SolidWorks®

Engineering Design Project

The Mountainboard



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About This Course

The SolidWorks Engineering Design Project, The Mountainboard and its supporting materials is designed to assist you in learning SolidWorks in an academic setting. The SolidWorks Engineering Design Project, The Mountainboard offers a competency-based approach to learning 3D design concepts and techniques.

Online Tutorials

The *SolidWorks Engineering Design Project* is a companion resource and supplement for the SolidWorks Online Tutorials.

Accessing the Tutorials

To start the Online Tutorials, click **Help, SolidWorks Tutorials**. The SolidWorks window is resized and a second window will appears next to it with a list of the available tutorials. As you move the pointer over the links, an illustration of the tutorial will appear at the bottom of the window. Click the desired link to start that tutorial.

Conventions

Set your screen resolution to 1280x1024 or better for optimal viewing of the tutorials.

The following icons appear in the tutorials:

Next Moves to the next screen in the tutorial.

- Represents a note or tip. It is not a link; the information is to the right of the icon. Notes and tips provide time-saving steps and helpful hints.
- You can click most toolbar buttons that appear in the lessons to flash the corresponding SolidWorks button. The first time you click the button, an ActiveX



control message may appear: An ActiveX control on this page might be unsafe to interact_with other parts of the page. Do you want to allow this interaction? This is a standard precautionary measure. The ActiveX controls in the Online Tutorials will <u>not</u> harm your system. If you click **No**, the scripts are disabled for that topic. Click **Yes** to run the scripts and flash the button. **Timer** shows the estimated time to complete the tutorial.

Open File or **Set option** automatically opens the file or sets the option.

Show me... shows a video about this step.

Q

A closer look at... links to more information about a topic. Although not required to complete the tutorial, it offers more detail on the subject.



Why did I... links to more information about a procedure, and the reasons for the method given. This information is not required to complete the tutorial.

Printing the Tutorials

If you like, you can print the Online Tutorials by following this procedure:

1 On the tutorial navigation toolbar, click **Show**

This displays the table of contents for the Online Tutorials.

2 Right-click the book representing the lesson you wish to print and select **Print** from the shortcut menu.

The **Print Topics** dialog box appears.

- 3 Select Print the selected heading and all subtopics, and click OK.
- 4 Repeat this process for each lesson that you want to print.

Using This Course

This course is not just this book. The *SolidWorks Engineering Design Project, The Mountainboard* is the focal point of the SolidWorks course — the road map for it. The supporting materials that are in the SolidWorks Online Tutorials give you a lot of flexibility in how you learn SolidWorks.

Learning 3D design is an interactive process. You will learn best when you explore the practical applications of the concepts you learn. This course has many activities and exercises that will allow you to put design concepts into practice. Using the provided files, you can do so quickly.

The lessons for this course are designed to balance lecture and hands-on learning. There are also assessments and quizzes that give you additional measures of your progress.

Lesson Structure

Each lesson contains the following components:

- □ Goals of the Lesson Clear objectives for the lesson.
- □ Before Beginning the Lesson Prerequisites, if any, for the current lesson.
- Review of Previous Lesson You reflect back on the material and models described in the previous lesson with questions and examples. Answer these questions to reinforce concepts.
- □ Lesson Outline Describes the major concepts explored in each lesson.
- □ Active Learning Exercises You create parts, assemblies and drawings that will make up the final project, The Mountainboard.
- □ 5-minute Assessments These review the concepts developed in the outline of the lesson and the active learning exercises.
- □ Exercises and Projects These exercises and projects provide additional material to practice the concepts learned in the lesson.
- □ Lesson Quizzes Fill in the blank, true/false and short answer questions compose the lesson quizzes.
- □ Lesson Summary Quick recap of the main points of the lesson.

Goals of This Lesson

- □ Become familiar with the Microsoft Windows interface.
- □ Become familiar with the SolidWorks interface.

Before Beginning This Lesson

- □ Verify that Microsoft Windows is loaded and running on your classroom/lab computer.
- Verify that the SolidWorks software is loaded and running on your classroom/lab computer in accordance with your SolidWorks license.
- □ Load the training files.

Outline of Lesson 1

- □ Active Learning Exercise Using the Interface
 - Starting a Program
 - Exiting a Program
 - Searching for a File or Folder
 - Opening an Existing File
 - Saving a File
 - Copying a File
 - Resizing Windows
 - SolidWorks Windows
 - Toolbars
 - Mouse Buttons
 - Context-sensitive Shortcut Menus
 - Getting Online Help

Active Learning Exercise — Using the Interface

Start the SolidWorks application, search for a file, save the file, save the file with a new name, and review the basic user interface.

The step-by-step instructions are given below.

Starting a Program

1 Click the **Start** button in the lower left corner of the window. The **Start** menu appears. The **Start** menu allows you to select the basic functions of the Microsoft Windows environment.

Note: Click means to press and release the left mouse button.

- 2 From the Start menu, click All Programs, SolidWorks, SolidWorks as shown below.
 - **Note:** Depending on how SolidWorks was installed on your computer, the version and the Service Pack number, 2011 SP2 for instance, may be included or not listed. If your computer is running the 32 bit version of Windows, the x64 Edition will not be listed as part of the program name.

The SolidWorks application program is now running.



Note: Your **Start** menu may appear different than the illustration depending on which version of the operating system is loaded on your system.

TIP: A desktop shortcut is an icon that you can double-click to go directly to the file or folder represented. If your system desktop has a shortcut to the SolidWorks application program, you can start the program by double-clicking the left mouse button on this shortcut. The illustration shows the SolidWorks shortcut.



Exit the Program

To exit the application program, click **File**, **Exit** or click \checkmark on the main SolidWorks window.

Searching for a File or Folder

You can search for files (or folders containing files) which is useful if you cannot remember the exact name of the file that you need.

Windows has different methods to search for files through the Start menu and Windows Explorer.

Search Box

The Search Box is located on the Start menu. To search from the start menu, click **Start** and then type in the file name or partial file name you are looking for. Using the Search Box from the Start menu, Windows will only search indexed files.

Windows Explorer

In Widows Explorer, there is a search box in the upper right corner of the window. When using this search box, Windows will search the folder or library that is active in Windows Explorer and search for file names, text in the file content and properties. When searching a library, Windows will also search all sub folders.

Search for a file

We will now use Search to find a file.

- 1 Right-click Start, and click Open Windows Explorer.
- 2 In the left pane of Windows Explorer, locate and select the Mountainboard Design Project folder.
- 3 In the Search Box in the upper right corner of the Window, type dumb* to locate a file with a name that starts with the letters dumb.

As you type, Windows will be searching the selected folder and sub-folders for any files that match what you have typed.

G V Search Results in Mountainboard Design R	Projec	:t_Student ►	- ++ du	mb*			×
File Edit View Tools Help			_				
Organize 🔻 Save search Burn							0
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🌗 Mountainboard	📄 🔚 Libraries 🚜 Homegroup 👎 Computer 📲 Custom 🏉 Internet						
Training Templates							
Mountainboard Design Project Teacher	Ŧ	•	III				•
1 item							

TIP: The asterisk (*) is a wild card. The wild card allows you to enter part of a file name and search for all files and folders that contain that piece.

Opening an Existing File

1 Double-click on the SolidWorks part file Dumbell.

This opens the Dumbell file in SolidWorks. If the SolidWorks application program is not running when you double-click on the part file name, the system starts the SolidWorks application program and then opens the part file that you selected.

TIP: Use the left mouse button to double-click. Doubleclicking with the left mouse button is often a quick way of opening files from a folder.

You could have also opened the file by selecting **File**, **Open** from the Windows Explorer menu.

Saving a File

1 Click **Save [**] to save changes to a file.

It is a good idea to save the file that you are working whenever you make changes to it.

Copying a File

Notice that Dumbell is not spelled correctly. It is supposed to have two "b's".

1 Click **File**, **Save As** to save a copy of the file with a new name.

The **Save As** window appears. This window shows you in which folder the file is currently located, the file name, and the file type.

2 In the File Name field enter the name Dumbbell and click

Save.

A new file is created with the new name. The original file still exists. The new file is an exact copy of the file as it exists at the moment that it is copied.

Resizing Windows

SolidWorks, like many applications, uses windows to show your work. You can change the size of each window.

- 1 Move the cursor along the edge of a window until the shape of the cursor appears to be a two-headed arrow.
- 2 While the cursor still appears to be a two-headed arrow, hold down the left mouse button and drag the window to a different size.
- **3** When the window appears to be the size that you wish, release the mouse button.

Windows can have multiple panels. You can resize these panels relative to each other.

- 4 Move the cursor along the boarder between two panels until the cursor appears to be two parallel lines with perpendicular arrows.
- **5** While the cursor still appears to be two parallel lines with perpendicular arrows, hold down the left mouse button and drag the panel to a different size.
- 6 When the panel appears to be the size that you wish, release the mouse button.

	Save As
у.	Search Lesson01 > • •
-	Organize 🔻 New folder
e v v	 SolidWorks Curriculum_and_Courseware_2011 Mountainboard Design Project_Student Images Lessons Lesson01 Exercises Lesson02 Lesson03 Lesson04 Lesson05
	File name: Dumbbell.SLDPRT
	Save as type: Part (*.prt;*.sldprt)
e	Description: Add a description
d	Save as copy References Hide Folders Save Cancel

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The SolidWorks User Interface

The SolidWorks user interface is a native Windows interface, and as such behaves in the same manner as other Windows applications. Some of the more important aspects of the interface are identified below.



SolidWorks Document Windows

SolidWorks document windows have two panels. One panel provides non-graphic data. The other panel provides graphic representation of the part, assembly, or drawing.

The leftmost panel of the window contains the FeatureManager[®] design tree, PropertyManager, ConfigurationManager and DisplayManager.

 Click each of the tabs at the top of the left panel and see how the contents of the window changes.

The rightmost panel is the Graphics Area, where you create and manipulate the part, assembly, or drawing.

2 Look at the Graphics Area. See how the dumbbell is represented. It appears shaded, in color, and in an isometric view. These are some of



the ways in which the model can be represented very realistically.

Task Pane

The SolidWorks Task Pane is a window menu that contains six or more panels: SolidWorks Resources, the Design Library, File Explorer, View Palette, Appearances, Scenes and Decals, and Custom Properties. The panels are used to access existing geometry. It can be opened/closed and moved from its default position on the right side of the interface.

Mouse Buttons

Mouse buttons operate in the following ways:

- □ Left Selects menu items, entities in the graphics area, and objects in the FeatureManager design tree.
- **Right** Displays the context-sensitive shortcut menus.
- □ Middle Rotates, pans, and zooms the view of a part or an assembly, and pans in a drawing.

Mouse Gestures

Mouse gestures provide a quick way to invoke up to eight different commands. They can be customized separately for sketches, parts, assemblies and drawings. To use mouse gestures, right-click in a blank area of the graphics window and then drag the mouse pointer over the desired command.





Toolbars

Toolbar buttons are shortcuts for frequently used commands. You can set toolbar placement and visibility based on the document type (part, assembly, or drawing).

SolidWorks remembers which toolbars to display and where to display them for each document type.

1 Click View, Toolbars.

A list of all toolbars displays. The toolbars with a check mark beside them are visible; the toolbars without a check mark are hidden.

Standard Views	X

- 2 Click the toolbar name to turn its display on or off. If it is not already on, click **View** to turn the **View** toolbar on.
- **3** Turn several toolbars on and off to see the commands.

Adding Commands to Toolbars

Toolbars start with the most frequently tools on them, but you can customize each toolbar by adding or removing commands as needed. To add additional commands, click **Tools**, **Customize**. Select the **Commands** tab.

Commands are organized by categories. Select a category, then drag the desired command to a toolbar.

Customize	ि <mark>४</mark>
Toolbars Commands Menus Keyboard Mouse Gestures Options	
Categories: Flyout Toolbars 2D To 3D Align Annotation Assembly Blocks Curves Dimensions/Relations DimApert Drawing Explode Sketch Fastening Features Layout Tools Line Format Macro Mold Tools Quick Snaps Reference Geometry Render Tools Screen Capture Selection Filter Sheet Metal Sketch Solid/Works Office Spline Tools Standard Standard Standard News Surfaces Table Toolbox Tools Select a category, then click a button to see its description. Drag the I	Buttons Image: A for the second sec
	OK Cancel Help

Heads-up View Toolbar

The Heads-up View toolbar is a transparent toolbar that contains many common view manipulation commands. Many of the icons (such as the Hide/Show Items icon shown) are Flyout Tool buttons that contain other options. These flyouts contain a small down arrow \mathbb{S}_{-} to access the other commands.

Command Manager

The Command Manager is a multifunction toolbar. Its contents can be adjusted quickly so that it may function in place of several toolbars.

When you click a tab on the Command Manager, the CommandManager updates to show that toolbar. For example, if you click **Sketch** tab, the **Sketch** toolbar appears in the CommandManager.



Arranging Toolbars

Toolbars may be positioned anywhere on the screen. If a toolbar displays its name, then it is floating and can be positioned anywhere on the screen. If a toolbar is positioned around the edge of the screen and is not displaying its name, it is docked.





Position the Toolbars

To make sure everyone's view of SolidWorks is the same, we will use the default setup of SolidWorks which uses the Command Manager initial toolbars and their locations.

- 1 Click View, Toolbars.
- **2** Select the following toolbars:
 - Command Manager
 - View (Heads-Up)
 - Task Pane
- **3** Clear the **MotionManager**.

>	CommandManager
►	Use Large Buttons with Text
æ	2D to 3D
	3Dcontrol
D	Align
\mathbf{A}	Annotation
٩	Assembly
A°	Blocks
γ	Curves
ر ا	Dimensions/Relations
\$	DimXpert
	Display States
	Drawing
-	eDrawings 2008
a ^r	Explode Sketch
6 1	Fastening Feature
	Features
\mathbf{B}_I	Formatting

4 The SolidWorks window should look like the image below.



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Context

Toolbar

offer more menu items.

Shortcut Menus

Shortcut menus give you access to a wide variety of tools and commands while you work in SolidWorks. When you move the pointer over geometry in the model, over items in the FeatureManager design tree, or over the SolidWorks window borders, right-clicking pops up a context toolbar and shortcut menu of commands that are appropriate for wherever you clicked. Clicking an item will pop up a context toolbar.

You can access the "more commands menu" by selecting the double-down arrows imes in the menu. When you select the double-down arrows, the shortcut menu expands to

The shortcut menu provides an efficient way to work

without continually moving the pointer to the main pull-down menus or the toolbar buttons.

While the context toolbar makes more efficient use of space and reduces mouse travel, it can be difficult to use if you are not familiar with the various icons. The shortcut menu can be shown, without the context toolbar by customizing our setup.

To remove the context toolbar and have all the commands shown in the shortcut menu, click **Tools, Customize**. Clear **Show in shortcut menu**.



Show in shortcut menu

Note: The Customize menu is only available when a SolidWorks document is open.

Shortcut Key

Pressing the "**S**" key on the keyboard will access a customizable toolbar. Like mouse gestures, this toolbar is different for sketches, parts, assemblies and drawings.

Getting Online Help

If you have questions while you are using the SolidWorks software, you can find answers in several ways.

Note: If the Help button **?** does not appear in the Standard toolbar, you can add it. To do so, click **Tools**, **Customize**, **Commands**, and the toolbar that you wish to add the button to. In this case, click **Standard**. The available buttons for that toolbar display. Drag the button to the toolbar at the top of the SolidWorks window.

1 Click **The International Click Tells** or **Help, SolidWorks Help Topics** in the menu bar.

The online help appears.

2 Click **2** on the PropertyManager.

Quick Tips

Quick Tips are part of the on-screen help system. They provide guidance to users unfamiliar SolidWorks by asking "What would you like to do?".

Clicking on the task you would like to accomplish will cause the appropriate commands to be highlighted.



5 Minute Assessment — #1

- 1 Search for the SolidWorks part file Paper Towel Base. How did you find it?
- 2 What is the quickest way to search for a file?
- **3** How do you open the file from the **Search Results** window?
- **4** How do you start the SolidWorks program?
- **5** What is the quickest way to start the SolidWorks program?

Lesson 1 Vocabulary Worksheet

N	Vame:0	Class:	Date:
D pr	Directions: Answer each question by writing th rovided.	e correct ans	swer or answers in the space
1	Shortcuts for collections of frequently used of	commands:	
2	Command to create a copy of a file with a ne	ew name:	
3	One of the areas that a window is divided int	:	
4	The graphic representation of a part, assemble	ly, or drawin	g:
5	Character that you can use to perform wild c	ard searches	:
6	Area of the screen that displays the work of a	a program: _	
7	Icon that you can double-click to start a prog	ram:	
8	Action that quickly displays menus of freque	ently used or	detailed commands:
9	Command that updates your file with change	es that you ha	ave made to it:
10	Action that quickly opens a part or program:		
11	The program that helps you create parts, asso	emblies, and	drawings:
12	Panel of the SolidWorks window that display assemblies, and drawings:	vs a visual re	presentation of your parts,
13	Technique that allows you to find all files and set of characters:	d folders that	t begin or end with a specified

Lesson 1 Quiz

N	ame:	Class:	Date:
Di pr	rections: Answer each question by wr ovided or circle the answer as directed	iting the correct an l.	swer or answers in the space
1	How do you start the SolidWorks app	lication program?	
2	Which command would you use to cre	eate a copy of your	file?
3	Where do you see a 3D representation	n of your model?	
4	Look at the illustration (at right). Wha of frequently used commands called?	t is this collection	Standard Views ↓
5	How would you find a file if you coul	d not remember th	e whole file name?
6	Which command would you use to pr	eserve changes tha	t you have made to a file?
7	Which character helps you perform a	wild card search?	
8	Circle the cursor that is used to resize	a window.	kg ((∿ =
9	Circle the cursor that is used to resize		
10	Circle the button that is used to get on	lline help.	ینې د رونې د

Lesson Summary

- □ The Start menu is where you go to start programs or find files.
- □ You can use wild cards to search for files.
- □ There are short cuts such as right-click and double-click that can save you work.
- □ File, Save allows you to save updates to a file and File, Save As allows you to make a copy of a file.
- □ You can change the size and location of windows as well as panels within windows.
- The SolidWorks window has a Graphics Area that shows 3D representations of your models.

Lesson 2: Basic Functionality

Goals of This Lesson

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



□ This part is the center anchor for each of the two bindings. The Mountainboard uses two of these parts, one for each binding.

Before Beginning This Lesson

□ Complete the previous lesson: Using the Interface.

Resources for This Lesson

This lesson plan corresponds to the following lessons in the SolidWorks Online Tutorial:

- \Box Lesson 1 Parts
- □ Lesson 3 Drawings
- □ Fillets

For more information about the Online Tutorials, See "Online Tutorials" on page 1.

Review of Lesson 1 — Using the Interface

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

- □ Use windows to view files.
- □ Use the mouse to select buttons, menus, and model elements.
- □ Run programs like SolidWorks mechanical design software.
- □ Find, open, and work with files.
- □ Create, save, and copy files.
- □ SolidWorks runs on the Microsoft Windows graphical user interface.
- □ Click **Start**, and type a file name in the Search Box to find files or folders.
- □ The mouse lets you move around the interface. Discuss the uses of:

Click

Double-click Right-click

- □ The quickest way to open a file is to double-click on it.
- □ Saving a file preserves the changes that you have made to it.
- □ SolidWorks windows display graphic and non-graphic model data.
- □ Toolbars display frequently used commands.

Outline of Lesson 2

- □ In Class Discussion The design process
 - Stating goals
 - Iterative nature of design
- Course Project Overview The Mountainboard
 - Project goals
- In Class Discussion The SolidWorks Model
 - Parts
 - Assemblies
 - Drawings
- □ Active Learning Exercise, Part 1 Creating a Basic Part
 - Create a New Part document
 - Overview of the SolidWorks Window
 - Sketch a Circle
 - Add Dimensions
 - Changing the Dimension Values
 - Extrude the first Feature
 - View Display
 - Save the Part
 - Calculate the weight of the part
 - Extruded Cut Feature
 - Mirror entities
 - Create slots
 - Round the Corners of the Part
 - Rotate the View
 - Save the Part
 - Determine mass properties
- □ Active Learning Exercise, Part 2 Create a drawing
 - Create a New Drawing document
 - · Create Front, Top, Isometric and Section views
 - Change drawing scale
 - Position views
- Exercises and Projects
- Lesson Summary

In Class Discussion — The Design Process

When starting a new design, it is important to state the objectives and scope of the project. This is called product definition.

What is the final project to be and what elements make up the completed project? What tasks need to be accomplished to reach the stated goals?

For example, if you were designing a toaster you might want to know:

- How many slices must be able to be toasted at once?
- What is the maximum amount of power it can consume?
- How fast does it have to make toast? How do you measure this?
- How much can the toaster weigh?
- What is the maximum price the toaster can be sold for?
- How big can the toaster be?
- What manufacturing methods will be used.
- Will renderings or animations be required to support the marketing operation?

If the goals are clearly stated, it is much easier to know when the design is successful and how close you are to completion during the design process.

The design process is iterative in that you will rarely be able to go from idea to product in one straight line. Parts created or decisions made later in the design process may cause parts created earlier to be redesigned or modified.

Course Project Overview — The Mountainboard

Throughout the lessons of this course, we will be designing and analyzing a mountain skateboard. Individual parts will be created and then assembled into several sub-assemblies. Drawings will be created for several of the parts so that they can be manufactured.

Once we have the parts and assemblies created, they need to be analyzed to make sure they are strong enough to meet their intended use.

Using PhotoView 360 and MotionManager, we will make photorealistic images and animations of the project to show off our work and prepare it for marketing.

The Mountain Board

The finished mountain board is comprised of the deck, truck, axle assembly, wheels and the bindings.

The Bindings

There will be two bindings, one rightfooted and the other left-footed. The binding anchor will hold the binding to the deck and allow for adjustment across the deck as well as rotation. The binding is covered with a rubber pad which is glued to the surface.



The Deck

The deck is a laminated, symmetric piece with holes to mount the two trucks and two bindings. It must be flexible enough to turn the trucks.

It will support an average rider of 75 kilograms but should be able to support riders up to 100 kilograms.

The Truck and Axle

The truck and axle assembly connects the wheels to the deck. It must provide a dampened suspension system to cushion the ride without allowing oscillations that could make the ride unstable.

The suspension must be adjustable to be able to tailor the ride to the weight and skill of the rider as well as the terrain.

Mounting positions must be included for the optional brake system.





The Wheels

Each of the four wheel assemblies consists of a two-part plastic wheel with a tire and tube. Each wheel has two bearings.

Mounting positions must be included for the optional brake system.



The Mountainboard

The completed mountainboard.





In Class Discussion — The SolidWorks Model

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

The SolidWorks model is made up of:

- □ Parts
- □ Assemblies
- □ Drawings

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

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

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

Active Learning Exercise, Part 1 — Creating a Basic Part

The first part created will be the Binding Anchor shown at right. We will use SolidWorks to create this part.

Design Intent

Before starting on the actual steps to create the Binding Anchor, we need to determine the design intent. This is a list of requirements the finished part needs to meet. The design intent will tell us what the finished part must be able to do.



- □ The Binding Anchor will position the binding on the deck.
- □ The Binding Anchor must allow the binding to be positioned both along the centerline of the deck as well as adjusting the angle to the deck to allow the rider to set a comfortable stance.
- □ The Binding Anchor clamps the binding to the deck.
- □ There must be no sharp edges to injure a rider.

The Binding Anchor will look like the drawing below. Step-by-step instructions are given below.









Task 1— Create a New Part Document

- 1 Create a new part. Click
 - New 📄 on the Standard toolbar. The New SolidWorks Document dialog box appears.
- 2 Click the **Training Templates** tab.
- 3 Select the **Part_MM** icon.
- 4 Click OK.

A new part document window appears.

AltISO M	ny templates	LIN_for	
Part_MM P	with views gr	aphics Previev	ıi

Overview of the SolidWorks Window

When you create a new sketch:

- □ A sketch origin appears in the center of the graphics area.
- "Editing Sketch" appears in the status bar at the bottom of the screen.
- □ Sketch1 appears in the FeatureManager design tree.
- □ The status bar shows the position of the pointer, or sketch tool, in relation to the sketch origin.



First Feature

The first feature requires:

- □ Sketch plane Top
- □ Sketch profile 2D Circle
- □ Feature type Extruded boss feature

Sketching verses Drawing

The basis of most SolidWorks features is the sketch. Sketching is different from drawing in that drawings are created to the correct size as the lines and circles are drawn on the screen. With sketches, you only get the lines and circles close to their correct size. Dimensions and relationships will be added to make the sketch the correct size.

Open a Sketch

- 5 In the FeatureManager design tree, select (click once) the Top plane.
- 6 Select the **Sketch** tab on the Command Manager.

7 Open a 2D sketch. Click **Sketch [** on the Command Manager.

The sketch opens on the Top plane.

Background

To make the images in this course easier to read, we will use a white background instead of the default. To change the

background scene, click **Apply Scene** for the **Heads**up View toolbar and select the scene you want to use.

View Orientation

When we open a sketch for the first feature, SolidWorks will automatically change the view orientation to be normal to the sketch plane. This makes it easier to see the sketch. It is like looking straight down on a piece of paper.

Confirmation Corner

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

Sketch Indicator

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

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

Sketch Entities

SolidWorks provides a variety of tools to create sketches. They can be found on both the **Sketch Entity** menu and most can also be found on the **Sketch** toolbar.





Sketch Menu

The Sketch Tools menu is found by clicking **Tools**, **Sketch Entities**. All of the sketch tools are listed in the menu.

Sketch Toolbar

The Sketch Toolbar contains most of the sketch entities. It can be customized by adding or removing buttons.

Sketch
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Command Manager

Selecting the Sketch tab on the Command Manager will display the sketch tools.

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Exit Sketch	Smart Dimension	🗖 + 💮	• 🖉 • 🙈	Trim Entities	Convert Entities	Offset		Linear Sketch Pattern	*	Display/Delete Relations	Quick Snaps	Rapid
-	-	(÷ ÷)	• *	-	-	Enuues	50	Move Entities	*	-	~	Sketch
Features	s Sketch	Evaluate	DimXpert	Office Pro	ducts							

Task 2 — Create the first sketch

The first feature will be a short cylinder, 75mm in diameter and 3.5mm thick.

The Circle

The circle tool 🕝 creates 2D circles. Using the mouse, press the left mouse button at the location for the center of the circle, then (holding down the left mouse button) drag until the circle is approximately the correct size. Release the left mouse button.

R = 32.045

1 Click the **Circle** tool 3 from the Sketch toolbar. The curser will show that the circle

tool is active by displaying a circle under the drawing tool \bigwedge^{∞} .

- 2 Move the cursor over the origin until an orange circle appears. The little yellow icon below the drawing tool will show that we are going to make the center of the circle coincident with the sketch origin.
- **3** Press the left mouse button and drag the circle until it is just about 32mm. The cursor feedback will show the radius of the circle.



Dimension the sketch

To make the circle the correct size, we will add a diameter dimension to the sketch.

4 Click Smart Dimension 🐼 on the Sketch toolbar. The cursor will look like this,

 $k_{indicating}$ indicating that the dimension tool is active.

5 Click on the circle, then move the cursor to the right and up on the screen. The preview dimension and witness lines will be visible. Click to set the dimension location.



6 Input the dimension by typing 82 in the SpinBox. Click to accept the dimension.

Spin boxes are used to input numerical data. They are called spin boxes because the numbers can be spin up or down using the arrows on the

right. 責

Sketch Status

Sketches are normally fully defined before creating a feature with them. To be fully defined, the sketch geometry must be geometrically defined and positioned.

- □ To be geometrically defined, there must be enough dimensions and/or relationships to keep the size and shape of the sketch from changing if we try to drag it.
- To be positioned the sketch must also have dimensions or relationships that keep it from moving.

Sketch Color

The color of the sketch entities shows the status of the individual entity.

Blue - Under defined

Black - Fully defined

Red - Over defined

Extrude

Once the sketch is completed, it can be extruded to create the first feature. There are many options for extruding a sketch including the end conditions, draft and depth of extrusion, which will be discussed in more detail in later lessons. Extrusions take place in a direction normal (perpendicular) to the sketch plane.

Task 3 — Extrude the first feature

Extruding the 2D sketch will produce a 3D solid. In this case, we will make a short cylinder.

1 Select the Features tab on the Command Manager.

Click **Extrude Boss/Base C** on the Features toolbar. The model will reorient to the Isometric view and show a preview of the extrusion.

2 Preview graphics.

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

A handles appear that can be used to drag the preview to the desired depth. The current depth of the preview can be seen in the PropertyManager.


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) Boss-Extrude / 💥 ග්ර

Sketch Plane

3.500mm

Direction 2

Thin Feature

Draft outward

From

Direction 1

🍂 🛙 Blind

CD1

- **3** In the PropertyManager, change the settings as shown.
 - End Condition = **Blind**
 - \checkmark_{D1} (Depth) = **3.5mm**
- 4 Create the extrusion. Click **OK**. ✓ . The extrusion now becomes a solid and a new feature, Extrude1 is displayed in the FeatureManager design tree.

```
TIP:
```

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

A second method is the set of **OK/Cancel** buttons in the confirmation corner of the graphics area.

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



Customize Menu

Blind Extrusions

Blind extrusions take the 2D sketch and move it, some specific distance, normal (perpendicular) to the sketch plane.

FeatureManager design tree

This extrusion is the first feature of our part. The FeatureManager design tree shows this feature by type and with a default name Boss-Extrude1.

The sketch of the circle (Sketch1) is listed under the feature. It is said to be absorbed by the feature.



View Display

The View toolbar provides a quick method to change the way the model is displayed on the screen. It provides one set of tools to Zoom, Pan and Rotate the model view and another to change the way the model is displayed. In most cases, models are created in Shaded view because it most closely resembles the real world.



Save the Part

Save your work frequently. If you have a computer problem, you may loose everything you did since the last time you saved your work.

5 Click Save 🔚 on the Standard toolbar, or click File, Save.

The **Save As** dialog box appears.

6 Type Binding Anchor for the filename.

Stave As	
🚱 🔍 🛡 🕌 « Mountainboard 🕨 Binding	 ✓ ✓ ✓ ✓<!--</th-->
Organize 🔻 New folder	≣≕ ▼ 🔞
lessons	^ Name
🌗 Mountainboard	
Axle-Truck	No items match your search.
🍌 Binding	
📔 Hardware	
Parts	
Spring Assembly	
Wheel Assembly	
퉬 Training Templates	▼ ∢ III ►
File name: BindingAnchor	•
Save as type: Part (*.prt;*.sldprt)	▼
Description: Add a description	
Save as copy	References
Hide Folders	Save Cancel

- 7 Save the file to the folder Binding found under SolidWorks Curriculum and Courseware_2011\Mountainboard Design Project\Mountainboard.
- 8 Click Save.

The sldprt extension is automatically added to the filename.

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

Note: All the files we create of the Mountainboard project should be saved in the appropriate folder under the folder ...\SolidWorks Curriculum and Courseware_2011\Mountainboard Design Project.

Changing views

The Standard Views toolbar or the Heads-Up View toolbar make it easy to change your view of the model by simply clicking on the view you would like to see.

Heads-up View Toolbar



Standard Views Toolbar



access the different views. The tools available through Mouse Gestures can be customized to show either four or eight tools. The tools available will also depend on whether you are in a sketch, part, assembly or drawing. Shown are the default Mouse Gestures for a part.



9 Change the view of the model to the Bottom view. Click on either the Standard Views or Heads-up View toolbar.

Task 4 — Add a second feature

The second feature will be another cylinder, slightly smaller than the first.

- 1 Change the display mode back to shaded. Click **Shaded With Edges** on the **View** toolbar.
- 2 Select the bottom face of the cylinder. It will turn blue to show that it is selected.
- **3** Start a new sketch by clicking the **Sketch [** on the Sketch toolbar or pop up toolbar.
- 4 Select **Circle** \bigcirc on the Sketch toolbar.
- **5** Draw a circle, slightly smaller than the size of the cylinder. It does not have to be centered on the cylinder. The circle is blue, indicating that the sketch is Under Defined.



Task 5 — Adding sketch relationships

Sketch relationships are used to force a behavior on a sketch element to capture design intent. Some are automatic, others can be added as needed.

When adding relationships, only those relationships that are appropriate for the sketch entities selected will be shown in the PropertyManager.

1 Click Add Relation how on the Sketch toolbar. Add Relation will appear in the PropertyManager.





Callouts

Callouts provide a display of existing conditions. The relationship callouts show which relationships exist and between which sketch entities.

Task 6 — Dimension the circle

The concentric relationship defines the position of the circle, but it is still blue (under defined) because it doesn't have a dimension for its diameter.

- 1 Click **Smart Dimension** 🐼 on the Sketch toolbar.
- 2 Click on the circle, move the cursor to the right the click again to set the dimension position.
- 3 Type **75** for the value. Click ✓. The circle will now be black to show that it is fully defined.

Status Bar

The status bar, located at the bottom right of the graphics window will also show the state of the sketch. Fully Defined



Change the viewpoint

When creating the first feature, our viewpoint

was automatically changed to the Isometric view. After the first feature, we must change the view to best see the preview of the new feature.

Ø75

Task 7 — Extrude the second feature

- 1 Change the viewpoint to Dimetric by clicking 🚺 on the Views Orientation toolbar.
- 2 Click **Extrude** on the Features toolbar.
- **3** Select **Blind** for the type of extrusion.
- 4 Type **3mm** for the depth. Check the preview shown in yellow. It shows that material will be added to the bottom of the first cylinder.
- 5 Click 🖌

Task 8 — Cut a recess in the top of the part

Material needs to be removed from the top of this part to:

- □ Reduce weight. Each part must be designed to be as light as possible so that the assembled mountainboard is not too heavy to be carried.
- □ Lower the tops of the screws used to bolt this part to the deck. This will reduce the chance of anything (pants leg, shoe laces, etc.) getting caught on the screw heads.

Removing material by extruding a cut

Material can be added or removed by extrusion. The process to add or remove material works the same in that you start by creating a 2D sketch. That sketch is then moved normal to the sketch plane. If you are creating a boss, the enclosed volume is added to the part. If you are creating a cut, the enclosed volume is removed from the part.

- 1 Orient the part to the Top view by clicking \square on the Views Orientation toolbar.
- 2 Select the top face of the model, and click **Sketch** [2] to start a new sketch.
- **3** Click **Circle (O)** on the Sketch toolbar.
- 4 Draw a circle from the center of the top face. Make its radius about **30mm**.
- 5 Dimension the circle to be **63mm** in diameter.
- 6 Reorient the model to the Isometric view.
- 7 Click Extruded Cut 间 to use the circle to cut away some material.
- 8 Check the preview, by default, cuts go into the existing part.
- **9** Type **3mm** for the depth.





10 Click *✓*. The **Cut Extrude** command removed a cylinder shaped volume from the part.



Calculating the weight of the part

In any design, it is important to keep track of the weight of each individual part. In the case of the Mountainboard, if the individual parts become too heavy, the total weight of the board may exceed a reasonable weight to be carried up the hill.

The weight of the part can be calculated by multiplying the volume of the part by the density of the material.

 \Box Weight = Volume x Density

Calculate the volume

The total volume is the sum of the volumes created by the two extrudes minus the volume of the cut.

- Total Volume = Volume of each of the two extruded cylinders volume of the extruded cut
- □ Volume of a cylinder = Area of the circle times the cylinder height = Pi times the diameter squared divided by 4, times the cylinder height = $(Pi * D^2/4) h$
- □ Total Volume = $(3.14 * 82^2 / 4)(3.5) + (3.14 * 75^2 / 4)(3) (3.14 * 63^2 / 4)(3) = 18,483.56 + 13,253.60 9,351.74 =$ **22,385.42**cubic millimeters

Find the density

The density of engineering materials can be found in many ways. There are numerous engineering handbooks or several sites on the internet. One such site is MatWeb (www.matweb.com).

The Binding Anchor will be made from 2014 Aluminum. Using MatWeb, the density for 2014 Aluminum is 2.8 g/cc. There are 1000 cubic mm in 1 cc, so the density would be:

2.8 g/cc x .001 cc/mm = .0028 g/mm3

Calculate the weight

Weight = Volume x Density = 22,385.42 mm³ x .0028 g/mm³ = **62.68 grams (2.21 oz)**.

Task 9 — Create the screw slots

To make the position of the binding adjustable, the binding anchor will have four slots. These will allow the position of the bindings to be moved along the centerline of the deck.

The slots are symmetrical, so we will use a function called mirroring to make sure the slots always remain symmetrical if we later need to change their size.

Create a sketch

- 1 Select the face of the part created by the cut.
- 2 Open a sketch by clicking 🛃 on the Sketch toolbar.
- 3 Change to the Top view by clicking 🗊 on the View Orientation toolbar.

The Sketch Mirror tool

Mirror Entities and **Dynamic Mirror Entities** create symmetric relationships between sketch entities about a centerline. The **Dynamic Mirror Entities** command can be used *while sketching* and the **Mirror Entities** *after sketching* with the same final results.

Lines and Centerlines

The Line tool draws straight lines. If the line is vertical, the cursor will show a yellow callout \blacksquare to indicate that a **Vertical** relationship will be added. If the line is horizontal, the cursor will show an \blacksquare to indicate that a **Horizontal** relationship will be added.

Centerlines are construction geometry. They are used to position other entities but do not result in features.

While you are sketching, the callouts will be yellow | - |, indicating which relationship will be added when you release the mouse button. The green callouts show relationships that have been added | - |.

Note: The color of the callouts indicating existing relationships, green in this case, can also be cyan. Their color depends on the color scheme chosen in the SolidWorks Options.

Create a centerline

4 Click **Centerline** on the Sketch toolbar.





5 Sketch a vertical Centerline from the sketch origin. The length is not important. Make sure the cursor displays a 1, indicating a Vertical relationship will be added.



6 Click **Dynamic Mirror Entities** (2) on the **Sketch** toolbar. A pair of parallel marks will appear at each end of the centerline to show that we are in the mirror mode.

7 Sketch a vertical line to one side of the centerline.As soon as you finish drawing the line, a mirror image will be drawn automatically on the other side of the centerline.

The callouts will show that a symmetric relationship has been added between the endpoints of each line.



Arcs

There are three tools provided to create arcs:

Tangent Arc \bigcirc — Adds a tangent relationship to the entity it is sketched from.

Center Point Arc \bigcirc — Defined by a center point and a radius.

3 Point Arc \square — Defined by two endpoints and a radius.

The choice of arc tools depends on the geometry that needs to be created.

Add an Arc

The sketch of the slot is composed of two straight lines and two arcs. The arcs must be tangent to both lines.

- 8 Click Tangent Arc i the Sketch toolbar.
- **9** Place the cursor over the end of the right vertical line and drag an arc up and around to the right until you get cursor feedback showing you have gone 180 degrees.

There will be three indicators that you have gone 180 degrees:

- A blue dashed line (inference line) from the center of the arc
- The angle symbol $rac{3}{2}$ under the drawing tool
- The arc degree feedback (A=180)

When you release the mouse button, a second symmetric arc will be drawn automatically.

10 Draw another vertical line, from the end of the arc, vertically downward until you get an blue inference line from the bottom end of the first line.



<u>D</u>2

03

17.532, 90°

- 11 Finish the sketch with another **Tangent Arc**.
- 12 Turn off the sketch **Dynamic Mirror Entities** tool by clicking **a** on the Sketch toolbar.



Review the progress

With sketch mirroring turned on, each entity we drew had a mirrored entity drawn on the other side of the centerline. The symmetric relationships added my the mirror tool will make these sketch elements retain the symmetry we desire.

Callouts show that the arcs are tangent to the two lines it connects to \bigotimes and symmetric to the other arcs and centerlines \square . The numbers next to each symmetric relationship show the pairs of symmetric elements.



With all the symmetric relationships, the number of

callouts displayed may make the sketch elements hard to see, so we can turn them off.

Turn off the callouts

To toggle off the callout display of existing relationships, click **View, Sketch Relations** from the menu. This command is a toggle that turns the callouts on or off.

- 13 Click View, Sketch Relations.
- 14 Turn off the callouts by clicking View, Sketch Relations.

Mirror after sketching

Mirrored entities can also be created after creating sketch entities. To mirror after sketching, select the centerline about which you want to mirror and all the entities you want to mirror.

To mirror after sketching, click **Mirror Entities (1)**.

Task 10 — Mirror the two slot sketches

To complete the pattern of slots, mirror the two slots across a horizontal centerline.

- 1 Sketch a horizontal centerline from the origin to the right.
- 2 Click Mirror Entities 🔊 on the Sketch toolbar.
- **3** Turn off the **Centerline** tool **i** by clicking on the tool again.
- 4 Select the four lines and four arcs as the Entities to mirror. Select the horizontal centerline for the entity to Mirror about.

Make sure you *do not* have the vertical centerline selected.

Click 🖌.

- AMirror 🧹 🗙 🛁 Message ~ Select entities to mirror and a sketch line or linear model edge to mirror about Options Entities to mirror Д Line2 Line3 Line4 -Line5 Copy Mirror about:
- 5 We now have a sketch that will cut the four slots.



View Relationships

6 Click View, Sketch Relations.

The callouts show that the all the arcs are tangent to the two lines they connect to \bigotimes and symmetric to the other arcs and centerlines \square . The numbers next to each symmetric relationship show the pairs of symmetric elements.

With this many symmetric pairs, the number of callouts can make it difficult to see the sketch. Turn off the callouts by clicking View, Sketch Relations.

Test the relationships

All four slots sketches should have symmetric relationships. Anything done to one slot should be mirrored into the other sketches.

8 Drag the point shown. All four slots should change shape together.





Task 11 — Add dimensions

To fully define the sketch, we must add dimensions to position the slots and define their size. Even though we only drew one of the slots, the dimensions can be on any of the slots.

1 Add dimensions to the upper right slot as shown.

To add the **16mm** dimension, select the two arcs, not the vertical line.



2 Add dimensions as shown to the upper left slot to position it.

Both of these dimensions go from the lower arc to one of the centerlines.

3 Fully defined. The sketch geometry should now be all black, showing that the sketch is fully defined.



Task 12 — Create the cuts

The four slots must cut completely through the Binding Anchor. When we create the cut, it must

be done so that if we need to change the thickness of the material later in the design process, the slots do not have to be redone.

- 1 Click Insert, Cut, Extrude from the menu.
- 2 Click 🜍 on the **View** toolbar to change the view to Isometric.
- **3** From the list in the PropertyManager select **Through All**.

Through All

The end condition **Through All** will make the cut go through all the geometry. If, when we later analyze the part for strength we determine that it needs to thicker, we will not have to redo the slots because they will go through the entire part, no matter how thick it is.



4 Complete the cut. Click \checkmark .

Renaming Features

All the features shown in the FeatureManager design tree can be renamed. Renaming features can make them easier to locate as the parts become more complex.

There are three methods to rename features:

- Click-Pause-Click. Click on the feature name, Pause, Click the name again, type the new name
- Click on the feature name, press **F2**, type the new name
- Right-click the feature name and select **Properties**. Change the name in the Properties dialog box

Task 13 — Rename the slots

- 1 In the FeatureManager design tree, click once on Cut-Extrude2, this is the slots we just created.
- 2 Press F2. The feature name now has a box around it and a flashing cursor.
- **3** Type Rounded Slots for the new name.
- 4 The feature has now been renamed to something more descriptive.

Notice that the feature's icon does not change. The i shows that this feature is a **Cut-Extrude**.





Cut-Extrude2



Filleting

Filleting refers to both fillets and rounds. The distinction is made by the geometric conditions, not the command itself. Fillets are created on selected edges. Those edges can be selected in several ways.

Both fillets (adding volume), shown in red, and rounds (removing volume), shown in yellow, are created with this command. The orientation of the edge or face determines which is used.



Task 14 — Round an outside corner

All the existing edges in this model are sharp. To meet our design intent, all the exposed edges need to be rounded.

- 1 Click Fillet 🙆 on the Features toolbar.
- **2** Select the edge shown.



FilletXpert

省 Fillet 🖌 🗙

Manual Fillet Type

- **3** Type **1.5mm** for the fillet radius.
- **4** The following should be set by default. If not, change them as follows:
 - Manual
 - Fillet Type Constant radius
 - Tangent propagation Selected
 - Full preview Selected

5 Preview. Once Full preview is selected, the outline of the fillet will be shown in yellow. The fillet radius will be shown in a callout, Radius: 1.5mm attached to the edge. Click V.

Task 15 — Add a fillet to the inside edge

- 1 Click Fillet 🙆 on the Features toolbar.
- **2** Select the edge shown.

3 Type **1mm** for the fillet

4 Click **v**. The inside edge

now has a fillet.

radius.



Select this edge







Editing Features

After a feature is created, it can easily be changed. For the extrudes and cuts we made, each end condition or depth could be changed to reflect changes in our design intent, or changes required by later analysis.

For the fillet just created, we could add additional edges to the feature or change the fillet radius.

To edit a feature, right-click the feature either in the graphics area or FeatureManager design tree and select **Edit Feature**.

Task 16 — Edit the Fillet

1 In the Feature Manager design tree, right-click the feature Fillet2 and select **Edit Feature** from the Context toolbar.

SolidWorks can be customized to show the Context toolbar with a text menu below it or just a text menu.

The PropertyManager will now show the same information as when we first applied the fillet.



2 Select the edge shown.



Note: One of these edges will require a round while the other will require a fillet. Both are being done in the same command.

- 3 Once selected, the preview will show that the edge will be rounded as part of the Fillet2 feature. Click ∠.
- 4 Save the part.



How much does it weight now?

The same principle used earlier of adding and subtracting volumes still applies, however it is now more complicated.

The volume removed by the slots is not too difficult as the area of each slot can be thought of as a rectangle and circle



The fillets are more complicated. The two rounded corners are each part of a torus (donut).

The volumes of the rounds are parts of the torus. The section views at right show what the two would look like. The volumes are <u>not</u> one-quarter of the volume of the torus. The equations to determine their volumes are available in both engineering and mathematics handbooks.

The inside corner fillet is even more complicated but still solvable by looking up the equation.



Using SolidWorks To Get The Weight

Rather than manually solve for the volume of our part and lookup the density of the material, SolidWorks provides tools to solve for the volume and weight of the part.

Add Material

SolidWorks provides a library of materials that can be assigned to parts. Once a material is applied, it will be used by SolidWorks to calculate the weight of the part.

Adding material to a part also changes its visual properties (what it looks like) and graphic properties like the crosshatch used in drawings.

The material assigned to the part can also be used for stress analysis and for photorealistic rendering.

To assign a Material to a part:

- Click **E** on the **Standard** toolbar.
- Click Edit, Appearance, Material in the menu.
- Right-click Material in the FeatureManager design tree and select Edit Material.

Task 17 — Add material to the part

We are going to manufacture this part from Aluminum 2014.

- Click Edit, Appearance, Material from the menu. The Materials Editor will open.
- 2 Click the Plus sign next to Aluminum Alloys to expand the list.
- 3 Select 2014 Alloy.

Select Metric (MKS) for Units.

Examine the Physical Properties. Density is listed as $.0028 \text{ g/mm}^3$. This is the same value we determined earlier.

Note: Certain graphics cards support RealView advanced graphics visual properties. To find out if yours does, consult the **Help** documentation inside SolidWorks.

Aluminium Alloys		Properties	Appearance	CrossHatch	n Custom	Application Data	Favorites
1060 Alloy		Material	properties				
1060-H12		Materia	ls in the defa	ult library can	not be edit	ted. You must first	copy the material to
1060-H12 Rod (SS)		a custo	m library to e	dit it.			
1060-H14		Model 1	runa: Í	Linear Elastic	Instranic	_	
1060-H16	_	Moder	iype,	Linear clasuc	1500 Opic		
50-H18	=	Units:		Metric (MKS)			
1060-H18 Rod (SS)		Catego	rv: [Aluminium All	ovs		
1060-O (SS)		corogo	···· [/	575		
1100-H12 Rod (SS)		Name:		2014 Alloy			
100-H16 Rod (SS)							
100-H26 Rod (SS)							
)-O Rod (SS)		Descrip	tion:				
5 Alloy		Source	. 1				
Alloy		Joarco	'				
.0-T43 Insulated Mold Casting (SS)		Sustain	ability:	Defined			
1.0-T6 Insulated Mold Casting (SS)			l				
01.0-T7 Insulated Mold Casting (SS)		Property			Value	Units 🥖	
2014 Alloy		Elastic Mo	dulus in X		744388.3	kgf/cm/z	
2014-0		Poisson's	Ration in XY		0.33	N/A	
2014-T4		Shear Mo	dulus in XY		285518.8	kgf/cm^2	
2014-T6		Mass Der	Mass Density		0.0028	kg/cm^3	
)18 Alloy		Tensile S	Tensile Strength in X		1687.06	kgf/cm^2	
24 Alloy		Vield Stre	Compressive Strength in X		984 12	kgf/cm*2	
024 Alloy (SN)		Thermal Expansion Coefficient in X		2.3e-005	5 /ºC		
HO		Thermal Conductivity in X		0.382409	409 cal/(cm·sec·°C)		
		Specific Heat		229.446	cal/(kg·°C)		
24-15							

4 Click **Apply** and then **Close** to apply the material.

The material is now listed in the FeatureManager design tree.



Mass Properties

Physical properties of a part can easily be calculated using the **Mass Properties** tool. This tool will not only calculate the volume and weight of the part, but many other properties needed during the design and analysis of a part.

To calculate Mass properties:

- Click Tools, Mass Properties from the menu
- Or, click <u>iii</u> on the Tools toolbar

Task 18 — Determine The Weight

1 Click **Tools, Mass Properties** from the menu.

The Mass Properties box will appear.

The Volume is calculated to be **19,765.727mm³** and the Mass is **55.344 grams**.

When we calculated the weight earlier it was 62.68 grams. This was before we removed material with the four slots and two rounds.

2 The Center of Mass is the balance point of the part. If we could suspend the part at this point, it would not want to tip over.

The center of mass is displayed numerically in the box and graphically by a purple triad.



Print	Сору	Close	Options	Recalculate	
	,				
Output coo	rdinate system:	default		•	
		BindingAnchor.	SLDPRT		
	Selected items:				
Include hid	den bodies/comp	onents			
Show outp	ut coordinate sys	stem in corner of	window		
Assigned m	ass properties				
Mass propertie	s of BindingAnch	or in work (Part	Configuration -	Default)	
Output coordi	nate System: o	default			
Density = 0.00	3 grams per cub	oic millimeter			
Mass = 55.344	grams				
Volume = 1976	5.727 cubic milli	meters			
Surface area =	4 11801.277 squ	are millimeters			
Center of mass	s: (millimeters)				
Y = -0.18	3				
Z = 0.000					
Principal axes o	of inertia and prir	ncipal moments o	finertia: (gram	s * square millim	eters)
Taken at the α Iv = (1.0	enter of mass.	n	Pv	= 26521 926	
Iy = (0.0)	00, 0.000, -1.00	0)	Py	= 27145.211	
Iz = (0.0	00, 1.000, 0.000) Í	Pz	= 53319.807	
Moments of ine	ertia: (grams * s	quare millimeter	5)		
Taken at the c	enter of mass an	d aligned with th	e output coordir	nate system.	
Lxx = 265	21.926	Lxy = 0.000	Lx	z = 0.000	
Lyx = 0.0 Lzx = 0.00	00	Lyy = 53319.8 Lzy = 0.000	Ly Ly	z = 0.000 z = 27145.211	
Momente of inc	rtiau (grame * a	auara millimeter	.)		
Taken at the o	utput coordinate	system.	,		
Ixx = 265	23.780	Ixy = 0.000	Ix	z = 0.000	
Iyx = 0.00	00	Iyy = 53319.8	07 Iy	z = 0.000	
Izx = 0.00	00	Izy = 0.000	Iz	z = 27147.065	

Change Units.

The units for the Binding Anchor part are in millimeters, grams, seconds, so the mass properties displayed in millimeters and grams. If we need the mass properties in different units, such as inch, pound, second, the conversion is simple.

- 3 Click the **Options** button.
- 4 Select Use custom settings. Select:
 - Length Inches
 - Mass Pounds
 - Per unit volume inches^3

Mass/Section Property Options
Units
Scientific Notation
Use document settings
Output Use custom settings
Length: Decimal places:
Inches
Mass:
pounds -
Per unit volume:
inches^3
Material Properties
Density: 0.0028 g/mm^3
Accuracy level
Lower (faster) Higher (slower)
OK Cancel Help

5 Click OK.

Note: The units have only been changed in this output. The part still uses millimeters as the unit of length

- 6 Click **Close** to close the Mass Properties.
- 7 Save the part.

Output coordinate System: -- default --Density = 0.101 pounds per cubic inch

Mass = 0.122 pounds

Volume = 1.206 cubic inches

Surface area = 18.292 square inches

Center of mass: (inches) X = 0.000 Y = -0.007 Z = 0.000

Active Learning Exercise, Part 2 — Creating a Drawing



Drawings are one way to communicate a design to the shop that will manufacture the part.

Task 1— Create a New Drawing Document

When a part is open, we can create a drawing directly from it.

1 Create a new drawing. X New SolidWorks Document Click Make Drawing Templates Training Templates from Part/Assembly 💷 💷 📟 B B 💷 on the Standard A-Scale 1to 2 B-Scale 1to 4 Drawing Preview toolbar. The New SolidWorks **Document** dialog box appears. 2 Click the **Training** Templates tab. 3 Select the A-Scale1to2 icon. Tutorials ОК Cancel Novice Help 4 Click OK.

Overview of the SolidWorks Drawing Window

- □ A new drawing sheet appears in the graphics area.
- □ The toolbars used in the drawing process are displayed as new tabs on the Command Manager.
- □ "Editing Sheet1" appears in the status bar at the bottom of the screen followed by the drawing scale.
- □ Sheet1 appears in the DrawingManager.
- □ The View Palette opens in the task pane.



Create Three Standard Views

Most part drawings contain the three standard views: Front, Top, Right. All three views can be created with a single command.

In the United States, the standard three views follow the conventions for Third Angle projections. The views are created as you would see the model as viewed from the Front, Top, or Right.



In other parts of the world, the standard is First Angle projection. With First Angle projections, the view is projected on a plane behind the model.



Note: In the following steps and all drawings created in this course, the paper background used in the default drawing templates has been removed. This is done to make the drawings easier to view and print.

- 5 Insert a model view. Drag the Front view from the View Palette onto the drawing sheet and drop it approximately in the position shown.
- 6 Once the first view is dropped, we can add any projected view by moving the cursor in the directions we want to project and dropping the new view.
- 7 Move the cursor

vertically from the Front view and the Top view will appear. Drop the Top view in the approximate position shown.



- 8 Move the cursor diagonally away from the Front view, up and to the right to create an Isometric view.
- 9 Click **OK** < to finish adding views.

Note: The geometry of this part would make a Right view redundant, so we did not add one.



View Properties

The properties for each drawing view can be set differently. When we first create a drawing view, it will have the properties set by the drawing template. Once a view is created, we can select the view and change individual properties for just this view.

10 Select the Isometric view. In the PropertyManager set the **Display** Style to Shaded With Edges.



11 Select the Front view. In the PropertyManager set the **Display** Style to Hidden Lines Visible.

Sheet Properties

Sheet properties control the setting for the drawing sheet. They can be used to change the size and scale of the sheet as well as setting the type of projection to be used.

The default scale of the drawing template was 2 to 1. This makes the drawing views too small for the drawing sheet.

Task 2 — Adjust the sheet scale

- 1 In the DrawingManager, right click Sheet1 and select **Properties** from the list.
- **2** Change the Scale to 1 to 1

4 Examine the drawing, the

3 Click OK.

In the DrawingManager, right-	Sheet Properties				X			
click Sheet1 and select	Sheet1		ype of projection	Nauto involuto de				
Properties from the list.	Scale: 1	. 1	 First angle Third angle 	Next view label:	A			
Change the Scale to 1 to 1.	Sheet Format/Size	. 4	• mire angle		<u>^</u>			
Click OK .	Sheer to madyate Standard sheet size Volume show standard format A1 (ISO) A2 (ISO) A3 (ISO) A4 (ISO)							
	C: \Program Files \SolidWorks \k Browse Display sheet format Custom sheet size Width: Height:							
	Use custom property values from model shown in:							
	Default		•	ОК	Cancel			
Examine the drawing, the view now fit the sheet better, but not in the correct position.	t they are			3				
				WESTIGNED West State TINUCCE 680m TEVE COLD TEVE TEVE State FCA/FE TEVE State COLARTE COLARTE OD COLARTE COLARTE	SolidWorks Binding Anchor			

Task 3 — Adjust the views

After adjusting the scale of the views, the individual views may no longer be in the correct location on the sheet. We can easily adjust their positions.

Moving Views

Views can be moved by simply dragging their borders.

When moving views, alignment between the Front and Top views will be maintained automatically.

1 Move the Front view. Move the cursor over the Front view. When it changes

to $\mathcal{R}_{\mathfrak{n}}$, press the left mouse button and

drag the view. As you drag the Front view, the Top view will also move to stay aligned about the Front view.

- 2 Move the Top view. When you drag the Top view, it is only permitted to move vertically as it must maintain its alignment to the Front view.
- 3 Click **Save**. The default name of the drawing will be Binding



Anchor.slddrw, the same name as the part, but with the extension for a drawing.

Section Views

Section views are used to show the detail at some point inside the model. The model is sectioned using a cutting plane and the unused section is removed. The exposed surface is then cross-hatched with a pattern that designates the material.



4 Create a section view from the Top view. Click **Section View** (1) on the Drawing toolbar or the **View Layout** tab of the Command Manager.

The **Line** tool will become active.

5 Draw a vertical line through the Top view at its center.

6 Move the cursor to the right of the view. The section view will move with the cursor. Click the sheet where you want to drop the section view.

7 The section line will be annotated with the first available letter, in this case "A" and the section view will be annotated "Section A-A" to relate it to the section line.







SECTION A-A

8 Adjust the views on the page by dragging them into the positions shown.



9 Save and Close the drawing.

5 Minute Assessment — #2

- 1 How do you start a SolidWorks session?
- 2 Why do you create and use Document Templates?
- **3** How do you start a new Part Document?
- 4 What features did you use to create the Binding Anchor?
- **5** True or False. SolidWorks is used by designers and engineers.
- 6 A SolidWorks 3D model consists of ______
- 7 How do you open a sketch?
- **8** What does the Fillet feature do?
- **9** What tool calculates the volume of a part?
- **10** What does the Cut-Extrude feature do?
- 11 How do you change an existing feature?

Exercises and Projects

The following exercises provide additional practice in sketching and creating simple extrudes and cuts.

Exercise 1: Sketching Lines

Create this part using the information and dimensions provided. Sketch and extrude profiles to create the part.

This lab reinforces the following skills:

- □ Sketching.
- Dimensions.
- □ Extruding a feature.
- 1 New part.

Open a new part using the Part_IN template.

2 Sketch.

Create this sketch on the Front Plane using lines, automatic relations and dimensions.

Fully define the sketch.





3 Extrude.

Extrude the sketch **1**" in depth.

4 Save and Close the part.

Exercise 2: Sketching Lines with Inferences

Create this part using the information and dimensions provided. Sketch and extrude profiles to create the part.

This lab reinforces the following skills:

- □ Sketching.
- Dimensions.
- □ Extruding a feature.
- 1 New part.

Open a new part using the Part_IN template.

2 Automatic relations.

Create this sketch on the Front Plane using lines and automatic relations. Show the **Perpendicular** and **Vertical** relations.

3 Dimensions.

Add dimensions to fully define the sketch.





- 4 Extrude. **Extrude** the sketch **0.5**".
- **5** Save and Close the part.

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Exercise 3: Sketching Horizontal and Vertical Lines

Create this part using the information and dimensions provided. Sketch and extrude profiles to create the part.

This lab reinforces the following skills:

- □ Sketching.
- Dimensions.
- □ Extruding a feature.
- 1 New part.
- 2 **Open** a new part using the Part_MM template.
- **3** Sketch and extrude.

Create this sketch on the Front Plane using lines, automatic relations and dimensions. Extrude the sketch **20mm** in depth.

4 Save and Close the part.

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Exercise 4: Sketch Practice

Create the part shown. Start on the Front plane.





Exercise 5: Multiple Bosses

Create the part shown. Start on the Top plane. Use **Mirror Entities** to create the second cylindrical boss.

The base and cylinders are extruded to a depth of **0.5 inches**. Corner fillet radius is **0.25 inches**.





Exercise 6: Angles

Create this part. Start on the Front plane. Extrude it to **0.25 inches** thick.





Exercise 7: Bracket

Create this part using the information and dimensions provided. Sketch and extrude profiles to create the part.

This lab reinforces the following skills:

- □ Sketching
- □ Bosses
- □ Holes

Design Intent

The design intent for this part is as follows:

- □ The boss is centered on the rounded end of the base.
- □ The hole is a through hole and is concentric to the boss.

Use the Part_MM template.

Dimensioned View

Use the following graphics and the design intent to create the part.





As an aid to constructing this part, visualize how it could be broken down into individual features:



Exercise 8: Basic Drawing

Create an A-size drawing of the Bracket part created in the previous exercise. Include a Front, Top, Right and Isometric views, Third Angle projection.



Lesson 2 Vocabulary Worksheet

N	ame: Class: Date:
F	ill in the blanks with the words that are defined by the clues.
1	The corner or point where edges meet:
2	The intersection of the three default reference planes:
3	A feature used to round off sharp corners:
4	The three types of documents that make up a SolidWorks model:
5	Controls the units, grid, text, and other settings of the document:
6	Forms the basis of all extruded features:
7	Two lines that are at right angles (90°) to each other are:
8	The first feature in a part is called the feature.
9	The outside surface or skin of a part:
10	A mechanical design automation software application:
11	The boundary of a face:
12	Two straight lines that are always the same distance apart are:
13	Two circles or arcs that share the same center are:
14	The shapes and operations that are the building blocks of a part:
15	A feature that adds material to a part:
16	A feature that removes material from a part:
17	An implied centerline that runs through the center of every cylindrical feature:

Lesson 2 Quiz

N	Name:Class:		_ Date:
D pr	Directions: Answer each question by writing the corr provided.	ect answer	or answers in the space
1	1 You build parts from features. What are features?		
2	2 Name the features that are used to create the Bind	ing Ancho	or in Lesson 2.
3	3 How do you begin a new part document?		
4	4 Give two examples of shape features that require a	sketched j	profile.
5	5 Give an example of an operation features that requ	ires a selec	cted edge or face.
6	6 Name the three documents that make up a SolidW	orks model	
7	7 What is the default sketch plane?		
8	8 What is a plane?		
9	9 How do you create an extruded boss feature?		
10	10 Why do you create and use document templates?		
11	11 What is a section view?		

Lesson Summary

- □ SolidWorks is design automation software.
- □ The SolidWorks model is made up of:
 - Parts
 - Assemblies
 - Drawings
- □ Features are the building blocks of a part.
- □ The weight of a part is its volume times the material density.
- Drawings are used to communicate the design to the shop.
- □ The views most commonly used to describe a part are:
 - Top View
 - Front View
 - Right View
 - Isometric View

Lesson 3: Basic Parts — The Binding

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



Before Beginning This Lesson

□ Complete the previous lesson: Basic Functionality.

Resources for This Lesson

This lesson plan corresponds to the following lessons in the SolidWorks Online Tutorial:

- □ Lesson 1 Parts
- □ Sheet Metal
- □ Assembly Mates

For more information about the Online Tutorials, See "Online Tutorials" on page 1.

Review of Lesson 2: Basic Functionality

Questions for Discussion

- 1 A SolidWorks 3D model consists of three documents. Name the three documents.
- 2 Parts are built from features. What are features?
- **3** Name the features that are used to create the Binding Anchor in Lesson 2.
- 4 What is the base feature of the Binding Anchor?
- **5** Why did you use the Fillet feature?
- **6** How did you create the Base feature?

Outline of Lesson 3

- □ Active Learning Exercise, Part 1— Creating a part
- □ Active Learning Exercise, Part 2 Create an assembly
- □ Exercises and Projects
- □ Lesson Summary

Active Learning Exercise, Part 1 — Create a Part

Follow the instructions in this lesson to create the Binding Base Plate. The Binding Anchor created in the last lesson will fit into the large hole in the center to hold the Binding Base Plate to the deck of the Mountainboard.





Design Intent

- □ There will be two versions of this part, one for the left foot and mirror part for the right foot.
- □ The part will be held in place by the Binding Anchor created in the last lesson.
- □ The front and back of the part will curve upward.
- □ Side tabs will help hold the foot centered on the binding.
- □ The side tabs will have slots to attach the flexible straps that go over the foot.

The Modeling Approach

The Binding Anchor, created in the previous lesson, was created by adding material on top of material to get the basic geometry. For the Binding Base Plate, the approach will be to create an oversized block of material and use a "cookie cutter" to cut away the material around the final part.

Task 1— Create the First Feature

- 1 Create a new part using the template Part-MM.slddot.
- **2** Create a sketch on the Top plane.
- 3 Click View, Sketch Relations to make the callouts visible.
- 4 From the origin, create a vertical and horizontal line as shown. The vertical line should be about 200 mm and the horizontal line 75 mm.

- **5** Create another horizontal line from the end of the vertical line. Make this line about 35mm.
- 6 Create a line from the end of the first horizontal line. Make sure that this line is NOT vertical.



Tangent Arc

Tangent Arcs are used to create an arc that begins tangent to a selected endpoint on the sketch.

- 7 Click **Tangent Arc**) on the **Sketch** toolbar.
- 8 Draw an arc from the endpoint of the angled line to the end of the top horizontal line.



started.

10 Add a tangent relationship. Notice that there is no tangent relationship between the arc and the top horizontal line. The automatic tangent relationship is only added to the starting end of the arc. To test this, drag the left end of the upper horizontal line to the left.

If a tangent relationship is required at the finish end, it must be added manually.



Adding Sketch Relationships

The Add Relation tool **L** allows us to add relationships to

geometry after it has been created. Only relationships that are appropriate for the selected geometry will be shown.

- 11 Click \square on the **Sketch** toolbar.
- **12** Select the arc and the upper horizontal line.
- **13** The only two choices available are **Tangent** and **Fix** as these are the only two relationships that can be established between an arc and a line.



. -

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87

225

R75

- 14 Click **Tangent .** The arc and upper horizontal line are now tangent. Check the relationships by dragging the same point as in the earlier step. The tangency will be maintained.
- **15** The callouts will show that the arc is now tangent at both ends.

- 16 Fully define the sketch by adding the dimensions shown.
- 17 All the sketch geometry should now be black.
- 18 Extrude the feature. Click **a** on the **Features** toolbar.



🔽 Boss-Extrude1	?
🖌 🗙 60°	
From	~
Sketch Plane	•
Direction 1	~
Mind Blind	•
1	
25.000mm	Å.
N	ţ.
Draft outward	



Flip Side To Cut

Normally, we use the **Extruded Cut** to remove the material inside a sketch. It can also be used to remove all the material outside a sketch. This is like using a cookie cutter where we are interested in keeping the shape inside the cutter. Selecting or clearing **Flip Side To Cut** determines if the material inside or outside of the sketch is removed.



Task 2 — Cookie Cutter Cuts

To get the shape we are interested in, we will use an **Extruded Cut** that will remove the unwanted material from our base feature.

Changing View Orientation

Is is frequently easier to sketch when looking directly at the sketch plane. To change the view orientation to look directly at the sketch plane, select the sketch plane then click

Normal to 🔳 on the Standard Views toolbar.

Create a Sketch

- 1 Select the face shown and open a new sketch.
- 2 Change the view so that you are looking normal to the sketch plane. Click on the Standard Views toolbar.
- 3 Select the Line \bigwedge tool.
- 4 Position the cursor over the lower edge of the part.



When you are over the edge, the cursor feedback will be \searrow_{\varkappa} . This is the feedback for coincident, meaning you are on the edge. Sketch a line to the right.



5 Add a Tangent Arc. Draw a tangent arc from the right endpoint of the line to the midpoint of the right vertical edge.

A = 24.17° R = 142.544

When the cursor is over the right vertical edge, the midpoint will be displayed with a square with two

diagonal lines \uparrow . Once the arc is drawn there will be a midpoint relationship \blacksquare established with the right vertical line.

- 6 Add a **Tangent Arc** from the left end of the line to the midpoint of the left vertical edge.
- 7 Dimension the sketch as shown.



Offset Sketch Entities

The **Offset Entities** tool is used to make a copy of sketch entities, or edges, offset from the original by some specified distance.

8 Click any of the three sketch entities.

9 Click Offset Entities on the Sketch toolbar. Select Chain will be selected by default, this will select all sketch entities that are continuously connected to the one we selected.



10 Type **3mm** for the offset. Select **Reverse**, if necessary, to make the preview appear above the other sketch entities.



Click **OK**. Each arc and the line are duplicated at an offset distance of 3 mm.

Close the Sketch

The sketch must be closed to extrude the cut. Add two lines to connect the ends of the arcs. These lines need to be perpendicular to the arcs. This means that the lines must point through the centers of their respective arcs.

11 Draw a line to connect the ends of the two arcs on the left. The line should become fully defined (black).



12 If the line does not become fully defined, add a relationship to make the line perpendicular to the arc. Because we cannot add a perpendicular relationship between the arc itself and the line we will add a coincident relationship between the line and the centerpoint of the are. By basic geometry this will make the line perpendicular to the arc.



- **13** Click **Add Relation L** on the Sketch toolbar.
- 14 Whenever the PropertyManager or a dialog box has an entry box colored in the light blue color shown at right, anything selected in the graphics area will be entered in the box.
- **15** Select the arc centerpoint and the line.
- 16 Click k to add a **Coincident** relationship. The line will now be black as it is fully defined.
- 17 Repeat the procedure to close the sketch between the arcs on the right.

Cut to the Outside

The **Extruded Cut** command can either cut what is enclosed by the sketch or everything outside the sketch.

- **18** Click **Extruded Cut [i]** on the Features toolbar.
- **19** Select **Through All** for the end condition.
- **20** By default, the cut will remove the material inside the sketch.



21 Select Flip side to cut. Now the cut will keep the material that is inside the sketch and remove the material that is outside the sketch.



22 Click **OK**.

The material remaining is our curved base plate.

23 Save the part to the folder Mountainboard Design Project\Mountainboard\ Binding folder.

Task 3 — Creating the Side Tabs

The two side tabs provide the mounting locations for the binding straps that go over the riders foot. They must be offset from the base plate to allow for the thickness of the strap. They must also be a uniform thickness as the final product will be cut from flat material and bent into the final shape.



Create First Offset

- 1 Select the face shown and open a sketch.
- Change the view orientation to Normal To by clicking on the Standard Views toolbar.
- 3 Select the Rectangle tool from the Sketch toolbar.

Select this face

Select **Corner Rectangle** Corner Rectangle for the **Rectangle Type** in the PropertyManager.

The **Rectangle** tool draws a rectangle with two lines horizontal and two vertical.

4 Start the rectangle with a coincident relationship to the bottom edge of the Binding Base Plate. Drag the rectangle until you get a coincident relationship with the top edge.



5 Add dimensions.

The two coincident relationships control the height and vertical position of the rectangle. To make it fully defined, only a width dimension and a single positioning dimension are need.

Add the two dimensions shown.



6 Extrude the sketch to a blind depth of 4mm.

Create the tab



Select

- 9 Add a relationship. Click Add Relation how on the Sketch toolbar. Select the right vertical line of the sketch and the right vertical edge of the offset.
- 10 Click **//** Collinear and click OK.
- 11 The rectangle will now stay the same width as the offset.
- 12 Dimension the height of the rectangle to **32mm**.
- 13 Extrude the sketch, **Blind** to a depth of **3mm**.

Task 4 — Binding Attachments

The binding straps will attach to the base plate through two curved slots. The slots need to be curved to allow the binding straps to rotate as the foot is pushed into the binding.

- 1 Select the outside face of the tab and open a sketch.
- **2** With the outside face of the tab still

selected, click 🔳.

Zoom to Selection

To get a closer look at a selected item,

click **Zoom to Selection** (a) on the

View toolbar. This will make the selected entity fill the graphics area.

3 Zoom in on the selection by clicking **Zoom to Selection** [**3**].

Sketching the Slots

The two slots are symmetrical. To create them, sketch one and mirror it get the second.

- 4 Create a centerline. Start the centerline at the midpoint of the top edge and make sure it is vertical.
- 5 To mirror as we sketch, click **Dynamic Mirror**

Entities (b) on the Sketch toolbar.

Centerpoint Arc

Drawing the Centerpoint Arc is a two step process. You first drag from the center of the arc to the start point of the arc. Release the mouse button, then press and drag the length of the arc.

6 Select the 😥 Centerpoint Arc from the Sketch toolbar.





- **7** Start the arc from the centerline and drag the radius/ start point as shown.
- 8 Release the mouse button. This will be the point where the arc begins.
- **9** Press the mouse button and drag until the arc is as shown, then release the mouse button.

- **10** Draw another centerpoint arc using the same centerpoint. This makes the arcs concentric.
- 11 Draw **Tangent Arcs** to close up the sketch. Add Tangent relationships to make sure all the arcs are tangent to the arcs they are connected to.

TIP: Toggle on **Sketch Relations** to make it easier to check the relationships.

12 Turn off **Dynamic Mirror Entities**.





A = 26.58°

Dimension the Sketch

13 Add the Dimensions shown.



Dimension Alignment

To create the 6mm dimension between the arcs, pick the arcs on each end of the slot. As you drag the cursor, the dimension preview will show:



To lock in the dimension alignment you want, click the right mouse button to set it. The cursor \overline{k} shows that clicking the right mouse button will lock the alignment. Once the dimension alignment is locked, the cursor changes to \overline{k} , to show that you can now unlock the dimension alignment by clicking the right mouse button. Once locked, the alignment of the dimension will stay active no matter where you then move the cursor.

14 Extrude a cut. Select **Up To Next**.

The end condition **Up To Next** will extrude the cut until it reaches the next surface that intercepts the entire profile. In this case it is the inside face of the tab. In some cases, such as this, more than one end condition will produce the same results but for different reasons. We could have used **Through All** and still had the same results.

Task 5 — Create the second tab



Bends

This part will be manufactured from a flat piece of metal. It will first be cut to shape, then the tabs will be bent, followed by the front a back curves. Because the part will be manufactured from a single piece of metal, the model must have a uniform thickness.

We could add the bends manually using fillets.

If we add a **4mm** fillet to the lower edge of the tab.



The profile of the tab shows that the thickness is no longer uniform. We must add a fillet to the inside edge of the tab at the correct radius to keep the material uniform.

To calculate the fillet radius:

The radius of the inside fillet = radius outside fillet - material thickness.

- □ Radius inside = Radius outside Thickness
- \Box Radius _{inside} = 4 3 = 1mm

If we applied a 1mm fillet to the inside edge.

The finished bend would now a uniform thickness.





Sheet Metal

SolidWorks sheet metal functions can create bends from existing square corners and calculate the correct amount of material needed to cut the flat blank.

Parts can be flattened to show the correct flat pattern.

Note: Parts must be a uniform thickness in order to insert bends. This is consistent with the process in the shop as the part will be made from a single piece of material that is a consistent thickness.

Task 6 — Add bends

As stated earlier, this part will be created from a single piece of flat metal. Using the Sheet Metal tools, we will add bends to this part so that it can be flattened to determine the size of the blank that will need to be created.

- 1 Select the top flat face of the Binding Base Plate. This will be the face that will be fixed. All the bends will move relative to this face.
- 2 Click Insert, Sheet Metal, Bends from the menu.



3 Set the bend radius. Change the Bend Radius to **3mm**, which is the material thickness.

Leave the remaining option as shown.

4 Click 🖌.

- 5 Sheet Metal Features. Four new features are added to the FeatureManager design tree:
- □ Sheet-Metall contains the sheet metal definitions such as the bend radius we entered in the last step.
- □ Flatten-Bends1 creates a flat pattern or the part.
- □ Process-Bends1 contains the information to bend the flat pattern into the final part.
- □ Flat-Pattern1 also creates a flat pattern of the part. Note that it is gray in color indicating that it is suppressed. This means that the feature is not in use.

Rollback

The model can be rolled back to a previous state by moving the rollback bar at the end of the FeatureManager design tree.

To rollback the FeatureManager design tree, move the cursor over the rollback bar (the line that is normally at the bottom of the FeatureManager design tree). The cursor will

change to a hand and, then drag the rollback bar to the desired position.

6 In the FeatureManager design tree, move the cursor over the rollback bar and drag the rollback bar to a position between Flatten-Bends1 and Process-Bends1.

The part will flatten.

Working With The Flattened Part

Features can be added to the flattened part just as they can with the bent up part.

Features added in the flattened state usually equate to machining that would be done before the part is bent in the shop.





👸 Sheet-Metal1

Kat-Pattern1



Task 7 — Add Fillets

The sharp corners need to be rounded both to make the part look better and for safety reasons.

1 Add **15mm** fillets to the corners shown.

2 Add **6mm** fillets to the corners of the tabs as shown.

3 Add **2mm** fillets to the corners shown.

4 Bend the part by moving the rollback bar to the end of the FeatureManager design tree. The part will bend to its final shape.



Task 8 — Center Hole

To hold the Binding Base Plate to the deck, it will have a hole into which fits the Binding Anchor created in the last lesson.

This hole would be created before the part was bent as it is much easier to clamp the part for the drilling operation when the part is flat.

1 Rollback the part to before Process-Bends1.

Note: Rollback can also be done through the right mouse button menus. Right-click on Process-Bends1 and select Rollback

2 Create a sketch on the top face of the part. Sketch a circle and dimension it as shown.

Note: We are cutting the hole just slightly larger than the size of the boss on the Binding Anchor part. We want the fit tight, but not so tight that it is hard to assemble.



3 Extrude a cut.

Now that the part has been turned into sheet metal by the **Insert Bends** command, a new end condition appears called **Link To Thickness**. This makes the cut depth the same as the material, even if the material thickness changes.

Note: Link To Thickness is only available in parts that have been turned into sheet metal parts by the process of **Insert Bends** or **Base Flange**.



4 Bend the part. Right-click the feature Process-Bends1 and select **Roll To End**.

This is just another way to move the rollback bar.



Task 9 — Attach Material

Because we are concerned with the overall weight of our product, material should be attached to each part as we build them, this will make it easier to check the weight of the entire product as we assemble it.

- 1 Attach the aluminum material 2014 Alloy to the part. Either click Edit Material **E** on the Standard toolbar or Edit, Appearance, Material from the menu.
- 2 In the **Materials Editor**, expand the Aluminum Alloys category by clicking the plus sign.
- 3 Select 2014 Alloy and click OK.
- 4 Save the part.

Active Learning Exercise, Part 2— Create an Assembly

Assemblies show the relationships between the various parts. We will create an assembly of the two parts of the Binding that we have made so far. Later, we will add additional components.

Task 1— Create an Assembly

- 1 To create a new assembly, click **Make Assembly from Part/Assembly** [SS] on the Standard toolbar.
- 2 Select the template Assembly_MM and click **OK**.

Because we selected to make an assembly, only assembly templates are shown.



3 **Insert Component** is automatically activated by SolidWorks when we open a new assembly. This is done just to save time. We only have one file open, so it is automatically selected and a preview of the part is attached to our cursor. The part will move with the cursor.

If the part was not open, we could select the **Browse** button to locate it.



Note: If you do not see the preview graphic when your cursor is in the graphics area, select **Graphics preview** in the PropertyManager.

Options	~
Start command when creating new assembly	
Graphics preview	

4 Place the part by moving the cursor to the assembly Origin. When the cursor is over the Origin, it will change to ¹/₂. Click on the Origin to place the part.

Position of the First Component

The initial component added to the assembly is by default, **Fixed**. Fixed components cannot be moved and are locked into place wherever they fall when you insert them into

the assembly. By using the & cursor during placement, the component's origin is at the assembly origin position. This also means that the reference planes of the component match the planes of the assembly, and the component is fully defined.

Degrees of Freedom

There are six degrees of freedom for any component that is added to the assembly before it is mated or fixed: translation along the X, Y, and Z axes and rotation around those same axes. How a component is able to move in the assembly is determined by its degrees of freedom. The **Fix** and **Insert Mate** options are used to remove degrees of freedom.



The Assembly Window

The Assembly window looks very much like the part window except that the menu items change to those functions appropriate to creating and using assemblies. There will be some different toolbars as well.



Task 2 — Inserting Parts Into an Assembly

Parts and assemblies can be added into an assembly in many ways:

- □ From the menu, click Insert, Component, Existing Part/Assembly.
- Drag a part or assembly from an open window into the assembly window.
- Drag a part or assembly from Windows Explorer into the assembly window.
- Drag a part of assembly from the Design Library or File Explorer in the Task Pane into the assembly window.
- Drag a part or assembly from a 3D Content Central or other web pages into the assembly window.

Assembly Toolbar

The Assembly toolbar contains commands specific to working in assemblies.



	Edit Component	Insert Components	Mate	Linear Compon	Smart Fasteners	Move Component	Show Hidden Components	Assembly Features	Reference Geometry	80 New Motion Study	Exploded View	Explode Line Sketch	
1	Assembly	Layout	Sketch	Evaluate	Office Pro	oducts							

- 1 **Open** the part Binding Anchor.
- 2 Tile the windows by clicking **Window**, **Tile Vertically** from the menu.
- 3 Drag the top level icon **%** from the FeatureManager design tree of the Binding Anchor into the graphics area of the assembly.

The Binding Anchor is now added to the assembly.

The Binding Base Plate is fixed, but the Binding Anchor still has all six degrees of freedom.

Maximize the assembly window by clicking **Maximize** on the Assembly window title bar.



Positioning Components

One or more selected components can be moved or rotated to reposition them for mating using the mouse, or the **Move Component** and **Rotate Component** commands. Also, moving under defined components simulates movement of a mechanism through dynamic assembly motion.

Move Component

Moves a component in one of several ways: along an entity such as an edge; along assembly X, Y, Z axes; by X, Y, Z distances; or to a specific coordinate.

Components can also be moved by dragging them with the left mouse button.

Rotate Component

Rotates the component in one of several ways: about its centerpoint; about an entity such as an edge or axis; or by some angular value about the assembly X, Y, Z axes.

Components can also be rotated by dragging them with the right mouse button.

Task 3 — Mate the Binding Anchor

The Binding Anchor holds the Binding Base Plate to the deck of the mountain board. It will require two mates to position it correctly, a **Concentric** mate to hold it in the center of the hole in the Binding Base Plate. The second mate will position the top face of the Binding Base Plate coincident to the underside of the lip of the Binding Anchor.

1 Move the Binding Anchor to a position near the Binding Base Plate. Select the Binding Anchor, then hold down the left mouse button and you will be able to drag the Binding Anchor to different parts of the screen.

Note: Notice that the Binding Anchor can pass through the Binding Base Plate. Even thought these are "solid" models, they are still just mathematical representations of volumes in space.

2 Most mates are between faces of parts. To make the selection of faces easier, we will turn on the Face Filter which will only allow us to select faces. Press F5 on the

keyboard to show the Selection Filter toolbar. Select the Filter Face tool 📑.

- **3** Click **Mate (s)** on the **Assembly** toolbar to add a mate.
- 4 Select the two faces shown.

When you pick the second face, the parts will move into alignment for a concentric mate and the **Mates** toolbar will appear.



Note: The Binding Anchor has been rotated in the graphic to make it easier to see the two faces. You will have to rotate the model to be able to select both faces.

Mate Pop-up

The **Mate Pop-up** toolbar is used to make selections easier by displaying the available mate types on the screen.



The mate types that are available vary by geometry selection and mirror those that appear in the PropertyManager. The dialog appears on the graphics but can be dragged anywhere.

Either the on-screen or PropertyManager dialog can be used.

- 5 Click $| \langle v \rangle |$ to apply the **Concentric** mate.
- 6 Try to move the Binding Anchor. Click 😥 on the **Assembly** toolbar and try to drag the Binding Anchor. It will only move through the hole in the Binding Base Plate and rotate about its axis as these are the only degrees of freedom that remain after applying the **Concentric** mate.
- 7 Click Insert, Mate from the menu.
- 8 Select the top flat face of the Binding Base Plate.



9 Rotate the model and pick the face shown on the Binding Anchor.

The Mate Pop-up will show that

Coincident *K* is selected.

Click	\checkmark	to apply the mate.
-------	--------------	--------------------

- **10** Try to move the Binding Anchor. It will now only rotate in the hole of the Binding Base Plate as it only has one degree of freedom.
- 11 Toggle off the **Face Filter**.



Task 4 — Save the Assembly

- 1 Click File, Save.
- 2 Name the assembly as Binding to the ...\Mountainboard\Binding folder. SolidWorks will add the extension SLDASM to indicate this is an assembly file.

Task 5 — Calculate the Weight of the Assembly

When each of the two parts were created, we added the material 2014 Aluminum Alloy. The weight of the two part assembly can be determined the same way the weight was calculated in a part, using the **Mass Properties** tool.

1 Click Tools, Mass Properties.

The weight of the assembly is **218.360 grams** (0.481 pounds).

- **Question:**What material would SolidWorks use if we forgot to apply a material to all the parts?
- Answer: Each part template has a default material. If we do not apply a different material, the weight will be calculated using the default material which has a density of **0.001 g/mm³**.
- 2 Open the Binding Base Plate part. In the FeatureManager design tree, rightclick the Binding Base Plate and select **Open Part**.

🔯 Mass Properties								
Print Capy	Close Options Recalculate							
Output coordinate system:	default 👻							
Selected items:	Binding.SLDASM							
Include hidden bodies/comp	✓ Indude hidden bodies/components							
Show output coordinate sys	Show output coordinate system in corner of window							
Assigned mass properties								
Mass properties of Binding (Assembly Configuration - Default)								
Output coordinate System: default								
Mass = 218.360 grams								
Volume = 77985.657 cubic millimeters								
Surface area = 53756.358 squ	Surface area = 53756.358 square millimeters							
Center of mass: (millimeters) X = -50.232 Y = 3.990								
Z = -113.589	-							
•								

- 3 Check the material density. Click **Tools, Options** from the menu.
- 4 Select the **Document Properties** tab. This tab lists properties associated with just this part.
5 Select **Material Properties**. Because we applied a material thought the Material Editor, the individual options are grayed out. The Material Editor assigned values for Density and Hatch Pattern.

Document Properties - Material Properties				
System Options Document Pro	perties			
Drafting Standard Annotations Virtual Sharps Virtual Sharps Tables Detailing Grid/Snap Units Model Display Material Properties Image Quality Sheet Metal Plane Display DimXpert Size Dimension Chain Dimension Geometric Tolerance Chamfer Controls Display Options	2014 Alloy Density: 0.0028 g/mm^3 Area hatch / fill None Solid @ Hatch Pattern: ANSI38 (Aluminum) ¥ Scale: 1 Angle: 0deg			
			ОК	Cancel Help

- 6 Click **OK** to close the **Options**.
- **7** Save and Close all open files.

5 Minute Assessment – #3

- 1 What features did you use to create Binding Base Plate?
- **2** What does the Fillet feature do?
- **3** Name three view commands in SolidWorks.
- 4 Where are the display buttons located?
- **5** Name the three SolidWorks default planes.
- 6 The SolidWorks default planes correspond to what principle drawing views?
- 7 True or False. In a fully defined sketch, geometry is displayed in black.
- 8 True or False. It is possible to make a feature using an over defined sketch.
- 9 Name the primary drawing views used to display a model.

Exercise 9: Base Bracket

This exercise reinforces the following skills:

- □ Sketching lines.
- □ Adding geometric relations.
- □ Sketching on standard planes.
- □ Sketching on planar faces.
- □ Filleting.
- □ Creating cuts, holes and bosses.

Design Intent

- □ Some aspects of the design intent for this part are:
- □ Thickness of the Upper and Lower features are equal.
- □ The holes in the Lower feature are equal diameter.
- □ The Upper and Lower features are flush along the back and right side.



Upper. feature

Open a new part using the Part_MM template.

- 1 Create the **Lower** feature.
- 2 Use lines to sketch this profile. Add dimensions to fully define the sketch.



3 Select a face as sketch plane.

4 Create the **Upper** boss feature.

Select the rear face that is hidden by the top face of the model as the sketch plane. Use **Select Other** or rotate the view to select it.

Sketch the lines and relate them to the existing

edges where they should be coincident.

5 Extrude.

Extrude *into* the first feature a depth of **20mm**.





Exercise 10: Guide

This lab reinforces the following skills:

- □ Sketch lines, arcs, circles and fillets.
- □ Relations.
- □ Extrusions.
- □ Fillets and rounds.

Design Intent

Some aspects of the design intent for this part are:

- □ Part is not symmetrical.
- □ Large circle is tangent to outer edge.
- □ Large circle is coincident with underside brace edge.
- □ Plate thicknesses are equal.

Procedure

Open a new part using the Part_MM template.

1 Sketch the profile.

Using the Front plane, create the profile.







2 Extrusion.

Extrude the sketch **10mm**.



7 Cuts.

Use symmetry with lines and arcs to create a **Through All** cut for the slot shape. Use a circle to create another cut concentric with the model edge.

This sketch requires the use of a Parallel relation. Check	
the Help, SolidWorks Help Topics for more	
information.	



8 Save and close the part.

Lesson 3 Quiz

N	ame: Class: Date:				
D pi	Directions: Answer each question by writing the correct answer or answers in the space provided.				
1	How do you begin a new part document?				
2	How do you open a sketch?				
3	What is the Base feature?				
4	What color is the geometry of a fully defined sketch?				
5	How can you change a dimension value?				
6	What is the difference between an extruded boss feature and an extruded cut feature?				
1	How do you extrude a cut so that the material outside the sketch is removed?				
2	What is a fillet feature?				
3	How do you start a new Assembly document?				
4	What are components?				

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

Lesson Summary

- □ Base Feature is the first feature that is created the foundation of the part.
- □ The base feature must always add material.
- □ Extruded Cuts can remove either the material inside or outside the sketch.
- □ Insert Bends can be used to turn a part into sheet metal.
- □ Sheet metal parts must be uniform thickness.
- □ An assembly contains two or more parts.
- □ In an assembly, parts are referred to as *components*.
- □ Mates are relationships that align and fit components together in an assembly.
- □ The first component placed into an assembly is fixed.
- Mass Properties can be used to determine the weight and center of gravity for an assembly.

Lesson 4: Revolved Features — The Wheel Hub

□ Students will be able to create and modify the following parts and assembly:



Before Beginning This Lesson

□ Complete the previous lesson: Basic Parts - The Binding

Resources for This Lesson

This lesson plan corresponds to the following SolidWorks Online Tutorials:

- **D** Revolves and Sweeps
- Department Pattern Features
- □ Import/Export
- \Box Toobox

For more information about the Online Tutorials, See "Online Tutorials" on page 1.

Review of Lesson 3 — The Binding

Questions for Discussion

- 1 What are the two ways material can be removed with an Extruded Cut?
- 2 What is the primary requirement for a part that is to be turned into sheet metal with the command **Insert, Bends**?
- **3** What do mates do in an assembly?
- **4** When calculating Mass Properties of an assembly, how is the density of each part determined?

Outline of Lesson 4

- □ In Class Discussion Toolbox
- □ Active Learning Exercise, Part 1 The Wheel Hubs
 - Revolved features
 - Hole Wizard
 - Trim Entities
 - Convert Entities
 - Patterns
 - Reordering features
- □ Active Learning Exercise, Part 2 Importing Data
 - Importing files
 - Neutral file formats
- □ Active Learning Exercise, Part 3 Create the Wheel Assembly
 - Adding Toolbox parts
 - Section views
- □ Active Learning Exercise, Part 4 Create an Exploded View of the Wheel Assembly
 - Create Exploded views
 - Animate Exploded views
- D Exercises and Projects Additional Mountainboard Parts
- □ Exercises and Projects Revolved Features
- □ Lesson Summary

In Class Discussion — Toolbox

Toolbox includes a library of standard parts that are fully integrated with SolidWorks. These parts are ready-to-use components — such as bolts and screws.



To add these parts to an assembly, select the type of part you want to insert, then drag the Toolbox part into your assembly. As you drag Toolbox parts, they snap to the appropriate surfaces automatically establishing a mate relationship. In other words, a screw recognizes that it belongs in a hole and snaps to it by default.

As you are placing the Toolbox parts, you can edit the property definitions to correctly size the Toolbox part to your needs. Holes created with the hole wizard are easy to match with properly-sized hardware from Toolbox.

The Toolbox Browser library of ready-to-use parts saves you the time that you would usually spend creating and adapting these parts if you built them yourself. With Toolbox, you have a complete catalog of parts.

Toolbox supports international standards such as ANSI, AS, BSI, CISC, DIN, GB, IS, ISO, JIS, KS and MIL. In addition, Toolbox also includes standard parts libraries from leading manufacturers such as PEM[®], Torrington[®], Truarc[®], SKF[®], and Unistrut[®].

Making Sure That the Screws Fit

Before you placed the washers and screws, you should have measured the depth of the holes and the thickness of the washer as well as the diameter of the holes.

Even if you measured before placing the hardware, it is a good practice to verify that the screw fits as you intended it to. Viewing the assembly in wireframe, viewing it from different angles, using **Measure**, or creating a section view are some ways to do this.



A section view lets you look at the assembly as if you took a saw and cut it open.



Active Learning Exercise, Part 1 — The Wheel Hubs

Each wheel assembly is made up of six different parts:

- □ Tire
- □ Inner Tube
- □ Wheel Hub
- □ Bearing
- Bolt
- □ Nut

In this lesson, we will create the wheel hubs then import the tire and tube from another source. The bearings, bolts and nuts will be created using the SolidWorks Add-in, Toolbox.





Design Intent

The design intent for the hub is:

- □ 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.
- \Box The bolt holes must capture the nuts to keep them from rotating during assembly.

X

Revolved Features

Material can be added or removed from a model by using the **Revolve** command. To this point, we added or removed material by way of extrusions that moved the sketch normal to the sketch plane.

Revolves move the sketch around a centerline, producing cylindrical or conical results.

Task 1— Create a New Part

1 Create a new part. Click New SolidWorks Document

=	
New () on the Standard toolbar.	Templates Training Templates
The New SolidWorks	AltISO My template Part_IN Part_IN_for with views graphics
Document dialog box appears.	Part_MM Part_MM_for Assembly_IN Assembly_MM
2 Click the TrainingTemplates tab.	graphics
3 Select the Part_MM icon.	
4 Click OK .	Novice Cancel Help

5 Save the new part as Wheel Hub. Save the file to the Mountainboard\Wheel Assembly folder.

Revolved Features

Revolved featured are created by rotating a sketch around a centerline. The centerline becomes the axis of revolution.

There are three simple rules for revolved features:

- □ A centerline or sketch line must be specified as the axis of revolution.
- □ The sketch must not cross the axis.
- The axis of revolution for the revolve must be selected before creating the revolved feature.



Task 2 — Create the Hub Center

The hub center will be created as a simple revolved feature. While we could create this feature by extruding a circle, revolved features are generally more appropriate for parts that rotate, such as this hub and wheel.

- $\begin{array}{c|c} & & & \\ & & \\ & & \\ m, \\ o \text{ the} \\ e \text{ to} \end{array}$
- 1 Create a sketch on the Right plane.
- 2 Sketch a rectangle, approximately 25 mm by 25 mm, with the Origin at the lower right.
- **3** Sketch a centerline from the Origin, horizontally to the right. The length is not important.

We will revolve the rectangle around this centerline to form a cylinder.

- 4 Dimension the top line **21mm**. This is the thickness of the hub center.
- **5** The hub diameter. When the rectangle is revolved around the centerline, the vertical dimension of the rectangle will represent the radius of the cylinder. What is generally more important to the design is the diameter rather than the radius.

Add a dimension from the top horizontal line to the *Centerline*. When the cursor is on the side of the centerline closest to the selected line, you get a radius dimension. When the cursor is on the side of the centerline away from the selected line, you get a diameter.

Dimension the diameter to 40mm.



6 Revolve the hub. Click **Revolved Boss**/

Base \bowtie on the Features toolbar.

The preview shows that the rectangular sketch will be revolved around the centerline.

Select **Blind** and **360 deg** for the revolve type and angle.

Click OK.

7 Rename this feature Hub.

Task 3 — Cut the center holes.

Cuts can also be created as revolved features. This closely represents the machining operation of a lathe.

- 1 Create a sketch on the Right plane.
- 2 Reorient the view to the Right view.
- 3 Sketch the profile shown, including the centerline from the Origin to the right.

4 Dimension the sketch as shown. The larger diameter cut will be to house the wheel bearing and the smaller bore will be a clearance hole for the axle shaft.









5 Revolve a cut. Select Revolved

Cut **(solution**) on the Features toolbar or **Insert, Cut, Revolve** from the menu.

6 Choose **Blind** and **360deg**.

Click 🗹 .

- 7 Rename this feature Bearing Cut.
- 8 Save this part as Wheel Hub.

Patterns

Patterns are the best method when creating multiple instance of one or more features. The use of patterns is preferable to other methods for several reasons:

□ Reuse of geometry

The original or seed feature is created only once. Instances of the seed are created and placed, with references back to the seed.

□ Changes

Due to the seed/instance relationship, changes to the seed are automatically passed on to the instances.

Use of Assembly Component Patterns

Patterns created at the part level are reusable at the assembly level as Feature Driven Patterns. The pattern can be used to place component parts or sub-assemblies.

Smart Fasteners

One last advantage is that Smart Fasteners can be used to automatically add fasteners to the assembly. Smart Fasteners are only used to populate holes.



Task 4 — Create a spoke

To create the three spokes, only one will be modeled. The remaining spokes will be created as a circular pattern.

- 1 Create a sketch on the Front plane and change the view orientation to the Front View.
- 2 Sketch a vertical centerline from the Origin.
- 3 Click Dynamic Mirror Entities 🔊 on the Sketch toolbar.
- 4 Sketch a line from the Hub outward. A symmetric line will be drawn automatically.
- 5 Turn off Dynamic Mirror Entities 🙆.
- 6 Add a Coincident relationship. Click Add Relation ho on the Sketch toolbar to add a relationship. Select one of the

lines and the Origin. Click 📈 to add a **Coincident** relationship.

7 Add an angular dimension. Click Smart Dimension (2), then select the two lines.

Because the two lines are not parallel, the dimension will be an angular dimension. Place the dimension then type **40deg** in the spin box.

8 Draw a Centerpoint Arc to close the top of the sketch. SelectCenterpoint Arc S on the Sketch toolbar.





Start the arc at the origin and drag it to the top end of one of the lines. Release the mouse button, then press the left mouse button again and drag to the top of the other line.

9 Dimension the arc. Set the radius equal to **43mm.**



Convert Entities

Convert Entities enables you to copy model edges into your active sketch. These sketch elements are automatically fully defined and constrained with an **On Edge** relation.

Trim Entities

Sketch entities can be trimmed shorter using the **Trim Entities** tool. The **Trim Entities** tool can remove sketch entities by several different methods.

The most common method is to trim to the closest entity. This method removes the entity from the point where it is selected to the nearest intersection with another sketch entity.

Task 5 — Close the sketch geometry

The remaining sketch entity needs to be an arc that is the same as the outside edge of the Hub. This could be drawn as another **Centerpoint Arc**, however it is more efficient to create it from the existing edge of the model using **Convert Entities**. By using **Convert Entities**, we make sure that the inside surface of the spoke is always the same radius as that of the Hub.

1 Click **Convert Entities ()** on the **Sketch** toolbar. Select the outside edge of the hub.

Click 🗹 .

The entire circular edge has been converted into a circle in our sketch.

- 2 To trim the circle, select **Trim Entities *** from the Sketch toolbar.
- 3 Select **Trim to Closest** in the Property manager. The cursor will change to the Trim Cursor ℝ_X.
- 4 Click the part of the circle to be trimmed away.



Task 6 — Extrude the Spoke

- 1 Extrude the sketch. **Extrude** to a **Blind** depth of **15mm**.
- 2 Rename this feature Spoke.
- 3 Save the Wheel Hub.

Construction Geometry

Construction geometry is used to locate other sketch entities or features.

Any piece of sketch geometry can be converted into construction geometry or vice-versa. Construction geometry is considered to be reference geometry and does not have to be fully defined.

To convert sketch geometry into construction geometry:

 \Box Select the geometry, then click \blacksquare on the **Sketch** toolbar.

□ Select the geometry, then in the PropertyManager select **For construction**.

Task 7 — Create the Bolt Hole

Create a bolt circle. Bolt circles are construction geometry used to position bolt holes around the center axis.

- 1 Create a sketch on the face shown.
- 2 Reorient to the Front view.
- 3 Sketch a circle with its center at the **Origin**.
- 4 Dimension the circle to diameter **63.5mm**.

- **5** Select the circle, then in the PropertyManager select **For construction**.
- 6 The circle will turn into a construction circle.







- 7 Sketch a line from the Origin, vertically upward until it passes the circle. The intersection of the line and circle will be the location for the bolt hole.
- 8 Use the PropertyManager to change the line into a construction line.



9 Exit the sketch by clicking **OK** in the Confirmation Corner.

Task 8 — Add the Bolt Hole

The hole in the spoke needs to be multi-functional. Because two hubs will be positioned back to back, the hole must have a hexagonal cutout to capture the nut as well as be sized so that the bolt head can turn to be tightened. We will first create a clearance hole for the bolt shaft, then the hexagonal cut.

Hole Wizard

The **Hole Wizard** is used to create specialized holes in a solid. It can create simple, tapered, counterbored and countersunk holes using a step by step procedure.

1 Select the face of the spoke.

REPRODUCIBLE

- 2 Click Insert, Feature, Hole, Wizard from the menu. This starts the Hole Wizard.
- 3 Click the Hole button. Using the pull-down lists, select:
 - Standard: Ansi Metric
 - Screw type: Screw Clearances
 - Size: **M6**
 - End Condition: Through All

- 4 Click on the **Positions** tab. You can now edit the sketch that determines the hole position(s). The message tells you to locate the hole center(s).
- 5 Turn off the **Point** * tool by clicking it once on the Sketch toolbar. The sketch entity **Point** is automatically turned on so that you can place several holes. We are only placing one hole so we can turn off the tool.
- 6 Position the hole. Click **L** add a relationship. Select the Point, Construction Line, and Construction Circle. There will only be one

relationship available, Intersection. Click $|\chi|$ then \checkmark .







7 Complete the wizard. Click . The correct size clearance hole for an M6 metric bolt is created.



Task 9 — Add the Hex Cut

The hex cut will be made by creating a hexagonal sketch and extruding a cut.

- 1 Create a sketch on the face shown.
- 2 With the face still selected, click **Normal To Standard Views** toolbar to change the view so that we are looking normal to the face.

Zoom to Selection

Zoom to Selection will zoom the view to whatever entity is selected. The selected item will fill the screen but have a clear area around it.

3 Click **Zoom to Selection** (a) on the View toolbar. The selected face will now fill the screen.



Polygon Tool

Regular (all sides equal) polygons can be sketched with the **Polygon** tool. All regular polygons are based on a construction circle. The polygon is defined by the circle being inscribed or circumscribed about the polygon.

- □ Inscribed circle the circle is tangent to the midpoint of each line.
- □ Circumscribed circle the circle is coincident to the endpoints of each line.



- Select the Polygon tool (from the Sketch toolbar or select Tools, Sketch Entities, Polygon from the menu.
- 2 Sketch a Polygon by selecting the center point and dragging to some radius. The size and position are not important as we will set those in the following steps.



🕑 Polygon

- **3** The default for polygons is 6 sides. If your polygon has a different number of sides, it can easily be changed in the PropertyManager.
- 4 Add a **Concentric** relationship between the construction circle and the circular edge of the hole.

5 Dimension the construction circle of the polygon to 10.5mm.

Why dimension the construction circle? Part of the design intent is that this hex cut capture the nut, but also that the bolt head must be able to rotate inside the cut. The dimension represents the size of a clearance hole for the bolt head.

We could have also dimensioned the hex hole from flat to flat. While this is satisfactory for SolidWorks, it doesn't represent the design intent as clearly as dimensioning the circle diameter.

6 Add a **Horizontal** relationship to one of the lines of the hexagon. This is necessary to fully define the sketch. Without this relationship, the sketch is free to rotate about its center.



- 7 Create a **Cut-Extrude** to a depth of **6.5mm**.
- 8 Hide the bolt circle sketch. We no longer need to see the bolt circle and centerline now that the holes have been created. Right-click on the bolt circle and select Hide. Depending on your menu setup, Hide may either be listed in the menu or just in the context toolbar





Note: We could also right-click the sketch in the FeatureManager design tree. When a sketch is hidden, the sketch icon becomes hollow.

Sketch visible	🧹 Sketch4
Sketch hidden	Sketch4

9 Rename this feature Hex-Cut

Entering Dimensions

Whenever we add a dimension or depth, we normally enter the value in the units of our part or assembly. There are times when we do not know the value in the default units. Rather than use a calculator to convert units, we can just enter the value we know with the units included.

If we do not add units, SolidWorks will assume that the value is in the default units.

Task 10 — Add a Chamfer

- 1 Select the **Chamfer** tool **(2)** from the **Features** toolbar.
- 2 Select Angle distance.
- 3 Select one of the edges of the hex cut.
- 4 Set the **Chamfer** dimension. We want a 1/32 inch chamfer. Rather than divide 32 into 1 to get the decimal equivalent and then convert it to millimeters, just type **1/32in**. When you press the **Enter** key, the 1/32 inches will be calculated and displayed as millimeters.

Note: If you do not add the "in" to specify that the units are inches, SolidWorks will interpret the dimensions as 1/32 millimeters.



5 Click 🖌 .

We really wanted to chamfer all six edges but only selected one.



Edit Feature

Edit Feature provides a simple method to change the information used to create a feature. To edit any feature, right-click the feature either in the FeatureManager design tree or the graphics area, and select **Edit Feature**.

Task 11 — Edit the Chamfer

- Right-click the feature Chamfer1 in the FeatureManager design tree and select Edit
 Feature 8
 The Chamfer PropertyManager will open.
- 2 Select the remaining five edges of the Hex-Hole.



- Click . All six edges of the hex hole are now chamfered.
- 4 Rename this feature Hex-Cut Chamfer.
- 5 Save the Wheel Hub.

Task 12 — Revolve the Rim

The wheel rim will be constructed as another revolved feature.

- 1 Reorient to the Right view.
- 2 Create a sketch on the Right plane.
- **3** Draw a centerline from the Origin horizontally to the left. This will become the axis of rotation for this sketch.
- 4 Create the geometry shown in Sketch A.



5 Add the dimensions shown in Sketch B.Remember, to dimension the four diameters, the centerline must be selected as one end of the dimension



6 Add the following dimensions.

- 7 Add the following sketch relationships to fully define the sketch.
- 8 Revolve the rim. Click Revolved
 Boss/Base in on the Features toolbar.

Parallel

- 9 Revolve Blind, 360 degrees.
- **10** Rename this feature Rim.



Task 13 — Pattern the Spokes

The Wheel Hub will have three spokes. We have created one and will create the other two as a pattern of the first. By creating the spokes as a pattern, we can quickly change the number of spokes as well as their design.

Circular Patters

Circular Patterns create copies, or instances, in a circular pattern controlled by a center of rotation, an angle and the number of copies. The instances are dependent on the originals. Changes in the originals are passed on to the instanced features.

Axes

SolidWorks has two types of axes:

□ Temporary Axes

These are created by SolidWorks any time a cylindrical or conical solid is created.

□ Axes

These are created manually by the user.

Viewing Axes

Axis visibility can be turned on or off from the **View** menu. Click **View**, **Temporary Axes** or **View**, **Axes** to toggle the axes on or off.

The Heads-up toolbar can also be used to toggle the axes on or off.

 View the temporary axes. Click
 View, Temporary Axes. This will be the axis that will be used to pattern the spokes.



2 Create a circular pattern. Click

Circular Pattern () on the Features toolbar.

- **3** Select the Temporary Axis in the graphics area.
- 4 Click the Features to Pattern box to make it active.
- 5 Click the Circular Pattern title at the top of the PropertyManager to fly-out the FeatureManager design tree.
- 6 Select Spoke, M6 Clearance Hole1, Hex-Cut and Hex-Cut Chamfer. These are all the features we want to pattern.
- 7 For the number of instances either type **3** or use the arrows to change the number to **3**.
- 8 Make sure the angle is set to 360 degrees and Equal spacing is selected.
- Click Interpretent with the two holes and the chamfer have been patterned.

Task 14 — Add Fillets

Fillets need to be added to the spokes to round all the edges. Rather than add the fillets to all three spokes individually, we will just add the fillets to the first spoke, then include the fillets into the pattern.





- SolidWorks Engineering Design and Technology Series
 - 1 Locate the first spoke. In the FeatureManager design tree, select the feature named Spoke. By selecting a feature in the FeatureManager design tree, the feature will be highlighted in the graphics area.

- 2 Add a **4mm** fillet to both sides of the spoke where it contacts the hub.
- **3** Rename this feature **Fillet R4**.

- 4 Add a **2mm** fillet to the three edges shown.
- **5** Rename this feature **Fillet R2**.

- Task 15 Add the Fillets to the Circular Pattern
 - 1 Right-click the feature CirPattern1 in the FeatureManager design tree and select **Edit Feature**.

We want to add the two fillets into the definition of the circular pattern, however they are grayed out and cannot be select.

They cannot be selected because they were created later in time than the circular pattern, so when the circular pattern was created, the fillets did not exist.

2 Exit the circular pattern by clicking **Cancel** X.









Reordering Features

Features can be reordered in the FeatureManager design tree by simply dragging and dropping the feature in a new location.

When dragging a feature, the cursor will change to \checkmark indicating that the new location will be under the feature highlighted. If you drag the feature too far up the

FeatureManager design tree, the cursor will change to 🔕, indicating that you cannot drop the feature at this location.

Parent/Child Relationships

The parents and children of any feature determine its relationships. Parents are used to create the new feature; the new feature is then dependent on the parent. For the child feature to exist, the parent feature must exist.

Parent/Child relationships can be determined by right-clicking a feature and selecting **Parent/Child** from the menu.

 In the FeatureManager design tree, right-click Fillet R4 and select Parent/Child from the menu.

The fillet has two parents, the Spoke and the Hub. This is logical as the fillet connect the two features. If we move the fillet up the FeatureManager design tree, it can never go before either the Spoke or the Hub.

The fillet has no children, in other words, no features depend on the fillet.

- 2 Reorder the features. Drag the feature Fillet R4 to a position between the Rim and the CirPattern1.
- 3 Drag the feature Fillet R2 to a position after Fillet R4.Both fillets now exist before the circular pattern so they can be included in the pattern.
- 4 In the FeatureManager design tree, right-click CirPattern1 and select Edit Feature.
- 5 Select Fillet R4 and Fillet R2 to add them to the **Features to Pattern**.









6 Click **OK**. The two fillets have now been added to the circular pattern and appear on all the spokes.



Task 16 — Add Spoke to Rim Fillets

The spoke to rim fillets provide an additional challenge because of the way the two features meet. We need a relatively large fillet to make the wheel look good.

Because the spokes were patterned before the rim was created, fillets between the two cannot be added to the pattern. We can add all the fillets to the three spokes individually, but this is a lot of work. We might try to fillet one spoke and pattern the fillets, however this doesn't work. Fillets need to be patterned with the underlying geometry.



To allow the fillets to be added to the pattern, we must roll back the model to before the circular pattern.

- 1 In the FeatureManager design tree, right-click CirPattern1 and select **Rollback** [m] from the menu.
- 2 Click Insert, Features, Fillet/Round from the menu.
- **3** Type **6mm** for the fillet radius.
- 4 Select the edge between the inside face of the rim and the side of the spoke as shown. Select the same edge on the other side of the spoke.


- 5 Click . The fillet is added and the adjacent face has been extended.
- 6 Rename the fillet **Fillet R6**.

- 7 Add a **3mm** fillet to the two edges show.
- 8 Rename the fillet to **Fillet R3**.
- 9 Roll the model forward to the end. Right-click in the FeatureManager design tree and select Roll to End.
- 10 Add the two fillets to the circular pattern. Right-click CirPattern1 and select Edit Feature. Select Fillet R6 and Fillet R3 to add them to the Features to Pattern.
- 11 Click 🗹 . All three spokes should now have the same fillets.

12 Add additional fillets to round remaining sharp edges. Add a 3 mm fillet. Select the face and edges shown. When you select a face, all edges of that face will be filleted.







13 Finish the front by adding a 0.5mm by 45 degree chamfer to edge of the hole where the bearing will be inserted.





Task 17 — Add Index Pins and Index Holes

Index pins and holes provide two functions, first they help to line up the parts as they are put together in the assembly process. Second, they prevent the two halves from rotating, relative to one another, when they are in use.

Shared Sketches

To this point, each sketch has been used to create a single feature. There are times when we need to capture the design intent for several features in a single sketch, such as the case of the index pins and index holes. The pins will be created as extrusions while the holes will be created as cuts, but they must have a relationship so that the pins will always align with the holes.

- 1 Change to the Back view by clicking **Back** on the Standard Views toolbar.
- **2** Create a sketch on the back face of the model.
- 3 Sketch a circle with its center on the Origin. Dimension the circle to 77mm.





Options

For construction

4 Change the circle to construction geometry. The pins and holes will be placed on this circle. We do not want this circle to create an extrusion or cut, we just want to use it to line up other sketch entities.

In the PropertyManager for the circle, select **For Construction**. The circle now changes to a construction line.

5 Create two construction lines. The end points of each line must be coincident with the construction circle and each line must be horizontal.





6 Add angular dimensions. To fully define the construction geometry add the angle dimensions show. Select the Smart Dimension tool ?, then select the Origin and each end of a line. Drag the dimension to the position shown.

Repeat for the other dimension.

The four endpoints will be the locations for the pins and holes.

- 7 Sketch a circle at the endpoint of each line.
- 8 Dimension the two top holes. Dimension the circle on the right 3.9mm and the circle on the left 3.8mm. The larger circle will be used to create the hole and the smaller circle the pin. The difference in dimensions allows for a slight clearance to make sure the pins don't stick.
- 9 Add equal relationships. Add an Equal relationship between the two circles on the right. Repeat with the two circles on the left. This will make the two holes the same size and the two pins the same size.

The sketch is now fully defined.



Sketch Contours

Sketch Contours allow you to select portions of a sketch that are generated by the intersection of geometry and create features. This way you can use a partial sketch to create features.

Another advantage of this method is that the sketch can be reused, creating separate features from different portions of the sketch.

10 Extrude the pins. Click **Extruded Boss/Base (ie)** to create an extrusion.

Set **Direction 1** to **Blind** and the depth to **4.5mm**. Rotate the model to allow you to see the preview. If we were to

click \checkmark , we would extrude four pins which is *not* what we want to do.





In the PropertyManager, click the down arrow next to **Select Contours** to expand the selection box. Click the two left circles to select their contours. Only the two circles will now be extruded, nothing else.

Click 🗹 .

- 11 Rename the feature Index Pins.
- 12 Cut the Index Holes. Click the plus sign next to Index Pins and select the sketch under the feature.
- 13 Click Extrude Cut 间.

Set **Direction 1** to **Blind** and the depth to **5.5mm**. Rotate the model to allow you to see the preview.

In the PropertyManager, click the down arrow next to **Select Contours** to expand the selection box. Click the two right circles to select their contours. Only these two circles will now be cut, nothing else.



14 Rename the feature Index Holes.

Click the plus sign next to Index Pins and Index Holes to show the sketch underneath. Notice that both features use the same sketch

and that the sketch icons show the hand *which* indicates sharing.





15 Add a **Chamfer** to the top of the Index Pins. You generally don't leave sharp edges on a small feature like the pin as the edge is likely to be damaged which would cause problems when assembling the parts.

Click **Chamfer C** then select the edges of the two pins. Select **Distance distance** for the type of chamfer and set the two distances to **1.5mm** and **1.0mm**.

Watch the preview as you may have to reverse the order of D1 and D2.





16 Save the Wheel Hub.



Task 18 — Hole For Tube Stem

The tire that will be used has a tube. There must be a cutout in the hub for the tube stem so that the tube can be inflated.

We will create a revolve cut opposite one of the spokes.

- 1 Change to the Back view by clicking **Back** for the Standard Views toolbar.
- **2** Create a sketch on the back face of the model.
- 3 Sketch a vertical centerline from the Origin.
- 4 Sketch a rectangle roughly in the position shown.



Coincident

- 5 Add two **Coincident** relationships as shown. This will insure that we cut all the material in the rim.
- 6 Add a **Collinear** relationship between the left side of the rectangle and the centerline.
- 7 Dimension the width of the rectangle to 5.8mm.
- 8 Create a revolved cut by clicking Revolve
 Cut on the Features toolbar. The cut can revolve 360 degrees even if it is only cutting material for 180 degrees.

9 Add a **0.5mm** by **45deg** chamfer to the edges of the cut as shown.

Task 19 — Add Lettering

Lettering can be added to parts as an extrusion to provide raised letters, or as a cut to provide engraved letters. We will add the name "SolidWorks" to the rim of the wheel for advertising purposes.

Because we want the letters to follow a curved path, we will create some construction geometry to guide the letter placement.







- Change to the Front view by clicking
 Front
- **2** Create a sketch on the front face of the rim.
- **3** Sketch a vertical centerline from the Origin.
- 4 Sketch a centerpoint arc with the center on the Origin. Make the arc wider than the spoke and change it to construction geometry.
- 5 Add a **Symmetric** relationship between the two endpoints of the arc and the centerline.

Click **L** to add a relationship. Select the





- two endpoints and the centerline. Click
- 6 Dimension the arc as shown.
- 7 Add text. Click Tools, Sketch Entities, Text from the menu.
- 8 Click the arc. This is the curve we want the text to follow.
- 9 Click in the Text box to make it active, then type SolidWorks for the text.
- **10** Select **Full Justify** for the text alignment and **Text outside the arc**.
- 11 The default font, set in the options, is too small. ClearUse document font, and click the Font... button.
- 12 Change the font size to 26 points and the style to Bold. Click OK



The text is just sketched lines and arcs that can be extruded or cut as we desire.

Note: Sketch text is created from the font definitions and will always be under defined in the sketch.

While the goal is to created fully defined sketches, having the sketch text under defined is acceptable.

13 We want to cut the text into the surface. Click

Extruded Cut [a]. Cut the letters to a blind depth of **1mm**.

14 Rename the feature **Text**.



Product Visualization

Model Display

Product visualization encompasses all the elements to display our models. There are three ways to display a model in SolidWorks; OpenGL, RealView and Rendered.

□ OpenGL

In Open Graphics Language, models can be displayed as shaded, wireframe or a combination of the two. It does not require special hardware and can be software emulated. All modern graphics cards, regardless of the cost, have OpenGL capabilities and therefore the calculations for OpenGL are normally done on the Graphic Processor Unit (GPU) of the graphics card. These effects show:

- Surface shading including color, ambience, diffusion, specularity and transparency.
- Basic texture mapping techniques.
- Diffused ground shadow.
- □ RealView

RealView incorporates OpenGL, but takes it to another level. More realistic effects can be achieved such as dynamic environmental reflections in the appearance, selfshadows, a ground shadow and a ground reflection, as well as more advanced texture mapping techniques. Most modern video cards, in the professional workstation class, can display all the effects of RealView.

RealView supports:

- Advanced Shading
- Reflections (environmental)
- Self shadows
- Ground reflection
- Advanced texture mapping and bump maps

□ Rendered

Rendering of the SolidWorks model is done using PhotoView 360. PhotoView 360 is distinguished from RealView in that it is not a process of the GPU, and photorealistic renderings can be achieved regardless of the graphics card used. This is because it is entirely calculated on the CPU on your PC and is rendered in a sequential event and is therefore not dynamic. PhotoView 360 renderings are more photorealistic than RealView because light rays and reflections are more accurately calculated. PhotoView 360 rendering supports: Self-reflection (reflection of objects in one another) HDRI (environmental) lighting

Changing Display Modes

Changing between OpenGL and RealView is done by either enabling RealView graphics. If RealView is disabled, you are in OpenGL.

Where to Find It

To toggle between OpenGL and RealView:

- □ Click View, Display, RealView Graphics from the menu.
- □ Click **RealView Graphics** on the Heads-Up toolbar.

Appearances

Appearances control the way the surfaces of the model look. They can be applied at the assembly, part, body, feature or face of a model. All surface of a model have an appearance applied to them.

Materials vs. Appearances

Appearances are different from materials. When we applied a material to a part, it defined the physical properties of the part such as density and strength. Included in the material definition is a default appearance, however this can be changed. As an example, if we were modeling a body component for an automobile, the material would be steel, however the final component would be painted, so the appearance could be something other than what was applied with the steel material.

To add an appearance:

- □ Click Edit, Appearance, Appearance from the menu.
- □ Click **(**) on the **Standard** toolbar.
- □ Select the Appearances, Scenes and Decals tab 🥘 in the Task Pane.

1 Add an appearance to the part. Select the **Appearances**, **Scenes**, **and Decals** tab of the Task Pane. Click the plus sign next to Appearances to expand the list. Click the plus sign next to Plastic and then select High Gloss. In the lower pane, drag the appearance light grey high glass plastic into the graphics area. This will apply the appearance to the entire part.



🐑 🖾 🔂 🍺

🗔 High Gloss

🗔 Textured

And I we dia the second second

« Appearances, Scenes, and Decals 🖓

2 Add an appearance to the Text feature. Select the Text feature in the FeatureManager design tree. Select the Appearances, Scenes, and Decals tab of the Task Pane. Click the plus sign next to Appearances to expand the list. Click the plus sign next to Plastic and then double-click red high gloss plastic.



Display Manager

The DisplayManager is like the FeatureManager design tree except is shows the different display properties of the file. These include appearances, scenery, decals, cameras and lighting associated with the active SolidWorks part or assembly.

The DisplayManager also makes it easy to:

- □ Understand the way in which appearance and decal inheritance works.
- □ Select and edit appearances and decals associated with the model.
- □ Access the appearance, scene, lighting, camera and decal properties.
- □ Transfer appearances and decals between components, features and faces.

The DisplayManager has four different sections:	DisplayManager tab
View Appearances	
□ View Decals	View Appearances
□ View Scene, Lights and Cameras	View Decals View scenes, Lights and Cameras
PhotoView 360 Options	PhotoView 360 Options

- 1 Examine the DisplayManager. Select the DisplayManager tab and then select **View Appearances**.
- 2 In turn, select **History**, **Alphabetical** and **Hierarchy**. **History** shows the time order in which the two appearances were added. **Alphabetical** lists the appearances by their names and **Hierarchy** according to the rules used to show which appearance is on top.



 Create a pattern of the Text feature. Turn on the Temporary Axes in the View menu and create a pattern of three instances of the Text.

In the PropertyManager for the Circular Pattern, select **Propagate visual properties**. This will make the red high gloss plastic appearance attach to the new pattern instance.





4 Now that the part is finished, we will add a scene. Select the **Appearances**, **Scenes**, **and Decals** tab of the Task Pane. Click the plus sign next to Scenes to expand the list. Click the plus sign next to Basic Scenes and then locate Backdrop Studio Room. In the lower pane, drag the appearance Backdrop Studio Room into the graphics area. This will apply the scene to the model. 5 Change display modes. Toggle RealView on and off either through the Heads-Up toolbar or the View menu. Toggle on Shadows in shaded mode from the Heads-Up toolbar. Compare the results. There are reflections in RealView that will move on the model as you rotate it. In OpenGL, there are no reflections visible.



6 Save and Close the Wheel Hub.

Active Learning Experience, Part 2 — Importing Data

Parts of any design may come from other sources. There is no need to reconstruct new geometry if it can instead be imported from a file that already exists.

Imported data can be used either through direct translation or by way of neutral file formats.

Direct translators allow a file saved in another CAD format to be opened directly in SolidWorks. SolidWorks has direct translators to open files created in Inventor, SolidEdge, Unigraphics, Pro-Engineer, CADKEY, Rhino and others.

Translation through neutral file formats requires the file created in another program to be saved as a neutral format. None of the existing CAD programs use the



neutral formats directly. The neutral formats only provide a "common ground" that both programs can use. SolidWorks can then read the neutral format and convert the data to SolidWorks data. The two most widely used neutral formats are IGES and STEP.

Task 1— Import the Tire

For the mountainboard, we are not going to design and manufacture the tire and the tube, rather we have found a supplier that makes both of these items. To include the tire and tube in our assembly, we need a CAD model. The supplier uses a CAD system that saves files in a format not supported by SolidWorks direct translators. To get the CAD model, the supplier has provided them in two neutral formats.

- 1 Click **File**, **Open** from the menu.
- 2 Select IGES (*.igs, *iges) from the Files of type list.

Examine the list to see the other files types that can be opened in SolidWorks.

- **3** Select the file Tire.igs from the LessonO4 folder.
- 4 Click the **Options** button.

The options allow us to change the settings for the import process.

The IGES export process breaks down the solid model into individual entities. The import process tries to reconstruct the model. Because of differences in the methods used by various CAD software to create models, there can be errors in the translation.

SolidWorks Files (*.sldprt; *.slda 🔻

SolidWorks Files (*.sldprt; *.sldasm; *.slddrw) Part (*.prt;*.sldprt) Assembly (*.asm;*.sldasm) Drawing (*.drw;*.slddrw) DXF (*.dxf) DWG (*.dwg) Adobe Photoshop Files (*.psd) Adobe Illustrator Files (*.ai) Lib Feat Part (*.lfp;*.sldlfp) Template (*.prtdot;*.asmdot;*.drwdot) Parasolid (*.x_t;*.x_b;*.xmt_txt;*.xmt_bin) STEP AP203/214 (*.step;*.stp) ACIS (*.sat) VDAES (*.vda) VRML (*.wrl) STL (*.stl) CATIA Graphics (*.cgr) ProE Part (*.prt,*.prt.*;*.xpr) ProE Assembly (*.asm;*.asm.*;*.xas) Unigraphics (*.prt) Inventor Part (*.ipt) Inventor Assembly (*.iam) Solid Edge Part (*.par;*.psm) Solid Edge Assembly (*.asm) CADKEY (*.prt;*.ckd) Add-Ins (*.dll) IDF (*.emn;*.brd;*.bdf;*.idb) Rhino (*.3dm) All Files (*.*)

- 5 Set **Options**. Select:
- □ Surface/solid entities
- □ Try forming solid(s)
- □ Perform full entity check and repair errors
- □ Automatically run Import Diagnostics.

These options tell SolidWorks to make the import entities into a solid model if it can and if there are errors, use the tools it has to fix any errors it detects.

Import Options		×
File Format		
General STL/VRML IDF	✓ Surface/solid entities ● Try forming solid(s) ■ B-Rep mapping ● Knit surface(s) ● Do not knit ✓ Merge Entities ■ Import as sketch(es) ● Import as 3D curves ■ Import multiple bodies as parts ✓ Perform full entity check and repair errors ✓ Automatically run Import Diagnostics (Healing) □ Customize curve tolerance: □ Unit ● File specified unit ● Document template specified unit □ IGES ■ Show IGES levels STEP ■ Map configuration data UG □ Import tool bodies.	
Reset	,	
		OK Cancel Help

- 6 Click **OK** then **Open**. Watch to progress as the new solid is created. Notice that this can be a long process depending on the complexity of the model and the speed of your computer.
- 7 When asked, click **Yes** to run Import Diagnostics on the

part. When Import Diagnostics complete, click \checkmark .

- 8 Examine the FeatureManager design tree. There is only one feature for the tire called Imported1. The translation process, through neutral translators, does not provide any of the individual feature information, only a single body. We can add additional features to this model, but we cannot see the original features used to create the geometry that is now Imported1.
- **9** Save the Tire as a SolidWorks part to the Mountainboard\Wheel Assembly folder.
- 10 Close the Tire part.

Task 2 — Import the Tube

The Tube was provided as a STEP file. STEP is also a neutral format like IGES but is newer and gaining more popularity.

- 1 Click File, Open from the menu.
- 2 Select STEP AP203/214 (*.step,*.stp) from the Files of type list.
- **3** Select the file Inner Tube.STEP from the LessonO4 folder.



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T	
% Tire (Default)	
Annotations	
→ B Material < not specified>	
——————————————————————————————————————	
🕂 ∔ Origin	
Imported1	

SolidWorks Files (*.sldprt; *.slda 🔻

SolidWorks Files (*.sldprt; *.sldasm; *.slddrw) Part (*.prt;*.sldprt) Assembly (*.asm;*.sldasm) Drawing (*.drw;*.slddrw) DXF (*.dxf) DWG (*.dwg) Adobe Photoshop Files (*.psd) Adobe Illustrator Files (*.ai) Lib Feat Part (*.lfp;*.sldlfp) Template (*.prtdot;*.asmdot;*.drwdot) Parasolid (*.x_t;*.x_b;*.xmt_txt;*.xmt_bin) IGES (*.igs;*.iges) ACIS (*.sat) VDAFS (*.vda) VRML (*.wrl) STL (*.stl) CATIA Graphics (*.cgr) ProE Part (*.prt, *.prt. *;*.xpr) ProE Assembly (*.asm;*.asm.*;*.xas) Unigraphics (*.prt) Inventor Part (*.ipt) Inventor Assembly (*.iam) Solid Edge Part (*.par;*.psm) Solid Edge Assembly (*.asm) CADKEY (*.prt;*.ckd) Add-Ins (*.dll) IDF (*.emn;*.brd;*.bdf;*.idb) Rhino (*.3dm) All Files (*.*)

4 Click **Open**. The Inner Tube will open much faster than the Tire because it is a much simpler part.

Again there is only one feature, Imported1.

5 Save and Close the Inner Tube as a SolidWorks part to the Mountainboard\Wheel Assembly folder.



Active Learning Experience, Part 3 — Create the Wheel Assembly

We now have the basic parts to create a wheel assembly. We will use two Wheel Hubs plus the Tire and Inner Tube.

Task 1— Create a Wheel Assembly

- 1 Before creating the assembly, make sure that the three parts we have created in this lesson are all in the same folder. The three parts Wheel Hub, Tire and Inner Tube should all be in the folder SolidWorks Curriculum and Courseware_2011\ Mountainboard Design Project\Mountainboard\Wheel Assembly. If they are located someplace else, use Windows Explorer to move them to the correct folder.
- 2 Create a new assembly. Click **File**, **New** and select the **Assembly_MM** template from the Training Templates tab.
- 3 Insert the first part. Click the **Browse** button in the PropertyManager. Locate the part Wheel Hub and click **Open**. The wheel hub will appear transparent, drag it to the

assembly Origin. When the cursor changes to \searrow release the mouse button.

- 4 The Wheel Hub is now fixed in space with its Origin mated to the assembly Origin and its three principal planes mated to the three planes of the assembly.
- 5 Add another instance of the Wheel Hub. The wheel assembly will use two wheel hubs mounted back to back. Because we already have one instance of the wheel hub in the assembly, we can insert another instance by dragging it from the FeatureManager design tree.

6 Hold down the **Ctrl** key and drag the Wheel Hub from the FeatureManager design tree and drop it in the graphics area.

The FeatureManager design tree shows two instances of the Wheel Hub. Instance one is fixed in space and the second instance still has some degrees of freedom as noted by the minus sign.





Smart Mates

Smart Mates simplify the mating process by allowing you to drag the face you want to mate onto another face. To add **SmartMates** you hold down the **Alt** key while dragging and dropping a selected face or edge.

These mates use the same **Mate Pop-up** Toolbar as the **Mate** tool uses to set the type and other attributes. All mate types can be created with this method.

Certain techniques generate multiple mates and do not use the toolbar. These require the use of the **Tab** key to switch mate alignment.

Filters

Selecting edges and faces is often tricky because of adjacent edges or faces. In order to restrict selection, the **Selection Filters** option is used.

The Filter toolbar may be shown or hidden by selecting View, Toolbars, Selection Filters from the menu.



7 Most of the mating entities we will be choosing are faces. To make it easier to select faces, we will turn on a filter so that only faces can be selected. Click **Toggle Selection**

Filters Toolbars on the Standard toolbar to open the Filter toolbar. Then click

Filter Faces 📑 .

Note: The Filter Faces option can also be turned on using the keyboard shortcut **X**.

The cursor will show that a filter is applied by changing to $k_{\rm T}$. When the cursor is

over a face, the cursor will change to $k \equiv 0$.

8 Add a **Concentric** mate. Select Face 1, then press and hold down the **Alt** key and drag Face 1 to Face 2. When the

cursor changes to $\overline{\begin{subarray}{c} $\mathbf{\dot{c}}$}$, indicating a concentric mate, release the **Alt** button.



9 Reverse the alignment. The two cylindrical faces can be concentric in two orientations. The initial orientation is based on which of the two possibilities is closest. Press the Tab key. This will toggle the two possible alignments.

When the two hubs are facing in opposite directions, release the mouse button.

10 The two Wheel Hubs will align concentric and the Mate **Pop-up** toolbar will appear.



- 11 Click \checkmark to accept the mate.
- 12 Move the second Wheel Hub. We need three mates to properly position the second Wheel Hub. First is the Concentric mate already added. Second is a Concentric mate between an Index Pin and the corresponding Index Hole. Finally a Coincident mate between the inside faces of the two Wheel Hubs to keep them together.

It is easier to select the Index Pin and Index Hole if the two parts are moved apart.

Drag the second Wheel Hub to the right to separate it from the first Wheel Hub. The exact distance is not important, only that you can select the pin and hole.





13 Mate an Index Pin to the corresponding Index Hole. Click

on the **Assembly** toolbar. Select the cylindrical face of an Index Pin. Rotate the view so you can see the corresponding Index Hole, then click the inside face of the hole. Click **OK**.



Note: Check to make sure that the hole for the Tube Stem lines up on the two Wheel Hubs. If it does not, edit the last mate and change the mate to the correct hole.

- 14 Mate the inside faces of the two Wheel Hubs together with a Coincident mate. Click so on the Assembly toolbar. Select the inside faces of the two Wheel Hubs, then click OK.
- 15 Save the assembly as Wheel_Assembly to the Mountainboard\Wheel Assembly folder.
- 16 Turn off the Face Filter.



Task 2 — Add the Tube

The Inner Tube must be mated to the Wheel Assembly. The difficulty is that it doesn't have any surfaces that lend themselves to mating to the wheel hub. The Tube will be mated to the assembly using reference geometry.

Adding Parts to an Assembly

Part can be added to an assembly in several ways:

- Using the menu. Click Insert, Component, Existing Part/Assembly.
- Drag from an open part/assembly window.
- □ Drag from Windows Explorer.
- 1 **Open** the part Inner Tube.
- 2 Tile the windows. Click Window, Tile Vertically.

3 Drag the top level icon from the Inner Tube FeatureManager design tree into the assembly window.



- 4 In the FeatureManager design tree, click the plus sign next to the part Inner Tube to expand the listing.
- 5 Click on the Assembly toolbar. Select the Front plane of the assembly and the Front plane of the Inner Tube. Coincident should be selected for Standard Mates.

Click **OK**. The Inner Tube will move into the plane of the hub.

6 Repeat the above step to mate the Top planes of the assembly and Inner Tube **Coincident**, then the Right planes, also **Coincident**.



7 Check the assembly. The Inner Tube should be correctly positioned on the Wheel Hub with the Tube Stem coming through the hole in the Wheel Hub.



Task 3 — Add the Tire

The procedure for adding and mating the Tire is essentially the same as for adding the Inner Tube.

- 1 **Open** Windows Explorer.
- 2 Locate the Tire.sldprt created earlier.
- 3 Drag the Tire.sldprt into the assembly window.



4 Add mates between the three principal planes of the Tire and the assembly just like we did for the inner tube.

Task 4 — Toobox setup

Assembly hardware is rarely designed and manufactured for a specific project as it is cheaper to buy available material from existing suppliers. Within the SolidWorks modeling environment the same is true; we do not want to have to create standard fasteners, we would rather just use pre-made models.

Toolbox

SolidWorks Toolbox is a time-saving library of standard parts that uses Smart Part Technology to automatically select the appropriate fasteners and assemble them in the proper sequence. Toolbox is a SolidWorks add-in program which means it works inside of SolidWorks.

Toolbox can create the fasteners in two different ways. For this course we want Toolbox to create a new part for each fastener and store it in a folder with the rest of our course files.

- 1 Turn on Toolbox. Click **Tools, Add-Ins** from the menu.
- 2 Select both SolidWorks Toolbox and SolidWorks Toolbox Browser. Click OK.

Toolbox will be added to the Design Library.

3 Click **Toolbox**, **Configure** from the menu.



4 Select Define user settings.

Select **Create Parts**. Click **Browse** in then navigate to the SolidWorks Curriculum and Courseware_2011\Mountainboard Design Project \Mountainboard\Hardware folder.

Select Error when writing to a read-only document.

Click Save, then Close.

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Toolbox

💻 Ansi Inch

🛓 🔤 Ansi Metric

Design Library

5 Open Toolbox Browser

Expand **Toolbox** in **Toolbox** on the Design Library Task Pane. The Toolbox Browser appears.

The Toolbox Browser is an extension of the Design Library that contains all available Toolbox parts.

The Toolbox Browser is organized like a standard Windows Explorer folder view.

- 6 Click the pushpin is to keep the Design Library open as we select other things.
- 7 Select the folder Ball Bearings under SKF®\ Bearings.

8 Drag a Radial Ball Bearing and drop it on the bearing cutout in the Hub.

The Radial Ball Bearing property box will appear.



9 Select the size **6001** and a Display of **Detailed**.

This bearing has a Bore of 12mm and an Outside diameter of 28mm which is what we need to fit the hole in the Wheel Hub.

Click ✓. A bearing part will be created and added to the assembly.

Note: Depending on where you drop the bearing, you may get the Mate Pop-up menu to let you mate the bearing. If you do not get the menu, don't worry, we will add the mates manually.

- **10** There will still be a bearing preview on your cursor, rotate the assembly so you can see the other side and drop a second bearing at the other bearing cutout.
- 11 Click **Cancel** X in the PropertyManager to stop adding bearings.

12 Add a **Concentric** and **Coincident** mates to position the bearing so it is bottomed in the bore of the Wheel Hub.



Mate References

Most Toolbox parts have mate references assigned so that they will automatically snap into position when added to an assembly. Bolts and Nuts have mate references that will mate them concentric to the bolt hole and coincident with the end surface of the hole.

Mate references are designated entities such as planes, edges or vertices that allow the part to be dragged from Toolbox, the FeatureManager design tree or Windows Explorer as if it were being dragged by that entity.

Adding Multiple Toolbox Parts At Once

Toolbox parts can be added to multiple locations at the same time. Instead of dragging the part from the Toolbox Browser, the mating location or locations are selected first. To insert the Toolbox item, right-click the part in the Toolbox Browser and select **Insert into assembly**.

- 1 Change the view orientation to the Front view.
- 2 Select the Ansi Metric, Bolts and Screws, Socket Head Screws folder under Toolbox.
- **3** Press and hold **Ctrl** and select the three edges of the bolt holes.
- 4 Right-click the nut Socket Head Cap Screw Ansi B18.3.1M and select Insert into assembly.



- **5** Select the following properties:
 - Size: **M6**
 - Length: 25
 - Drive Type: **Hex**
 - Thread Length: 25
 - Thread Display: Simplified

Click 🗹 .

Cap Screws have been inserted into each of the three holes.





Thread Display

While fasteners such as bolts and screws are fairly detailed parts, they are also very common ones. In general, bolts and screws are not the parts that you design. Instead you will use off-the-shelf hardware components. It is a well-established design practice to not draw all of the details of fasteners, but to specify their properties and show only an outline — or simplified — view of them.

The three display modes for bolts and screws are:

- Simplified Represents the hardware with few details. Most common display. Simplified display shows the bolt or screw as if it were unthreaded.
- Cosmetic Represents some details of the hardware. Cosmetic display shows the barrel of the bolt or screw and represents the size of the threads as dashed lines.
- Schematic Very detailed display which is rarely used. Schematic shows the bolt or screw as it really appears. This display is best used when designing a unique fastener or when specifying an uncommon one.



Task 5 — Examine the Mates

Each of the three Cap Screws added by Toolbox was placed with two mates, a **Concentric** mate to center the bolt in the hole, and a **Coincident** mate where the bolt would stop if it were pushed into the hole. To view the mates:

- 1 Click the FeatureManager design tree tab.
- 2 Click the plus sign next to Mates to expand the mate group.
- 3 Examine the entries. Each Cap Screw has two mates.

Coincident11 (Wheel Hub<1>,Socket Head Cap Screw_AM<1>)
 Concentric8 (Wheel Hub<1>,Socket Head Cap Screw_AM<1>)
 Coincident12 (Wheel Hub<1>,Socket Head Cap Screw_AM<2>)
 Concentric9 (Wheel Hub<1>,Socket Head Cap Screw_AM<2>)
 Coincident13 (Wheel Hub<1>,Socket Head Cap Screw_AM<3>)

Task 6 — Add the Nuts

The three nuts can be added with the same procedure. One difficulty will be aligning the flats of the nuts with the holes as the Toolbox nuts do not have mate references to create this alignment.

- 1 Change the view orientation to the Back view.
- 2 Select the Ansi Metric, Nuts, Hex Nuts folder under Toolbox.
- **3** Press and hold **Control** and select the three edges of the bolt holes.

Note: You must be careful to select the edge of the bolt holes and not one of the edges of the bolt. If you have the wrong edges selected, one or more of the bolts will not be able to be mated in the assembly and will cause an error.



- 4 Right-click the nut **Hex Nut Style 1-ANSI B18.2.4.1M** and select **Insert into assembly**.
- 5 Select the sizes shown, then click ✓.
 Three nuts will be inserted into the assembly.

6 Examine one of the nuts. Zoom in on any of the three nuts and observe its position. Note that the flats on the nut do not line up with the flats in the hole.

The mate references contained in the file that creates the nuts do not contain anything that will make these faces parallel. We have to do this manually by adding a **Parallel** mate between a flat on the nut and the flat in the hole. Zoom in on one of the nuts.



Should be parallel



7 Turn on the **Filter Face** by clicking **i** on the **Filter** toolbar.

Select Other

There are many times that the face we are trying to select is hidden behind another object. In the case of the nuts in our current assembly, it is difficult to select either the flats on the nuts or the flats in the holes.

Select Other is used to select hidden faces or the model without reorienting it.

To select faces that are hidden or obscured, you use the **Select Other** option. When you position the cursor in the area of a face and press the right mouse button, **Select Other**

is available as an option on the shortcut menu. The face closest to the cursor is hidden and listed in the dialog under --*Hidden Faces*--. Other visible faces are numbered and listed in the dialog. Moving over them in the dialog highlights them on the screen.

The reason the system hides the closest face is since that one was visible, if you wanted to select it you would have simply picked it with the left mouse button.

- 9 Click Insert, Mate to open the Mate PropertyManager.
- Place the cursor as shown and right-click. Choose Select Other. The top face of the nut will become transparent and we are looking at the faces on the inside of the nut.

The cursor will change to . To select a face under

the cursor, you can click the left mouse button. To remove a face so you can see deeper into the model, click the right mouse button.

- 11 Move your cursor over the list. Each face will highlight when the cursor moves over it in the list.
- 12 Click the left mouse button over the face of the B18.2.4.1M Hex nut to accept this face. This face will now be listed in the Mate PropertyManager.
- 13 Select one of the flat faces of the Hex-Cut. Depending on the orientation of the model and the nut, you can either pick the face directly or use Select Other.





16 Save the assembly.

14 Apply a **Parallel** mate. The nut will now rotate to the correct position.

15 Repeat the above steps to align the other two nuts.





Section View

Section View cuts the view using one or more section planes. The planes can be dragged dynamically. Reference planes or planar faces can be used.

To check our work and see how the parts fit together, we can use the section view.

- 1 Orient the model to the Isometric view.
- 2 Click Section View 🛐 on the Heads-Up toolbar or View, Display, Section View on the menu.
- In the PropertyManager, click to select the Right plane as the section plane.





- 4 Reorient the model to the Right view.
- **5** Examine the model. Check the fit of the bolts and bearings.

While in the section view, you can zoom and pan to get a better look at the individual features.

- 6 To return to the normal view, either click Section View and on the Heads-up View toolbar or clear View, Display, Section View on the menu.
- **7** Save the assembly.



Active Learning Experience, Part 4— Create an Exploded View of the Wheel Assembly

Exploded Views

Exploded views are created to make it easier to see how an assembly is put together and to see the parts that are normally hidden from view.

You make **Exploded Views** of assemblies by moving the assembly components one at a time or in groups. The assembly can then be toggled between normal and exploded view states. Once created, the **Exploded View** can be edited and also used within a drawing. **Exploded Views** are saved with the active configuration.

You can only create one exploded view for each configuration.

Task 1— Create An Exploded View

- 1 Orient the assembly to the Isometric view.
- 2 Click Insert, Exploded View from the menu. This opens the Assembly Exploder.
- **3** Exploded Views are created one step at a time. To create a step, select a component either in the graphics area or the FeatureManager design tree.

4 Select the bearing that is visible. A manipulator triad will appear. To move a component, drag one of the manipulator handles.

- 5 Drag the blue manipulator handle to the left. The bearing will move in the Z direction. A ruler will appear to help determine the distance.Drag the bearing about 140mm.
 - **Note:** The exact distance you move the individual components is not important as you are trying to show a picture of how the components fit together.
- 6 When you drop the part, it will change color to magenta and there will be a blue drag handle to further refine the position.
- 7 When you have the part in the position you desire, click any clear area of the graphics area or select the next part you want to move.
- 8 Expand the flyout FeatureManager design tree.
- **9** Press and hold the **Ctrl** key and select the three bolts.







- 10 Drag the blue manipulator about100mm and all three bolts will move.When done, click in the graphics area to complete the step.
- Reorient the assembly so you can see the three nuts. Hold down the Control key and select them all.
- **12** Drag the nuts away from the assembly.
- **13** Select the second bearing and move it toward the three nuts.
- 14 Select one of the Wheel Hubs and move it by the blue manipulator handle.
- **15** Repeat for the other Wheel Hub.
- 16 We want the Inner Tube to make two moves. First will be along the same direction as the other components, then we want it to move up along the Y direction.
- 17 Select the Inner Tube and move it by the blue manipulator.



18 Select the Inner Tube again, this time drag the Inner Tube by the green manipulator handle.



Explode Steps

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🖅 🚽 Explode Step1

🛓 🛃 Explode Step2

🗄 🚽 Explode Step5

🚽 Explode Step6

🤏 Wheel Hub-2

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Adjusting the Steps

Now that all the parts have been exploded, we need to adjust their positions so that we can clearly see each component of the assembly.

The explode steps have been listed in the PropertyManager. If you click the plus sign next to any step, you will see the component or components that are moved during that step.

To change the distances, either right-click a step and select **Edit Step**, or just select the step. In each case the component or components will change color to magenta and the blue drag handle will appear.

- 1 Select Explode Step1 in the PropertyManager. Drag the bearing to a new position.
- 2 Repeat with each Explode Step until all the components are spaced as shown.
- Click to finish exploding the assembly.
- 4 **Save** the assembly.


Task 2 — Collapse the Exploded View

The Exploded View information is stored in the ConfigurationManager.

- 1 Click the ConfigurationManager tab [at the top of the FeatureManager design tree.
- 2 Click the plus sign in front of Default to expand the configuration tree.
- 3 Click the plus sign in front of ExplView1 to show the individual steps in the Explode sequence.
- 4 Right-click ExplView1 and select **Collapse**.

The exploded assembly will collapse to the assembled form. To explode the assembly, right-click ExplView1 and select **Explode**.

Exploded Animations

The Explode and Collapse sequence can also be animated where the steps will take place in sequence. This can make it easier to see the process.

Animation Controller

The Animation Controller controls the recording and playback of the animation.



- **5** Right-click ExplView1 and select **Animate Collapse**.
- 6 The assembly will collapse and the Animation Controller will appear.
- 7 Click the Reciprocate on the Animation Controller. The animation will continue to explode and collapse.
- 8 To end the animation, click **Stop**
- **9 Close** the Animation Controller.

Note: When the Animation Controller is open, you cannot access other SolidWorks functions. You must close the Animation Controller first.

- **10** Collapse the assembly.
- **11 Save** the assembly.

5 Minute Assessment – #4

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

2	Examine the three illustrations at the right. Which one is not a valid sketch for a revolve feature? Why?			
3	What does the Convert Entities sketch tool do?	Â	В	• c

- 4 In an assembly, parts are referred to as ______.
- **5** True or False. A fixed component is free to move?
- **6** True or False. Mates are relationships that align and fit components together in an assembly.
- 7 How many components does an assembly contain?
- 8 In which window do you find ready-to-use hardware components?
- **9** True or False: Parts from Toolbox automatically size to the components they are being placed on.
- **10** True or False: Toolbox parts can only be added to assemblies.

Exercises and Projects — Additional Mountainboard Parts

The following parts will be needed for the suspension of the mountainboard. The compression spring will be made in a latter lesson.



Exercise 11: Fender Washer

The Fender Washer is a simple revolved part.



- 1 Revolve the washer from the sketch shown.
- 2 Save the part as Fender Washer to the Mountainboard\Parts folder.



Exercise 12: Spring Dampener

Create the Spring Dampener as a revolved part.

1 Create a sketch on the Front plane.

2 Create two centerlines from the Origin. Select the horizontal centerline and click **Dynamic**

Mirror Entities 🙆.

3 Sketch a vertical line from the Origin, then from its end sketch a horizontal line.





Exercise 13: Spring Retainer



1 Create a base revolve using the dimensions shown.



2 Create a recess cut 13mm in diameter and 0.75mm deep.



3 Create a Through All cut, 5mm in diameter.

4 Create a new plane **25mm** from the top face of the model.

5 Extrude a Blind cut.

Sketch a circle **6mm** in diameter on the new plane. Extrude a cut to a Blind depth of **19mm**.

6 Add a 1 mm fillet and a0.5mm by 45° chamfer to the edges shown.









 Add appearances to the part. Make the overall part black high gloss plastic and the center hole and the chamfers on each end yellow high gloss plastic.



- 8 Save the part as Spring Retainer to the Mountainboard\Parts folder.
- 9 **Close** the part.

Exercises and Projects — Revolved Parts

Exercise 14: Flange

Create this part using the dimensions provided. Use relations wisely to maintain the design intent.

This lab uses the following skills:

- □ Revolved features.
- □ Circular patterning.

Units: inches

Design Intent

The design intent for this part is as follows:

- □ Holes in the pattern are equally spaced.
- □ Holes are equal diameter.
- □ All fillets are equal and are **R0.25**".

Note that construction circles can be created using the **Properties** of a circle.

Dimensioned Views

Use the following graphics with the description of the design intent to create the part.

Top View







Front View



Exercise 15: Wheel

Create this part using the dimensions provided. Use relations wisely to maintain the design intent.

This lab uses the following skills:

- □ Revolved features.
- □ Optional: Text in a sketch.

Units: millimeters

Design Intent

The design intent for this part is as follows:

- □ Part is symmetrical about the axis of the hub.
- □ Hub has draft.



Dimensioned Views

Use the following graphics with the description of the design intent to create the part.

Front and Top views, and Section A-A from Front view.



Optional: Text in a Sketch

Text can be added to a sketch and extruded to form a cut or a boss. The text can be positioned freely, located using dimensions or geometric relations, or made to follow sketch geometry or model edges.

1 Construction geometry.

Sketch on the front face and add construction lines and arcs as shown.

TIP: Use **Symmetric** relationships between the endpoints of the arcs and the vertical centerline.



2 Text on a curve.

Create two pieces of text, one attached to each arc. They have the following properties:

Text: Designed using

- Font: Courier New 11pt
- Alignment: Center Align
- Width Factor: 100%
- Spacing: **100%**
- Text: SolidWorks
 - Font: Arial Black 20pt.
 - Alignment: Full Justify
 - Width Factor: 100%
 - Spacing: not applicable when using Full Justify
- **3** Extrude.

Extrude a boss with a **Depth** of **1mm** and **Draft** of **1°**.

4 Save the part and close it.



Lesson 4 — Quiz

Name: C1	1966.	Date
	lass	Date

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

- 1 How do you start a new Assembly document?
- **2** What are components?
- 3 The **Convert Entities** sketch tool projects selected geometry onto the ______ plane?
- 4 True or False. Edges and faces can be selected items for Mates in an assembly.
- **5** A component in an assembly displays a (-) prefix in the FeatureManager. Is the component fully defined?
- **6** What actions do you perform when an edge or face is too small to be selected by the cursor.
- 7 How do you establish a mate relationship between a Toolbox part and the part it is being placed on?
- 8 How would you determine the correct length of a machine screw that fastens two parts using a washer, lock washer, and nut?
- 9 How do you specify the location of a Toolbox part?
- **10** True or False. Screw threads are always displayed in Schematic mode showing all details.

Lesson Summary

- □ Revolve feature is created by rotation a 2D profile sketch around an axis of revolution.
- □ The profile sketch can use a sketch line (that is part of the profile) or a centerline as the axis of revolution.
- □ The profile sketch *cannot* cross the axis of revolution.



- □ Files can be imported from other CAD software using neutral file formats.
- □ Toolbox provides ready-to-use parts such as bolts and screws.
- □ Toolbox parts are placed by dragging and dropping them into assemblies.
- □ An assembly contains two or more parts.
- □ In an assembly, parts are referred to as components.
- □ Mates are relationships that align and fit components together in an assembly.
- Exploded views can be animated to more clearly show the assembly steps.

□ You will be able to create, modify and analyze the following part. This is the deck of the mountainboard.



Before Beginning This Lesson

□ Complete the previous lesson: Revolved Features — The Wheel Hub.

Resources for This Lesson

This lesson plan corresponds to:

- Design Tables
- □ *SolidWorks SimulationXpress* in the SolidWorks Online Tutorials.

For more information about the Online Tutorials, See "Online Tutorials" on page 1.

Review of Lesson 4: Revolved Features

Questions for Discussion

- 1 Describe the steps required to create a revolved feature
- **2** Describe an assembly.
- **3** What does the command Convert Entities do?
- **4** What does a selection filter do?
- 5 What does it mean when a component in an assembly is "fixed"?
- **6** What are mates?
- **7** What are degrees of freedom?
- 8 How are degrees of freedom related to mates?

Outline of Lesson 5

- □ In Class Discussion Mechanics of Solids and FEA
- □ Active Learning Exercise, Part 1 Create the Deck
 - Create a layout sketch
 - Extrude as thin feature
 - Cut as a thin feature
 - Add Chamfers
 - Using the Hole Wizard
 - Copy sketch
 - Apply a texture
 - Apply a material
- □ Active Learning Exercise, Part 2 Initial Analysis
 - Using SolidWorks SimulationXpress
 - Add Fixtures
 - Add Loads
 - Analyze the model
 - Examine the results
- □ Active Learning Exercise, Part 3 Configurations
 - Create part configurations
 - Suppress features
 - Create split lines
- □ Exercises and Projects Thin Features
- □ Lesson Summary

In Class Discussion — Mechanics of Solids

- □ Mechanics of Solids
 - Exterior Loads
 - Interior forces
 - Material properties
- Exterior Loads

Exterior loads are determined by using a free body diagram and the application of Newton's Laws.

- □ Free Body Diagram
 - Free Body Diagrams remove the external connections of the model and replace them with the resultant forces and moments acting on the body.
- □ Newton's Laws

Sir Isaac Newton (1642-1727) formulated the fundamental principles of mechanics in three laws.

• First Law

If the resultant force acting on a particle is zero, the particle will remain at rest (if originally at rest) or will move with constant speed in a straight line (if originally in motion).

What this says is that for a body at rest or in constant motion, the sum of the forces in any direction must add up to zero. Also, the moments about any point must also be zero.

• Second Law

If the resultant force acting on a particle is not zero, the particle will have an acceleration proportional to the magnitude of the resultant and in the direction of the resultant force.

If you push on a body, like the mountainboard, it will continue to move faster and faster. If there was no friction or wind resistance, it would just keep accelerating. In the real world, as the mountainboard picks up speed, friction and wind resistance increase until they equal the force you are pushing with. At this point we go back to the first law where the sum of the forces are equal and the mountainboard continues at a constant speed.

• Third Law

The forces of action and reaction between bodies in contact have the same magnitude, same line of action and opposite sense.

□ Internal Forces

- Stress measure of force per unit area. Units are normally pounds per square inch or Newtons per square meter.
- Strain measure of elongation measured in units of length/length such as inches per inch or millimeter per millimeter.

□ Part Deformation

The deformation resulting from totaling all the internal strain.

□ Finite Element Analysis

Finite Element Analysis or FEA is a numerical method to determine properties across a section of interest. FEA is used to solve many problems in machine design, acoustics, electromagnetics, solid mechanics, fluid dynamics and many others. FEA uses numerical techniques to solve field problems described by a set of partial deferential equations.

Active Learning Exercise, Part 1 — Create the Deck

Follow the instructions in this lesson you will create the mountainboard deck, shown below. Once complete, basic analysis will be done to check the strength of the part.



Design Intent

The design intent for the Deck is:

- □ The Deck will be created as a laminated piece.
- □ The Deck is symmetric front to back and side to side.
- □ Mounting holes must be provide to attach the Bindings.
- □ Mounting holes must be provided to attach the Truck.
- □ The Deck must support an average rider of 75 kilograms but should be able to support riders up to 100 kilograms.

Thin Features

Thin Features are made by using an open sketch profile and applying a wall thickness. The thickness can be applied to the inside or outside of the sketch, or equally on both sides of the sketch. Thin features creation is automatically invoked for open contours that are extruded or revolved. Closed contours can also be used to create thin features.

Thin features can be created for extrudes, revolves, sweeps and lofts.

Layout Sketches

Layout sketches can be used to capture some or all of the design intent.

Task 1— Create a Thin Extrusion

- 1 Create a new part using the Part_MM template.
- 2 Open a sketch on the Front plane.
- **3** Draw two construction lines, one vertically from the Origin and the second horizontal through the Origin.

The deck is symmetrical so we will use the vertical construction line to mirror sketch entities.

The horizontal construction line is used to set the bottom of the sketch and make it easier to create a sketch that is close to the correct size.

4 Add a midpoint relationship between the horizontal construction line and the Origin. Press and hold the **Control** key and select both the Origin and the horizontal construction line.

In the PropertyManager select \swarrow to add the **Midpoint** relationship.

- 5 Add a dimension to the line of **1000mm**.
- 6 Mirror as we sketch. Click the vertical centerline, then click **Dynamic Mirror Entities** in the Sketch toolbar to begin mirroring the sketch.
- 7 From one end of the horizontal centerline draw a vertical centerline and dimension it 70mm.
- 8 Click the Point * tool on the Sketch toolbar and click on the horizontal centerline. The sketch mirror will create a second point on the other side of the Origin.



- 9 Dimension the distance between the two points 650mm.
- Sketch a line, tangent arc, and another line as shown.
 Dynamic Mirror Entities will create a mirror image.
- 11 Turn off Dynamic MirrorEntities by clicking a in the Sketch toolbar.
- 12 Sketch a tangent arc to connect the two halves of the sketch.
- 13 Add relationships. Select the left arc and Control select the horizontal construction line. In the PropertyManager select



14 Select the left arc again and control select the left point we added. In the PropertyManager select **Coincident**.

Tangent A

- **15** This makes the arc tangent at this specific point.
- **16** Select the two points shown.In the



17 Add the two dimensions shown.



Make sure that the 175mm dimension is aligned to the line segment After selecting the line, the dimension can be changed to the three possible types of dimensions (horizontal, vertical, aligned) based on the position you drop the dimension. The three possibilities are shown below. To lock the type of dimension, right-click when the cursor position gives you the desired type of dimension. Once locked, the cursor can be moved without changing the type of dimension.



18 Add a dimension from the center arc to the horizontal centerline. Place the dimension below the horizontal centerline and accept the existing dimension.

Arc Conditions

By default, when we dimension from a circle or arc to something else, the dimension will be from the circle or arc centerpoint. This can be changed to the Minimum or Maximum arc conditions by either of two methods:

- Drag the dimension's extension line to the arc or circle
- **□** Edit the Properties of the dimension.

When dimensioning to an arc, Minimum and Maximum conditions still apply. The Minimum condition in this case, appears to be going to a blank space. Why?

Even though we see an arc, the underlying geometry is a circle.



- **19** Change the dimension to the Minimum arc condition. Select the Leaders tab in the PropertyManager.
- 20 Select Min for First arc condition.
- **Note:**If you do not see the three radio buttons for First arc condition, you did not select the arc. You must select on the arc itself, not its center point or endpoint.
- 21 Click OK.
- 22 Double click the dimension and change it to 40mm.
- 23 Click OK.



24 The sketch is now fully defined.





Boss-Extrude

Sketch Plane

Mid Plane

Ŧ

🗙 66

From

Direction 1

- 25 Create a thin extrusion. Click Extrude Boss/Base a on the Features toolbar.
- 26 Select Mid Plane for Direction 1 and set the Depth to 230 mm.

Mid plane depth refers to the total depth of the extrusion. In this case, the extrusion will be 115mm to each side of the sketch.



Because we are extruding an open sketch, **Thin Feature** is checked by default. For thin features, **T1** is the thickness of the extrusion in the sketch plane. Thickness can be added to either side of the sketch or both. Type **12mm** for the thickness.

Check the direction of the thickness. The sketch should be at the bottom of the thickness,

not the top. To change the direction on which the thickness is added, click *M* in the PropertyManager.



- 29 Click . The extrusion is created, 230mm wide and 12mm thick.
- **30 Save** the part as Deck.sldprt to the Mountainboard folder.

Task 2 — Round the Ends

Open sketches can also be used to create cuts. When using an open sketch to create a cut, you must decide which side of the cut to keep and which to remove.

Reorient The View "Normal To"

The **View Normal To** option is used to change the view orientation to a direction normal to a selected planar geometry. The geometry can be a reference plane, sketch, planar face or feature that contains a sketch.

Clicking the **Normal To** icon a second time will flip the orientation around to the opposite side of the plane.

Zoom To Selection

Zoom to Selection Solution in on the selected entity. You can select the entity in either the Graphics Area or the Feature Manager design tree.

 To round the end of the Deck, we want to sketch on the face shown. To make it easier to sketch, change the view to Normal To the face. Select the face and

click **Normal To** on the Heads-up View toolbar.

We are now looking normal to the face on which we want to sketch.

- With the face still selected, click Zoom to Selection (a) on the View toolbar. This will zoom in to the selected face.
- **3** Create a sketch on this face.
- 4 Click on the View toolbar to show Shaded With Edges. This makes it easier to see where each face ends.
- 5 Click the **Centerline** tool **i** and draw a centerline from the midpoint of the vertical edge.
- 6 Click Dynamic Mirror Entities 🙆 on the Sketch toolbar.





REPRODUCIBLE

- 7 Sketch a Line and Tangent Arc as shown.
- 8 Turn off Dynamic Mirror Entities.
- 9 Sketch a Tangent Arc to connect the two existing arcs.

- **10** Add a **Tangent** relationship between the arc and the edge of the board as shown.
- 11 Add a **Coincident** relationship between the endpoint of the line and the vertex as shown.
- **12** The sketch should now be fully defined.
- **13** Add the three dimensions shown.



Tangent

Coincident

- 14 Use the open sketch to cut off the end of the Deck.Click to create a Insert, Cut, Extrude from the menu.
- 15 Reorient the model to the Isometric view and Zoom in. Using an open sketch will automatically choose to cut Through All in both directions. Locate the Flip Side To Cut arrow, it points to the side of the sketch that will be removed. Make sure it is pointing to the outside.



16 To change the side to remove, you can click on the Flip Side ToCut arrow or select Flip side to cut in the PropertyManager.





17 Click \checkmark to complete the cut.

Mirroring Features

Features can be mirrored about a plane or planar face. The mirrored features maintain a relationship with the original features such that any changes to the original features are also made on the mirrored features.

- **18** Click **Insert, Pattern/Mirror, Mirror** from the menu.
- **19** Click on heading **Mirror** in the PropertyManager, this will cause the FeatureManager design tree to "fly-out" over the graphics area.

20 In the FeatureManager design tree, select the Right plane, then select Extrude1. The Right plane is the mirror plane and Extrude1 is the feature we are mirroring. The preview should look like this:



- 21 Click 🗹 .
- 22 Save the part.

Task 3 — Smooth Out The Edges

The transition between the cut we just made and the side of the Deck created a hard edge. Both for aesthetics and safety we would like a smoother transition. This can be accomplished is a large radius fillet.

- 1 Click **Fillet (**) on the **Features** toolbar to create a fillet.
- 2 Select the four edges shown.
- **3** Set the fillet radius to **250mm**.
- 4 Click OK.
- 5 Add a **3mm** by **45deg** chamfer to the top and bottom edge of the Deck.



- 6 Click on the **Heads-up View** toolbar to remove the edge display.
- 7 Save the part.



Task 4 — Add The Binding Mounting Hole Patterns

Mounting holes must be added for the bindings and the trucks. Because the Deck is symmetrical, we could add the holes to one side of the Deck and mirror them to the other. Another approach is to copy the features.

Copy Sketches and Features

Sketches and features can be copied and pasted into new locations using the same method we would use in any other Windows based program.

When we copy and paste features, we are copying both 2D and 3D information. The 2D information is contained in the sketch. The 3D information is the feature definition and end conditions

To copy a sketch or feature, select the sketch or feature, then:

□ Click **Edit**, **Copy** from the menu.

 $\Box \text{ Type } \mathbf{Ctrl} + \mathbf{C}$

To past a sketch or feature, select the plane or face where you want to paste the sketch or feature, then:

- □ Click **Edit**, **Paste** from the menu.
- □ Type **Ctrl + V**
- 1 Create the hole pattern for the Binding. Select the face shown and open a sketch.
- 2 Change the view orientation by clicking

Normal To </u> on the View toolbar. Then,

click **Zoom to Selection** [**Q**].

3 Click to show the model Shaded With Edges.



- 4 Sketch a centerline between the two midpoints as shown.
- 5 Select Dynamic Mirror Entities 🔊
- 6 Sketch one circle.
- 7 Turn off Dynamic Mirror Entities.



- 8 Sketch a second centerline, vertically from the midpoint of the first centerline.
- 9 Select this new centerline, then holdControl and select the two circles.
- **10** Click **Mirror Entities** (1) to mirror the two circles.



- **11** Dimension the sketch as shown.
- 12 Create a cut, Through All.
- **13** Rename this feature Binding Holes.



Task 5 — Copy The Binding Holes Sketch

- 1 Copy the sketch for the Binding Holes. Click the plus sign next to the Binding Holes feature and select the sketch.
- 2 Click **Edit**, **Copy** from the menu. This places a copy of the sketch on the Windows clipboard.

3 Select the face shown. This is where we will paste the sketch.

- 4 Click **Edit**, **Paste** from the menu. The sketch will appear on the selected face but its position will depend on where you picked the face to select it. A new, under defined, sketch appears in the FeatureManager design tree.
- 5 Edit the new sketch. Right-click the new sketch and select **Edit Sketch**.
- 6 Change the view orientation to **Normal To** the sketch.
- 7 The sketch has been copied with all the dimensions. What could not be copied were the relationships to things outside the sketch. In this case, that would be the midpoint relationships between the ends of the horizontal centerline and edges of the sketch plane.
- 8 Click Add Relation on the Sketch toolbar. Select the endpoint of the centerline and the edge shown. Add a Midpoint relationship.









Notice that the Sketch is not fully defined. To determine what's wrong, try to drag the right end of the horizontal centerline.



- 9 Because the horizontal centerline in the original sketch went from midpoint to midpoint, it never had a horizontal relationship, so there was none to copy. To fix this, add a Midpoint relationship between the right endpoint of the centerline and the edge shown. The sketch will now be fully defined.
- **10** Create a **Through All** cut with this sketch. We now have holes for both bindings.
- **11 Save** the part.

Task 6 — Add The Truck Mounting Holes

The Truck mounts to the underside of the Deck by way of four bolts. There are also four additional holes used to position and adjust the suspension springs and dampers.

The Truck assembly will look like the model at right when completed.

All holes in the Deck will be through hotels.





- 1 Select the face shown.
- 2 Reorient the view to **Normal To** and zoom in.
- 3 Start the Hole Wizard by clickingin the Features toolbar.

- **4** Select the following settings:
 - Type Hole
 - Standard Ansi Metric
 - Screw Type Screw Clearances
 - Size M6
 - End Condition Through All

Click the **Positions** tab.



5 One Sketch Point will appear on the surface at the point where you selected the surface. The Sketch Point tool will be active in order to allow us to place additional holes.

Click again on the surface to create a second **Sketch Point**.

6 Clear the **Point** \ast tool.



The America Hole Specification

7 We need a total of four holes of this size that will be symmetric about the centerline of the deck.

Draw a centerline from the midpoint of the edge shown.

8 Mirror the two sketch points.

Clear the **Centerline** i tool. Press and hold **Control** and select the two points and the centerline.

Click **Mirror Entities** (1) to mirror the two points.

Dimension the points as shown.
 Add a Horizontal relationship between two points as shown.





10 Click **OK** in the **Hole Wizard** dialog. These are the four holes that will be used to mount the Truck.


- **11** Repeat the above procedure to create four additional holes for the spring adjustment.
- **12** Use the following settings and the dimensions shown at right. All holes line up vertically.
 - Type Hole
 - Standard Ansi Metric
 - Screw Type Screw Clearances
 - Size **M8**
 - End Condition- Through All

13 The finished pattern should look like this.



000

once.

14 Mirror both hole

patterns to the other

end of the Deck.

Mirror both hole

patterns around the

Right plane using the

Mirror command only

15 Save the part.

5 Minute Assessment — #5

- **1** What is a thin feature?
- **2** How do you lock a dimension orientation so that it remains horizontal, vertical or aligned.
- **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?
- 4 True or False: The Mirror command can only mirror a single feature at a time.

Active Learning Experience, Part 2 — Initial Analysis

As the design progresses, each component must be checked to insure that it is strong enough to handle the loads applied to it. In order to determine the internal elements of the part's strength, stress and strain, we must first understand the external loads applied to the part.

Mechanics

Mechanics is defined as the science which describes and predicts the condition of rest or motion of bodies under the action of forces.

For the Mountainboard, we will first look at the forces acting when the board is not moving or moving at a constant velocity. This would be the case of a rider standing on the board while it was either not moving or moving at a constant speed.

Newton's Laws

Newton (1642-1727) formulated the fundamental principles of mechanics in three laws.

First Law

If the resultant force acting on a particle is zero, the particle will remain at rest (if originally at rest) or will move with constant speed in a straight line (if originally in motion).

What this says is that for a body at rest or in constant motion, the sum of the forces in any direction must add up to zero. Also, the moments about any point must also be zero.

Second Law

If the resultant force acting on a particle is not zero, the particle will have an acceleration proportional to the magnitude of the resultant and in the direction of the resultant force.

If you push on a body, like the mountainboard, it will continue to move faster and faster. If there was no friction or wind resistance, it would just keep accelerating. In the real world, as the mountainboard picks up speed, friction and wind resistance increase until they equal the force you are pushing with. At this point we go back to the first law where the sum of the forces are equal and the mountainboard continues at a constant speed.

Third Law

The forces of action and reaction between bodies in contact have the same magnitude, same line of action and opposite sense.

Free Body Diagrams

The Free Body Diagram isolates the part and shows all the external loads applied to it. For the Deck, we would have F_1 and F_2 which would be the weight of the rider transmitted to the Deck through the rider's feet. F_3 and F_4 would be the reaction of the ground through the wheels, axles and trucks.



The Third Law says that the ground must push back with a force equal to the riders weight.

For a 75 kilogram rider standing on the board, $F_1 + F_2 = 75$ kg. If the rider's weight is evenly distributed, then $F_1 = F_2 = 37.5$ kg. Because the Deck is symmetrical and the rider's weight is applied symmetrically it should be obvious that $F_3 = F_4 = 37.5$ kg.



How do the results change when the rider's weight is not symmetrical? If the rider puts all his weight on one foot then $F_1 = 75$ kg and $F_2 = 0$. How do we determine F_3 and F_4 since they are no longer equal?

The Second Law says that the sum of the forces must be zero if the mountainboard is at rest.

If we consider the Y direction (up) to be the positive direction:

Sum of the forces in the Y direction = $\Sigma \mathbf{F_v} = \mathbf{F_3} + \mathbf{F_4} - \mathbf{F_1} - \mathbf{F_2} = 0$

 $F_3 + F_4 - 75 - 0 = 0$

 $F_3 + F_4 = 75$

Moments

Moments are the product of a force acting at a distance.

By summing the forces we insured that there is no translation of the Deck. Moments must also be equal to zero to insure that there is no rotation.

To determine the values of F_3 and F_4 we can sum the moments about any point. Moments are the rotational forces calculated by multiplying the force by the distance from the point we are calculating.

The moments can be calculated about any point on the model. To make the calculations easier, we will calculate the moments about the point where F_3 acts on the Deck. Since F_3 acts at this point, the distance is zero and it creates no moment. The distances in the following diagram are in millimeters, but we will calculate moments using meters so that the result will be in kg-m. Sum the moments about F_3 with counterclockwise as positive:



Sum of the moments about $\mathbf{F_3} = \Sigma M_{F3} = (-\mathbf{F_1} * .215) + (-\mathbf{F_2} * .640) + (\mathbf{F_4} * .860) = 0$ (-75 * .215) + (0 * .640) + ($\mathbf{F_4} * .860$) = 0 -16.125 + 0 +($\mathbf{F_4} * .860$) = 0 $\mathbf{F_4} = 16.125 / .860 = 18.75 \text{ kg}$

Therefore: $F_3 = 75 - F_4 = 75 - 18.75 = 56.25$ kg



Stress

Stress is calculated as the force per area. During the design process we will need to make sure the different components are strong enough so that they will not fail while in use. The internal force must counteract the external forces to keep the part from moving.

We can take a section through the model at any point and calculate the forces and moments that exist at that point.

If we cut the Deck in half and just look at the left half, there are two external forces F_1 and F_3 . These must be resisted by a resultant internal force and moment F_R and M_R .



At this point in the model, $F_{R} = F_{1} - F_{3} = 75 - 56.25 = 18.75$ kg.

If we consider counterclockwise rotation to be positive and clockwise rotation as negative:

 $M_{R} = (F_{1} * .215) - (F_{3} * .430) = (75 * .215) - (57.35 * .430) = -8.54$ kg-m.

The negative value means that the moment acts in a clockwise direction.

Finite Element Analysis

The basic concept behind Finite Element Analysis or FEA is to continue to divide the model into smaller elements so that we can determine the stress throughout the model. In addition to stress, FEA can determine a variety of other quantities such as deformation and strain.

Mesh

The process of dividing the model into smaller finite elements is called meshing. The meshing process usually uses tetrahedrons as the mesh elements.



When meshed without the holes, the Deck looks like this:



Active Learning Experience, Part 3 — Configurations

Configurations

Configurations allow you to represent more than one version of the part in the same file. Configurations are used with parts and assemblies to hide or suppress features or components. You can also have different values for dimensions in each configuration. You can set up different schemes or representations and name them for quick and easy retrieval. A part or assembly can have multiple configurations.

Task 1— Create A Configuration

To simplify the initial analysis process we will make a configuration of the Deck that has no holes and only has half of the model. Because the model is symmetrical and the initial loads are symmetrical, there is no need to use up processing time doing both halves of the Deck.

1 Open the ConfigurationManager by clicking the

tab at the top of the FeatureManager design tree.

There is only a Default configuration until we add additional configurations.

2 Right-click the top level icon S Deck Configuration(s) and select Add Configuration.



-© Add Configuration

- **3** Type the following into the **Add Configuration** dialog box.
 - Configuration Name: **FEA**
 - Description: Configuration for Finite Element Analysis
 - Comment: Suppressed holes and chamfers. Cut part in half.

Click OK.

- 4 The new configuration is added to the ConfigurationManager and is active.
- 5 Click the tab for the FeatureManager design tree $\overline{\mathbf{w}}$

Task 2 — Suppress The Chamfer and Holes





The chamfer around the edge of the Deck does not affect

the strength of the Deck. It is only there to reduce the sharp edge to avoid injury to the rider and to reduce damage to the edge during impact. The small surface the chamfer creates can cause the mesh elements to become very small and therefore cause the mesher to create many more elements than would otherwise be necessary for a good analysis. We will suppress the chamfer on the configuration used to do the analysis.

All the holes in the Deck reduce its strength. We will first do an analysis of the Deck without the holes, then a second analysis with the hole to see the difference.

Finally, we will cut the Deck in half to analyze only one side. We can do this with parts that are symmetrical in geometry and symmetric in loading.

Suppress/Unsuppressed Features

Suppress is used to temporarily remove a feature. When a feature is suppressed, the system treats it as if it doesn't exist. That means other features that are dependent on it will be suppressed also. In addition, suppressed features are removed from memory, freeing up system resources. Suppressed features can be unsuppressed at any time.

To suppress a feature:

- □ Select the feature, then click **Edit**, **Suppress**, and choose a scope from the menu
- □ Right-click the feature, and select **Suppress**
- \Box Select the feature, and click **Suppress** $[\square]$ on the **Context** toolbar.
- □ Click **Suppressed** in the Feature Properties dialog box.
- 6 Right-click the Chamfer1 feature in the FeatureManager design tree and select **Suppress** from the **Context** toolbar.
- 7 The Chamfer feature will turn grey 🖉 Chamfer1 to show that it is suppressed, and the Chamfer will disappear from the model.

- 8 More than one feature can be suppressed at the same time. Press and hold **Control** and select the features Binding Holes and Cut-Extrude2. Click **Edit**, **Suppress**, **This Configuration** from the menu.
- 9 When a parent feature is suppressed, the child features must also be suppressed. Press and hold Control and select both the M6 and M8 Clearance Holes. Click in the Context toolbar.

Mirror2 will also be suppressed because it is a child of the clearance holes.

Task 3 — Cut The Deck In Half

- 1 Open a sketch on the Top plane.
- **2** Change the Top view.
- **3** Sketch a rectangle as shown making the right side coincident with the Origin.
- 4 Add two **Collinear** and one **Tangent** relationship to fully define the sketch.

5 Extrude a cut, **Through All**. This cuts away half of the Deck.



Task 4 — Add Split Lines

When we do the analysis of the Deck, will will place loads to simulate the rider's foot and the reaction of the truck. These loads must be applied to existing faces. The faces that currently exist are not the correct size or shape to represent these loads. To solve this we will split some of the existing faces into multiple faces.

Split Lines

Split lines are used to divide model faces into two. Split lines are created like any other sketched feature. They can be one or more connected sketch entities. They must be oriented so that they will pass through model faces when projected normal to the sketch plane.

To insert a split line:

- □ Click Insert Curve, Split Line
- □ Click **Split Line i** on the Curves toolbar.
- 1 Create a sketch on the face shown.

We want to sketch a rectangle at an angle to the centerline of the deck to more closely represent the rider's foot pressure.



Parallelogram Tool

The **Parallelogram** tool is used to sketch four sided closed shapes that can be rectangles or parallelograms.

Both shapes have opposite sides parallel. The difference between the two is the angle between the sides. Rectangles can only have 90° corners. Parallelograms can be any angle less than 180° .

Why can't we use the Rectangle tool? When sketching with the **Rectangle** tool, the sides are constrained to be either vertical or horizontal, there are no other options.



4 Release the mouse button, then click again and drag sideways to sketch the adjacent side.



5 Dimension the sketch as shown. It should now be fully defined.



- 6 Click **Insert, Curve, Split Line** from the menus.
- **7** Select the sketch face.
- 8 **Projection** should be selected by default. This will project the sketch onto the selected face.



9 Click OK.

The parallelogram area is now a separate face. We will apply the rider's foot pressure to this area during analysis.

- **10** Create another split face where the Truck will attach to the Deck.
- 11 Save the part.



Task 5 — Review the Configurations

- 1 Click the 😰 tab to change to the ConfigurationManager.
- 2 Double-click the Default configuration.

The entire Deck with holes and chamfers will appear.



3 Change to the FeatureManager design tree.

Only Cut-Extrude3 and the two split lines are suppressed. Cut-Extrude3 is the feature that cuts the Deck in half.



4 Click the 🖺 tab to change to the ConfigurationManager, then double-click the FEA configuration.

The Deck will appear without the holes, chamfer and left half.

- 5 Change to the FeatureManager design tree.
- 6 The holes and chamfer will be suppressed and Cut-Extrude3 and the split lines will be unsuppressed.



Task 6 — Add A Material and Appearance

The next task is to add a material and appearance to the model. The material will be used for weight calculations, the Bill of Materials that will be added to the drawings and to do the stress analysis. In the FeatureManager design tree, right-click **Material** and select **Edit Material**.

- 1 Click the plus sign next to Plastics to expand the list.
- 2 Select Acrylic (Medium-high impact).

3 The material definition includes not only the physical properties of the material, but also the visual properties.

For a simple strength analysis we need the Elastic Modulus, Poissons Ratio, Tensile Strength and Yield Strength.

ABS ABS PC Acrylic (1tedium-high impact)) CA Delrin 2700 NC010, Low Viscosity Acetal C Epoxy, Unfilled Epoxy, Unfilled EPDM Material properties Melamine resin Nylon 101 Nylon 101 Nylon 610 PA Type 6 PBT General Purpose PC High Viscosity PE High Density PE High Density PE HolyNedium Density Perspex (TM) GS Acrylic Cast Sheet PF Polybutadiene (PB) PBTP PDI Polybutadiene (PB) PEI Polybethrer Resin Polybethrer Resin Polybethrer Resin Polybethrer Kestone (PEEX) Polybethrer Kestone (PEEX) Polybethrer Merkone (PEEX) Polybethrer Merkone (PEX) Polybe	astics	Properties A	ppearance Cro	ssHatch Custor	m Application Data	Favorites
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→ Epoxy, Unfilled → Epoxy, Unfilled → Epoxy → PDM → Melamine resin Nylon 101 Nylon 6/10 → PA Type 6 → PPBT General Purpose → PC High Viscosity → PE Low/Medium Density → PE Low/Medium Density → Perspex (TM) GS Acrylic Cast Sheet → PF → Polybutadiene (PB) → PBTP → Polybutadiene (PB) → PEI → Polybester Resin → Polybether Polyol → Polybetheretherketone (PEEK) → Polybutheretherketone (PEEK) → Polybethylene Cross-Linked	CA	Model Typ	be: Linear	r Elastic Isotropic	-	
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- 4 Click **Apply** and **Close** to accept this material.
- **5** When we applied the material Acrylic (Medium-high impact), it had an appearance associated with it. This appearance is very transparent and will make doing the analysis difficult, so we will add a different appearance that will make the part easier to see.

Click the **Appearances**, **Scenes**, **and Decals** tab in the Task Pane. Expand the Appearances, Plastic, High Gloss folder, then double-click **white high gloss plastic**. This will apply the appearance to the entire part.



Default appearance

Acrylic (Medium-high impact)

Lighting

When viewing a model in either OpenGL or RealView, the model is lit by a combination of lights that are assigned with the scene. Earlier, we applied the Plain White scene which has three directional lights. This combination of lights may make the model too bright with the white plastic appearance, and therefore difficult to see.

As each scene can have different lights assigned, changing scenes will change the lighting. Individual lights can be turned on or off, repositioned and have their intensity changed through the individual light properties.

Lights assigned to the model are found in the DisplayManager under **View Scene, Lights and Cameras**. To edit the properties of a light:

- Double-click the light
- □ Or, right-click the light and select Edit [Directional, Spot, Point] Light.

To turn a light on or off, right-click the light and click **On (Off) in SolidWorks**.

- 6 Turn off two light. Select the DisplayManager tab and then click View Scene, Lights and Cameras. Right-click light Directional2 and then click Off in SolidWorks.
- 7 Repeat for light Directional3.
- 8 Adjust the brightness of the remaining directional light. Doubleclick light Directional1. Adjust the **Brightness** to 0.6 or any value that makes it easy for you to see the model.







Task 7 — Do A Simple Analysis

Using SolidWorks SimulationXpress, conduct a simple analysis of the Deck.

SolidWorks SimulationXpress

SolidWorks SimulationXpress is a first pass stress analysis tool for SolidWorks users. It helps you judge whether you part will withstand the loading it will receive under real-world conditions.

SolidWorks SimulationXpress is a subset of the SolidWorks Simulation product.

SolidWorks SimulationXpress uses a wizard to provide an easy to use, step-by-step method of performing design analysis. The wizard requires several pieces of information in order to analyze the part: *materials*, *fixtures* and *loads*. This information represents the part as it is used.

1 Click **Tools**, **SimulationXpress** to open the wizard in the Task Pane.

The wizard has several numbered steps showing the process we will take to do the analysis. As each step is completed, a green check will appear to the right of the step indicating that the action for that step is complete.

Note: There is a link to additional training on SolidWorks SimulationXpress Welcome page.

🚮 🚮 🗁 🚅 🔮 🜱 🐋 SolidWorks SimulationXpress *9* 🗩 × Welcome to SolidWorks SimulationXpress SimulationXpress helps you predict how a part will perform under load and helps you detect potential problems early in the design cycle. In SimulationXpress, you apply loads and fixtures to your part, specify its material, analyze the part, and view the results. All of this information is included in the Simulation study Note: Most analysis problems require a comprehensive analysis product for more accurate and complete real-world simulations before final sign-off on a design. Click here for your free online training on SolidW fundamentals. Options Next

Options

The **Options** dialog contains settings for the System of units and Results location.

2 Click Options. Set the units to English (IPS) and select Show annotation for maximum and minimum in the stress plots.

SimulationXpress Op	tions
System of units:	English (IPS)
Results location:	C:\SolidWorks Curriculum_and_Courseware_2
Show annotation	for maximum and minimum in the result plots
	OK Cancel

- 3 Set the path to the **Results location** to C:\SolidWorks Curriculum and Courseware_2011\ Mountainboard Design Project \Lessons\Lesson05.
- 4 Click **OK** and then **Next**.

Procedure

The introductory screen introduces the steps in the wizard. As we work through the wizard we will do the six steps shown to add fixtures, loads and material. We then run the analysis, view the results and them possibly optimize the part.

This procedure creates a SimulationXpress Study which creates a tab at the bottom of the graphics window. The details of the study are shown in the lower pane of the FeatureManager design tree. As we progress through the study, the details will be added to the Simulation study tree.



Fixtures

Fixtures are used to "fix" faces of the model that

should not move during the analysis. You must restrain at least one face of the part to avoid failure due to rigid body motion.

5 Click Add a fixture.

7 Click 🗹 .

6 Click the face shown. Because we are analyzing only half of the model due to symmetry, we will fix the face that would connect the two halves.





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SolidWorks SimulationXpress	9 🗩
	×
1 Fixtures	
2 Loads	
3 Material	
5 Results	
6 Optimize	
Apply fixtures to keep the part from moving when loads are appl Warning: Faces with fixtures are treated as perfectly rigid. This cause unrealistic results in the vicinity of the fixture. Examples: <u>Fixed Holes</u> <u>Fixed vs. Supported</u> <u>Fixed vs. Attached Parts</u>	ied. can
Note: More flexible fixture types are available in Solidworks Simulation Professional.	
Add a fixture Back Start	over

8 Click **Next** in the wizard.

The restraint is added as Fixed-1.

We could have added additional restrains if they were necessary for the analysis. In this case, they are not necessary.



Note that the green check mark has appeared next to Fixtures in the wizard.

	SolidWorks SimulationXpress	ØE
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Loads

Loads is used to add external forces and pressures to faces of the part. **Force** implies a total force, for example **200lbf** applied to the face in a specific direction. **Pressure** implies that the force is evenly distributed on the face, for example, **300psi**, and is applied normal to the face.

- 9 Click Add a force.
- **10** Select the split face representing the rider's foot



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Туре

11 Select **Selected direction**, then select the Top plane in the FeatureManager design tree.

We must apply the force normal to the Top plane as it represents the weight of the rider, so it must be in the direction of gravity.

12 Select **N** from the pull down list and type **367.7** for the value of the force

Note: We had to input the force in Newtons. One kilogram force is the equivalent of 9.807 Newtons.



13 Examine the model. Make sure the arrows are pointing toward the board. If they point away from the board, select **Reverse direction**.



KimulationXpress Study (-FEA-)

∏ Fixtures

🛛 🧭 Fixed-1 🛃 External Loads

[Deck (-[SW]Acrylic (Medium-high impact)-)

🛨 Force-1 (:Per item: -367.7 N:)

14 Click ✓. The load representing the weight of the rider has been added as Force-1.

🗙 🖃

Face<1>

Normal
 Selected direction

Top Plane

V N

Reverse direction

Per item

367.7

Туре

15 Click Add a force in the wizard. Repeat the above procedure to add a 367.7N force acting on the circular split face representing the reaction force at the Truck. Make this force normal to the Top plane.



16 After applying the second force, click **Next**.

Material

The next phase is selecting the Material. You can choose from libraries of standard materials or add your own.

- 17 Because we had applied a material to this part in SolidWorks, the material will already be selected in the wizard. Since we have a material applied, there is already a green check mark on the Material tab.
- 18 Click Next.



Run

SolidWorks SimulationXpress prepares the model for analysis and then it calculates displacements, strains, and stresses.

We could change the mesh size by selecting Change settings. As this is our first analysis, we will use the default settings.

19 Click **Run Simulation** to begin the solution.

takes very little time.



Once the solution completes, the part will be ani am tat ed to show how it deforms.

- 21 Examine the animation. The face where the Deck was cut should remain stationary while the rest of the Deck deflects. This is about what we expected.
- 22 Click Stop animation.
- 23 Click Yes, continue for Does the part deform as you expected.

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Results

Once the simulation has been run, four results are provided.

- □ Stress
- Displacement
- Deformation
- □ Factor of Safety

The results are also displayed in the Simulation study tree. To view the results, either doubleclick the result in the Results folder or select it in the SimulationXpress wizard.



SolidWor	ks SimulationXpress	Ø ₽
		×
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2 Loads		Image: A start of the start
3 Material		V
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5 Results		\checkmark
6 Optimize		
Results 🔁 Show von Mises str	ess	
Show displacemen	t	
Show where factor	of safety (FOS) is below:	1
Based on the specified ((FOS) found in your desi	parameters, the lowest fa gn is 2.71736	ictor of safety
Use these controls to vie	ew the animation.	
🚺 Play animation		
🔲 Stop animation	1	(
Done viewing result	ts	
Back	- Contraction	Start Over

Factor of Safety

SolidWorks SimulationXpress uses the maximum Von Mises stress criterion to calculate the factor of safety distribution. This criterion states that a ductile material starts to yield when the equivalent stress (von Mises stress) reaches the yield strength of the material. The yield strength (SIGYLD) is defined as a material property. SolidWorks SimulationXpress calculates the factor of safety at a point by dividing the yield strength by the equivalent stress at that point.

At any location, a factor of safety that is:

- □ Less than 1.0 indicates that the material at that location has yielded and the design is not safe.
- **Equal to 1.0** indicates that the material at that location is at the yield point.
- Greater than 1.0 indicates that the material at that location has not yielded.

Select Show where factor of safety (FOS) is below: 1 in the wizard.

2 The initial analysis shows a Factor of Safety of **2.72** as shown in the callout. This indicates that for the deck is nearly 3 times stronger than it needed to carry the applied load.

Does this mean that we can reduce the strength of the deck by making it thinner or



changing material? No, not yet. Remember that this was just a preliminary analysis for the very simple loading condition of a rider standing on the deck with weight equally distributed. Before making design changes, we need to analyze the deck under the extreme loads we expect the deck to encounter.

3 There are other ways to look at the results: stress and deformation.

The following are some examples of the different ways to display the results. To access the other plots, either select them in the wizard or double-click the result in the Results folder of the SimulationXpress study tree. The Stress Distribution and Deformed Shape graphics can be animated and saved as *.avi files.

□ Stress Distribution

The stress distribution shows a maximum stress of 2,402 psi.



Scientific Notation

Scientific Notion is used to represent both large and small numbers more easily. With Scientific Notion, numbers are always displayed with one digit to the left of the decimal point and the remaining digits to the right of the decimal point.

The number to the left of the decimal is called the coefficient and must be equal to or greater than 1 and less than 10.

The coefficient is followed by the digits to the level of accuracy the number represents. They are followed by the power of ten to which the number must be multiplied. The power of ten is represented by "e" which stands for exponent. You can think of this as the number of places you must move the decimal point such that e+002 means one with the decimal point moved two places to the right (100), e+006 = 1,000,000. The same is true for negative exponents, except that you move the decimal point to the left. So, e-002 = .01 and e-006 = .000001, etc.

Engineering Notation

In some applications, Engineering Notation is used instead of Scientific Notation. Engineering notation is similar to Scientific Notation except that powers of ten are limited to multiples of three. Some examples:

Number	Scientific Notation	Engineering Notation
1,234	1.234e+003	1.234e+003
12,345	1.2345e+004	12.345e+003
123,456	1.23456e+005	123.456e+003
1,234,567	1.234567e+006	1.234567e+006

□ Deformation

This is best seen by viewing the animation.



Generate Report

Generate Report creates a report can be viewed in Microsoft Word.



D eDrawing[®] of analysis results.

eDrawings will be covered in more detail, in a following lesson.



3 Close and SolidWorks SimulationXpress wizard and save the results.

Updating the Model

Changes performed in SolidWorks are detected by SolidWorks SimulationXpress. Changes can be made to the model, materials, restraints or loads. The existing analysis can be **Updated** to show the newest results.

Task 8 — Change the Model and Re-run the Analysis

For simplicity, we suppressed the holes in the Deck for the first analysis. The holes however are stress risers and should be included in the analysis. We will unsuppressed the holes and re-run the analysis.

1 Make the FEA configuration active if it is not already.

Click the ConfigurationManager tab on the FeatureManager design tree. Then doubleclick the FEA configuration to make it active.

- 2 In the FeatureManager design tree select:
 - Binding Holes
 - Cut-Extrude2
 - M6 Clearance Hole1
 - M8 Clearance Hole1
 - Mirror2
 - **TIP:** To select more than one item in the FeatureManager design tree you can either hold down the Control key as you select multiple items or if the things you want to select are sequential you can select the first item, then hold down the Shift key and select the last item. These are both Windows techniques for selecting multiple items.
- 3 Right-click on any of the selected features and select Unsuppress.
- 4 Start SolidWorks SimulationXpress. Click **Tools**, **SimulationXpress** from the menu.
- 5 Because we save the results for the previous analysis, the wizard gives us a choice to either delete those results and start over, or to edit the existing study.

Click Next.



- 6 Several items; **Fixtures**, **Loads** and **Material** still have green check mars as these items have not changes.
- 7 The Run and Results items do not have green checks indicating that there have been changes that make the results invalid.
- 8 Click the **Run**, then **Run Simulation**. The simulation will run and update all of the results.
- 9 Stop the animation and double-click the **Factor of Safety** plot.

The holes have reduced the Factor of Safety from 2.72 to 1.55.

			×
1	Fixtures		v
2	Loads		1
3	Material		V
4	Run		
5	Results		
6	Optimize		
Wa	arning: Faces with fixtures are tr	eated as perfectly	rigid. This re. Examples
	n cause unrealistic results in the Fixed Holes Fixed vs. Supported Fixed vs. Attached Parts	e vicinity of the fixtu	
No	n cause unrealistic results in the <u>Fixed Holes</u> <u>Fixed vs. Supported</u> <u>Fixed vs. Attached Parts</u> ote: More flexible fixture types are mulation Professional.	available in Solid	Works
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	n cause unrealistic results in the Fixed Holes Fixed vs. Supported Fixed vs. Attached Parts ote: More flexible fixture types are mulation Professional. Add a fixture Edit an existing fixture Next	available in Solid	Works

10 Double-click the Stress results. The maximum stress in now4,200psi.



11 Zoom in to the four Binding Holes where the maximum stress tag is pointing.

We can see that there is more stress across the board than along its axis.



Question: Does this seem reasonable?

<u>Answer:</u> Yes. We applied the weight of the rider across most of the width of the Deck. The Deck is supported very close to its centerline by the two Trucks which makes the rider's weight try to bend the Deck from side to side. This tries to elongate the hole in a direction across the Deck.

- 12 Click **Close** then **Yes** to save the SolidWorks SimulationXpress results.
- **13** Change to the Default configuration of the Deck.
- **14** Save and Close the model.

Exercise 16: Thin Bracket

Create the part shown.

The exercise reinforces:

- □ Thin features
- □ Mirror features
- □ Apply material to a part.
- 1 Create a new part using the Part_MM template.
- **2** Create a sketch on the front plane. The sketch is symmetrical.

3 Extrude the sketch as a thin feature.

Extrusion depth is 150mm, using Mid Plane.





4 Add fillets to the extrusion.

sketch.

Right-click the base feature Extrude-Thin1 and select Edit Feature.

Select Auto-fillet corners and set the radius to 5mm.

20

5 Click **Detailed Preview** for the see the effect of the fillet.Click **OK** to finish the edit.

- 6 Use the Hole Wizard to add a M12 Clearance Hole.
- 7 Mirror the hole around the Front plane.
- 8 Mirror the two holes about the Right plane.
- **9** Add the material Chrome Stainless Steel to the part.

10 If your computer supports RealView

graphics, click \bigcirc to turn on the RealView display.

11 Check the mass of the part. If you have done the previous steps correctly, the part should weigh **2.229 kilograms**.



Lesson 5 Quiz

Name: _____ Class: ____ Date:____

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

- 1 How is the ConfigurationManager used in SolidWorks?
- 2 Can SolidWorks SimulationXpress be used to analyze parts where the sum of the forces do not add up to zero?
- **3** What is a Free Body Diagram?
- 4 Name an advantage to using the Hole Wizard as compared to creating a sketch and either extruding or revolving a cut.
- 5 What does it mean when the Factor of Safety is less than one?
- **6** How is the number 345,678 expressed in Engineering Notation?
- 7 How is the number 345,678 expressed in Scientific Notation?
- 8 What is the shape of the finite elements used by SolidWorks SimulationXpress?
- **9** True or False: When a feature is Suppressed, it is removed from memory and not calculated.
- 10 Name two things that can be controlled by configurations.

Lesson Summary

- □ Thin features are created from open profile sketches.
- □ The Hole Wizard is used to easily make holes that conform to the various engineering standards.
- □ Mirror can be used to mirror features across a plane or planar face.
- Different configurations can have different:
 - Suppressed features
 - Dimension values
- □ SolidWorks SimulationXpress will do a first pass stress analysis.
- □ Material applied to a part can be used by SolidWorks SimulationXpress.
- □ SolidWorks SimulationXpress will provide the following output:
 - Factor of Safety
 - Stress distribution
 - Deformation
 - HTML Report
 - eDrawing

□ You will be able to create and modify the following parts:



Before Beginning This Lesson

□ Complete the previous lesson: Thin Features — The Deck.

Resources for This Lesson

This lesson plan corresponds to *Multibody Parts* in the SolidWorks Online Tutorials. For more information about the Online Tutorials, See "Online Tutorials" on page 1.

Review of Lesson 5 — Thin Features

Questions for Discussion

- 1 Different part configurations can have different_____, ____?
- **2** Thin features can be created from:
 - a) Open sketch
 - b) Closed sketch
 - c) Either an opened or closed sketch.
- **3** True or False: SolidWorks SimulationXpress can be used for linear static analysis of parts.
- 4 How do you "lock in" a dimension orientation?
- **5** Where can you apply a material to a part so that it can be used in SolidWorks SimulationXpress?
- 6 Split lines are used to do what?
- 7 What is the only end condition available for a cut made with an open sketch?
Outline of Lesson 6

- In Class Discussion
 - Multibody solids
 - Boolean operations
 - Finite Element Analysis
- □ Active Learning Exercise, Part 1 The Axle
- □ Active Learning Exercise, Part 2 Stress Analysis
- □ Active Learning Exercise, Part 3— The Truck
- □ Exercises and Projects Multibody Parts
- □ Lesson Summary

In Class Discussion — Multibody Solids

□ What is a sold model?

A solid model is the most complete type of geometric model used in CAD systems. It contains all the wire frame and surface geometry necessary to fully describe the edges and faces of the model. In addition to the geometric information, it has the information called topology that relates the geometry together. An example of topology would be which faces (surfaces) meet at which edge (curve). This intelligence makes operations such a filleting as easy as selecting an edge and specifying a radius.

□ What is a multibody solid?

Multibody solids occur when there is more than one solid body in a part. In cases where discrete features are separated by a distance, this can be the most efficient method in designing a part

Multibody Solids

Multibody solids occur when there is more than one continuous solid in the same part file. Often times, multibody techniques are useful for designing parts that require specific distance separations of features. These bodies can be accessed and modified separately and later merged into a single solid.

Multibody solids are created in several ways. The following commands have the option of creating multiple solid bodies from a single feature:

- □ Extruded bosses and cuts (including thin features)
- □ Revolved bosses and cuts (including thin features)
- □ Swept bosses and cuts (including thin features)
- □ Lofted bosses and cuts
- □ Thickened cuts
- □ Cavities

The most direct way to create a multibody solid is by clearing the **Merge result** check box for specific boss and cut features.

□ Boolean operations.

There are three Boolean operations that can be used to combine multibody solids within SolidWorks. They are Add, Subtract and Common.

Active Learning Exercise, Part 1— The Axle

The axle is a symmetric part. Because of symmetry, only half of the part need be created feature by feature, as the mirror half can be created using the **Mirror** function.



Design Intent

- □ The axle serves as the connect between the Wheel Assemblies and the Truck
- **□** The part will be machined from aluminum stock
- □ The part will pivot about the King Pin that goes through the Truck.
- □ Mounting must be provided for the optional brake system.

Task 1— The Axle

We will create several of the features of the Axle by creating two individual solid bodies, then combining them into the final shape. Each body will represent the way the feature looks in one of the standard orthogonal views (Front, Top, Right).

- 1 Create a new part using the Part_MM template.
- 2 Save the new part with the name Axle.
- 3 Create the following sketch on the Front plane.



Note: The center of the 67mm arc is coincident to the left vertical line.

- 4 Extrude the sketch to a depth of **11mm**.
- 5 Create the following sketch on the Top plane.



- 7 Click Extrude 💽.
- 8 Extrude to a Blind depth of 46mm. Clear Merge result. Click OK.



🕞 Boss-Extrude 🕺	?
🖌 🗙 რი	
From	1
Sketch Plane 🔻]
Direction 1	1
🗾 Blind 🗸]
*	
Merge result	
\$	
Draft <u>o</u> utward	

9 Examine the results. The FeatureManager design tree shows two solid bodies. Because the results of the two extrusions were not merged, each remains separate.

Each solid body is named for the last feature that created it.

10 Look at the graphic. Notice that there is no edge where the two bodies intersect.



Combined Bodies

The **Combined Bodies** technique is used to create a single solid by adding, subtracting or intersecting the volumes of solid bodies. These are also known as Boolean operations.

The **Combine** tool is used to combine the volumes of multibody solids into a single solid body. The bodies can be combined in different ways using different operations.

The **Combine** tool has three options:

□ Add

The **Add** operation uses the **Bodies to Combine** list to merge the bodies into a single solid by adding all volumes. This operation is also known as a **union** in other systems.

□ Subtract

The **Subtract** operation uses the **Main Body** and **Bodies to Combine** list to merge the bodies into a single solid by subtracting the bodies to combine from the main body.

Common

The **Common** operation uses the **Bodies to Combine** list to merge the bodies into a single solid by finding the volume that is common to all. This operation is also known as an **intersection** in other systems.

- To **Combine** solid bodies:
- □ Click **Combine** 👘 on the Features toolbar
- □ Click Insert, Features, Combine
- 1 Click **Combine [a]** on the Features toolbar.
- 2 Select the two solid bodies.
- **3** Select **Common**. This will create a single solid body from the volume that is common to the two bodies.



4 Click **Show Preview**. The preview shows the common volume



5 There are four possible choice for combining the two solidbodies.



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- **9** Create the following sketch on the Front plane. Both arcs are of equal radius.
- **10 Extrude** the sketch to a depth of **50mm**. Clear Merge results.







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Extrude2. Next, the two combined volumes will be added together.

- 5

- 12 Click Insert, Features, Combine.
- 13 Select the solid bodies Extrude3 and Extrude4.
- 14 Select Common.
- 15 Click 🖌 .
- 16 Again, notice that there is no edge present where the two remaining solids pass through one another.
- 17 Click **Combine** [17
- 18 Select Add, then select the two solid bodies Combine1 and Combine2.
- 19 Click 🗹 .
- 20 With the two solids added together, the only changes noted are the new edge where the solids join and the Solid Bodies folder has been reduced to only a single body, Combine₃.



- **21** Create a new sketch on the end face shown. Sketch plane 22 Sketch a rectangle and relate it to the existing edges of the model with **Collinear** relationships. Collinear **23** Extrude the sketch to a depth of 70mm, into the model. Select Merge results. 24 Create a 20mm fillet to fill in the area behind the last extrusion. Radius: 20mm **25** Add an **11mm** fillet to the edge shown. Radius: 11mm
- Mountainboard Design Project with SolidWorks

- **26** Create a sketch on the face shown.
- 27 Reorient the model to the Front view.



28 Create the sketch as shown.

Foreshorten Radius

Large radius values can create dimensions that extend off the screen. In a drawing they may extend past the viewing area.

Radius dimensions can be foreshortened through the dimension's properties.



- Select the 95mm radius dimension.
 Select the Leader tab in the PropertyManager.
- 2 Select Foreshorten radius, then click \checkmark .



🚰 Fillet / X

Manual

FilletXpert

- **3** The radius dimension is now displayed with a broken leader
- 4 **Extrude** the sketch *into* the model 6mm.

Full Round Fillets

The Full Round Fillet option creates a fillet that is tangent to three adjacent sets of faces. Each face set can contain more than one face. However, within each face set, the faces must be tangent continuously.



A Full Round Fillet does not

need a radius value. The radius is determined by the shape of the faces you select.

- 1 Select **Fillet (?)** on the Features toolbar.
- 2 Select Full round fillet.
- 3 Select Face 1.
- 4 In the PropertyManager click in the middle box to make it active (light blue background), then select Face 2.
- 5 Make the bottom box active and select Face 3.



- **Note:** Each face will be color coded corresponding to the colors in the PropertyManager:
 - Face <1> Blue
 - Face <2> (Center Face) Indigo
 - Face <3> Magenta

6 Click 🗹 .

Task 2 — Trim the Edge

The last extruded feature we created is too long and needs to be trimmed back to the existing edge of the previous extrusion.

- 1 Add a **5mm** fillet to the edge shown.
- 2 Create a sketch on the Top plane.
- **3** Reorient the model to the Top View.
- 4 Change the display to Hidden Lines Removed by clicking on the View toolbar.
- 5 Click **Convert Entities**

on the **Sketch** toolbar.

- 6 Select the five edges shown.
- 7 Click 🖌 .
- B Drag the endpoint of the line shown to shorten it. The exact length is not important, we are just leaving enough room to draw the next arc.





Note: Even though the endpoint is black, indicating fully defined, it can be moved.

9 Sketch a Tangent Arc from the end of the line we just shortened to the existing arc and add a Tangent relationship to insure it is tangent at both ends.







10 Extrude a cut, Through All. If necessary, select Flip side to cut so that we are left with the part as shown.

By using the existing edges and converting them into the sketch, we have insured that this cut will be correct even if the four extruded features later change size.



Task 3 — Filleting The Edges

There are many edges that need to be filleted. Most can be done as constant radius fillets, however some will need to be created with other filleting methods.



- □ The system defaults to five control points, one at each end of the edge and the other three located at equidistant increments of 25%, 50%, and 75% along the edge between the vertices. You can increase or decrease the number of control points.
- □ You can change the position of any control point by changing the percentage assigned to that control point. You can also drag any control point, and its assigned percentage will update accordingly.

Fillet

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Fillet Type

Manual

Constant radius
Variable radius

FilletXpert

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- □ Although there is a visual display of the control points, they are only active if you select them and assign a radius value.
- □ Inactive control points are red. Active control points are black, and have a callout attached to them indicating the assigned radius and percentage values.
- 2 Click the **Fillet** tool.
- **3** Select Variable Radius for Fillet Type.

- 4 Select the four edges shown.
- Callouts will appear at each end of the individual line segments. The last line segment selected will also show the intermediate points.
- 6 Individual radius values can be entered using the callouts.
- Double-click each callout and enter the radius values shown. There are only two radius values, 5mm and 1mm.
- 8 In the **Items to Fillet** box, select each edge in turn. Notice that the intermediate points will appear on the edge in the graphics area. Select the





longest edge. If you picked edges in order from right to left, it will be Edge3. If you picked from left to right it will be Edge2.

9 In the graphics area, select the middle intermediate point (50%). This will make the callout visible. Double-click the callout and enter 1mm for the radius.



10 Click 🗹 .

We now have a radius that blends smoothly from 5mm at one end to 1mm at the other end.

11 Add another Variable Radius fillet. There are only two radius values, 4mm and 5mm.



12 Add a constant radius fillet of **5 mm** to the edge shown.

Task 4 — Create The Connections

The final steps are to create the connections to the other components and the second half of the model.

- 1 Create a sketch on the end face shown.
- 2 Sketch a circle with a diameter of20mm and make it Concentric to the fillet edge shown.
- 3 Extrude a boss to a blind depth of 3mm.
- 4 Select the end face of the boss and click
 Hole Wizard in on the Features toolbar.



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Hole Type

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Type:

Being Edited

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Section Dimensions Value

13mm

67mm

118deg

End Condition

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Q13

Dimension

Diameter

Drill Angle

Depth

The Specification

Type Positions

- 5 Select the Legacy Hole button.
- 6 From the Hole type list, select **Simple Drilled**.
- 7 Under Section Dimensions, double-click the value for Diameter and type 13mm.
- 8 Double-click the value for **Depth** and type **67mm**.
- **9** Switch to the **Positions** tab.
- 10 Deselect the **Point** tool by clicking ***** in the **Sketch** toolbar.

- 11 Add a **Concentric** relationship between the point and the edge of the boss.
- 12 Click 🗹 .
- 13 Add 1mm fillets to the edges of the boss.





Task 5 — Add Tapped Hole

The wheel axle will be inserted into the hole we just created. To keep the axle in place, we will use a set screw. The set screw will be threaded into a tapped hole and tightened until it keeps the axle shaft from rotating or pulling out of the hole.

1 Select the face shown.



- 2 Click **Hole Wizard** on the Features toolbar.
- 3 Click the **Straight Tap** Button.
- 4 Make the following selections:
 - Standard: Ansi Metric
 - Screw type: Tapped hole
 - Size: M6x1.0
 - End Condition: Up To Next
 - Cosmetic thread without thread callout
- **5** Click the **Positions** tab.
- 6 Clear the **Point** [*] tool.
- 7 Add a **Coincident** relationship between the point and the edge shown.
- 8 Dimension the point **28mm** from the end face.
- 9 Click 🗹 .





Task 6 — Add Countersunk Holes

In the final assembly, the spring and dampener assembly will be mounted between the Truck and the Axle. There are two springs and two possible positions for each.

- 1 Reorient the model to the Bottom view.
- 2 Select the bottom face and click Insert, Feature, Hole, Wizard.
- 3 Select the **Countersink** button.
- 4 Make the following selections:
 - Standard: Ansi Metric
 - Screw type: Flat Head Screw ANSI B18.6.7M
 - Size: **M5**
 - End Condition: Through All
- **5** Click the **Positions** tab.



- 6 There will be a preview of the first hole and the **Point** tool will be selected. Select the bottom face to place a second hole.
- 7 Turn off the **Point** tool.
- 8 Sketch a horizontal **Centerline** from the midpoint of the left edge.
- **9** Add **Coincident** relationships between the points and the centerline.
- **10** Draw a vertical centerline from the Origin.
- 11 Add the dimensions shown. Dimension from the points to the vertical centerline and move the dimension to the left of the centerline before placing it.







12 Click ✓. The two holes are correctly created an placed on the part.



Task 7 — Brake Mounting Pad

If the optional Brake Kit is installed, the Brake Arms are bolted to the Axle. These pads will be an integral part of the Axle.

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- 1 Reorient the model to the Back view.
- **2** Create a sketch on the back face.
- 3 Sketch a rectangle.
- 4 Dimension the sketch as shown.
- 5 Extrude the sketch to a Blind depth of 6mm.
- 6 Examine the model. Reorient the model to the Right view. There is a gap between the extrusion and the rest of the axle. We could have avoided this by creating the brake pad before the fillet. We can also fix this by extruding the brake pad in two directions.



7 Right-click the brake pad extrusion and select Edit Feature.

- 8 Select Direction 2.
- **9** For **Direction 2**, select **Up To Surface**. Select the surface of the fillet.



10 When extruding in both directions, each direction can have a different end condition. Choosing **Up To Surface** for **Direction 2** will fill the existing gap.



11 Click 🗹 . The gap is now filled.

Task 8 — Add A Tapped Mounting Hole

The brake mounting pad needs a single tapped mounting hole to attach the brake caliper.

- 1 Reorient the model to the Back view.
- 2 Select the face of the Brake Mounting pad and then click Hole Wizard in on the Features toolbar.



- **3** Select the **Tap** hole.
- 4 Make the following selections:
 - Standard: Ansi Metric
 - Screw type: **Tapped hole**
 - Size: M6x1.0
 - End Condition: Up To Next
 - Cosmetic thread without thread callout
- 5 Click on the **Positions** tab.
- 6 Turn off the **Point** tool as we only need one hole.



7 Dimension the **Point** as shown.





- 8 Click 🗹 .
- **9** Add **1mm** fillets all around.

Task 9 — Mirror The Body

To this point, we have only created half of the Axle. Using the **Mirror** command, we can create the other half of the Axle in one simple step.

1 Click Insert, Pattern/Mirror, Mirror.

Mirror face

2 Select the face shown. The is the face we will mirror the part about.

- 3 Make the **Bodies to Mirror** box active.
- 4 Select the part in the graphics ares.
- 5 Make sure that **Merge solids** is selected.
- 6 Click 🗹 .

Task 10 — Create King Pin Hole

The Axle will be connected to the Truck through a King Pin.

- 1 Create a sketch on the face shown.
- 2 Sketch a vertical centerline from the Origin.
- 3 Dimension the centerline to **32mm**.
- 4 Sketch a circle and create a **Coincident** relationship with the end of the centerline.
- 5 Dimension the circle to **12mm**.
- 6 Extrude a cut Through All.



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7 Save the part







Task 11 — Apply Material

- 1 Click Edit Material 🔚 on the Standard toolbar, or right-click Material in the FeatureManager design tree and select Edit Material.
- 2 Expand the Aluminum Alloys by clicking the plus sign next to the group.
- 3 Select 6061 Alloy.
- 4 Click **Apply** and **Close**.

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5052-H38, Rod (SS)		a custo	m library to e	dit it.					
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5 Check the weight of the part. Click Tools, Mass Properties.

The axle weighs 343.297 grams (0.757 pounds).

- 6 Click Close.
- 7 Save the part.

🚳 Mass Properties	
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Output coordinate system: default	•
Axel.SLDPRT Selected items:	
☑ Include hidden bodies/components	
Show output coordinate system in corner of window	
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Mass = 343.297 grams	
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5 Minute Assessment — #6-1

- 1 What are the three Boolean operations that can be done with multi-bodies?
- 2 What SolidWorks tool is used on multibody solids to do Boolean operations?
- **3** What determines the radius of a Full Round Fillet?
- **4** What type of fillet can be used to have the fillet radius change along the length of an edge?
- 5 What mirroring option is used to mirror half of a part to get the full part?

Active Learning Exercise, Part 2 — Stress Analysis

We did a rudimentary analysis of the Deck using SolidWorks SimulationXpress. While SolidWorks SimulationXpress gave us a quick solution, it was limited in its capability. When more advanced analysis is required we use SolidWorks Simulation.

Terms in analysis

When we analyze a part, there are several properties we will check.

Stress

Stress is the intensity of *internal* force. Generally speaking, this is the applied force divided by the area over which it applies.

Stress =
$$\frac{Force}{Area}$$

Applying a 10 kilogram force to a rod with a 1 square centimeter cross section would yield a stress of $10 \text{ kg/} \text{ cm}^2$.



Stress can be either tensile or shear and is a vector quantity having both a magnitude and direction.

Tensile Stress

Tensile stress, or compressive stress, can be thought of as pushing things together or pulling them apart. If the block at right was glued to the plate at the bottom and we applied a force to lift the block, we would put the glue joint in tension.

Shear Stress

Shear stress is like trying to slide a block on a surface. If we tried to pull the block to the side, the glue joint would be in shear.



Elongated Lenght

Initial Length

Elongation

Von Mises Stress

Von Mises Stress is a non-negative, scalar value. Von Mises stress is a commonly used stress measure because the structural safety of many engineering materials showing elastoplastic properties, such as steel, is well described by von Mises stress magnitude.

Strain

Strain is defined as the intensity of deformation. This is a measure of the change in length of material as a force is applied, measured in units of length per units of length such as inches per inch or centimeter per centimeter.

Strain = Elongation InitialLength

If the bar at right was 30cm long before the load was applied and 33cm after the load was applied:

Strain =
$$\frac{33-30}{30} = \frac{3}{30} = 0.1$$

Modulus of Elasticity or Young's Modulus

To determine the rigidity of material, plots are created between Stress and Strain. With SolidWorks Simulation, we analyze materials where the plot of Stress versus Strain is a straight line. In other words, there is a linear relationship between stress and strain.

The slope of the Stress-Strain curve is called the Modulus of Elasticity or Young's Modulus.



Poisson's Ratio

As material elongates due to an applied tensile force, its cross sectional area is reduced. Poisson's Ratio is the ratio of the strain in the axial direction to the strain in the cross section direction.



Displacement

Deformation is the actual movement of each element in the model.

What is SolidWorks Simulation?

SolidWorks Simulation is a design analysis tool based on a numerical technique called Finite Element Analysis or FEA. SolidWorks Simulation belongs to the family of engineering analysis software products developed by SRAC, now part of Dassault Syetèmes SolidWorks Corporation.

SolidWorks Simulation comes in different "bundles", or applications, designed to best suit the needs of different users. With the exception of SolidWorks SimulationXpress, which is an integral part of SolidWorks, all SolidWorks Simulation bundles are add-ins. A brief description of the capabilities of different bundles is as follows:

□ SolidWorks SimulationXpress

The static analysis of parts with simple types of loads and supports.

SolidWorks Simulation

The static analysis of parts and assemblies.

SolidWorks Simulation Professional

The static, thermal, buckling, frequency, drop test and optimization analysis of parts and assemblies.

□ SolidWorks Simulation Premium

All capabilities of SolidWorks Simulation Professional plus nonlinear analysis and fatigue; advanced dynamic analysis available in the GeoSTAR interface.

Before we proceed with the lesson, let us construct a foundation for our skills in SolidWorks Simulation by taking a closer look at what Finite Element Analysis is and how it works.

What Is Finite Element Analysis?

In mathematical terms, FEA, also known as the Finite Element Method, is a numerical technique of solving field problems described by a set of partial differential equations. Those types of problems are commonly found in many engineering disciplines, such as machine design, acoustics, electromagnetism, soil mechanics, fluid dynamics, and others. In mechanical engineering, FEA is widely used for solving structural, vibration, and thermal problems.

FEA is not the only tool available for numerical analysis. Other numerical methods used in engineering include the Finite Difference Method, Boundary Element Method, or Finite Volumes Method. However, due to its versatility and high numerical efficiency, FEA has come to dominate the software market for engineering analysis. Using FEA, we can analyze any shape, use various ways to idealize geometry and produce results with the desired accuracy. FEA theory, numerical problem formulation, and solution methods become completely transparent to users when using SolidWorks Simulation.

A powerful tool for engineering analysis, FEA is used to solve problems ranging from very simple to very complex. Design engineers use FEA during the product development process to analyze the design-in-progress. Time constraints and limited availability of product data call for many simplifications of the analysis models. At the other end of scale, specialized analysts implement FEA to solve very advanced problems, such as vehicle crash dynamics, metal forming, or analysis of biostructures.

Steps in the FEA process.

Regardless of the project complexity or the field of application, the fundamental steps in any FEA project are always the same. The starting point for any analysis is the geometric model. In our case, this is a SolidWorks model of a part or an assembly. To this model, we assign material properties, and define loads and restraints. Next, as always the case when using a tool based on the method of numerical approximations, we discretize the model intended for analysis.

The discretization process, better known as meshing, splits the geometry into relatively small and simply-shaped entities, called finite elements. The elements are called "finite" to emphasize the fact that they are not infinitesimally small, but only reasonably small in comparison to the overall model size.

When working with finite elements, the FEA solver approximates the wanted solution (for example, deformations or stresses) for the entire model with the assembly of simple solutions for individual elements.

From the perspective of FEA software, each application of FEA requires three steps:

□ Preprocessing

The type of analysis (e.g., static, thermal, frequency), material properties, loads and restraints are defined and the model is split into finite elements.

□ Solution

Computing the desired results.

□ Postprocessing

Analyzing the results.

We follow the preceding three steps every time we use SolidWorks Simulation.

From the perspective of FEA methodology, we list the following FEA steps:

- Building the mathematical model
- Building the finite element model
- Solving the finite element model
- Analyzing the results

Build Mathematical Model

Analysis with SolidWorks Simulation starts with the geometry represented by a SolidWorks model of a part or assembly. This geometry must be meshable into a correct and reasonably small, finite element mesh. By small, we do not refer to the element size, but the number of elements in the mesh. This requirement of meshability has very important implications. We must ensure that the CAD geometry indeed meshes and that the produced mesh provides the correct solution of the data of interest, such as displacements, stresses, temperature distribution, and so on.

Often, but not always, this necessity of meshing requires modifications to the CAD geometry. Such modifications can take the form of defeaturing, idealization, and/or clean-up, described as follows:

Defeaturing

Defeaturing refers to the process of suppressing or removing geometry features deemed insignificant for analysis, such as external fillets, rounds, logos, and so on.

Idealization

Idealization presents a more aggressive exercise that may depart from solid CAD geometry as, for example, when representing thin walls with surfaces.

Clean-up

Clean-up is sometimes required because the meshable geometry must satisfy much higher quality requirements than those commonly followed in Solid Modeling. For clean-up, we can use CAD quality-control tools to check for problems, like sliver faces or multiple entities, that the CAD model could tolerate, but would make meshing difficult or impossible.

It is important to mention that we do not always simplify the CAD model with the sole objective of making it meshable. Often, we simplify a model that would mesh correctly "as is", but the resulting mesh would be too large and, consequently, the analysis would run too slowly. Geometry modifications allow for a simpler mesh and shorter computing time. Successful meshing depends as much on the quality of the geometry submitted for meshing as on the sophistication of the meshing tools implemented in the FEA software. Having prepared a meshable geometry, we define material properties, loads, supports and restraints, and provide information on the type of analysis that we wish to perform.

This procedure completes the creation of a mathematical model. Note that the process of creating the mathematical model is not FEA-specific. FEA has not yet entered the picture.



Build Finite Element Model

We now split the mathematical model into finite elements through a process of discretization, better known as meshing. Discretization visually manifests itself as the meshing of geometry. However, loads and supports are also discretized and, after the model has been meshed, the discretized loads and supports are applied to nodes of the finite element mesh.



Solve Finite Element Model

After creating the finite element model, we use a solver provided in SolidWorks Simulation to produce the desired data of interest.

Analyze Results

The analysis of results is often the most difficult step of all. The analysis provides very detailed results data, which can be presented in almost any format. Proper interpretation of results requires that we appreciate the assumptions, simplifications, and errors introduced in the first three steps: building the mathematical model, building the finite element model, and solving the finite element model.

Errors in FEA

The process of creating a mathematical model and discretizing it into a finite element model introduces unavoidable errors. Formulation of a mathematical model introduces modeling errors, also called idealization errors. Discretization of the mathematical model introduces discretization errors, and solution introduces numerical errors.

Of these three types of errors, only discretization errors are specific to FEA. Therefore, only discretization errors can be controlled using FEA methods. Modeling errors, affecting the mathematical model, are introduced before FEA is utilized and can only be controlled by using correct modeling techniques. Solution errors, which are round-off errors accumulated by solver, are difficult to control, but fortunately are usually very low.

Limitations of SolidWorks Simulation

With any FEA software, we need to take advantage of its strengths as well as work within its limitations. Analysis with SolidWorks Simulation is conducted under the following assumptions:

- □ material is linear
- □ deformations are small
- □ loads are static

These assumptions are typical of the FEA software used in the design environment, and the vast majority of FEA projects are run successfully within these limitations.

Linear Material

In all materials used with SolidWorks Simulation, stress is linearly proportional to strain.



Using a linear material model, the maximum stress magnitude is not limited to yield or to ultimate stress as it is in real life.

For example, in a linear model, if stress reaches 100,000 psi under a load of 1,000 lb., then stress will reach 1,000,000 psi under a load of 10,000 lb. 1,000,000 psi is, of course, a ridiculously high stress value.

Material yielding is not modeled. Whether or not yield, in fact, takes place can only be interpreted based on the stress magnitudes reported in results.

Most analyzed structures experience stresses below yield stress, and the factor of safety is most often related to the yield stress.

Therefore, the analysis limitations imposed by linear material seldom impede SolidWorks Simulation users.

Small Deformations

Any structure experiences deformation under load. In SolidWorks Simulation, we assume that those deformations are small. What exactly is a small deformation? Often it is explained as a deformation that is small in relation to the overall size of the structure.



The preceding figure shows a cantilever beam in bending with small deformations and large deformations.

If deformations are large, then the SolidWorks Simulation assumptions generally do not apply, even though SolidWorks Simulation has some large displacement analysis capabilities.

Note that the magnitude of deformation is not the deciding factor when classifying deformation as "small" or "large". What really matters is whether or not the deformation changes the structural stiffness in a significant way.

Small deformation analysis assumes that the structural stiffness remains the same throughout the deformation process. Large deformation analysis accounts for changes of stiffness caused by deformations.

Static Loads

All loads, as well as fixtures, are assumed not to change with time. This limitation implies that loads are applied slowly enough to ignore inertial effects. Dynamic loading conditions can not be analyzed with SolidWorks Simulation.

While all loads, in reality, change with time, modeling them as static loads is most often acceptable for the purpose of design analysis. Gravity loads, centrifugal forces, pressure, bolt preloads, and so on can be successfully represented as static loads.

Dynamic analysis is generally required only for fast-changing loads. A drop test or vibration analysis definitely requires that we model dynamic loads.

Task 1— Prepare the model for analysis

The Axle is a symmetric part. To analyze the entire part would be redundant as the loads and stresses will be the same on both sides of the plane of symmetry. Just as we did when analyzing the Deck, we will cut the part in half and only analyze one half. This will reduce the time to run the analysis and calculate the results as there will be half as many finite elements.

- 1 In the FeatureManager design tree, select the ConfigurationManager tab.
- 2 Right-click the top level icon and select Add Configuration.
- **3** Name the new configuration FEA.
- 4 Type Half of finished model for the description.
- **5** Click ✓ to create the configuration.



6 There are several small fillets that do not affect the strength of the part and will create difficulties when creating a mesh. Suppress the fillets around the Brake Mounting Pad and the end boss.



- **7** Select the bottom face of the model and open a sketch.
- 8 Reorient the model to the Top view.





9 Sketch a rectangle and add relationships to the origin and three edges of the model so that the rectangle covers half of the model.



10 Extrude a **Cut** through the entire mode. Click ✓.



11 We now have two configuration of the part, Default and FEA.



Task 2 — Setup the analysis

As stated earlier, SolidWorks Simulation is an Add-in product to SolidWorks. It is only available after it is turned on.

- 1 Click **Tools**, **Add-Ins** from the SolidWorks Menus.
- 2 Select SolidWorks Simulation. Click OK.

- **3** Examine the CommandManager, there will be a new tab for SolidWorks Simulation. There will also be a new menu titled **Simulation**.
- 4 To do an analysis, we must create a Study. On the menu, select **Simulation**, **Study**.
- 5 In the PropertyManager, type axle_static for the study name.
- 6 Under **Type**, select **Static**.
- 7 Click 🖌 .

- 8 The SolidWorks Simulation Manager will appear below the FeatureManager design tree and shows the study with folders for the different parameters used in the study.
- **9** Notice that the material we assigned in SolidWorks (6061 Alloy) has been automatically applied to the analysis.






- **10** Below the SolidWorks Simulation Manager and the graphics area, a new tab will appear for each study we create. To change studies, we just select the tab for the study of interest.
- 11 On the menu, click **Simulation**, **Options**.
- **12** Select the **Default Options** tab.
- 13 Select Metric for the Unit system, mm for the Length/ Displacement units and Kgf/cm^2 for Pressure/Stress. Click OK.

Unit system SI (MKS) English (IPS) Metric (G)	
Units	
Length/Displacement:	mm 💌
Temperature:	Celsius 👻
Angular velocity:	Hertz 🔹
Pressure/Stress:	Kgf/cm^2 👻

Task 3 — Add loads and fixtures

Because this part is symmetrical, we are only analyzing half of it. We are going to apply three fixtures, **Symmetrical**, **On cylindrical face** and **Fixed**.

Being symmetrical, nothing can move across the plane of symmetry, so looking at the image, the symmetric fixture prevents any part of the model from moving either across or away from the plane of symmetry.

If we look normal to the plane of symmetry, the model is free to move within the plane. Once the load is applied, the U-shaped cross section may have the tops of the two vertical bosses get closer or farther apart, or the sides and bottom of the U could bend.

The second fixture will be on a cylindrical face. This will represent the radial constraint of the bolt that will hold the Axle to the Truck.

The third restraint will be fixed, just to keep the model from moving. We will only apply the fixed restraint to a single vertex to prevent the fixture from causing the model to deform unnaturally.



- In the SimulationManager, right-click the Fixture item. Select Advanced Fixtures.
- 2 Under Advanced, select Symmetry.
- **3** Select the three faces shown.
- 4 Click 🗹 .



Note: You may want to select **Show preview** if it is not already turned on. This will show symbols representing the restraints and forces as they are applied. The preview is not shown in the above graphic and the two that follow just to make it easier to see which faces and vertex have been selected.



Select

Fixture

- 5 Right-click the Fixtures item again, and select Advanced Fixtures.
- 6 Under Advanced, select On Cylindrical Face.
- **7** Select the two cylindrical faces as shown.
- 8 Click Radial . Set the value to 0. This keeps the two cylindrical faces from moving away from the axis of the bolt. The surfaces are still free to move along the axis or to rotate about the axis.
- 9 Click 🗹 .

- 10 Right-click the Fixtures item again, and select **Fixed Geometry**.
- 11 Under Standard, select Fixed Geometry.
- **12** Select the vertex shown.
- 13 Click 🗹 .



- 14 In the SimulationManager, right-click the External Loads and select Force.
- **15** Select the face shown.
- 16 Under Force/Torque, select Selected direction.
- 17 Click in the Direction box to make it active, then select the Top plane from the Fly-out FeatureManager design tree.
- 18 Under Force, click Normal

to plane [Y]. Type 50 to apply 50 kgf normal to the Top plane. *If necessary*, **Reverse direction** for the force to be applied upwards.

- 19 Click 🗹 .
- **20** Examine the

SimulationManager. The material, fixtures and force that we applied are all listed.

🗙 🖃

Type Split

Force/Torqu

Torque

Face<2>

Normal

Top Plar
 O Per item

Units

Force

8

50

Total

Metric (G)

Selected direction

⊤ kgf

▼ kqf

50 • kgf

•



nal to Plane (kgf): 50

Task 4 — Run the analysis

Before the analysis can be run, the model must be meshed.

- 1 Right-click the **Mesh** item and select **Create Mesh**.
- 2 The default values for mesh element size and tolerance are normally a good place to start. Click



3 The model will mesh. During the process, the Mesh Progress dialog will keep us advised of the progress.



4 We now have everything needed to run the analysis. In the SimulationManager, rightclick the study axle_static and select **Run**.

Task 5 — Analyze the results

When the analysis has finished running, the results are placed in the Results folder in the SimulationManager. There will be a single plot for stress, displacement and strain however you can make additional results plots to suit your needs.

 Click the plus sign next to the Results folder. Double-click Stress1.



- 2 Examine the plot. The color of the model represents the stress and corresponds to the scale on the right. Our default settings show von Mises stress measured in kg/cm².
- 3 A red arrow indicates where on the scale the Yield Stress is located. In this case, the Yield Stress is higher then the top value on the scale, so the arrow is not shown. That means that the all the stress in the part is below the Yield stress. This is good.



- 4 There is a plot title in the upper left corner of the graphics area. Notice that the deformation scale is 103.247. This means that we are seeing slightly more than 100 times the actual deformation of the part. This is done just to make it easier to visualize the result.
- 5 Each plot can be customized to display the results in different ways. Right-click Stress1 in the Results folder and select Settings. From the Fringe Options list, select Line. This plot shows lines of constant stress something like a topographical map.



6 Double-click Displacement 1 under the Results folder. The colors now represent the amount of displacement of each element. The scale is in mm and we are looking at just about 100 times the actual deformation.

Notice that there is practically no movement along the plane of symmetry.



7 To display the result in different units, right-click the Displacement1 plot under the Results folder and select Edit Definition.... Select in from the list in the Units box.



Click 🖌 .

8 We can see from the scale that the maximum displacement is 5.673e-003 or about 0.005 inches.

 9 Right-click Displacement1 and select Settings.... Select
 Mesh from the Boundary options list.

Select Superimpose model on the deformed shape.

This will allow us to see both the deformed model as well as the undeformed model.

Select **Translucent (Single color)**, then click the color box and choose a different color, such as yellow.



Adjust the transparency slider to be able to see the undeformed shape.

Click 🖌 .

10 Reorient the model to the Left view.

We can see that the U shape is now wider at the top and the bottom has bowed upward.

Note: The Fixture icons have been hidden to make the graphic easier to see.



Task 6 — Create a plot

When we used SolidWorks SimulationXpress, one of the result we got was the Factor of Safety. We can create a similar plot in SolidWorks Simulation.

- 1 Reorient the model to the Isometric view.
- 2 Right-click the Results folder and select **Define Factor of Safety Plot**. The Factor of Safety dialog is a wizard that will lead us through the steps to create the plot.
- **3** Use the default Max von Mises stress for the first step. We will create a Factor of Safety plot by having the maximum von Mises stress divided by the yield stress.



4 Click Next 🗐.

- 5 Select **Yield strength** for the stress limit. Units and material have already been specified in our study so they should both be correct.
- 6 Click Next 💮.



1 Safety result Based on the maximum von Factor of safety:

1.9169

- 7 Select Factor of safety distribution.
- 8 Click 🗹 .

- 9 This plot shows us that the minimum Factor of Safety is 1.9.
- **10** Select the **Model** tab.
- 11 Select the ConfigurationMan ager tab.
- 12 Double-click the Default configuration.
- 13 Save and Close the Axle part.



5 Minute Assessment — #6-2

V	What happens during discretization or meshing.
]	The slope of the Stress-Strain curve is called?
١	What are the three conditions that must be met to use SolidWorks Simulation?

Active Learning Experience, Part 3— The Truck

The Truck connects the Deck to the Axle. Together, they form a joint that allows the wheels to turn. It also contains the suspension adjustments.

Design Intent

- □ One face of the Truck will be solidly mounted to the Deck.
- Bearings will be mounted in the Truck to connect it to the Axle, allowing rotation.
- Hex nuts will be molded into the Truck to allow adjustment of the suspension springs.



Procedure

When designing products, we first capture the functional aspects of the part. Once we have all the necessary features to allow the part to do its job, we can then refine, or optimize, the part to make it better. The optimization process may require us to make changes that will make the part stronger, lighter, easier to manufacture, or just more appealing to the eye.

The approach to create the Truck will be similar to creating the **Axle** in that the first several features will be created as separate bodies, then combined.

Task 1— Create the basic shape.

- 1 Create a new part using the Part_MM.slddot template.
- 2 Create a sketch on the Front plane.
- **3** Use the centerline and sketch mirroring to create the sketch.
- 4 Add a dimension between the arc and the base line. By default, the dimension will be placed at the center of the arc.



5 With the dimension selected, drag the end of the extension line from the center of the arc to the arc itself.



6 Once the dimension goes to the top of the arc, it can be changed to the correct value of **72mm**.

- 7 Sketch a circle and add the two dimensions shown. The center of the circle should be coincident to the centerline.
- 8 To fully define the sketch, add aConcentric relationship between the circle and the arc.



Mid Plane Extrusion

The **Mid Plane** extrusion creates the feature

so that it has an equal amount of material to each side of the sketch plane. The distance specified is the total depth of the extrusion.

9 Extrude the sketch. Select Mid Plane for the direction and type100mm for the depth.





🕞 Boss-Extrude

🖌 🗶 66

- 10 Click OK.
- 11 Create the following sketch on the Right plane.
- 12 The top of the sketch should be **Collinear** with the top of the model and the two top lines should have an **Equal** relationship.



13 Extrude the sketch Up To Vertex. Select the Vertex shown. Select Direction 2 and choose Up To Vertex. Select the her Vertex shown.



- 14 Clear Merge result and click OK.
- **15** Create a new Sketch on the **Top** plane.

- **16** Sketch the following geometry.
- 17 The sketch is symmetrical. Use the centerline and **Dynamic Mirror Entities**.



Note: When you add the first of each pair of fillets, you will get a warning that "At least one segment being filleted has a midpoint or equal length relation. Geometry may have to move to satisfy this relation when the fillet is created. Do you want to continue?"

Click Yes. Once you add the second fillet of the pair, the midpoint relationship will be restored and solved.

19 Add a **Tangent** relationship.

The sketch should now be fully defined.



Question: Why didn't we use Blind for the end condition?

Answer: We wanted to make sure that this extrusion always goes to the top of the part, even if the other two extrusions change.

- 21 Combine the three bodies. Click **Combine** i and select Common. Select the three solid bodies from the Solid Bodies folder.
- 22 Click **OK**.
- 23 Save this part as Truck to the Mountainboard\Axle-Truck folder.



Examine the part. The technique we used was to create three solid bodies that represented the Front, Right and Top view of the combined body.



- 2 Sketch a circle as shown, centered on the existing hole in the Truck.
- **3** Dimension the hole to **28mm**.



- 4 Create a Blind cut to a depth of 8mm.
- 5 Reorient the model to the Back view and repeat the cut.

Question: Could we mirror the cut.

<u>Answer:</u> No, the part is not symmetrical about the Front plane. If we mirror the cut, it will not be to the same depth.



Task 3 — Hex Nuts

The Hex Nuts will be molded into the Truck during the manufacturing process. We will create holes in the Truck to account for them.

- 1 Orient the model to the Top view.
- 2 Create a sketch on the top face of the model.
- **3** Sketch a centerline vertically from the Origin.
- 4 Click Dynamic Mirror Entities 🙆.
- 5 Click the Polygon tool (and sketch two polygons. If the Polygon tool is not on the Sketch toolbar, select Tools, Sketch Entities, Polygon from the menu.
- 6 The number of sides to the polygon is adjusted in the PropertyManager. Select **6**.
- 7 Turn off Dynamic Mirror Entities.
- 8 Add a **Horizontal** relationship between the Origin and the centers of the two polygons you created. You can select the two centerpoints and the Origin in the same command.

Notice that there are no centerpoints on the two polygons created by the **Sketch Mirror**.

9 Select one edge from each of the two polygons you drew and add a Vertical relationship. This keeps the polygons from rotating.



🔁 Po	olygon	?
~		
Optio	ons	~
	For construction	
Para	meters	\$
۲	6	
	 Inscribed circle Circumscribed circle 	
٩	-40.0384191	÷
٩	6.12271357	Å. T
\bigcirc	15.37021952	÷
17	318.09279179°	÷
	New Polygon	

10 Select the to circles that are inscribed in the polygons and add an equal relationship. With the circles equal, we only need to add size dimensions to one of the polygons.

TIP: You may have to zoom in on the polygons to be able to select the circles.

- 11 Dimension the sketch as shown.
- **12** Extrude a cut, **Through All**.
- **13** Rename this feature Hex Cuts.



Task 4 — Create Standoffs

When assembled, the axle flanges will be positioned between the two bearings. To reduce the contact area, we will add standoffs to the sides of the Truck.

- 1 Create a sketch on the face shown.
- 2 Click **Normal To** \bigcirc on the Standard View toolbar to orient the view to the selected face.
- 3 The face we are sketching is hidden. Change the display to Hidden Lines
 Visible by clicking on the Views

toolbar.



- 4 Create a "headstone" sketch.
- 5 Add a **Concentric** and **Collinear** relationship.
- 6 Dimension the arc to **18mm**.
- 7 We need to make sure the standoff has a hole that matches the center hole in the Truck.
- 8 Select the edge of the hole and clickConvert Entities 1.



- 9 Extrude the sketch to a **Blind** depth of **2.5mm**.
- **10** Create a matching standoff on the other inside face of the Truck.



Task 5 — Initial Analysis

Design is an iterative process. Once we have all the key elements in our design it is time to refine it.

Depending on the design intent, refinements may include such things as:

- □ Reducing the weight
- □ Reducing the amount of material
- □ Reducing the size of the part
- □ Improvements that make the part easier to make

We have everything in this version of the Truck except the holes that will be used to mount it to the Deck. We will check the weight of the part, then do a static analysis to make sure the part is strong enough.

- 1 Click Edit Material 🚼 to add material to the part.
- 2 Expand Plastics and select Nylon 6/10.
- 3 Click Apply and Close.
- 4 Click **Tools, Mass Properties**. The two things we are currently interested in, are volume and mass.

The volume is **209,105 cubic millimeters** (12.760 cubic inches). The more we can reduce this, the less material will be required. The less material, the cheaper the part is to produce.

The mass is 292.748 grams

(0.645 pounds). This will be one factor in the overall weight of the mountain board. The lighter the individual parts, the lighter the overall weight of the mountainboard and the easier it will be to carry.

5 Close Mass Properties.

Mass Properties			
Print Copy Close Options	Recalculate		
Output coordinate system: default	•		
Truck.SLDPRT Selected items:			
☑ Include hidden bodies/components			
Show output coordinate system in corner of window			
Assigned mass properties			
Mass properties of Truck (Part Configuration - Default)	<u>^</u>		
Output coordinate System: default			
Density = 0.001 grams per cubic millimeter	E		
Mass = 292.748 grams			
Volume = 209105.499 cubic millimeters			
Surface area = 45222.777 square millimeters			
Center of mass: (millimeters)			
Y = 19,204 Z = -0.359			
< III			

6 Change the appearance. As Nylon 6/10 has a default appearance that is transparent, apply an appearance of light gray low gloss plastic to make the model easier to see. Once the model is fully developed, we will add a different appearance that will represent what the final part will look like.

Task 6 — First stress analysis.

To determine if the part has sufficient strength, we will use Finite Element Analysis to examine the stress distribution and deformation of the model.

When in use, the Truck has loads applied from the weight of the rider plus numerous impact loads from running over objects, taking jumps and cornering. The actual computation of the magnitude and direction of these forces is beyond the scope of this course, so we will use a set of loads that were determined elsewhere.

Lets look at the Truck in the final assembly. One wheel has been removed for clarity.

If we look at just the vertical forces caused by the weight of the rider and the mountainboard itself, the ground must react with an equal and opposite force. This force is transmitted through the axle to the Truck.



Ground Reaction Force

Using vector addition, this force can be broken into two forces, one which pushes the Truck into the Deck, and a second force that tries to bend the vertical bosses of the Truck.



We will apply two forces to the Truck, one to represent the force normal to the Deck and one to represent the bending force.

1 Create a study. Click **Simulation**, **Study** in the menu.



- 2 Name the study Static_1. Select Static. Click OK.
- **3** To save time when setting up repeated studies, we can set our preferences for the studies.

In the menu, select **Simulation**, **Options**.

4 Select the **Default Options** tab.

Select **Units** in the left pane.

Set the System of units to Metric (G), the Length units to mm and Pressure/Stress to Kgf/cm^2.

Click OK.

Default Options - Units			×
System Options Default Options	Unit system SI (MKS) English (IPS) Metric (G) Units Length/Displacement: Temperature: Angular velocity: Pressure/Stress:	mm ▼ Celsius ▼ Hertz ▼ Kgf/cm^2 ▼	

- 5 In the SimulationManager, right-click Fixtures and select Fixed Geometry.
- 6 Select the bottom face of the Truck. Under Standard, select Fixed Geometry. Click OK.





✓ X →□
Message

Static_1

Static

Name

Туре

7 In the SimulationManager, right-click External Loads and select Force.

🖌 🗶 🚽

8 The first force we will add is the force that pushes the Truck into the Deck.

Select the two surfaces shown. These are the two faces the bearings will pressed into. The load from the Axle will be transmitted through these faces.

- 9 The units should be set to **Metric (G)**.
- 10 Select Force.
- 11 Select **Selected direction**, then select the Front Plane in the FeatureManager design tree.

This will set the force the force direction relative to the Front Plane.

Select Total as the 100 kgf is to be shared by both surfaces.

12 Select Along Plane Dir 2 and enter 100.

Check the preview icons to make sure they are pointing the correct way (down). *If necessary*, use **Reverse direction**. Force/Torqu Force Torque Face<1> Face<2> Normal Selected direction Front Plan Per item Total Units ~ Direction Metric (G) Force **** 🔻 kgf 100 🔻 kgf Reverse direction 3 🔻 kgf

Click .
13 To apply the bending force, we must split a surface to limit the area where the force applies.





- 14 Create a sketch on the face shown. Sketch a circle and make it Coradial with the circular edge of the standoff.
- 15 Split the face by clicking Insert, Curve, Split Line.
- 16 We will apply the bending force to this face and the face on the other vertical boss where the bearing bottoms out.
- 17 In the SimulationManager, right-click External Loads and select Force.
- **18** Select the two faces shown.
- 19 Select Normal. Set the Units to Metric (G) and type 50 for the Normal Force and select Per item. As we are applying this force to two faces, the total force will be 100 kgf.
- 20 Click 🗹 .



21 We now have all the forces and restraints to run the first analysis.

22 We applied a material (Nylon 6/10) to the part in SolidWorks so it is already applied to the part. We do not have to add the material again.

- 23 The next step is to mesh the model. In the SimulationManager, right-click Mesh and select Create Mesh.
- 24 Accept the default values and click **OK**.
- **25** Run the analysis by right-clicking the study Static_1 in the SimulationManager and selecting Run.









Task 7 — Examine the Plots

We can get an understanding of what is happening to our part under load by examining the various plots created by SolidWorks Simulation. This is not an automatic process, rather an engineering task which requires you to look at the results and draw you own conclusions.

1 Examine the different plots.

The Plots

Once an analysis has been run, plots are automatically created in the Results folder.

To display a plot:

- Double-click it in the SimulationManager.
- Or, right-click the plot and select **Show**.

Stress Plot

The stress plot shows the force per unit area. In the Metric system this is kilograms-force per square centimeter.

By looking at the color code, we can see that the stress is very small on the base plate. On the two vertical bosses, the stress increases as we move from the holes where the load is applied toward the base plate. We have the highest stresses where the vertical plates connect to the base plate. Much of this load is from bending.





Displacement Plot

The Displacement Plot shows how much each element of the model moves because of the applied loads. This plot is to an exaggerated scale. Look at the information in the upper left, it shows that the displacement is shown more than 72 times the actual displacement.

From the color code we can see that the base plate doesn't move. This is what we would expect because we applied a **Fixed** restraint to the bottom face.

Strain Plot

The Strain Plot shows the strain for each individual finite element.





Task 8 — Create a Factor of Safety Plot

- 1 Right-click the Results folder and select Define Factor of Safety Plot.
- 2 Select Max von Mises stress, then click Next.
- 3 The Units should be set to kgf/cm^2 by the default settings we selected earlier. Click Next.
- 4 Select Factor of safety distribution, then OK.

Design Check Plot

The Design Check plots the Factor of Safety (FOS) for the model. It shows how much more the loads can increased before the part yields or fails.

Looking at the color scale we can see that the minimum FOS is 17 (Red color). This means that the loads can be increased 17 times before the part yields or fails.

With a minimum FOS of 17, we can reduce some of the material in the part to make it lighter. Our concern will be to make sure that



the stress in the vertical bosses is transmitted to the base plate.

Ribs

The rib tool, **Insert, Features, Rib...**, allows you to create ribs using minimal sketch geometry. The tool prompts you for the thickness, direction of the rib material, how you want to extend the sketch if necessary, and whether you want draft.

Rib Sketch

The rib sketch can be simple or complex. It can be as simple as a single sketched line that forms the rib centerline, or it can be more elaborate. Depending on the nature of the rib sketch, the rib can be extruded parallel or normal to the sketch plane. Simple sketches can be extruded either parallel to or normal to the sketch plane. Complex sketches can only be extruded normal to the sketch plane. Here are some examples



Task 9 — Add ribs to vertical bosses.

To help support the load transmission from the vertical bosses to the base plate, we will add some ribs between the two.

There are several different methods to create these ribs, but the easiest to use is the **Rib** tool. With the **Rib** tool we will create one rib in the middle of the part, then pattern it to create the remaining ribs.

- 1 Select the **Model** tab to close the static study.
- 2 Create a sketch on the Right plane.
- 3 Change the model view to the Right view.
- 4 Sketch a single line making each end point coincident to the edges shown.





Both Sides

Parallel to Sketch

× 60

Extrusion direction

8

1.00deg

Flip material side

Parameters

- 5 Click the **Rib** tool 🍓
- 6 Select **Both Sides**. This will add material to both sides of the sketch line.
- **7** Select **Parallel to Sketch**. This will make the rib extrude in the plane of the sketch. Notice that this is different from all other extrusions which only extrude normal to the sketch plane.
- 8 Type **2.5mm** for the rib thickness.
- 9 Examine the sketch in the graphics area. Make sure the gray arrow is pointing towards the intersection of the vertical boss and base plate. If it is not, right click on the arrow or select Flip material side in the PropertyManager.

÷



10 Click . The Rib tool creates an extrusion that fills in material up to the next geometry it encounters.

Note: The rib is shown in red for clarity, your rib will be the same color as the rest of the part.



- 11 Now that we have one rib, the remaining ribs can be made using a pattern. Click **Insert, Pattern/Mirror, Linear Pattern**.
- **12** Select the two edges shown for direction 1 and 2.
- 13 Set the spacing to 12mm for each direction and the number of instances to 5.
- 14 Select Pattern seed only. This will create the pattern in the second direction with only the original seed element (the rib). If we did not check this box, SolidWorks would use all five instance of the rib created in the first direction to create the ribs in the second direction. This would create ribs on top of ribs and would not be very efficient.



15 Click in the box Features to Pattern to make it active, then select the Rib either in the graphics area or the FeatureManager design tree. You should now have a preview of the ribs.

Click OK.



16 Repeat the above procedure to create ribs for the other vertical boss.



Task 10 — Remove material from the base.

We can reduce the weight of the Truck by removing material from the three thick areas and then adding ribs to maintain the stiffness of the truck and restore some of the strength.

The first part of the task is to remove material from the base plate.

- 1 Create a sketch on the bottom face of the Truck.
- With the bottom face still selected, create an offset from the edges. Click Offset Entities

on the Sketch toolbar.

3 Select Reverse to get the sketch inside the edge of the truck and type 2.5mm for the offset distance.

⑦ Offset Entities ✓ X →	?		
Parameters	~		
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Add dimensions		/	
Reverse			
Select chain			
Bi-directional			
Make base			
Construction		· · ·	
Cap ends		N.	
<u>Arcs</u>			
Unes			

- 4 Show the sketch used to create the hex holes. Right-click the sketch under the feature Hex Cuts and select **Show**.
- 5 Sketch two circles centered on the two hex holes on the right.
- 6 Add an equal relationship. Dimension one of the circles to **22 mm**.
- 7 Trim the overlapping part of the circles to leave a sketch that looks like the figure 8.
- 8 Sketch a vertical centerline from the origin. The length is not important.



9 Select the centerline and the two arcs that make up the figure 8, then click Mirror

Entities 🔼.

Offset From Surface

The Offset From Surface end condition is used to locate the end of an extrusion as a measurement from a plane, face or surface rather than the sketch plane of the feature.

This allows a feature to terminate at a set distance from the selected surface. This can be used to create a cut that will always leave a





- 10 Click Extruded Cut 间.
- 11 Select Offset From Surface for the end condition, then select the top face of the truck.
- **12** Type **2.5mm** for the distance.
- **13** Check the preview.



- 14 Click OK.
- **15** Hide the sketch under Hex Cut feature.
- 16 Create four bosses for the holes that will connect the truck to the deck of the mountainboard.

Create a sketch on the bottom face of the Truck.

- 17 Sketch a vertical centerline through the origin.
- 18 With the centerline selected, click Dynamic Mirror Entities 🔬



- 19 Sketch two circles to the right of the centerline, SolidWorks will create two mirror images of the circles.
- 20 Add an **Equal** relationship between the two circles you sketched. Because we used the **Mirror Entities** command, the two circles that were drawn by SolidWorks will be equal to the ones we drew.
- 21 Add a **Vertical** relationship between the centers of the two circles you drew.
- 22 Turn off Dynamic Mirror Entities.
- 23 Add a dimension to one of the circles. For the value type 9.5mm. It doesn't matter which of the four circles you add the dimension to, the symmetric and equal relationships will take care of the remaining circles.
- **24** Add the dimensions shown to fully define the sketch.



Up To Surface

These new bosses we are creating need to be the same height as the bosses for the hex nuts. To make sure they are always the same height, we can extrude our sketch **Up To Surface**.

25 Extrude the sketch. Click Extrude

Boss/Base **a**. Select Up To Surface for the end condition, then select the face shown.

Notice that the face changes to magenta, the same as the color in the PropertyManager.

Click 🗹 .



Task 11 — Create strenghtening ribs.

We can't just leave the base plate hollowed out because it will not have enough stiffness to keep the vertical bosses upright. To strengthen it, we will add a web of ribs that look like this.

While this may look complicated at first, we can create all the ribs at one time using the **Rib** tool.

Examine the ribs. Notice that with the exception of the center vertical rib, all the remaining ribs radiate from a single point between the hex holes on each side.

To construct this set of ribs, we will start with a layout sketch.



All ribs go through



Construction Geometry

Construction geometry can be created to capture different relationships. This construction geometry can be very useful as it doesn't actually create anything. A simple example in the physical world would be the case of laying tile on a floor. To get the first row of tiles straight, we could use of a chalk line. The chalk line on the floor is our construction geometry to show where the first tiles go. When the floor is complete, we don't see the chalk line.

- 1 Create a sketch on the bottom face of the model.
- 2 Click **[1]** to start a centerline. A centerline is a line used for construction.
- **3** Sketch a centerline between the two vertices as shown.
- 4 Click **Point *** on the Sketch toolbar.

5 Move the cursor over the centerline until the midpoint symbol appears. Click on the midpoint symbol to create a point at the midpoint of the centerline.

6 Sketch a centerline from the midpoint of the top arc to the midpoint of the bottom inside edge. This represents the center of the part.



7 Sketch the centerlines shown.
 Each centerline starts at the point.
 Except for the horizontal and vertical centerlines, the other end of each centerline is coincident to the endpoint of an arc on the inner edge of the base.

8 Create the centerline and point on the right side of the model and create the remaining centerlines.

9 Sketch lines on top of the centerlines. These lines do not have to extend all the way to the edges because the rib tool will automatically extend the rib up to the next geometry. Each line however must cross all the other lines in the sketch. If they do not, the rib will stop at the next line rather than extending all the way to the existing part geometry.



Both Sides.

Normal to Sketch

Rib

🗙 66

arameters

2.000n

sion direction:

Flip material side
 1.00deg
 Type:
 Linear
 Natural

1

- 10 Click Rib and set the rib width to 2mm.
 Select Both Sides and Normal to Sketch. This will center the ribs on the sketch lines and extrude the ribs in a direction normal to the sketch plane instead of in the sketch plane as was done in the previous rib.
- 11 Examine the ribs, they should look like this.
- 12 If part of a rib is missing, edit the sketch used to create the rib and extend the line into the area where the rib is missing. As an example:



Task 12 — Add ribs to vertical boss.

In this task, we will need to remove material from each of the vertical bosses, then add ribs to stiffen the part.

Sketch face

Select

- 1 Orient the model to the Front view.
- **2** Create a sketch on the face shown.

- 3 Select the three edges shown, then click
 Convert Entities 1. This will give us three lines in our sketch that are tied to the underlying edges.
- 4 Right-click the top circular edge and click **Select Tangency**. Three edges are selected.
- 5 Click Offset Entities . Set the offset to 2.5mm to the inside and click OK.
- 6 We now have five lines and an arc that are all fully defined. While they are fully defined, their lengths can still be adjusted.

Extend Tool

Extend can be used to lengthen sketch geometry.

7 Click Extend Entities in on the Sketch toolbar or Tools,
 Sketch Tools, Extend from the menu. When you move the cursor over a line, the extended line will be previewed. Extend Entities will extend the line until it intersects the next sketch entity.




profile.

8 Extend the two lines as shown.



9 Trim the sketch to obtain a single closed

10 Create an additional 2.5mm offset from Offset this edge the edge of the circular cutout. This will be used to hold the bearing.



- Extrude a cut using Offset FromSurface. Type 2.5mm for the offset distance.
- **12** Select the inside face of the vertical boss as the offset surface.



- **13** Add a **5.0mm** fillet to the bottom edge.
- 14 Repeat the above steps to the other vertical boss.

Task 13 — Add ribs.

- 1 Orient the model to the Front view.
- **2** Create a sketch on face shown.
- Click Point * and add a point
 Coincident to the center of the hole.
- 4 Create the following sketch.

Notice that all the radial lines would pass through the center of circular hole if they were extended. In the last example we used construction geometry to set up the relationship. For this sketch, just add a **Coincident** relationship between each radial line and the point at the center of the circle.



TIP: Use **Mirror Entities** and **Symmetric** relationships to reduce the amount of sketching.

5 Add the additional arcs and lines to complete the sketch.



6 Create ribs. Use the rib tool to create ribs that are **2mm** thick and have **2°** of draft.

- 7 Add fillets. Add an 8mm fillet to the three edges as shown. This reduces the stress at the intersection of the ribs and the base plate as well as making the part look better.
- 8 Repeat the process for the other vertical boss.





Task 14 — Remove more material

The final operation is to remove some material along the top of the vertical bosses. As we could see in the first stress analysis, there is very little stress in this area, so we can remove the material to reduce the weight of the part.

- 1 Click **Chamfer (2)** on the Features toolbar.
- **2** Select the edge shown.

- **3** Select **Distance distance** for the type of chamfer.
- 4 Type **5.5mm** for the first distance and **10mm** for the second distance.
- 5 Examine the callout to make sure the 10mm is applied to the top side of the vertical boss and the 5.5mm is down the side. If the directions are reversed, type 10mm for direction 1 and 5mm for direction 2.
- 6 Click 🗹 .
- 7 Apply another **Chamfer** to the other vertical boss.



Select this edge

Task 15 — Create cuts for mounting hardware.

When we removed material from the base plate, we added four cylindrical bosses to support the mounting bolts that will hold the Truck to the Deck. We will add holes to the four bosses.

- 1 Orient the model to the Bottom view.
- **2** Select the bottom face of the model.

- **3** Click the **Hole Wizard (a)** on the Features toolbar.
- 4 Click the **Hole** button.

Set the properties of the hole as follows:

- Standard: Ansi Metric
- Screw Type: Screw Clearances
- Size: **M4**
- End Condition: Up To Surface

- 5 Once you select **Up To Surface**, you must select the surface. Rotate the model and select the top face of the base plate.
- 6 Click the **Positions** tab.
- 7 Place the holes. This portion of the wizard is used to locate and fully define the center point of the holes. A sketch point is added as the hole center point.
- 8 Multiple instances of the hole can be created in one command by inserting additional points at different locations.
- 9 Wake up the centerpoint. We want each hole to be centered on its respective boss so they must each be concentric to the boss. Move the cursor over the circular edge of a boss and pause. The centerpoint will be calculated and displayed.



The America Hole Specification





- **10** Move the cursor to the centerpoint and select it. This will add a concentric relationship between the point and the circular edge.
- 11 Repeat this procedure for the remaining three bosses.
- **12** Click **Point** \ast to turn it *off*.
- 13 There is one additional point on the bottom face. This was created when we initially selected the bottom face. Select this point and delete it as it is not needed.
- 14 Complete the hole by clicking **OK**.





Remove these ribs



Task 16 — Model refinement

Examine the model. The holes we just added come very close to some of the ribs. When the Mountainboard is assembled, a bolt will go through each of these holes and screw into a hex nut. We must have enough room around each of the holes to fit the nut.

We can fix this either by spacing the ribs differently or just removing the specific instances of the rib pattern where there isn't enough room for the nut. We will remove two ribs. Because these ribs were created by a pattern, we will remove them from the definition of the pattern.

 Locate the pattern feature that created this series of ribs, it should be LPattern1.

Note: If you added and removed patterns while creating the Truck, the feature name might be different.

2 Right-click the pattern feature and select **Edit Feature**.

- 3 Locate the section **Instance to Skip** in the PropertyManager and click the down arrow. This will expand the **Instance to Skip** section and place a magenta dot over each pattern instance in the graphics area.
- 4 Select the two magenta dots as shown. The preview will show that the rib instances will be removed.
- 5 Click **OK**. The two ribs have been removed and we now have enough room for the nuts.





Task 17 — Analyze the model.

Now that the model is complete, we analyze it again to make sure it is still strong enough with the material removed.

- 1 Check the weight of the part. Click **Tools**, **Mass Properties**.
- 2 The part now weights 158.308 grams (0.349 pounds), just a little over half the 293 grams we started with.
- 3 Close Mass Properties.
- 4 Click the SolidWorks Simulation tab Static_1 below the FeatureManager design tree.

🚳 Mass Properties		
Print Copy	Close Options	Recalculate
Output coordinate system:	default	-
Selected items:	Truck.SLDPRT	
Include hidden bodies/comp	ponents	
Show output coordinate system in corner of window		
Assigned mass properties		
Mass properties of Truck (Part Configuration - Default)		
Output coordinate System: default		
Density = 0.001 grams per cubic millimeter		
Mass = 158.308 grams		
Volume = 113077.191 cubic millimeters		
Surface area = 82113.834 squ	are millimeters	
Center of mass: (millimeters) X = -0.000 Y = 20.416 Z = -0.528		
•		•

- 5 Re-mesh the model. Right-click the Mesh icon and select Create Mesh.
- **6** We may be warned that remeshing the model will delete the previous results.

Click OK.

7 This part is much more complicated than the last time it was meshed, so we will not use the default mesh. Select **Curvature**

based mesh and set the element size to 8mm. Click 🗹 .

Note: Meshing techniques and settings are a topic in themselves and are beyond the scope of this project. For now we will just accept that these settings are needed to obtain a proper mesh of this model.



- 8 Rerun the analysis. Right-click the study by right-clicking Static_1 in the SimulationManager and selecting **Run**.
- Examine the plots. The stress plot shows that the maximum stress of 4.742 x 10² kgf/cm² is still below the Yield Strength of the material.



10 The Design Check plot shows a minimum factor of safety of 3.



Continued refinement.

While this process looks very easy and we got a result that gives us an indication of the stress in the model, we don't know how accurate they are. As mentioned earlier, FEA is a method of approximations. We generally do not just run one analysis of the model, rather we run several analyses to see if the results are consistent or converging.

Task 18 — Refine the analysis

To refine the results, we will run the model again using a smaller mesh size.

1 Remesh the model with a smaller element size. Right-click the Mesh icon in the SolidWorks Simulation Manager and select **Create Mesh**.

2 Type **4.0mm** for the mesh element size and **0.4mm** for the minimum element size. This will create smaller elements.

3 Click **OK** to create the mesh.

This mesh uses elements half the size of the previous mesh. The result is about four times the total elements than used in the first run. Also note that the mesh has adjusted its size around some areas such as the holes.



4 Run the analysis. This will take longer than the previous analysis due to the higher number of elements.





Examine the results. The Maximum stress is 6.853x 10² kgf/cm², this is higher than the previous result.



6 The FOS is 2.1 which is less then with the courser mesh. As the maximum stress was higher, the actual FOS is lower than before.

Are we done yet?

If we were going through the full development of this part, we would have to do additional analysis. We would continue to refine the mesh on this part until the results between runs were more consistent. If the Factor of Safety then became too low, we would have to do more refinement of the model such as adding material in the high stress areas to spread out the load.

When we set up this problem, we used SolidWorks Simulation. This limits us to:

• Static loads

When this part is in use, loads are usually not static. As the mountainboard goes over bumps or the rider takes jumps, the loads may be impact loads or rapidly varying.

Linear Material

Most plastics do not exhibit linear stress-strain curves.

• Small Deformations

The material will flex considerably under impact loads, so we have probably exceeded the small deformations limit.

We also made some assumptions about the size of the load. While these load assumptions and limitations of SolidWorks Simulation were acceptable during the early development phase of this part, we would need to use additional tools to get us closer to the loads this part will see in use. The calculations and theory needed are beyond the scope of this course, so we will assume that the design is satisfactory as far as we have taken it.

Task 19 — Adjust the part's appearance

The manufactured part will be a black textured color. In this task we will change the appearance of the part to a black texture and adjust its reflective properties.

- 1 Select the Model tab below the FeatureManager design tree to exit simulation.
- 2 Select the **Appearance** tab on the Task Pane.
- **3** Expand the **Appearances** and **Plastic** folders and select **Textured**.
- 4 Press and hold the **Alt** key and drag the appearance PW-MT11250 into the graphics area. This will apply the appearance to the entire part. The PropertyManager will now show the properties of this appearance. If we needed to change the color of the part, it could be done here, however the default color is the color we were trying to achieve, so it can be used as is.
- 5 Click 🖌 .
- 6 Save and Close the part.



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5 Minute Assessment – #6-3

- 1 What are the three primary requirements to use SolidWorks Simulation?
- **2** True or False: When creating a linear pattern, in two directions, the directions must be 90 degrees apart?

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

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

Exercises and Projects — Multibody Parts

Exercise 17: Combining a Multibody Part

Create this part by following the steps as shown.

This lab reinforces the following techniques:

- □ Multibody solids
- □ Combining
- □ Hole Wizard

Units: inches

Procedure

Open a new part using the Part_IN template and name it Mbody1.

1 Sketch first profile.

Use lines, fillets and offsets.

Extrude the profile **2.25in** using a mid-plane end condition.



- 3 Combine bodies.Combine the two solid bodies into one.
- **4** Add features Add boss, cut, hole wizard and fillet features.



•

- 5 Finish part with **0.0625**" radius fillets and rounds.
- 6 Save and close part.



Exercise 18: Bridging a Multibody Part

Create this part by following the steps as shown.

This lab reinforces the following techniques:

- □ Multibody solids
- □ Bridging
- Units: millimeters

Design Intent

The design intent for this part is as follows:

- □ Part is *not* symmetrical.
- \Box Holes are though all.
- □ All fillets and rounds are **5mm** radius.

Procedure

Open a new part using the Part_MM template and name it Mbody2.

1 Create a multibody part.





2 Finish part with bridge technique.



3 Save and close part.

Lesson 6 Quiz

Name:	Class	s: Date:
-		

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

- 1 What SolidWorks tool is used on multibody solids to do Boolean operations?
- 2 When you create a Full Round Fillet, what determines its radius?
- **3** What type of fillet can be used to have the fillet radius change along the length of an edge?
- 4 What mirroring option is used to mirror half of a part to get the full part?
- 5 What are the three steps of the FEA process?
- 6 What happens during discretization or meshing?

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

- 8 What are the three conditions that must be met to use SolidWorks Simulation?
- **9** If you apply a material in SolidWorks, do you have to apply it again in SolidWorks Simulation?
- **10** True or False: When creating a linear pattern, in two directions, the directions must be 90 degrees apart?
- 11 Relative to the sketch plane, which direction can you extrude a rib?
- 12 What are the three Boolean operations that can be done with the **Combine** command?

___:

Lesson Summary

- □ Using multibody techniques, we can create a part or feature by creating extrusions based on the standard views of the final part, then combine them into a single body.
- □ SolidWorks Simulation is used to analyze parts and assemblies to determine the internal stress, strain, deformation and factor of safety.
- □ Ribs can be used to create extrusions both parallel to the sketch plane as well as normal to the sketch plane.
- □ Design is an iterative process. It can take several refinements before the final part is created.

Lesson 7: Sweeps and Lofts — Springs and Binding

Goals of This Lesson

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



Resources for This Lesson

This lesson plan corresponds to the following modules in the SolidWorks Online Tutorials.

- **Gamma** *Revolves and Sweeps*
- \Box Lofts
- □ Multibody Parts

For more information about the Online Tutorials, See "Online Tutorials" on page 1.

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

Questions for Discussion

- 1 What is a multibody solid?
- **2** What is a linear material?
- **3** List some differences between SolidWorks SimulationXpress and SolidWorks Simulation.
- 4 List some types of refinements we may apply to our models once they have all the functional features.

Outline of Lesson 7

- □ In Class Discussion
- □ Active Learning Exercises, Part 1 Creating a Spring
 - Creating a sweep
 - Composite curves
 - Create a user defined plane
 - Create an axis
- □ Active Learning Exercises, Part 2 Create an Assembly
 - Create an exploded view
 - Animate the exploded view
- □ Active Learning Exercises, Part 3 Binding Straps
 - Sweep with guide curves
 - Full round fillet
 - Loft features
- □ Active Learning Exercises, Part 4 Multibody Parts
 - Creating Multibody parts
 - Saving solid bodies as separate part files
 - Hiding components in an assembly
 - Edit a part inside an assembly
- □ Exercises and Projects Sweeps
 - Sketch the Sweep Section
 - Create the Sweep Path
- □ Lesson Summary

In Class Discussion

Extrusions and Revolves can be used to create a large number of features, however they have limitations when it comes to complex shapes.

Quick Review:

□ Extrusions: A sketch is moved along a straight path to add material (Extruded Boss) or remove material (Extruded Cut).



 Revolves: A sketch is rotated about a centerline or edge to add material (Revolved Boss) or remove material (Revolved Cut).



More complicated shapes can be created with Sweeps and Lofts.

as a Swept Cut.

Sweeps

Sweeps take a sketch, called a Profile and move it along a **Path**. It must be a closed, non-self-intersecting boundary. However, the sketch can contain multiple contours either nested or disjoint.



Lofts

Lofts create a solid by connecting a series of loft profiles. These profiles can be different shapes.



Active Learning Exercises, Part 1— Creating a Spring

Create the spring. The Spring will be created as a sweep feature.

In the previous lessons, material was added either through extrusions or revolves.

Task 1— Crate A Helix

A spring is created by forming a rod into the shape of a helix. To create a spring, we will sweep a circle along a helix.

- 1 Open a new part using the Part_MM template.
- **2** Create a sketch on the Top plane.
- **3** Sketch a circle and dimension its diameter as **25mm**.

Creating A Helix

The **Helix** command creates a helical 3D curve based on a sketched circle and defined by values for a combination of height, pitch and number of revolutions.

- 4 Click Insert, Curve, Helix/Spiral from the menu.
- 5 Select Height and Revolutions from the Defined By list.
- 6 Type **45mm** for **Height** and **5.5** for **Revolutions**.
- 7 Select Counterclockwise for the rotation direction and Odeg for the Start angle.
- 8 Click **OK**. The helix will be the sweep path for the spring.





Select here

Task 2 — Create a Profile

The sketch used as the sweep profile must be created on a plane that is on the end of the sweep path. To create a plane at the end of a line or edge, simply select the line or edge near the end where you want to create the plane, and insert a sketch.

1 Create a sketch for the profile on an plane normal to the end of the helix. Select the helix near the end closest to the

origin, then click Sketch 🛃

2 Sketch a circle. The position should be approximately as shown. We do not have to position it exactly as we will do that with a relationship.

Pierce Relationships

The **Pierce** relationship is used in sweeps. It can be thought of as a 3D Coincident relationship. In our spring, the circle will follow the helix. Think of the helix as a thin piece of wire and the profile sketch as being drawn on a piece of paper. If we stick the wire through the paper at the center of the circle, this would be a pierce relationship. When we perform the sweep, the paper will slide along the wire, held at the pierce point.

3 Add a **Pierce** relationship. Select both the *center* of the circle and the *helix*. There should only be one relationship shown,

Pierce. Apply the **Pierce (1)** relationship.

- 4 Dimension the circle to a diameter of **4mm**. The sketch should be fully defined.
- 5 **Exit** the sketch.

When we create extruded or revolved features, only one sketch is involved, so we normally select **Extrude** or **Revolve** while we are still in the sketch. Because a sweep requires more than one sketch (profile and path) we must not be in **Edit Sketch** mode to start the sweep.

6 Create the sweep. Click **Swept Boss/Base** G on the **Features** toolbar.



- **7** Select the circle for the profile and the helix for the path. The callouts and color help to identify which sketch is 号 Sweep which. × **Profile and Path** -0 Sketch2 Plane Magenta Path(Helix/Spiral1) Blue Profile(Sketch2)
- 8 Set Options to **Follow Path**. This will keep the profile sketch normal to the path curve as it sweeps.



9 Click 🖌 .

We now have the center section of the spring. We need to make some additions to create a realistic spring. Compression springs have the last turn at each end at a tighter pitch and the ends are ground flat to create more contact area.



Task 3 — Create the Lower Helix

1 Delete the Sweep feature. Select Sweep1 in the FeatureManager design tree and press Delete. We only want to delete the sweep feature, not the underlying sketches and helix. Make sure that Also delete absorbed features is cleared (not checked).

Confirm Delete	
Do you really want to delete this: Sweep1 (Feature)	Yes to A
And all dependent items:	<u>No</u>
	<u>C</u> ancel
	Help
Also delete absorbed <u>f</u> eatures	
Don't ask me again	

Confirm Delete	X
Do you really want to delete this: Sweep1 (Feature) And all dependent items: Sketch2 (Sketch) Helix/Spiral1 (Feature) Sketch1 (Sketch)	Yes to All No Cancel Help
 ✓ Also delete absorbed features ✓ Also delete all child features □ Don't ask me again 	<u>Treb</u>

2 Delete the profile sketch (Sketch2).

Profile sketches must be located at the end of the sweep path. Because we are going to add on to both ends of the helix, we will not be able to use the existing sketch. We will create a new profile after the completed helix is created.

- 3 Create a sketch on the Top plane
- 4 Sketch a circle and dimension it to **25mm**.
- 5 Click Insert, Curve, Helix/Spiral.
- 6 Define the helix by Pitch and Revolution. Set the pitch to 4mm and 1.0 revolutions. The Start angle must be 0.00deg to make the helix end where the original helix begins.
- 7 Select Reverse direction. Check the preview graphics to make sure the new helix is going in the correct direction. It should look like an extension to the original helix.
- 8 Click ✓ .The new helix should join the existing helix as shown.



Task 4 — Create the Upper Helix

The upper helix will be create the same as the lower helix except that we need to create a new plane to create the circle used to make the helix. At the lower end, we used the Top plane to create the circles used for the two helixes. There is no plane at the top end of helix so we must create one. We also have a special consideration that we want to make sure the plane is always at the top of the helix, even if we later change the height of the first helix.

1 Click Insert, Reference Geometry, Plane.

- 2 Expand the Flyout FeatureManager design tree by clicking the plus sign next to the spart icon. Select the Top plane in the FeatureManager design tree and the top end of the helix.
- **3** Because we selected a plane and a point, SolidWorks will default the type of plane to **Parallel**.
- 4 Click 🗹 .
- **5** Create a sketch on this new plane.
- 6 Sketch a circle, centered on the Origin and dimension the diameter to **25mm**.

- 7 Create a helix by selecting Insert, Curve, Helix/Spiral.
- 8 Set the Helix options as shown.Notice that we have to start at 180° because the original helix was 5.5 turns.
- 9 Click 🗹 .
- **10** Create a sketch at the end of the bottom helix.
- 11 Sketch a circle, dimension it as **4mm** in diameter.
- 12 Add a **Pierce** relationship between the center of the circle and the bottom helix.
- **13** Exit the sketch and create a sweep.

Look at the preview, the sweep does not extend past the end of the bottom helix. We can only have one path for a sweep and there is no way to designate all three helixes in the sweep command.

- 14 **Cancel** the **Sweep** command by clicking X.
- **15 Delete** the sketch with the circle profile.

Task 5 — Create a Composite Curve

We could not use the three helixes for the sweep path because the path must be a single curve. To make a single curve from the three, we will create a combination called a Composite Curve.

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	😕 Helix/Spiral	?
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Path(Helix/Spiral2)

Profile(Sketch5)

Composite Curve

A **Composite Curve** enables you to combine reference curves, sketch geometry , and model edges into a single curve. This curve can then be used as a guide or path when sweeping or lofting.

- 1 Click Insert, Curve, Composite from the menu.
- 2 Either in the graphics area or the FeatureManager design tree, select the three Helixes.

Cor 🔁	nposite Curve	?
🧹 🎽		
Entitie	es to Join	~
V	Helix/Spiral1 Helix/Spiral2 Helix/Spiral3	
	пеіх/эрігагэ	



3 Click 🗹

The three Helix/Spiral features and their sketches will now be absorbed by the CompCurvel feature. If you select CompCurvel in the FeatureManager design three, the entire helix will highlight in the graphics area showing that the three curves have been combined.

Task 6 — Create The Sweep

- 1 Create a sketch at the end of the composite curve.
- 2 Sketch a circle, dimension it as **3.9mm** in diameter.
- 3 Add a **Pierce** relationship between the center of the circle and the composite curve.
- 4 **Exit** the sketch.
- **5** Create a **Sweep** using CompCurve1 as the path.
- 6 Hide the sketch plane. Right-click on the plane and select Hide.



Task 7 — Create End Cuts

Actual springs would have the top and bottom ground flat. To do this on the model we will create cuts using straight lines.

- 1 Create a sketch on the Front plane.
- 2 Reorient the model to the Front view.
- **3** Draw a vertical centerline from the Origin downward.
- 4 Draw a horizontal line.
- 5 Add a **Midpoint** relationship between the end of the centerline and horizontal line.
- 6 Dimension the sketch as shown.
- 7 Click Insert, Cut, Extrude on the menu.





- 9 Click 🗹 .
- 10 Repeat this procedure to the other end of the spring. The total length of the spring should be 53mm.
- 11 Save this part as Spring in the ...\Mountainboard\Spring Assembly folder.



Task 8 — Create Reference Geometry

Thinking ahead to the task of creating an assembly using this spring, we will need to create some reference geometry to make it easier to mate the spring in the assembly.



- 3 Click 🗹 .
- 4 Rename the Axis **Centering**.

- 5 Click Insert, Reference Geometry, Plane.
- 6 Select the bottom face of the spring.

With only one plane selected, the default plane type will be distance. We want this plane to be at the mid-point of the spring which is 53mm tall. We could divide 53 by 2 to get the distance, then enter the value in the property manager.

An easier way is to do the math right in the property manager by typing 53/2. As soon as you press return, SolidWorks will replace the equation with the result 26.5mm.









8 Click 🗹 .

Rename the plane Center Plane.

9 Save the part.



Active Learning Experience, Part 2— Create An Assembly

Creating the final assembly of the mountainboard will be easier if we create the subassemblies as we go along.

1 Create a new assembly using the Assembly_MM template.

Because the Spring is still open, it will be listed in the Open documents box in the PropertyManager. Select the Spring. A copy of the Spring will be on the screen and move with the cursor.

2 Move the cursor over the Origin of the assembly. When the cursor changes to click on the Origin.

This will fix the Origin of the spring to the Origin of the assembly with corresponding planes coincident (Front to Front, Top to Top, etc.)



3 Add the Spring Dampener. Click **Insert Components** in the Assembly toolbar. Click **Browse** and select the Spring Dampener that you created in the practice exercise in Lesson 4.

Click in the graphics area to insert the part.

Note: If you did not create the Spring Dampener in the practice exercises in Lesson 4, the completed part is located in the LessonO4\Exercises\Built Parts folder. Built versions of the Spring Retainer and Fender Washer, which we will also need for this assembly, are located there as well.

- 4 Turn on the Axes and Temporary Axes by clicking **View**, 6ar - Axes and View, Temporary Axes from the menu or selecting them in the **Heads-up** toolbar. **5** Click **Mate (N)** on the Assembly toolbar. 6 Select the axis of the Spring and the temporary axis of the Axes Spring Dampener. The default mate will be **Coincident**. Temporary Axes Click 🗹 . This positions the Spring Dampener inside the Spring. ▦ 7 Click **Mate** (on the Assembly toolbar. Ø -(D1) 8 Select the Centering Plane in the Spring Centering and the Top plane of the Spring Dampener. The default mate will be **Coincident**. Click 🗹 .
 - **Note:** Normally, mates are added to reflect the mechanical conditions of the assembly. In some cases, such as the Spring Dampener, the part may float in the actual assembly. The Spring Dampener can move along the axis as it will be held by bolts at either end. Because we want to contain the Spring Dampener in our model we use reference geometry to hold it in position.
- 9 Add a Spring Retainer. Click Insert, Component, Existing Part/Assembly.
- 10 Click **Browse** and locate the Spring Retainer. Click in the graphics area to drop the part.
- 11 Mate the Temporary Axis of the Spring Retainer to the Centering axis of the Spring.

- Select the two faces shown, then click Mate .Add a Coincident mate.
- 13 Hide the Centering Plane.

In the FeatureManager design tree, expand the Spring features by clicking the plus sign next to the Spring part.

14 Right-click the Center Plane and select **Hide** from the menu.



15 Add another Spring Retainer to the assembly. Hold the **Control** key and drag the Spring Retainer from the FeatureManager design tree into the graphics area


Mate alignment:

- SolidWorks Engineering Design and Technology Series
 - 16 Mate the second Spring Retainer to the Centering axis of the Spring and the top ground face, just as was done to the first Spring Retainer.

Note: To reverse the direction of the Spring Retainer you have to change the alignment. Click either **Align** or **Anti-Align** to reverse the direction

- 17 Add a Fender Washer to the assembly and mate it to the top Spring Retainer.
- **Save** the assembly as Spring Assembly to the ...\Mountainboard\Spring Assembly folder.

Task 9 — Create An Exploded View

To show how this assembly is put together, create an exploded view.

Exploded Assemblies - Review

You can make **Exploded Views** of assemblies by exploding the assembly component by component. The assembly can then be toggled between normal and exploded view states. Once created, the **Exploded View** can be edited and also used within a drawing. **Exploded Views** are saved with the active configuration.

You can only create one exploded view per configuration of the assembly.

Setup for the Exploded View

Before adding the **Exploded View**, it is good practice to create a configuration for the storage of an **Exploded View**.









- 1 Add a new configuration. Switch to the ConfigurationManager, right-click the top level icon and select **Add Configuration**.
- 2 Type the name Exploded and add the configuration.
- **3** The new configuration is the active one.





4 Make the Axes visible by clicking **View Axes** from the Heads-up toolbar.

Assembly Exploder - Review

The Assembly Exploder is used to create the individual steps in an Exploded View.

Each step requires three actions:

- Select a component.
- Move the component.
- Adjust the component position, if necessary.
- 1 Orient the model to the Isometric view, then zoom out. We need to zoom out because the assembly will take up more room on the screen as we move the components away from their mated position.
- 2 Click Insert, Exploded View.
- 3 Select the Fender Washer. A **Triad** will appear on the part. The **Triad** is used to move the part in specific directions.



- 4 Drag the green arrow to move the Fender Washer along the Y-axis. The exact distance is not important. We will adjust all the positions later to make the exploded view look correct.
- **5** When you finish dragging the part, the Triad will disappear and a blue arrow will appear. You can use the blue arrow to further adjust the position of the part.
- 6 When you are done adjusting the position, click in any open space in the graphics area to end this step or click the next component to explode.

- 7 Click on the upper Spring Retainer. Drag it by the green arrow of the **Triad** to a position below the Fender Washer.
- 8 We want the Spring Dampener to move to a position alongside the Spring. We will make it do three moves during the explode. First it will move vertically along the axis until it is above the Spring. Then it will move radially away from the axis. Finally it will move to a position alongside the Spring.
- 9 Click on the Spring Dampener, then drag it by the green arrow of the Triad until it is above the Spring.
- 10 Click in an empty area of the graphics window to finish the step.







- 11 Click on the Spring Dampener, then drag it by the red arrow of the **Triad** until it is outside of the Spring.
- **12** Click in an empty area of the graphics window to finish the step.



13 Click on the Spring Dampener, then drag it by the green arrow of the Triad until it is alongside the Spring.When complete the exploded assembly should look like this.







14 Create another move step to move the lower Spring Retainer in the -Y direction.

When looking at the exploded view, we may not have moved all the pieces far enough. In the above graphic, the Fender Washer and upper Spring Retainer are too close to the Spring. When the Spring Damper is exploded, it moves through the other two components.

Editing the Explosion Steps

The PropertyManager lists all the steps of the explosion.

To simply adjust the position of the components in a step, just click on the step. The component will turn color to magenta and the blue drag arrow will be visible.

To make the Triad visible, right-click the step and select Edit Step.

- 15 Select Explode Step1. Drag the Fender Washer further from the Spring.
- **16** Select Explode Step2. Drag the upper Spring Retainer further from the Spring.
- 17 When all the components are where you want them, click \checkmark
- 18 Turn off the Origin, Axes and Temporary Axes by clearing them on the **View** menu.
- **19 Hide** any planes that are showing.



entering

Explode/Collapse Assembly - Review

The Exploded View information is stored in the ConfigurationManager with the configuration. You can create one exploded view for each configuration of the assembly.

Once created, the assembly can be exploded or collapse through the ConfigurationManager.

To explode or collapse the assembly, right-click

ExplodeView and select either **Explode** or **Collapse**.

When using **Explode** or **Collapse**, all parts move



from their collapsed position *directly* to their explode position regardless of the steps taken to get from collapse to explode.

Note: There will only be a choice to **Explode** or **Collapse**, never both. If the assembly is already exploded, the only choice will be to **Collapse**. If collapsed, the only choice will be to **Explode**.

Animate Explode/Collapse

The explode and collapse steps can be shown as an animation. When animated, each step will be done in order so each part follows the path created by the explode steps.

To animate the explode or collapse, right-click **P** ExplodeView and select either **Animate explode** or **Animate collapse**.

Task 10 — Explode and Collapse the assembly

- 1 In the ConfigurationManager, right-click ExplView1 and select **Collapse**. All assembly parts move directly to the assembled positions.
- 2 Right-click ExplView1 and select **Explode**. All parts return directly to the exploded positions.
- **3** Right-click ExplView and select **Animate collapse**.

The parts will move in turn based on the steps used to create the exploded view and the **Animation Controller** will appear.

Animation Controller

The **Animation Controller** proves simple controls, similar to a CD or DVD player, to control the playback of animations.

If PhotoView 360 is installed, the animations can be recorded with photorealistic rendering.



1 Click **Reciprocate** ↔, then **Play** . The assembly will continue to explode and collapse.

- 2 Click Stop 🔳 .
- 3 Click Normal \rightarrow to reset the Animation Controller.
- 4 Click Save Animation in the Animation Controller.
- 5 The default name for the animation file will be the same as the assembly. Save the file as a Microsoft AVI file (*.avi) and Renderer SolidWorks screen.

Type **10** for **Frames per second**.

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📆 Save Animation to File

- 6 Click **Save**. Select **Microsoft Video 1** as the Compressor.
- 7

Note: You may get a warning message about Microsoft Video 1 compressor causing corrupt AVIs. For now, we will use it and observe the results.

Video Compression				
Compressor:	ОК			
Microsoft Video 1	Cancel			
Compression Quality: 85	Configure			
	About			
Key Frame Every 8 frames				

- 8 Click OK.
- **9** Watch the progress on the screen. The **Animation Controller** showed that the animation was four seconds long. MotionManager will record 41 frames. One frame at time zero, then one frame every 1/10 of a second (10 frames per second) until the animation is complete.
- 10 Use Windows Explorer to locate the AVI file.
- 11 Double-click the file. The default media player should start and play the animation.
- **12 Close** the media player.
- **13** Save and Close the assembly.

Active Learning Experience, Part 3 — Binding Straps

The Binding straps consist of four pieces, two straps that attached to the Binding Base Plate, a foam pad that goes over the rider's foot, and a catch to hold the ends of the straps together.

To make it easier to fit the parts together, the two straps and the foam pad will be constructed in the same part as



multibody solids. Once we are sure everything fits correctly, each of the final multibodies will be saved as a separate part.

Task 11 — Create the Binding Straps

1 **Open** the part Binding Start Sketch. This part only contains sketches, and reference geometry. It is used to set the spacing and interrelationships between the different parts of binding.

The rectangles and centerlines represent the positions of the

posts of the Binding Base Plate. The straps will later attach to these posts.

The four blue curves will become the paths and guide curves when we sweep the straps.

- **2** Create a sketch on the Top plane.
- 3 Sketch a rectangle as shown. The size and position are not critical.
- 4 We now have to position the sketch using relationships to the path and guide curve.
- 5 Right-click on the line shown and choose SelectMidpoint from the menu.
- 6 Click Add Relationship **[**].



- 7 Select the curve shown. Make sure you select the *curve* and not an *endpoint*. If you select the endpoint instead of the curve, you will not be able to add a **Pierce** relationship.
- 8 Select **Pierce** in the PropertyManager.
- **9** Add another **Pierce** relationship between the corner point of the rectangle and the guide curve.

- 10 Dimension the width of the rectangle to 4mm.
- **11 Exit** the sketch.
- **12** Rename the sketch to **Profile1**.
- 13 Create a sweep. Click Insert, Base/Boss, Sweep.





- Sweep ?
- as the path.

14 Select Profile1 as the profile sketch and CompCurve1

- 15 Click 🗹 . Examine the feature. Notice that there is a twist in the sweep.
- **16** To correct the twist, we will use the guide curve.

Guide Curves

Sweeps can contain multiple guide curves which are used to shape the solid. As the profile is swept, the guide curves control its shape.

- 17 Right-click the sweep and selecting Edit Feature from the menu.
- 18 Expand the sections Options and Guide Curves by clicking the arrow ≥.
- **19** Select CompCurve2 for the guide curve.
- 20 Select Follow path and 1st guide curve for Orientation/ twist Type. This will cause the profile to keep the corner point on the guide curve and prevent the twist we saw in the previous step.
- 21 Click 🗹 . The sweep now stays flat.





Full Round Fillets - Review

The **Full Round Fillet** option creates a fillet that is tangent to three adjacent sets of faces. Each face set can contain more than one face. However, within each face set, the faces must be tangent continuously.

A **Full round Fillet** does not need a radius value. The radius is determined by the shape of the faces you select.

- 22 Select Fillet 🙆.
- 23 Select Manual, then Full round fillet for Fillet Type. The Items To Fillet box will expand to require three faces to be selected.



Task 12 — Create the Second Strap

Creating the second strap will be essentially the same as creating the first strap except that we will not round off the end because we will later attach the clasp assembly. Also, because we are creating this second strap in the same part as the first strap we must make sure that the two part geometries do not merge.

Parallelogram - Review

The **Parallelogram** tool is part of the Rectangle tool and is used to create rectangles that *do not* have two sides vertical and two sides horizontal. When creating a parallelogram, opposite sides will have a parallel relationship.

The Parallelogram tool and can also be used to create parallelograms where the corners do not meet at 90 degrees.

- 1 Create a sketch on the Top plane.
- 2 Use the **Parallelogram** tool to create a rectangle. Click the Rectangle tool and then select the **3 Point Center Rectangle** in the PropertyManager.



Note: To sketch a parallelogram:

Drag from point 1 to point 2 and then release the mouse button.

Place the cursor at point 2 and drag to point 3 and release the mouse button.



- 3 Create **Pierce** relationships between the midpoint of one side of the rectangle and ComCurve3, and between the corner point of the rectangle and CompCurve4.
- 4 Dimension the width of the rectangle to 4mm.



- **5 Exit** the sketch.
- 6 Rename the sketch Profile2.
- 7 Create a sweep using the sketch Profile2 as the profile and CompCurve3 as the path and CompCurve4 as the Guide Curve.
- 8 Clear **Merge results**. We want to keep the geometry of this strap separate from the first strap.



 Profile and Path Profile 2 CompCurve 3 Options Correntation/twist type: Follow path and 1st guit Merge tangent faces Show preview Merge result 	🗲 Swee	ep	?		
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- 9 Click 🗹 .
- 10 Examine the FeatureManager design tree. Near the top is a folder called Solid Bodies. Click the plus sign to expand the folder. Each solid body takes the name of the last feature used to create it. Fillet1 is the first strap that was created with Sweep1 and Fillet1. Sweep2 is the strap we just created.



11 Rename the solid bodies to Strap_right and Strap_left.



12 Save the part.

Task 13 — Create the Foam Pad

The binding needs a pad to spread the load out on the top of the rider's foot.

This part will be molded in the flat state shown at right. When installed in the binding, it will take a shape defined by the straps and the rider's foot. We will create this pad two different ways. First we will create it in the flat state as this is what will be manufactured. We will also create the part in its bent state in order to show the pad in our product literature.



Lofting

Lofting enables you to create features that are defined by multiple sketches. The system constructs the feature - either a boss or a cut - by building the feature between the sketches.

1 Open the part Foam Pad.sldprt. This part contains the section profiles.

Each profile is a different size but is similar in shape. When lofting, the individual sections can be different shapes, this is what makes lofting so powerful. The one important consideration is that each section have the same number of sketch segments.

2 Create a loft. Click Insert, Boss/ Base, Loft.



3 Select each sketch in order at the point indicated. You do not have to pick on an exact point. The idea is that where we select is close to the intersection of the horizontal and vertical lines.
Select close to this intersection

4 The preview should look like this.

5 Click

6 Save the part.

Question
What happens if you select the wrong points or they are out of order?

Answer

The preview shows the selected points with the cyan control handles. These can be dragged to different positions.

If you selected the sections in the wrong order, you can change the order in the PropertyManager by selecting the profile you want to reorder, then selecting

Move Up () or **Move Down ()** until it is in the correct position.

- 7 To edit the Loft, right-click Loft1 and select Edit Feature.
- 8 Drag the third control handle to the other end of the horizontal line as shown.

Profile(Plus3)

- 9 Click is no longer smooth as different nodes on each section are now connected.
- **10** Edit the Loft1 feature and move the control point back to its original position.
- 11 Apply material to the part. ClickEdit Material : on the Standard toolbar.



- 12 Expand the material category **Other Non-metals** and select **Rubber**.
- 13 Click 🗹 .
- 14 Save and Close the part.

Active Learning Experience, Part 4 — Multibody parts

We created the two binding straps in the same part file. We will now add the deformed foam pad to this part. When complete, each of the three multibodies will be saved to a separate file.

The advantage of creating the parts in a single part file is that the final three parts will be parametrically linked back to our original multibody part. Any changes that need to be made, can be made in the multibody part which will then propagate the changes into the individual parts.

Task 1— Create a curved foam pad

- 1 **Open** the part Binding Start Sketch that we used to create the two straps.
- 2 Select the sketches Loft Section 1 through Loft Section 7. Right-click any of the selected sketches and select Show.
- 3 These are the same loft sections as used in the flat Foam Pad. The only difference is that they are all on planes that are normal to the edges of the straps. Some of the sections are also rotated to keep the rectangular slot aligned with the straps.



- 4 Create a Loft between the seven sketches. Click Lofted Boss/Base [] on the Features toolbar.
- 5 Select the seven sketches in turn. Make sure you select each sketch near the same point. Zoom in as necessary to make sure you select the correct point.



- 6 Clear Merge results.
- 7 Click 🗹
- 8 Examine the Solid Bodies folder in the FeatureManager design tree. There should be three solid bodies.
- 9 Rename the solid body Loft1 to Foam_Pad.
- Hide the sketches. Expand the Loft Feature in the FeatureManager design tree by clicking the plus sign next to the Loft1. Select the seven sketches. Right-click any selected sketch and click Hide.



Saving Solid Bodies as Parts

You can save one or more of the solid bodies in a multibody part as separate part files. There are several commands to do this, each with different characteristics. Some commands give you the option to also generate an assembly from the saved parts.

Default Templates

The commands in this section create new SolidWorks documents - either a part, an assembly or both as appropriate. You have the option of specifying a document template or allowing the system to use the default template. This choice is determined by the settings in **Tools, Options, System Options, Default Templates**.

Insert Into New Part

Insert into New Part allows you to save individual solid bodies as part files. Each resulting part file is linked by an external reference back to the source part.

Insert into New Part does not create a feature in the source part. The solid bodies are saved as they are after the last part feature is rebuilt. Any changes you make to the source part *will* propagate to the saved parts.

Task 2 — Save The Solid Bodies As Parts

- 1 Set the default part template. Click **Tools**, **Options** from the menu.
- 2 Select the System Options tab, then Default Templates.
- **3** For Parts, click and browse to the Training Templates folder and select the Part_MM.prtdot template.

Note: We could also set the default assembly and drawing templates through this same procedure.

4 Click OK.

- 5 In the Solid Bodies folder, right-click Strap_right and select Insert into New Part. The Save As dialog will open.
- 6 Name the new file Strap_right.sldprt and save it to the Mountainboard\Binding folder.
- 7 The FeatureManager design tree shows only a single feature, showing that this part is referenced back to the Stock-Binding Start Sketch part.
- 8 Press **Control-Tab** to shift back to Binding Start Sketch.
- 9 Save the remaining two solid bodies to separate parts called Strap_left.sldprt and Foam_Pad_curved.sldprt.

Task 3 — Create The Strap Assembly

- 1 Create a new assembly. Click **File**, **New** and choose the **Assembly_MM.asmdot** template.
- 2 All the open parts will be listed in the Open documents section of the PropertyManager.
- **3** Select Strap_right. The part Strap_right will now be previewed on the cursor.
- 4 Click the plus sign next to the Assembly icon on the fly-out FeatureManager design tree to show the existing components.





5 Click on the Origin. This will insert the part Strap_right on the assembly Origin with its three planes aligned to the corresponding planes in the assembly.



6 Click Insert, Component, Existing Part/Assembly from the menu and insert the Strap_left and Foam_Pad_curved part using the same method.

Because the three parts all came from the same original file, their origins and default planes all line up. This allowed us to insert them into the assembly without having to add additional mates.

7 Notice that each part has an (f) in from of it. This means that the parts are fixed in space and can not be moved.



8 Save the assembly as Strap Assembly.

Task 4 — Add The Clasp Assembly

- 1 Open the assembly Clasp Assembly.sldasm.
- 2 Tile the SolidWorks windows by clicking Windows, Tile Vertically.
- 3 Drag the top level icon for the Clasp_ assembly into the Strap Assembly.





Hide Components

Hiding a component temporarily removes the component's graphics but leaves the component active within the assembly. A hidden component still resides in memory, still has its mates solved, and is still considered in operations like mass property calculations.

To **Hide** a component:

- □ Click **Hide/Show Components** (Magnetic Component is visible, it will hide it. If the component is hidden, it will show it.
- Click on a component and then click Hide component or Show component is on the Context toolbar.
- □ Right-click the component and select **Hide** or **Show**.
- □ Right-click the component and select **Properties** from the **Component** list. Select the **Hide Component** check box.
- □ From the pull-down menu, choose **Edit**, **Hide** or **Edit**, **Show**.
- 4 Hide the parts Strap_Right and Foam_Pad-curved. Select the two parts, either in the graphics area or the FeatureManager design tree, then click so on the Assembly toolbar.

Mate Considerations

The sweep provides some additional challenges for mating as there are few planar surfaces or linear edges. To mate the Clasp Assembly we will have to mate some edges and points.

5 Add a **Tangent** mate between the top of the strap and the top of the slot in the clasp.

We use a **Tangent** mate instead of a **Coincident** mate because the top face of the strap is not planar.



Select

Select

6 Add a **Coincident** mate between the midpoint of the strap and the midpoint of the edge of the clasp.

- 7 While it may look like we have enough mates, if you try to drag the clasp assembly, it can still rotate.
- 8 Add a **Coincident** mate between the vertex of the strap and the edge of the clasp.

The clasp is now fully mated to the strap.

9 Show all the parts. Select the Strap_right and Foam_Pad_curved then click Hide/Show
Components (%).

Task 5 — Add an exploded view

Exploding the straps will be a little different than with previous assemblies because we want the straps to move as if they are being retracted from the clasp assembly. These directions are not along the assembly X, Y, or Z axes.

- 1 Click Insert, Exploded View from the menu.
- 2 Select the component Strap_right, the **Triad** will appear and be aligned with the assembly axes.
- **3** Right-click the center of the **Triad** and select **Align to**. Select the face shown. The Triad will align itself to this face.
- 4 Drag the red arrow of the Triad to move the Strap_right clear of the other components. Notice that it moves as if we were sliding it out of the clasp assembly.
- 5 Repeat this procedure to move the Strap_left to the left and clear of the other components.
- 6 Finally, move the Clasp Assembly vertically.
- 7 Collapse the assembly and Save it.



Task 6 — Insert A Sub-Assembly Into An Assembly

Assemblies can be added to other assemblies in the same way parts are added to assemblies. Assemblies inside other assemblies are call sub-assemblies, however, they are exactly the same file type within SolidWorks.

1 **Open** the Binding assembly.

- 2 Tile the windows vertically.
- 3 Drag the top level icon of the Strap Assembly into the Binding.
- 4 Maximize the Binding window by clicking **Maximize** \Box on the Window Title Bar.
- 5 Click 🔊 to add a **Mate**.
- 6 Expand the Strap Assembly in the FeatureManager design tree and select the Top plane of the Strap Assembly.
- 7 Select the top face of the Binding Base Plate.
- 8 Click 📂 to add a **Distance** mate. Type **12mm**

for the distance. Click 🗹 .

This mate sets the vertical height of the strap assembly.

9 Add a Tangent mate between the face shown and the top face of the Strap_right.

Note: We cannot use a **Coincident** mate as the top face of the strap is not planar.

- 10 Add a **Coincident** mate between the vertex and face shown. The **Strap Assembly** should now be fully mated into the **Binding** assembly.
- **11** Save the assembly.





Task 7 — Create Strap Buttons

Parts can be built in the assembly. This allows the leveraging of the geometry of other parts within the assembly. To hold the straps to the Binding Base Plate, each strap needs two posts that will fit through the curved slots.

Edit Part

While you are in an assembly, you can switch between editing the assembly — adding mate relations, inserting components, etc., — and editing a specific part. Editing a part while in the



context of an assembly allows you to take advantage of geometry and dimensions of other components while creating matching or related features. Using geometry outside the part creates **External References** and **In-context Features**.

Two commands, **Edit Part** and **Edit Assembly**, are used to switch back and forth between editing one component in an assembly and editing the assembly itself. When you are in edit part mode, you have access to all the commands and functionality the part modeling portion of SolidWorks. Plus, you have access to other geometry in the assembly.

To edit a part in an assembly, select the part you wish to edit, then either:

- Click Edit, Part or Edit, Assembly
- □ From the right-mouse menu, select Edit Part or Edit Assembly
- □ From the Assembly toolbar, click the **Edit Component** [vol.
 - **TIP:** The **Edit Component (Solution**) tool is a toggle. It switches you between **Edit Part** mode and **Edit Assembly** mode. It also acts as a visual indicator of which mode you are in. It is depressed when you are in **Edit Part** mode.
 - Note: The ToolTip on the we tool says Edit Component. In an assembly, both parts and sub-assemblies are considered components. To see the edit part color click Use specified colors when editing parts in assemblies found under Tools, Options, System Options, Colors.

Other indicators that you are in **Edit Part** mode are the status bar which reads **Editing Part**, and the window banner which looks like this:

Strap_right -in-Binding.SLDASM

1 Select the part Strap_right, then click to the **Edit Part** tool.

The strap we are editing turns blue

as well as its representation in the FeatureManager design tree.



Note: The colors of the components and their transparency are controlled in **Tool**, **Options, System Options, Color**. The colors and transparency on your system may be different from the colors shown in these graphics.

To set you system to show the same colors shown here set the following Solidworks System Options:

Color:

Set the color for **Assembly, Edit Part** to Royal Blue (fifth column, fourth row).

Select Use specified colors when editing parts in assemblies.

Display/Selection:

For **Assembly transparency for in context edit**, select **Opaque assembly** from the list.

2 Select the face of the Binding Base Plate as shown and click **Insert, Sketch**.

Even though we are editing the part Strap_right, we can create a sketch from a plane in another part.

- Click Normal To on the Standard Views toolbar. This will change our viewpoint so we are looking directly at the selected face.
- Click Zoom To Selection on the View toolbar.
 This will make the selected face fill the screen.
- **5** Sketch a vertical centerline from the midpoint of tab.
- 6 Turn on **Dynamic Mirror Entities** by clicking 🔊 on the Sketch toolbar.







7 Sketch a circle in one of the slots. The second circle will be drawn by the mirror command.

8 Add two Tangent relationships between one of the circles and the sides of the slot.



- **9** Dimension the distance between the circles.
- **10** Turn off **Dynamic Mirror Entities**.
- 11 **Hide** the Binding Base Plate. We don't have to hide this part, however it makes it easier to see the preview.



12 Click Extrude.

13 Click $\overline{\mathbf{N}}$ to reverse the direction of the extrusion.

Note: The default direction for extrusions is away from existing geometry.

- 14 Select **Up To Next** for direction. **Up To Next** extends the feature from the sketch plane to the next surface that intercepts the entire profile. (The intercepting surface must be on the same part.)
- 15 Click 🗹 .
- 16 Right-click the Strap_right part and selectOpen Part from the menu.

We do not need the surrounding geometry to create the tops on the two pins, so it is easier to work in the part instead of the assembly.

17 Select the top face of one of the pins and open a sketch.





18 Click the Offset Entities tool. Type 1mm for the distance.Click OK.

Select the top of the other pin, click **Offset Entities** , then

click 🖌 .



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2	< -⊨	
araı	neters	~
ò	1.000mm	4
	✓ Add dimensions	
	Reverse	
	Select chain	
	Bi-directional	
	Make base construction	
	Cap ends	
	Arcs	
	🔘 Lines	

Note: You can only offset one entity at a time, so we had to do the top of each pin separately.

19 Extrude the sketch to a depth of **1mm**. Add a **.5mm** fillet to the top of each button.



- 20 Return to the Binding assembly by clicking Window from the menu and selecting Strap_right -in- Binding.SLDASM from the list.
- 21 We are still in the Edit Part mode. Click
 Edit Component (State 1) to return to the Edit
 Assembly mode.
- 22 In the FeatureManager design tree, select the Binding Base_Plate and click Hide/Show

Components (B) to show the part.

- **23** Repeat the above procedure to add pins to the Strap_Left part.
- **24 Save** the Binding assembly.



Task 8 — Create The Binding Pad

The Binding Pad will be made by cutting a flat piece of material to shape, then gluing it to the Binding Base Plate. This is another case where we might create two versions of this part. One would be the flat piece used for manufacturing and the other would be the part in its curved shape to be used for illustrations.

We will create the curved version of the part in the context of the Binding assembly.



- 1 Click Insert, Component, New Part from the menu.
- 2 The cursor changes to \searrow indicating that we need to select a plane for the first sketch. Whichever plane or face we select will become the Front plane of the new part. When

the cursor is over a valid face, it will change to k_{s}^{\bullet} .

- **3** Select the thin face shown.
- 4 As soon as the face is selected, we are in **Edit Sketch** mode, editing the new part. Everything is still gray as there is no geometry in our new part.

The FeatureManager design tree shows the part in the blue color.

Select

Internal Parts

The name assigned to new parts include braces [] surrounding the name. This indicates an internal part and is done automatically for all new parts created incontext to offer you the flexibility to easily discard parts that you don't want and not be concerned about renaming as you work.

- **Renaming** Right-click the part and choose **Rename Part** to set the name of the part.
- □ Saving Right-click the part and choose Save Part (in External File) to save the part to a true part file (*.sldprt) outside the assembly. Saving the assembly will generate the same options.
- 1 In the FeatureManager design tree, right-click [Part1^Binding] and select **Rename Part**. Type Binding Pad as the new file name.
- 2 Right-click the part [Binding Pad] and select Save Part (in External File).

- 3 Select the file Binding Pad and click Same As Assembly. This will save the Binding Pad to a separate part file in the same folder as the assembly.
- 4 Click OK.
- 5 Notice that the braces are now gone from the part name in the FeatureManager design tree as this is now a separate part.

Convert Entities - Review

Convert Entities enables you to copy model

edges into your active sketch. These sketch elements are automatically fully defined and constrained with an **On Edge** relation. **Convert Entities** is like **Offset Entities** except that the offset distance is always zero.

To convert entities, select the edge or edges, then:

- □ Pick **Convert Entities ()** tool from the Sketch toolbar
- □ Click Tools, Sketch Tools, Convert Entities from the menu.
- 1 Reorient the view to the Right View.
- 2 Click **Convert Entities [**], and then select the six edges

shown. Click 🗹 .

- **3** Open the part Binding Pad. Right-click Binding Pad in the FeatureManager design tree and select Open Part.
- 4 Right-click Sketch1 and select Edit Sketch.
- 5 Drag one of the endpoints shown onto the other endpoint.

Note: Even though the endpoints were black, indicating that they were fully defined, they can be dragged to new locations. This is only true when the sketch entities were converted from edges.

- 6 **Exit** the sketch.
- 7 In the FeatureManager design tree, select the sketch.





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- 8 Click **Insert, Curve, Composite**. This will combine the six line and arc segments into a single curve that be can use as a path for a pattern feature.
- 9 Click 🗹 .
- 10 Create the following sketch on the Front plane.

11 Drag the corner of the sketch onto the point in the composite curve where the left end of the straight line meets the tangent arc. The sketch should be fully defined.



12 Extrude the sketch 130mm.

We will later cut away the extra material. For now we just need to make sure the extrusion will extend past the end of the Binding Base Plate.



13 Pattern the feature. Click Insert, Pattern/Mirror, Curve Driven Pattern.

Select CompCurvel in the FeatureManager design tree for the direction. Type **34** for the number of instances and **5.5 mm** for the spacing. Select **Tangent to curve** for the **Alignment method**.





Note: Each instance of the extrusion was 6 mm wide but we patterned it with a spacing of 5.5 mm. This makes sure that the final pattern does not have gaps when it makes the turn up the slope at the end of the Binding Base Plate.

Pattern Alignment

Alignment method controls Tangent to curve the orientation of the patterned instances. Selecting Tangent to curve will cause the pattern instances to rotate as the pattern changes direction. Align to seed If Align to seed is selected, the pattern instances will maintain the same orientation as the seed feature. 14 Click 🗹 . This creates the pattern in just one direction. To create the rest of the pattern, we will create another pattern

from the same extrude feature and pattern it in the other direction.

- 15 Create another pattern. Click Insert, Pattern/Mirror, Curve Driven Pattern.
- 16 Select CompCurvel in the Feature Manager design tree for the direction and Extrudel for the Feature to Pattern. Type 8 for the number of instances and 5.5 mm for the spacing. Select Tangent to curve for the Alignment method.



- 17 Click is the full pad, ready to be trimmed.
- **18** Return to the Binding assembly.
 - **TIP:** Hold **Control** and press the **Tab** key. This will cycle through the open SolidWorks documents.
- 19 Because the Binding Pad is different from when we left the Binding assembly, the assembly needs to be rebuilt. Click **Yes**.



The Binding Pad should completely cover the top of the Binding Base Plate.

- 20 The only part we need to use to trim the Binding Pad is the Binding Base Plate.Hide all the other parts.
- 21 Reorient the model to the Bottom view.



- 22 Create a sketch on the bottom face of the pad.
- 23 Select the edge of the hole in the Binding Base Plate.
- 24 Click **Offset Entities** and create an offset **5mm** to the outside of the hole.



25 Create an **Extruded Cut** using the end condition **Through All**.

Note: Because we are in the **Edit Part** mode, **Through All** means through all of the part we are editing. It *does not* affect any other part.

- **26** Create another sketch on the bottom face of the Binding Pad.
- 27 Create a sketch on the bottom face of the Binding Pad that is offset **2mm** from the edges of Binding Base Plate.

Note: After creating offset lines and arcs from the edges of the Binding, you must inspect the sketch at each intersection of a line and an arc to make sure that there is a single end point. If not, trim and extend the lines and arcs as necessary to create a single closed sketch.

- 28 Extrude a cut **Through All**. We want to cut away everything that is outside the sketch so remember to check **Flip side to cut**.
- 29 Return to the Binding assembly.Click () to exit Edit Part.



- **30** Show all the parts.
- **31 Save** the assembly. This completes the Binding assembly.



5 Minute Assessment – #7

- 1 Unlike an extruded feature, a swept feature requires a minimum of two sketches. What are these two sketches?
- 2 What information does the pointer provide while sketching an arc?
- **3** What does Convert Entities do?
- 4 How many loft profiles are required to create a loft feature?
- 5 What are the functions of Guide Curves when creating a sweep?
Exercises and Projects — Sweeps

Exercise 19: Sweeps without Guides

Create these two parts using swept features. These require only a path and a section, no guide curves.

Units: millimeters

Cotter Pin

The Cotter Pin uses a path that describes the inner edge of the sweep.



Paper Clip

The Paper Clip is defined by a path that describes the centerline of the sweep.



Thanks to Paul Gimbel, TriMech Solutions, LLC for submitting these examples.

Exercise 20: Attachment

Create this part using the step by step instructions provided. Use relations or link values where applicable to maintain the design intent.

This lab uses the following skills:

- □ Sketching
- □ Planes
- □ Extruding
- □ Sweeping
- Multi-thickness Shelling
- Variable-radius Fillet



26

140°

60

Design Intent

The design intent for this part is as follows:

- □ Part is symmetrical.
- □ Wall thickness is uniform.

Procedure

- 1 Open a new part using the Part_MM template and name it Attachment.
- 2 Layout sketch.

Sketch a layout of the part on the Front reference plane. The sketch sets the locations and dimensions for the two main features.

Name the sketch Layout.

Note: The **26°** angle is dimensioned to the Right reference plane.

3 Plane normal to curve.

Create a plane that is normal to the endpoint of the upper line of the Layout sketch.

Name the plane cyl plane.

4 Plane through 3 points.

Create another sketch on the Top reference plane and add a short vertical line from the Origin.

Exit the sketch.

Using the **Through Lines/Points** method, select the endpoints of this line and the sharp corner of the Layout sketch, to define another plane.

Name this plane intake.

5 Sketch the profile.

Sketch on the intake plane to create the profile of the nozzle.

Use symmetry to create the sketch and tie it to the Layout sketch.



6 Axis.

Create an axis defined by the intersection of the Front and Top reference planes.

This will be the vector for the extrude direction.

7 Extrude.

Extrude the profile sketch using the **Blind** end condition. Select the axis for the **Direction of Extrusion**. Set the **Depth** to **28mm**. 8 Cylinder.

On the cyl plane, sketch a **34mm** diameter circle, centered on the end of the upper line in the Layout sketch.

This circle will be used to extrude a cylinder.

Up to Surface

From the on-line help it is easy to see that the end condition **Up to Surface** meets our needs. **Up to Surface** extends the extrusion from the sketch plane to the selected surface. The surface can be a face, a reference plane, or a stand-alone surface.

9 Up to Surface.

Click **Insert, Boss, Extrude**. Verify from the preview that the boss is extruding in the correct direction. If it is not, click **Reverse Direction**.

From the End Condition: list, select Up to Surface.

Select the front face of the swept first feature.

Click **Draft On**, set the angle to **2°**, and check **Merge result**.

Click 🖌 .



10 Multiple-thickness Shell.

Shell the solid **2mm** to the inside, selecting the end faces for removal. Select the cylindrical face and set it to **4mm**.



12 Variable radius fillet

Add a variable radius fillet to the set of tangent edges shown. The fillet varies from **5mm** to **10mm** at the middle, and back to **5mm**.



TIP: This technique simplifies assigning the values to the vertices:

- 1. Click **Fillet** 🙆.
- 2 Click Variable radius.
- 3 Right-click an edge, and pick **Select Tangency**.
- 4 Set the **Radius** to **5mm**, and click **Set All**.
- 5 Set the number of control points to **1**.
- 6 Click in the **Items To Fillet** list.
- 7 Use the arrow keys on the keyboard to cycle through the list of selected edges. As you do so, the control point will move from one edge to another.
- 8 When the control point appears on the correct edge, click it in the graphics area. Then, use the callout to assign the **10mm** radius.
- 9 Click 🖌 .
- 13 Add **5mm** radius fillets to the edges shown.



14 Inner fillets and rounds.

Add fillets and rounds of **3mm** on the inner edges of the part as shown in the section view at the right.

15 Save and close the part.



Exercise 21: Hanger Bracket

Create this by following the steps as shown.

This lab uses the following skills:

- □ Multibody solids
- □ Sweep using guide curves
- □ Merging bodies

Units: inches

Design Intent

The design intent for this part is as follows:

- □ All fillets and rounds are **0.125**".
- Part is symmetrical with respect to the parting line.
- \Box Draft is **3°**.

Procedure

1 Open a new part using the Part_IN template and name it Hanger Bracket.





2 Create sweep ends.

Create two extruded solid bodies to represent the ends of the sweep.

3 Create sweep path.

The path and the guide curve must each be in separate sketches.

Create the path sketch using the existing geometry.



4 Create guide curve.

Create the guide curve sketch using the existing geometry including the path sketch.

TIP: If you sketched *all* the geometry in one sketch, it can still be used. Change the two lines and the arc that form the guide curve to construction geometry. Open a new sketch for the guide curve.

Use **Convert Entities to** copy the guide geometry into the new sketch.





Exercise 22: Tire Iron

Create this by following the steps as shown.

This lab uses the following features:

- □ Sweep feature
- □ Revolve feature
- □ Sketch fillets
- Polygon tool
- Dome feature
- □ Reference planes

Design Intent

The design intent for this part is as follows:

- □ Regular end is symmetrical using angled cuts.
- □ Wrench end is created using a hexagon cut.
- □ Section is constant diameter.



Procedure

- 1 Open a new part using the Part_IN template and name it Tire Iron.
- **2** Create the sweep path.

Create the sketched lines then add the fillet.





3 Insert sweep.

Create a new reference plane and use it to sketch the sweep section sketch. Sweep the profile along the path.



4 Revolved feature.

Create a revolved feature on the angled end of the sweep feature. This boss will hold the hexagon cut.



Lesson 7 Quiz

 Name:
 Class:
 Date:

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

1 Describe the steps required to create a swept feature.

- 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.



3 Describe the steps required to create a Loft feature.

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

Part 3:

- **5** True or False. The location where you select each profile determines how the Loft feature is created.
- 6 What two sketches are required to create a Sweep feature?
- **7** Where can you find additional sketch tools that are not located on the Sketch Tools toolbar?
- 8 Multiple choice. Circle the best answer. Examine the illustration at the right. How should you create this object?
 - a. Use a **Revolve** feature
 - b. Use a **Sweep** feature



- 9 True or False. A SolidWorks part can contain more than one closed volume.
- **10** What is the name of the entity created by combining curves, sketch geometry and model edges into a single curve?
- 11 When exploding components in an assembly, how do you reorient the direction of the Triad?
- 12 How many exploded views can be created of an assembly?

Lesson Summary

- □ Sweeps
 - Sweeps are created my moving a profile sketch along a path.
 - Guide curves can be used to control the twist of a sweep.
- □ Composite curves can combine individual entities into a single curve that can be used as a sweep path.
- □ Lofts create a solid by connecting multiple profiles.
- □ Multibody solids can be saved as individual parts.
- Exploded views can be created to show how the components of an assembly go together.
- Parts can be created in the context of an assembly so that we can take advantage of the geometry of other parts.

Goals of This Lesson

□ Upon successful completion of this lesson, you will create the final assembly of the all the parts created in the previous lessons into the finished mountainboard:



Before Beginning This Lesson

□ Complete lessons 1 through 7. The parts and assemblies needed to complete this lesson were created in the previous lessons.

Review of Lesson 7 — Sweeps and Lofts

- 1 What is the primary difference between a sweep and a loft?
- **2** What are Composite Curves?
- **3** How is the radius of a Full Round Fillet determined?
- 4 How do you hide a component in an assembly?
- **5** Can you hide a feature in a part?
- 6 How do you create an AVI recording of an assembly explode or collapse?

Outline of Lesson 8

- □ In Class Discussion Assemblies and Mates
- □ Active Learning Exercises, Part 1 Assembly
- □ Active Learning Exercises, Part 2 Information From The Assembly
- □ Active Learning Exercises, Part 3 Exploded View
- □ Exercises and Projects Assembly Drawings
- □ Lesson Summary

In Class Discussion — Assemblies and Mates

Of all the parts designed by engineers, very few are used by themselves. Most become part of an assembly that may then become part of a larger assembly.

Examine the things you see in your training room and notice how most objects you see are an assembly of individual parts. Try to locate some part that is used by itself. There might be a few, such as a rubber eraser or a ruler, but you probably won't find many. Look for assemblies. Most of what you see will be assemblies.

When we put together an assembly in SolidWorks, the assembly will follow the same general rules as creating an assembly in the physical world. We will first put together the sub-assemblies, then we will add sub-assemblies together until we get to the final product.

Think of the way an automobile is assembled. All the sub-components of the car are assembled by the various manufacturers around the world. Things like the audio system, transmission, alternator, seats and head lamps. When the car is assembled on the assembly line, they start with the frame because it is the foundation of car, all the other sub-assemblies are somehow attached to the frame.

The assembly of the Mountainboard will follow as similar approach. Three of the subassemblies have already been created, the Wheel Assembly, the Spring Assembly and the Binding. We will put together the remaining sub-assemblies, then create the final assembly. The final assembly will start with the Deck as it forms the foundation of the Mountainboard.

Mates

The various parts and sub-assemblies are connected together with fasteners in the physical world. In SolidWorks we use mating relationships, or just mates for short. While we will add fasteners to our assembly, they do not actually hold the assembly together.

The general philosophy for adding mates will be to add mates that represent the real world fasteners. One limitation is that we do not need to represent all fasteners with mates, we only need enough mates to fix the part as it would be in the actual assembly. Adding additional mates would be redundant and make the assembly harder to solve.

Active Learning Exercises, Part 1 — Assembly

Now that we have created all the individual parts and some sub-assemblies, it is time to put the entire project together.

We have already put together three assemblies, the wheel, the binding and the spring and damper. We will first create an entire axle assembly using the Truck, Axle,

Wheel Assembly, Spring Assembly and some other hardware that will be provided to us. Once together, we can combine this Axle Assembly with the Deck and the bindings to complete the mountainboard.



Task 1— Create the Axle Assembly

The first part of this task is to create a new assembly.

- 1 Click **File**, **New** from the menu and select the **Assembly_MM** template.
- 2 Click **Browse** and locate the part Axle in the Mountainboard folder.
- When the preview image appears with your cursor, move the cursor over the assembly origin until the sicon appears, then click the origin.
- 4 Save the new file as Axle Assembly to the Mountainboard folder.
- **5** Click **Insert Component** [Section on the Assembly toolbar.

- 6 Click **Browse** and locate the part Axle Shaft in the Mountainboard\Parts folder.
- **7** Drop the part into the assembly.



Smart Mates

Mates can be added between components while dragging and dropping them. This method, called **SmartMates**, can be done in two ways. First is to drag the appropriate vertex, edge or face of the part from an open part window onto the corresponding vertex, edge or face in the assembly window. The second method uses the **Alt** key in conjunction with standard drag and drop techniques.

These mates use the same **Mate Pop-up** toolbar as the **Mate** tool uses to set the type and other attributes. All mate types can be created with this method.

Certain techniques generate multiple mates and do not use the toolbar. These require the use of the **Tab** key to switch mate alignment.

Mating Entities	Type of Mate	Pointer
2 linear edges	Coincident	
2 planar faces	Coincident	And A
2 vertices	Coincident	<i>ی</i> ۲ 0-0
2 conical, or 2 axes, or 1 conical face and 1 axis	Concentric	<i>ی</i> ر ۱۹۰۵
2 circular edges	Concentric and Coincident	1 /3 •

and the second

Task 2 — Mate the Axle Shaft

- 1 Add a **Concentric** mate using the Smart Mate technique as follows:
 - Click and hold the cylindrical face of the axle Shaft.
 - Press and hold the **Alt** key as you drag the component.
 - Move the component over the cylindrical face of the axle.
 - Drop the part when the **Concentric** pointer **** = appears, indicating a concentric mate.
 - Confirm the **Concentric** type from the **Mate Pop-up** Toolbar.
 - A **Concentric** mate is added between the Axle Shaft and the Axle parts.
- 2 When assembled, the Axle Shaft should slide into the Axle until it bottoms. Change the view orientation to the Front view and zoom in on the axle.
- 3 Change the display to **Hidden**

Lines Visible by clicking in on the View toolbar.



4 Click Mate Solution on the Assembly toolbar and select the edge of the Axle Shaft and the conical face

shown. Click ✓ to accept the **Coincident** mate.

5 Hold down the **Control** key and drag another copy of the Axle

Select

Shaft from the FeatureManager design tree into the graphics area. This will insert another instance of the Axle Shaft part into the assembly.

- 6 Apply the same **Concentric** and **Coincident** mates this shaft.
- 7 **Open** the part King Pin Sleeve from the Mountainboard\Hardware folder.
- 8 Tile the two windows vertically by clicking **Windows**, **Tile Vertically** from the menu.

9 Drag the edge of the King Pin Sleeve shown to the edge of the Axle indicated. When you are on the correct edge the cursor will change to the Pin in Hole .



10 Press the **Tab** key to reverse the orientation of the mates. When the cursor shows the **Pin in Hole** and the orientation is correct, drop the King Pin Sleeve. It will receive both a **Concentric** and **Coincident** mate.



11 Finish the assembly by adding two Socket Set Screws. These set screws are used to keep the Axle Shafts from backing out of the Axle. The set screws are located in the Mountainboard\ Hardware folder as Socket Set Screw Cup Point_AM.

Mate each set screw using one **Concentric** and one **Coincident** mate.

12 Save the assembly.



Task 3 — Create the Truck Assembly

- 1 Click File, New from the menu and select the Assembly_MM template.
- 2 Click **Browse** and locate the part Truck in the Mountainboard folder.
- When the preview image appears with your cursor, move the cursor over the assembly origin until the pointer appears, then click the origin.
- 4 Save the new file as Truck Assembly to the Mountainboard folder.
- **5** Click **Insert Component** [Second to on the Assembly toolbar.
- 6 Click **Browse** and locate the part 8mm Threaded Insert in the Mountainboard\Parts folder.
- **7** Drop the part into the assembly.
- 8 These inserts will actually be molded into the Truck, so there will be an exact fit. We need three coincident mates to position the insert.



9 Mate the Insert to the Truck with three Coincident mates. Mate two adjacent faces on the insert to two adjacent faces on the Truck. The third mate will be between the top of the Insert and the top face of the Truck.



10 Insert three more instances of the 8mmThreaded Insert into the assembly and mate them to the three remaining hex holes.



Mate References

Mate References allow you to realize the benefits of SmartMates without the requirement of having the part you want to mate open. By identifying a face, edge or vertex in the part as the mate reference, you can use SmartMates while dragging and dropping the part from Windows Explorer, File Explorer or the Design Library.

Primary, Secondary, and Tertiary References

When you insert a part with a mate reference, the software identifies potential mate partners for the specified entity. If the primary entity is not valid for the entity your pointer is over, then the secondary entity is used. If neither the primary nor secondary entities are valid, then the tertiary entity is used.

As you move the cursor in the assembly window, the pointer changes and the preview snaps into place when a potential mate partner is found.

Task 4 — Add Mate References to the **Truck** bearing.

- **1 Open** the part Bearing from the Mountainboard\Parts folder.
- 2 Click Insert, Reference Geometry, Mate Reference.
- **3** Select the edge and two faces as the Primary, Secondary and Tertiary references.
- 4 Click 🗹 .



- 5 The MateReferences will be listed in the FeatureManager design tree.
- 6 Save and Close the part.
- Drag a Bearing from the Mountainboard
 Parts folder in the File Explorer to the edge shown The Bearing will snap into position and both a Coincident and Concentric mate will be created. If the



→ Origin → MateReferences → 100 Default-<1> → 100 Extrude1 → 100 Cut-Extrude1 → 100 Mirror1

Bearing is sticking out of the hole, press the **Tab** key to reverse the alignment.

- 8 There are two configurations of this bearing, select the 28mm OD x 12mm ID and click OK.
- **9** After you have placed the first Bearing, add a second bearing to the other side of the **Truck**.
- 10 Save the assembly as Truck Assembly.



Task 5 — Assemble the entire wheel assembly

We now have all the sub-assemblies for the mountainboard.



We will first put together all the sub-assemblies that make up the wheel assemblies. Then we will add the wheel assemblies and bindings to the deck to complete the mountainboard.



- 1 Create a new assembly using the Assembly_MM.asmdot template.
- 2 If the Axle Assembly is not open, click **Browse** and locate the Axle Assembly in the Mountainboard folder.
- **3** Drop the Axle Assembly on the origin of the new assembly.
- 4 **Save** the new assembly to the Mountainboard folder as Truck_Axle_Wheel.sldasm.
- 5 Insert two instances of the Spring Assembly into the Truck_Axle_Wheel assembly.

Suspension Adjustments

The springs and dampeners can be installed in two locations to adjust the ride of the mountainboard. To create a stiffer ride, they are mounted in the set of holes further away from the center. This does two things, first is to compress the spring further as the angle between the Deck and Axle increases. This creates more force than if the springs were mounted closer to the center. Second, being further away from the center, the lever arm is longer which creates a larger restoring moment.



- 6 Mate each of the Spring Assemblies to the outer holes in the Axle. The end of the Spring Assembly with the Fender Washer should be up.
- Drag the Truck Assembly into the Truck_Axle_Wheel assembly.
- Mate to this hole

Fender washers

- 8 Add a **Concentric** mate between the hole in the Bearing and the King Pin Sleeve.
- 9 Add a **Coincident** mate between the two faces show to position the axle the Truck.





Task 6 — Add the remaining hardware

While our CAD model is held together with mating relationships, we need to add the remaining fasteners that will hold the real model together. The parts provided have mate references so that we can drag and drop them from the Design Library to our assembly.

1 Add the King Pin and the B18.2.2.4M Hex flange nut to the assembly. Select B18.3.3M-10x80 SHSS--N for the King Pin configuration. Both parts can be found in the Mountainboard\Parts folder.



Add, from Toolbox, two B18.3.5M 4x0.7x10 Socket FCHS-10N screws to hold the bottom of the spring assemblies to the axle.



- 3 **Open** the part Adjusting Screw from the Mountainboard\Parts folder. This part is used to preload the springs to further adjust the ride of the mountainboard. There are two configurations of this part, the Default configuration that has all the threads modeled and a Simplified configuration without the threads. The Default configuration will take considerably longer to rebuild because of all the additional surfaces that must be calculated.
- 4 Drag two instances of the Simplified configuration of the Adjusting Screw into the Truck_Axle_Wheel assembly.



Default

Simplified

5 Mate each Adjusting Screw with aConcentric and Distance mate as show.Set the distance mate to 10mm.



Task 7 — Add the Wheel Assembly

To complete this assembly, we only need to add two wheel assemblies and a lock nut on each axle shaft.

- 1 Click Insert, Component, Existing Part/Assembly from the menu.
- 2 Click **Browse** and locate the Wheel Assembly. Insert two instances of the Wheel Assembly into the Truck_Axle_Wheel assembly.
- **3** We will need two mates to position each of the wheel assemblies. Add a **Concentric** mate between the inside face of one of the wheel bearings and the cylindrical face of the Axle Shaft.

Note: Make sure that the valve stems are facing out.

- 4 The Axle Shaft has a step face. When assembled, the wheel is pushed onto the shaft until the face of the bearing is stopped by the step.
- 5 Add a Coincident mate between this stepped face and the side of the bearing.



TIP: To select the stepped face, you will have to zoom in close as the face is only .5mm wide.

- 6 Finish the assembly by adding a B18.2.4.5M-Hex jam nut, M10 x 1.5 to each axle Shaft to hold on the wheels.
- **7** Save the assembly.



Task 8 — Clean up the FeatureManager design tree

The FeatureManager design tree shows the assembly as being made up of six sub-assemblies and eight fasteners. We can make the FeatureManager design tree easier to understand if we group all the fasteners together in a folder.

- 1 Right-click any of the fasteners in the FeatureManager design tree and select Add to New Folder.
- 2 Name the folder Fasteners.
- **3** Drag each of the fasteners into this folder. Drag the part to the folder.

When the cursor changes to \Leftarrow you can drop the part and it will go into the folder.

- 4 The FeatureManager design tree is now easier to understand. We can collapse the folder by clicking the minus sign. This makes the FeatureManager design tree even shorter.
- 5 Save the assembly.

🎨 Truck_Axle_Wheel (Default <default_display state-1="">)</default_display>
Annotations
🕀 🙀 Lights, Cameras and Scene
🛁 🗼 Origin
🖬 🧐 (f) Axle Assembly<1> (Default <default_display state-1="">)</default_display>
亩 🧐 (-) Spring Assembly<1> (Exploded <display state-1="">)</display>
亩 🧐 (-) Spring Assembly<2> (Exploded <display state-1="">)</display>
Truck Assembly <1> (Default <default_display state-1="">)</default_display>
🚋 🌇 (-) King Pin<1> (B18.3.3M - 10 × 80 SHSSN)
⊕
⊕
😐 🍟 (-) B18.3.5M - 5 × 0.8 × 10 Socket FCH5 10N<3>
🙃 🌇 (-) Adjusting Screw_&<1> (Simplified)
🙃 🈘 (-) Adjusting Screw_&<2> (Simplified)
💼 🧐 (-) Wheel Assembly<1> (Default <default_display state-1="">)</default_display>
💼 🧐 (-) Wheel Assembly<2> (Default <default_display state-1="">)</default_display>
🖶 📅 (-) Hex Nut Jam_AM<2> (B18.2.4.5M - Hex jam nut, M10 x 1.5, with 16mm WAFD-N)
① _ 例例 Mates

Truck_Axle_Wheel (Default <default_display state-1="">)</default_display>
Annotations
🔄 😡 Lights, Cameras and Scene
🚽 💭 Origin
🕣 🧐 (f) Axle Assembly<1> (Default <default_display state-1="">)</default_display>
🗄 🧐 (-) Spring Assembly<1> (Exploded <display state-1="">)</display>
🗄 🧐 (-) Spring Assembly<2> (Exploded <display state-1="">)</display>
Truck Assembly <1> (Default <default_display state-1="">)</default_display>
⊕ 🧐 (-) King Pin<1> (B18.3.3M - 10 × 80 SHSSN)
📮 🦲 Fasteners
🕀 👒 (-) Adjusting Screw_&<2> (Simplified)
🖶 👒 (-) Adjusting Screw_&<1> (Simplified)
🖶 📅 (-) Hex Nut Jam_AM<1> (B18.2.4.5M - Hex jam nut, M10 x 1.5, with 16mm WAFD-N)
🖶 📲 (-) Hex Nut Jam_AM<2> (B18.2.4.5M - Hex jam nut, M10 x 1.5, with 16mm WAFD-N)
⊕ 📅 (-) B18.2.2.4M - Hex flange nut, M6 x 1N_&<1> (PreviewCfg)
⊕ 🕎 (-) B18.3.5M - 5 × 0.8 × 10 Socket FCHS 10N<1>
⊕ 🕎 (-) B18.3.5M - 5 × 0.8 × 10 Socket FCHS 10N<2>
⊕ 📅 (-) B18.3.5M - 5 × 0.8 × 10 Socket FCHS 10N<3>
🖶 🎨 (-) Wheel Assembly<1> (Default <default_display state-1="">)</default_display>
⊕ 🎨 (-) Wheel Assembly<2> (Default <default_display state-1="">)</default_display>
亩 III Mates

Task 9 — Create an exploded view

Using the skills developed in the previous lessons, create a configuration named Exploded and an exploded view of the assembly.

When you are done creating the exploded view, collapse the view.

Animations of the explode and collapse sequence are included in the LessonO8\Built Parts folder.



Task 10 — Create a drawing of the Wheel Assembly

The procedure to create an assembly drawing is the same as creating a drawing of a part.

Bill Of Materials

Assembly drawings may also contain a Bill of Materials which is a list of the parts and sub-assemblies. The Bill of Materials, called a BOM for short, may have different columns to list additional information about each part or sub-assembly. The most common information would be an item number, quantity and description. Other information that may be included are: material, weight, cost, vendor, or stock size.

Balloons

Balloons are used to identify parts and sub-assemblies in the drawing. The numbers in the balloons correspond to the item number in the Bill Of Materials.

Balloons can be formatted to show the item number, quantity or a custom property.



1 Before we create the drawing, we will set an option that will control the size of new views in the drawing. Click **Tools, Options** and select the **System Options** tab.

Select Drawings and then clear Automatically scale new drawing views.

If **Automatically scale new drawing views** is selected, SolidWorks will override the scale of the drawing template to make the drawing view fit. In our case, we want to have the drawing views use the scale of the drawing template.

System Options - Drawings	
System Options Document Prop	erties
General Drawings — Display Style — Area Hatch/Fill Colors Sketch — Relations/Snaps	Eliminate duplicate model dimensions on insert Eliminate duplicate model notes on insert Mark all part/assembly dimension for import into drawings by default Automatically scale new drawing views Show contents while dragging drawing view Display new detail circles as circles

- 2 Click **OK** to close the **Options**.
- 3 With the Truck_Axle_Wheel assembly still open, click Make Drawing from Part/ Assembly and toolbar.
- 4 Select the Drawing template B-Scale1to4.

5 In the Task Pane, select the Isometric view, then drag it onto the drawing.

Click 🗹 .

- 6 We want to show the exploded view instead of the collapsed view. Rightclick Drawing View1 in the FeatureManager design tree and select **Properties**.
- 7 Select Use named configuration: and select Exploded from the list.

Select **Show in exploded state**. This will change the view from collapsed to exploded.

Note: Show in exploded state does not create an exploded view, it only displays the exploded view if one exists in the selected configuration. You must create the exploded view manually.

Click OK.

- 8 Position the view as shown by dragging its border.
- 9 Select the view and then click Insert, Tables, Bill of Materials.



Drawing View Pro	perties		8
View Properties	Show Hidden Edges	Hide/Show Components	Hide/Show Bodie
View informati	on		
Name: Drawin	ng View1	Type: Named View	
Model informa	tion		
View of: Document:	Truck_Axle_Wheel_ C:\SolidWorks Curric	dulum_and_Courseware_20	11\Mountain
Configuration	information 's "in-use" or last save	d configuration	
Explode	coloded state		•
Display State	Display State-1		•
Balloons	on text to specified tabl	Show Envelo Align breaks Display shee Display boun	with parent t metal bend notes ding box
		OK Cancel	Help


Lesson 8: Final Assembly

- **10** Select the following choices:
 - Table Template: bom-standard
 - Table Anchor: Attach to anchor
 - BOM Type: Indented, No numbering
 - Configurations: Explode

Click У .



11 The Bill Of Materials will be inserted so that it fits into the upper right corner of the drawing. If the table is located to the wrong side of the upper right corner, select the BOM table and then change the alignment in the PropertyManager.





If it is too wide it may cover part of the drawing view. Adjust the column widths by dragging the column borders. When the cursor is over a column border it will change to the pointer shown below.



- 12 Examine the BOM. Each sub-assembly and independent part has its own item number.
- 13 Select the drawing view and click AutoBalloon \mathcal{P} on the Annotations toolbar.

- 14 Select **Bottom** 😹 for Balloon Layout.
- 15 From the pull-down lists in the PropertyManager, select:
 - Style: Circular Split Line
 - Size: 2 Characters
 - Balloon text: Item Number
 - Lower: Quantity

- **16** Drag the balloons to a position that is centered under the drawing view.
- 17 Click 🗹



18 Save and Close the drawing.

Task 11 — Add the bindings to the deck

We only created one binding for the left foot. We could make a right footed version of each part individually, or we can create all the right handed parts in a single operation in the assembly.

- 1 Create a new assembly using the Assembly_MM template.
- 2 Insert the Deck part and mate it to the origin.
- **3** Save the assembly as Mountainboard.sldasm. This will be our top level assembly.



- 4 Add the Binding assembly. Click **Insert Components** [29] and then **Browse**. Locate the Binding assembly in the Mountainboard\Binding folder and click **Open**.
- 5 Drop the Binding assembly into the Mountainboard assembly. There are no **SmartMates**, so you can place the Binding anywhere.
- 6 We will create all the mates between the Deck and the Binding Anchor. This is consistent with the way the actual binding works.



Because the end face of the slot is a cylinder (even thought we only see half of it) we can have two alignments. While we cannot see it, when SolidWorks created the end faces of the slot, it created full cylinders. Part of the cylinder is trimmed away so that we only see the end face.



As the hole is also a cylinder, the two alignments are created by mating to either the near or far faces.



TIP: It will be easier to select the proper faces if you **Hide** the Strap Assembly.

Task 12 — Align the Binding Assembly

When a sub-assembly is inserted into another assembly, it behaves as if it were welded together. Because we added mates between the Binding Anchor and the Deck, the angle between the Binding and the Deck will be whatever angle exists between the Binding Base Plate and the Binding Anchor. To change this angle, we have two options:

- Change the angle in the Binding assembly
- Make the Binding assembly Flexible in the Mountainboard assembly

For the first binding, we will change the angle in the Binding assembly.

Lesson 8: Final Assembly

- 1 In the FeatureManager design tree, right-click the Binding assembly and select **Open Assembly**. The Binding assembly will open in its own window.
- 2 Add an **Angle** mate between the Right plane of the Binding Base Plate and the Right plane of the Binding Anchor. Set the angle to 60°.
- 3 Click either Aligned or Anti-Aligned to get the orientation the same as shown.

Note: The Strap Assembly has been hidden to make it easier to see the angle.

4 Use the Window menu to return to the Mountainboard assembly. The Binding should be oriented as shown.





Mirroring Components

Many assemblies have some degree of left-right symmetry. Components and subassemblies can be mirrored to reverse their orientation. This can also generate "opposite hand" parts.

When you mirror components in an assembly, they fall into two categories:

- □ Those parts whose orientation in the assembly is mirrored and whose geometry is also mirrored they have right and left-hand versions.
- □ Those parts whose orientation in the assembly is mirrored but whose geometry is not hardware, for example.

Mirror Components

Mirror Components allows you to generate an "opposite hand" component or subassembly at the assembly level. Options allow for simply reversing or mirroring components.

Task 13 — Insert a mirrored Binding

The second binding will be a mirror image of the first binding. Some of the parts will need to be mirrored, others will not.

Mirrored Parts

- Binding Base Plate
- Binding Pad
- Strap_right
- Strap_left
- Foam_Pad

Parts Not Mirrored

- Binding Anchor
- Clasp Assembly
- 1 Click Insert, Mirror Components from the menu.
- 2 Select the Right plane of the Deck as the Mirror plane.
- 3 Select the Binding in the fly-out FeatureManager design tree for the **Components to Mirror**.
- 4 Click Next 🗐
- **5** Expand the listing in the **Components to Mirror** section of the PropertyManager by selecting each plus sign. We will select those components that we want to be mirrored.
- 6 Select everything that gets mirrored.

Select each of the following components in the **Orient Components** box and then click **Create opposite hand version**:

- Binding
- Binding Base Plate
- Strap Assembly
- Strap_right
- Strap_left
- Foam_Pad
- Binding Pad
- 7 As the components are selected we can see a preview to determine if the parts are correct.







- **9** The mirrored components are new parts that must be named. The default is to add Mirror as a prefix to the existing file names so that the mirrored assembly of Binding will be MirrorBinding.
- 10 Click 🖌 .



11 Some of the mates could not be created automatically. We will have to fix this manually. Notice that all the mates listed are in subassembles of the Binding.

Click **OK**.



- 12 The mirrored binding has been created.
- 13 Try to move the new binding. The new mirrored Binding does not move because subassemblies are rigid by default. Being rigid means that all the components maintain a fixed position relative to the other components.



14 Open the MirrorBinding in its own window. Notice that there are only four mates. The Binding assembly has seven mates. If you examine the mates for the Strap Assembly and the MirrorStrap Assembly, you will see that the Strap Assembly has three mates while the MirrorStrap Assembly has none.

As long as we are in the top level assembly (mountainboard), the missing mates will not cause a problem, however for completeness, the missing mates need to be added back into the mirrored assemblies.

15 Add the appropriate mates between the MirrorBinding Anchor and the MirrorBinding Base Plate that were used to mate the original Binding. Also add the appropriate mates in the MirrorStrap Assembly to properly position the Clasp Assembly.

Task 14 — Add the wheel assemblies

The last major assembly to be added will be the wheel assemblies. They will be mated to the Deck using mates similar to the effects of using fasteners. The fasteners used to hold the wheel assemblies to the Deck would pull the face of the Truck in contact with the underside of the **Deck**. This is a **Coincident** mate. The fasteners also line up the holes in the Truck with the holes in the Deck. While there will be four bolts used to mount each wheel assembly on the real Mountainboard, we only need to add two **Concentric** mates to hold the alignment. Any additional mates would just be redundant.

1 Insert one instance of the Truck_Axle_Wheel assembly into the Mountainboard assembly.

2 Mate the top face of the Truck to the mounting face on the bottom of the Deck with a **Coincident** mate as shown.



3 Add Concentric mates between the two holes in the Truck and the corresponding holes in the Deck. Make sure the Truck is oriented so that the rounded face is near the end of the Deck.



Note: If you get an mate error when adding the second **Concentric** mate, check the positioning dimensions of the holes both the Deck and the Truck. They must be exactly the same or the two **Concentric** mates will fight for control.

- 4 Add another instance of the Truck_Axle_Wheel assembly to the Mountainboard assembly and mate it to the other end of the Deck.
- **5** Save the assembly.

Task 15 — Add fasteners

The only remaining tasks to complete the assembly are to add the fasteners that hold the Bindings and Trucks to the Deck. Except for specialized fasteners, we generally purchase fasteners and other common components for existing suppliers.

In addition to Toolbox, we have additional resources to locate



components that we would buy instead of manufacture. To attach the Bindings to the deck, we need a special type of nut called a T-nut which will not extend significantly below the bottom of the Deck. There are no T-nuts in Toolbox, so we will find one in 3D ContentCentral[®].

3D Content Central[®]

3D ContentCentral provides access to 3D models from component suppliers and individuals in all major CAD formats. These models can be downloaded and saved locally. Many suppliers provide both 2D and 3D models.

We can access 3D ContentCentral through the Design Library in the Task Pane. You must have internet access to use 3D ContentCentral.

Note: In the next few steps, we will use 3D ContentCentral to download a fastener. If you do not have internet access, the part is provided for you in the Mountainboard\Hardware folder.

1 In the Design Library, click the plus sign next to 3D ContentCentral.



The first time you logon to 3D ContentCentral you will be asked to provide an email address and accept the licensing agreement.

2 Click the plus sign next to User Library, then click Home Page. In the lower pane, click Click here for User Library to go to the start page.



3 Click Hardware, then click Nuts.

4 There are several pages of nuts, locate the T-nut shown.

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	tro//www.3dcontentral.com/oartr/supplier/lises_libran/8/M/M/Models.acm/title=Nutr + D to + Y 20 Google	0 -
		_ <u>88</u> >>
	NUIS: Parts & Assemblies	. 😅
MS35650-304	Title: MS35650-304 By: Torv Suarez Description: NUT, HEX, #10-32 Company:EMTEQ Engineering Category: Hardware: Nuts Supplier name: Supplier name: QPL Catalog: User: Lbrary Supplier part number:	
12/11/2008 Multiple Configur Downloads: 4818 (1 Ratings) Contributed on: 6/28/2002	Tations T-nut Title: [b]Don Hosteller [br]Design Engineer [br]Wieland Designs, Inc. [b] [br] [br] [d]T-nut By: Don Hosteller utilizing sheet metal bends and solid modeling. Uses design table for several size Company: Weland Description:: T-rut utilizing sheet metal bends and solid modeling. Uses design table for several size configurations. Designs, Inc. Category: Hardware Nuts Catalog: User Ubrary	
Downloads: 97 (3 Ratings) Contributed on: 12/23/2010	Title:Mutter DIN-934 By:Georg Staudinger Description:Mutter DIN-934 Company:Staudinger Metallou GmbH Dimensionen: M2.5 - M16 Supplier part number:Mutter DIN-934 Category:Edware: Nuts Catalog: User Library	E
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http://www.3dconten	tcentral.com/parts/supplier/User-Library/8/44/44/Models/part. 😜 Internet Protected Mode: On 🆓 👻 🔍 100)% 🔻

Note: 3D ContentCentral is continually being expanded so you may have to search the listing for this specific T-nut.

5 Click on the image of the T-nut.

This T-nut comes in several sizes, select the size .25 x .313 std. head from the list.

Notice the three images of a mouse near the bottom of the window. These show you which button or buttons to press to rotate, zoom or pan the model.

[b]Don Hostetler[br]Design Engineer[br]Wieland Designs, Inc.[/b][br][d]T-	nut utilizing shee - Windows	Internet Explorer		X	_
G v ★ http://www.3dcontentcentral.com/parts/supplier/User-Librar	y/8/44/44/Models/part.aspx?	d=11 🔻 🗟 😽 🗙 🚼 G	oogle	م	•
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Home > User Library > Hardware > Nuts > [b]Don Hostetler[br]Design Engineer[br]Wiela	nd Designs, Inc.[/b][br][br][d]T	nut utilizing sheet metal bends a	nd solid modeling. U	ses design table for several size	
[b]Don Hostetler[br]Design Engineer[br]Wieland Designs, Inc.[/b][br][br][d]T-nut utilizing sheet metal bends and solid modeling. Uses design table for several size configurations.	Modified on: August 12,2002 Downloads: 4818	Ra Co Alt	nting: 🗙 🗙 🗙 🗙 mments: 1 ternate Versions:	Post Alternate Version	E
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- 6 Select Configure & Download.
- 7 Select .25 x .313 std. head for the size.
- 8 Select SolidWorks Part/ Assembly for the format and 2011 for the Version.
- 9 Clear the options to download as a Zipped file and to download all configurations.

Configure	Download
Change the options below to customize the model for downloading. Click the Update Preview button to apply your changes to the 3D or 2D viewer on the left. 	Download the model according to the specified sizing parameters in either 3D or 2D format. Format: ③ 3D ④ 2D Format: SoldWorks Part/Assembly (*.sldprt) ▼ Version 2011 ▼ ☑ Zipped ☑ Download all configurations
	Remind me to rate this model

10 Click **Download**.

11 Drag the Toolbox icon to the SolidWorks graphics area.



- 12 Save the part to the Mountainboard\Hardware folder.
- 13 Drag three more instances of the T-nut into the assembly.
- 14 Close 3D Content Central.
- 15 Using SmartMates, drag the edge shown, while holding down the Alt key, into the proper hole. Use the Tab key to reverse the direction of the T-nut so that the shaft goes into the hole.
- **16** Reorient the model so you can see the tops of the four T-nuts.



Note: It will be easier to insert the screws in the next step if you hide the Binding Strap assembly.

- 17 Click the pushpin in the Design Library to keep it open. Expand Toolbox, then Ansi Inch, then Bolts and Screws by clicking the plus sign next to each. Select Machine Screws.
- 18 Hold down Control and select the top circular edge of each of the four T-nuts. In the lower pane of the Design Library, right-click the Truss Head Screw and select Insert Into Assembly.



- 19 Select:
 - Size 1/4-20
 - Length .625
 - Drive Type Cross
 - Thread Length 0.625
 - Thread Display Simplified

Click 🖌 .

All four Truss Head Screws will be inserted.

20 Add T-nuts and Truss Head Screws to the MirrorBinding.

Note: When you insert the Truss Head Screws you may be asked if you would like to create new copies or to use the existing copies of the parts, use the existing parts.



21 Use Toolbox to add 8 - 32 x 1.125 inch Truss Head Screws and 8 - 32 Machine Screw Nut Hex to the eight locations to hold the Truck to the Deck.



Task 16 — Create a hardware folder

The FeatureManager design tree is getting relatively long. We can add all the hardware to a new folder to make it easier to find things.

- 1 Select all the hardware in the FeatureManager design tree.
- 2 Right-click any of the selected pieces of hardware and select Add to New Folder.
- 3 Name the new folder Hardware.
- **4 Save** the assembly.

Active Learning Exercise, Part 2 — Information From The Assembly

Now that the Mountainboard is assembled, we can get several pieces of information from the assembly.

AssemblyXpert

Information can be extracted from an assembly to determine some of its parameters such as size, depth and references. For statistics on the quantities of certain types of part components and sub-assemblies, **AssemblyXpert** can be used.

Find References

Find References can be used to extract the exact locations of component part and assembly files. The listing provides a full path name for each reference used. The **Copy Files** button can be used to copy the files to another, common, folder.

Task 1— Determine the size of the assembly

1 Examine the size of the Mountainboard assembly. Click **Tools**, **AssemblyXpert** from the menu.

- 2 Assembly Statistics show us that there are 183 total components in our assembly. This is the total of all parts and sub-assemblies
- There are 163 parts which means that there must be 183
 163 = 20 sub-assemblies. This is shown in the fifth line.
- 4 There are 41 unique parts. The difference between this and total number of parts (183) is a result of using many of our parts, most fasteners for example, more than once in the assembly.
- 5 Click **OK** to close the **Assembly Statistics**.



6 Click **File**, **Find References** to list the references and full path locations of the components used in the assembly.

	📝 Indude broken references 🛛 💿 Nested view 🖉 Flat view
ame	In Folder
🏽 Mountainboard_&.SLDASM	C:\SolidWorks Curriculum_and_Courseware_2011\Mountainboard Design Project_Teacher\Mountainboard-complet
🔖 Deck_&.SLDPRT	C:\SolidWorks Curriculum_and_Courseware_2011\Mountainboard Design Project_Teacher\Mountainboard-complet
Binding_&.SLDASM	C:\SolidWorks Curriculum_and_Courseware_2011\Mountainboard Design Project_Teacher\Mountainboard-complet
😵 Binding Base Plate_&.SLDPR	C:\SolidWorks Curriculum_and_Courseware_2011\Mountainboard Design Project_Teacher\Mountainboard-complet
Binding Anchor_&.SLDPRT	C:\SolidWorks Curriculum_and_Courseware_2011\Mountainboard Design Project_Teacher\Mountainboard-complet
Strap Assembly_&.SLDASM	C:\SolidWorks Curriculum_and_Courseware_2011\Mountainboard Design Project_Teacher\Mountainboard-complet
■ Strap_right_&.SLDPRT	C:\SolidWorks Curriculum_and_Courseware_2011\Mountainboard Design Project_Teacher\Mountainboard-complet
🕵 Binding Start Sketch_&.	C:\SolidWorks Curriculum_and_Courseware_2011\Mountainboard Design Project_Teacher\Mountainboard-complet
Binding_&.SLDASM	C:\SolidWorks Curriculum_and_Courseware_2011\Mountainboard Design Project_Teacher\Mountainboard-complet
□ Strap_left_&.SLDPRT	C:\SolidWorks Curriculum_and_Courseware_2011\Mountainboard Design Project_Teacher\Mountainboard-complet
🕵 Binding Start Sketch_&.	C:\SolidWorks Curriculum_and_Courseware_2011\Mountainboard Design Project_Teacher\Mountainboard-complet
Binding_&.SLDASM	C:\SolidWorks Curriculum_and_Courseware_2011\Mountainboard Design Project_Teacher\Mountainboard-complet
🖃 🍕 Foam_Pad_&.SLDPRT	C:\SolidWorks Curriculum_and_Courseware_2011\Mountainboard Design Project_Teacher\Mountainboard-complet

- 7 Examine the list. All the files should be in the SolidWorks Curriculum and Courseware_2011\Mountainboard Design Project\
 Mountainboard folder or one of its sub-folders. If we had made a mistake and saved one or more of the files to other location, or we did not setup Toolbox correctly, we could use Copy Files to copy all the files used in the Mountainboard assembly to a common location.
- 8 Click **Close** to close the Search Results.

Task 2 — Check the weight of the assembly

We can use the **Mass Properties** tool to get the final weight of the entire Mountainboard assembly. Before we do however, we need to make sure that there is a correct material assigned to each of the 43 unique parts.

If we have been adding material to the parts as they are created, there should not be too many that do not have material already assign. To check and assign the materials, we do not have to leave the Mountainboard assembly, we can add the material while we are still working with the assembly.

Edit Component - Review

While you are in an assembly, you can switch between editing the assembly — adding mate relations, inserting components, etc. — and editing a specific part. Two commands, **Edit Part** and **Edit Assembly**, are used to switch back and forth between editing one component in an assembly and editing the assembly itself. When you are in **Edit Part** mode, you have access to all the commands and functionality the part modeling portion of SolidWorks.

To edit a part while in the assembly:

- □ Click **Edit**, **Part** from the menu.
- □ Or, from the right-mouse menu, select Edit Part or Edit Assembly.
- Or, from the Assembly or Context toolbars, click Edit Component [Solar]

- 1 Check each part in the Mountainboard assembly to make sure it has the proper material assigned as shown in the table below.
- 2 To add or change a material, right-click the part in the FeatureManager design tree and select **Edit Part** from the menu.

3 After editing the material, return to the **Edit Assembly** mode by selecting the part and clicking the **Edit Component** (s) tool.

Component	Material
Deck	Acrylic (Medium-high impact)
Binding	
Binding Base Plate	2014 Alloy
Binding Anchor	2014 Alloy
Strap Right	Rubber
Strap Left	Rubber
Foam Pad	PP Copolymer
Binding Pad	Rubber
Clasp Assembly	
Clasp-1	Chrome Stainless Steel
Clasp-2	Chrome Stainless Steel
Claps-pin	Chrome Stainless Steel
Axle Assembly	
Axle	6061 Alloy
Axle Shaft	Alloy Steel
King Pin Sleeve	Alloy Steel
Spring Assembly	
Spring	Chrome Stainless Steel
Spring Dampener	PE High Density
Spring Retainer	ABS PC
Fender Washer	Alloy Steel
Truck Assembly	
Truck	Nylon 6/10
Bearings	Chrome Stainless Steel
Threaded Insert	Chrome Stainless Steel
Wheel Assembly	
Wheel Hub	PVC Rigid
Inner Tube	Rubber
Tire	Rubber
SKF-6001 Bearing	Chrome Stainless Steel
Hardware	
All fasteners	Alloy Steel

Note: Remember that several of the fasteners are located in subassemblies, insure that all the fasteners are Alloy Steel.

- 4 Once all the materials are applied, we can check the weight of the finished Mountainboard. Click Tools, Mass Properties from the menu.
- **5** The mass of the assembly is 8,946.489 grams or 8.95kg which meets our original design intent.

🚳 Mass Properties	x
Print Copy Close Options Recalcul	ate
Output coordinate system: default	•
Mountainboard_&.SLDASM Selected items:	
☑ Include hidden bodies/components	
Show output coordinate system in corner of window	
Assigned mass properties	
Mass properties of Mountainboard_& (Assembly Configuration - Default)	*
Output coordinate System: default	
Mass = 8946.684 grams	Ξ
Volume = 5835333.489 cubic millimeters	
Surface area = 2547945.651 square millimeters	
Center of mass: (millimeters) X = -0.008 Y = 9.349 Z = 0.185	
•	

6 To check the mass in other units, click **Options**.

Select **Use custom settings** and choose **Inches** and **Pounds** from the lists.

Click OK.

The weight of the finished mountainboard is 19.724 pounds.

Click Close.

Mass/Section Property Options
Units
Scientific Notation
Use document settings
 Use custom settings
Length: Decimal places:
Inches 🔻 3
Mass:
pounds 💌
Per unit volume:
millimeters^3 🔹
Material Properties
Density: 0.001 g/mm^3
Accuracy level
Lower (faster) Higher (slower)
OK Cancel Help

Active Learning Exercises, Part 3 — Create an Exploded View

Task 3 — Create an exploded view

Create an exploded view of the entire Mountainboard. The end result will be to explode the components on the front of the Mountainboard and leave the components on the rear as assembled. This will allow us to see most components exploded as well assembled.

Exploded views created within sub-assemblies can be imported and reused. As we have already created exploded views of the Truck_Axle_Wheel assembly and the Binding,



we can use them to simplify the process.

- 1 Create a new configuration of the Mountainboard called Exploded.
- 2 Change the sub-assemblies for the front Truck_Axle_Wheel and the Binding to their Explode configurations. In the FeatureManager design tree, right-click the assembly instance of the Truck_Axle_Wheel that is positioned at the front of the Mountainboard and select Properties. Select the Exploded configuration and then OK.
- 3 Repeat this procedure for the Binding.
- Insert an Exploded view into the Mountainboard assembly. Click
 Insert, Exploded View from the menu.

General properties			
Component Name: Truc	k_Axle_Wheel_&	Instance Id: 1 Full	Name: Truck_Axle_Wheel
Component Reference:			
Component Description:	Truck_Axle_Whe	el_&	
Model Document Path:	C:\SolidWorks C	rriculum_and_Courseware	_2011\Mountainboard Desig
(Please use File/Replace of	command to replace	model of the component(s))
Display State specific prop Hide Component Referenced Display Sta	erties		
<explode>_Ap</explode>	pearance Display	State	
Change display propertie	sin: This disels		
Change display propertie	s in: This displa	y state 💌	
Change display propertie Configuration specific proy Referenced configuratio	s in: This displa perties m	y state	Suppression state Suppressed Resolved Lightweight
Change display propertie Configuration specific prop Referenced configuratio	s in: This displa perties n	y state	Suppression state Suppressed Resolved Lightweight Solve as Rigid Flexible

- 5 Make sure that **Select sub-assembly's parts** is cleared.
- 6 We only need to move two sub-assemblies, the Binding and the front Truck_Axle_Wheel assembly. Select the Binding assembly
- 7 We want to move the Binding away from the surface of the Deck in a direction normal to that surface. The Move Triad is currently

oriented to the assembly coordinate system, so if we move the Binding along one of the three principal directions, it will not be where we want it.

Aligning The Move Triad - Review

The move triad can be realigned by either right-clicking the center yellow ball and selecting **Align to Component [component name]**, or by dragging the yellow ball onto a planar surface or linear edge.

1 Drag the cyan ball of the move triad to the face of the Deck to which the Binding is mated. The move Triad will realign to this face.

- 2 Use the triad to drag the Binding away from the Deck. The exact position is very subjective, but we want to make sure that it is clear that the Binding has moved away from the Deck. Click anywhere in the graphics area to complete the step.
- 3 Select the front Truck_Axle_Wheel assembly. Right-click the cyan ball in the move Triad and select

Align with Component Origin and select the Truck_axle_Wheel-1 assembly.





Lesson 8: Final Assembly

4 Use the Triad to drag the sub-assembly away from the Deck. We need to move this subassembly further from the Deck than the Binding because we will also explode this sub-assembly. When we explode the subassembly, some parts will move back in the direction of the **Deck**. Click anywhere in the graphics area to complete the step.



Re-use Sub-assembly Explode

When a sub-assembly has had exploded steps created, they can be re-used in a higher level assembly. This can save a considerable amount of work.

To re-use a sub-assembly's explode steps, while creating explode steps, select the subassembly and click **Re-use Sub-assembly Explode**.

- 5 Select the front Truck_Axle_Wheel assembly.
- 6 Click **Re-use Sub-assembly Explode**. All the steps of the sub-assembly explode will be recreated.
- 7 Use the Fly-out FeatureManager design tree to select front right Wheel Assembly.



8 Click **Re-use Sub-assembly Explode**. This explodes a sub-assembly of the sub-assembly.



- **Note:** We had to select the Wheel **Assembly** using the FeatureManager design tree rather than the graphics area to make sure we just selected the Wheel Assembly. If we tried to select the Wheel Assembly in the graphics area, we would have selected the Truck_Axle_Wheel assembly instead.
- **9** Complete the exploded view of the Mountainboard by exploding the front Binding and the fasteners that hold the front Binding and Truck.
- **10** Adjust the positions of individual parts or assemblies for clarity.
- **11 Save** the assembly.



Task 4 — Create the assembly drawing

Create a drawing of the Mountainboard on a B-size sheet. Include both an exploded view and an assembled view.



1 Save and Close all open documents.

5 Minute Assessment – #8

- 1 Where do you find ready-to-use hardware components?
- **2** True or False: Parts from Toolbox automatically size to the components they are being placed on.
- **3** How do you size Toolbox components as you are placing them?

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

- **5** True or False: A fixed component is free to move.
- **6** What is a Bill Of Materials?
- 7 What information can be shown in a balloon?

Exercises and Projects — Assembly Drawings

Exercise 23: Wheel Assembly

Create a drawing of the Wheel Assembly.

Include a Bill of Materials and Balloons.



Exercise 24: Create an AVI of an assembly explosion sequence.

Record the explosion and collapse sequences as an AVI.

Exercise 25: Additional Practice

Create additional assembly drawings and AVIs of the other sub-assemblies of the Mountainboard.

Lesson 8 Quiz

 Name:
 Class:
 Date:

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

- 1 When calculating the mass properties of a part, what density is used if there is no material applied to the part?
- 2 How do you apply a mate using Smart Mates?
- **3** True or False: Smart Mates can only add one mate at a time?
- **4** What allows a part to be automatically mated when dragging it from the Windows Explorer?
- **5** How many mate references can be added to a part?
- 6 How do you change the orientation of the Move Triad?

Lesson Summary

- □ Assemblies are created by mating together parts and sub-assemblies.
- □ Assembly components are put together in a similar manner to the physical world.
 - Individual parts are used to create small assemblies.
 - Small assemblies are put together to make larger assemblies.
 - The top level assembly starts with a major piece that does not move.
- □ Toolbox can be used to add standard hardware to an assembly.
- 3D Content Central provides additional hardware and parts from a variety of manufactures.
- □ Assembly drawings usually contain a Bill Of Materials that lists all the components in the assembly.
- □ Exploded views created in sub-assemblies can be reused in a higher level assembly.
- □ Balloons are used to identify individual components on the assembly drawing.
- □ **Material Properties** can be used to determine the weight and center of gravity of the entire assembly.

Lesson 9: Presenting Results

Goals of This Lesson

- □ Create photorealistic renderings.
- □ Create animations.
- □ Create eDrawings[®] from existing SolidWorks files.
- □ View and manipulate eDrawings.
- □ Email eDrawings.

Before Beginning This Lesson

- □ Complete the previous lesson Final Assembly.
- An email application needs to be loaded on the your computer. If email is not present on your computer, you will not be able to complete *More to Explore* which is an exercise that teaches you how to email an eDrawing.
- □ Verify that eDrawings2011 is set up and running on your computer.
- □ Verify that PhotoView 360 is set up and running.
- □ Verify that Adobe Acrobat Reader is installed.



Resources for This Lesson

This lesson plan corresponds to the *eDrawings* and Model Display modules in the SolidWorks Online Tutorials. For more information about the Online Tutorials, See "Online Tutorials" on page 1.

Review of Lesson 8— Assemblies

- □ Assemblies are created from parts and sub-assemblies
- □ The final assembly should be created in an order similar to the order used to assembly the final product in the shop.
- □ Smart Mates can be used to speed and simplify the assembly process.
- Sub-assemblies can be mirrored in an assembly. Some parts are mirrored and some are just copied.
- Explode and collapse sequences can be saved as AVI files if you have Animator installed.
- □ Assembly Statistics can be used to determine the number of parts and sub-assemblies in a higher level assembly.
- □ A Bill of Materials is used in an assembly drawing and lists the parts and subassemblies contained in the assembly.

Outline of Lesson 9

- □ In Class Discussion Project documentation
- □ Active Learning Exercises, Part 1 Screen Outputs
- □ Active Learning Exercises, Part 2 Drawings and eDrawings
- □ Active Learning Exercises, Part 3 Basic PhotoView 360 Rendering
- □ Active Learning Exercises, Part 4 Render the Truck
- □ Active Learning Exercises, Part 5 Texture Appearances
- □ Active Learning Exercises, Part 6 Adding Decals
- □ Active Learning Exercises, Part 7 Adding Appearances to assembly components
- □ Active Learning Exercises, Part 8 Final Rendering
- □ Active Learning Exercises, Part 9 Animations
- □ More to Explore Create a presentation to display the work created in this course
- □ Lesson Summary

In Class Discussion — Output

During the product design process, there are numerous requirements for output of the design details.

During Design

• Product design review

During the design process there will be multiple reviews of the progress. These can be informal between members of the workgroup or more formal presentations to the project leadership.

For lower level reviews these might be done at your workstation, for higher level reviews they could be done in a large conference room or presentation space.

• Collaboration

During the design process, work is generally shared between different engineers and designers. Each will be responsible for certain parts of the project, but the end result must be a product where all the parts fit together that achieves the design intent.

• Sub-contractors

Parts of the overall product design might be done by sub-contractors.

• Document Control

The Document Control organization needs to keep records of the design process and any changes that are made to the product.

- □ Manufacturing
 - Drawings

Drawings are still used to provide information to the shop where the product will be manufactured.

- Files for direct input to manufacturing machines Electronic files are used to provide direct input to computer controlled manufacturing equipment.
- Files to support manufacturing

Drawings are used to provide information to Quality Control inspectors as a source to determine if parts are manufactured correctly.

- □ Marketing
 - Product brochures
 - Trade shows
 - Web pages
 - PowerPoint presentations
- □ Product Support
 - Technical Manuals
 - Assembly Procedures
 - Training Manuals

Types of Output

□ Screen Prints

Screen prints can be used for simple presentation where full photorealistic rendered images are not required. If Realview is available, the screen prints will be Realview images.

Photorealistic prints

Photorealistic images can be produced using PhotoView 360.

□ Animations

Product animations can be created with the SolidWorks MotionManager.

□ 2D drawings

Two dimensional drawings can be created for parts and assemblies.

□ 3D drawings (eDrawings)

EDrawings provide a method to transfer both 3D parts and assemblies as well as drawings that contain 3 dimensional information.

D PDF

The Portable Document Format is used to provide an output that can be read by anyone with free reader software. The PDF format allows people at the receiving end to read a document without having the source program such as SolidWorks.

SolidWorks Files

SolidWorks files can be sent to other people that use SolidWorks. If the person receiving the files doesn't have SolidWorks, the files can still be read using the free version of eDrawings.

□ Non-SolidWorks files - STL, IGES, STEP

Non-SolidWorks files are used to transmit the 3D models to people that have other CAD software that need to have the 3D data available.

□ Image files

Image files include several widely used formats including TIFF and JPG. They can be created directly from SolidWorks. BMP, PNG, TIFF and others can be created when using PhotoView 360.

PhotoView 360 and MotionManager

Ideally, you want to view your designs in as realistic a manner as possible. Being able to view designs realistically reduces prototyping costs and speeds time to market. PhotoView 360 lets you use realistic surface appearances, lighting and advanced visual effects to display your models. SolidWorks MotionManager lets you capture and replay motion. Together, PhotoView 360 and SolidWorks MotionManager can display a model close to real life.

PhotoView 360 uses advanced graphics to create photorealistic images of SolidWorks models. You can select appearances to display the model as the built part would appear — if it existed. For example, if a part is being designed to have a chrome finish, you can display it in chrome. If chrome does not look right, you can change the display to brass or some other appearance.

In addition to advanced appearances, PhotoView 360 also has advanced lighting, reflectance, texture, transparency, and roughness display capabilities.

SolidWorks MotionManager is effective in realistically communicating the basic design intent of a SolidWorks part or assembly. You can animate and capture motion of SolidWorks parts and assemblies that you can play back. This allows you to communicate design intentions, using SolidWorks MotionManager, as a feedback tool. Often, an animation is a quicker and more effective communication tool than static drawings.

You can animate standard behaviors such as explode and collapse or other behaviors such as rotate.

SolidWorks MotionManager generates Windows-based animations (*.avi files). The *.avi file uses a Windows-based Media Player to playback the animation. You can use these animation files for product illustrations, design reviews, and so forth.
Active Learning Experience, Part 1 — Screen Outputs

Task 1— Create printed versions of the Axle.

While working on the Axle Assembly, you will need to show your manager your progress. You are also collaborating with another engineer on the design of the Mountainboard, so you need some images of the part you are working on to show him.

- 1 Open the Axle Assembly.
- 2 Orient the model in the Isometric view.
- 3 Click View, Display, Shaded so that we do not see the edges of model.
- 4 Turn off RealView graphics. Click View, Display, and make sure that RealView Graphics is not selected.
- 5 Click File, Print from the menu.
- 6 Select your printer.
- 7 Click Page Setup.

Print	x
Document Printer	
Name: HP DeskJet 1220C	Properties
Status: Ready	Page Setup
Type: HP DeskJet 1220C	Preview
Where: USB002	Freview
Comment:	
Document Options	System Options
Header/Footer	Margins
Print range © Entire model	Number of copies: 1
 Current sheet 	Print background
Current screen image Selection	Print to file
O Sheets:	Convert draft quality drawing
Enter sheet numbers/ranges. For example: 1,3,5-8,10	nerra conigri quanty
	Help

- 8 For Resolution and Scale select Scale and type 80 in the spin box. We have to scale this down because the part would be larger than a standard sheet of paper.
- 9 Click OK.
- 10 Click Header/Footer.
- 11 Click Custom Header.
- 12 Click the Font button. Change the font to Arial, Bold, 16 point. Click OK.

 Use system settings Use this document settings Set each drawing sheet individuall Settings for: 	<u>у</u>
Resolution and Scale ○ Scale to fit ● Scale: 80 ● %	Drawing Color Automatic Color / Gray scale Black and white
Paper Size: Letter Source: Automatically Select	Orientation Portrait Landscape
Preview OK	Cancel Help

13 Type Axle Assembly in the **Center** box. Click **OK**.

Custom Header					×
To insert the page r filename: position t appropriate button.	umber, numbe ne insertion po	er of pages, da pint in an edit b	ite, time or ox and select	t the	OK Cancel
		3	Font		Help
Left section:	Ce	nter section:		Right section	:
	*	Axle Assem	bly		*
	-		~		-
Custom Faster					X

- 14 Click Custom Footer.
- **15** Type your name in the center box.
- 16 Click in the **Right section** box, then

click to add the date. The actual date will not appear in the **Custom Footer** box, only the control code & [date]. When the document is printed, the actual date will appear. Click **OK**.

17 Click **OK** to print the document.

Custom Footer		×
To insert the page numbe filename: position the ins appropriate button.	r, number of pages, date, time or ertion point in an edit box and sele	ect the OK Cancel
	Font	Help
Left section:	Center section:	Right section:
	Student Name	&[date] ^



RealView Graphics

RealView graphics can produce a much more realist image of the 3D model. It supports real-time reflections and much more realistic surface characteristics.

To use RealView graphics, your computer must have a video graphics card and the appropriate drivers that support RealView. If your graphics card supports RealView, the

following icon \bigotimes will appear on the View toolbar. If your graphics card does not support RealView, the icon will be grayed out.

To view parts and assemblies with Realview graphics, materials must first be added to the parts.

Note: If your computer does not support RealView graphics, you will not be able to do the following steps.

- 1 Click RealView Graphics 😡 on the View toolbar.
- 2 Change the appearance applied to the Axle to Chrome Stainless Steel.
- 3 Select the **Appearances**, **Scenes and Decals** tab in the Task Pane. Expand the Appearances folder and click Chrome. Drag the appearance chromium plate onto the Axle part. When you release the mouse button, the **Appearance Target** will



4 Rotate the view and notice how the reflections move as if created by a scene behind the viewer

- 5 Change the scene. Click
 Apply Scene from the Headsup View toolbar and select
 Factory Background from the list.
- 6 Rotate the model and notice that the background and reflections both move.



- 8 Select Print background.
- 9 Click **Custom Header**. Add the text "RealView Graphics" to the Right section.
- **10** Click **OK** three times to exit all the open dialogs and print the image.



11 Compare this image to the first image and notice the improvement.

There are still many adjustments we could make to this image to make it look even better, but for now we will use this.



Task 2 — Create image files from a part

Creating a Slideshow

Another form of review can take place using prepared slides and Microsoft[®] PowerPoint. Creating image files of our model and inserting them into PowerPoint slides is a simple task.

Saving the Image

You can save SolidWorks and PhotoView 360 images to an image file format that can be used for design proposals, technical documentation and product presentations.

SolidWorks images can be saved as:

- JPEG (*.jpg)
- TIFF (*.tif)
- Adobe Illustrator (*.ai)
- Adobe PhotoShop (*.psd)

PhotoView 360 Images can be rendered and saved to the following file types:

- Flexible Precision Image (*.flx)
- TARGA (*.tga)
- Windows Bitmap (*.bmp)
- High Dynamic Range (*.hdr)
- JPEG (*.jpg)
- Portable Network Graphics (*.png)
- SGI RGB (*.sgi)
- TIFF (*.tif)
- Open EXR (*.exr)

- 1 Click File, Save As. Save As is used to save a file to a new name or file type.
- 2 Select **Tif (*.tif)** in the **Save as type** list.
- 3 The default file name should be Axle Assembly.TIF.
- 4 Navigate to the Lesson09 folder. Click **Save**.

We now have a TIFF image that can be used in a presentation or written report.

Organize New folder Image: Second S	Organize New folder		
Image: Name Date Image: Name Image: Name Image: Name Image: Name <t< th=""><th>organize</th><th></th><th>- (</th></t<>	organize		- (
File name: Axle Assembly_&.TIF Save as type Tif (*.tif)	Lesson08 Lesson09 Built Parts Exercises Mountainboard Axle-Truck Binding Hardware	Name Built Parts Exercises	Date mc 3/14/201 3/14/201
Description: Add a description Options	File name: Axle Assembly_&.TIF Save as type Tif (*.tif) Description: Add a description Options]	

- 5 Start Microsoft[®] Paint. Click Start, All Programs, Accessories, Paint.
- 6 Open the image Axle Assembly.TIF.
- 7 Examine the image, it should be just like the image we printed except that there will not be a header or footer.
- 8 Close Microsoft Paint.
- 9 Start Microsoft PowerPoint.
- **10 Open** the presentation Mountainboard.ppt found in the LessonO9 folder. This is a blank presentation with a a title slide and body slide.
- 11 Select the body slide.
- 12 Click "Click to add title" and type Axle Assembly.
- 13 Click in the center of the slide. Click **Insert, Picture** from the menu.
- 14 Navigate to the Lesson09 folder and select the file Axle Assembly.tif. Click **Open**.

- **15** The image will appear on the slide, but will be too large. Resize the image by dragging the corner handles.
 - **TIP:** If you drag the corner handles, the image will maintain the same length to width ratio. If you drag one of the side handles, you will change the image proportions which is not desired in this case.



- 16 Create a new slide by clicking **Home**, **New Slide** from the menu.
- In SolidWorks click Image Capture and on the Standard toolbar or View, Screen
 Capture, Image Capture from the menu. This will copy an image of the graphics area to the Windows clipboard.
- **18** In PowerPoint, select the new slide and click **Edit**, **Paste**. Resize the image as necessary.

By using the screen capture and paste, we did not create another file.

19 Save the presentation and **Close** PowerPoint.

Active Learning Exercises, Part 2 — Drawings and eDrawings

Drawings

Drawings created in SolidWorks depend on references to the part or the assembly models used to create them. When sending a drawing to another engineer, customer or supplier, you send all the referenced files if you intend for them to work on the drawing.

If the intent is for the others to only view the drawing, there are many ways to send the drawing as a self contained file.

Some of the choices are:

- □ Send the SolidWorks drawing and view it using SolidWorks in the Quick view mode.
- □ Send the SolidWorks drawing and view it using SolidWorks Viewer.
- □ Create an eDrawing
- □ Create a TIF file.
- □ Create a PDF file.
- 1 Open the drawing Truck.slddrw from the Lesson09 folder.
- 2 If SolidWorks cannot locate the referenced file, it will ask you to locate it.

Click No.

SolidWorks 2011	
Unable to locate the file Curriculum_and_Cou \Lesson09	K:\\$olidTester\CopyFilesHere2\\$tudent\Lesson Files\\$olidWorks irseware_2007-2008\Mountainboard Design Project\Lessons \Truck.SLDPRT. Would you like to find it yourself?
🔲 Don't ask me again	Yes No

- Because the drawing cannot locate the part, it will open with blank views. This is what someone would see if you just sent them the drawing without the part.
- 4 **Close** the drawing without saving.



Quick View

SolidWorks files can be opened as view-only using the selection **Quick View**. When a file is opened as view-only, none of the parametric data is loaded. Instead only the visualization data stored with the file is shown. You can still zoom and pan. If you have opened a part or assembly as view-only, you can also rotate the model.

When a file is loaded as view-only, the FeatureManager design tree will be empty.

SolidWorks Viewer

SolidWorks Viewer is a free program used to view SolidWorks files. When using SolidWorks viewer, you can zoom, pan and rotate (parts and assemblies only) the model.

If you send a SolidWorks file to someone who does not have SolidWorks, they can download the SolidWorks Viewer from the SolidWorks web site, <u>www.solidworks.com</u>.

- 1 Click File, Open.
- 2 Locate the file Truck.slddrw in the LessonO9 folder but do not open it.
- 3 Select Quick View.

🛐 Open		X
🔾 🗸 🖓 🖉 Mountainboa	rd Design Project_Student + Lessons + Lesson09 +	 ✓ ✓
Organize 🔻 New folder		i = 🔻 🗔 🔞
Lesson01 Lesson02 Lesson03 Lesson04 Lesson05 Lesson06 Lesson07 Lesson08 Lesson08	Name Exercises Truck-SLDDRW Truck-1.SLDDRW	
Date modified: 5 Size: 4	/24/2007 7:34 AM Date created: 3/14/2011 7:33 PM Compu 82 KB Date accessed: 3/14/2011 7:33 PM	iter: TRAINING01 (this computer)
	Quick view References Advanced Configurations: Lightweight	Display States (linked) ▼ ✓ Do not load hidden components
File name:	Truck.SLDDRW	▼ Drawing (*.drw;*.slddrw) ▼ Open ▼ Cancel

- 4 Click **Open**. The drawing will be opened and the FeatureManager will be empty. If you try to locate the references, **File**, **Find References** will be grayed out. This is the same results as using the SolidWorks Viewer.
- 5 **Close** the drawing.

Saving As Image Files

If we save our parts, assemblies or drawings as image file, we do not save any of the intelligence of the 3D files.

There are two different ways to save files as images, raster or vector.

Raster Files

The image files we created are stored as raster information, that is, they store information about each pixel in the image. Raster images *do not* scale well. As we zoom in on a raster image, the individual pixels get larger and we lose resolution.

Raster files can be quite large as they have to keep information about every pixel regardless of whether or not there any useful information at a particular pixel.

Vector Files

Vector files store information based on where entities start and how to get to where they end (a vector). These images *do* scale well and are better suited to CAD drawings.

Vector files can be considerably smaller than raster files as they only store information about the actual entities in the file.

- 1 Open the drawing Truck-1.slddrw. Do not open the file as Quick View.
- 2 Click File, Save As.
- 3 Select **Tif (*.tif)** in the **Save as type** list.

The default file name should be Truck-1.tif.

4 Navigate to the Lesson09 folder. Click **Save**.

We now have a TIFF image of the drawing. TIFF files are raster files.

- 5 Click File, Save As.
- 6 Select Adobe Portable Document Format (*.pdf) in the Save as type list.
- 7 The default file name should be Truck-1.pdf
- 8 Navigate to the Lesson09 folder. Click **Save**.

We now have a PDF image of the drawing. PDF files are vector files.

- **9** Start Microsoft Paint.
- **10 Open** the file Truck-1.tif.
- 11 Examine the drawing. At full screen it looks pretty good.

12 Zoom in to the title block area. Notice that the text is hard to read because it is made of individual pixels that are now too large to properly display the characters. Look also at the curved lines in the drawing, they are not smooth because of the pixilation.

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PDF Format

The Portable Document Format or PDF, created by Adobe, can be used to save essentially any file format to a standard format that can be read with a free viewer. PDF insures that the image seen by the receiver is the same as that created in the source program. It also eliminates the need for the person receiving the file to have the same program as the person creating the source file.

Acrobat Reader

Acrobat Reader is a free program crated by Adobe. If not already loaded on your computer, it may be downloaded from <u>www.adobe.com</u>.

- 1 Use Windows Explorer to locate the file Truck-1.pdf in the LessonO9 folder.
- 2 Double-click Truck-1.pdf. This will start Adobe Acrobat Reader.
- 3 Zoom in on the same area of the drawing as we did in the last section. Notice that no matter how far you zoom in, the text is still clear because this is vector information instead of raster information.



4 **Close** Acrobat Reader and Microsoft Paint.

Task 3 — Create an eDrawing[®]

While the PDF format created a good, readable file, it still has limitations when used to share data. We see things in the world around us as 3D, so 2D drawings by their very nature are not as easily understood as 3D data. With eDrawings, we can create easy to use and easy to understand documents to convey our design data.

eDrawings

eDrawings is a free viewing and publishing application, created by SolidWorks Corporation, for sharing and archiving 2D and 3D product design data.

eDrawings can be created by:

- □ Click File, Publish to eDrawings.
- □ Click File, Save As and select eDrawings (*.edrw).
- 1 Click File, Save As.
- 2 Select eDrawings (*.edrw) from the Save as type list.
- 3 The default file name should be Truckl.edrw.
- 4 Navigate to the Lesson09 folder.
- 5 Click **Options**.

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Save as type: eDrawings (*.edrw)						
Description: Add a description						
Options	٦		Pas	sword		

6 Select Okay to measure this eDrawings file and Save shaded data in drawings.



7 Click **OK** then **Save**.

The eDrawing will be created.

8 Locate the file Truck-1.EDRW in the LessonO9 folder and double-click it.

9 eDrawings will open its own window.



- 10 The eDrawing looks very much like a standard 2D drawing, however there is much more intelligence behind it. Click Play
 Play
- 11 eDrawings will display all the views in rotation, stopping at each view to show any dimensions.
- 12 When you have seen all the views, click **Stop** \square_{Stop} .
- **13** To return to the view of the drawing sheet click **Home**
- 14 Select the Top view in the graphics area.
- **15** Click the **Rotate** tool. You can now rotate the model just like you could inside SolidWorks.
- 16 Click Home

Solid Works

A Truck-1

SCALE 12 PROVE

The 3D Pointer

You can use the **3D Pointer** to point to a location in all of the drawing views in drawing files. When you use the **3D Pointer**, linked crosshairs appear in each of the drawing views. For example, you can place the crosshairs on an edge in one view and the crosshairs in the other views point to the same edge.

The crosshairs colors indicate the following:

Color	Axis
Red	X-Axis (perpendicular to YZ plane)
Blue	Y-Axis (perpendicular to XZ plane)
Green	Z-Axis (perpendicular to XY plane)

1 Click View, 3D

Pointer Pointer . This displays a 3D pointer which shows the same point in all views at the same time.

- 2 In any of the views, drag the intersection of the three axis and notice how the pointer moves in all the views at the same time.
- 3 Click View,

3DPointer Pointer to turn off the 3D pointer.



Sending eDrawings

eDrawings can be sent as email in various forms.

If the receiving party has the eDrawing viewer, the eDrawing can be sent as an eDrawing file (.edrw [drawing], .eprt [part], .easm [assembly]).

If the receiving party does not have the eDrawing viewer it can be sent with the file. Files containing the viewer can be either executable (.exe), Zip (.zip), or HTML (.htm).

Task 4 — Send the eDrawing as email

1 Click File, Send from the menu or just click

Send \bigotimes_{send} on the toolbar.

2 Select **HTML**, then **OK**.



- 3 An email message will be generated with the eDrawing in HTML format as an attachment.
- 4 Address the email to yourself and send it.
- **5** When you get home, check the email to see what the receiving party would get.
- 6 Click **File**, **Close** to close the eDrawing of the Truck-1.

Paste	B I aby - A	· · · · · · · · · · · · · · · · · · ·	
Send	From To Cc Bcc Subject:	Road Runner Student@home.com Truck-1_&	
You ha you mu SolidW installe Double SolidW	ve been sen ust have the 'orks eDrawi d when you e-click the en 'orks eDrawi	t an eDrawings file as an HTML file. To view the eDrawings file, SolidWorks eDrawings Viewer installed. If you do not have the ngs Viewer installed, it will be automatically downloaded and open the HTML file in Internet Explorer. Inclosed *.htm file to view the eDrawings file and install the ngs Viewer if necessary.	

eDrawings As A Viewer

eDrawings can open any SolidWorks file from SolidWorks 97 Plus or later.

Task 5 — Open a SolidWorks assembly with eDrawings

1 Click **File**, **Open** and navigate to the **Truck_Axel_Wheel** assembly in the Mountainboard folder.

Click Open.

SolidWorks eDrawings Professional x64 Edition - [Truck_Axle_Wheel_&.SLDASM]	x
🔋 😢 File View Tools Window Help	x
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Available: Enterprise PDM 2011 SP3 is available for download → 🕲 DRAWINGS® 🖉 🕰	.#

- 2 Select the **Components** tab , then click **Move**. The **Move** tool can be used to move the individual components of the assembly.
- 3 Drag the Truck away from the assembly.
- 4 Experiment by dragging other components.
- **5** Right-click on one of the tires and select **Move Wheel Assembly**. You can now drag the assembly instead of just the single part.
- 6 To undo the move, right-click on the same tire and select **Undo Move**.
- **7** To return the assembly to its original condition, click **Home**
- 8 Right-click the Truck and select **Hide**.
- 9 To hide assemblies, right-click the assembly in the eDrawing Manager and select Hide.

Measure and Markup

eDrawings can be used as a design review tool. Rather than requiring everyone in the review chain to have SolidWorks, models can be reviewed by anyone with a copy of eDrawings viewer.

Measure and **Markup** are part of eDrawings Professional. These functions must be turned on by the person creating the ePart, eAssembly, or eDrawing using eDrawings Professional. Once turned on, anyone with the standard eDrawings viewer may use the functions.

Measure provides a capability similar to the Measure tool inside SolidWorks.

Markup allows the reviewers to add comments with arrows or clouds. The markup may be saved as a separate file so that only the markup comments need be sent back to the originator rather than the entire eFile.

Markup Options

Markup options allow you to establish the name of each reviewer and the color that will be used to display the comments.

- 1 Click **Tools**, **Options**. Select the **Markup** tab.
- 2 Type your name, or Student, for Name.
- **3** Select the color to be used for your comments by clicking in the color box.

We can also change the line size and text font, however we will leave then at the default for now.

- 4 Click **OK** to close the eDrawing Options.
- 5 Select the Markup tab 🥖 , then click Text with

Leader 🖊

Options 💌
General Markup Simulation Import
Comment preferences
Name: Student
Phone number:
Email address:
Color
Century Gothic, 6pt
Apply changes to:
All comments in this document
New comments
Replying to comments Copy original comment when replying Original comment color
OK Cancel Reset Help

- 6 Click on one of the Wheel Hub parts near the SolidWorks text, then move the cursor to the position where you would like to drop the text and click.
- **7** Type "Add color to wheel hub molding".





9 Examine the Markup tab in the eDrawingManager. AllMarkup comments will be listed on this tab.



10 Because the Mountainboard is used in a dirty environment, we do not want to use open type bearings where the dirt can get between the balls and the race.

Add another comment to change to a sealed bearing.

- 11 Click File, Save Markup.
- 12 Select Student then OK.



8 Click < .

- **13** Select the Axle-Truck folder in the Mountainboard folder, to store the markup file. The file will be saved with a *.markup extension.
- 14 Now that the markup file is saved external to the eDrawing file, we can delete the individual markups.

Right-click each comment and select Delete Comment.

The Markup tab should now be empty.

Markup Comments

Markup comments from other reviewers can be sent back using email. Only the markup file need be sent back as it can be loaded into the original copy of the eDrawing.

TIP: When sending the eDrawing to each reviewer, tell them which color to use for their comments.

- 15 We can restore not only our own comments, but also those of other reviewers.
- 16 Click File, Open Markup.
- 17 Navigate to the Axle-Truck folder and select the file Truk_Axle_Wheel.sldasm.Student.markup.

Click Open.

The comments are restored.

18 Close all open files.

5 Minute Assessment – #9-1

- 1 How do you create an eDrawing?
- 2 How do you send eDrawings to others?
- **3** What is the quickest way to return to the default view?
- 4 True or False: You can make changes to a model in an eDrawing.
- **5** True or False: You need to have the SolidWorks application in order to view eDrawings.
- **6** What eDrawings feature allows you to dynamically view parts, drawings, and assemblies?

Active Learning Exercises, Part 3 — Basic Rendering with PhotoView 360

Photorealistic Rendering

Photorealistic rendering is the process of photography except that we are using a computer model instead of a physical model.

Prior to actually producing a product we may need to show the customer what the product will look like or we may need to produce marketing materials.

The PhotoView 360 software

PhotoView 360 is a software solution from SolidWorks, fully integrated into the SolidWorks software to create photorealistic images directly from SolidWorks models. Renderings may be created from SolidWorks parts and assemblies, but not drawings. PhotoView 360 can produce photorealistic images to add visual impact to presentations and documents.

Some of the key features of PhotoView 360 are:

Photorealistic images directly from SolidWorks models

PhotoView 360 interacts with the 3D geometry created with the SolidWorks software. All changes to SolidWorks models are accurately represented in the PhotoView 360 images.

Fully integrated into SolidWorks

PhotoView 360 software is supplied as a SolidWorks add-in. You access all the controls for PhotoView 360 rendering from PhotoView 360 items on the main SolidWorks menu bar, the Render Tools toolbar or the Task Pane. The menu bar is displayed whenever a SolidWorks part or assembly document is open.

Appearances

One set of appearances is supplied with SolidWorks and used in both SolidWorks and PhotoView 360 to specify model surface properties such as color, texture, reflectance and transparency. Other appearances can be downloaded from various web sites, created using image creation software, or by scanning.

Lighting

Lights may be added in the same way a photographer adds lights when taking photographs. PhotoView 360 uses the same lights as SolidWorks but also uses lighting from the rendering environment or scene. PhotoView 360 has the sophistication to trace light rays and reflections.

Scenes

Each SolidWorks model is associated with a PhotoView 360 scene, for which you can specify properties such as environments and backgrounds. Scenes help to put products in context.

Decals

Images, such as company logos, can be applied to models.

Output

The rendered output from PhotoView 360 can be viewed on the screen, or saved to a graphics file.

Starting PhotoView 360

When PhotoView 360 is installed, the menu and toolbar do not automatically appear as part of the SolidWorks screen. They must be turned on.

To turn on PhotoView 360:

□ From the Tools menu, select Add-Ins..., select PhotoView 360.

Task 1— Start PhotoView 360

1 Click Tools, Add-Ins.

In the Add-Ins dialog box, select PhotoView 360. Click OK.



PhotoView 360 User Interface

The PhotoView 360 software uses the same user interface as the SolidWorks software. No new interface techniques are required.



Render Tools Toolbar

The Render Tools toolbar will appear whenever a part or assembly document is active. It can be moved, resized or docked like all other SolidWorks toolbars. If this toolbar is turned off, it can be turned

on by right-clicking an existing toolbar and selecting **Render Tools** or by using the **View**, **Toolbars** menu.

DisplayManager

There is an additional tab (on the Feature Manager design tree called the Display Manager. The Display Manager provides a view of the appearances, decals, lights, cameras and scenery associated with the active Solid Works part or assembly.

The DisplayManager indicates which items of geometry are attached to which appearance and decals.

The DisplayManager also makes it easy to:

- Understand the way in which appearance and decal inheritance works.
- Select and edit appearances and decals associated with the model.
- □ Transfer appearances and decals between components, features, and faces.

Getting Help

Help with tools associated with rendering are part of the standard SolidWorks help system.

To Get Help

Select SolidWorks Help from the Help menu, then select Model Display in the Contents tab.



Dynamic Help

Dynamic help is provided to assist you in understanding the effects of various display controls. Dynamic help is enabled in several DisplayManager tabs.

Whenever you select, or hover over, an item on the Illumination or Surface Finish tab, dynamic help will appear.

Move the cursor over any active illumination property to display dynamic help pertaining to that property.

With the cursor over the **Specular amount** property, dynamic help shows the way the model will reflect light as the slider is moved.

Options

PhotoView 360 has its own options. Options allow you to customize the PhotoView 360 settings to reflect your preferences. Options are divided into Output Image Settings, Render Quality, Bloom, Contour Rendering and Direct Caustics.

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0.60000024	lower value reduces highlights.
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0.150	directional, spot, or point light must illuminate the model and be enabled in
Blurry reflections	PhotoView.
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For a complete listing of all the settings available though the PhotoView 360 Options dialog refer to the Help menu.

To set PhotoView 360 Options:

- □ Click PhotoView 360, Options
- Or, click PhotoView 360 Options 🍖 in the DisplayManager.
- Or, on the PhotoView 360 toolbar, click Options 6.

Task 2 — Set PhotoView 360 Options

Before beginning a project in PhotoView 360, system options need to be set to make sure everyone sees the same results.

1 **Open** the part Axle.

Note: PhotoView 360 options can only be set if a part or assembly document is open.

(

2 Set the PhotoView 360 Options.

Click **Options (b)** on the Render Tools toolbar.

Under Output Image Settings set the options as follows:

- Image Width: 600
- Image Height: 400
- Image format: Windows BMP
- Default image path: Click **Browse** and select the **Mountainboard** folder
- 3 Set the **Render Quality** properties.
 - Preview render quality: Good
 - Final render quality: Good
 - Gamma: **1.6**
- 4 Other options.
 - Bloom: Cleared
 - Contour Rendering: Cleared
 - Direct Caustics: Cleared
- 5 Click \checkmark to close the **Options**.

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Bloom	≈
Contour Rendering	*
Direct Caustics	⇒

PhotoView 360 Rendering Procedure

For each SolidWorks model to be rendered, the process requires the same steps. Most of the steps are done within SolidWorks with only a few steps requiring PhotoView 360. The following steps are repeated until a satisfactory output is obtained:

Position the Model

Use a standard view, or use zoom, rotate and move to position the model in the desired position.

□ Apply Appearance

Apply appearances to the model, features and/or selected faces.

□ Set the Scene

Select one of the various preset scenes, or set your own background and scenery.

□ Set Lighting

Adjust the environmental lighting or add direct lights.

Preview Render

The preview is a test of the settings before final render.

□ Render the Model

The model is rendered to the Final Render window.

□ Save Output

To be used, the final render must be saved as a separate image file.

Post Processing

The rendered image output is not always the final product. Often the image output is used with other programs for additional effects.

Appearances

Appearances affect the way a surface reacts to light. They may be applied to parts, features or faces. Appearances are of two general types, Procedural and Textures.

To apply a appearances, select the object to which the material will be applied. Then, double-click the appearance in the lower pane of the task pane, or drag the material onto the selected part, feature or face.

To apply materials:

- □ Click PhotoView 360, Appearance.
- □ Or, click **Appearance** (●) on the Render Tools toolbar.

Appearances, Scenes and Decals Pane

The **Appearances**, **Scenes**, **and Decals Pane** lists all the appearances that are available to be applied to the model.

The top pane is the **Appearances Library** where appearances are listed in **Appearance Folders**. The appearance tree shows all the folders currently loaded. Each folder can be expanded by clicking the plus sign next to it to show the sub-folders. The bottom pane is the **Appearance Selection** area.

Procedural Appearances

Procedural appearances are defined by some procedure and consist of one or more colors and the way the appearance reacts to light. Procedural appearances can be thought of as 3D, that is they go all the way through the part. This is like adding dye to a plastic to be injected; the color will go all the way through the finished part.

Texture Appearances

Texture appearances are applied like wallpaper. During application, they can be stretched, shrunk, rotated, and reoriented to make them fit the surface. The pattern will be duplicated as many times as necessary to cover the entire surface.



Appearance Hierarchy

Appearances follow a rule-based hierarchy in both SolidWorks and PhotoView 360. Appearances applied to a face override appearances applied to a feature, which override appearances applied to the part.



5 Apply appearances to the part.

Click the plus sign to the left of the **Metal** folder, then select the sub-folder **Aluminum**.

In the appearance selection area, locate **polished aluminum**. Press and hold the **Alt** key and drag the material **polished aluminum** into the graphics area. This will apply the material to the entire part.

The PropertyManager will show the material is applied to the entire part.

Click 🗹 .



Scenes

PhotoView 360 scenes are made up of the things we see in the rendering that are not the model. They can be thought of as a virtual sphere around the model. Scenes are composed of backgrounds, foreground effects, and scenery. PhotoView 360 has numerous predefined scenes to make initial renderings quick and easy.

The **Scenes** folder in the Task Pane lists all the predefined scenes that are available to be applied to the model.

The top pane is the **Scene Library** where scenes are listed in **Scene Folders**. The scene tree lists all the scene folders currently loaded. Each folder can be expanded by clicking the plus sign next to it to show the sub-folders. The bottom pane is the **Scene Selection** area.

To apply a scene:

- Right-click the scene in the scene selection area, then click Add Scene to part [assembly].
- □ Or, double-click a scene.
- □ Or, drag the scene into the graphics area.
- 7 Select a scene.

Click the plus sign next to **Scenes** and then

select the **Presentation Scenes** folder. The different presentation scenes will appear in the lower pane.

Drag the scene Factory Background into the graphics area.

Render Preview

The PhotoView 360 rendering can be previewed in two ways, as a separate window or integrated into the SolidWorks graphics area. These previews can be used to test the render settings with a quick sequential rendering. Both previews are done as progressive renders in that the entire screen area is rendered at once and then refined until the selected quality is achieved. Only one of the two previews can be active at one time.

Both previews yield the same results, so it is an individual choice of which to use. For the remainder of this course, the Preview Window will be used.

Preview Window

To start the Preview Window:

- □ Click PhotoView 360, Preview Window from the menu.
- □ Or, click **Preview Window (b)** on the Render Tools toolbar.



Integrated Preview

To start the integrated preview:

- □ Click PhotoView 360, Integrated Preview from the menu.
- □ Or, click Integrated Preview **(I)** on the Render Tools toolbar.

When we turn on either preview, we will get a message recommending the use of perspective for a more realistic display.

Use Perspective Views in Renderings			
For a more realistic rendering, we recommend that you switch to perspective view or view the model through a camera with Perspective enabled. Click View > Display > Perspective or View > Lights and Cameras > Add Camera.			
Don't ask me again			
ОК			

Perspective

When creating CAD models, our viewports are set to orthographic, so there is no perspective. With orthographic views, parallel edges appear parallel on the screen which makes it easier to visualize the relationships in the model. When we look at objects in the physical world, we see perspective which aids in our perception of depth.

Perspective may be toggled on or off by:

- □ Clicking View, Display, Perspective on the menu.
- □ Or, clicking **Perspective** on the Heads-up toolbar.
- 1 Preview Render. Click **Preview Window** (1) on the Render Tools toolbar. Click **OK** to clear the message about perspective.
- 2 Turn on perspective by clicking View, Display, Perspective.

The Axle will be rendered in the Preview Window. Examine the rendering, you should be able to see the reflections of the holes on the flat surfaces of the model.

We have now done the basic steps of adding appearances and a scene to the model, and have a good start.

3 Create a final rendering. As the preview render looks about right, the next step is to create a final rendering and save it as a separate image file.



4 Click **Final Render** 2 on the Render Tools toolbar. The Final Render window will open and the image will be rendered.

Final Render Process

The render process takes place in two phases. In the first phase, the screen will be drawn four times with increasing detail. During this phase, PhotoView 360 is calculating the lighting and reflections. In the second phase, the actual image will be rendered. You will see one or more squares rendering until the image is complete. These little squares are referred



to as buckets and the number of buckets is dependant on the number of CPU cores and threads on your computer, one buck per thread.

- 1 Examine the Final Render window. At the top is information about the settings and time it took to render the image. There are also numbers from 0 to 9. PhotoView 360 stores the last ten rendered images. To see the other images, click any of the ten numbers.
- 2 Save the image to an image file. To be able to use the rendered image we need to save it as a separate file. Click Save Image in the upper right corner of the Final Render window. Save the file to the lesson folder. The file type will be Windows BMP by default because that is what we set in the PhotoView 360 options.

🛞 Save Image				×
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File name: Axle Asse Save as type: Windows	embly BMP (*.BMP)			•
Alide Folders			Save	Cancel

- **3** Using Microsoft Paint, open the Axle Assembly image and examine the results. Now that there is a separate image, it can be used in other documents.
- 4 Close all open files.

Lighting

Proper lighting can greatly enhance the quality of the rendering. The same principles used by photographers also work well in PhotoView 360.

When rendering with PhotoView 360, the primary light comes from the environment. In the physical world, if we take a picture without using a flash, the light used for the picture is environmental lighting which comes from a multitude of sources. In your classroom, you have light coming in through the windows and from overhead lights. This light reflects and refracts off or through many surfaces to illuminate the things we see. To understand how this works in PhotoView 360, we need to understand the components of a scene.

The scenes used in SolidWorks have several components as shown in the image at right. The environment is represented by the green sphere. It wraps around the model and our viewpoint. These environmental images give us not only what we see from our viewpoint, but also a world behind our viewpoint that allows reflections to be seen in the model. It is also these images that are used to light our rendered models.

The blue plane in the image is the background and can be seen behind our model. The checkered plane is the floor and will be positioned under the model to allow shadows and reflections.



In addition to the environmental lighting, discrete lights can also be used, just as they are when viewing the model in OpenGL or RealView.If we are going to use discrete lights in addition to our environmental light, to additional lights are created and positioned in SolidWorks. These lights must specifically be turned on in PhotoView 360 to be seen. PhotoView 360 has a few additional controls to refine the quality of the light and shadows.

Types of Lights

SolidWorks and PhotoView 360 use four types of lights:

□ Ambient

Ambient light illuminates the model evenly from all directions. In a room with white walls, the level of ambient light is high, because the light reflects off the walls and other objects.

Directional

Directional light comes from a source that is infinitely far away from the model. It is a collimated light source consisting of parallel rays arriving from a single direction, like the sun. The central ray of a directional light points toward the center of the model.

□ Spot

A spot light is a restricted, focused light with a cone-shaped beam that is brightest at its center. A spot light can be aimed at a specific area of the model. You can adjust the position and distance of the light source relative to the model, and the cone-angle through which the beam spreads.

Point

A point light comes from a very small light source located at a specific coordinate in the model space. This type of light source emits light in all directions. The effect is like a tiny light bulb floating in space.

Other Light Sources

In addition to the environmental lighting and the discrete lights, appearances can be used to provide light. Several appearances are specifically set up as light sources such as LEDs or area lights, but all appearances have a property called Luminous Intensity that can be adjusted to allow the appearance to act as a light source.

Creating Lights

SolidWorks creates multiple lights with each new part, depending on the scene selected. To create additional lights, right-click the Lights folder in the DisplayManager under the **Scene, Lights, and Cameras** tab. From the menu you can add additional spot, directional or point lights.

Note: There is only one ambient light. You cannot add any more, nor can you delete it.

Photographic Lighting

Model lighting is very subjective and is as much art as it is science. To obtain the best results, you should think like a photographer. There are many books on the subject of lighting, with different techniques, but most are based on a combination of using three basic lights.

□ Key light

This is a strong, front light to provide overall illumination of the model. The Key light is sometimes also called a Primary light.

Fill light

This light is generally of less intensity than the primary light and is used to lighten shadows by reducing the overall contrast between light and dark areas of the model.

□ Backlight

A light usually above and slightly behind the model to help outline the shape and make the model easier to see against the background.

Special Lights

In addition to the basic three lights, special lights are used to focus attention on some part or feature of the model, or to create some desired effect. For example, a point light might be positioned inside a lamp, to simulate the illumination coming from the lamp itself.

Active Learning Exercise, Part 4 — Rendering the Truck

In the last task, we further examined the steps used to create a rendered image. This time we will add some additional elements to gain some additional control.

To review, we must do the following steps:

- □ Position the model
- □ Apply Appearance
- □ Set the Scene
- □ Set the Lights
- \Box Render the model

Task 1— Apply Appearance to the Truck.

- 1 **Open** the part Truck.
- 2 Orient the part to the Isometric view.
- **3** In Lesson 6, we added an appearance to the **Truck**, so we do not have to do it again. If you did not add the appearance, you must do the following two steps.
- 4 Select the **Appearances**, **Scenes and Decals** tab on the Task Pane.
- **5** Locate the material **PW-MT11250** found under **Plastic, Textured** and drag it into the graphics area.



Note: Why are we using a different appearance than the part will be made from? With photorealistic rendering, our main concern is what the surface of the model will look like, not the actual material itself. For instance, if we make a part from steel and then apply a layer of paint, it is the paint that we will see, not the steel.
6 Turn on RealView by either clicking View, Display, RealView Graphics from the menu or click RealView Graphics on the Heads-Up toolbar.

Because we have not specified a scene, the model will be shown with the scene that was part of the original template used when creating the file. The appearance of the part is acceptable except that it is hard to see the details in several areas because of the lack of light.



7 Preview Render the model.Use the Preview Window to see how the rendered model will look.

We can see several differences from the RealView image. Once significant difference is the lighting as we do not see the bright areas on the model. This difference is due to the RealView model using the direct lights and the rendered preview using only the environmental lighting.



- 8 At the top of the FeatureManager design tree, select the DisplayManager (and then click View Scene, Lights and Cameras ().
- 9 There are two directional lights that are part of this scene. When PhotoView 360 is turned on, there will be two icons next to each light. The left icon is the light state (On/Off) in OpenGL and RealView and the right icon is the light state in PhotoView 360.

 Scene, Lights, and Cameras

 Scene, Lights, and Cameras

 Lights

 Lights

 Scene (Backdrop - Studio Ro

 Lights

 Scene Illumination

 Ambient

 Scene Directional1

 Scenera

We can see that both directional lights are on in OpenGL/RealView Note but off in PhotoView 360 .

10 Right click each of the directional lights and click **On in PhotoView** from the menu.

11 Examine the Preview Window. We now have more light and a shadow because the two directional lights are now on in PhotoView 360.

Image Review

Rendering is an iterative process. Rarely will you get the image you are looking for on the first attempt. We could continue to make adjustments to make it look better.

Some things we can see in the rendering:

• We can get a good idea of the light positions from the shadows, but we

Lesson 9: Presenting Results



may want the shadows to be in different positions. The shadow from the front flange makes it difficult to see the details of the back flange.

- The highlighted surfaces look good, but we can't see the vertical surfaces very well nor the rib structure.
- The shadows are much too sharp. You only get sharp shadows with tightly focused lights. Most shadows have softer edges.

Mountainboard Design Project with SolidWorks

Task 2 — Changing the light intensity

We are going to do two things to refine our rendering. First is to change position and intensity of the directional lights. We will use these lights to shine into some of the dark areas. Second, we will adjust the shadows.

Basic

Light Properties

Each light has a set of properties that control its behavior in PhotoView 360 as well as OpenGL and RealView. The Basic tab controls the light properties for OpenGL and RealView plus the position of the light. The PhotoView tab controls the light intensity and shadow properties for PhotoView 360 only.

Light Intensity

As we have seen, the individual lights can be turned on and off in PhotoView 360 separately from OpenGL/RealView. Besides just turning the lights on and off, their intensity and shadows are also controlled separately.

- 1 In the DisplayManager double-click the light Directional1. The PropertyManager for this light will open.
- 2 The position of the light can be changed by dragging the manipulator or by the sliders and entry boxes in the PropertyManager.
- **3** For this rendering, drag the position of Directional1 to the approximate position shown. Observe the Preview Window to see the effects as it will update as you make changes. This position will make the light high over the model to get light behind the front flange so that we can see the back flange easier.
- 4 Select the PhotoView tab. Both **On in PhotoView** and **Shadows** should be selected. Increase the **Brightness** to 2 w/srm^2.

Click 🗹

5 Double-click the light Directional². This will open the properties of the other directional light and show the manipulator.

Note: We could also right-click the light and select Edit **Directional Light** to access the properties of the light.

6 Adjust the PhotoView 360 Brightness to a value of 2 w/srm².



504

7 Adjust the light position to -50 degrees Longitude and 13 degrees Latitude. Make sure that Lock to model is cleared. By clearing Lock to model, the light position is relative to our point of view, so it does not change when we rotate the model.

Click У .

8 Examine the preview. The directional light makes the rib structure easier to see, but it is also casting a shadow on the model and the floor. While some shadows are good, we have too many and the shadow in the front of the model is distracting.

Shadows

Shadows are important to the process of creating realistic renderings. They can be used to define spatial relationships

Without shadows, the relative position between model and the surface may be difficult to understand. The model may



look like it is sitting on the surface, but without shadows you can't tell. Adding shadows may show that the part is actually floating above the surface.

Shadow Control

In the physical world, all lights cast shadows. In the computer world, we can have lights that do not cast shadows. Shadows are controlled individually in the properties for each light.

Task 3 — Shadow Control

There are too many shadows in the rendering which causes shadow clutter. We will remove the shadow from the light Directional1, The shadow from the light Directional2 is very sharp so we will soften it just a little.

1 Edit light Directional1.

Select PhotoView tab.

Clear **shadows**. This will turn off the shadow for just this one light. Click **OK**.



- 2 Edit light Directional2. Set the **Shadow softness** to **3.00deg** and the **Shadow quality** to **32**. Shadow softness will blur the edges if the shadow. Increasing Shadow quality reduces the noise in the shadow areas and makes the rendering look smoother.
- **3** Final Render the model.

The one shadow on the back surface of the part and the floor, caused by the directional light, is now softer. The second shadow is not gone.



Dimetional 2
✓ X □
Basic PhotoView
Photoview Controls
On in PhotoView
Brightness
2 w/srm^2
Shadows 🕆
Shadow softness:
3.00deg 🚔
Shadow quality:
32

4 Save the part.

Active Learning Exercises, Part 5 — Texture Appearances

Texture Appearances

Texture appearances are like elastic wallpaper. They are applied to the outside of the model. They can be stretched and rotated to completely cover all the surfaces.

The textures are tileable, that is the pattern repeats so that you cannot see where one instance of the pattern stops and the next one starts. PhotoView 360 installs a variety of texture appearances, however, it is easy to create additional texture materials. We can do this from any image type that PhotoView 360 recognizes.

Task 4 — Add A Texture Appearance

We will add a texture appearance to the Deck, then customize it.

- 1 **Open** the part Deck.
- 2 Select the **Appearances**, **Scenes**, **and Decals** tab on the Task Pane.
- 3 Select the appearance **carbon fiber dyneema plain** in the **Plastics, Composite** folder. Hold the **ALT** key and drag the appearance into the graphics area.
- 4 The appearance is previewed on the Deck with a blue and magenta box that can be used to resize the mapping. We will use this resize feature once we change the material to something else.





If we can't find something we like, we can make our own material from available 5 graphic images. A variety of images are provided in the folder SolidWorks Curriculum and Courseware2011 Mountainboard Design Project\Images\Appearance Images.



multi134.jpg







multi173.jpg



multi136.jpg

multi099.jpg

multi015.jpg





multi149.jpg

multi036.ipa

multi101.jpg



multi203.jpg





multi150.jpg

multi044.jpg

multi105.ipg





multi127.jpg

multi047.gif







multi056.aif

multi132.jpg

multi217.jpg

multi222.jpg



multi240.jpg





multi175.jpg









The Deck is the one part of this product that can be customized with a variety of designs and decals.

We will use the red006 image in the following steps to create a new texture appearance. You can use any of the images supplied, or download an image from one of the many sites available on the web. This is your chance to show some individuality and show what you think the final Mountainboard should look like. Find an image that will capture the way you think this product should look.

Appearance Files

PhotoView 360 texture appearances require two files, an image file and appearance settings file.

- □ The image file (*.jpg, *.png, *.bmp, *.tga, *.tif) contains the pattern that will be used on the surface.
- \Box The appearance settings file (*.p2m) stores the location of the image file and the information on how the surface will reflect light.

Removing Appearance

Appearances can be removed from a part, feature or face by:

□ Editing the appearance and selecting a different appearance

Right-click the appearance and select Edit Appearance.

□ Detaching the current appearance

Right-click the appearance and select **Remove Appearance**.

Task 5 — Change the appearance applied to the Deck

We will use the appearance we have applied to the Deck as a starting point from which we will create a new appearance.

- 1 In the Appearances PropertyManager, select the Advanced button, then select the Color/Image tab. Under Appearance, click Browse and navigate to the SolidWorks Curriculum and Courseware_2011\Mountainboard Design Project\ Images\Appearance Images folder.
- 2 Select the image red006.jpg or any other image you would like to use.



3 Click Open.

4 Because this is a new appearance, we must save the new appearance file. It will have a file extension of *.p2m. Save the file to the same folder as the image with the same name.

🕅 Save As					X
💭 🗸 🖟 « Images 🕨 Appearance Imag	es 🔻	4 7	Search Appearance Im	ages	٩
Organize 🔻 New folder			:==	•	0
 SolidWorks Backup Directory SolidWorks Common SolidWorks Curriculum_and_Cout SolidWorks Curriculum_and_Cout Mountainboard Design Project_ Mountainboard Design Project_ 	Name red006.p2m		A	Date 2/21/	modifi 2008 2:
Appearance Images Decal Images Scene Images	•	I			4
File name: red006.p2m Save as type: Appearance Files (*.p2m) Description: Add a description					•
) Hide Folders			Save	Cancel	

- **5** SolidWorks will ask if you want to open this new folder in the **Appearance folder**, click **Yes**.
- **6** The new folder will appear in the Appearances section of the Task Pane with the new material listed.
- 7 n the Appearance section of the PropertyManager, we can see the two elements of the material in the PropertyManager. The image (*.jpg) is what the material looks like, and the material file (*.p2m) contains the settings for how to apply the image to our model.

Арр	earance	~
	Appearance file path:	
	c:\solidworks curriculum_and_courseware_2011\mountainboard design project_teacher\images\appearance images\red006.p2m	
	Browse	
V (iolor	≽
Ima	ge	*
	Image file path:	
	C:\SolidWorks Curriculum_and_Courseware_2011\Mountainboard Design Project_Teacher\Images\Appearance Images\red006.jpg	
	Browse	

- 8 The image is interesting, but we would only like to see one pattern instance stretched over the top surface.
- **9** Select the **Mapping** tab in the Appearance PropertyManager.



red006 🗙 -)=

Basic

Color/Image

V Illumination

Advanced

🔥 Mapping

- 10 Clear Fixed aspect ratio, this will allow us to control the width and height separately.
- 11 We can either use the drag box in the graphics area to adjust the size of the image or type values directly into the PropertyManager. Notice that next to the Height and



Width boxes is a blue and magenta image showing which length is controlled by each number.

12 Use the drag box to adjust the image so that it looks like the image. You could also type

800mm for the Width and 225mm for the Height. Click 🗹 .

13 Select the **Surface Finish** tab. Select **None** from the list.

Click 🖋 .



We have stretched a single instance of the pattern to cover the entire top face of the Montainboard.

Experiment with the setting to see if you can make the rendering look better.



Note: In most situations, you will not determine the correct settings for your rendering on the first try. There may be a lot of trial and error associated with finding the correct look.

15 In the final product, the pattern would not actually go through the entire deck, rather it would be a thin laminated layer. Because we applied a texture material, the material is wallpapered to each face individually. To correct this problem, we will add a different appearance to the chamfer feature and the side faces.

- 16 In the FeatureManager design tree select the feature Chamfer1. In the Appearances, Scenes and Decals tab of the Task Pane, select the Plastic, High Gloss folder.
- 17 Right-click the appearance **blue polished ABS plastic** and click **Add appearance to selection**. This will apply the material to just the chamfer feature.
- 18 In the DisplayManager, click View Appearances . Both appearances will be listed. Right-click the blue polished ABS plastic appearance and click Edit Appearance. Select the Color/ Image tab.
- **19** Change the color to black by moving the three color sliders all the way to zero.
- 20 Click 🗹 .

21 Examine the model.

chamfers.

appearances.

22 The chamfers are black, but we still have to do the thin faces between the

23 Stay on the DisplayManager tab so

Right-click one of the faces between the two chamfers and click **Select Tangency**. Don't worry if you don't get them all, we can add faces later.

that you can see the two



24 In the DisplayManager, right-click the appearance blue polished ABS plastic, then click Attach to Selection.

The individual faces are added to the material selection.





25 Render the model to check your work.



Active Learning Experience, Part 6 — Add Decals

In this exercise we will add a decal to the top of the Deck. In many cases, this will be a manufacture's logo or product identification.

Decals

Decals are similar to texture appearances except for two differences:

□ We only use one instance of the image.

□ We can mask out parts of the image to let the material behind the decal show through.

Decals are separate files, just like texture materials. They are made with essentially the same procedure used to create texture appearances.

Task 1— Create a Decal file

We have a JPG image file with our logo that we want to add to the top face of the Deck.

1 We will create a new decal using essentially the same method we used to create a new texture appearance in that we will use an existing decal and modify it to use our own image.



- 2 Click **Edit Decal [a]** on the Render Tools toolbar.
- **3** The **Decal** PropertyManager opens. We will make a decal from the file provided.
- 4 Under Image file path, Click Browse....
- 5 Navigate to the SolidWorks Curriculum and Courseware_2011\Mountainboard Design Project\Images\Decal Images folder.
- 6 Select the file SolidWorks.jpg, then click **Open**.

🕎 Open	×
🚱 🔍 🗣 📕 « Images 🕨 Decal Images 🔹 🔸	Search Decal Images 🔎
Organize 🔻 New folder	
 MSOCache NVIDIA PerfLogs Program Files Program Files (x86) Program Data Recovery SolidWorks 2010 Data SolidWorks 2011 Data SolidWorks 2011 Data 	SolidWorks
SolidWorks.JPG Rating: ជំ ជំ ជំ ជំ ជំ ជំ JPEG image Dimensions: 584 x 342	
Configurations:	Display States (linked) 🔻 Do not load hidden components
File name: SolidWorks.JPG 👻	All Image Files (*.bmp;*.hdr;*.jp 💌 Open 💌 Cancel

Note: The image has a tan background color. This was done in the image editing software to make it easier to mask out the background.

514



7 Click Save Decal....

Save the decal file as SolidWorks.p2d to the ...\SolidWorks Curriculum and Courseware_2011\Mountainboard Design Project\Images\Decal Images folder.

Click Save.

🕎 Save As	×
💭 🗸 🕌 « Images 🕨 Decal Images	✓ ✓ Search Decal Images
Organize 🔻 New folder	ii - 🔞
 SolidWorks Common SolidWorks Curriculum_and_Cour Mountainboard Design Project_ Mountainboard Design Project_ Images Appearance Images Decal Images Scene Images Scene Images 	No items match your search.
File name: SolidWorks.p2d	•
Save as type: Decal Files (*.p2d)	•
Description: Add a description	
Remove bend lines	
Hide Folders	Save Cancel

- 8 Click **Yes** to have PhotoView 360 open the Decals folder in the Task Pane.
- **9** The **Decal** PropertyManager will show the image.



10 In the graphics area, zoom in on the area shown and select the face indicated.



11 The Preview shows the decal on the selected face.



Mapping

Η

🟡 0.00deg

Projection

0.000mm

0.000mm

Size/Orientation

Fixed aspect ratio Fit width to selection

Fit height to selection

114.25241828mm 1 66.90809427mm

Aspect ratio: 1.71 : 1

Mirror horizontally Mirror vertically Reset to Image

Selected Reference

•

COMPANY

*

<u>.</u>

12 Select the Mapping tab in the **Decal** PropertyManager.

> Notice that the colors used in the Preview correspond to the color shown in the Mapping tab indicating:

- Width Blue
- · Height Magenta
- · Horizontal Location Red
- Vertical Location Green

- 15 Make sure Fixed aspect ratio is selected.
- 16 Type 270.00deg for Rotation Angle the press Enter.
- 17 The preview will show that the decal has been rotated to the correct orientation. If the decal is upside down, use 90 deg instead of 270 deg.



- 18 Adjust the size and position of the decal by typing the following values. Watch the preview as you enter the values. You can also adjust the size by using the drag box, just as we did for the texture material.
 - Horizontal Location:10 mm
 - Width: 50mm





19 Examine the model.

The decal is located and sized correctly but we only want the black ellipse and the image inside of it. We do not want the tan colored area. To remove the image outside of the ellipse, we will use a mask.

- In the DisplayManager, select View
 Decals 1
 Right-click the decal
 SolidWorks and select Edit Decal.
- 21 Select the **Image** tab.



22 Select Image mask file.

Click **Browse...** under **Image mask file** and locate the file SolidWorks-mask.jpg in the Images\Decal Images folder.



Image Mask

The image mask is a black and while image that will be overlaid on the decal image. Where the mask is white, the decal will show through. Where the mask is black, the decal will not show.

23 The Resulting image shows that all the area with the pink and white cross hatch will be masked out. Only the black ellipse and the image inside of it will be placed on our model.



24 Examine the model.

The tan area has now been masked out leaving only the ellipse and the image inside.

Notice that we have applied the decal inside SolidWorks without using PhotoView 360.



- **25** Render preview. Check the rendered preview to see how the final rendering will look.
- 26 Save and Close the Deck.



Active Learning Experience, Part 7 — Adding Appearances to assembly components

We will add appearances to all the remaining components of the mountainboard.

Adding Appearances to Assemblies

Appearances can be applied at the part or assembly level. At the part level we can add the appearance to the entire part, a feature, body or face. At the assembly level we can add the appearance to the entire assembly or individual components. We can not add appearances to features, bodies or faces while at the assembly level.

Where we apply the appearance depends on the different renderings we intend to create. If we apply an appearance to the entire part while in a part file, every time that part is used in an assembly it will render with the appearance.

If we apply appearance to a part while in the assembly, only that part instance will render in that appearance.

Note: When we created the parts for the mountainboard, we applied appearances to many of the parts when we created them. If you have already applied appearances to the parts, you will not have to apply them again in the following steps, however the steps are provided for completeness and in some cases, different choices of appearances are used.

Task 1— Add appearances to the Spring Assembly

- 1 **Open** the Spring Assembly.
- 2 Select the Appearances, Scenes, and Decals tab (in the Task Pane then click the push pin is to keep the pane open .

3 Locate the appearance **chromium plate** in the **Metals, Chrome** folder. Drag the appearance onto the Spring part in the FeatureManager design tree.

Click 🗹 . This will apply the appearance to just the Spring part.



- 4 Use the same procedure to apply the other appearances to the parts in this assembly:
 - Fender Washer Metal, Steel, brushed steel
 - Spring Dampener Plastics, High Gloss, yellow high gloss plastic
 - Spring Retainer Plastics, High Gloss, black high gloss plastic

Note: You can apply the material to one Spring Retainer, then select the second Spring Retainer, right-click the material and select **Attach to Selection**.

- **5 Render** the model.
- 6 The spring is chromium plate which shows reflections. If you look carefully at the reflections you can see that the default scene actually wraps around the model and behind your point of view.
- 7 Save and Close the Spring Assembly.



Task 2 — Add appearances to the Wheel Assembly

We can add appearances to the hardware and tire in the assembly, but we must add appearances at the part level for the Wheel Hub and Tube because we need multiple appearances on each of these parts.

The text on the Wheel Hub needs to be a different color to stand out from the rest of the Hub. The Inner Tube was modeled as a single part even though a real inner tube would be made up of several pieces: rubber tube, valve stem, valve, valve stem cap. We only need to add appearances to the surfaces that will show. because most of the inner tube is unseen because it is inside the tire. We will only add appearances to the valve stem and valve stem cap.

- 1 **Open** the Wheel Assembly.
- **2** Apply the following appearances:
 - Tire Rubber, Matte, matte rubber
 - Bearings, nuts and bolts Metals, Steel, polished steel
- **3 Open** the Wheel Hub in its own window by right-clicking the Wheel Hub in either the graphics area or FeatureManager design tree and selecting **Open Part**.
- 4 Apply the appearance **light grey medium gloss plastic** from the **Plastics, Medium Gloss** folder to the entire part. The default color should be gray, but it is a little too dark. Change to a lighter gray by adjusting the Red, Green and Blue color sliders to 230.
- 5 In the FeatureManager design tree select the two features Text and CirPattern2.
- 6 In the Task Pane, double-click the appearance **grey high gloss plastic.** This will apply the appearance to just these two features.

- 7 Check the render preview of the part to check your work.
- 8 Save the part.
- 9 Return to the Wheel Assembly by clicking Window and selecting Wheel Assembly from the list.
- **10 Open** the Inner Tube in its own window.



- 11 Zoom in on the valve stem and cap area.
- **12** Select the three yellow surfaces shown.
- 13 Apply the appearance polished brass from the Metals, Brass folder by double-clicking the appearance in the Task Pane.
- 14 Select all the visible faces of the valve stem cap.
- **15** Apply the material **black medium gloss plastic** from the plastics, Medium Gloss folder.
- 16 Save the part.
- 17 Return to the Wheel Assembly by clicking Window and selecting Wheel Assembly from the list.
- 18 Select the Appearances, Scenes, and Decals tab on the Task Pane and then expand Scenes folder. Select Presentation Scenes. Drag the scene Courtyard Background into the graphics area.
- **19** Examine the preview render to check your work.
- 20 Save the assembly.



Mountainboard Design Project with SolidWorks



Task 3 — Add materials to the Bindings

The two bindings and the hardware inserted at the top level assembly are the only components that do not yet have appearances applied.

- 1 Open the Binding assembly.
- **2** Apply the following materials:
 - Binding Pad matte rubber
 - Clasp assembly **polished steel**
 - Strap right glossy rubber
 - Strap left -glossy rubber
 - Binding Base Plate polished aluminum
 - Foam Pad curved white high gloss plastic
 - Binding Anchor polished aluminum
- **3** Save and Close the assembly.



4 Apply the same appearances to the MirrorBinding assembly.

Active Learning Exercises, Part 8 — Final Rendering

The final step is to apply appearances to the fasteners in the top level assembly, then add scenery and do the final rendering.

Task 1— Apply appearances to the fasteners and hardware

- 1 **Open** the Mountainboard assembly.
- 2 Select all the fasteners and hardware. Because we put all these files in their own folder they are easier to find and select. Click the plus sign next to the Hardware folder to expand the listing. Select the first fastener in the list, then press and hold the **Shift** key and select the last fastener in the list. This will select the two fasteners you selected plus everything in between.
- **3** Double-click the appearance **polished steel** from the **Metal, Steel** folder.

Task 2 — Add a scene

To help add realism to our final rendering, we would like to show the mountainboard outside on a trail.

We will first add a standard scene, then use a digital photograph as a background to add realism to the final output.

- 1 Reorient the Mountainboard to the Isometric view.
- 2 Examine the **Basic Scenes** in the **Appearances**, **Scenes**, **and Decals** tab of the Task Pane and look at the available choices.

None of these studios are really what we want, so we can pick anyone of the existing bases then modify it.

- **3** Drag the **Rooftop** scene into the graphics area. The rooftop environment will place a sky around the model so that reflections on the shiny surfaces will look correct.
- In the DisplayManager, select View Scene, Lights and Cameras <a>[mailtoin]. Right click
 Scene (Rooftop) and click Edit Scene.
- 5 For Background, select Image from the list and then click Browse. Select the image Trail.jpg from the Images\Scene Images folder of the course files. The trail image is the background while the image sky.hdr is the environment.
- 6 Under Floor select **Floor shadows**. The floor itself will be invisible, but will show where the shadows fall.
- **7** Adjust the position of the Mountainboard to an appropriate position over the trail.



8 Examine the preview render the model. Even though we turned on the shadows in the scene properties, the only shadows are from the environment and they are hard to see.

If we were on the trail, we would expect light from the sun to cast shadows on the ground. To simulate the sun, we will use one of the directional light.

- 9 In the DisplayManager, right-click one of the directional lights and click On in PhotoView.
- **10** Edit the lights properties and select the PhotoView tab. Select **Shadows**.



- 11 Select the Basic tab and adjust the light position so that the shadows from the light are visible and in a suitable position.
- 12 When you are satisfied with the preview, create a final rendering and save it with the other mountainboard files.

Task 3 — Additional PhotoView 360 practice

Now that we have a suitable rendering, it is time to experiment with changes. Try some of the following:

- □ Add your school logo as a decal.
- □ Use a different image as the background scene.
- □ Change the lighting.
- □ Change the material color of the wheels.



□ Create a new appearance and apply it to the Deck.



Active Learning Exercise, Part 9 — Animations

To show off our mountainboard we will create an animation that we can send to prospective clients.

Storyboard

To create a good animation, we first need an idea of what we want the finished animation to look like. We put our ideas into a storyboard which lays out the different elements of the animation. There is no set format for storyboards, and the amount of detail depends on the level of complication of the final video. For for very short animations we just may write down a few steps in a numbered list. For more complicated animations we may layout a timeline.

Our first animation will be relatively simple so the storyboard will be like this:

- Start with the mountainboard in the center of the screen but zoomed out so it is quite small.
- Zoom in until the mountainboard fills the screen.
- Rotate the mountainboard one full turn so we can see it from all sides.
- Zoom out so that we can explode the assembly without parts going out of view.
- Explode the assembly.
- Rotate the mountainboard one full turn so we can see it from all sides.
- Collapse the assembly.
- Zoom out until we are back to where we started.

Task 1— Establish viewpoints

Viewpoints establish the camera position. While we could do these "on the fly" while creating the timeline, it is easier to save the different viewpoints as named views.

- 1 Orient the mountainboard to the Isometric view.
- 2 Make the **Exploded** configuration active.
- **3** Zoom out so that the mountainboard is very small in the center of the screen.
- 4 Click View, Modify, Orientation.
- **5** Click the pushpin \rightarrow to keep the box on the screen.
- 6 Click New View 👸 in the Orientation box.
- 7 Type Start for the name of the view and click OK.
- 8 The named view Start will appear as a view in the Orientation box.
- **9** Zoom in until the mountainboard fills the screen.
- 10 Click New View 隊 in the Orientation box.
- 11 Type **Zoomed In** for the name of the view and click **OK**.
- 12 Click the ConfigurationManager tab.

- **13** Explode the assembly.
- 14 **Zoom** and **Move** the assembly until it fits on the screen.
- **15** Click **New View** 隊 in the box.
- 16 Type **Explode** for the name of the view and click **OK**.
- **17** Collapse the assembly.
- 18 Use View, Orientation to zoom to the Start view.

MotionManager Interface

The animation process is based on a key frame-based interface. You decide how your assembly should look at various times, and then Animator computes the sequences needed to go from one position to the next.



Change bars and key frames are color coded to show their function.

MotionManager Toolbar

The MotionManager has its own toolbar located above the timeline.



Animation Mode

Animation Mode allows the animation to;

- □ Play once through (Normal),
- □ Play beginning to end repeatedly (Loop)
- □ Play forward to the end then back to the beginning (Reciprocate)

⇒	Normal
Ģ	Loop
⇔	Reciprocate

Task 2 — Animate Viewpoints

1 Select the **Motion Study 1** tab at the bottom of the graphics window.

This will open the Timeline.

2 In the MotionManager FeatureManager design tree, right-click **Orientation and Camera Views** and clear **Disable View Key Creation**.

When a new animation is started, the **View Orientation** is locked to prevent the MotionManager from placing keypoints every time you rotate, pan or zoom the model.

- 3 Drag the **Timebar** to four seconds.
- 4 Click once in the graphics area then press the spacebar to open the **View Orientation** dialog.

Note: We had to click once in the graphics area to set the focus. We essential told SolidWorks to interpret commands for the graphics area instead of the animation timeline.

5 Double-click the named view **Zoomed In**.

A key will be inserted at the four second point. The heavy black Changebar indicates the view will change from the Start view to the

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Zoomed In view from time zero to four.

Animation Wizard

The **Animation Wizard** can be used to automate some of the animation steps. The wizard can be used to animate rotation, explode, collapse and physical simulations.

Note: Explode and **Collapse** are only available after an exploded view has been created.

Task 3 — Use the Animation Wizard to create a view rotation

The Animation Wizard makes it easy to rotate the model abut the three axis of our screen.

Select an Animation Type

- 1 Click **Animation Wizard** ion the Animator toolbar.
- 2 Select **Rotate Model** for the type of animation.
- 3 Click Next.



- 4 Select the **Y-axis** for the axis of rotation.
 - **Note:** The axes of rotation are based on the computer screen. The X axis is a horizontal axis through the center of the screen. The Y axis is a vertical axis thought the center of the screen. The Z axis is normal to screen center.
- **5** Type **1** for the number of rotations and select **Clockwise** for the direction.
- 6 Click Next.



7 We want to pause the animation for one second between the time the model is zoomed in and when the rotation starts. Type 4 for the Duration and 5 for the Start Time.

We were fully zoomed in at 4 seconds so starting the rotation at 5 seconds causes the pause from 4 to 5 seconds.

Click Finished.

8 Review the animation. Click



Animation Control Options	
	To control the speed of the animation, set the duration of the entire animation below. Duration (seconds): 4 To delay the movement of objects at the beginning of the animation, set the start time. Start Time (seconds): 5
	< Back Finish Cancel Help

The MotionManager will step through the animation. It may be a little jerky at this point, but don't worry about it.

- **9** We want to hold the Zoomed In view for one second before we change the view to the named view Exploded. Drag the **Timebar** to **10** seconds.
- 10 Right-click the **Timebar** in line with the **View Orientation** feature and select **Place Key**.



- 11 Drag the **Timebar** to **13** seconds.
- 12 Click once in the graphics area then press the spacebar to open the **View Orientation** dialog.
- **13** Double-click the named view **Explode**.

Another Key Frame will be added and a new Changebar.

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14 Review the animation. Click **Play from Start b** on the **Animation** toolbar.

Task 4 — Animated Explode

- 1 Click Animation Wizard 🔛 on the Animator toolbar.
- 2 Select **Explode** and then **Next**.

Select an Animation Type	×
	This wizard will help you to create simple animations automatically. To begin, select the type of animation you want to create and click Next.
Delete all existing paths	 Import motion from Motion Analysis
Motion Analysis is available only if the have been calculated in a motion st	he SolidWorks Motion add-in is loaded and results tudy. Back Next > Cancel Help
Animation Control Options	X
	To control the speed of the animation, set the duration of the entire animation below. Duration (seconds): 5 To delay the movement of objects at the benirpring of the animation set the start time

< Back

Finish

Cancel

Help

3 Type 5 for Duration and 14 for Start Time.

Click Finish.

4 Examine the Timeline. All the component that move during the explosion have changbars.



- **5** Use the **Animation Wizard** to **Collapse** the assembly from time **20** seconds for a duration of **5** seconds.
- 6 Place another View Orientation key at 25 seconds.
- 7 Move the timebar to 28 seconds and change the view to **Zoomed in**.

- 8 Use the Animation Wizard to rotate the assembly from **19** to **22** seconds.
- **9** Review the animation. Click **Play from Start** | **b** on the **MotionManager** toolbar.

Task 5 — Save the animation as an AVI file

To make the animation viewable on other computers or viewing systems, we must save it as an AVI file. This file type can be viewed by Windows Media Player as well as many other programs.

- 1 Click **Save** on the MotionManager toolbar.
- 2 Save the animation to the file Mountainboard.avi.
- **3** Select **SolidWorks screen** for Render.
- 4 Type **10** for **Frames per second** and select **Entire animation**.
- 5 Click Save.

🛐 Save Animation to File	×
Save in: 🌗 Mountainboard	- 🕝 🏚 📂 🛄+
Name	Date modified
Axle-Truck Binding Hardware Parts	3/29/2010 3:02 PM 3/29/2010 3:02 PM 4/1/2011 1:38 PM 4/1/2011 1:40 PM
June 2017	5/29/2010 5:02 PIVI +
File name: Mountainboard.avi Save as type: Microsoft AVI file (* avi) Renderer: SolidWorks screen Image Size and Aspect Ratio F ## 1507 \$10	Save Schedule Cancel Help rame Information rames per second 10
✓ Fixed aspect ratio	Entire animation
Use camera aspect ratio) Time range
O Custom aspect ratio (width : height) 1507 : 591 ▼ ✓	22 to 22

CODEC

Video files can be quite large, so to reduce their size, file compression is generally used. CODEC is short for COmpressor/DECompressor. CODEC is any technology used to compress and decompress data. Different technologies perform this in different ways with either hardware, software, or a combination of the two.

The CODECs available on each computer may be different and will depend on the video products that have been loaded. It is important to keep this in mind when choosing the CODEC to compress your video files because the destination computers must also have the same CODEC loaded.

6 Select **Microsoft Video 1** for the Compressor. This CODEC is supplied with the Windows operating system so it should be available on any computer you wish to play the animation.

Video Compression	×
Compressor:	ОК
Microsoft Video 1	Cancel
Compression Quality: 85	Configure
Key Frame Every 8 frames	About

- 7 Click OK.
- 8 Observe the video recording process. The MotionManager is essentially saving a series of images, that will be shown in rapid succession during playback.

We set the frame rate to 10 frames per second so the MotionManager records an image of the assembly, then moves all parts to where they would be 1/10 second later and records another image. Our total animation is 28 seconds long, so animator will record 281 images (28 seconds x 10 frames per second + one frame at time zero).

9 When the animation process is finished, Open Microsoft Media Player, or any other media player you have available, and play back the animation.

Notice that the motion is much smoother than when we previewed it using the MotionManager. During preview, the frame rate may be slower as each position must be calculated. When we playback the recorded AVI file, the frames do not have to be calculated, just displayed.

10 Save and Close all files.

MotionManager and PhotoView 360

PhotoView 360 can be used in conjunction with MotionManager to photorealisticly render each frame of the animation. This process can take considerable time as 281 renderings would have to be done to show our animation rendered photorealisticly.

5 Minute Assessment - #9-2

- **1** What is PhotoView 360?
- 2 List the rendering effects that are used in PhotoView 360?
- **3** The PhotoView 360______ allows you to specify and preview materials.
- **4** Where do you set the scene background?
- **5** What is SolidWorks MotionManager?
- 6 List the five types of animations that can be created using the Animation Wizard.

More to Explore

Exercise 26: Drawings

Create a set of drawings for the Mountainboard.

Exercise 27: Exploded Views

Create assembly instructions for the Mountainboard using exploded view drawings.

Exercise 28: Create a PowerPoint[®] Presentation

Create a presentation covering the design process of the Mountainboard.

Exercise 29: Written Report

Write a report detailing the design of the Mountainboard. Use images of the various parts and assemblies. Include a description and images of the analysis done on the different parts.

Exercise 30: PhotoView 360

Create a marketing brochure using photorealistic images of the Mountainboard and its components.

Exercise 31: PhotoView 360 and Animator

Create a web page showing the rendered Mountainboard and an animation of the explode and collapse steps.
Lesson 9 Vocabulary Worksheet

N	Name: Class: Date:	
Directions: Answer each question by writing the correct answer or answers in the space provided.		
1	1 The ability to dynamically view an eDrawing:	
2	2 Halting a continuous play of an eDrawing animation:	
3	3 Command that allows you to step backwards one step at a time through an eD animation:	rawing
4	4 Non-stop replay of eDrawing animation:	
5	5 Rendering of 3D parts with realistic colors and textures:	
6	6 Go forward one step in an eDrawing animation:	
7	7 Command used to create an eDrawing:	
8	 8 Graphic aid that allows you to see the model orientation in an eDrawing create SolidWorks drawing: 	ed from a
9	9 Quickly return to the default view:	

10 Command that allows you to use email eDrawings with others:

Lesson 9 Quiz

ections: Answer each question by writing the correct answer or answers in the space vided. What is the window that shows you a thumbnail view of the whole eDrawing?
What is the window that shows you a thumbnail view of the whole eDrawing?
Which command displays wireframe as solid surfaces with realistic colors and extures?
How do you create an eDrawing?
What action does the Home command perform?
Which command performs a non-stop replay of eDrawing animation?
Frue or False — eDrawings only displays part files, but not assemblies or drawings.
Frue or False — You can hide assembly components or drawing views.
n an eDrawing created from a SolidWorks drawing, how do you view a sheet other han the one currently displayed?
What visual aid helps you identify model orientation in a drawing?
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?
What is PhotoView 360?
What is SolidWorks MotionManager?

- 14 Image Background is the portion of the graphics area not covered by the _____.
- **15** True of False. PhotoView 360 output renders to the graphics window or renders to a file.
- 16 SolidWorks MotionManager produces what type of file?
- 17 List the five types of animations that can be created using the Animation Wizard.
- **18** For a given animation, list three factors that affect the file size when the animation is recorded.

Lesson Summary

- □ eDrawings can be created quickly from part, assembly, and drawing files.
- □ You can share eDrawings with others even if they don't have SolidWorks.
- Email is the easiest way to send an eDrawing to others.
- □ Animations allow you to see all views of a model.
- You can hide selected components of an assembly eDrawing and selected views of a drawing eDrawing.
- □ PhotoView 360 creates photorealistic renderings of parts and assemblies.
- PhotoView 360 allows you to add materials, scenes, decals and lights to create images that look like photographs.
- □ PhotoView 360 can output to the computer screen, printer or image files.
- □ MotionManager is used to create animations of parts or assemblies.
- □ Animations are saved as AVI files.

animate	View a model or eDrawing in a dynamic manner. Animation simulates motion or displays different views.
appearance	Appearances are applied to parts, features, faces, bodies, components or assemblies to control visual properties.
assembly	An assembly is a document in which parts, features, and other assemblies (sub-assemblies) are mated together. The parts and sub- assemblies exist in documents separate from the assembly. For example, in an assembly, a piston can be mated to other parts, such as a connecting rod or cylinder. This new assembly can then be used as a sub-assembly in an assembly of an engine. The extension for a SolidWorks assembly file name is SLDASM. See also sub-assembly and mate.
axis	An axis is a straight line that can be used to create model geometry, features, or patterns. An axis can be made in a number of different ways, including using the intersection of two planes. See also temporary axis, reference geometry
block	A block is a user-defined annotation for drawings only. A block can contain text, sketch entities (except points), and area hatch, and it can be saved in a file for later use as, for example, a custom callout or a company logo.
boss/base	A base is the first solid feature of a part, created by a boss. A boss is a feature that creates the base of a part, or adds material to a part, by extruding, revolving, sweeping, or lofting a sketch, or by thickening a surface.
broken-out section	A broken-out section exposes inner details of a drawing view by removing material from a closed profile, usually a spline.
chamfer	A chamfer bevels a selected edge or vertex.
click-click	As you sketch, if you click and then release the pointer, you are in click-click mode. Move the pointer and click again to define the next point in the sketch sequence.

click-drag	As you sketch, if you click and drag the pointer, you are in click-drag mode. When you release the pointer, the sketch entity is complete.
closed profile	A closed profile (or closed contour) is a sketch or sketch entity with no exposed endpoints; for example, a circle or polygon.
collapse	Collapse is the opposite of explode. The collapse action returns an exploded assembly's parts to their normal positions.
component	A component is any part or sub-assembly within an assembly.
configuration	A configuration is a variation of a part or assembly within a single document. Variations can include different dimensions, features, and properties. For example, a single part such as a bolt can contain different configurations that vary the diameter and length. See design table.
Configuration Manager	The ConfigurationManager on the left side of the SolidWorks window is a means to create, select, and view the configurations of parts and assemblies.
coordinate system	A coordinate system is a system of planes used to assign Cartesian coordinates to features, parts, and assemblies. Part and assembly documents contain default coordinate systems; other coordinate systems can be defined with reference geometry. Coordinate systems can be used with measurement tools and for exporting documents to other file formats.
degrees of freedom	Geometry that is not defined by dimensions or relations is free to move. In 2D sketches, there are three degrees of freedom: movement along the X and Y axes, and rotation about the Z axis (the axis normal to the sketch plane). In 3D sketches and in assemblies, there are six degrees of freedom: movement along the X, Y, and Z axes, and rotation about the X, Y, and Z axes. See under defined.
design table	A design table is an Excel spreadsheet that is used to create multiple configurations in a part or assembly document. See configurations.
DisplayManager	The DisplayManager provides and outline view of display properties of the active document. These properties include appearances, decals, scenes, lights and cameras.
document	A SolidWorks document is a file containing a part, assembly, or drawing.
drawing	A drawing is a 2D representation of a 3D part or assembly. The extension for a SolidWorks drawing file name is SLDDRW.
drawing sheet	A drawing sheet is a page in a drawing document.

eDrawing	Compact representation of a part, assembly, or drawing. eDrawings are compact enough to email and can be created for a number of CAD file types including SolidWorks.
face	A face is a selectable area (planar or otherwise) of a model or surface with boundaries that help define the shape of the model or surface. For example, a rectangular solid has six faces. See also surface.
feature	A feature is an individual shape that, combined with other features, makes up a part or assembly. Some features, such as bosses and cuts, originate as sketches. Other features, such as shells and fillets, modify a feature's geometry. However, not all features have associated geometry. Features are always listed in the FeatureManager design tree. See also surface, out-of-context feature.
FeatureManager design tree	The FeatureManager design tree on the left side of the SolidWorks window provides an outline view of the active part, assembly, or drawing.
fillet	A fillet is an internal rounding of a corner or edge in a sketch, or an edge on a surface or solid.
graphics area	The graphics area is the area in the SolidWorks window where the part, assembly, or drawing appears.
helix	A helix is defined by pitch, revolutions, and height. A helix can be used, for example, as a path for a swept feature cutting threads in a bolt.
instance	An instance is an item in a pattern or a component that occurs more than once in an assembly.
layer	A layer in a drawing can contain dimensions, annotations, geometry, and components. You can toggle the visibility of individual layers to simplify a drawing or assign properties to all entities in a given layer.
line	A line is a straight sketch entity with two endpoints. A line can be created by projecting an external entity such as an edge, plane, axis, or sketch curve into the sketch.
loft	A loft is a base, boss, cut, or surface feature created by transitions between profiles.
mate	A mate is a geometric relationship, such as coincident, perpendicular, tangent, and so on, between parts in an assembly. See also SmartMates.
mategroup	A mategroup is a collection of mates that are solved together. The order in which the mates appear within the mategroup does not matter.

material	Materials are applied to parts to define the engineering properties. These properties include such properties as density and yield strength.
mirror	(1) A mirror feature is a copy of a selected feature, mirrored about a plane or planar face. (2) A mirror sketch entity is a copy of a selected sketch entity that is mirrored about a centerline. If the original feature or sketch is modified, the mirrored copy is updated to reflect the change.
model	A model is the 3D solid geometry in a part or assembly document. If a part or assembly document contains multiple configurations, each configuration is a separate model.
mold	A mold cavity design requires (1) a designed part, (2) a mold base that holds the cavity for the part, (3) an interim assembly in which the cavity is created, and (4) derived component parts that become the halves of the mold.
named view	A named view is a specific view of a part or assembly (isometric, top, and so on) or a user-defined name for a specific view. Named views from the view orientation list can be inserted into drawings.
open profile	An open profile (or open contour) is a sketch or sketch entity with endpoints exposed. For example, a U-shaped profile is open.
origin	The model origin appears as three gray arrows and represents the $(0,0,0)$ coordinate of the model. When a sketch is active, a sketch origin appears in red and represents the $(0,0,0)$ coordinate of the sketch. Dimensions and relations can be added to the model origin, but not to a sketch origin.
over defined	A sketch is over defined when dimensions or relations are either in conflict or redundant.
parameter	A parameter is a value used to define a sketch or feature (often a dimension).
part	A part is a single 3D object made up of features. A part can become a component in an assembly, and it can be represented in 2D in a drawing. Examples of parts are bolt, pin, plate, and so on. The extension for a SolidWorks part file name is SLDPRT.
pattern	A pattern repeats selected sketch entities, features, or components in an array, which can be linear, circular, or sketch-driven. If the seed entity is changed, the other instances in the pattern update.
planar	An entity is planar if it can lie on one plane. For example, a circle is planar, but a helix is not.

plane	Planes are flat construction geometry. Planes can be used for a 2D sketch, section view of a model, a neutral plane in a draft feature, and others.
point	A point is a singular location in a sketch, or a projection into a sketch at a single location of an external entity (origin, vertex, axis, or point in an external sketch). See also vertex.
profile	A profile is a sketch entity used to create a feature (such as a loft) or a drawing view (such as a detail view). A profile can be open (such as a U shape or open spline) or closed (such as a circle or closed spline).
Property Manager	The PropertyManager is on the left side of the SolidWorks window for dynamic editing of sketch entities and most features.
rebuild	The rebuild tool updates (or regenerates) the document with any changes made since the last time the model was rebuilt. Rebuild is typically used after changing a model dimension.
relation	A relation is a geometric constraint between sketch entities or between a sketch entity and a plane, axis, edge, or vertex. Relations can be added automatically or manually.
revolve	Revolve is a feature tool that creates a base or boss, a revolved cut, or revolved surface by revolving one or more sketched profiles around a centerline.
round	A round is an external rounding of an edge on a surface or solid.
section	A section is another term for profile in sweeps.
section view	A section view (or section cut) is (1) a part or assembly view cut by a plane, or (2) a drawing view created by cutting another drawing view with a section line.
shaded	A shaded view displays a model as a colored solid. See also HLR, HLG, and wireframe.
sheet format	A sheet format typically includes page size and orientation, standard text, borders, title blocks, and so on. Sheet formats can be customized and saved for future use. Each sheet of a drawing document can have a different format.
shell	Shell is a feature tool that hollows out a part, leaving open the selected faces and thin walls on the remaining faces. A hollow part is created when no faces are selected to be open.

sketch	A 2D sketch is a collection of lines and other 2D objects on a plane or face that forms the basis for a feature such as a base or a boss. A 3D sketch is non-planar and can be used to guide a sweep or loft, for example.
SmartMates	A SmartMate is an assembly mating relation that is created automatically. See mate.
sub-assembly	A sub-assembly is an assembly document that is part of a larger assembly. For example, the steering mechanism of a car is a sub- assembly of the car.
surface	A surface is a zero-thickness planar or 3D entity with edge boundaries. Surfaces are often used to create solid features. Reference surfaces can be used to modify solid features. See also face.
sweep	A sweep creates a base, boss, cut, or surface feature by moving a profile (section) along a path.
template	A template is a document (part, assembly, or drawing) that forms the basis of a new document. It can include user-defined parameters, annotations, or geometry.
toolbox	A library of standard parts that are fully integrated with SolidWorks. These parts are ready-to-use components — such as bolts and screws.
under defined	A sketch is under defined when there are not enough dimensions and relations to prevent entities from moving or changing size. See degrees of freedom.
vertex	A vertex is a point at which two or more lines or edges intersect. Vertices can be selected for sketching, dimensioning, and many other operations.
wireframe	Wireframe is a view mode in which all edges of the part or assembly are displayed. See also HLR, HLG, shaded.