assembly explode or collapse?

Answer:

- Create the explosion steps in the assembly.
- In the ConfigurationManager right-click the `ExplView` and select **Animate Explode** or **Animate Collapse**.
- Click **Save AVI**  on the Animation Controller.

5 Minute Assessment – #8 Answer Key

1 Where do you find ready-to-use hardware components?

Answer: In the Design Library under Toolbox and 3D Content Central.

2 True or False: Parts from Toolbox automatically size to the components they are being placed on.

Answer: False

3 How do you size Toolbox components as you are placing them?

Answer: Use the window that pops up to change the part properties.

4 In an assembly, parts are referred to as ______________?

Answer: Components

5 True or False: A fixed component is free to move.

Answer: False

6 What is a Bill Of Materials?

Answer: The Bill Of Materials, or BOM, is a list of all the parts and sub-assemblies used in an assembly.

7 What information can be shown in a balloon?

Answer: Item number, quantity, or a custom property.

Lesson 8 Quiz – Answer Key

1 When calculating the mass properties of a part, what density is used if there is no material applied to the part?

Answer: A default density is set in the Document Properties Options. Once a material is applied, this default value is overridden.

2 How do you apply a mate using Smart Mates?

Answer: Hold down the **Alt** key and drag the face or edge to be mated onto the edge or face it is to mate to.

3 True or False: Smart Mates can only add one mate at a time?

Answer: False. Smart Mates can add up to three mates with a single drag and drop operation.

4 What allows a part to be automatically mated when dragging it from the Windows Explorer?

Answer: Mate References stored in the individual part files.

5 How many mate references can be added to a part?

Answer: Each mate reference feature, can have primary, secondary and tertiary entities selected.

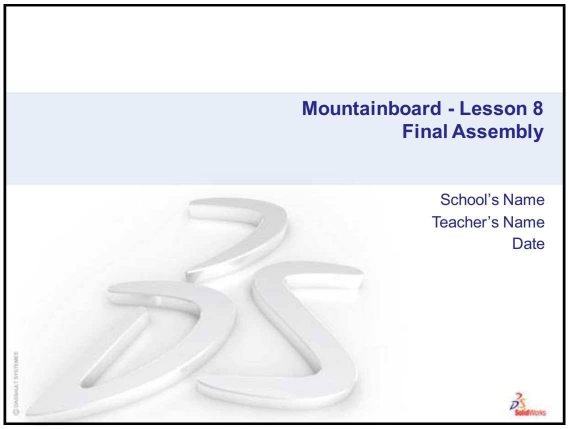
6 How do you change the orientation of the Move Triad?

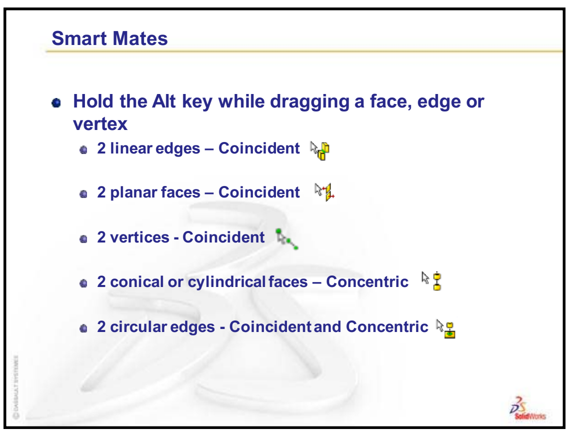
Answer:

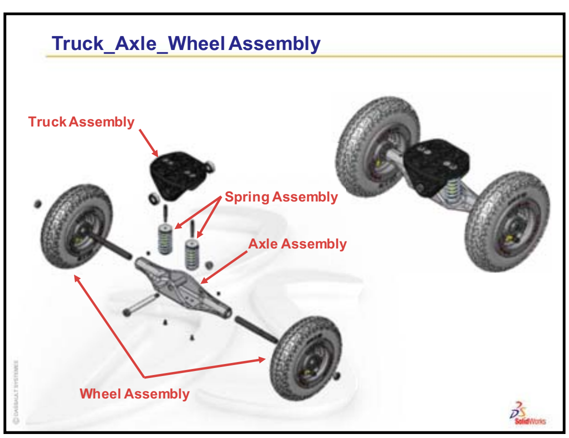
Right-click the center of the Triad and select Align To. Select a planar face on linear edge. The Triad will align itself to the face or edge.

Thumbnail Images of PowerPoint Slides

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

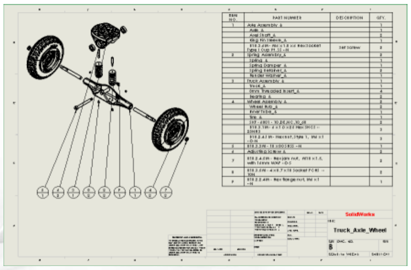






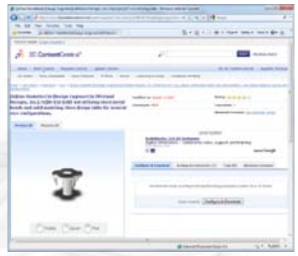
Assembly Drawings

- BOM - Bill of Materials
- Balloons



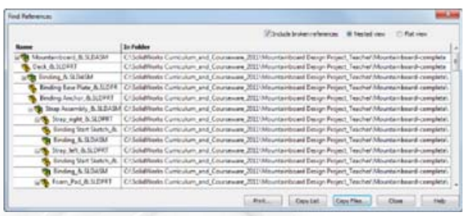
3D Content Central

- Online source of 3D models from component suppliers and individuals



Find References

- Locates the files being used by an assembly or drawing



Assembled Mountainboard

Exploded Mountainboard

9

Presenting Results

Notes to the Teacher

This lesson provides the mechanics of how to create various types of output files from SolidWorks. With these outputs, the students can make a variety of presentations and/or reports. It is left to the discretion of the Teacher as to what projects to assign the students.

Some suggestions:

- Have individual students or groups of students give an oral presentation about the Mountainboard design process.
- Do a design review using email. Have an edrawing sent to various students for markup. Return the markup files via email and present the results in class.
- Have students write a report about the design process including pictures and analysis results.
- Have students create marketing materials using the rendered images.
- Create a web page about the Mountainboard. Include rendered images and animations.
- Create assembly instructions for the Mountainboard using exploded drawings.

Changes for 2011

With SolidWorks 2011, PhotoWorks has been replaced by PhotoView 360. Most of the procedures to use PhotoView 360 are the same or similar to those used with PhotoWorks as almost everything is now done in core SolidWorks. Some of the things that have changed:

- There is no longer a Render Wizard.
- The primary light source in a rendering is environmental lighting as discrete lights are off by default.
- The RenderManager has been replaced by the DisplayManager which lists the visualization items of appearances, scenes, decals, light and cameras.
- You can no longer render to a printer or file, you just render to the render window and then save the images you want to separate files.
- There are far fewer options with PhotoView 360 because the primary setup is done in SolidWorks instead of PhotoView 360.

Customization

PhotoView 360 and the MotionManager provide significant latitude for each student to customize and display their finished project. The goal should be that no two finished and rendered mountainboards look alike.

The animation created during the lesson should just be considered a starting point for the students. They should be encouraged to create additional animations of the different sub-assemblies and render the animations if time and computer resources allow.

5 Minute Assessment – #9-1

1. How do you create an eDrawing?

   Answer: There are several ways:

   In SolidWorks, click **Publish an eDrawing** .

   Or, click **File, Publish to eDrawings** from the menu.

   Or, in SolidWorks click **File, Save As**. From the **Save as type** list, select eDrawing.
2. How do you send eDrawings to others?

   Answer: Email**.**
3. What is the quickest way to return to the default view?

   Answer: Click **Home** .
4. True or False: You can make changes to a model in an eDrawing.

   Answer: False. However if the eDrawing is review-enabled, you can measure geometry and add comments using markup tools.
5. True or False: You need to have the SolidWorks application in order to view eDrawings.

   Answer: False.
6. What eDrawings feature allows you to dynamically view parts, drawings, and assemblies?

   Answer: Animation.

5 Minute Assessment – #9-2 Answer Key

1. What is PhotoView 360?

   Answer: PhotoView 360 is a software application that creates photorealistic images from SolidWorks models.
2. List the rendering effects that are used in PhotoView 360?

   Answer: Appearances, Backgrounds, Decals, Lights and Shadows.
3. The ______________________ allows you to specify and preview appearances.

   Answer: Appearances, Scenes, and Decals tab.
4. Where do you set the scene background?

   Answer: Edit Scenes PropertyManager.
5. What is SolidWorks MotionManager?

   Answer: SolidWorks MotionManager is a software application that animates and captures motion of SolidWorks part and assemblies.
6. List the four types of animations that can be created using the Animation Wizard.

   Answer: Rotate Model, Explode View, Collapse View, Import Basic Motion/Motion Analysis.

Lesson 9 Vocabulary Worksheet — Answer Key

1. The ability to dynamically view an eDrawing: **Animate**
2. Halting a continuous play of an eDrawing animation: **Stop**
3. Command that allows you to step backwards one step at a time through an eDrawing animation: **Previous**
4. Non-stop replay of eDrawing animation: **Continuous Play**
5. Rendering of 3D parts with realistic colors and textures: **Shaded**
6. Go forward one step in an eDrawing animation: **Next**
7. Command used to create an eDrawing: **Publish**
8. Graphic aid that allows you to see the model orientation in an eDrawing created from a SolidWorks drawing: **3D Pointer**
9. Quickly return to the default view: **Home**
10. Command that allows you to use email eDrawings with others: **Send**

Lesson 9 Quiz — Answer Key

1 What is the window that shows you a thumbnail view of the whole eDrawing?

Answer: Overview window.

2 Which command displays wireframe as solid surfaces with realistic colors and textures?

Answer: **Shaded**.

3 How do you create an eDrawing?

Answer: Click **Publish an eDrawing** .

4 What action does the **Home** command perform?

Answer: Returns to the default view.

5 Which command performs a non-stop replay of eDrawing animation?

Answer: **Continuous Play**.

6 True or False — eDrawings only displays part files, but not assemblies or drawings.

Answer: False.

7 True or False — You can hide assembly components or drawing views.

Answer: True.

8 In an eDrawing created from a SolidWorks drawing, how do you view a sheet other than the one currently displayed?

Answer: Answers will vary but may include:

- In the Sheets tab of the eDrawing Manager, double-click the sheet you want to view.
- Click the sheet tab located below the graphics area of the eDrawings viewer.

9 What visual aid helps you identify model orientation in a drawing?

Answer: 3D Pointer.

10 Holding **Shift** and pressing an arrow key rotates a view 90-degrees at a time. How would you rotate a view 15-degrees at a time?

Answer: Press an arrow key without holding **Shift**.

11 What is PhotoView 360?

Answer: PhotoView 360 is a software application that creates realistic images from SolidWorks models.

12 What is SolidWorks MotionManager?

Answer: SolidWorks MotionManager is a software application that animates and captures motion of SolidWorks part and assemblies.

13 Where do you modify the scene background?

Answer: Scene PropertyManager - Background.

14 Image Background is the portion of the graphics area not covered by the ___________.

Answer: Model.

15 True of False. PhotoView 360 output renders to graphics window or renders to a file.

Answer: False.

16 SolidWorks MotionManager produces what type of file?

Answer: *.avi.

17 List the four types of animations that can be created using the Animation Wizard.

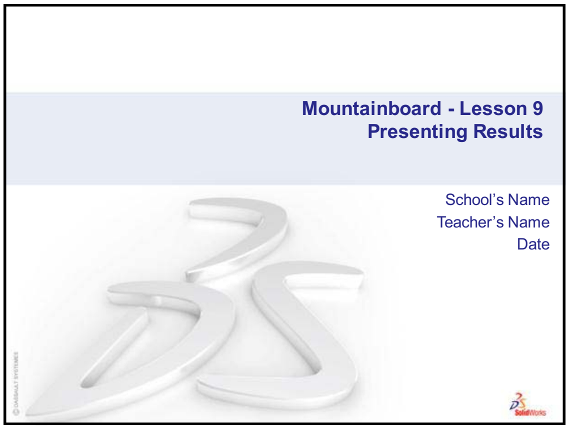
Answer: Rotate Model, Explode View, Collapse View, Import Basic Motion/Motion Analysis.

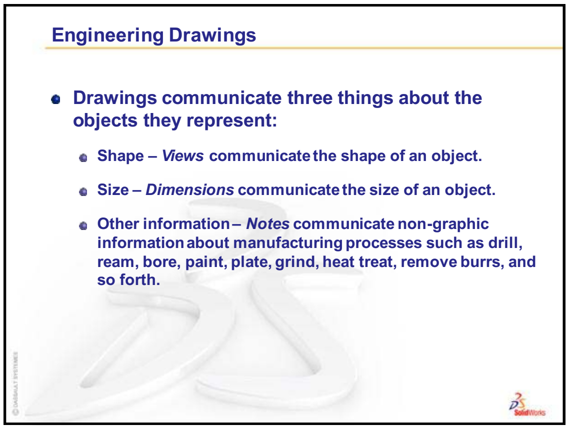
18 For a given animation, list three factors that affect the file size when the animation is recorded.

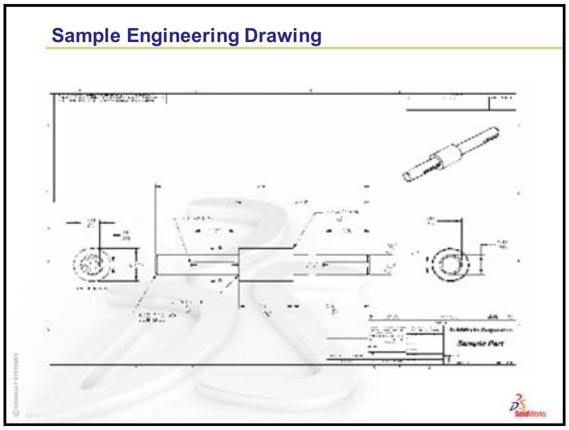
Answer: Possible answers include number of frames per second, type of renderer used, amount of video compression, number of key frames, and screen size. If the rendering is done with the PhotoView 360 buffer, the material, scene, and lighting effects such as shadows all affect file size.

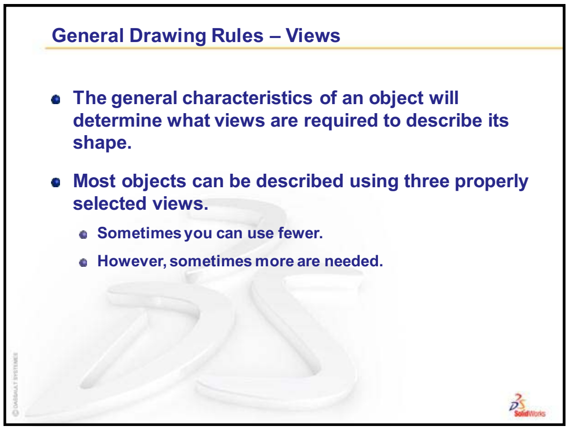
Thumbnail Images of PowerPoint Slides

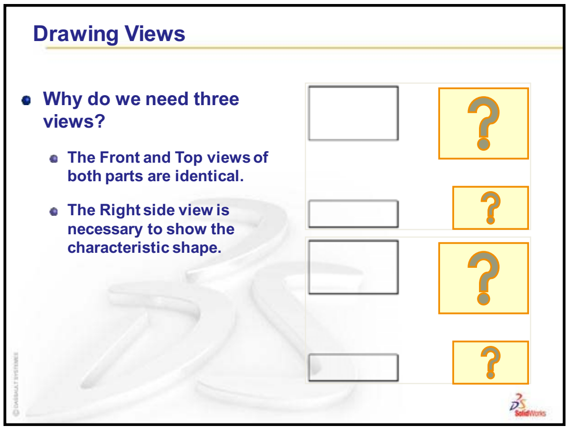
The following thumbnail images, arranged left to right, show the PowerPoint slides provided with this lesson.

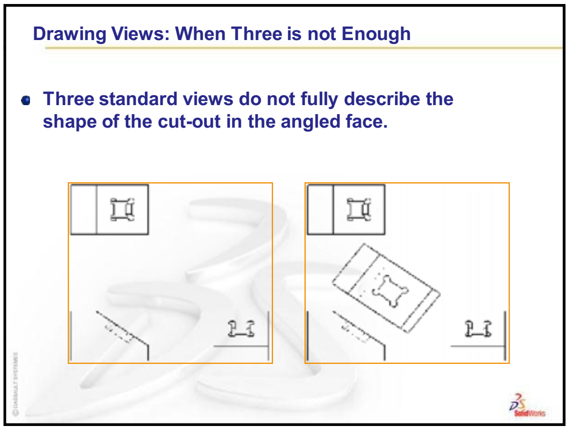






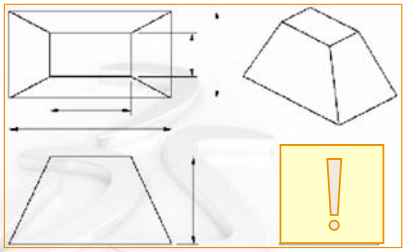






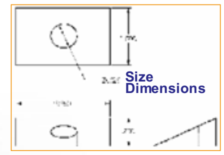
Drawing Views: When Three is too Many

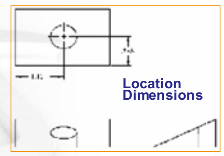
- The Right side view is unnecessary.



Dimensions

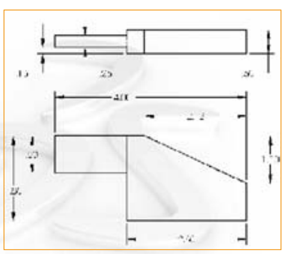
- There are two kinds of dimensions:
  - Size dimensions – how big is the feature?
  - Location dimensions – where is the feature?





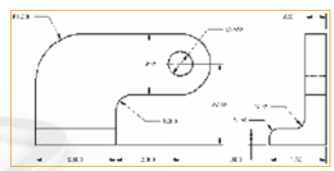
General Drawing Rules – Dimensions

- For flat pieces, give the thickness dimensions in the edge view, and all other dimensions in the outline view.



General Drawing Rules – Dimensions

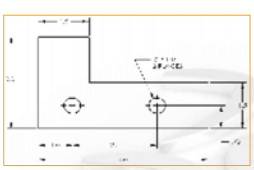
- Dimension features in the view where they can be seen true size and shape.



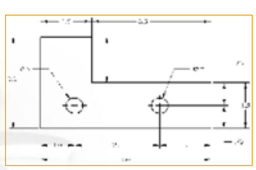
- Use diameter dimensions for circles.
- Use radial dimensions for arcs.

General Drawing Rules – Dimensions

- Omit unnecessary dimensions.



This

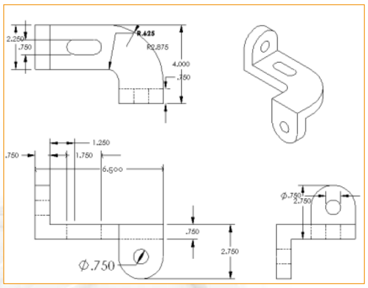


Not This

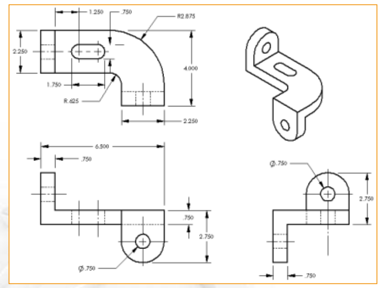
Dimension Guidelines – Appearance

- Place dimensions away from the profile lines.
- Allow space between individual dimensions.
- A gap must exist between the profile lines and the extension lines.
- The size and style of leader line, text, and arrows should be consistent throughout the drawing.
- Display only the number of decimal places required for manufacturing precision.
- Neatness counts!

Drawing Appearance – Not Good



Drawing Appearance – Much Better



What is a Drawing Template?

- A Drawing Template is the foundation for drawing information.

A drawing template specifies:

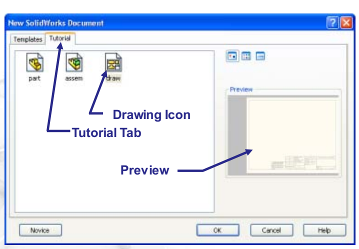
- Sheet (paper) size
- Orientation - Landscape or Portrait
- Sheet Format
  - Borders
  - Title block
  - Data forms and tables such as bill of materials or revision history

Drawing Templates Choices in SolidWorks

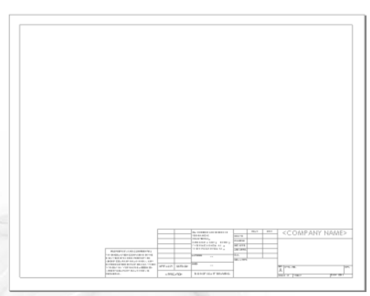
- Standard SolidWorks drawing template
- Tutorial drawing template
- Custom template
- No template

To Create a New Drawing Using a Document Template:

1. Click New on the Standard toolbar.
2. Click the Tutorial tab.
3. Double-click the drawing icon.



Sample Drawing Template



Edit Sheet vs. Edit Sheet Format

There are two modes in the drawing:

- Edit Sheet
  - This is the mode you use to make detailed drawings
  - Used 99+% of the time
  - Add or modify views
  - Add or modify dimensions
  - Add or modify text notes
- Edit Sheet Format
  - Change the title block size and text headings
  - Change the border
  - Incorporate a company logo
  - Add standard text that appears on every drawing

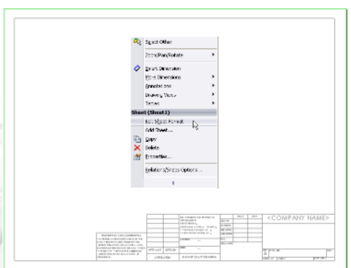
Title Block

- Contains vital part and/or assembly information.
- Each company can have a unique version of a title block.
- Typical title block information includes:

Company name	Material & Finish
Part number	Tolerance
Part name	Drawing scale
Drawing number	Sheet size
Revision number	Revision block
Sheet number	Drawn By/Checked By

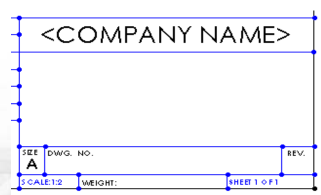
To Edit the Title Block:

1. Right-click in the graphics area, and select Edit Sheet Format from the shortcut menu.



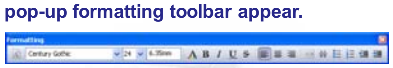
Editing the Title Block:

2. Zoom in on the title block.

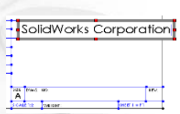


Editing the Title Block:

3. Double-click the note that says <COMPANY NAME>. The PropertyManager and the pop-up formatting toolbar appear.

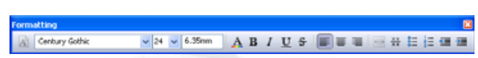


4. Enter your school name in the text insertion box.



Editing the Title Block:

5. Set the text justification to Align Left and change the size and style of the text font.



6. Click OK to apply the changes and close the PropertyManager.

Editing the Title Block:

7. Position the note so it is centered in the space.

Customizing the Part Name

Advanced Topic

- The name of the part or assembly shown on the drawing changes with every new drawing.
- It is not very efficient to have to edit the sheet format and the title block each time you make a new drawing.
- It would be nice if the title block would automatically be filled in with the name of the part or assembly that is shown on the drawing.
- This can be done.

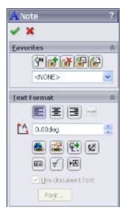
Editing the Part Name:

Advanced Topic

1. Click Note on the Annotation toolbar, or click Insert, Annotations, Note.

   The PropertyManager appears.

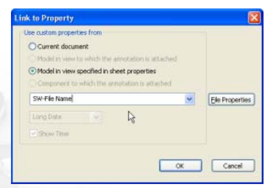
2. Click the Link to Property button .



Editing the Part Name:

Advanced Topic

3. Click Model in view specified in sheet properties, and choose SW-File Name from the list of properties.
4. Click OK to add the property.



Editing the Part Name:

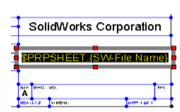
Advanced Topic

5. In the PropertyManager, set any other text properties such as justification, or font.

Editing the Part Name:

Advanced Topic

6. Click OK to apply the changes and close the PropertyManager.



Advanced Topic

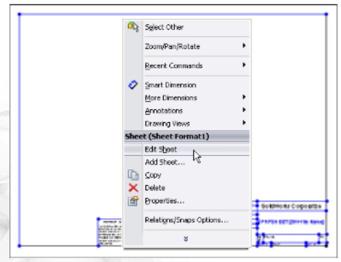
Advanced Topic

7. **Results.**

Currently the title block shows the text of the property. However, when the first view is added to the drawing, that text will change to become the file name of the referenced part or assembly.

Switching to Edit Sheet Mode:

1. **Right-click in the graphics area, and select Edit Sheet from the shortcut menu.**
2. **This is the mode you must be in when you make drawings.**



Detailing Options

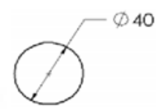
Dimensioning Standards

- **Dimensioning standards determine things such as arrowhead style and dimension text position.**
- **The Tutorial drawing template uses the ISO standard.**
- **ISO stands for International Organization for Standardization.**
- **ISO is widely used in European countries.**

Detailing Options

Dimensioning Standards

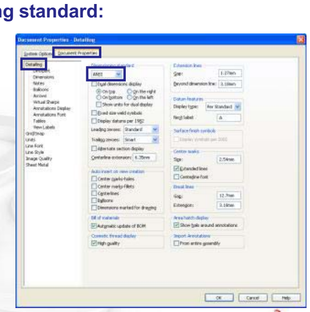
- **ANSI is widely used in the United States.**
- **ANSI stands for American National Standards Institute.**
- **Other standards include BSI (British Standards Institution) and DIN (Deutsche Industries-Normen).**
- **Customize the drawing template to use the ANSI standard.**



Detailing Options

Setting the dimensioning standard:

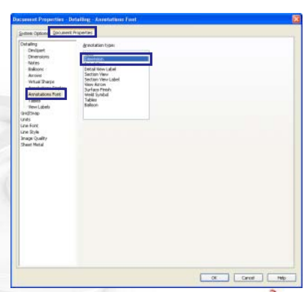
1. **Click Tools, Options.**
2. **Click the Document Properties tab**
3. **Click Detailing.**
4. **Select ANSI from the Dimensioning standard list.**
5. **Click OK.**



Detailing Options

Setting text fonts:

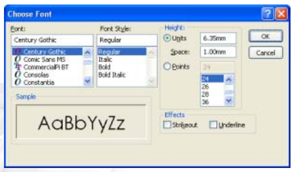
1. **Click Tools, Options.**
2. **Click the Document Properties tab**
3. **Click Annotations Font.**
4. **Select the annotation type from the list.**



Detailing Options

Setting text fonts continued:

5. The Choose Font dialog box opens.
6. Make the desired changes and click OK.



Saving a Custom Drawing Template:

1. Click File, Save As...
2. From the Save as type: list, click Drawing Templates.

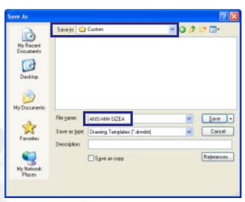
The system automatically jumps to the directory where the templates are installed.

3. Click to create a new folder.

Saving a Custom Drawing Template:

4. Name the new folder *Custom*.
5. Browse to the *Custom* folder.
6. Enter *ANSI-MM-SIZEA* for the file name.
7. Click Save.

Drawing templates have the suffix *.drwdot

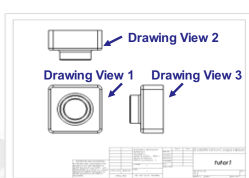


Creating a Drawing – General Procedure

1. Open the part or assembly you wish to detail.
2. Open a new drawing of the desired size.
3. Add views: usually three standard views plus any specialized views such as detail, auxiliary, or section views.
4. Insert the dimensions and arrange the dimensions on the drawing.
5. Add additional sheets, views and/or notes if required.

To Create Three Standard Views:

1. Click Standard 3 View .
2. Select Tutor1 from the Window menu.
3. Click OK.



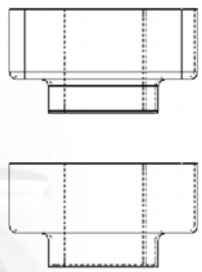
The drawing window reappears with the three views of the selected part.

Working with Drawing Views

- To select a view, click the view boundary. The view boundary is displayed in green.
- Drawing views 2 and 3 are aligned with view 1.
- Drag Drawing View1 (Front). Drawing View 2 (Top) and Drawing View 3 (Right) move, staying aligned to Drawing View1.
- Drawing View 3 can only be dragged left or right.
- Drawing View 2 can only be dragged up or down.

Working with Drawing Views

- Hidden line representation.
  - Hidden Lines Visible is usually used in orthographic views.
  - Hidden Lines Removed is usually used in isometric views.
- Tangent edge display.
  - Right-click inside the view border.
  - Select Tangent Edge, Tangent Edges Removed from the shortcut menu.



Dimensioning Drawings

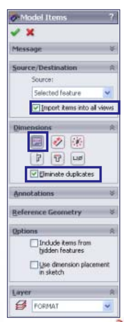
- The dimensions used to create the part can be imported into the drawing.
- Dimensions can be added manually using the Dimension tool .

Associativity

- Changing the values of imported dimensions will change the part.
- You cannot change the values of manually inserted dimensions.

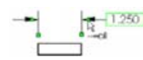
To Import Dimensions into the Drawing:

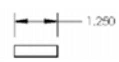
1. Click Model Items on the Annotation toolbar, or click Insert, Model Items.
2. Click the Import items into all views check box.
3. Click the option for Marked for drawing and Eliminate duplicates check box.
4. Click OK.



Manipulating Dimensions

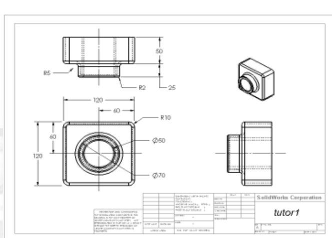
- Moving dimensions:
  - Click the dimension text.
  - Drag the dimension to the desired location.
  - To move a dimension into a different view, press and hold the Shift key while you drag it.
- Deleting dimensions:
  - Click the dimension text, and then press the Delete key.
- Flipping the arrows:
  - Click the dimension text.
  - A green dot appears on the dimension arrows.
  - Click the dot to flip the arrows in or out.





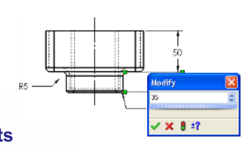
Finish the Drawing

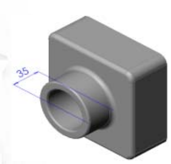
- Position the views.
- Arrange the dimensions by dragging them.
- Set hidden line removal and tangent edge display.



Associativity

- Changing a dimension on the drawing changes the model.
  - Double-click the dimension text.
  - Enter a new value.
  - Rebuild.
- Open the part. The part reflects the new value.
- Open the assembly. The assembly also reflects the new value.



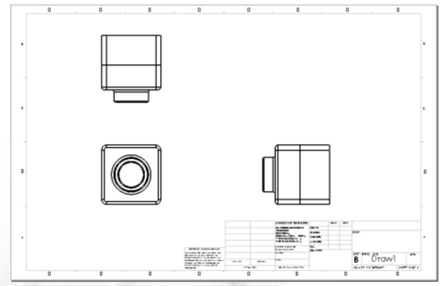


Multi-sheet Drawings

Drawings can contain more than one sheet.

- The first drawing sheet contains Tutor1.
- The second drawing sheet contains the Tutor assembly.
- Use the B-size landscape (11" x 17") drawing Sheet Format.
- Add 3 standard views.
- Add an Isometric view of the assembly. The Isometric view is a model view.

Three View Drawing of Assembly

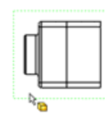


Model Views

- A model view shows the part or assembly in a specific orientation.
- Examples of model views are:
  - Standard Views such as Front, Top or Isometric view.
  - User-defined view orientations that were created in the part or assembly.
  - The current view in a part or assembly.

To Insert a model View:

1. Click model View , or click Insert, Drawing view, Model.
2. Click inside the border of an existing view.



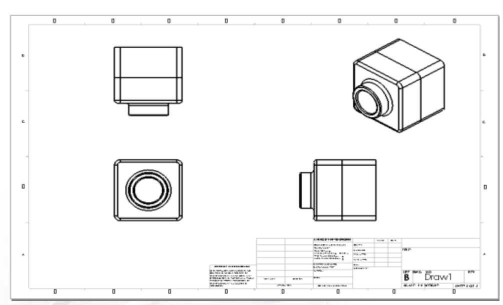
Important: Do not click directly on one of the parts in the assembly. Doing so will create a named view of that specific part.

Inserting a Model View:

3. A selection of model view icons appears in the PropertyManager.

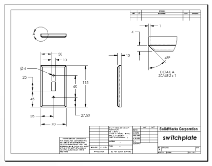
   Select the desired view, in this case, Isometric , from the selection.
4. Place the view in the desired location on the drawing.

Isometric View Added to Drawing



Specialized Views

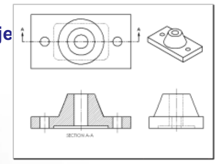
Detail View – used to show enlarged view of something.



1. Click , or click Insert, Drawing View, Detail.
2. Sketch a circle in the "source" view.
3. Position the view on drawing.
4. Edit the label to change scale.
5. Import dimensions or drag them into view.

Specialized Views

Section View – used to show internal aspects of obje

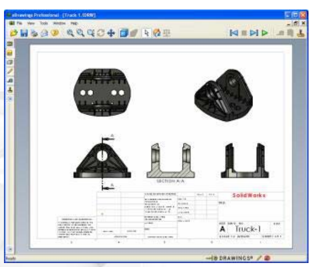


1. Click , or click Insert Drawing View, Section.
2. Sketch line in the "source" view.
3. Position the view on drawing.
4. Section view is automatically crosshatched.
5. Double-click section line to reverse arrows.

eDrawings

Animate, View, and Email eDrawings.

- Allows others to view parts, assemblies, and drawings outside of SolidWorks.
- Files are compact enough to email.



Publishing eDrawings

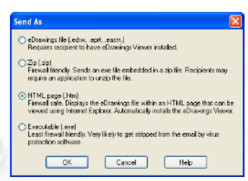
- Creating an eDrawing is quick and simple.
- Click to publish an eDrawing from any SolidWorks file.
- You can create eDrawings from AutoCAD® drawings, too.

View eDrawings Dynamically

- Click Continuous Play Play to view a continuously running animation of the eDrawing.
- Step through the eDrawing animation using Next Next and Previous Previous .
- Click Stop Stop to end the animation.

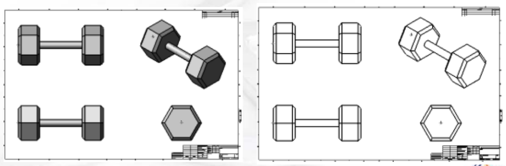
Sending eDrawings

- Click Send or File, Send to email an eDrawing.
- Several email-compatible formats.
- The recipient does not need to have the SolidWorks application to view the file.



Shaded View

- By default eDrawing views are shaded.
- Click Shaded to view an eDrawing as wireframe.
- Click again to view an eDrawing as shaded.



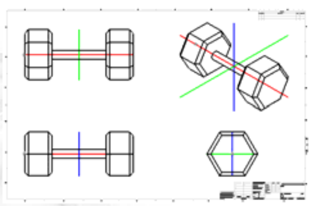
Resetting the View

- Click Home to reset the view to the default.
- Home allows you to look at the eDrawing and then quickly return it to the default view.

3D Pointer

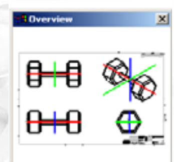
Helps you to see the orientation of the model in an eDrawing created from a drawing file.

- Click to display the 3D pointer.
  - Red — X-Axis
  - Blue — Y-Axis
  - Green — Z-Axis



Overview Window

- Small thumbnail view of the eDrawing.
- Click Overview Window to display the Overview window.



What is PhotoView 360?

A software application that creates realistic images from SolidWorks models.

PhotoView 360 uses rendering effects such as:

- Appearances
- Lights
- Shadows
- Backgrounds

Appearances

Appearances specify the visual properties of a model's surface.

Properties are:

- Color
- Mapping
- Surface Finish
- Illumination

Image Background

The portion of the graphics area not covered by the model.

- Background styles vary in complexity and rendering speed.
- Background styles controlled by Scene Editor.
- Incorporate advanced rendering effects into a PhotoWorks Scene.
  - Shadows
  - Reflections

SolidWorks MotionManager

What is SolidWorks MotionManager?

- SolidWorks MotionManager animates and captures motion of SolidWorks parts and assemblies.
- SolidWorks MotionManager generates Windows-based animations (*.avi files). The *.avi file uses a Windows-based Media Player.
- SolidWorks MotionManager can be combined with PhotoWorks.

Renderer Options

The Renderer affects the quality of the saved image. There are two options:

- SolidWorks screen
- PhotoView 360 buffer

Factors Affecting File Size

- Number of frames per second
- Renderer used
  - PhotoView 360 buffer creates a larger file than SolidWorks screen
- If using PhotoView 360 buffer:
  - Appearances
  - Background
  - Shadows
  - Multiple-light sources
- Video compression
- Key frames

Output

- Different forms of output are required for the different phases of the design process
  - During Design
  - Manufacturing
  - Marketing
  - Project Support

Types of Output

- Screen prints
- Photorealistic prints and image files
- Animations
- 2D Drawings
- 3D Drawings (eDrawings)
- PDF (Portable Document Format)
- SolidWorks Files
- Non-SolidWorks Files

Screen Prints

- Prints as you would any other document through File, Print menu
  - Scaled or full size
  - Color will be the same as on the computer screen
  - Use Realview if available to make it look more realistic
  - Add headers and footers to help identify the print

Screen Print

- RealView Off

Screen Print

- RealView On

Image Files

- From SolidWorks
  - JPEG (*.jpg)
  - Adobe PhotoShop (*.psd)
  - TIFF (*.tif)
  - Adobe Illustrator (*.ai)
  - TIFF (*.tif)
  - Adobe Illustrator (*.ai)
- From PhotoView 360
  - Windows Bitmap (*.bmp)
  - TARGA (*.tga)
  - JPEG (*jpg)
  - TIFF (*.tif)
  - High Dynamic Range (*.hdr)

Image Files

- Raster Images
  - Store information about each pixel
  - Do not scale well
  - Generally larger file sizes
- Vector Images
  - Store information about where entities start and end
  - Scale well
  - Generally smaller file sizes

Image in PowerPoint Slide


Truck


The Deck


Wheel Assembly


Spring Assembly


Binding Assembly


Mountainboard

CODEC

- COmpressor/DECompressor
  - Used to reduce video file size
  - CODEC used to record video file must also be available on machine used to play the video file