SolidWorks®

Engineering Design Project

The Mountainboard

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U.S. Patents 5,815,154; 6,219,049; 6,219,055; 6,603,486; 6,611,725; 6,844,877; 6,898,560; 6,906,712; 7,079,990; 7,184,044; 7,477,262; 7,502,027; 7,558,705; 7,571,079; 7,643,027 and foreign patents, (e.g., EP 1,116,190 and JP 3,517,643).

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Document Number: PMS0718-ENG

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Introduction

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 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 appears: 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 not harm your system. If you click **No**, the scripts are disabled for that topic. Click **Yes** to run the scripts and flash the button.

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

Video example shows a video about this step.

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.

1

Lesson 1: Using the Interface

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

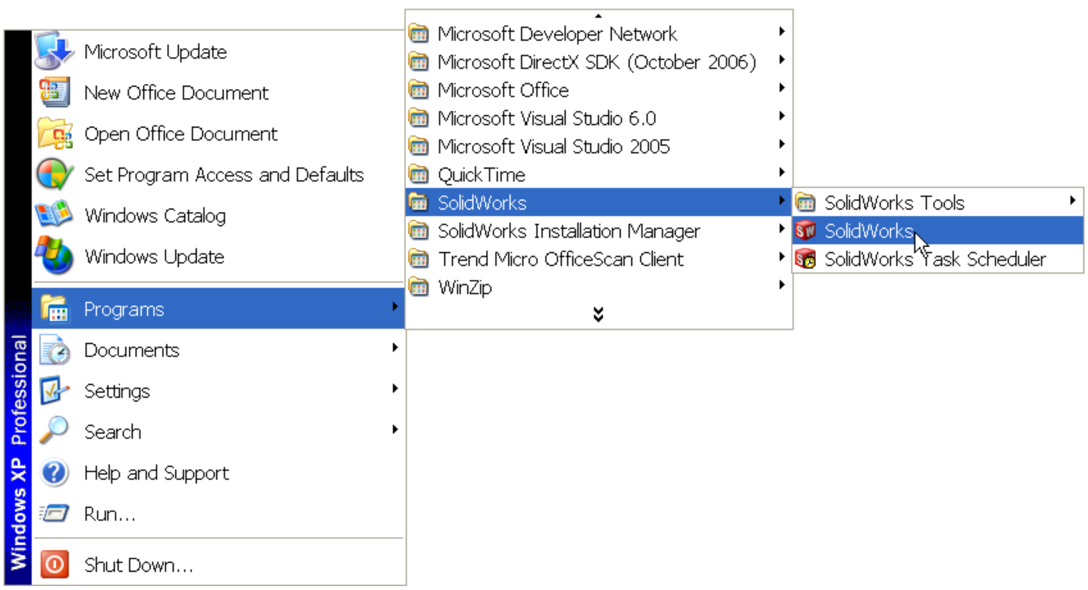
1 Click the **Start** button  in the lower left corner of the window. The **Start** menu appears. The **Start** menu allows you to select the basic functions of the Microsoft Windows environment.

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, 2010 SP2.1 for instance, may be included or not listed.

The SolidWorks application program is now running.



Note: Your **Start** menu may appear different than the illustration depending on which versions 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  on the main SolidWorks window.

Searching for a File or Folder

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

Windows has two methods to search for files, Microsoft Desktop Search and the Search Companion.

Microsoft Desktop Search

Microsoft Desktop Search is normally loaded when SolidWorks is installed on your computer. After installation, it must create an index, similar to the index in the back of most reference books. When you search using Microsoft Desktop search, the search program just looks in the index for the location of the information you are searching for and displays the result.

While Microsoft Desktop Search makes the search process much faster, it depends on creating and maintaining the index. If the index has not been created, we will not get a valid result from the search.

Search Companion

The Search Companion does not require any setup. When used to search, it starts at the beginning of the search path and checks everything. This is very much like searching for a topic in a book by starting at the first page and checking each page until you find the topic.

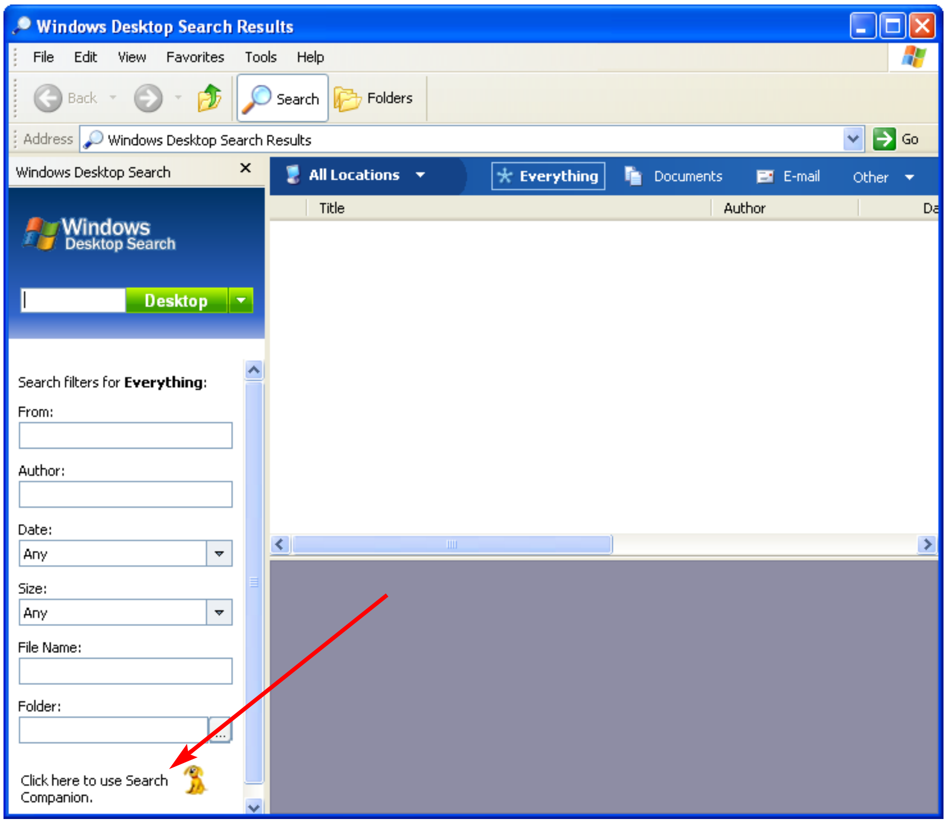
Search for a file

We will now use Search to fine a file.

1 Click **Start**, **Search**. Search for the SolidWorks part `dumbell`.

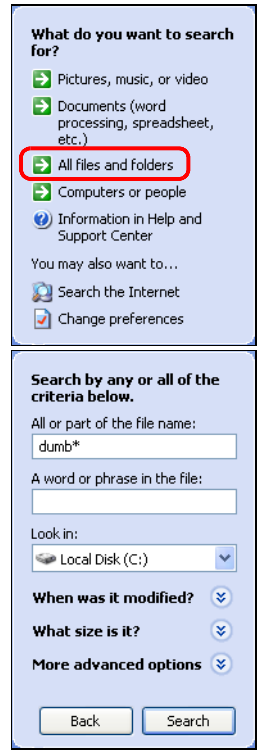
As we have not set up Microsoft Desktop Search, we will use the Search Companion.

2 If Microsoft Desktop Search is loaded on your computer, you will get the following window. Click **use Search Companion**.



Note: If Microsoft Desktop Search is not installed, you will go directly to the Search Companion.

3 Click **All files and folders**, then enter dumb* in the **All or part of the file name:** field.



Specifying what to search for and where to search for it is known as defining the search criteria.

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.

4 Click **Search.** The files and folders that match the search criteria appear in the **Search Results** window.

TIP: You can also begin a search by right-clicking on the **Start** button and selecting **Search**. Right-click means to press and release the right button on your mouse.

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. Double-clicking with the left mouse button is often a quick way of opening files from a folder.

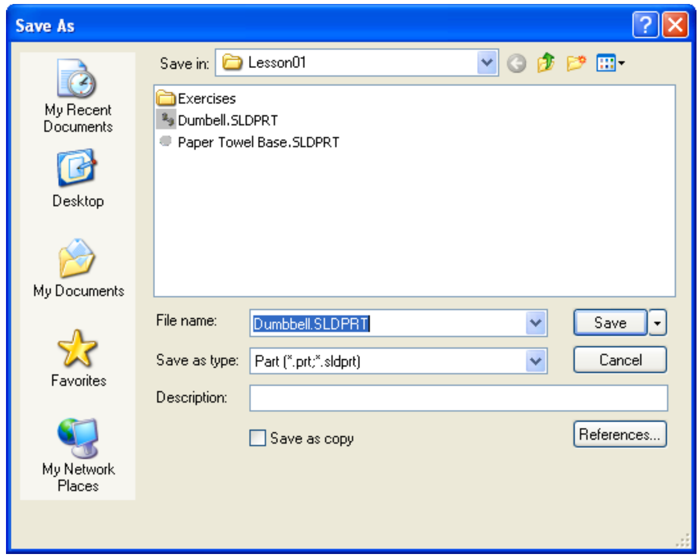
You could have also opened the file by selecting **Open**, **Open from Web Folder**, or a file name from the **File** menu in SolidWorks. SolidWorks lists the last several files that you had open.

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.

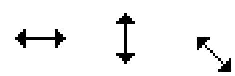
2 In the **File Name** field enter the name Dumbbell and click **Save**.

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.

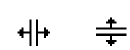


2 While the cursor still appears to be a two-headed arrow, hold down the left mouse button and drag the window to a different size.

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.

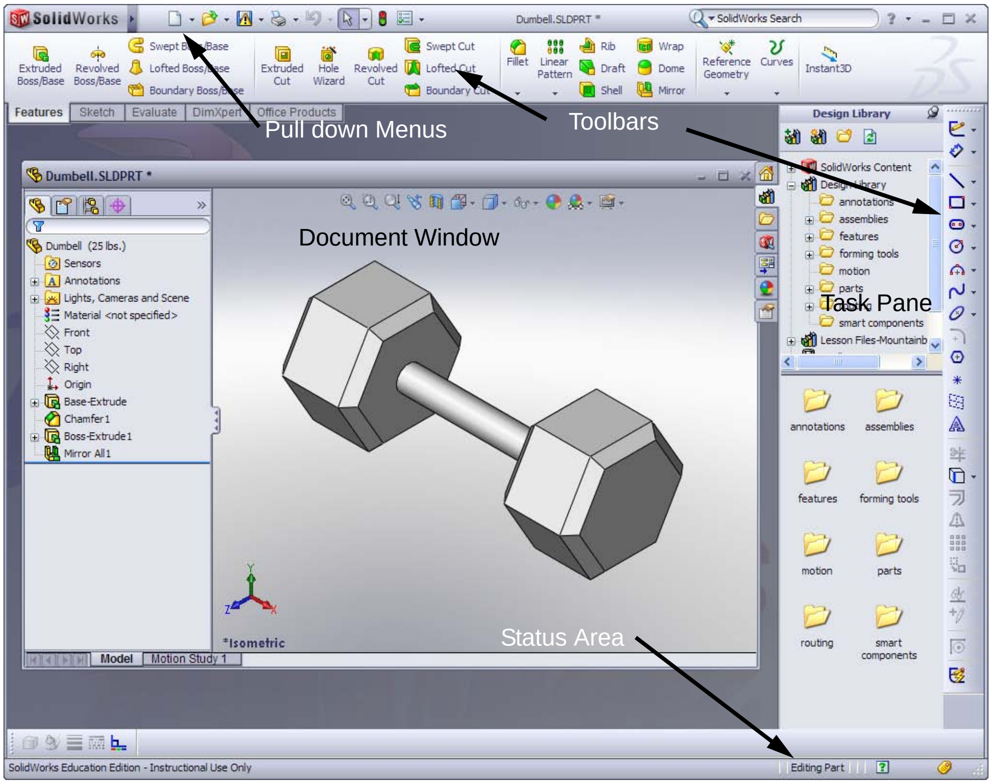


5 While the cursor still appears to be two parallel lines with perpendicular arrows, hold down the left mouse button and drag the panel to a different size.

6 When the panel appears to be the size that you wish, release the mouse button.

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

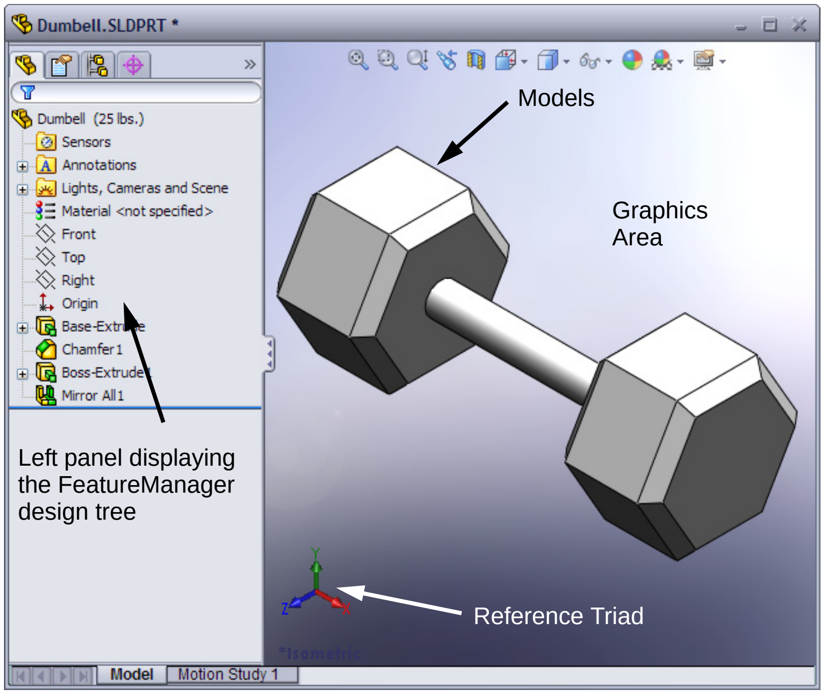
SolidWorks document windows have two panels. One panel provides non-graphic data. The other panel provides graphic representation of the part, assembly, or drawing.

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

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

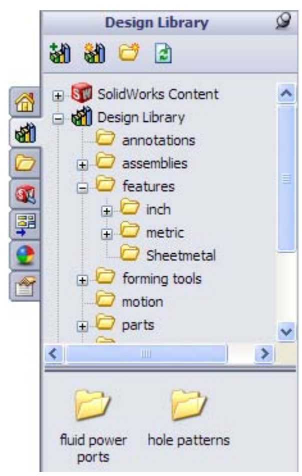
The rightmost panel is the Graphics Area, where you create and manipulate the part, assembly, or drawing.

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 seven or more panels: SolidWorks Resources, the Design Library, File Explorer, Search, View Palette, RealView 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.

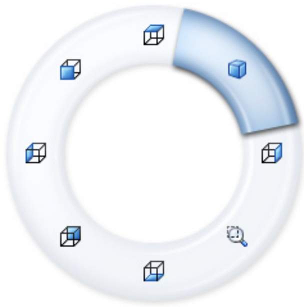


Mouse Buttons

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



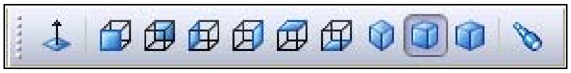
Mouse gestures provide a quick way to invoke up to eight different commands. They can be customized separately for sketches, parts, assemblies and drawings. To use mouse gestures, right-click in a blank area of the graphics window and then drag the mouse pointer over the desired command.

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.

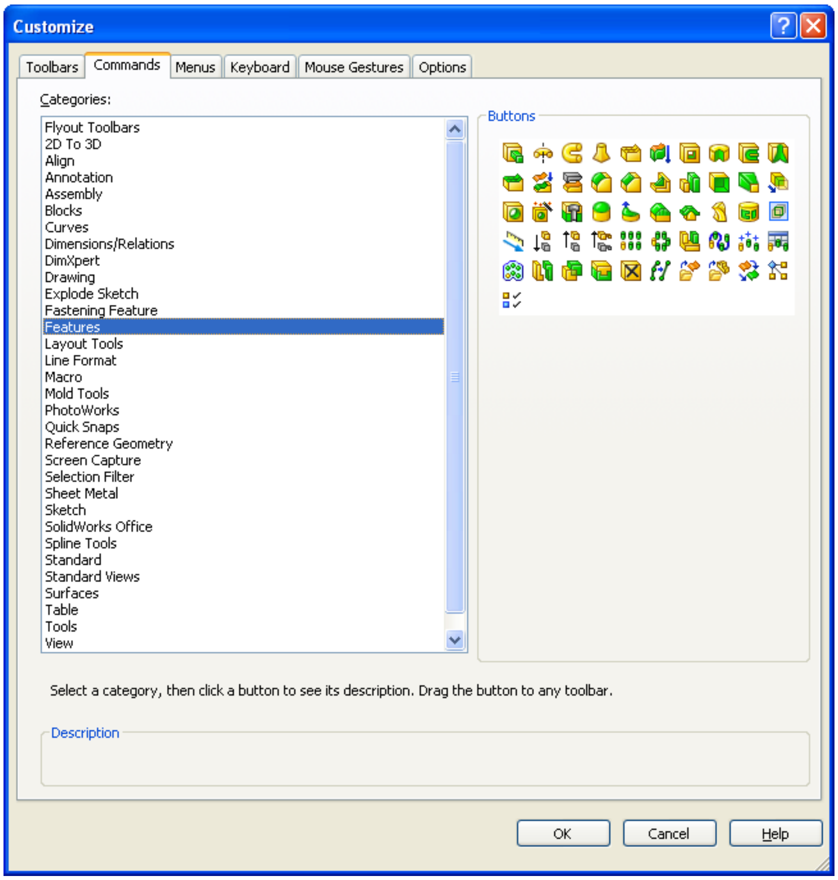
2 Click the toolbar name to turn its display on or off. If it is not already on, click **View** to turn the **View** toolbar on.

3 Turn several toolbars on and off to see the commands.

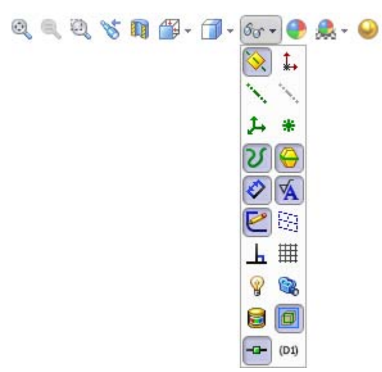
Adding Commands to Toolbars

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.



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  to access the other commands.

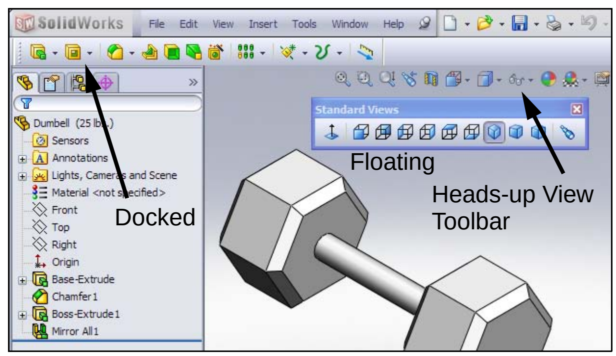
Command Manager

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.

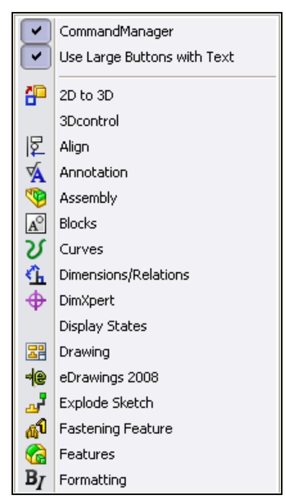
Arranging Toolbars

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.

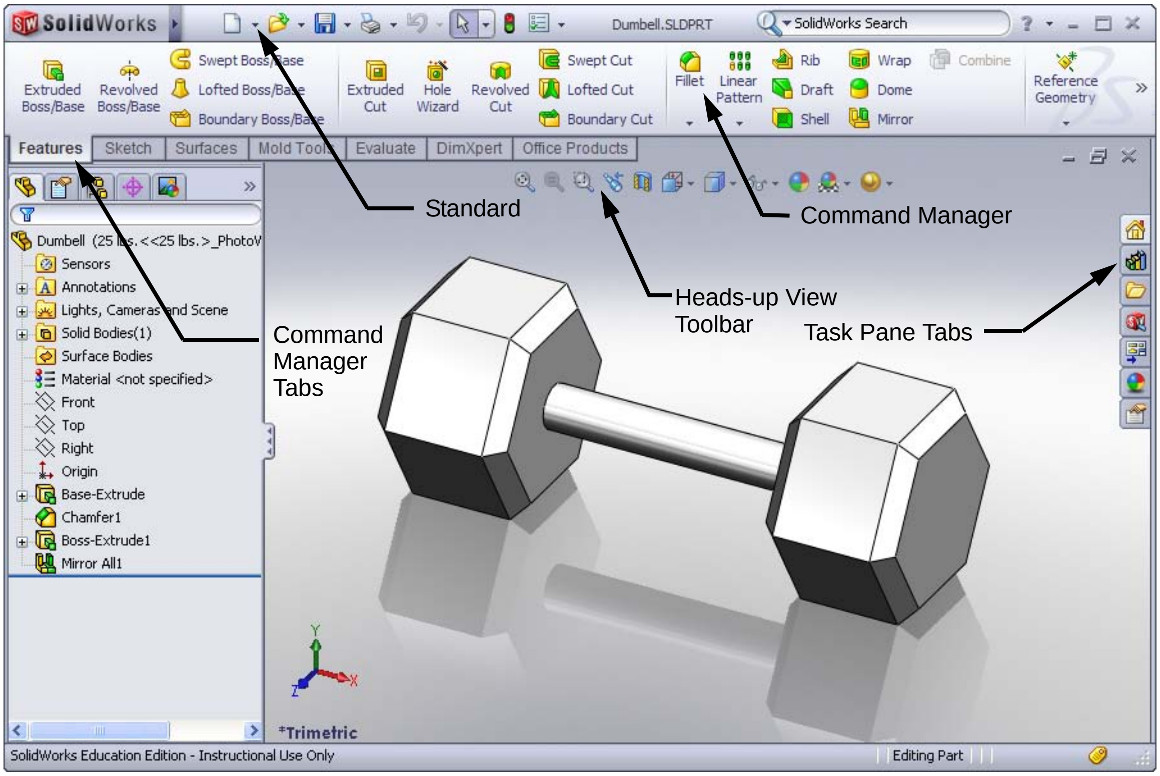


1 Click **View, Toolbars**.

2 Select the following toolbars:
   - **Command Manager**
   - **View (Heads-Up)**
   - **Task Pane**

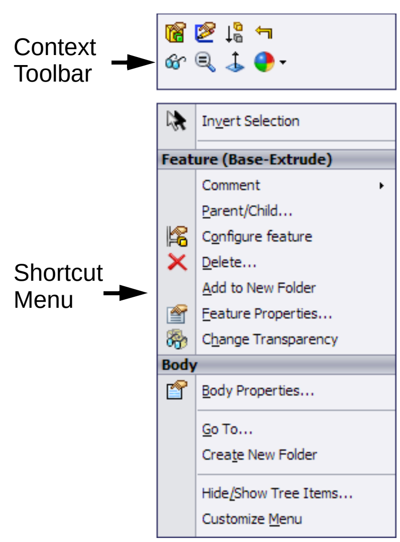
3 Clear the **MotionManager**.

4 The SolidWorks window should look like the image below.



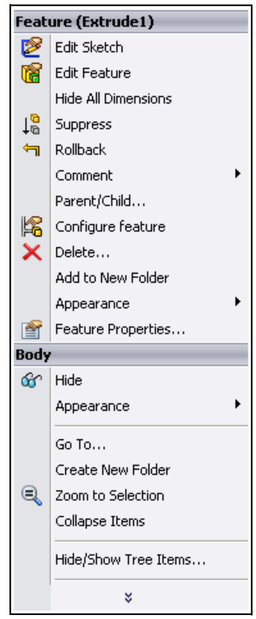
Shortcut Menus

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.



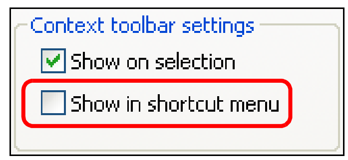
You can access the "more commands menu" by selecting the double-down arrows in the menu. When you select the double-down arrows, the shortcut menu expands to offer more menu items.

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.



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  or **Help, SolidWorks Help Topics** in the menu bar.

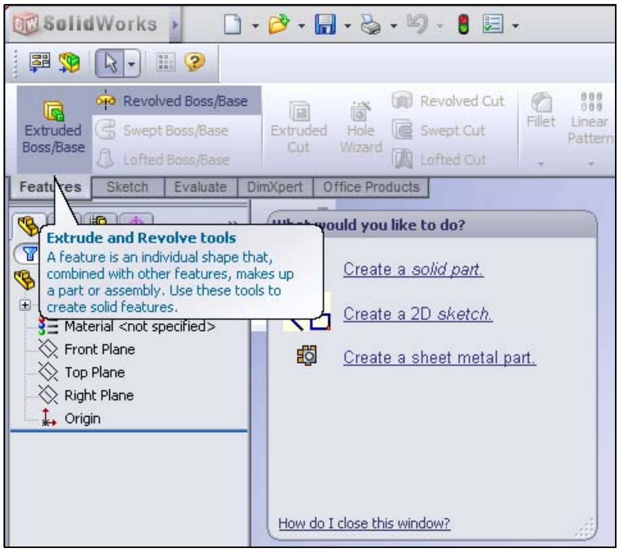
The online help appears.

2 Click  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 bring up the Search window?

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

Name: ________________________________Class: _________ Date:_______________

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

1 Shortcuts for collections of frequently used commands: _________________________

2 Command to create a copy of a file with a new name: ___________________________

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

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

5 Character that you can use to perform wild card searches: _______________________

6 Area of the screen that displays the work of a program: ________________________

7 Icon that you can double-click to start a program: ______________________________

8 Action that quickly displays menus of frequently used or detailed commands: _______
____

9 Command that updates your file with changes that you have made to it: ____________
____

10 Action that quickly opens a part or program: __________________________________

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

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

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

Lesson 1 Quiz

Name: ________________________________ Class: _________ Date: _______________

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

1 How do you start the SolidWorks application program?

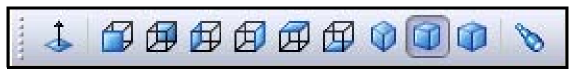
____

____

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

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

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



____

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

____

6 Which command would you use to preserve changes that 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.

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

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

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.

2

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:

- *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**, **Search** 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 PhotoWorks 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.

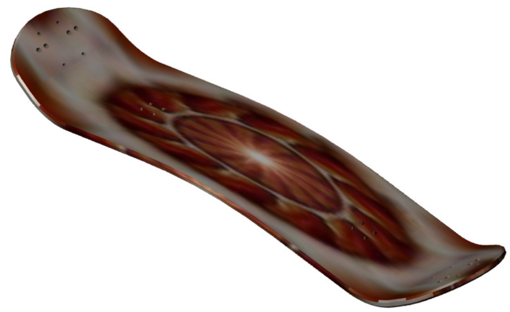
The Bindings

There will be two bindings, one right-footed 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.

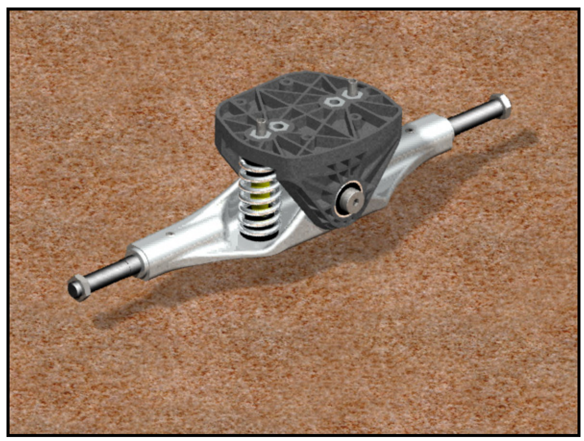


The Truck and Axle

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 (sub-assemblies) 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

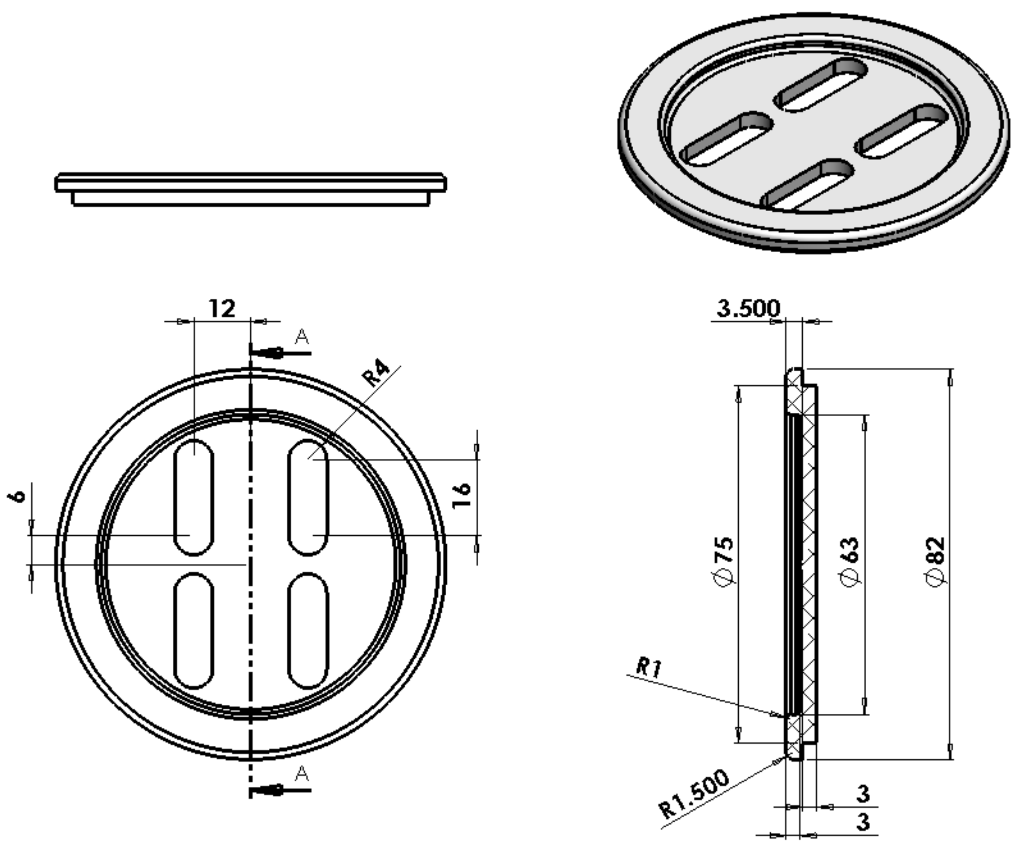
The first part created will be the Binding Anchor shown at right. We will use SolidWorks to create this part.

Design Intent

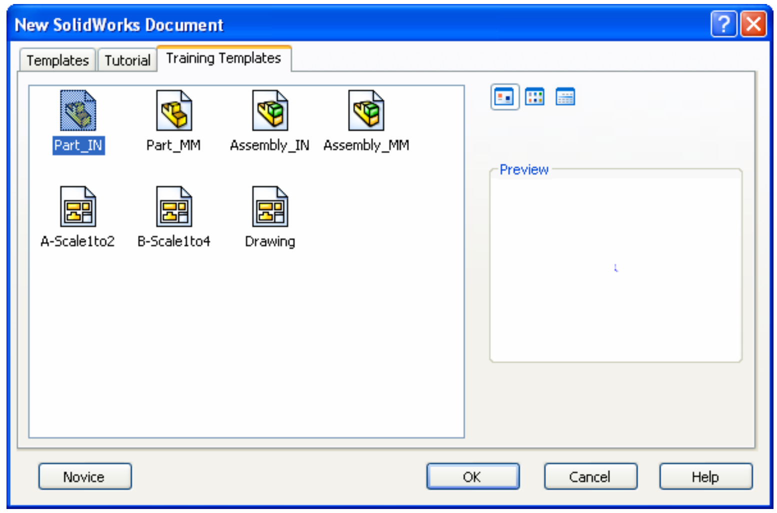
Before starting on the actual steps to create the Binding Anchor, we need to determine the design intent. This is a list of requirements the finished part needs to meet. The design intent will tell us what the finished part must be able to do.

- 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.

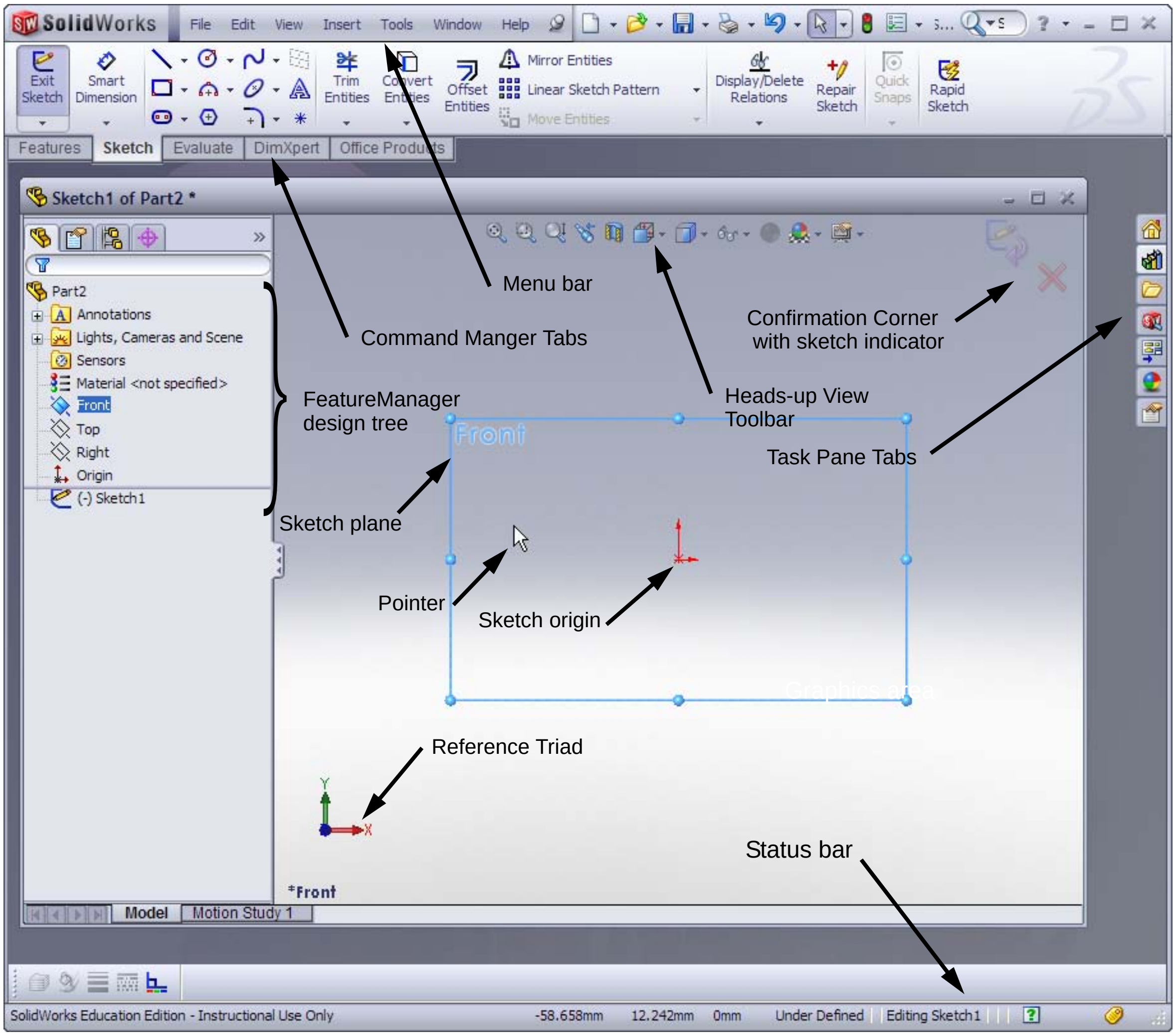
2. Click the **Training Templates** tab.
3. Select the **Part_MM** icon.
4. Click **OK**.

   A new part document window appears.

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** on the **Heads-up View** toolbar and select the scene you want to use.

View Orientation

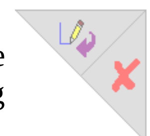
When we open a sketch for the first feature, SolidWorks will automatically change the view orientation to be normal to the sketch plane. This makes it easier to see the sketch. It is like looking straight down on a piece of paper.

Confirmation Corner

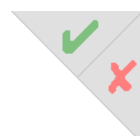
When many SolidWorks commands are active, a symbol or a set of symbols appears in the upper right corner of the graphics area. This area is called the **Confirmation Corner**.

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.

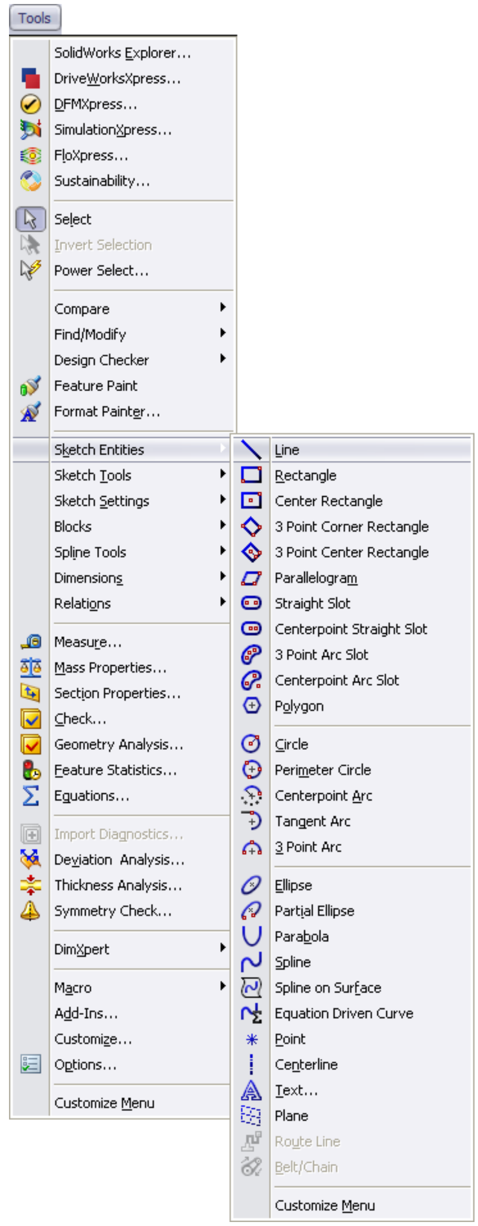


Sketch Entities

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.

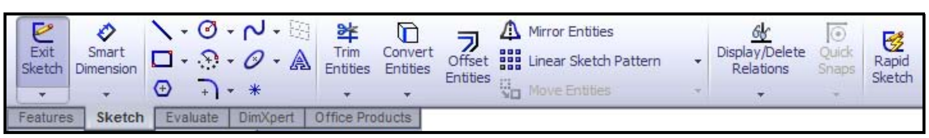


Sketch Toolbar

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

Command Manager

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



Task 2 — Create the first sketch

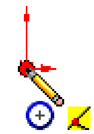
The first feature will be a short cylinder, 75mm in diameter and 3.5mm thick.

The Circle

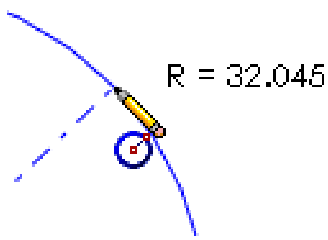
The circle tool  creates 2D circles. Using the mouse, press the left mouse button at the location for the center of the circle, then (holding down the left mouse button) drag until the circle is approximately the correct size. Release the left mouse button.

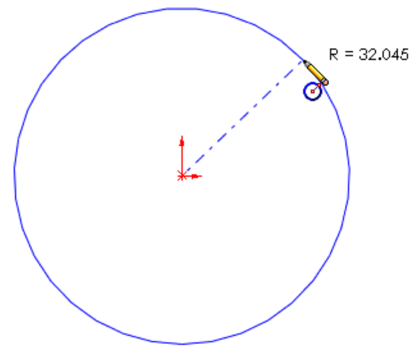
1 Click the **Circle** tool  from the Sketch toolbar. The curser will show that the circle tool is active by displaying a circle under the drawing tool .

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.



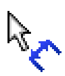
3 Press the left mouse button and drag the circle until it is just about 32mm. The cursor feedback will show the radius of the circle.



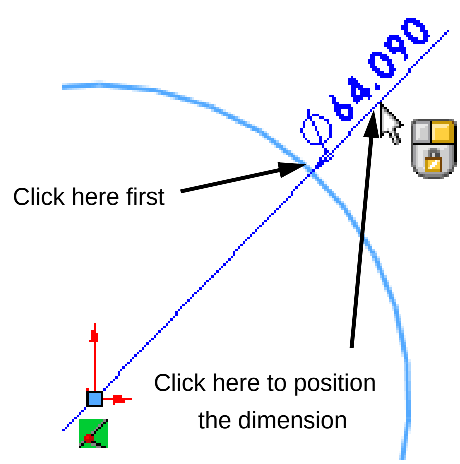


Dimension the sketch

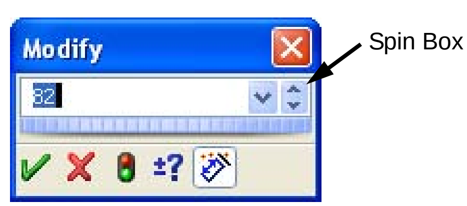
To make the circle the correct size, we will add a diameter dimension to the sketch.

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

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

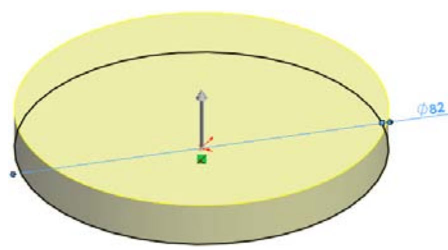
Red - Over defined

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.



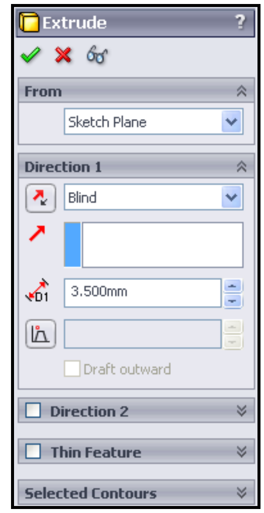
1 Select the Features tab on the Command Manager.

Click **Extrude Boss/Base** 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.



3 In the PropertyManager, change the settings as shown.

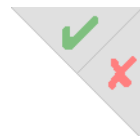
- End Condition = **Blind**
-  (Depth) = **3.5mm**

4 Create the extrusion. Click **OK**. . The extrusion now becomes a solid and a new feature, `Extrude1` is displayed in the FeatureManager design tree.

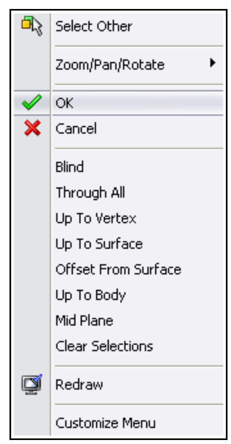
TIP:

The **OK** button  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.



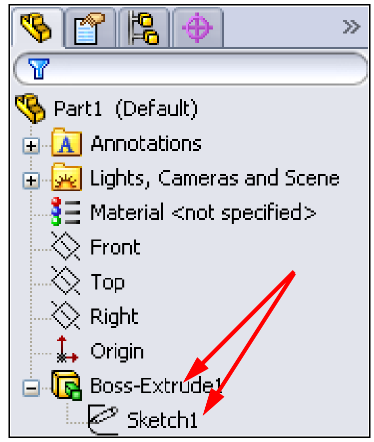
Blind Extrusions

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

FeatureManager design tree

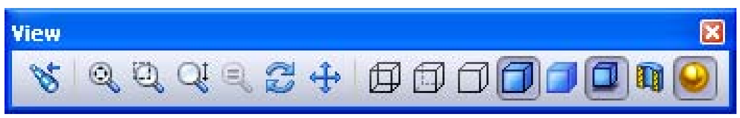
This extrusion is the first feature of our part. The FeatureManager design tree shows this feature by type and with a default name Boss-`Extrude1`.

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

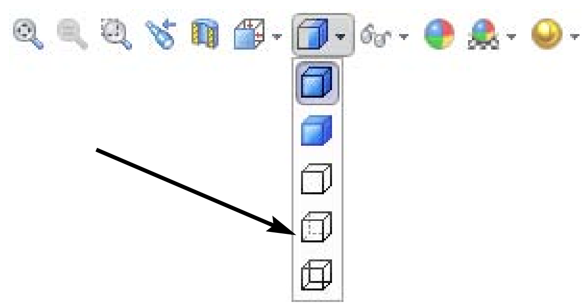


View Display

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.

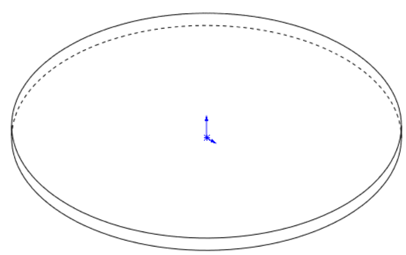


The Head-up View toolbar also provides the same functions. To change the view, click the **Display Style** pull-down list.



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

Hidden Lines Visible allows you to easily select hidden back edges of the part.

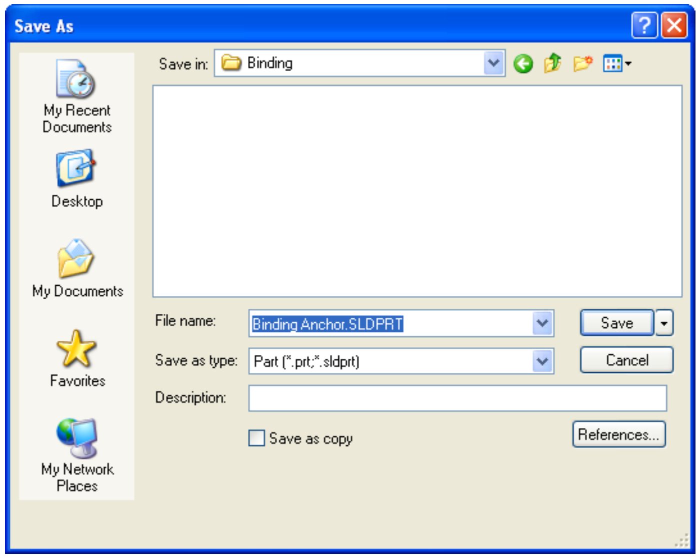


Save the Part

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.

7 Save the file to the folder `Binding` found under `SolidWorks Curriculum and Courseware_2010\Mountainboard Design Project\Mountainboard`.

8 Click **Save**.

The sldprt extension is automatically added to the filename.

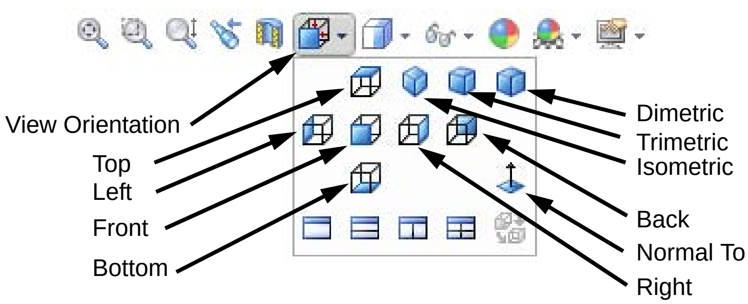
The file is saved to the current folder. You can use the Windows browse button to change to a different folder.

Note: All the files we create of the Mountainboard project should be saved in the appropriate folder under the folder …`\SolidWorks Curriculum and Courseware_2010\Mountainboard Design Project`.

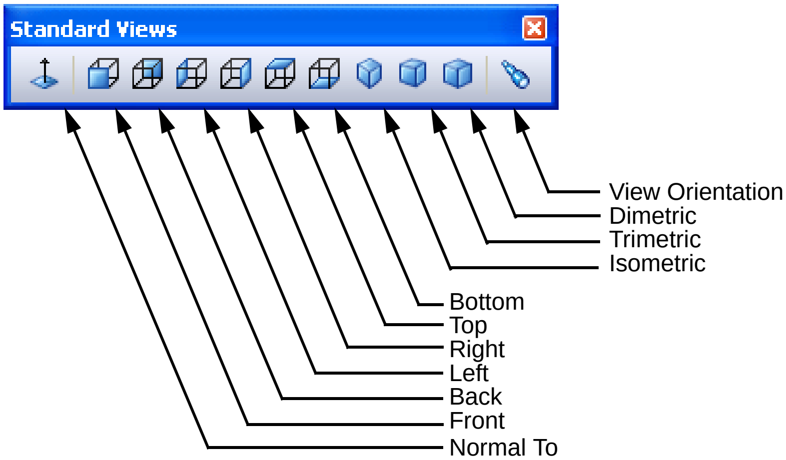
Changing views

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

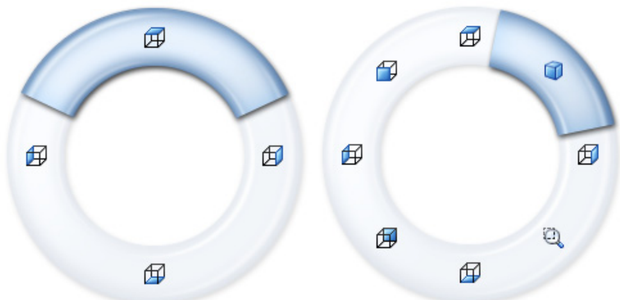


Standard Views Toolbar



Mouse Gestures

Mouse Gestures can also be used to 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.

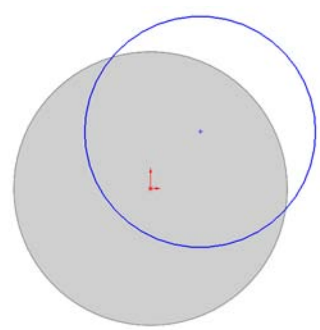
1 Change the display mode back to shaded. Click **Shaded With Edges** on the **View** toolbar.

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** 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.



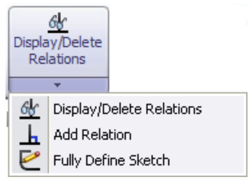
Task 5 — Adding sketch relationships

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.

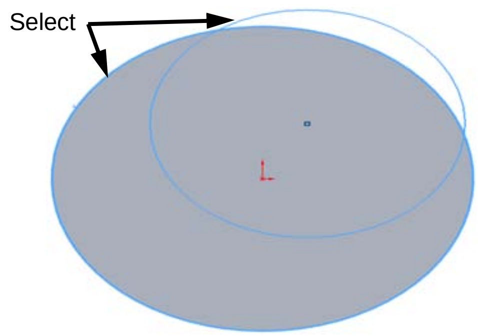
When adding relationships, only those relationships that are appropriate for the sketch entities selected will be shown in the PropertyManager.

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

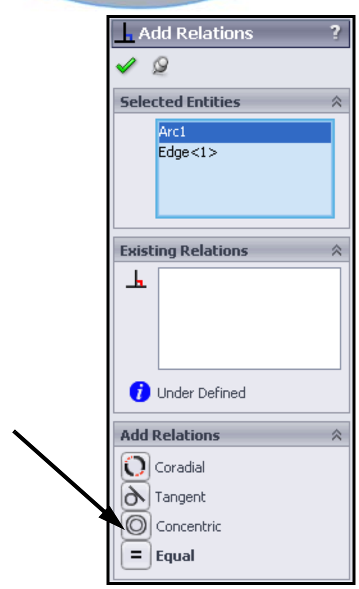
Note: When using the Command Manager, click **Display/Delete Relations**, then select **Add Relations** from the dropdown list.

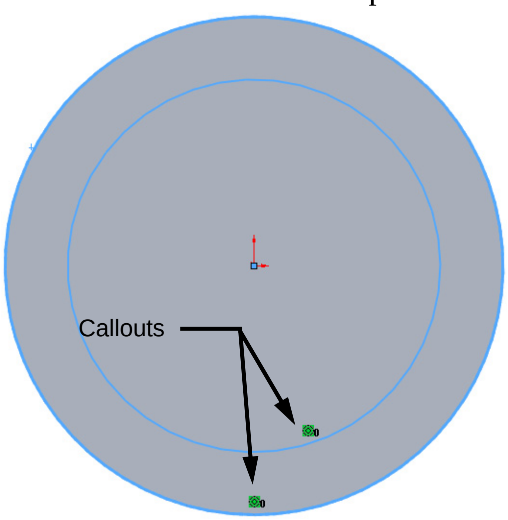


2 Select the circle and the edge of the cylinder.



3 Click **Concentric** in the **Add Relations** box. Then click **OK**. The circle will move to a position where it is centered on the cylinder. Callouts will show on the sketch to show the relationships.





Callouts

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

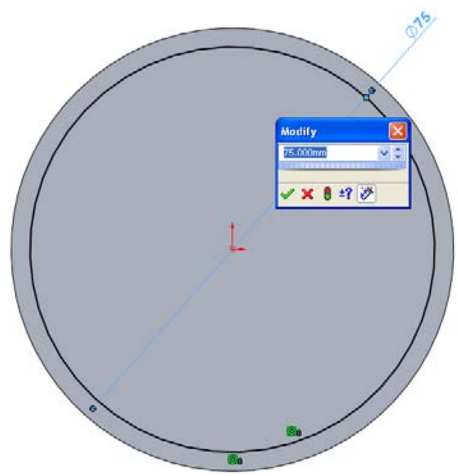
Task 6 — Dimension the circle

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.

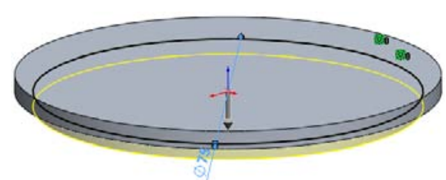
Task 7 — Extrude the second feature

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 on the **Views Orientation** toolbar.

2 Select the top face of the model, and click **Sketch** to start a new sketch.

3 Click **Circle** on the Sketch toolbar.

4 Draw a circle from the center of the top face. Make its radius about **30mm**.

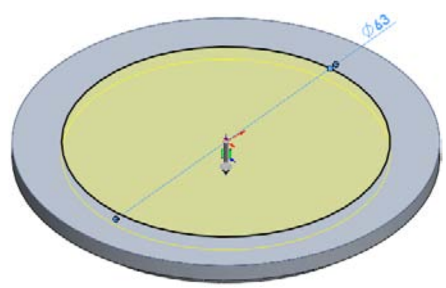
5 Dimension the circle to be **63mm** in diameter.

6 Reorient the model to the Isometric view.

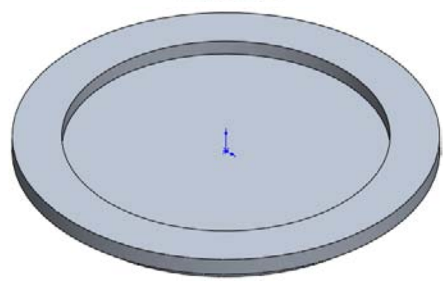
7 Click **Cut Extrude** to use the circle to cut away some material.

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.

- 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).

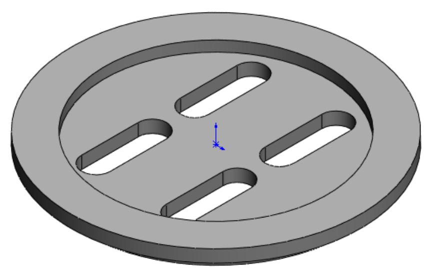
The `Binding Anchor` will be made from 2014 Aluminum. Using MatWeb, the density for 2014 Aluminum is 2.8 g/cc. There are 1000 cubic mm in 1 cc, so the density would be:

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

Calculate the weight

Weight = Volume x Density = 22,385.42 mm^3 x .0028 g/mm^3 = **62.68 grams (2.21 oz)**.

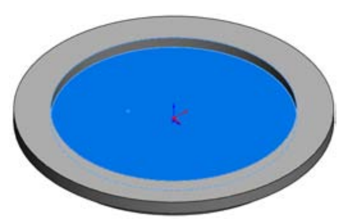
Task 9 — Create the screw slots



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 to indicate that a **Vertical** relationship will be added. If the line is horizontal, the cursor will show an 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.

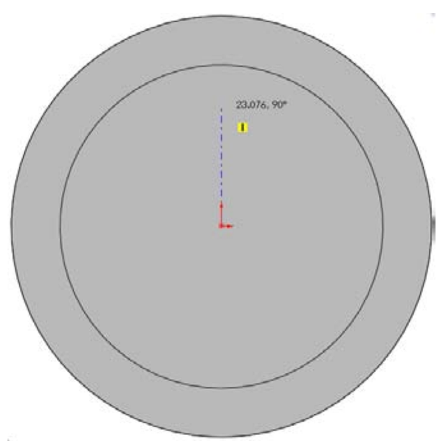
While you are sketching, the callouts will be yellow , indicating which relationship will be added when you release the mouse button. The green callouts show relationships that have been added .

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.

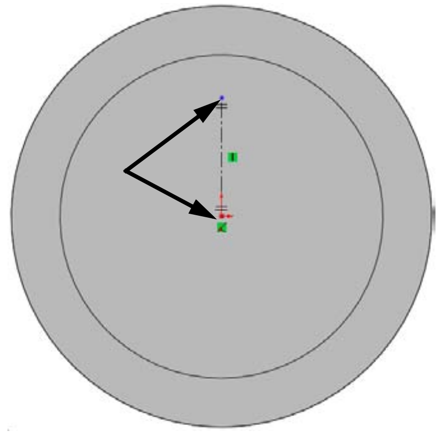
Create a centerline

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 , indicating a **Vertical** relationship will be added.

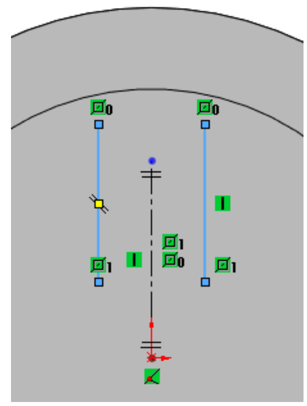


6 Click **Dynamic Mirror Entities** on the **Sketch** toolbar. A pair of parallel marks will appear at each end of the centerline to show that we are in the mirror mode.



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 — Adds a tangent relationship to the entity it is sketched from.

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

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

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

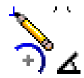
Add an Arc

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

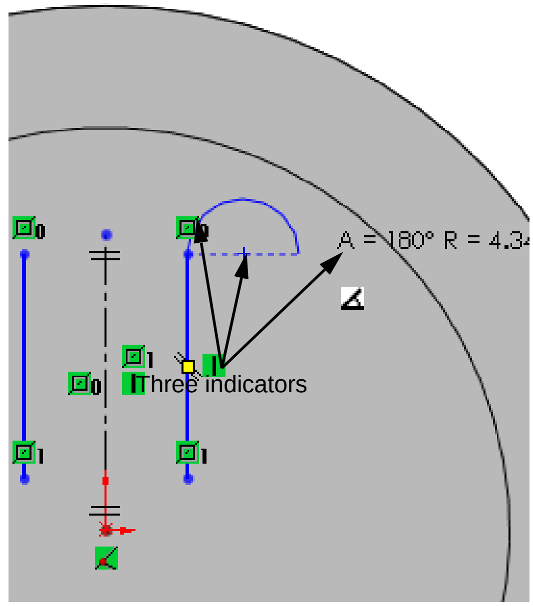
8 Click **Tangent Arc** the **Sketch** toolbar.

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.

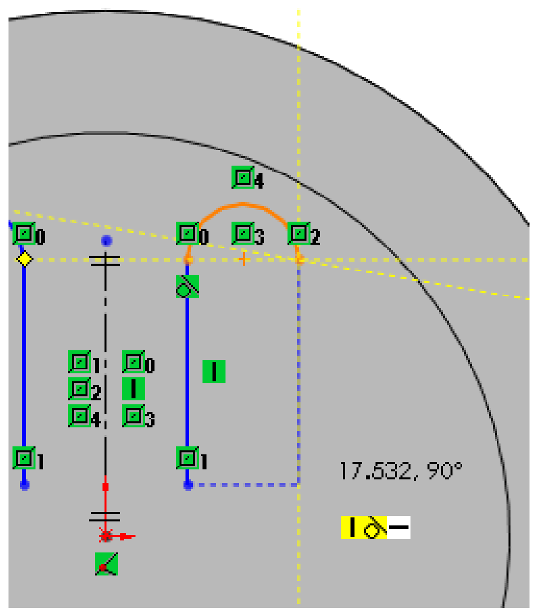
There will be three indicators that you have gone 180 degrees:

- A blue dashed line (inference line) from the center of the arc
- The angle symbol  under the drawing tool
- The arc degree feedback (A=180)

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

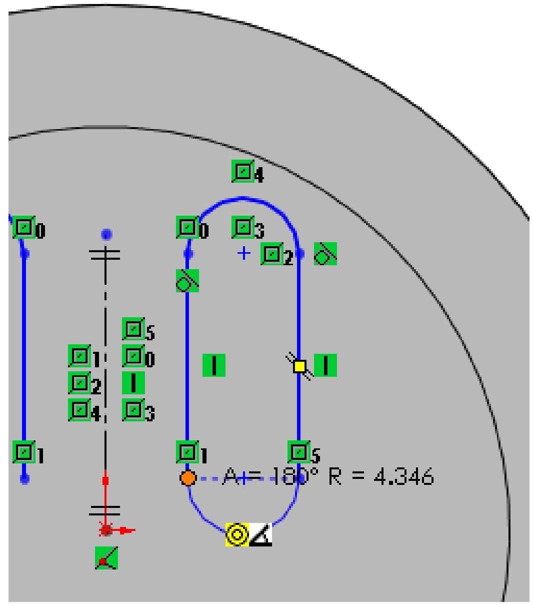


10 Draw another vertical line, from the end of the arc, vertically downward until you get an blue inference line from the bottom end of the first line.

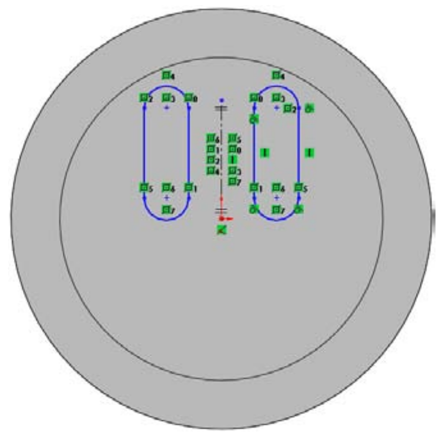


11 Finish the sketch with another **Tangent Arc**.

12 Turn off the sketch **Dynamic Mirror Entities** tool by clicking 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 and symmetric to the other arcs and centerlines. 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.

12 Click **View, Sketch Relations**.

13 Turn off the callouts by clicking **View, Sketch Relations**.

Mirror after sketching

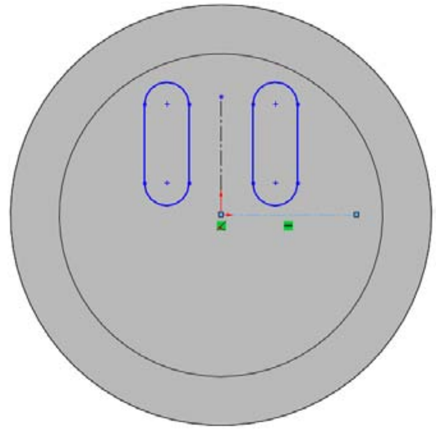
Mirrored entities can also be created after creating sketch entities. To mirror after sketching, select the centerline about which you want to mirror and all the entities you want to mirror.

To mirror after sketching, click **Mirror Entities** .

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.



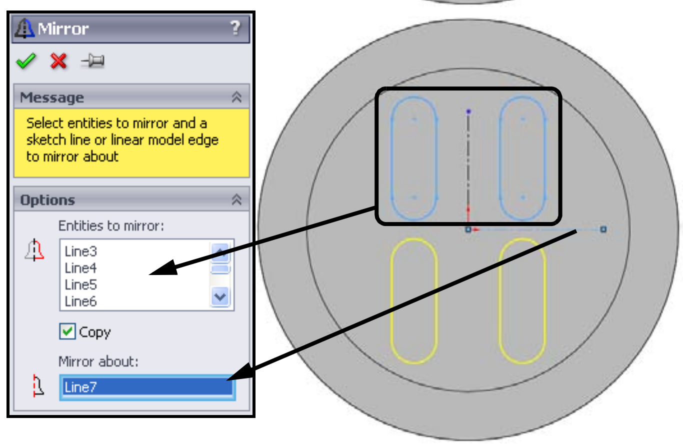
2 Click **Mirror Entities**  on the Sketch toolbar.

3 Turn off the **Centerline** tool  by clicking on the tool again.

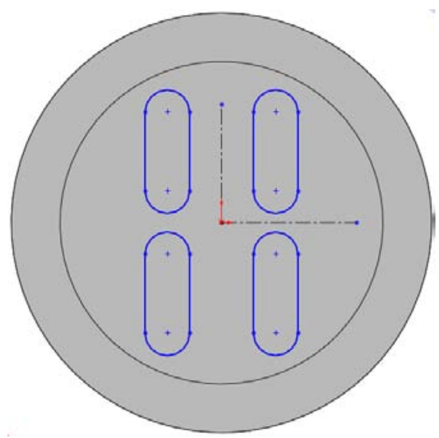
4 Select the four lines and four arcs as the **Entities to mirror**. Select the horizontal centerline for the entity to **Mirror about**.

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

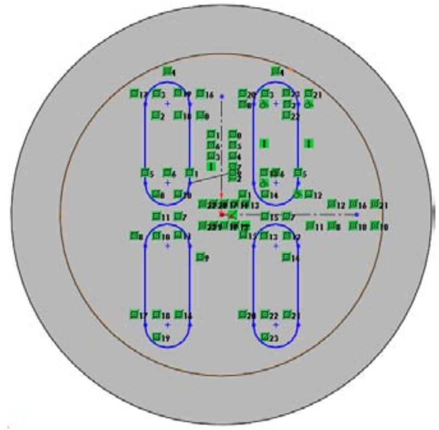
Click .



5 We now have a sketch that will cut the four slots.



View Relationships

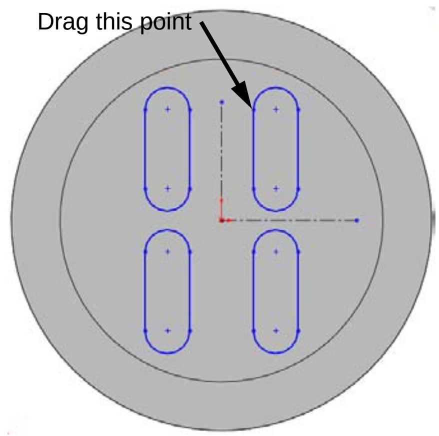


6 Click **View, Sketch Relations**.

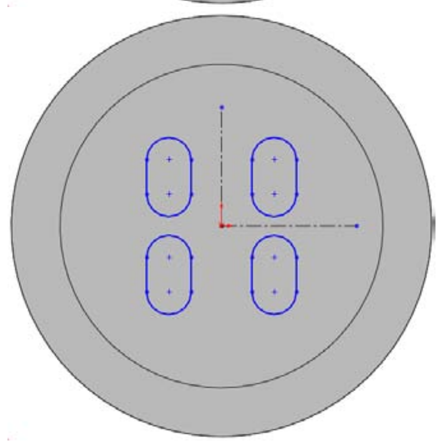
The callouts show that the all the arcs are tangent to the two lines they connect to and symmetric to the other arcs and centerlines . The numbers next to each symmetric relationship show the pairs of symmetric elements.

7 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.



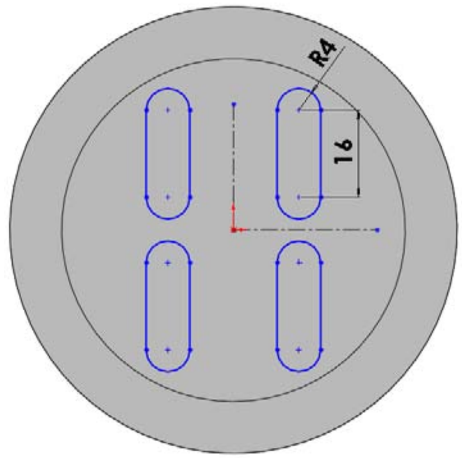
8 Drag the point shown. All four slots should change shape together.

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.

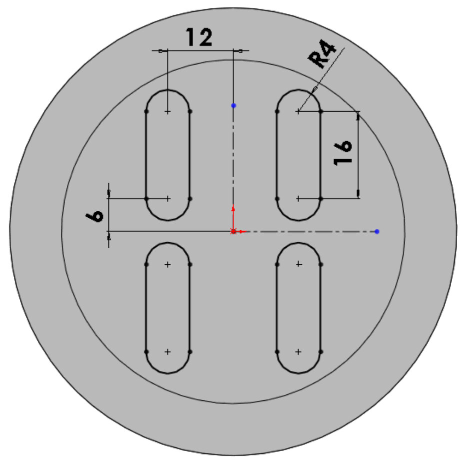
   To add the **16mm** dimension, select the two arcs, not the vertical line.



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.

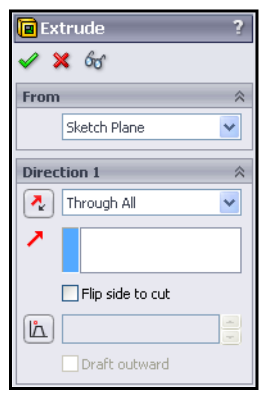
3. Fully defined. The sketch geometry should now be all black, showing that the sketch is fully defined.



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.

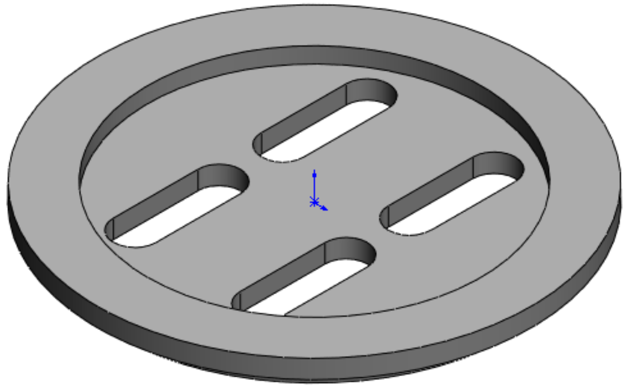
1. Click **Insert, Cut, Extrude** from the menu.
2. Click on the **View** toolbar to change the view to Isometric.
3. From the list in the PropertyManager select **Through All**.



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 .



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-Extrude4`, 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.

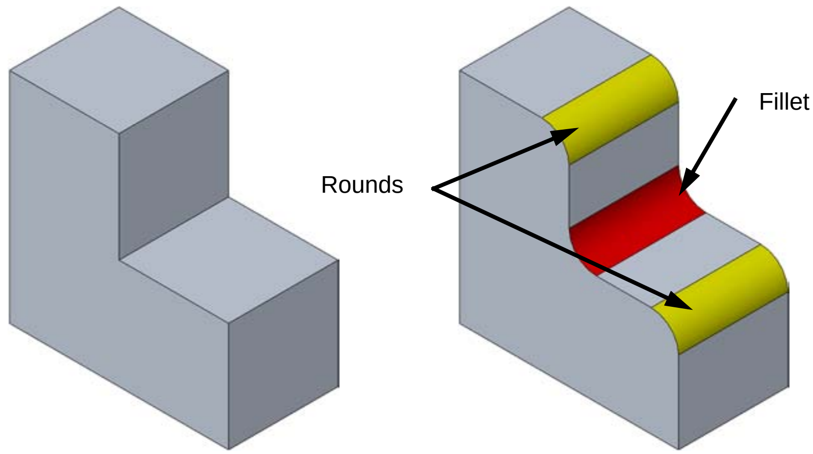
4 The feature has now been renamed to something more descriptive.

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

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.



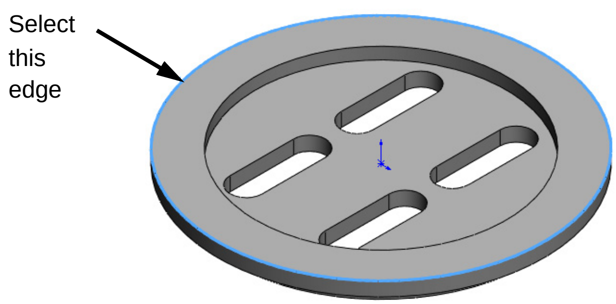
Original shape | Fillets and Rounds added

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.

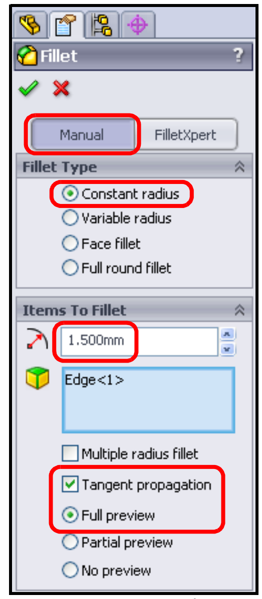
2 Select the edge shown.



3 Type **1.5mm** for the fillet radius.

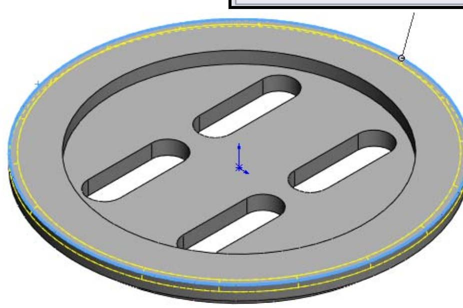
4 The following should be set by default. If not, change them as follows:

- **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 ✓.

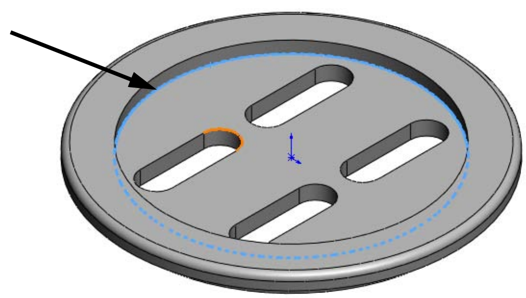


Task 15 — Add a fillet to the inside edge

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

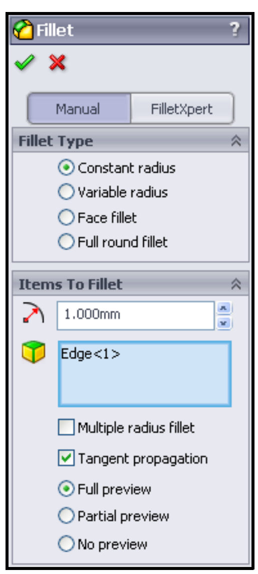
2 Select the edge shown.

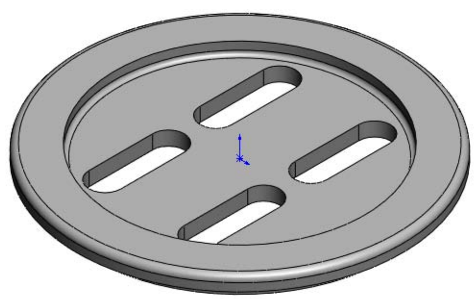
Select this edge



3 Type **1mm** for the fillet radius.

4 Click ✓. The inside edge now has a fillet.





Editing Features

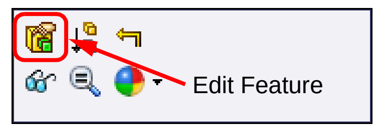
After a feature is created, it can easily be changed. For the extrudes and cuts we made, each end condition or depth could be changed to reflect changes in our design intent, or changes required by later analysis.

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**.

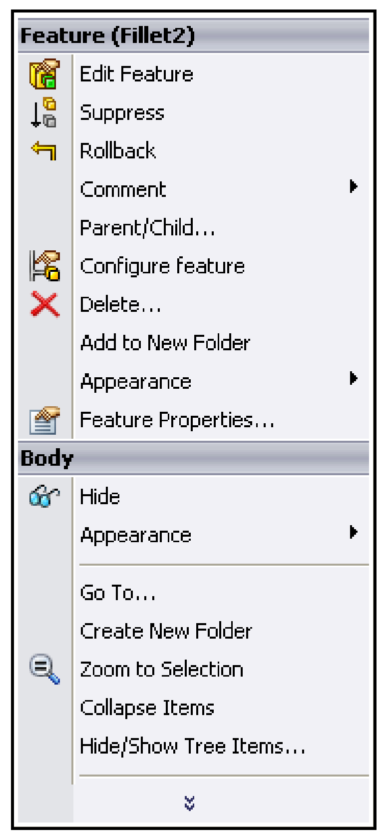
Task 16 — Edit the Fillet

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

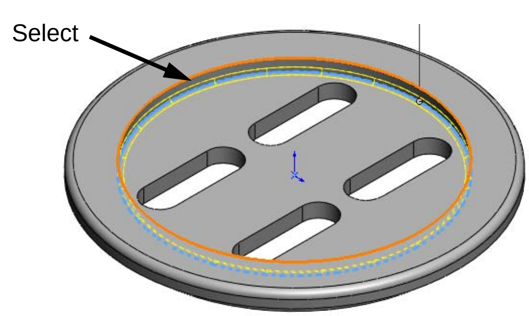


SolidWorks can be customized to show the Context toolbar with a text menu below it or just a text menu.

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



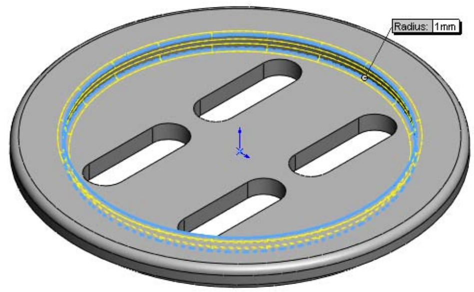
2 Select the edge shown.



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 .

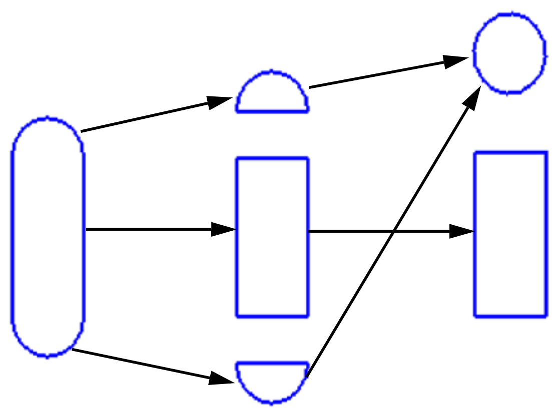
4 **Save** the part.



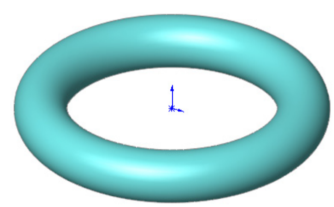
How much does it weight now?

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

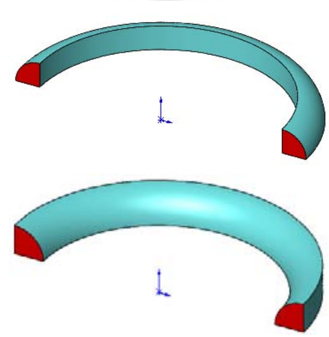
The volume removed by the slots is not too difficult as the area of each slot can be thought of as a rectangle and circle

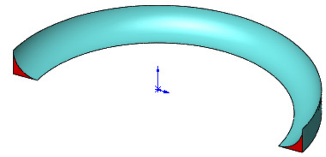


The fillets are more complicated. The two rounded corners are each part of a torus (donut).



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 ***not*** 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  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.

1 Click **Edit, Appearance, Material** from the menu.

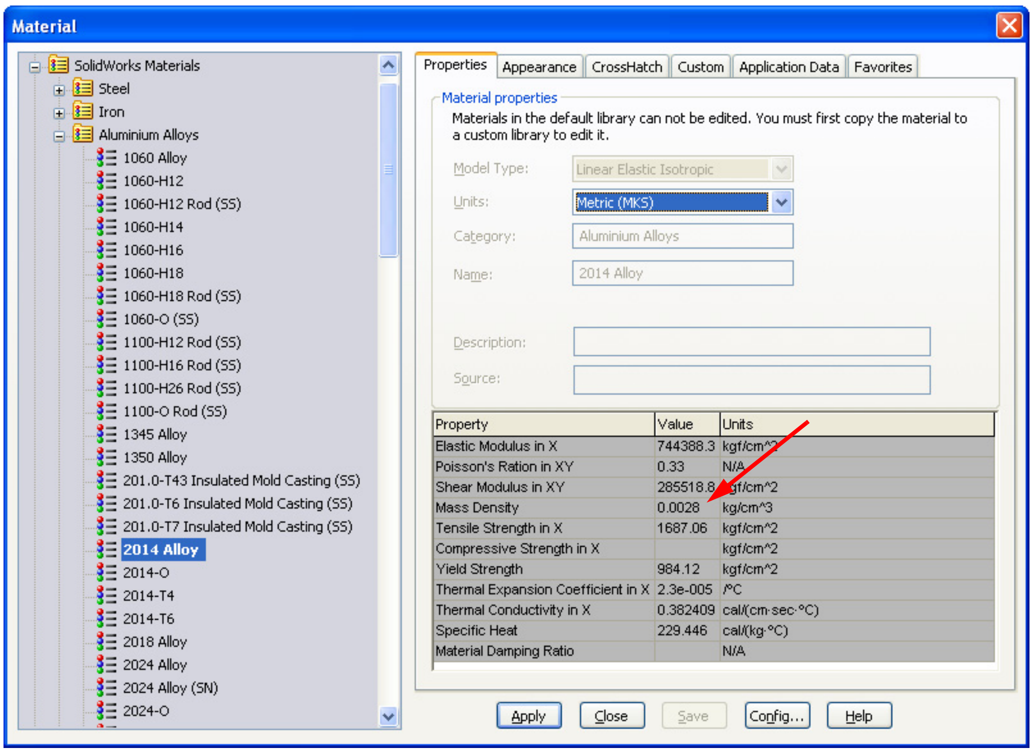
The **Materials Editor** will open.

2 Click the Plus sign next to **Aluminum Alloys** to expand the list.

3 Select **2014 Alloy**.

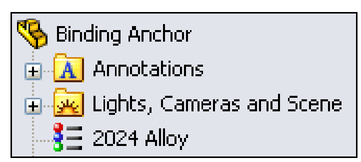
Examine the Physical Properties. Density is listed as .0028 g/mm^3. This is the same value we determined earlier.

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



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  on the Tools toolbar

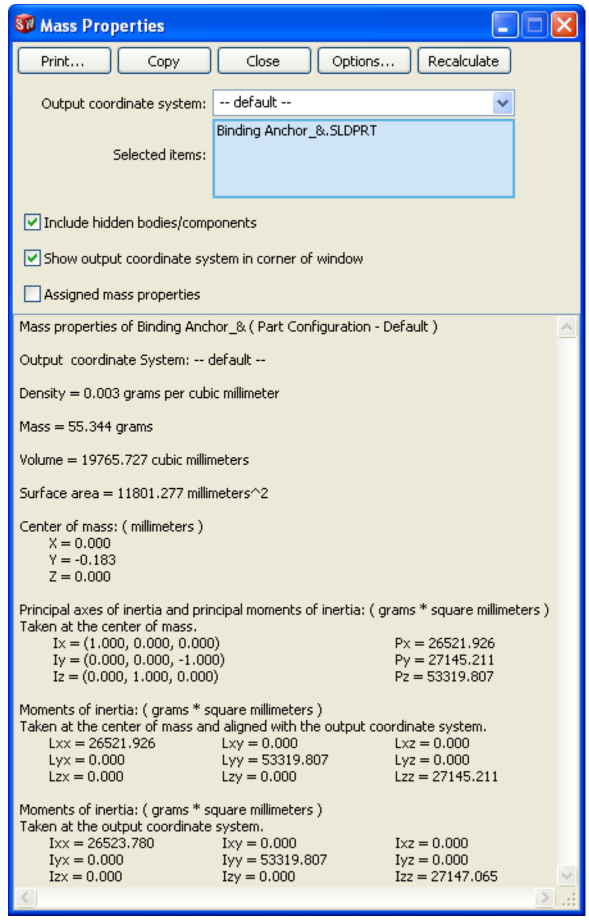
Task 18 — Determine The Weight

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

   The Mass Properties box will appear.

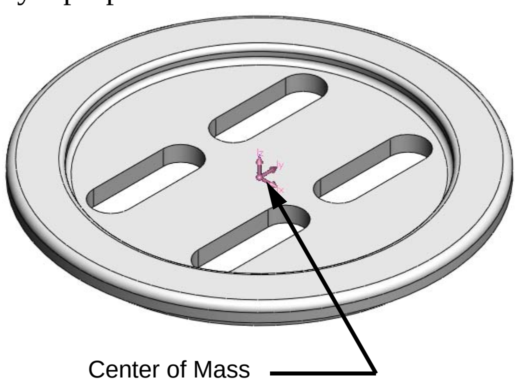
   The Volume is calculated to be **19,765.727 mm^3** and the Mass is **55.344 grams**.

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



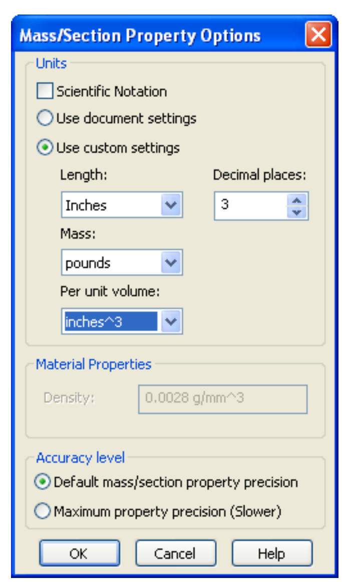
2. The Center of Mass is the balance point of the part. If we could suspend the part at this point, it would not want to tip over.

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



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**

5 Click **OK**.

```
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
```

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

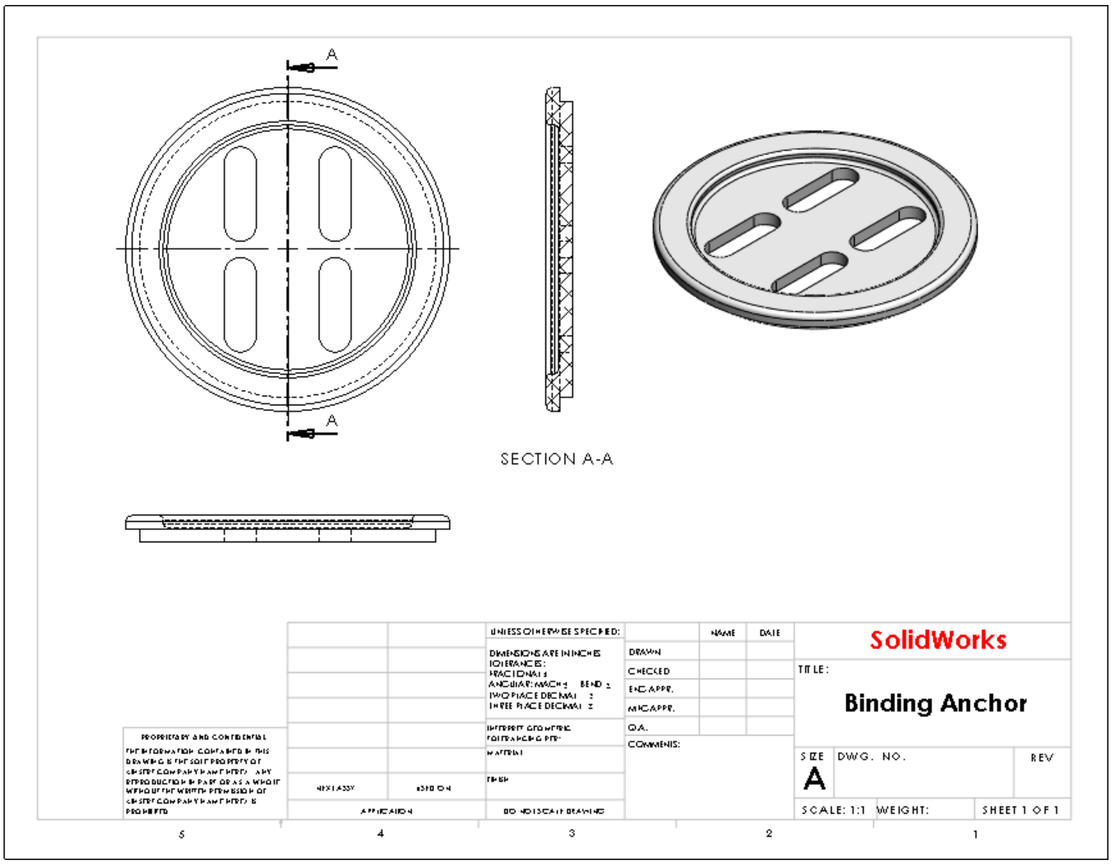
---

6 Click **Close** to close the Mass Properties.

7 **Save** the part.

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.

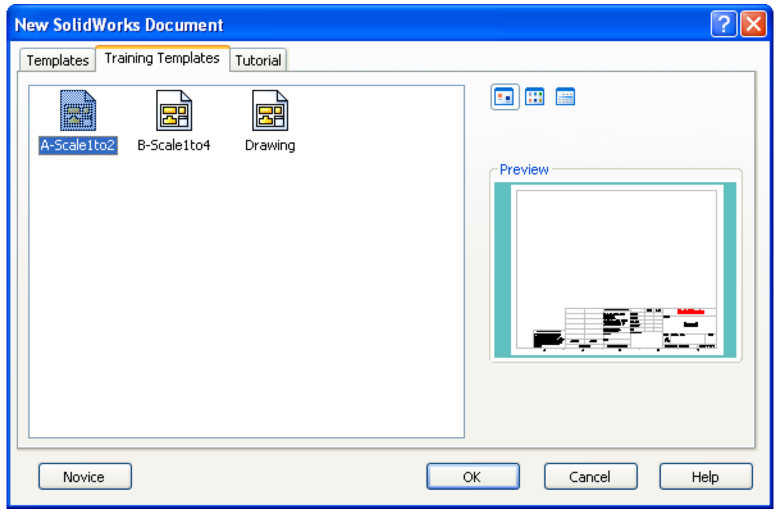
1 Create a new drawing. Click **Make Drawing from Part/Assembly**  on the **Standard** toolbar.

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

2 Click the **Training Templates** tab.

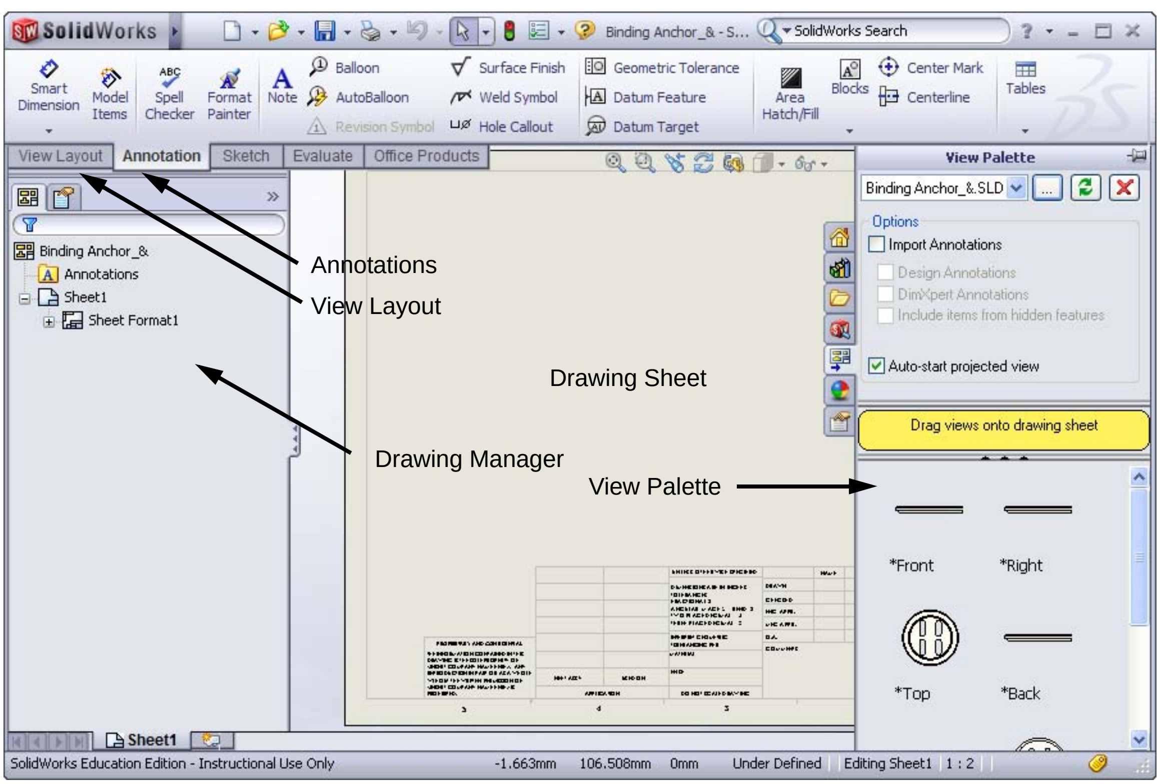
3 Select the `A-Scale1to2` icon.

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.

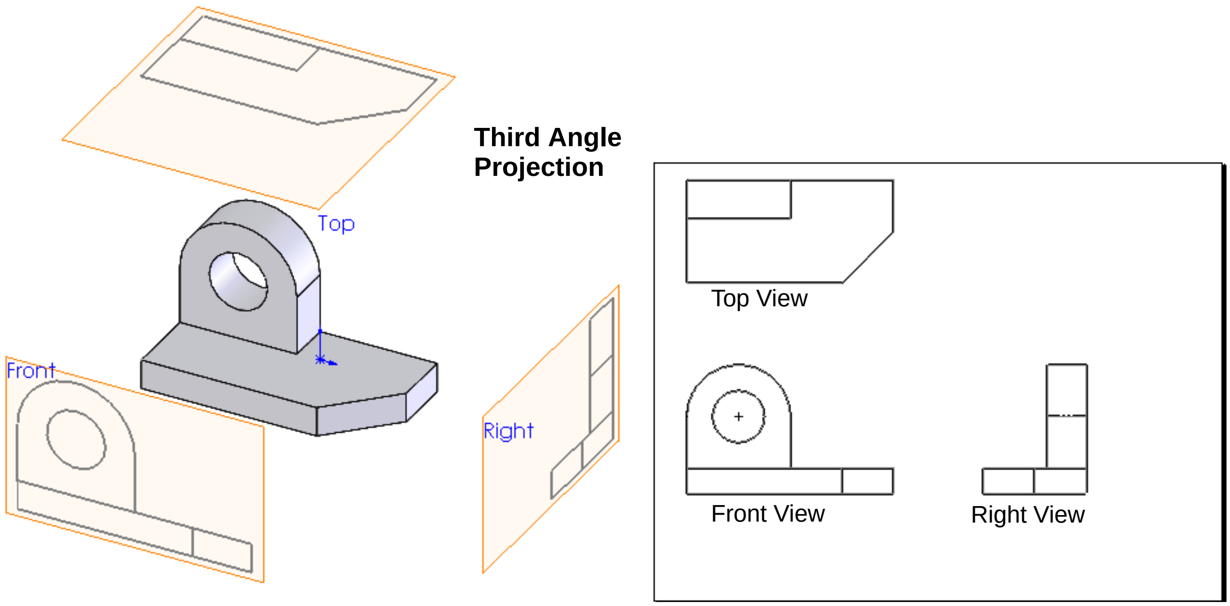


Create Three Standard Views

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

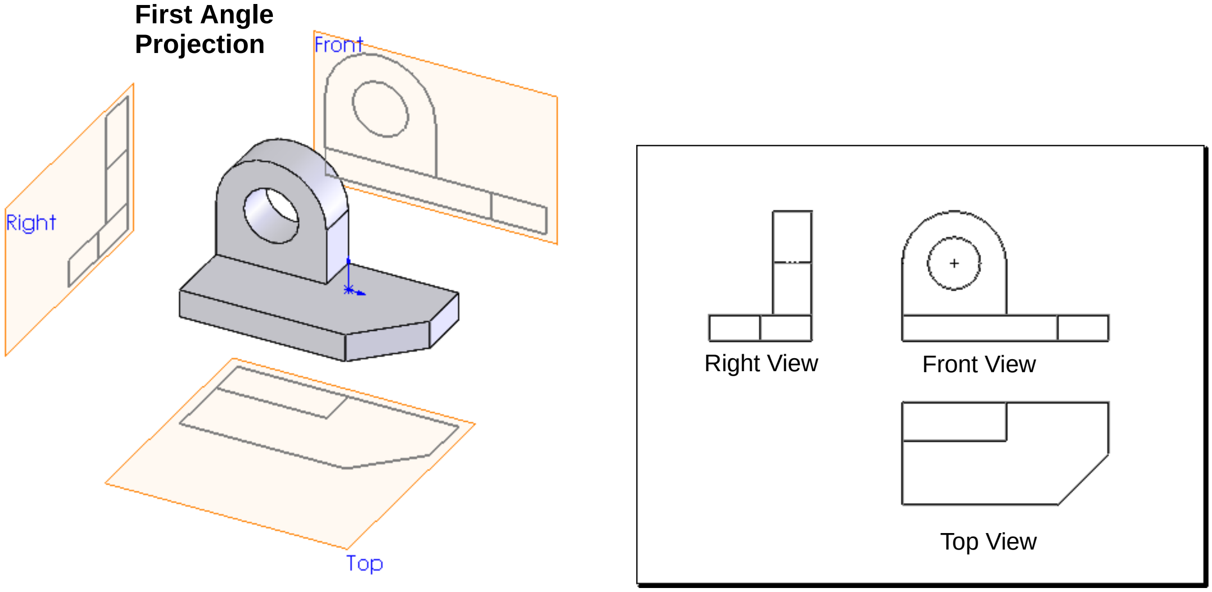
In the United States, the standard three views follow the conventions for Third Angle projections. The views are created as you would see the model as viewed from the Front, Top, or Right.

Third Angle Projection



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.

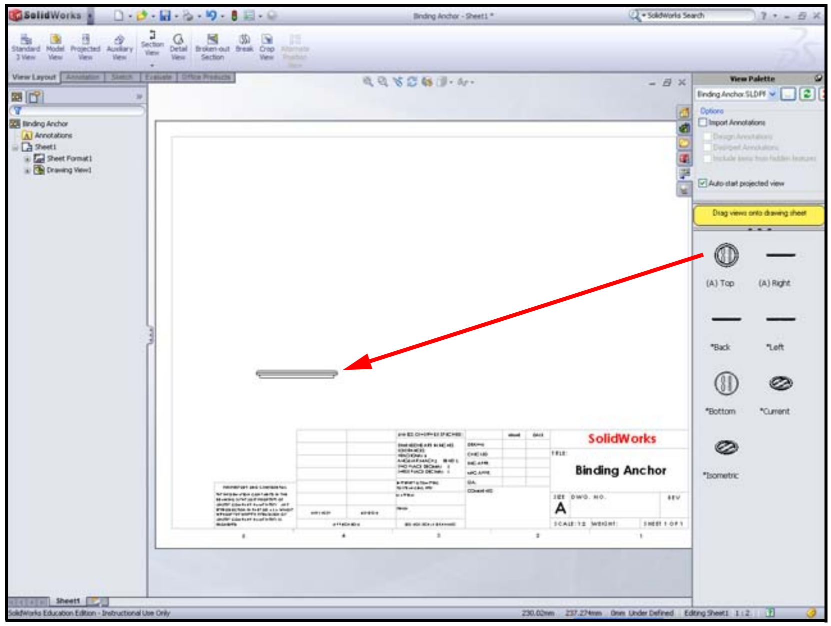
First Angle Projection



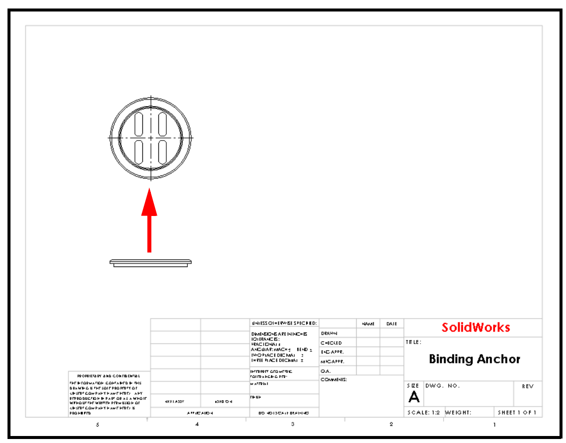
Note: In the following steps and all drawings created in this course, the paper background used in the default drawing templates has been removed. This is done to make the drawings easier to view and print.

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.

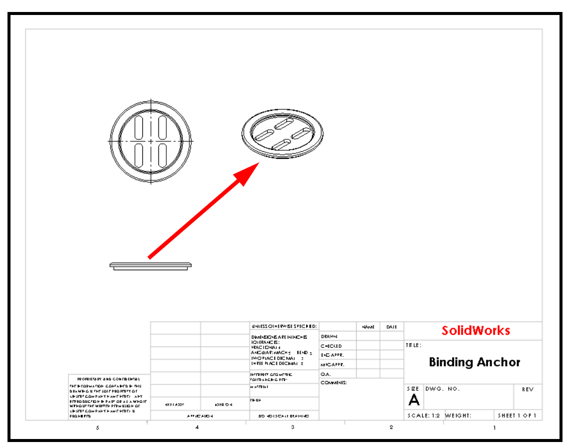


7 Move the cursor vertically from the Front view and the Top view will appear. Drop the Top view in the approximate position shown.



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**.

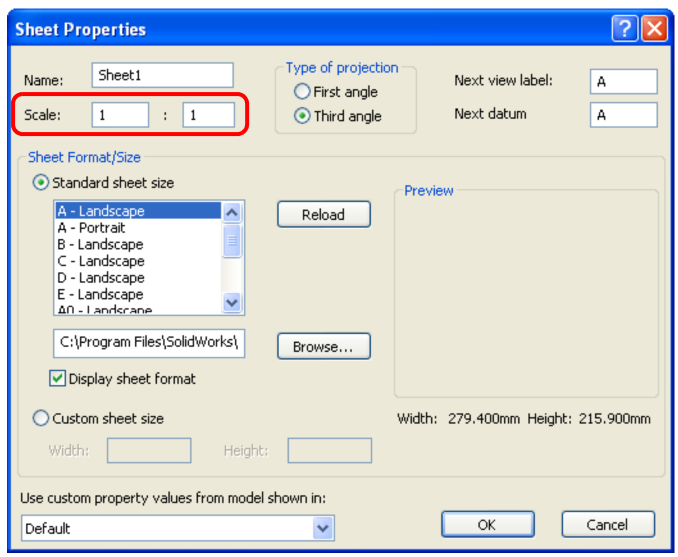
11 Select the `Front` view. In the PropertyManager set the **Display Style** to **Hidden Lines Visible**.

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

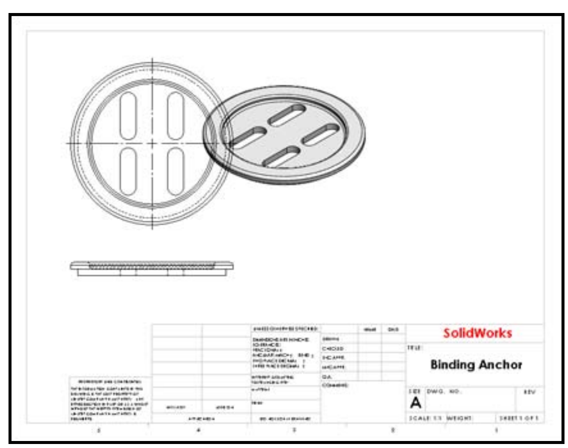


1 In the DrawingManager, right-click `Sheet1` and select **Properties** from the list.

2 Change the Scale to 1 to 1.

3 Click **OK**.

4 Examine the drawing, the view now fit the sheet better, but they are not in the correct position.



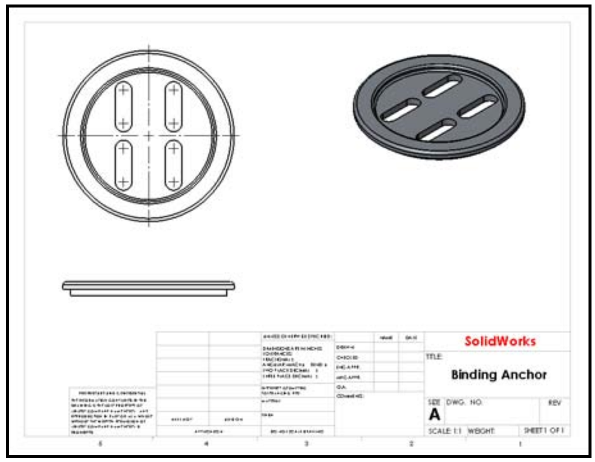
Task 3 — Adjust the views

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.

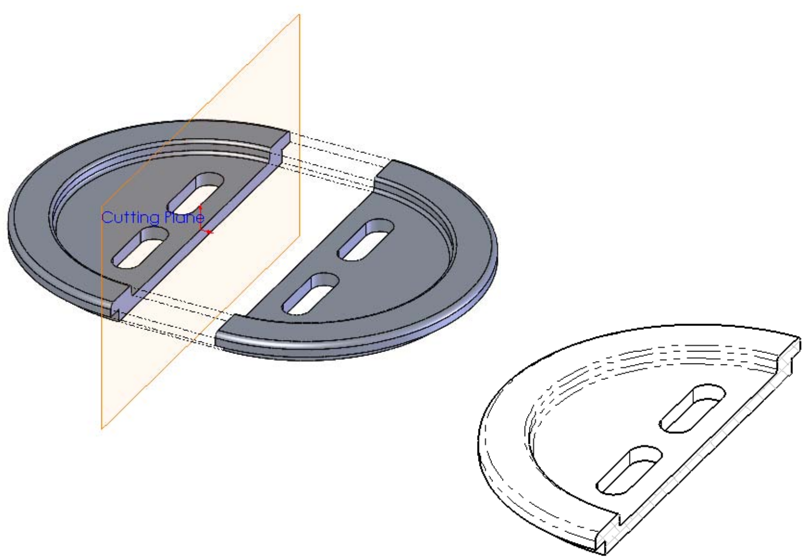
When moving views, alignment between the `Front` and `Top` views will be maintained automatically.



1. Move the `Front` view. Move the cursor over the `Front` view. When it changes to  , 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

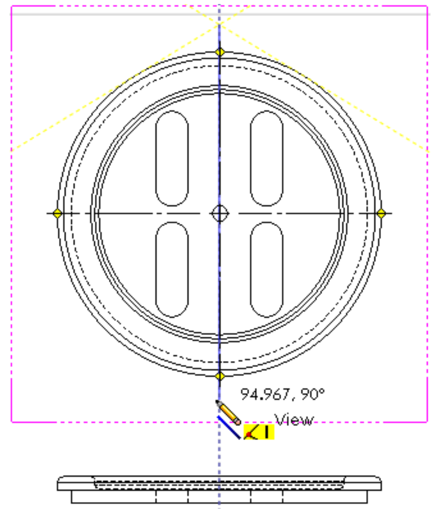
Section views are used to show the detail at some point inside the model. The model is sectioned using a cutting plane and the unused section is removed. The exposed surface is then cross-hatched with a pattern that designates the material.



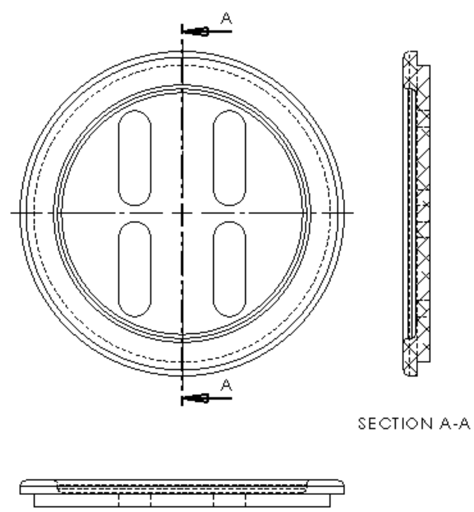
4 Create a section view from the Top view. Click **Section View**  on the Drawing toolbar or the **View Layout** tab of the Command Manager.

The **Line** tool will become active.

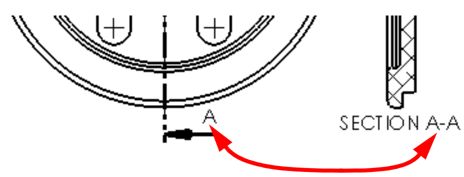
5 Draw a vertical line through the Top view at its center.



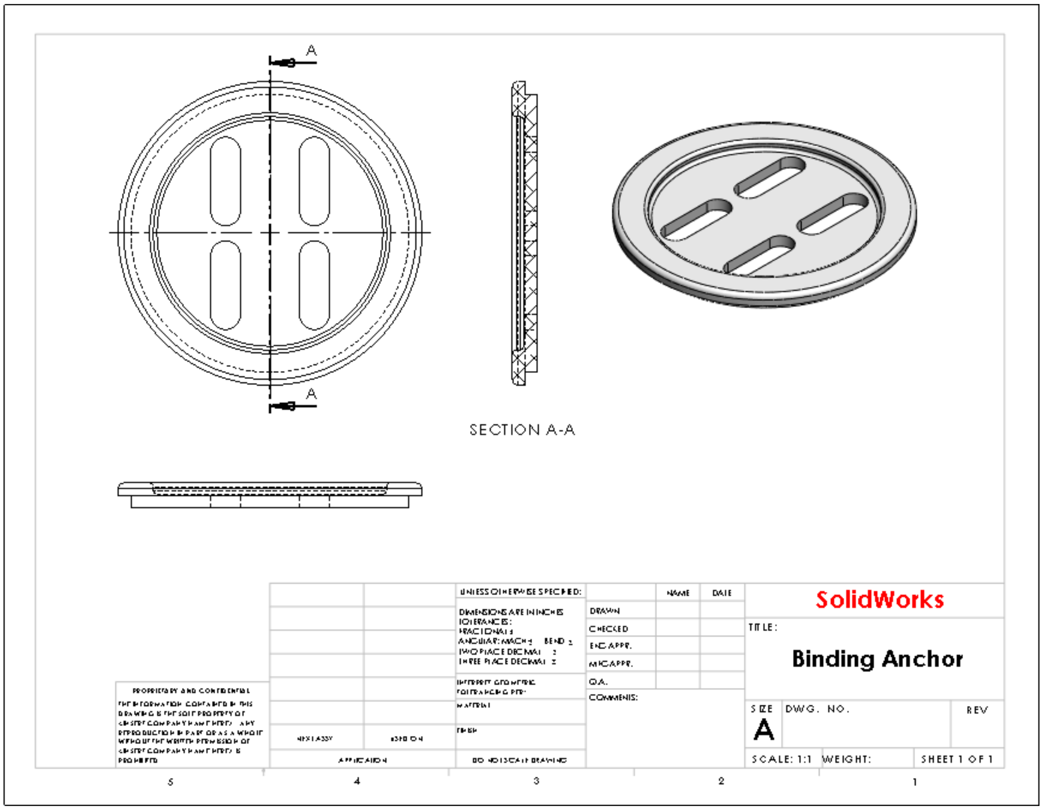
6 Move the cursor to the right of the view. The section view will move with the cursor. Click the sheet where you want to drop the section view.



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.



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



9 **Save** and **Close** the drawing.

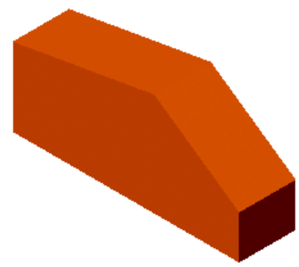
5 Minute Assessment — #2

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.

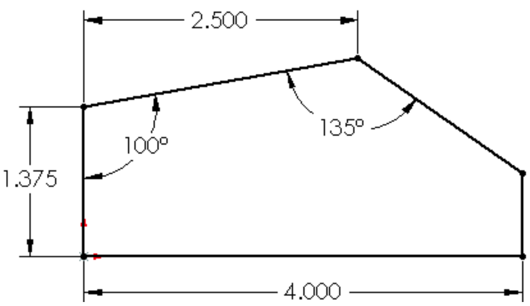
1 New part.

Open a new part using the `Part_IN` template.

2 Sketch.

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

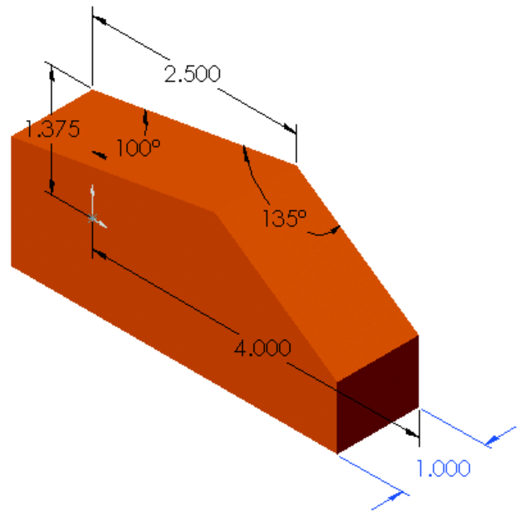
Fully define the sketch.



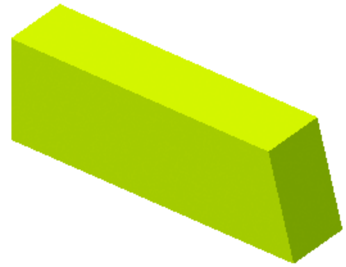
3 Extrude.

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:

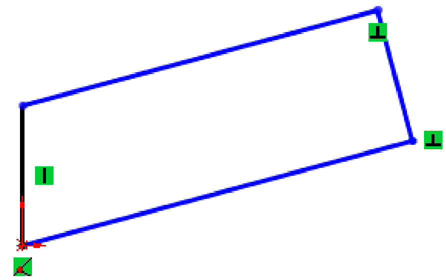
- Sketching.
- Dimensions.
- Extruding a feature.

1 New part.

Open a new part using the `Part_IN` template.

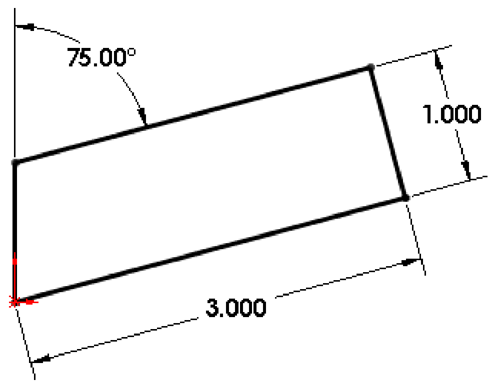
2 Automatic relations.

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



3 Dimensions.

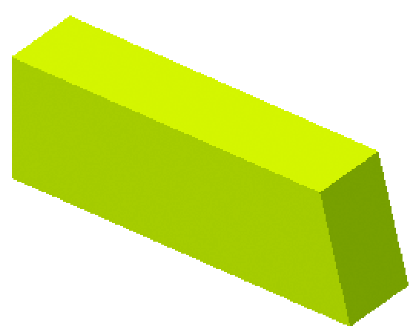
Add dimensions to fully define the sketch.



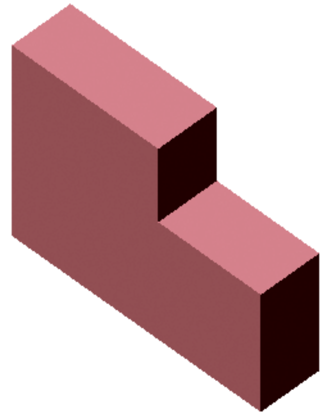
4 Extrude.

Extrude the sketch **0.5”**.

5 **Save** and **Close** the part.



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:

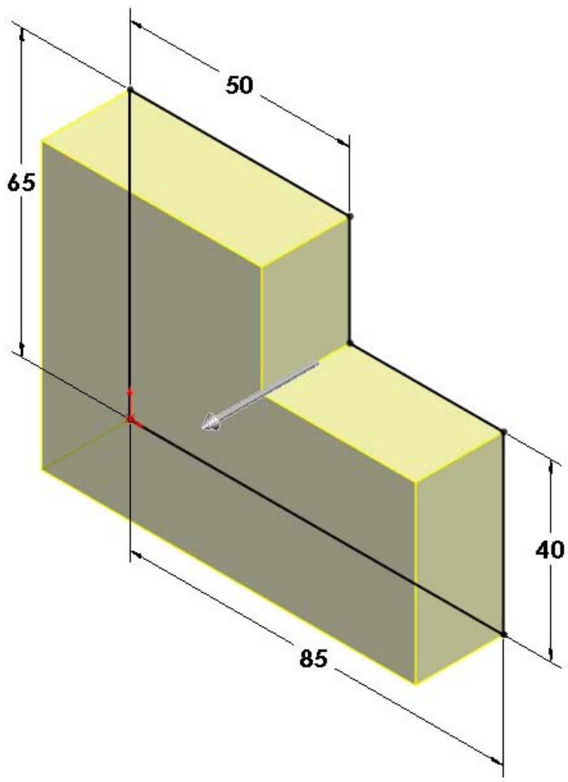
- Sketching.
- Dimensions.
- Extruding a feature.

1 New part.

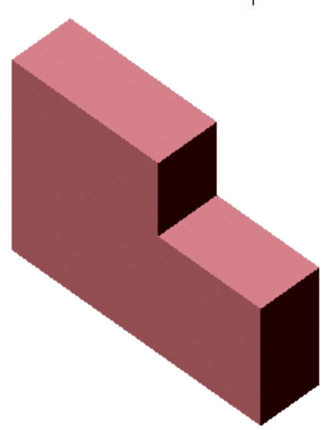
2 **Open** a new part using the `Part_MM` template.

3 Sketch and extrude.

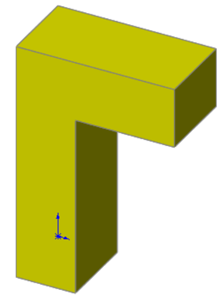
Create this sketch on the `Front` Plane using lines, automatic relations and dimensions. Extrude the sketch **20mm** in depth.



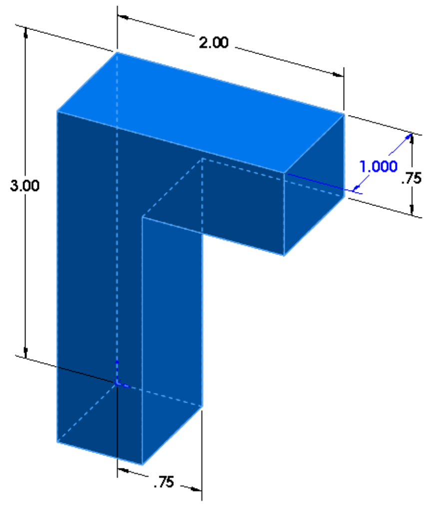
4 **Save** and **Close** the part.



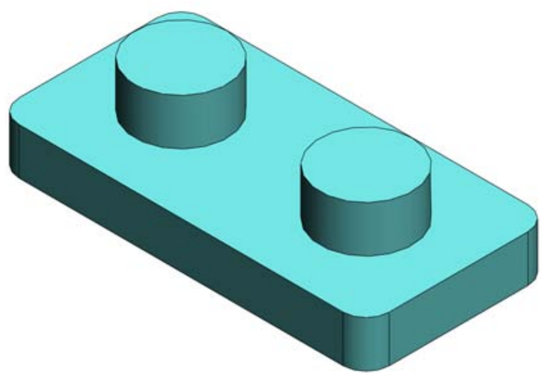
Exercise 4: Sketch Practice



Create the part shown. Start on the Front plane.

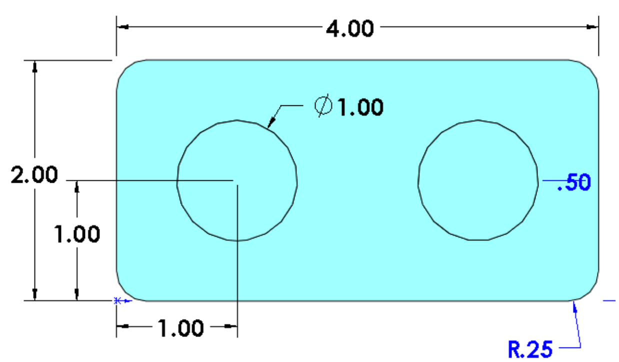


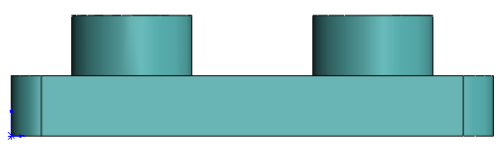
Exercise 5: Multiple Bosses



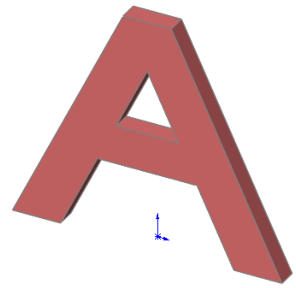
Create the part shown. Start on the Top plane. Use **Mirror Entities** to create the second cylindrical boss.

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

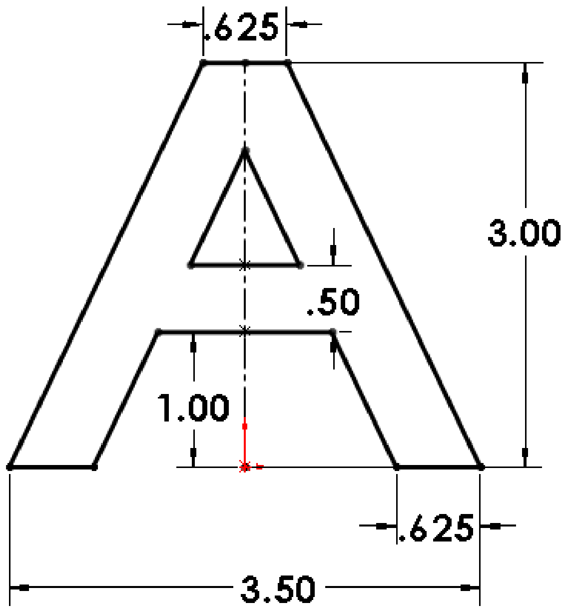




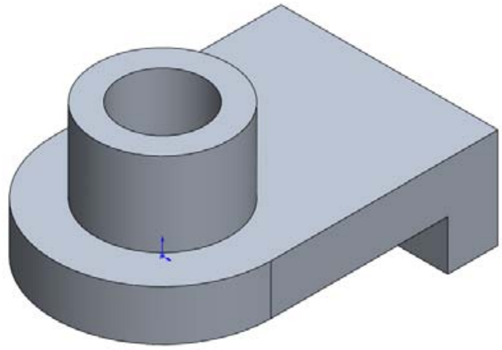
Exercise 6: Angles



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

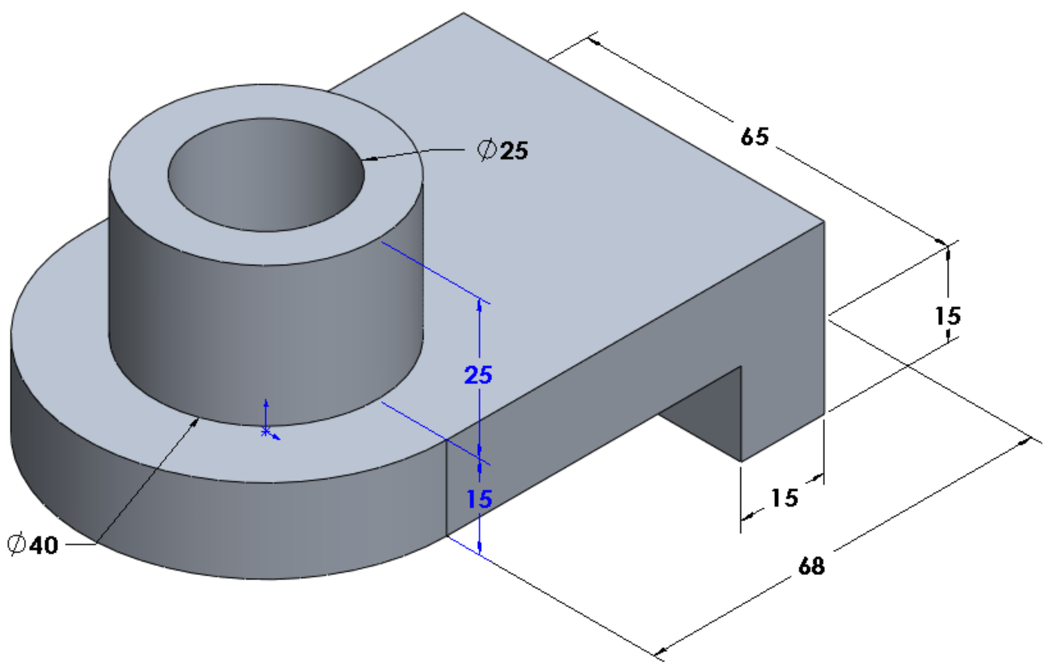
The design intent for this part is as follows:

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

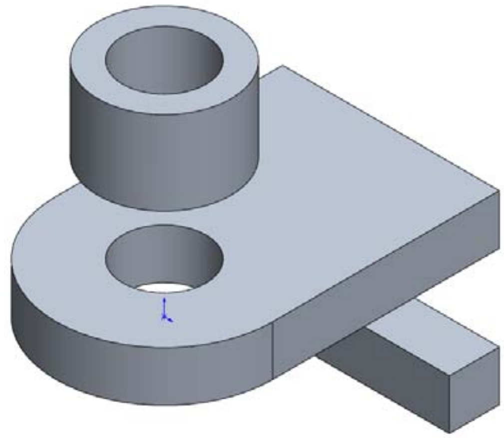
Use the `Part_MM` template.

Dimensioned View

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



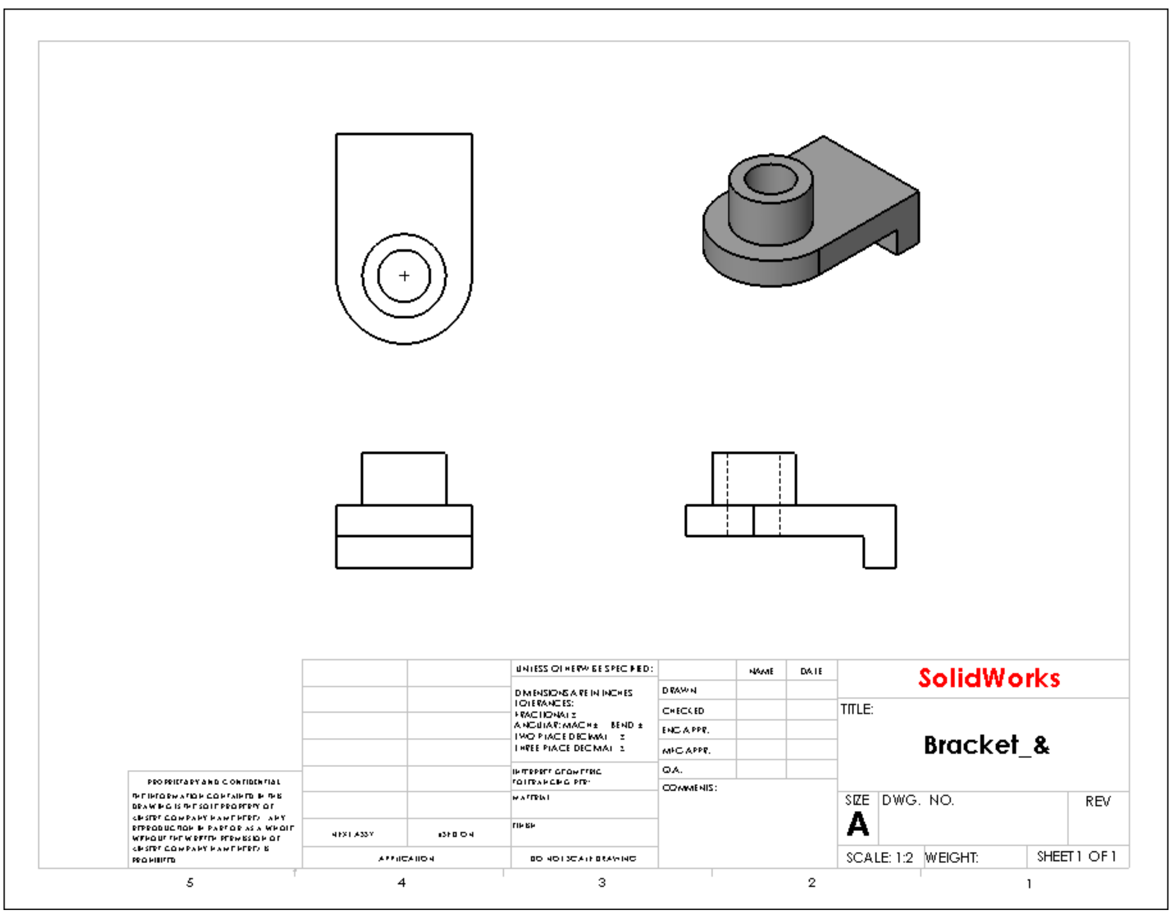
As an aid to constructing this part, visualize how it could be broken down into individual features:



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

Name: ______________________________Class: _________ Date:______________

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

1 The corner or point where edges meet:______________________________
2 The intersection of the three default reference planes:__________________
3 A feature used to round off sharp corners: __________________________
4 The three types of documents that make up a SolidWorks model: ___________
5 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

Name: ________________________________Class: _________ Date:________________

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

1 You build parts from features. What are features? ______________________________
____

2 Name the features that are used to create the `Binding Anchor` in Lesson 2. _________
____

3 How do you begin a new part document? _____________________________________
____

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

5 Give an example of an operation features that requires a selected edge or face. _______
____

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

7 What is the default sketch plane? ____
____

8 What is a plane? ____
____

9 How do you create an extruded boss feature? _________________________________
____

10 Why do you create and use document templates? ______________________________
____

11 What is a section view? ____
____

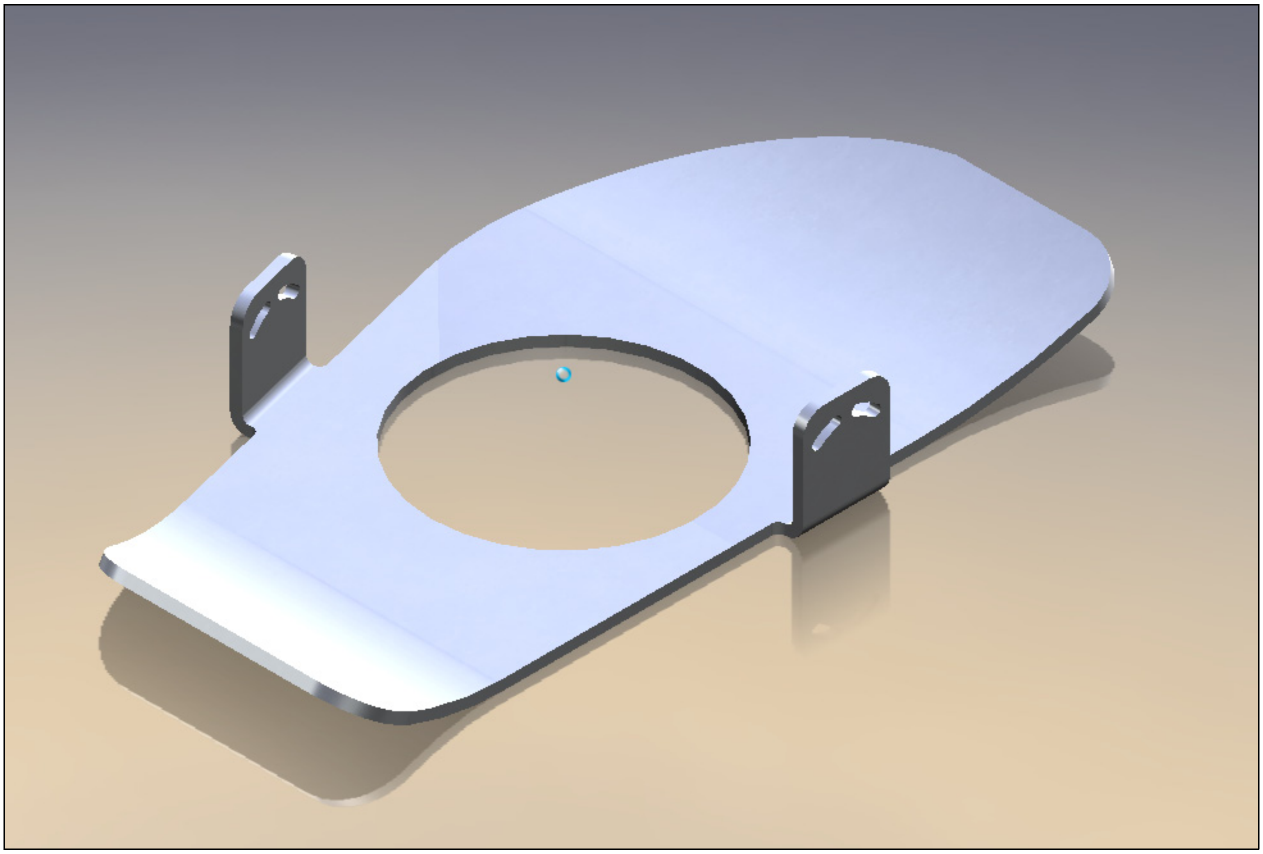
Lesson Summary

- 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

3

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

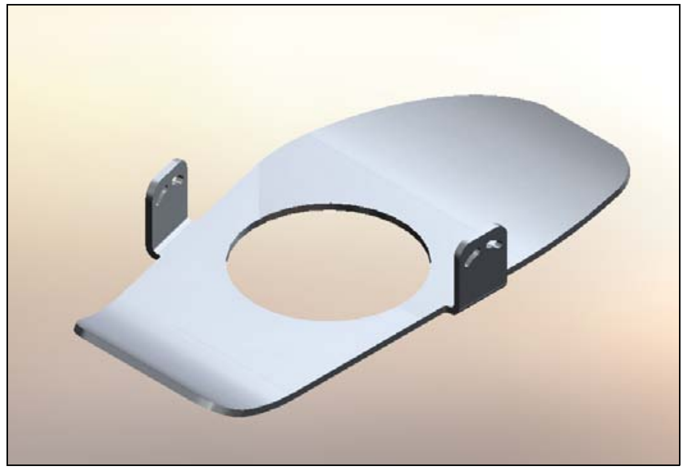
1. A SolidWorks 3D model consists of three documents. Name the three documents.
2. Parts are built from features. What are features?
3. Name the features that are used to create the `Binding Anchor` in Lesson 2.
4. What is the base feature of the `Binding Anchor`?
5. Why did you use the Fillet feature?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

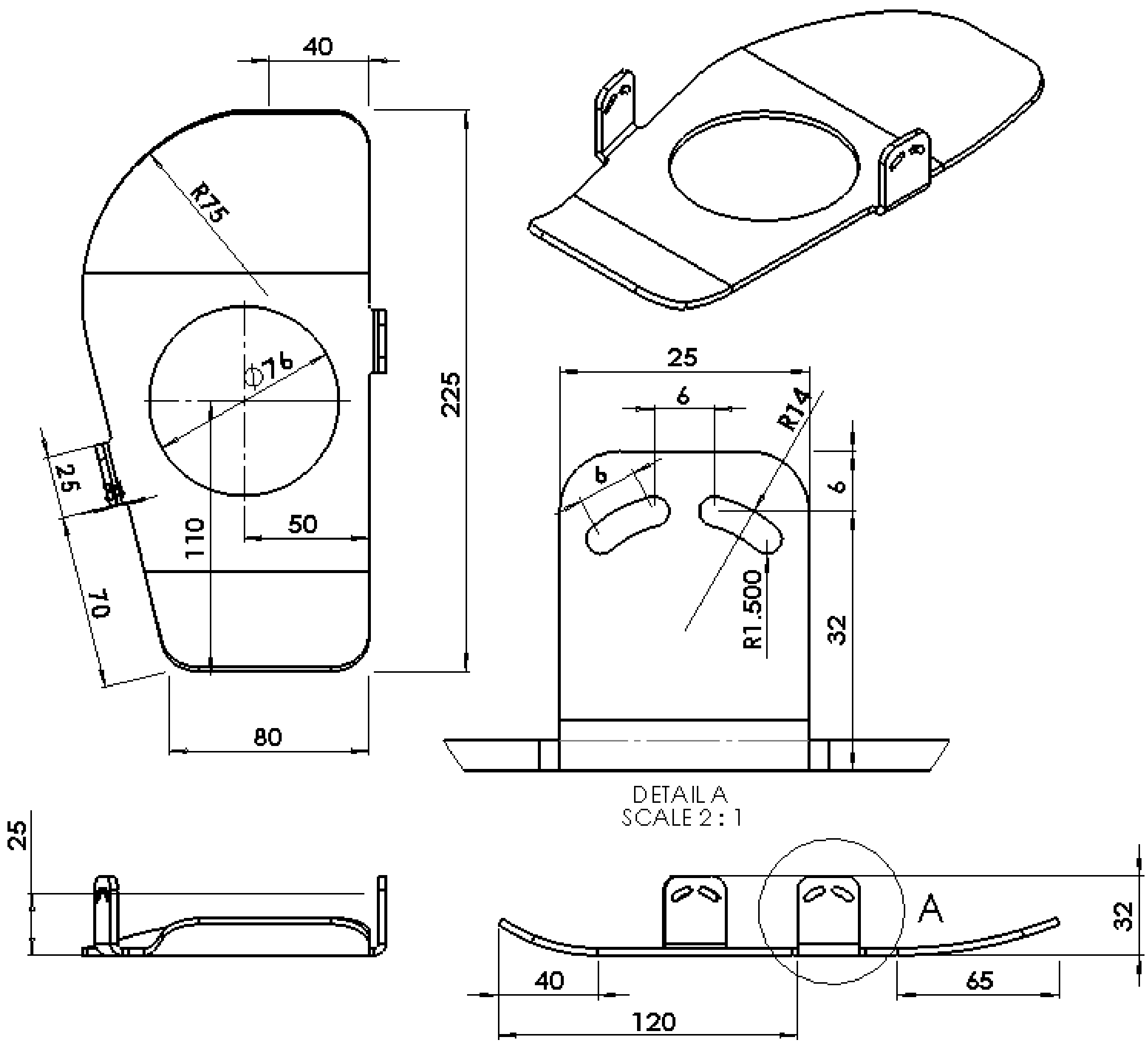
Outline of Lesson 3

- 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

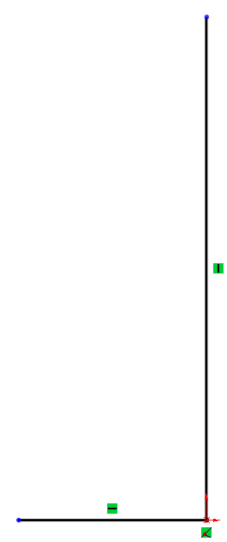
- There will be two versions of this part, one for the left foot and mirror part for the right foot.
- The part will be held in place by the Binding Anchor created in the last lesson.
- The front and back of the part will curve upward.
- Side tabs will help hold the foot centered on the binding.
- The 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.

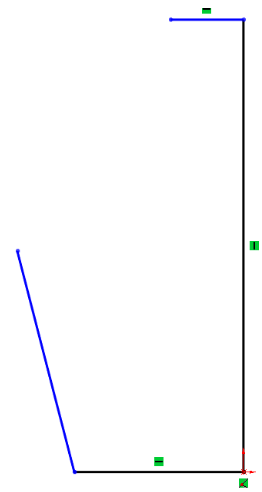
Task 1— Create the First Feature

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 35 mm.

6 Create a line from the end of the first horizontal line. Make sure that this line is *NOT* vertical.

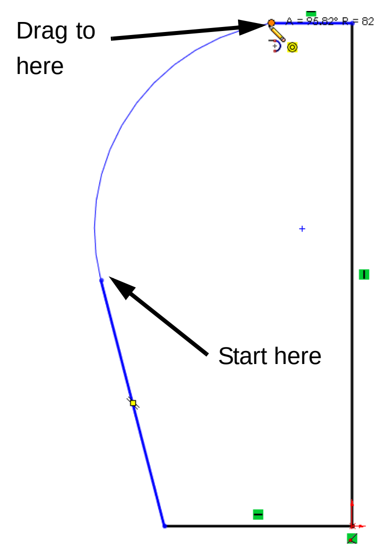


Tangent Arc

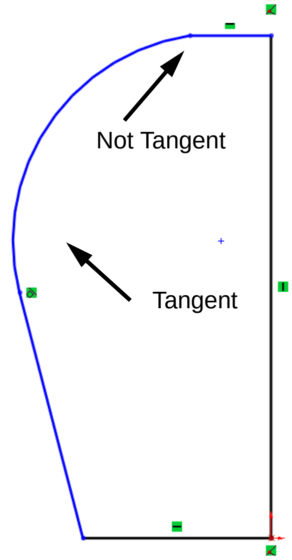
Tangent Arcs are used to create an arc that begins tangent to a selected endpoint on the sketch.

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.

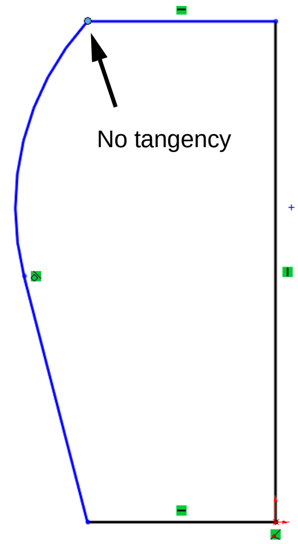


9 Check the relationships of the arc. There was a tangent relationship added between the arc and the line where the arc started.



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.



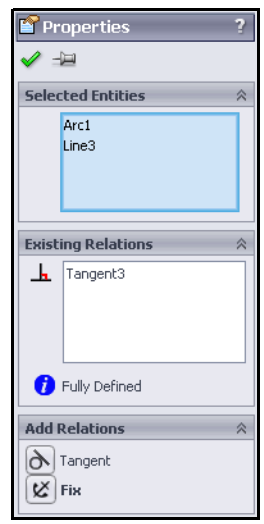
Adding Sketch Relationships

The **Add Relation** tool 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 on the **Sketch** toolbar.

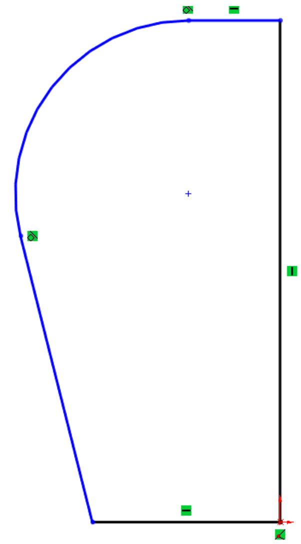
12 Select the arc and the upper horizontal line.

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.



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.

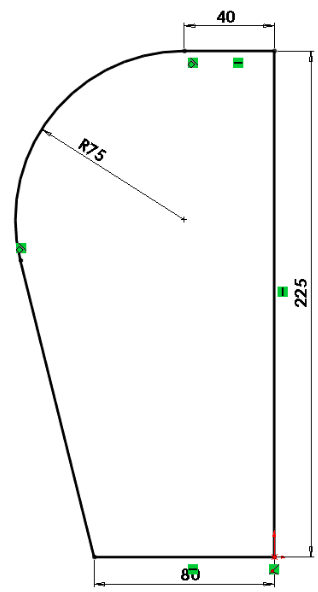
15 The callouts will show that the arc is now tangent at both ends.



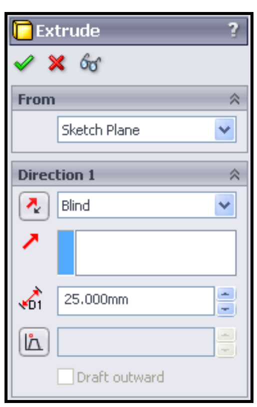
16 Fully define the sketch by adding the dimensions shown.

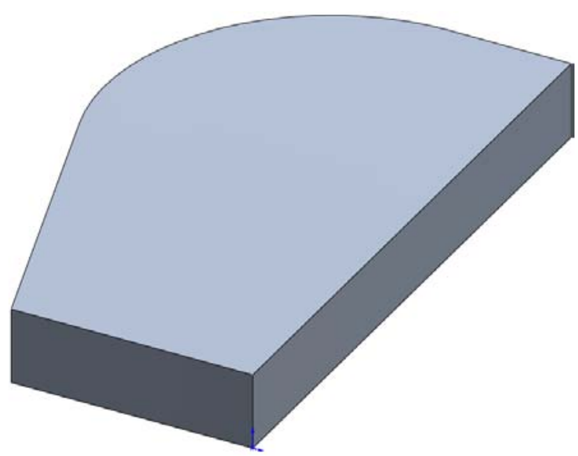
17 All the sketch geometry should now be black.

18 Extrude the feature. Click on the **Features** toolbar.



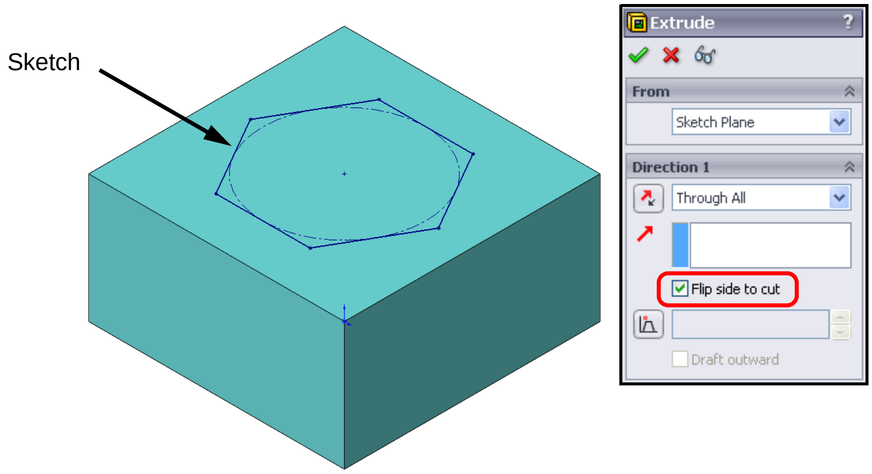
19 Create a **Blind** extrusion to a depth of **25mm**.

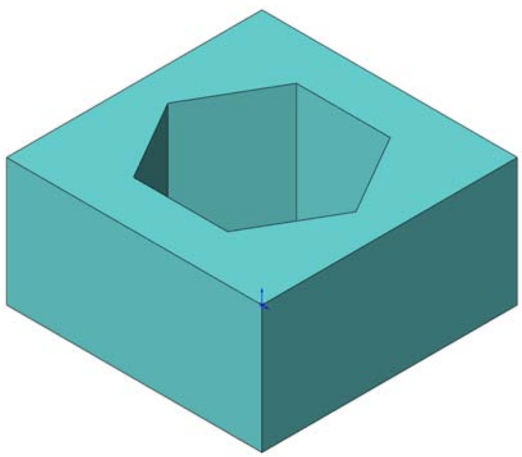




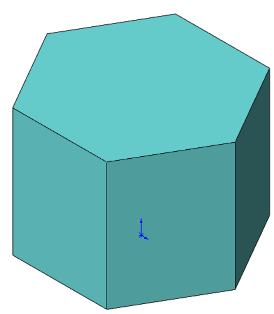
Flip Side To Cut

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.





Flip Side To Cut - Cleared



Flip Side To Cut - Selected

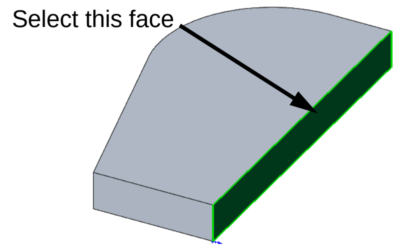
Task 2 — Cookie Cutter Cuts

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.

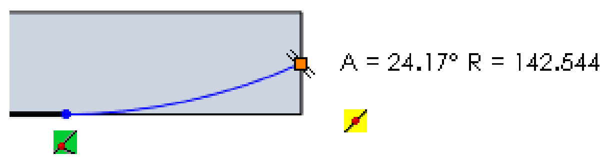
2 Change the view so that you are looking normal to the sketch plane. Click on the **Standard** Views toolbar.

3 Select the **Line** tool.

4 Position the cursor over the lower edge of the part.

When you are over the edge, the cursor feedback will be . 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.



When the cursor is over the right vertical edge, the midpoint will be displayed with a square with two diagonal lines . Once the arc is drawn there will be a midpoint relationship 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.

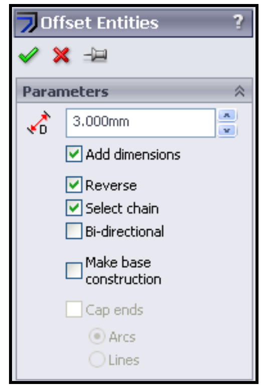
7 Dimension the sketch as shown.

Offset Sketch Entities

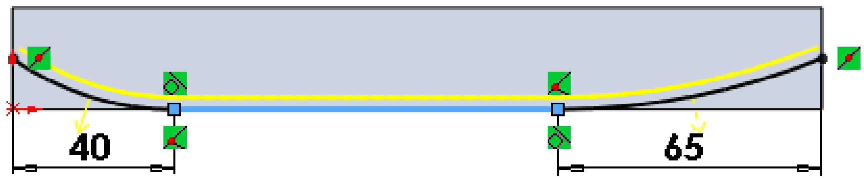
The **Offset Entities** tool is used to make a copy of sketch entities, or edges, offset from the original by some specified distance.

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.

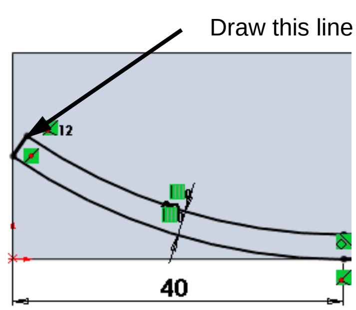


Click **OK**. Each arc and the line are duplicated at an offset distance of 3 mm.

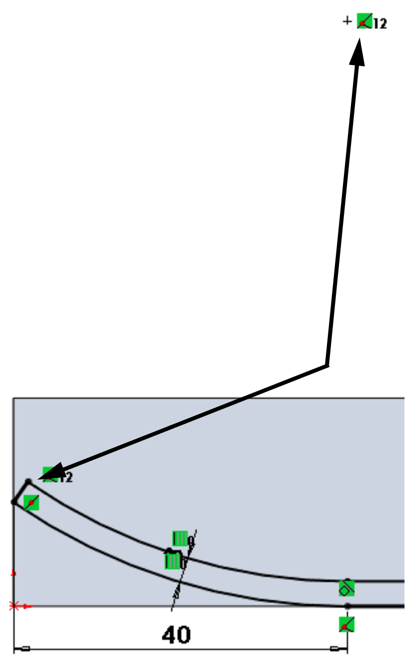
Close the Sketch

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** 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 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.

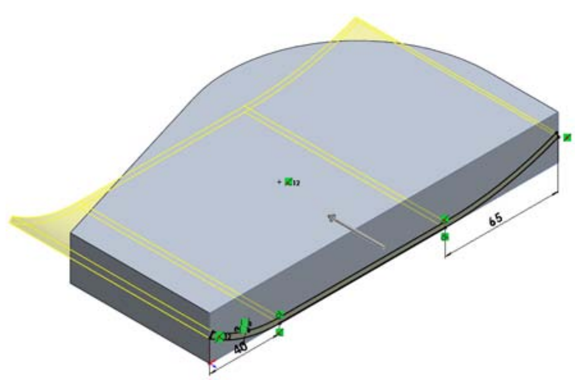
Cut to the Outside

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

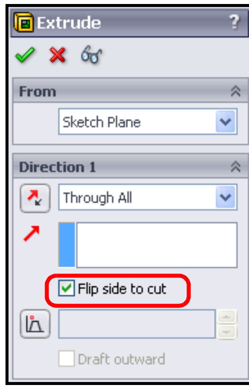
18 Click **Extruded Cut** on the Features toolbar.

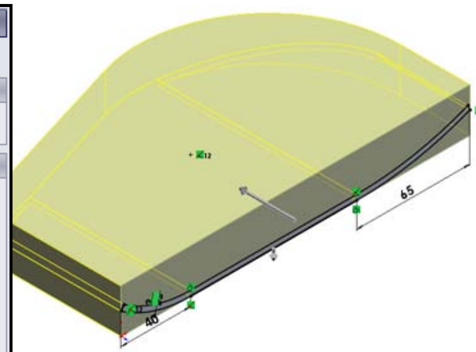
19 Select **Through All** for the end condition.

20 By default, the cut will remove the material inside the sketch.



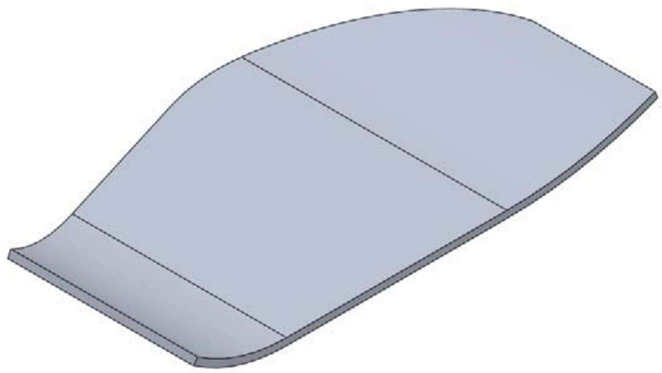
21 Select **Flip side to cut**. Now the cut will keep the material that is inside the sketch and remove the material that is outside the sketch.





22 Click **OK**.

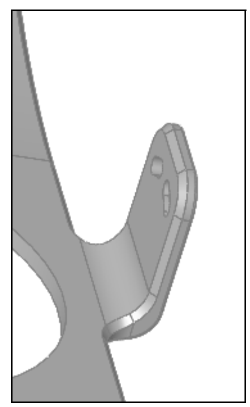
The material remaining is our curved base plate.



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

Task 3 — Creating the Side Tabs

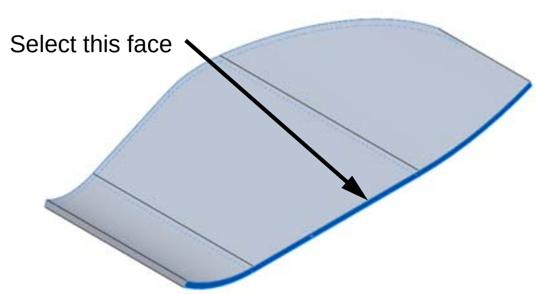
The two side tabs provide the mounting locations for the binding straps that go over the riders foot. They must be offset from the base plate to allow for the thickness of the strap. They must also be a uniform thickness as the final product will be cut from flat material and bent into the final shape.



Create First Offset

1 Select the face shown and open a sketch.

2 Change the view orientation to **Normal To** by clicking on the **Standard Views** toolbar.

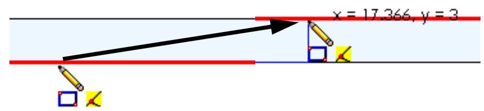


3 Select the Rectangle tool from the **Sketch** toolbar.

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.

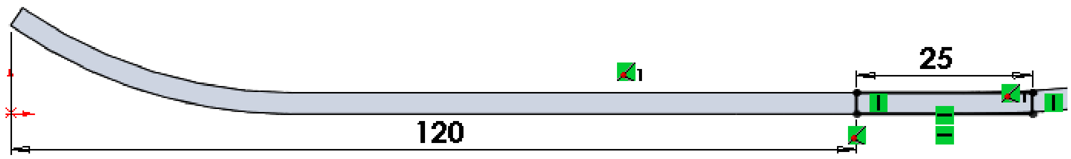
4 Start the rectangle with a coincident relationship to the bottom edge of the Binding Base Plate. Drag the rectangle until you get a coincident relationship with the top edge.



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.

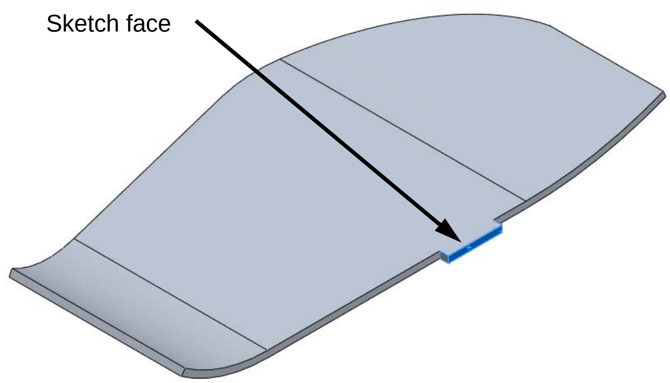
Add the two dimensions shown.



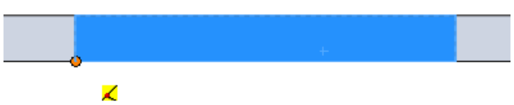
6 **Extrude** the sketch to a blind depth of **4mm**.

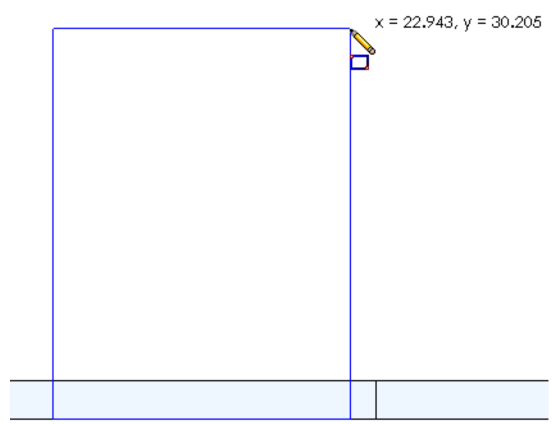
Create the tab

7 Create a new sketch on the new face created by the offset.



8 Sketch a rectangle. Start the rectangle at the lower left corner of the offset. The cursor feedback will show the yellow coincident callout to indicate we are on the endpoint of the edge.





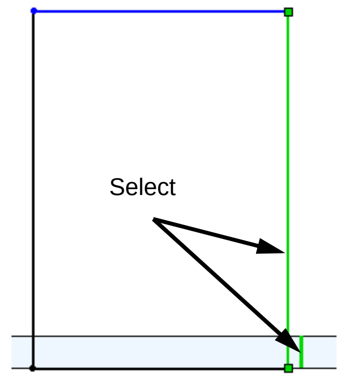
9 Add a relationship. Click **Add Relation** on the **Sketch** toolbar. Select the right vertical line of the sketch and the right vertical edge of the offset.

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**.

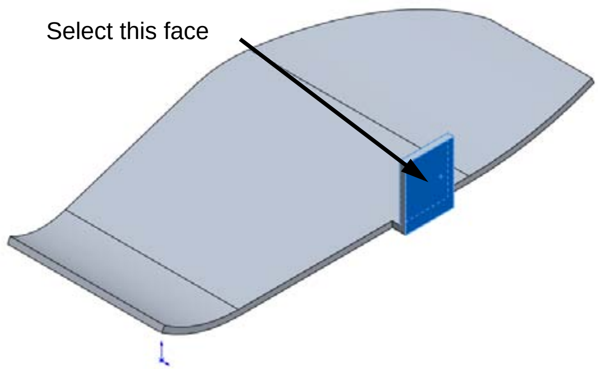


Task 4 — Binding Attachments

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

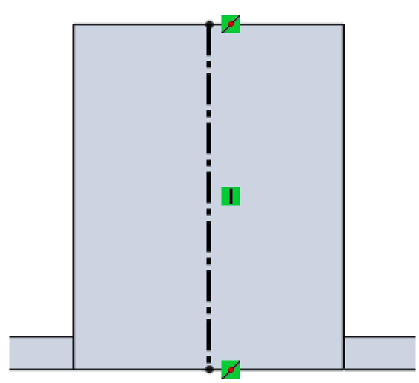
3 Zoom in on the selection by clicking **Zoom to Selection** .

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** on the Sketch toolbar.



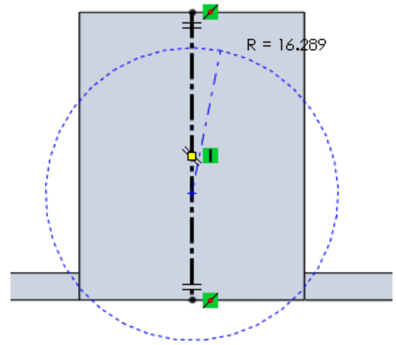
Centerpoint Arc

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.

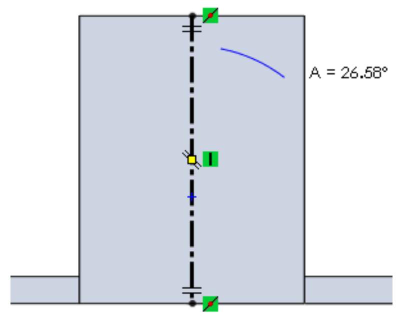
6 Select the **Centerpoint Arc** from the **Sketch** toolbar.

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.

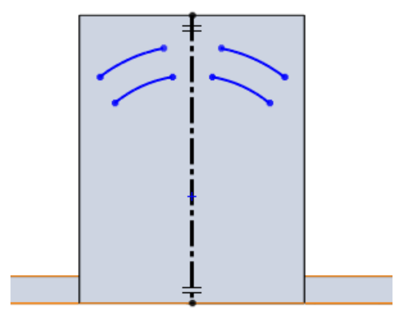


9 Press the mouse button and drag until the arc is as shown, then release the mouse button.



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.

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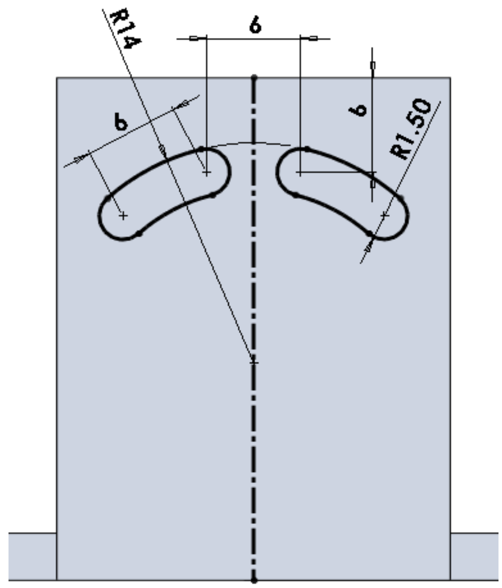
12 Turn off **Dynamic Mirror Entities**.

Dimension the Sketch

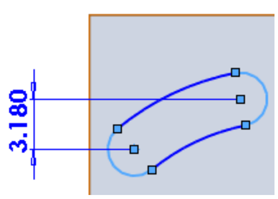
13 Add the Dimensions shown.



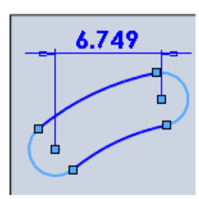
Dimension Alignment

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:

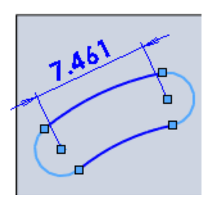
Vertical dimensions



Horizontal dimension



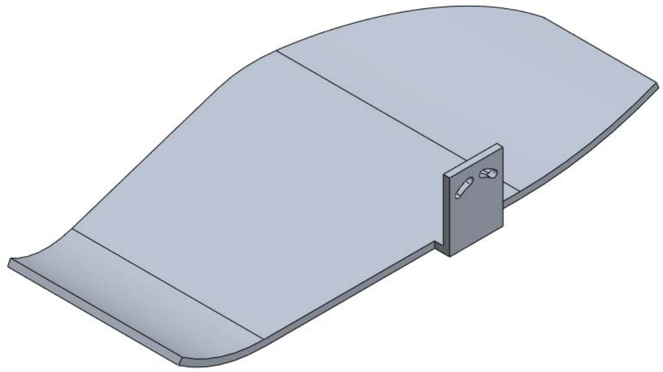
Aligned dimension



To lock in the dimension alignment you want, click the right mouse button to set it. The cursor  shows that clicking the right mouse button will lock the alignment. Once the dimension alignment is locked, the cursor changes to , 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.

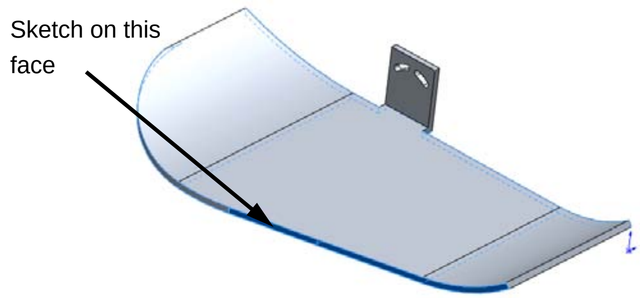
14 Extrude a cut. Select **Up To Next**.

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.

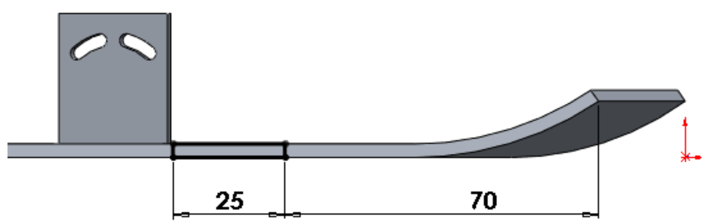


Task 5 — Create the second tab

Create another tab on the other side of the Binding Base Plate. Create the offset feature on the face shown.



Position the offset **70mm** from the vertex shown.

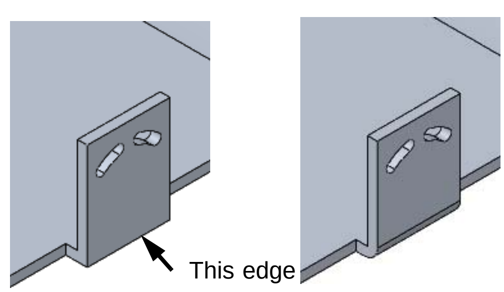


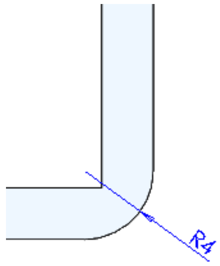
Bends

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.

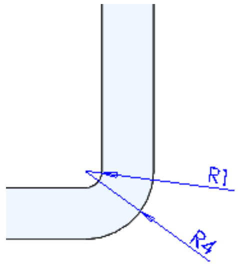
To calculate the fillet radius:

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

- Radius $_{inside}$ = Radius $_{outside}$ - Thickness
- 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.

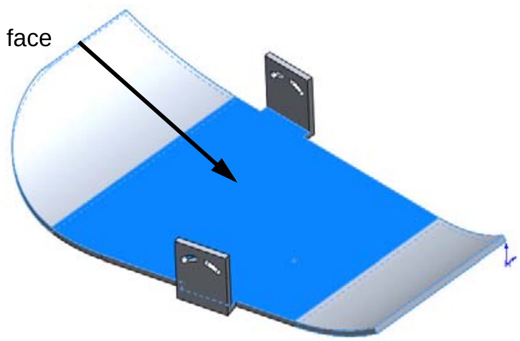
Note: Parts must be a uniform thickness in order to insert bends. This is consistent with the process in the shop as the part will be made from a single piece of material that is a consistent thickness.

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.

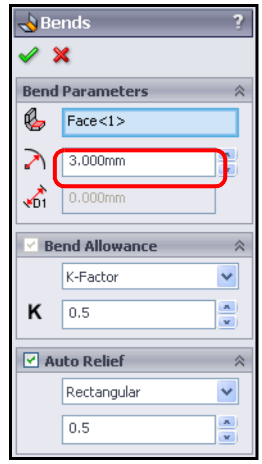
2 Click **Insert, Sheet Metal, Bends** from the menu.



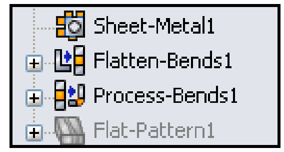
3 Set the bend radius. Change the Bend Radius to **3mm**, which is the material thickness.

Leave the remaining option as shown.

4 Click .



5 Sheet Metal Features. Four new features are added to the FeatureManager design tree.



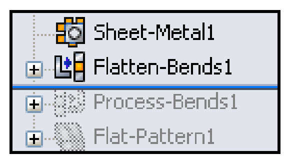
- `Sheet-Metal1` contains the sheet metal definitions such as the bend radius we entered in the last step.
- `Flatten-Bends1` creates a flat pattern or the part.
- `Process-Bends1` contains the information to bend the flat pattern into the final part.
- `Flat-Pattern1` also creates a flat pattern of the part. Note that it is gray in color indicating that it is suppressed. This means that the feature is not in use.

Rollback

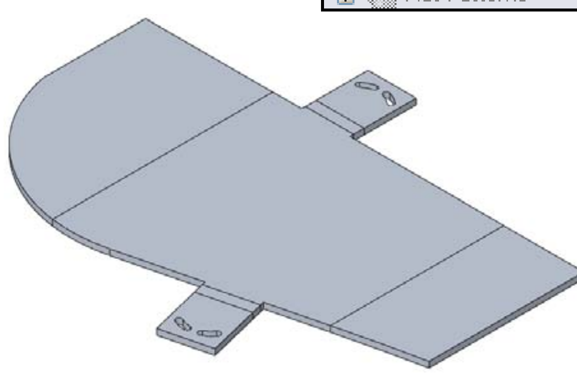
The model can be rolled back to a previous state by moving the rollback bar at the end of the FeatureManager design tree.

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 , 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

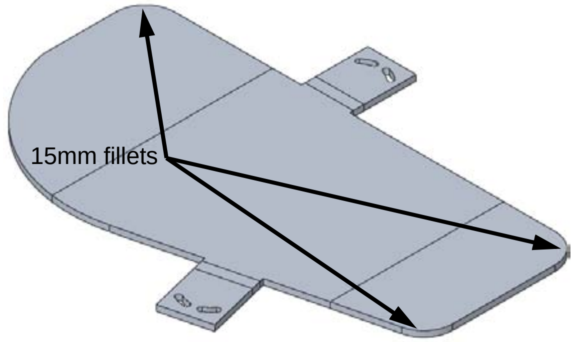
Features can be added to the flattened part just as they can with the bent up part.

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

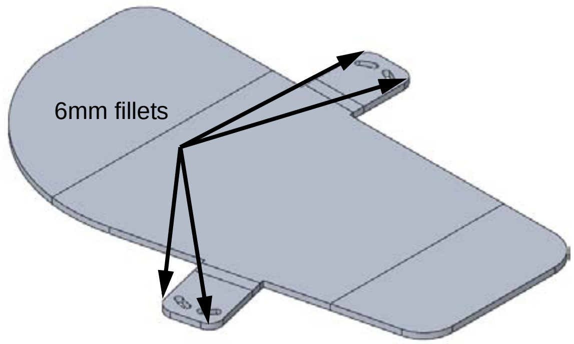
Task 7 — Add Fillets

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

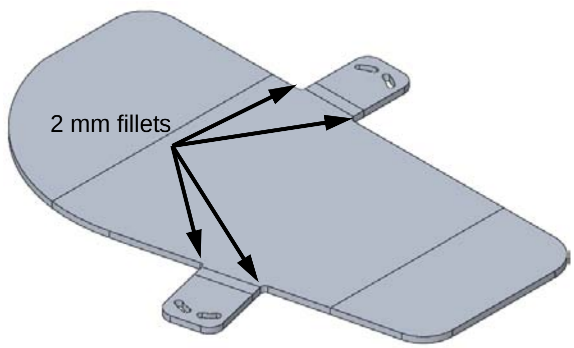
1. Add **15mm** fillets to the corners shown.



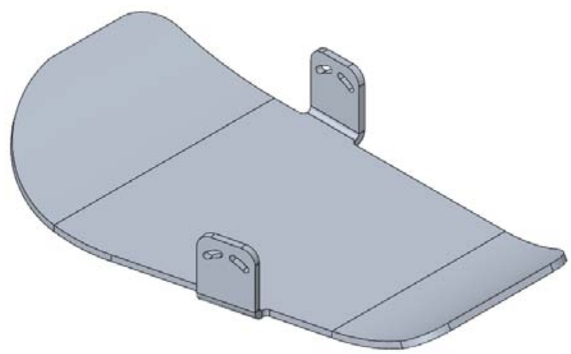
2. Add **6mm** fillets to the corners of the tabs as shown.



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

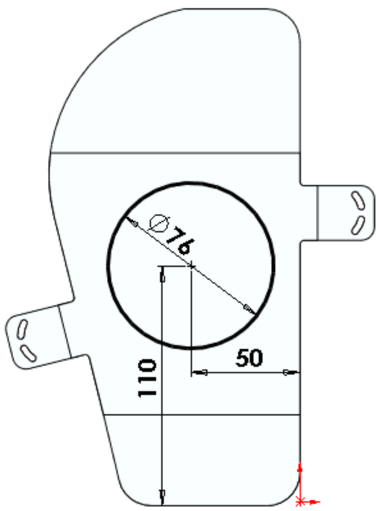
To hold the Binding Base Plate to the deck, it will have a hole into which fits the Binding Anchor created in the last lesson.

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.

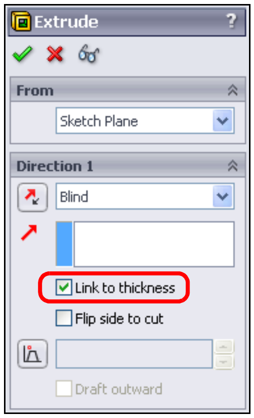
Note: Rollback can also be done through the right mouse button menus. Right-click on Process-Bends1 and select **Rollback**

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.

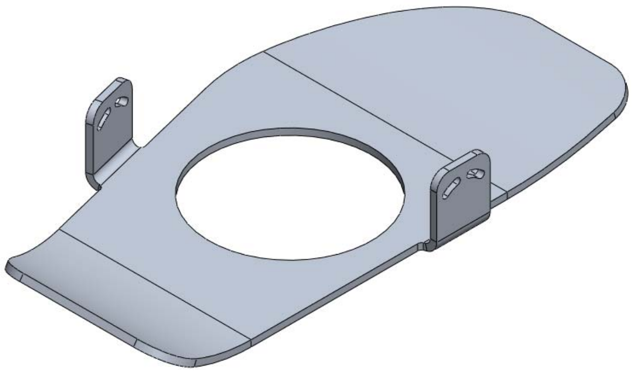


Now that the part has been turned into sheet metal by the **Insert Bends** command, a new end condition appears called **Link To Thickness**. This makes the cut depth the same as the material, even if the material thickness changes.

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 material Aluminum 2014 to the part. Either click **Edit Material** 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.

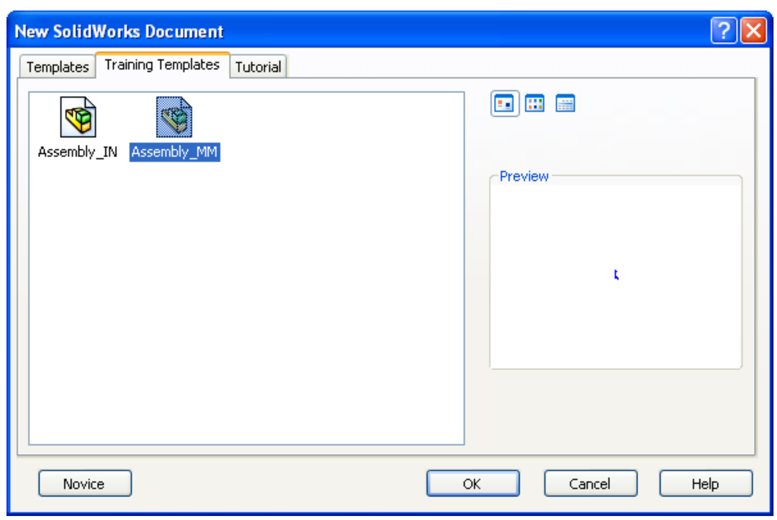
Active Learning Exercise, Part 2— Create an Assembly

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

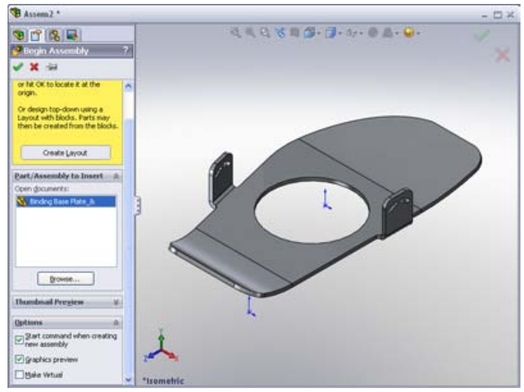
1 To create a new assembly, click **Make Assembly from Part/Assembly** on the Standard toolbar.
2 Select the template Assembly_MM and click **OK**.

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

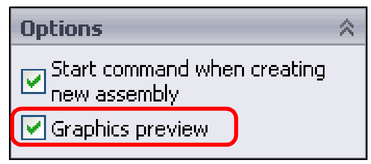


3 **Insert Component** is automatically activated by SolidWorks when we open a new assembly. This is done just to save time. We only have one file open, so it is automatically selected and a preview of the part is attached to our cursor. The part will move with the cursor.

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.



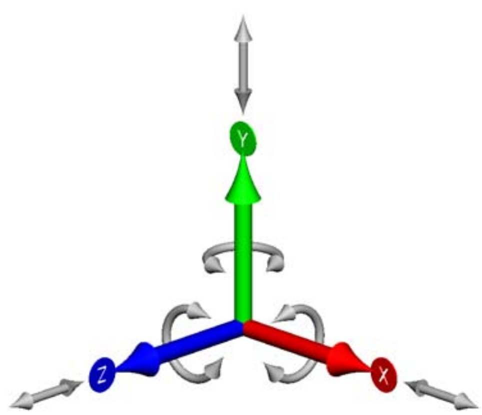
4 Place the part by moving the cursor to the assembly Origin. When the cursor is over the Origin, it will change to . Click on the Origin to place the part.

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.

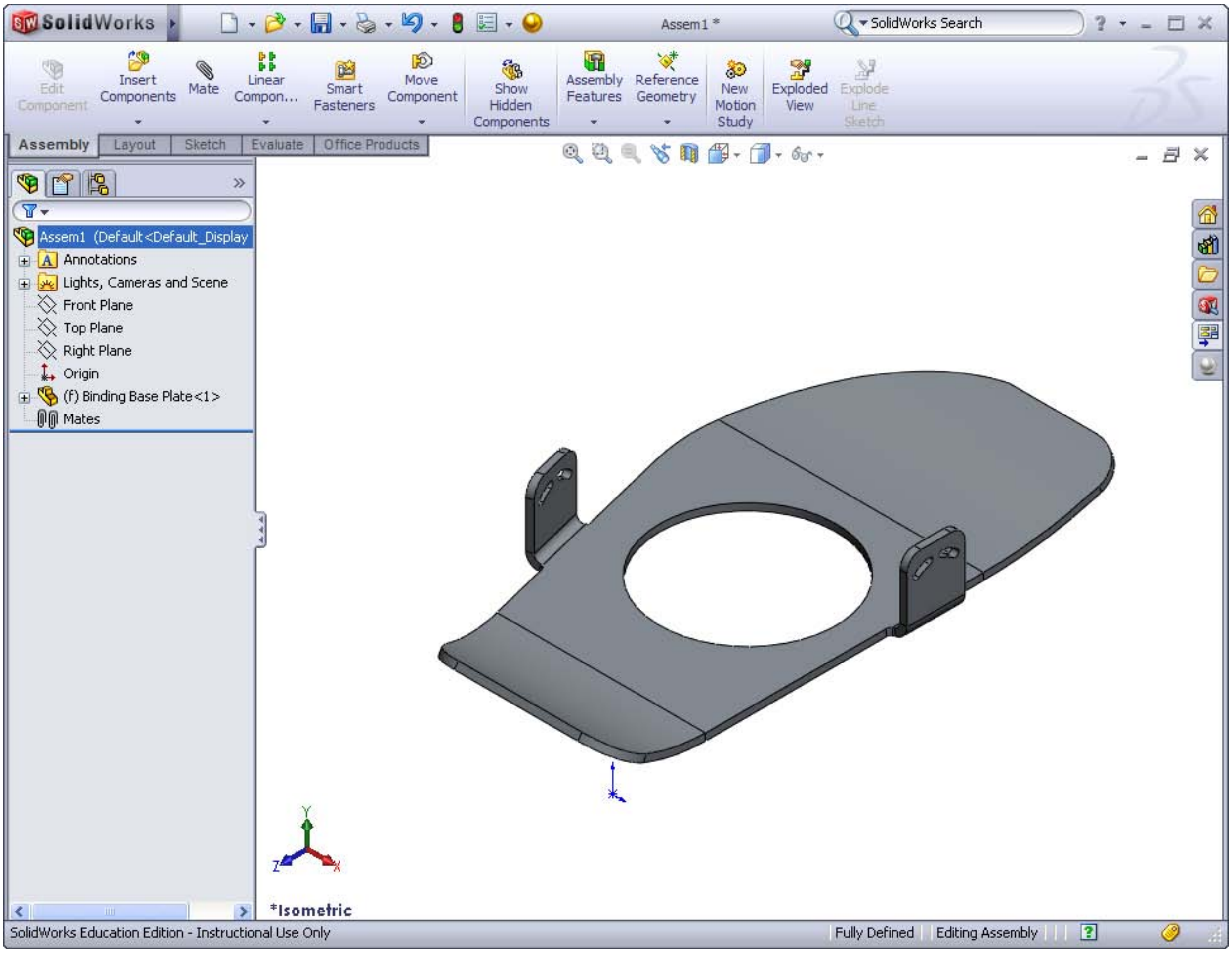
Degrees of Freedom

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.



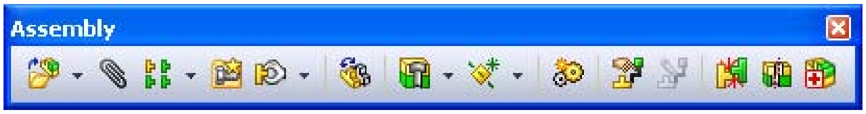
Task 2 — Inserting Parts Into an Assembly

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.



1 **Open** the part Binding Anchor.

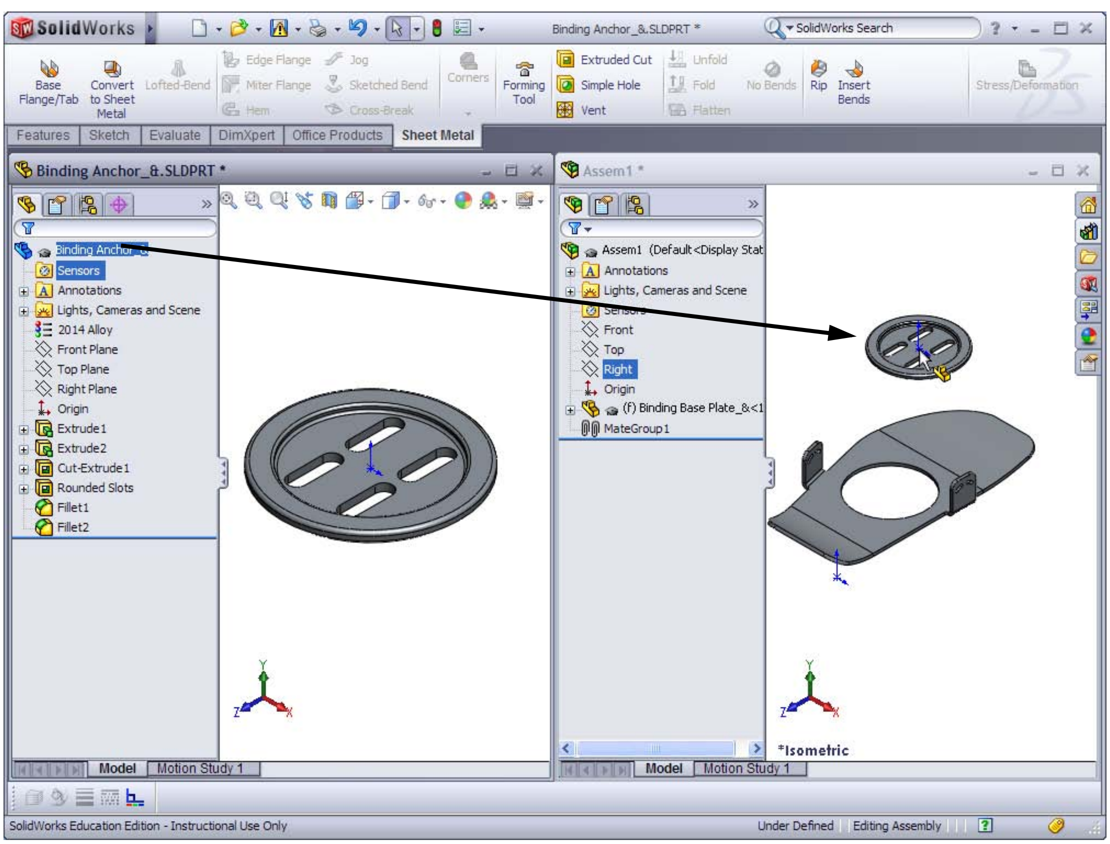
2 Tile the windows by clicking **Window, Tile Vertically** from the menu.

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.

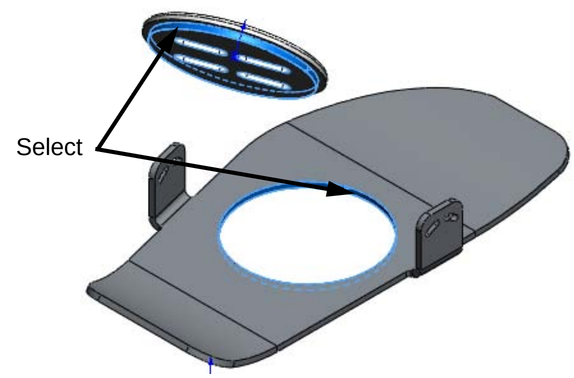
Note: Notice that the Binding Anchor can pass through the Binding Base Plate. Even thought these are “solid” models, they are still just mathematical representations of volumes in space.

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** on the **Assembly** toolbar to add a mate.

4 Select the two faces shown.

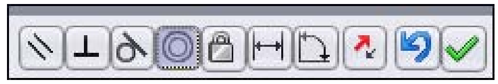
When you pick the second face, the parts will move into alignment for a concentric mate and the **Mates** toolbar will appear.



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.

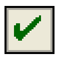
Mate Pop-up

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.

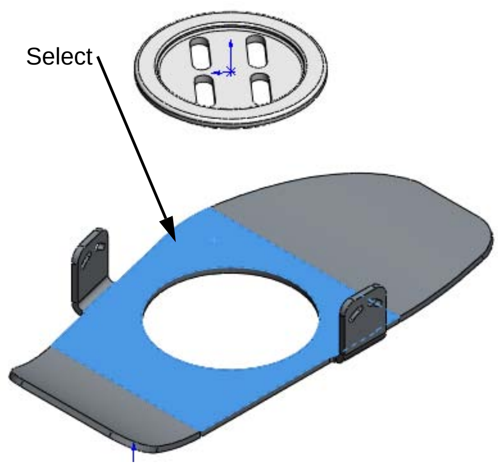
Either the on-screen or PropertyManager dialog can be used.

5 Click  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** is selected.

Click 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**.

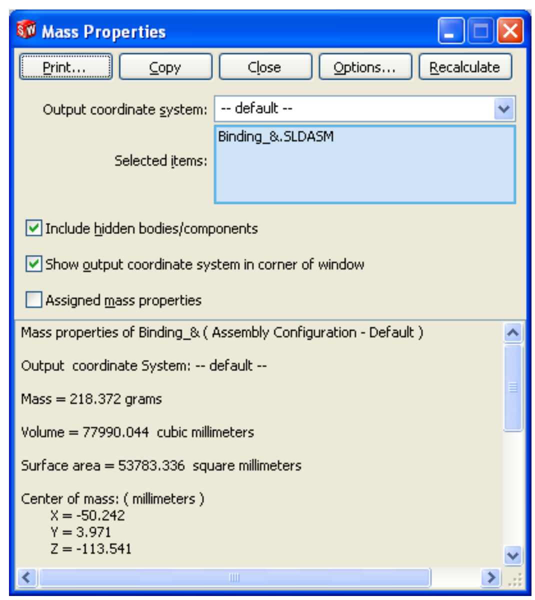
Task 4 — Save the Assembly

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 **218.372 grams** (0.481 pounds).

Question: What material would SolidWorks use if we forgot to apply a material to all the parts?

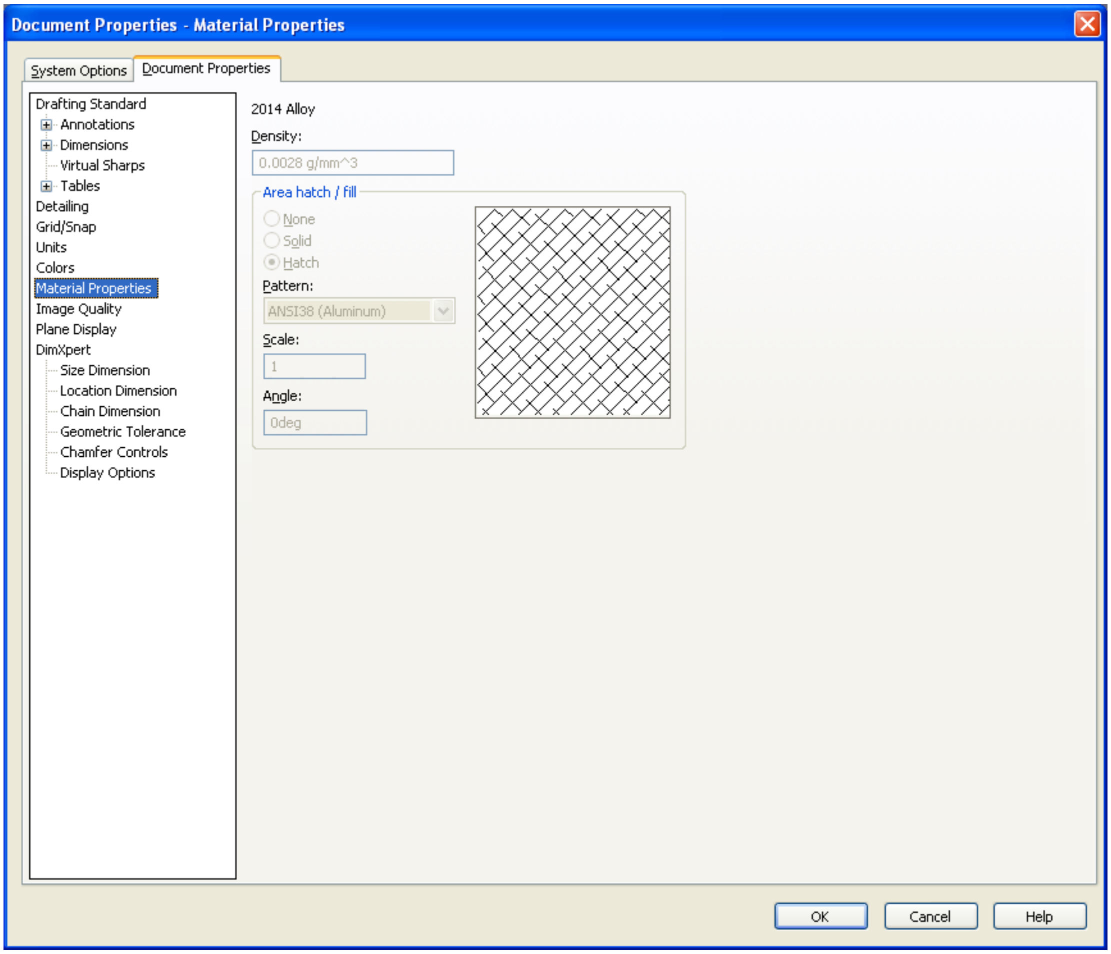
Answer: Each part template has a default material. If we do not apply a different material, the weight will be calculated using the default material which has a density of **0.001 g/mm^3**.

2 Open the Binding Base Plate part. In the FeatureManager design tree, right-click the Binding Base Plate and select **Open Part**.

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.



6 Click **OK** to close the **Options**.

7 **Save** and **Close** all open files.

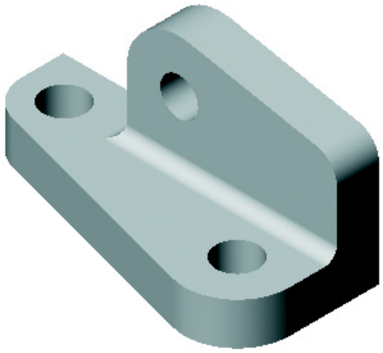
5 Minute Assessment – #3

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.

Exercises and Projects

Exercise 9: Base Bracket

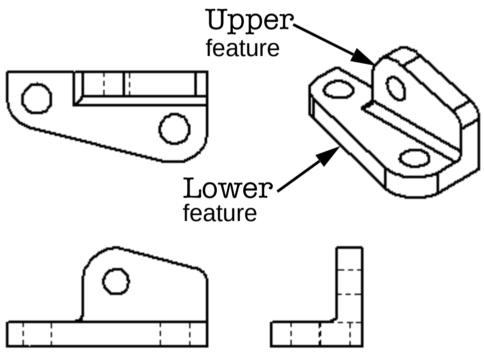
This exercise reinforces the following skills:



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

Design Intent

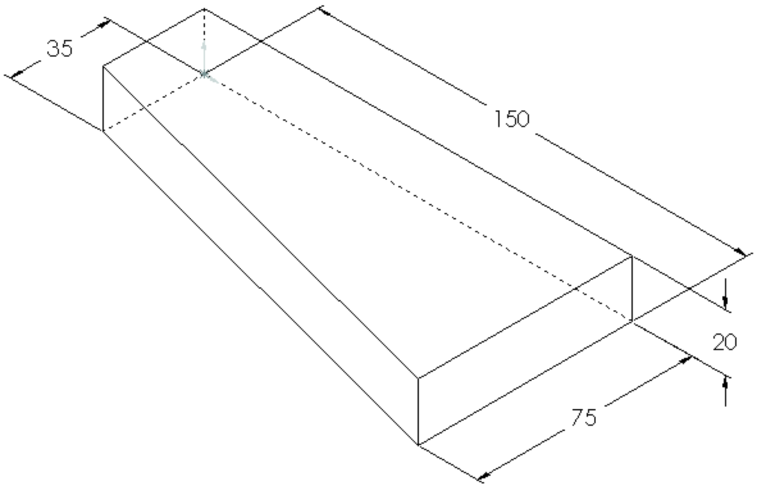
- ❑ Some aspects of the design intent for this part are:
- ❑ Thickness of the `Upper` and `Lower` features are equal.
- ❑ The holes in the `Lower` feature are equal diameter.
- ❑ The `Upper` and `Lower` features are flush along the back and right side.



Open a new part using the `Part_MM` template.

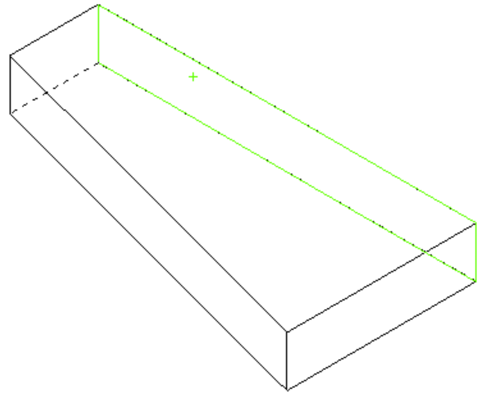
1 Create the **`Lower`** feature.

2 Use lines to sketch this profile. Add dimensions to fully define the sketch.



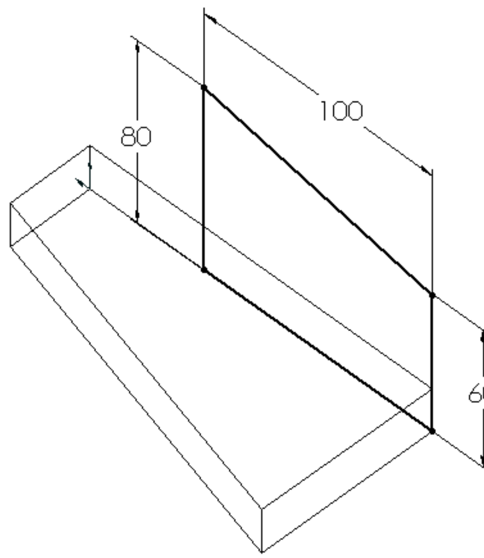
3 Select a face as sketch plane.

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.



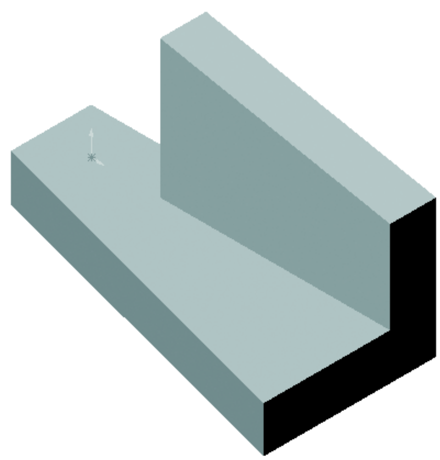
4 Create the **`Upper`** boss feature.

Sketch the lines and relate them to the existing edges where they should be coincident.



5 Extrude.

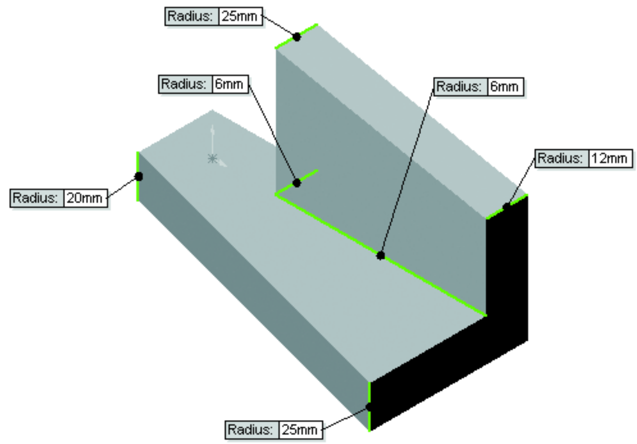
Extrude *into* the first feature a depth of **20mm.**



6 Create fillets and rounds.

Add the fillets in as few steps as possible.

Rename the features according to fillet size.

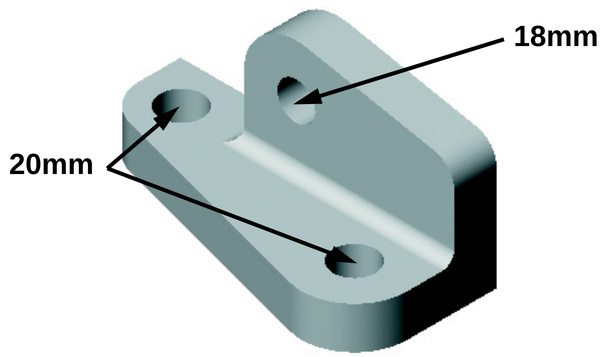


7 Holes.

Add the holes using as few features as possible. Make sure that the holes lie concentric to the fillet radii.

For the Hole Wizard, use ANSI Metric Drill Sizes.

8 **Save** and **Close** the part.



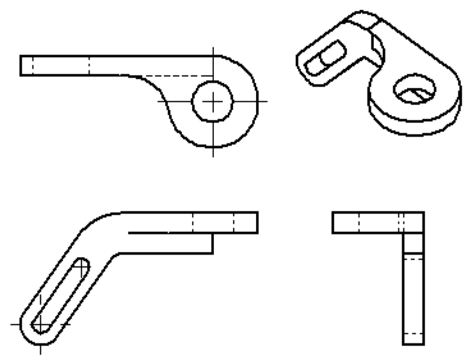
Exercise 10: Guide

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:



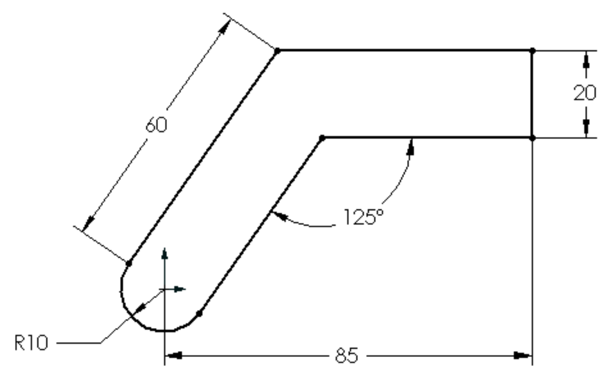
- ❑ Part is not symmetrical.
- ❑ Large circle is tangent to outer edge.
- ❑ Large circle is coincident with underside brace edge.
- ❑ Plate thicknesses are equal.

Procedure

Open a new part using the `Part_MM` template.

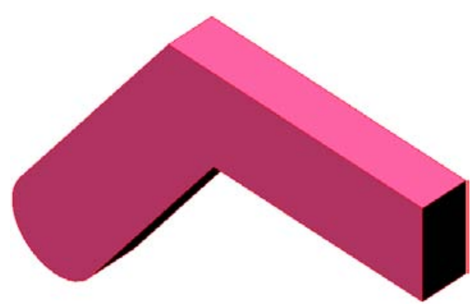
1 Sketch the profile.

Using the `Front` plane, create the profile.



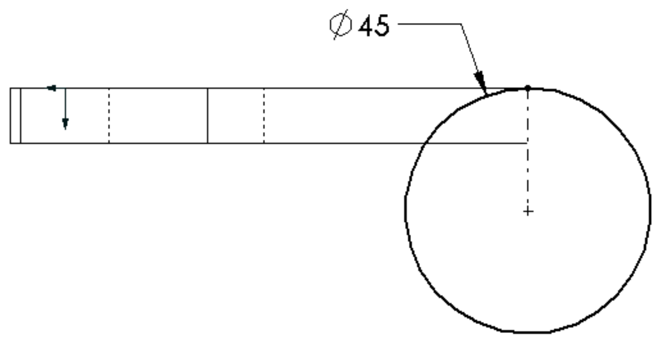
2 Extrusion.

Extrude the sketch **10mm**.



3 Upper sketch.

Start a sketch on the top face of the model. The circle is tangent to one edge and coincident to another edge.

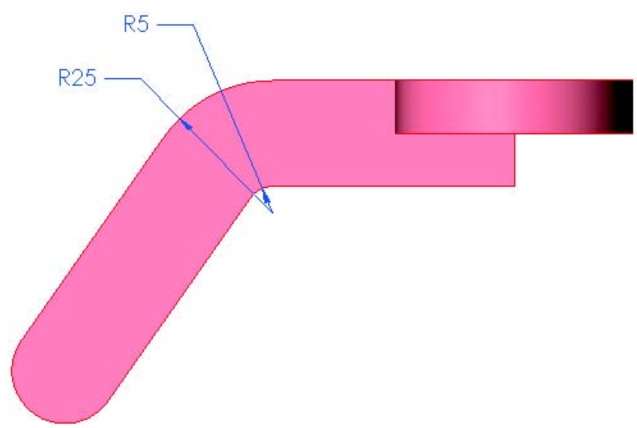


4 Extrude equal thickness.

Extrude the circle the same thickness as the first feature.

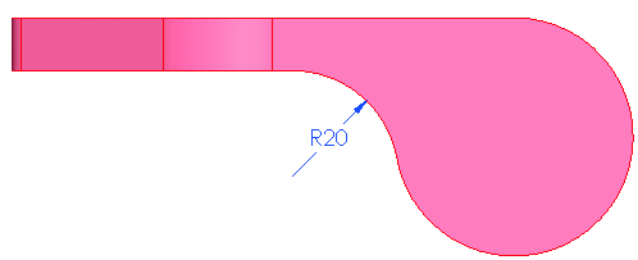
5 Add two fillets.

Add two fillets as shown.



6 Last fillet.

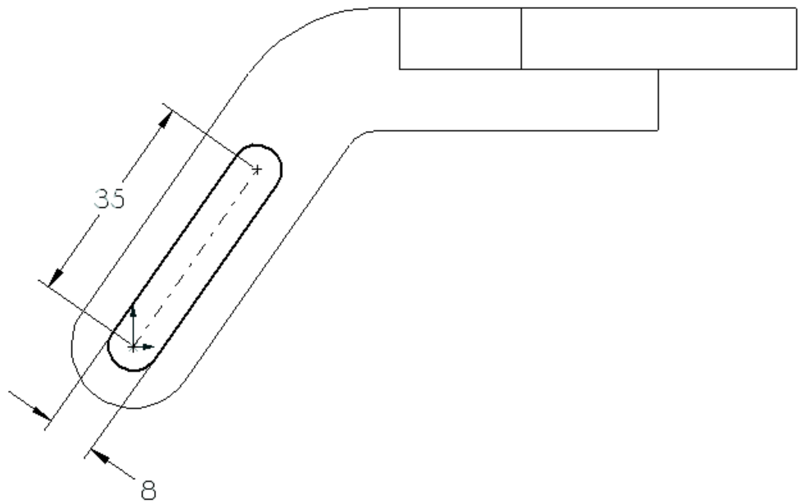
Create a third fillet with a **20mm** radius.

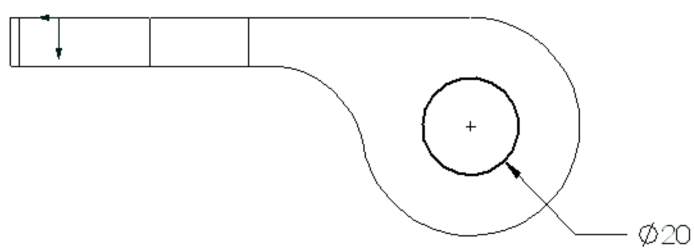


7 Cuts.

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.

Note: 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

Name: ______________________________Class: _________ Date:______________

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

1 How do you begin a new part document? ___________________________________
____

2 How do you open a sketch? ____
____

3 What is the Base feature? ____
____

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

5 How can you change a dimension value?___________________________________
____

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

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?_________________
____

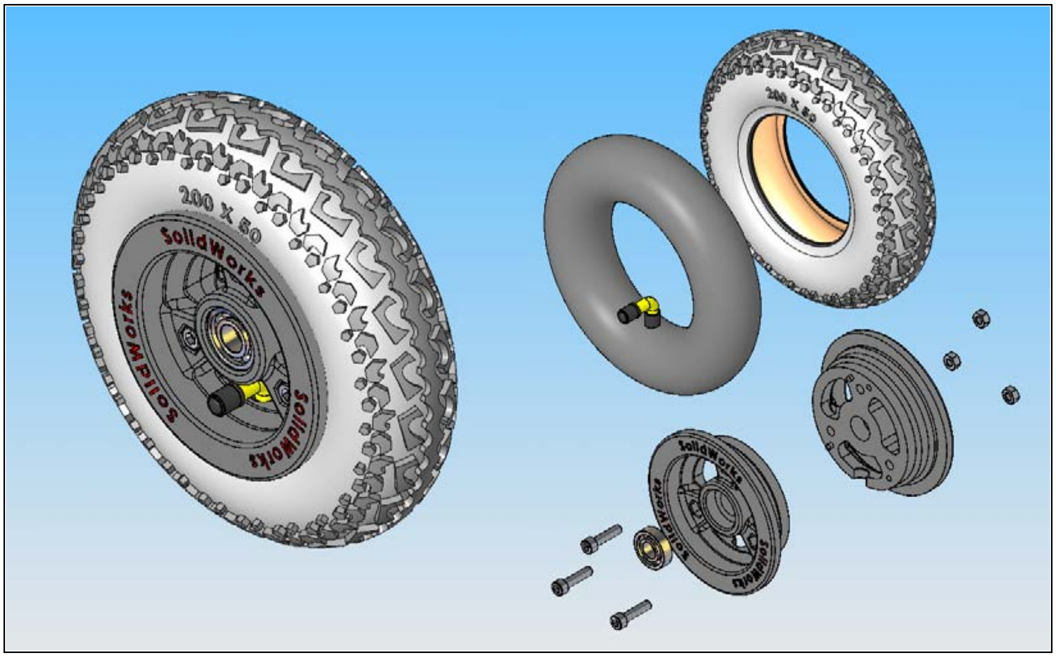
Lesson Summary

- 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.

4

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:

- *Revolves and Sweeps*
- *Pattern Features*
- *Import/Export*
- *Toobox*

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

Review of Lesson 3 — The Binding

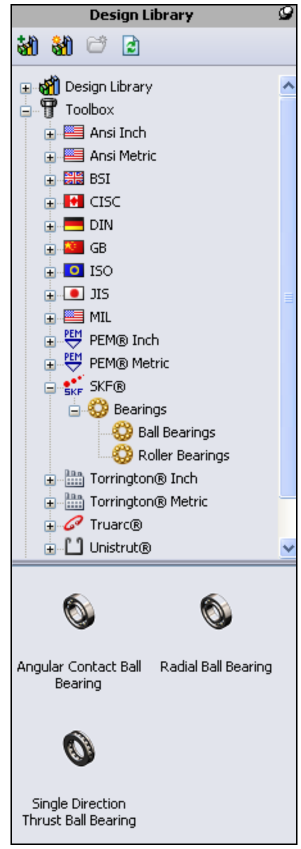
Questions for Discussion

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?

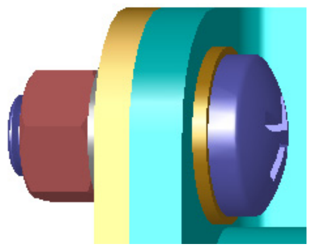
Outline of Lesson 4

- 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
- 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.



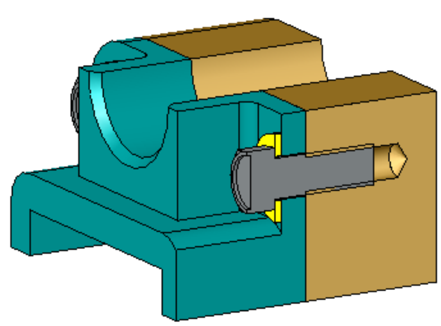
To add these parts to an assembly, select the type of part you want to insert, then drag the Toolbox part into your assembly. As you drag Toolbox parts, they snap to the appropriate surfaces — automatically establishing a mate relationship. In other words, a screw recognizes that it belongs in a hole 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, BSI, CISC, DIN, GB, ISO, JIS 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

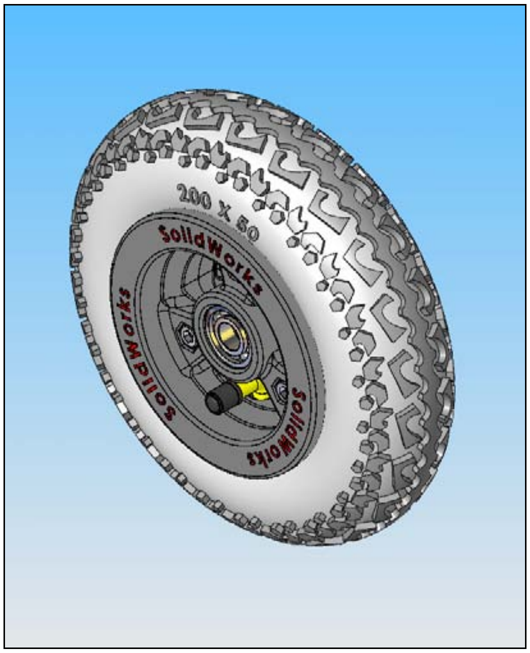


Before you placed the washers and screws, you should have measured the depth of the holes and the thickness of the washer as well as the diameter of the holes.

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

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

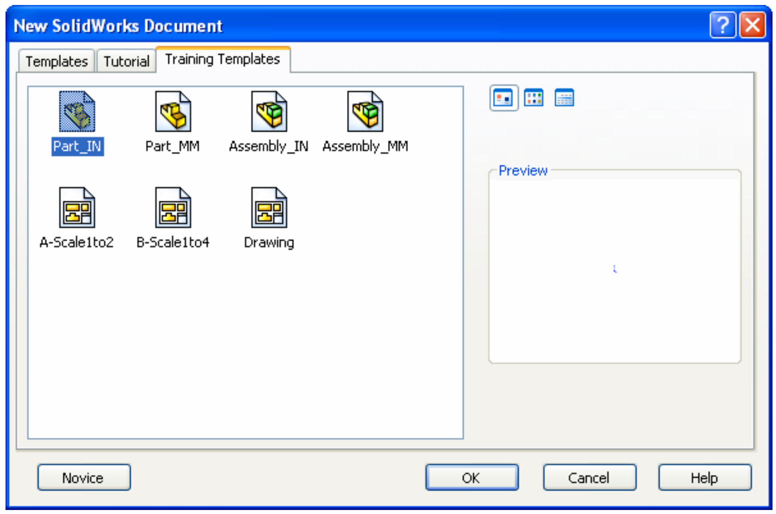
1 Create a new part. Click **New** on the Standard toolbar.

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

2 Click the **Training Templates** tab.

3 Select the **Part_MM** icon.

4 Click **OK**.



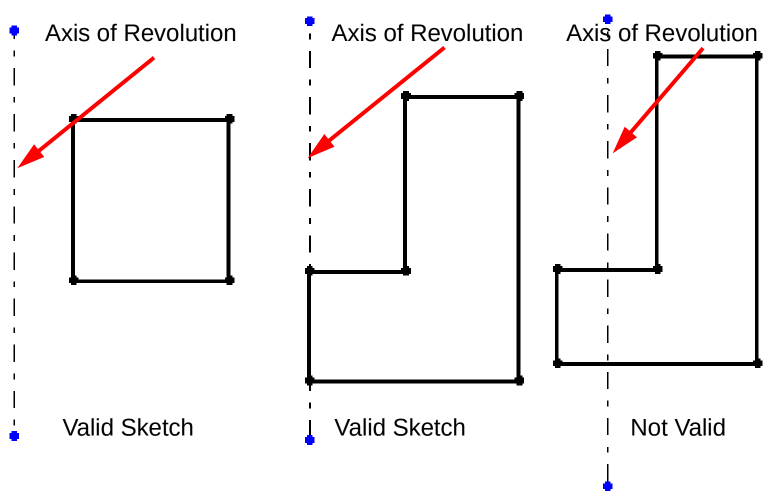
5 **Save** the new part as `Wheel Hub`. Save the file to the `Mountainboard\Wheel Assembly` folder.

Revolved Features

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

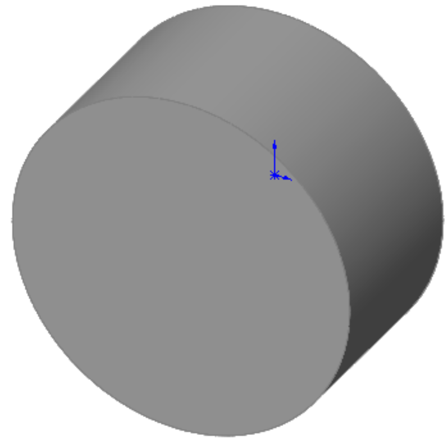
There are three simple rules for revolved features:

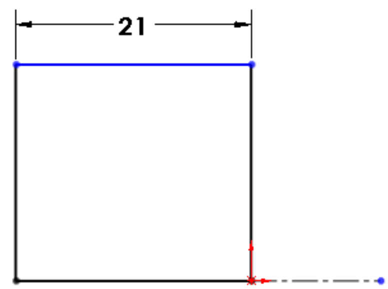
- A centerline or sketch line must be specified as the axis of revolution.
- The sketch must not cross the axis.
- The axis of revolution for the revolve must be selected before creating the revolved feature.



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.



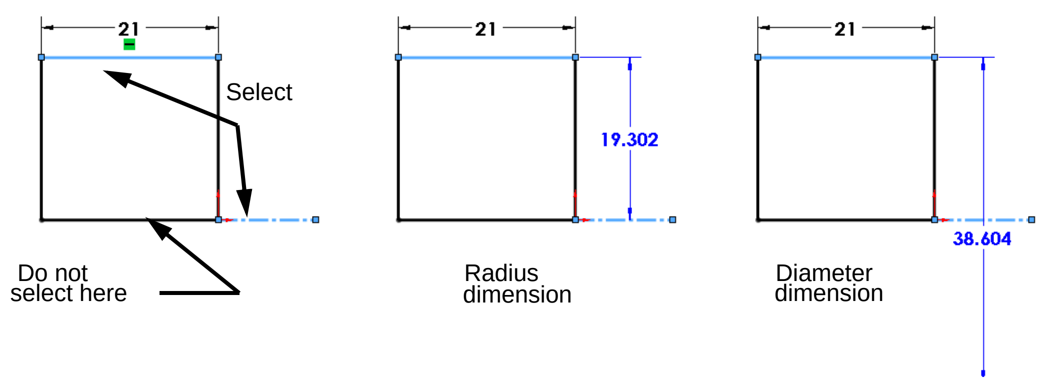


1. Create a sketch on the Right plane.
2. Sketch a rectangle, approximately 25 mm by 25 mm, with the Origin at the lower right.
3. Sketch a centerline from the Origin, horizontally to the right. The length is not important.

   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**.

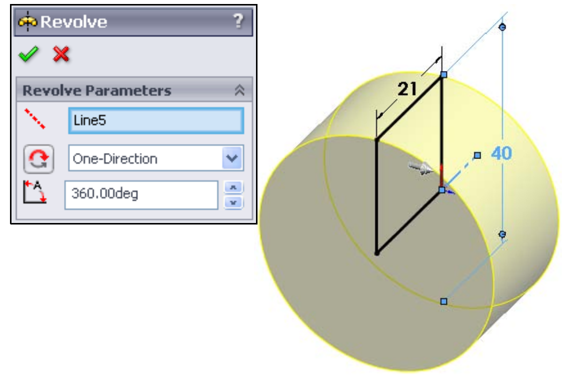


6 Revolve the hub. Click **Revolved Boss/ Base** on the Features toolbar.

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

Select **One-Direction** and **360 deg** for the type and angle.

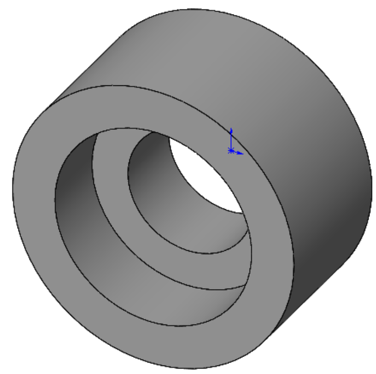
Click **OK**.



7 Rename this feature Hub.

Task 3 — Cut the center holes.

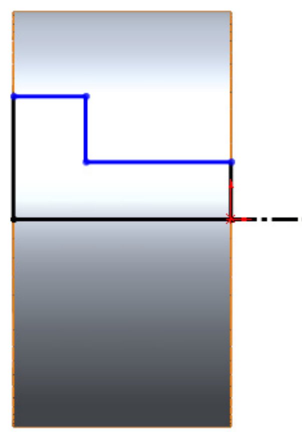
Cuts can also be created as revolved features. This closely represents the machining operation of a lathe.



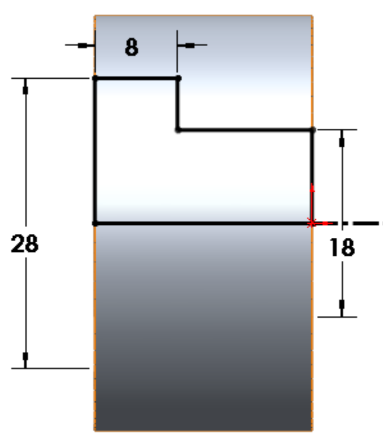
1 Create a sketch on the Right plane.

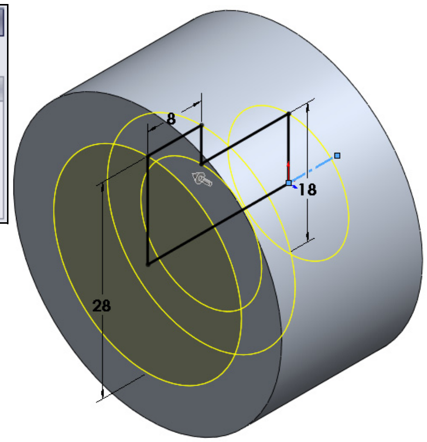
2 Reorient the view to the Right view.

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

6 Choose **One-Direction** and **360deg**.

Click .

7 Rename this feature `Bearing Cut`.

8 **Save** this part as `Wheel Hub`.

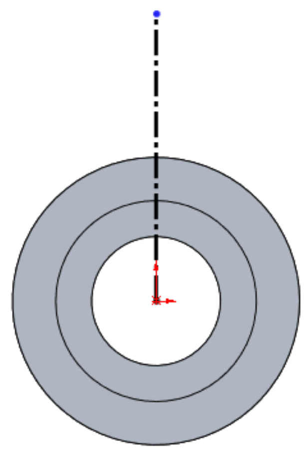
Patterns

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.



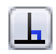
1 Create a sketch on the Front plane and change the view orientation to the Front View.

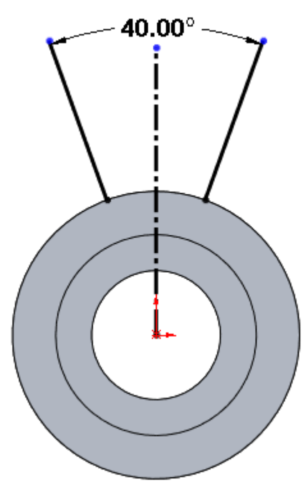
2 Sketch a vertical centerline from the Origin.

3 Click **Dynamic Mirror Entities**  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**  on the **Sketch** toolbar to add a relationship. Select one of the lines and the Origin. Click  to add a **Coincident** relationship.



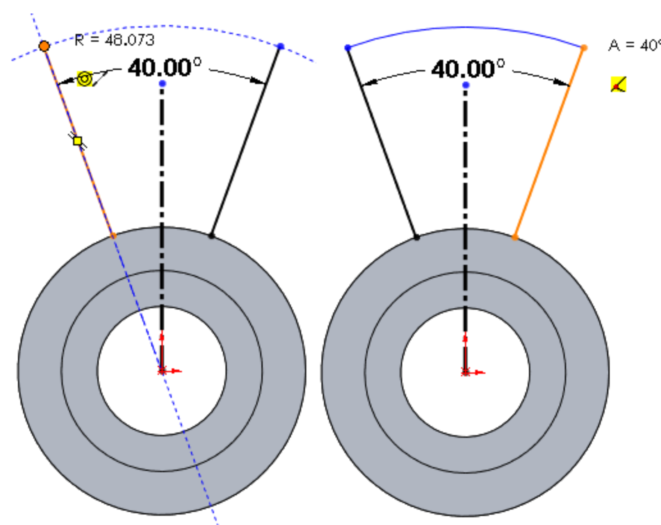
7 Add an angular dimension. Click **Smart Dimension** , then select the two lines.

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. Select **Centerpoint Arc**  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.

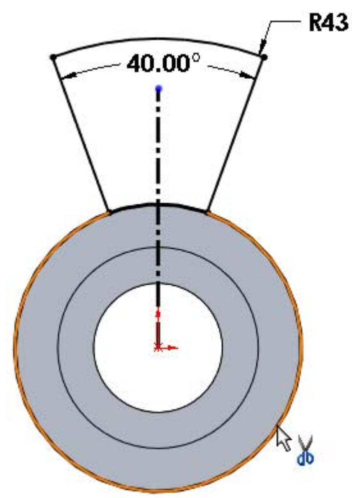
   Click .

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

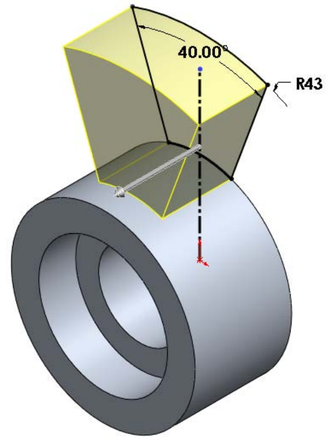
2 To trim the circle, select **Trim Entities** from the Sketch toolbar.

3 Select **Trim to Closest** in the Property manager. The cursor will change to the Trim Cursor .

4 Click the part of the circle to be trimmed away.



Task 6 — Extrude the Spoke



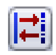
1. Extrude the sketch. **Extrude** to a **Blind** depth of **15mm**.
2. Rename this feature `Spoke`.
3. **Save** the `Wheel Hub`.

Construction Geometry

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

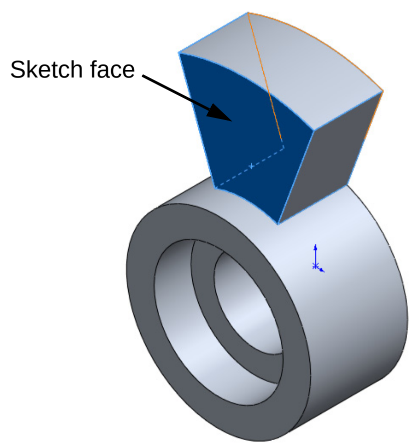
Any piece of sketch geometry can be converted into construction geometry or vice-versa. Construction geometry is considered to be reference geometry and does not have to be fully defined.

To convert sketch geometry into construction geometry:

- ❑ Select the geometry, then click  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.

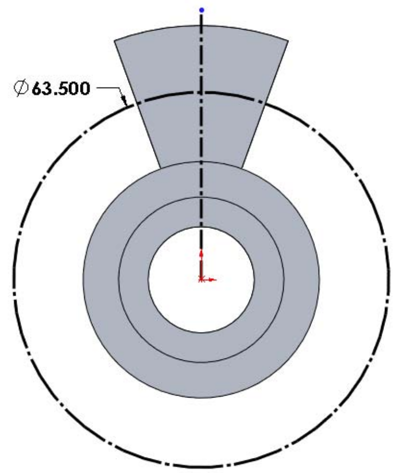


1. Create a sketch on the face shown.
2. Reorient to the `Front` view.
3. Sketch a circle with its center at the **Origin**.
4. Dimension the circle to diameter **63.5mm**.

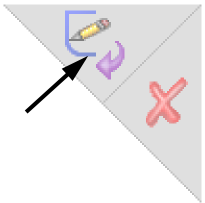
5. Select the circle, then in the PropertyManager select **For construction**.
6. The circle will turn into a construction circle.

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

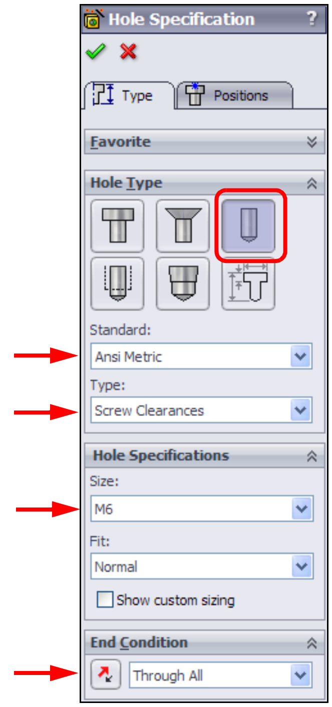
The **Hole Wizard** is used to create specialized holes in a solid. It can create simple, tapered, counterbored and countersunk holes using a step by step procedure.

1 Select the face of the spoke.

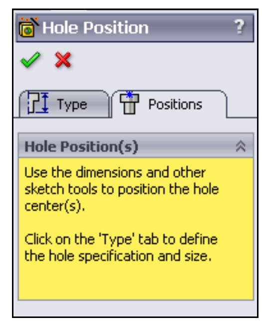
2 Click **Insert, Feature, Hole, Wizard** from the menu. This starts the **Hole Wizard**.

3 Click the **Hole** button. Using the pull-down lists, select:

- Standard: **Ansi Metric**
- Screw type: **Screw Clearances**
- Size: **M6**
- End Condition: **Through All**

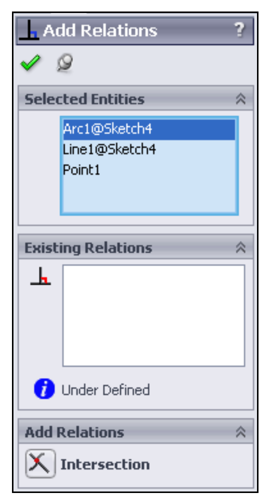


4 Click on the **Positions** tab. You can now edit the sketch that determines the hole position(s). The message tells you to locate the hole center(s).

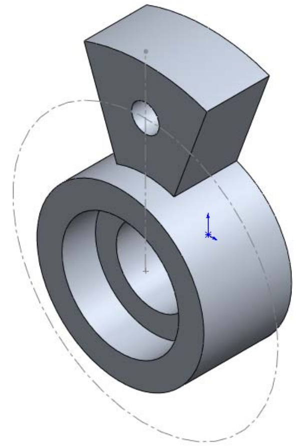


5 Turn off the **Point** tool by clicking it once on the Sketch toolbar. The sketch entity **Point** is automatically turned on so that you can place several holes. We are only placing one hole so we can turn off the tool.

6 Position the hole. Click to add a relationship. Select the Point, Construction Line, and Construction Circle. There will only be one relationship available, **Intersection**. Click then .



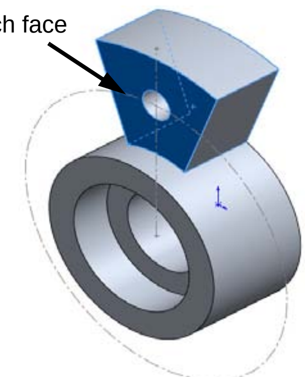
7 Complete the wizard. Click . The correct size clearance hole for an M6 metric bolt is created.



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**  on the **Standard Views** toolbar to change the view so that we are looking normal to the face.

Zoom to Selection

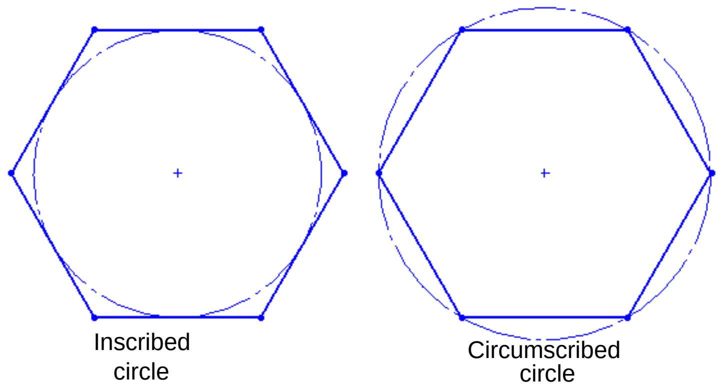
Zoom to Selection will zoom the view to whatever entity is selected. The selected item will fill the screen but have a clear area around it.

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

Polygon Tool

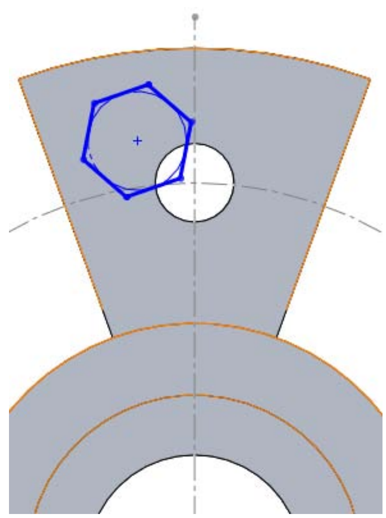
Regular (all sides equal) polygons can be sketched with the **Polygon** tool. All regular polygons are based on a construction circle. The polygon is defined by the circle being inscribed or circumscribed about the polygon.

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

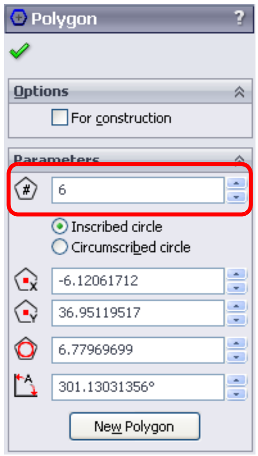


1 Select the **Polygon** tool  from the **Sketch** toolbar or select **Tools, Sketch Entities, Polygon** from the menu.

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.

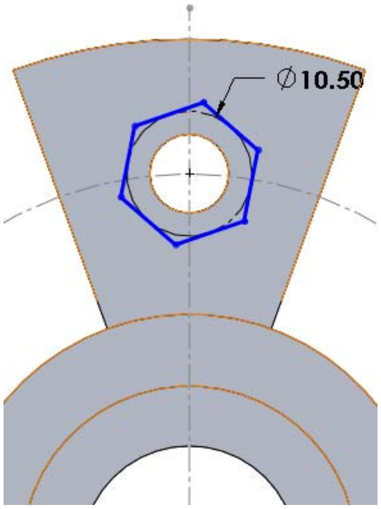


3 The default for polygons is 6 sides. If your polygon has a different number of sides, it can easily be changed in the PropertyManager.



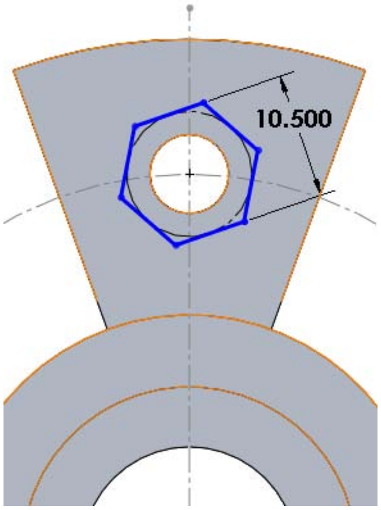
4 Add a **Concentric** relationship between the construction circle and the circular edge of the hole.

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.

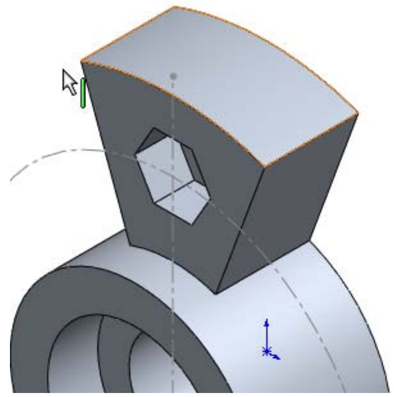
We could have also dimensioned the hex hole from flat to flat. While this is satisfactory for SolidWorks, it doesn't represent the design intent as clearly as dimensioning the circle diameter.



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 as .



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 

Sketch hidden 

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.

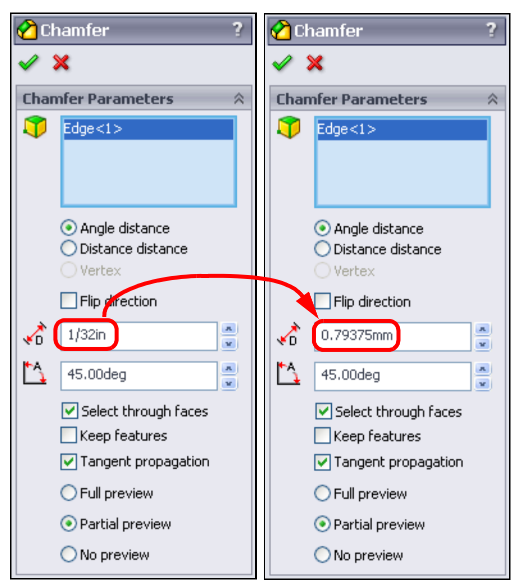
Task 10 — Add a Chamfer

1 Select the **Chamfer** tool  from the **Features** toolbar.

2 Select **Angle distance**.

3 Select one of the edges of the hex cut.

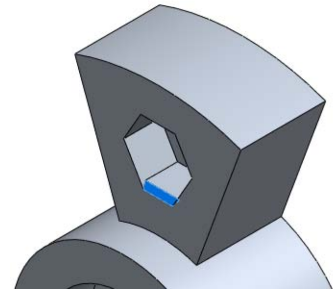
4 Set the **Chamfer** dimension. We want a 1/32 inch chamfer. Rather than divide 32 into 1 to get the decimal equivalent and then convert it to millimeters, just type **1/32in**. When you press the **Enter** key, the 1/32 inches will be calculated and displayed as millimeters.



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.



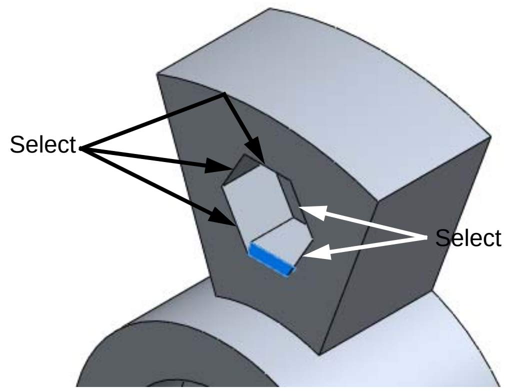
Edit Feature

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

1 Right-click the feature `Chamfer1` in the FeatureManager design tree and select **Edit Feature** . The Chamfer PropertyManager will open.

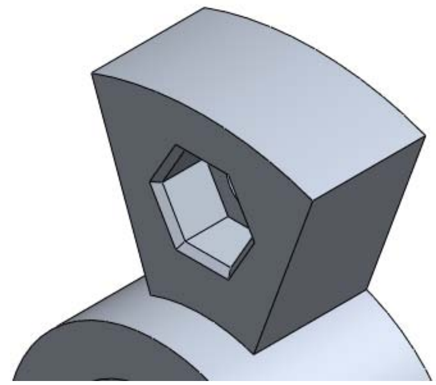
2 Select the remaining five edges of the `Hex-Hole`.



3 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

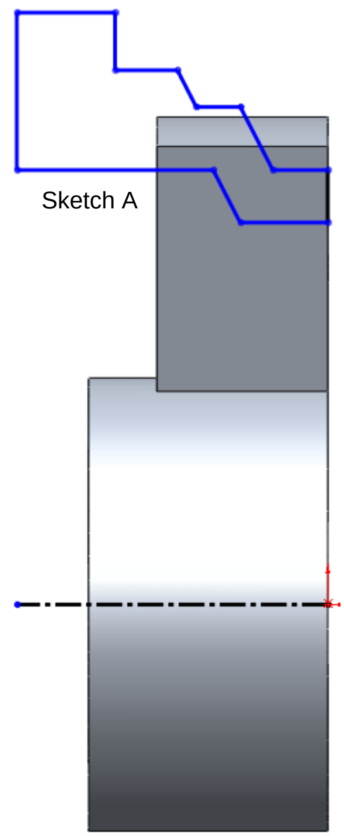
The wheel rim will be constructed as another revolved feature.

1 Reorient to the Right view.

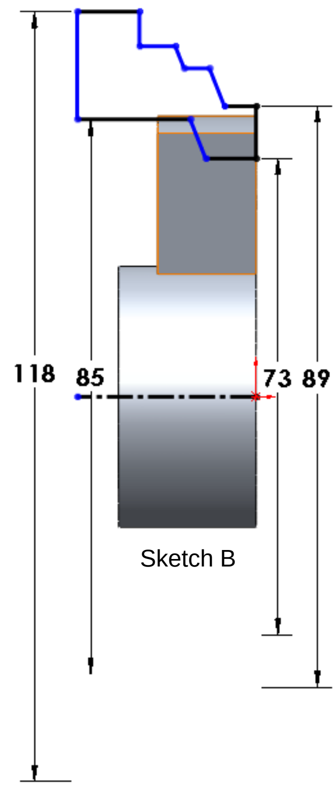
2 Create a sketch on the Right plane.

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.

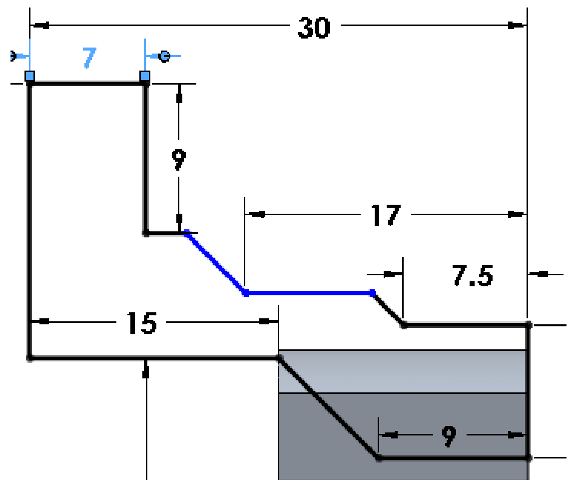


5 Add the dimensions shown in Sketch B.Remember, to dimension the four diameters, the centerline must be selected as one end of the dimension



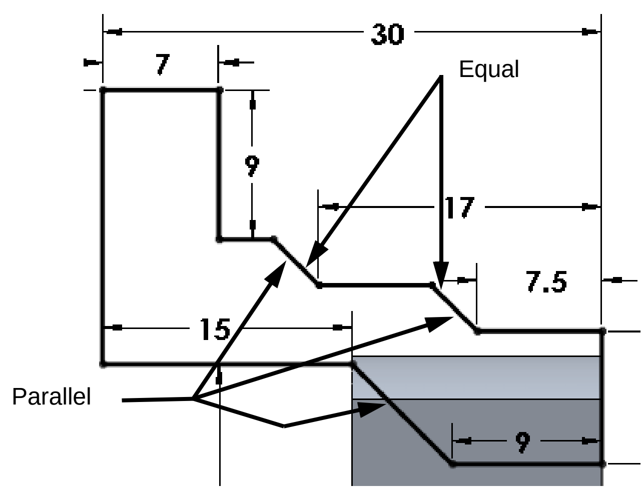
Sketch B

6 Add the following dimensions.



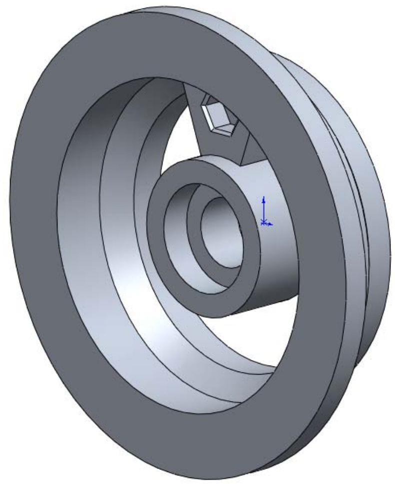
7 Add the following sketch relationships to fully define the sketch.

8 Revolve the rim. Click **Revolved Boss/Base** on the Features toolbar.



9 Revolve **One-Direction**, **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:

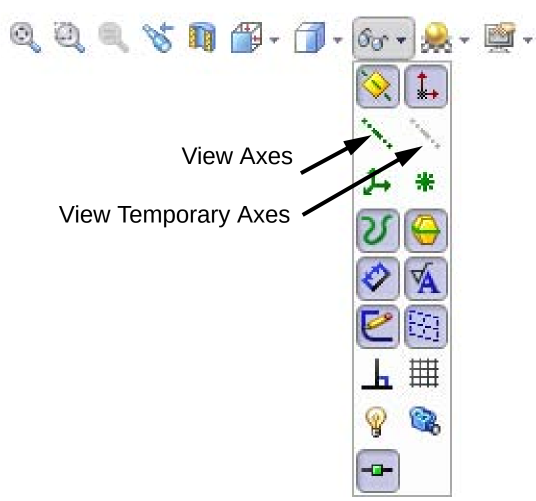
- Temporary Axes
  These are created by SolidWorks any time a cylindrical or conical solid is created.
- Axes
  These are created manually by the user.

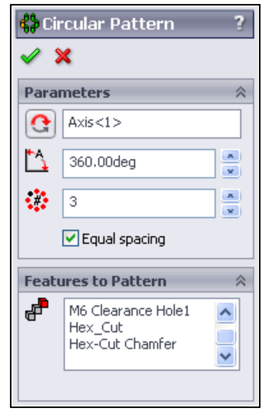
Viewing Axes

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.

1 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.

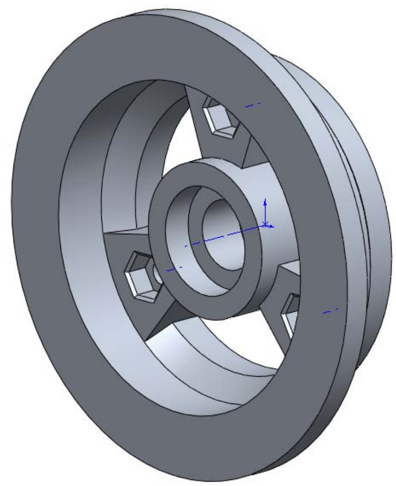
5 Click the **Circular Pattern** title at the top of the PropertyManager to fly-out the FeatureManager design tree.

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.

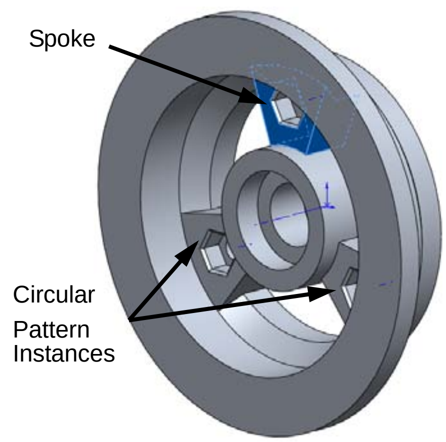
9 Click . The spoke, along with the two holes and the chamfer have been patterned.



Task 14 — Add Fillets

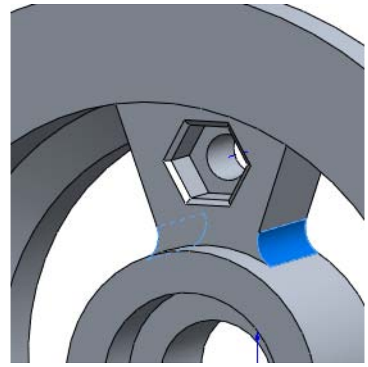
Fillets need to be added to the spokes to round all the edges. Rather than add the fillets to all three spokes individually, we will just add the fillets to the first spoke, then include the fillets into the pattern.

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.



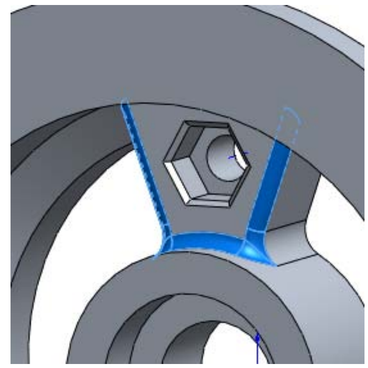
2 Add a **4mm** fillet to both sides of the spoke where it contacts the hub.

3 Rename this feature **Fillet R4**.



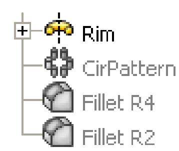
4 Add a **2mm** fillet to the three edges shown.

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** .

Reordering Features

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

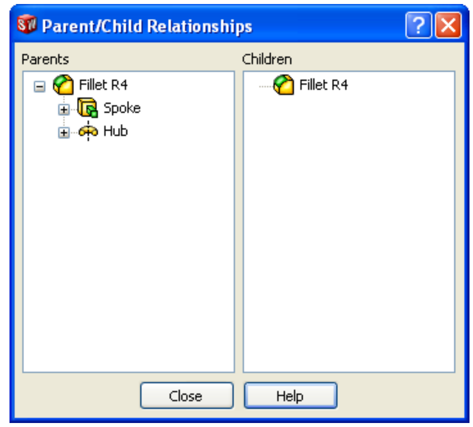
When dragging a feature, the cursor will change to  indicating that the new location will be under the feature highlighted. If you drag the feature too far up the FeatureManager design tree, the cursor will change to , indicating that you cannot drop the feature at this location.

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.

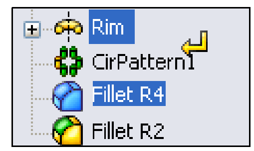
1 In the FeatureManager design tree, right-click Fillet R4 and select **Parent/Child** from the menu.



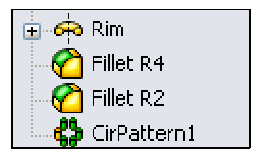
The fillet has two parents, the Spoke and the Hub. This is logical as the fillet connect the two features. If we move the fillet up the FeatureManager design tree, it can never go before either the Spoke or the Hub.

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.



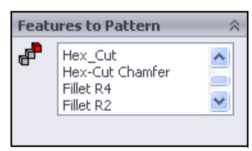
3 Drag the feature Fillet R2 to a position after Fillet R4.



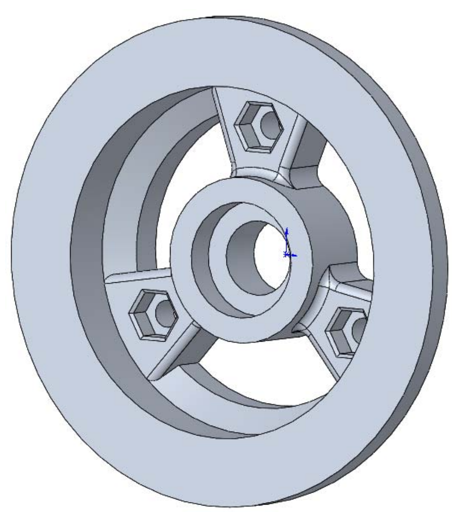
Both fillets now exist before the circular pattern so they can be included in the pattern.

4 In the FeatureManager design tree, right-click CirPattern1 and select **Edit Feature**.

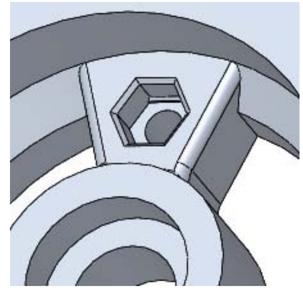
5 Select Fillet R4 and Fillet R2 to add them to the **Features to Pattern**.



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

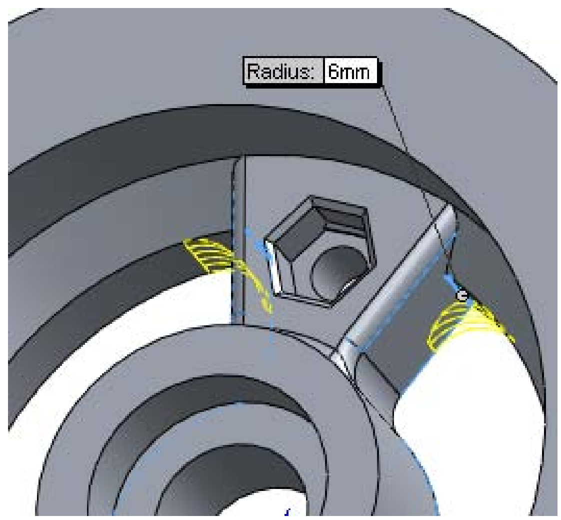


The spoke to rim fillets provide an additional challenge because of the way the two features meet. We need a relatively large fillet to make the wheel look good.

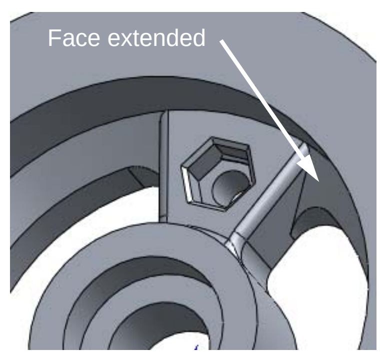
Because the spokes were patterned before the rim was created, fillets between the two cannot be added to the pattern. We can add all the fillets to the three spokes individually, but this is a lot of work. We might try to fillet one spoke and pattern the fillets, however this doesn't work. Fillets need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

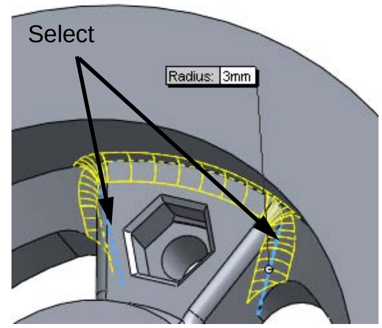


7 Add a **3mm** fillet to the two edges show.

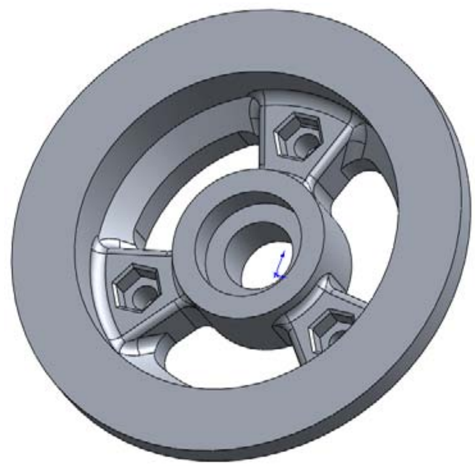
8 Rename the fillet to **Fillet R3**.

9 Roll the model forward to the end. Right-click in the FeatureManager design tree and select **Roll to End**.

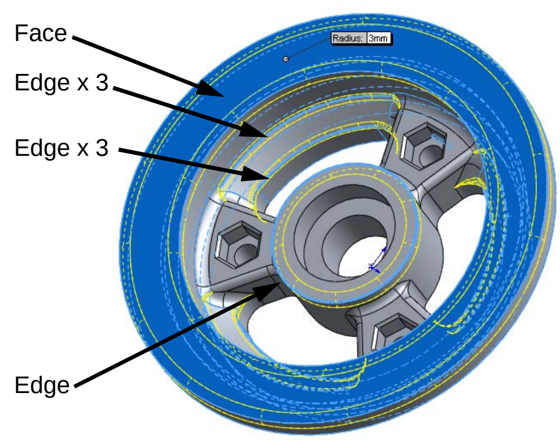
10 Add the two fillets to the circular pattern. Right-click `CirPattern1` and select **Edit Feature**. Select `Fillet R6` and `Fillet R3` to add them to the **Features to Pattern**.



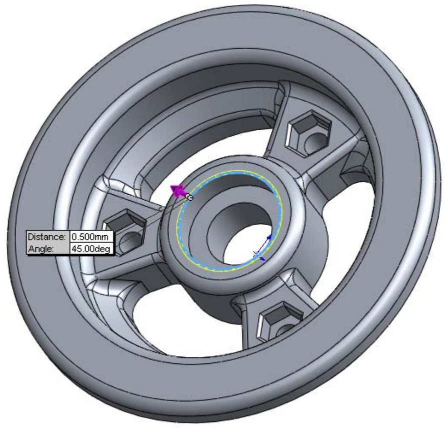
11 Click . All three spokes should now have the same fillets.

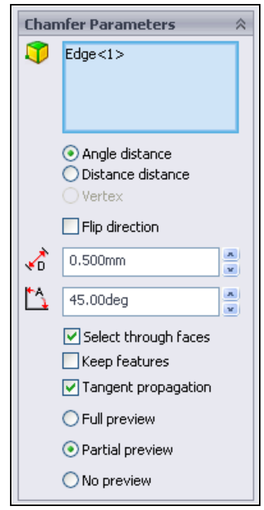


12 Add additional fillets to round remaining sharp edges. Add a 3 mm fillet. Select the face and edges shown. When you select a face, all edges of that face will be filleted.



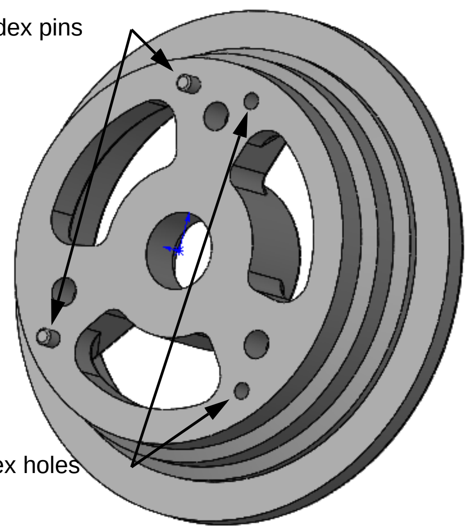
13 Finish the front by adding a 0.5 mm by 45 degree chamfer to edge of the hole where the bearing will be inserted.





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.



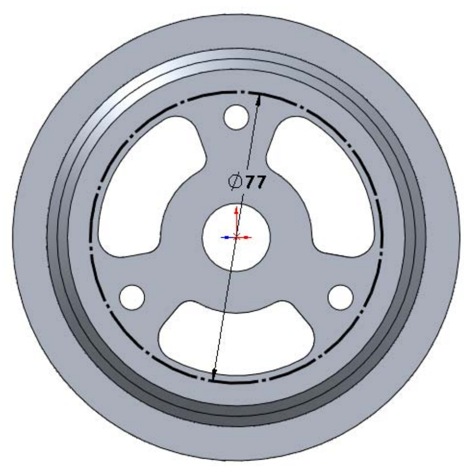
Shared Sketches

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.

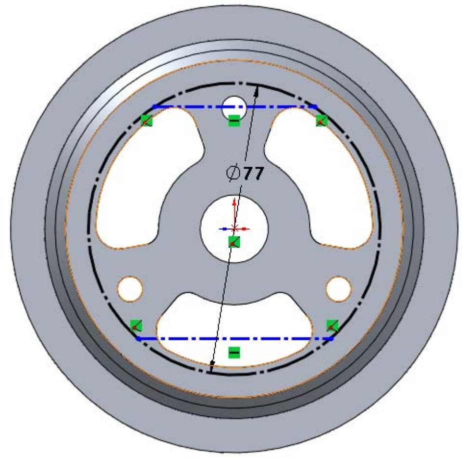
3 Sketch a circle with its center on the Origin. Dimension the circle to **77mm**.



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.

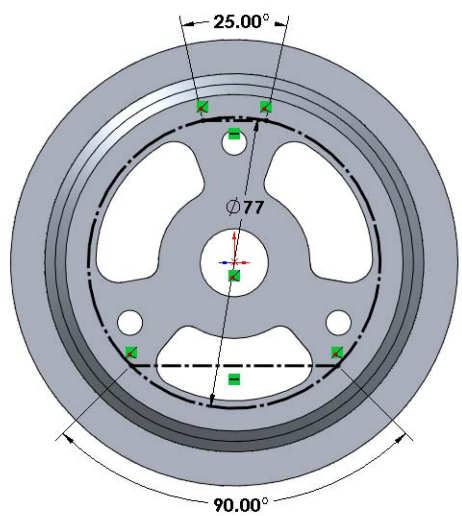
In the PropertyManager for the circle, select **For Construction**. The circle now changes to a construction line.

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.

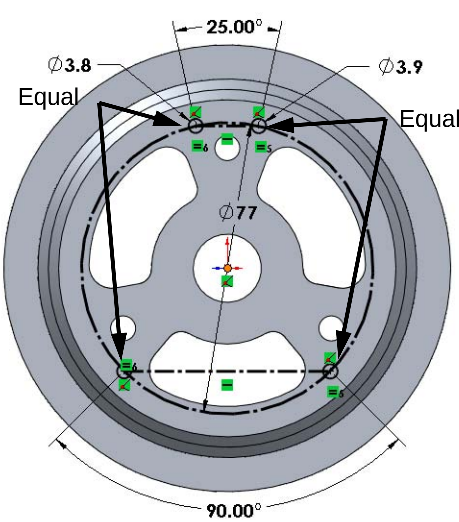


7 Sketch a circle at the endpoint of each line.

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.



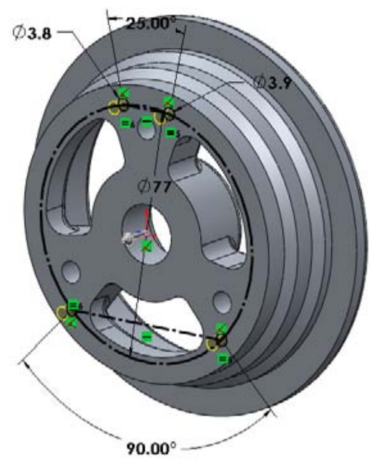
Sketch Contours

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** 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 , 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** .

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

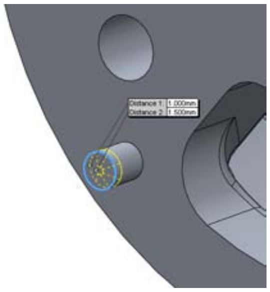
In the PropertyManager, click the down arrow next to **Select Contours** to expand the selection box. Click the two right circles to select their contours. Only these two circles will now be cut, nothing else.

Click .

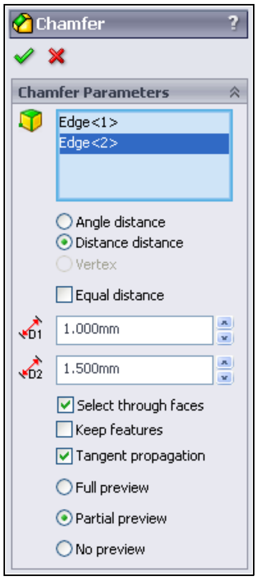
14 Rename the feature `Index Holes`.

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.

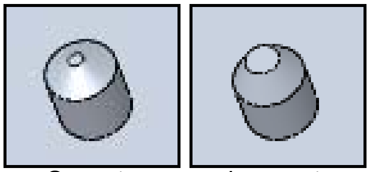
15 Add a **Chamfer** to the top of the `Index Pins`. You generally don't leave sharp edges on a small feature like the pin as the edge is likely to be damaged which would cause problems when assembling the parts.



Click **Chamfer** 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.



Correct Incorrect

Click .

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.

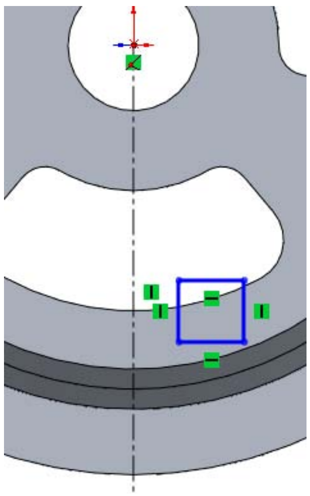
We will create a revolve cut opposite one of the spokes.

1 Change to the `Back` view by clicking **Back** on the Standard Views toolbar.

2 Create a sketch on the back face of the model.

3 Sketch a vertical centerline from the `Origin`.

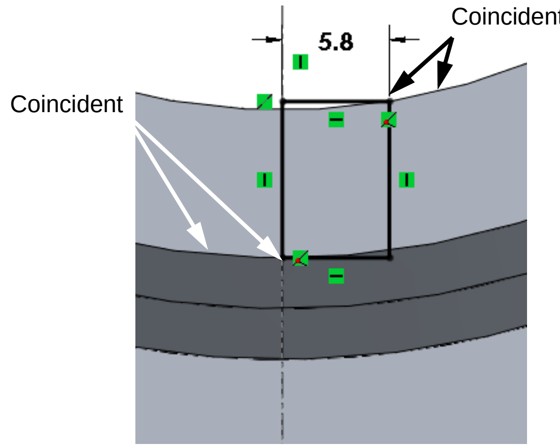
4 Sketch a rectangle roughly in the position shown.



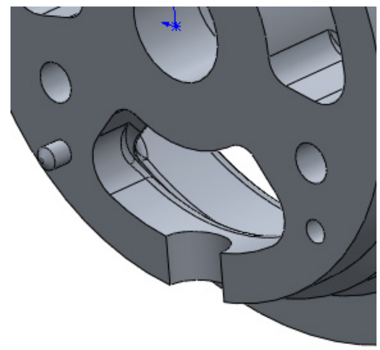
5 Add two **Coincident** relationships as shown. This will insure that we cut all the material in the rim.

6 Add a **Collinear** relationship between the left side of the rectangle and the centerline.

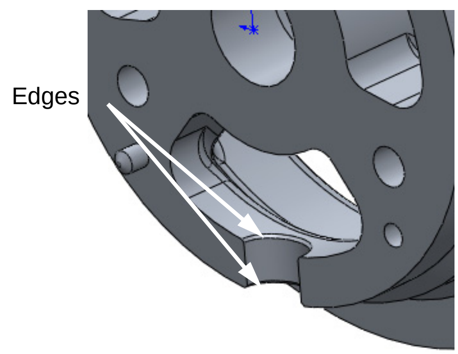
7 Dimension the width of the rectangle to **5.8mm**.



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.



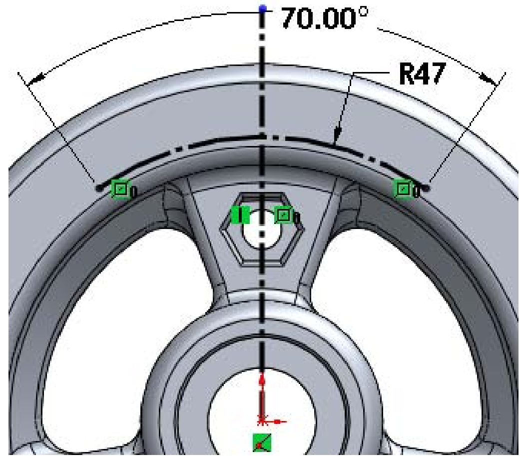
9 Add a **0.5mm** by **45deg** chamfer to the edges of the cut as shown.



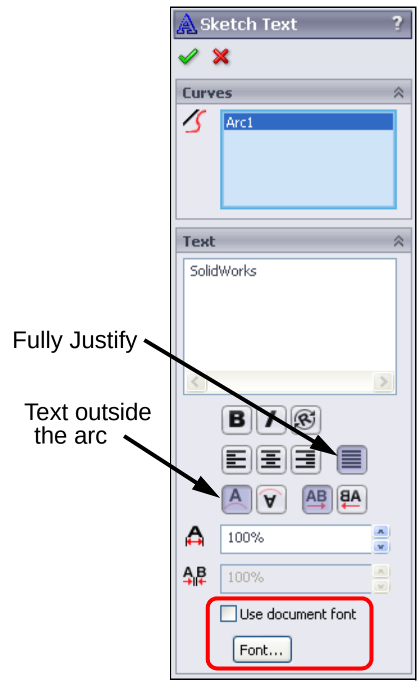
Task 19 — Add Lettering

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.

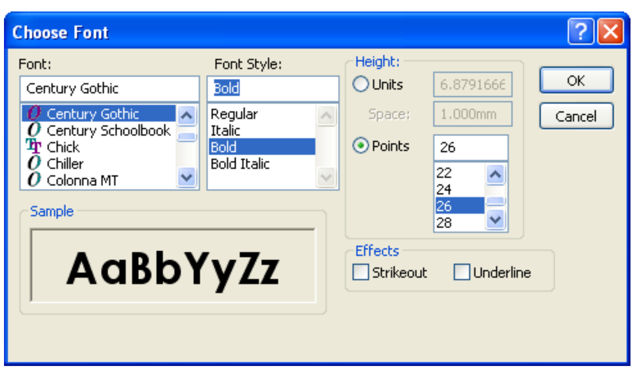
Because we want the letters to follow a curved path, we will create some construction geometry to guide the letter placement.



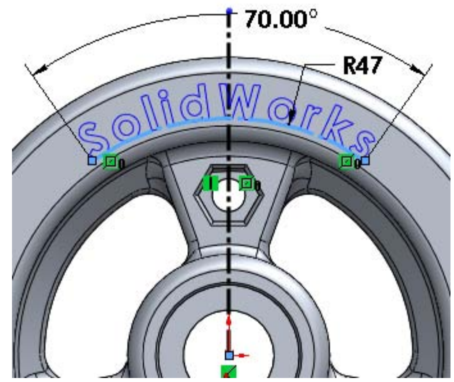
1 Change to the Front view by clicking **Front** .
2 Create a sketch on the front face of the rim.
3 Sketch a vertical centerline from the Origin.
4 Sketch a centerpoint arc with the center on the Origin. Make the arc wider than the spoke and change it to construction geometry.
5 Add a **Symmetric**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



13 We want to cut the text into the surface. Click **Extruded Cut**. Cut the letters to a blind depth of **1mm**.

14 Rename the feature **Text**.

Appearances

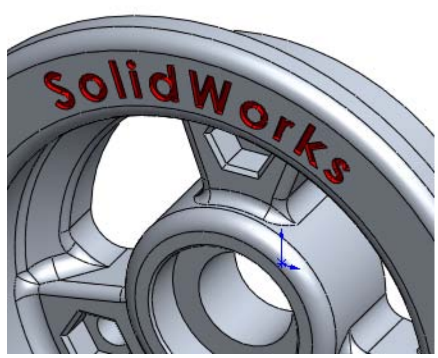
Appearances control the way the surfaces of the model look. They can be applied at the assembly, part, body, feature or face of a model.

To add an appearance:

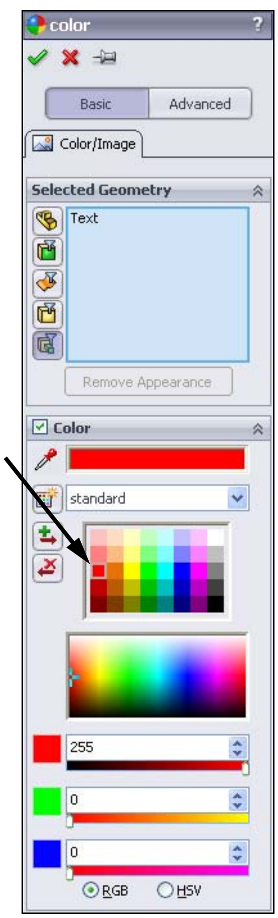
- ❑ Click **Edit, Appearance, Appearance** from the menu.
- ❑ Click on the **Standard** toolbar.
- ❑ Select the Appearances tab in the Task Pane.

1 Add color to the `Text` feature. Select the `Text` feature in the FeatureManager design tree. Click **Edit, Appearance, Appearance** from the menu.

The default appearance is called “color” which is suitable for our needs right now. Select the color **Red**. Click . The geometry of the `Text` feature is now red. This gives the impression that the cut letters have been painted red.

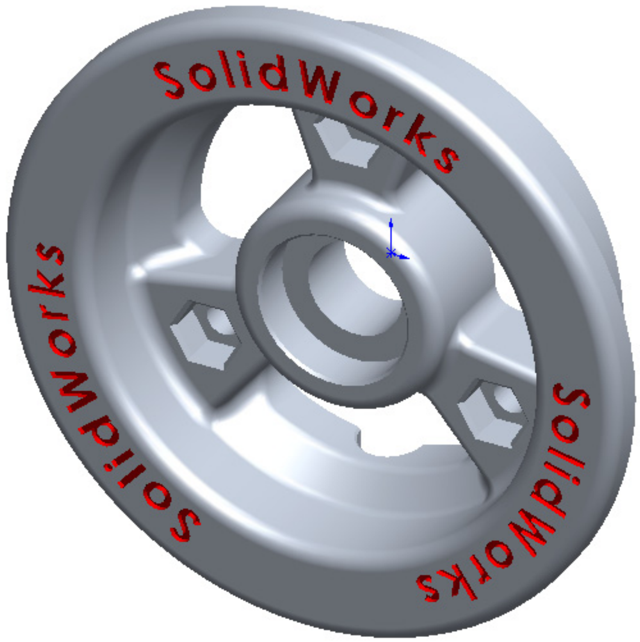


Red



2 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.

3 **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 and Rhino.

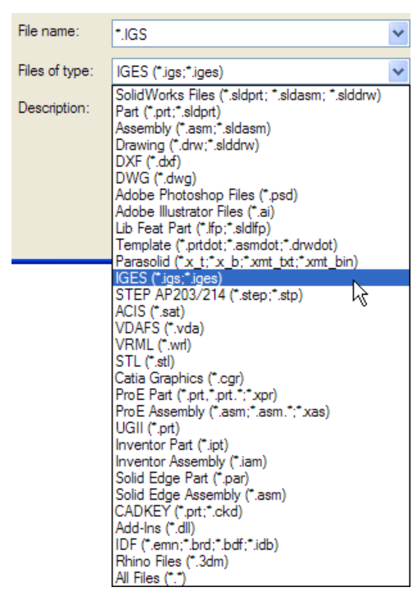
Translation through neutral file formats requires the file created in another program to be saved as a neutral format. None of the existing CAD programs use the neutral formats directly. The neutral formats only provide a “common ground” that both programs can use. SolidWorks can then read the neutral 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.



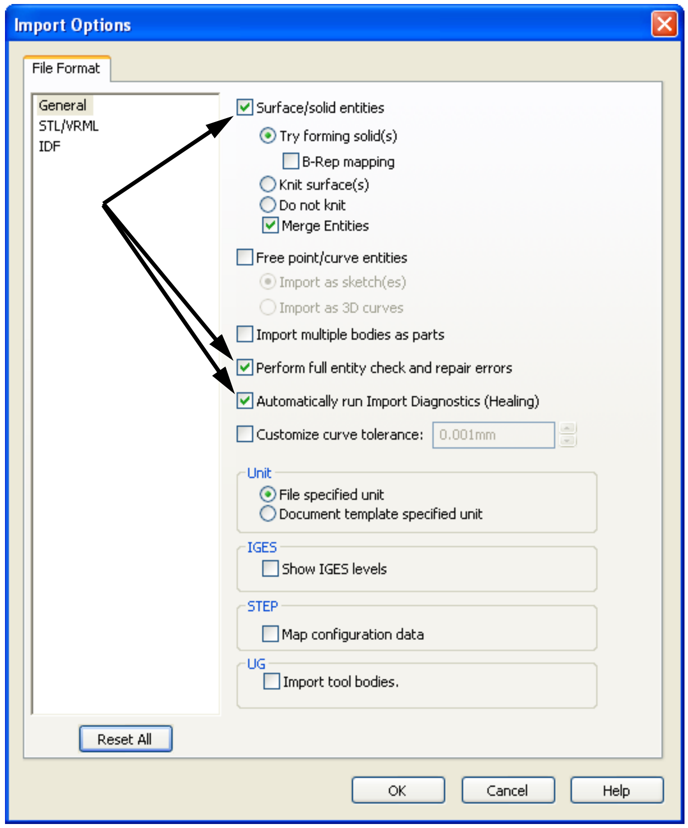
Examine the list to see the other files types that can be opened in SolidWorks.

3 Select the file `Tire.igs` from the `Lesson04` 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.

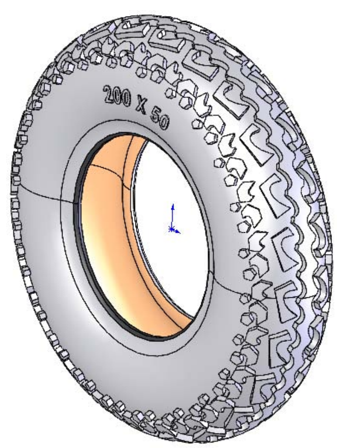


5 Set the options. Select:

- ❑ **Surface/solid entities**,
- ❑ **Try forming solid(s)**,
- ❑ **Perform full entity check and repair errors**
- ❑ **Automatically run Import Diagnostics**.

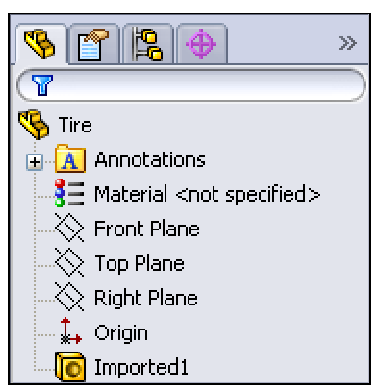
These options tell SolidWorks to make the import entities into a solid model if it can and if there are errors, use the tools it has to fix any errors it detects.

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 .

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 change anything about `Imported1`.



9 **Save** the Tire as a SolidWorks part to the `Mountainboard\Wheel Assembly` folder.

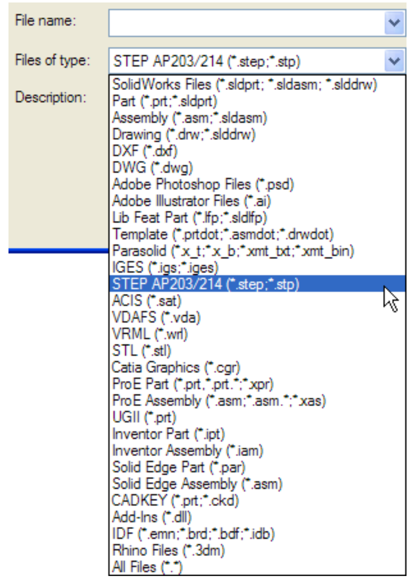
10 **Close** the Tire part.

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.

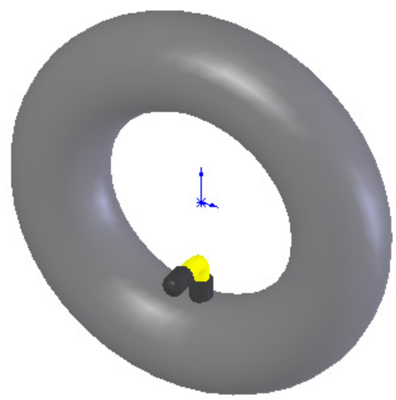
2 Select **STEP AP203/214 (*.step,*.stp)** from the **Files of type** list.



3 Select the file Inner Tube.STEP from the Lesson04 folder.

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_2010\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.

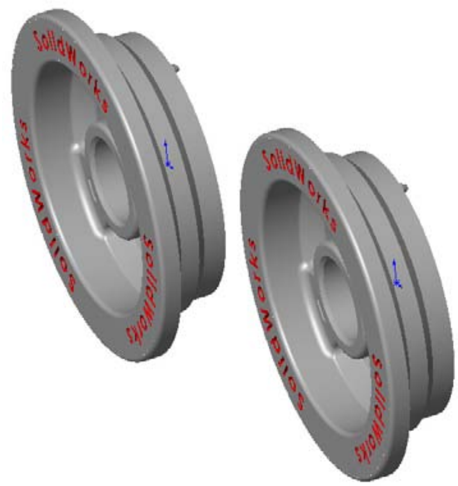
3 Insert the first part. Click the **Browse** button in the PropertyManager. Locate the part Wheel Hub and click **Open**. The wheel hub will appear transparent, drag it to the assembly Origin. When the cursor changes to  release the mouse button.

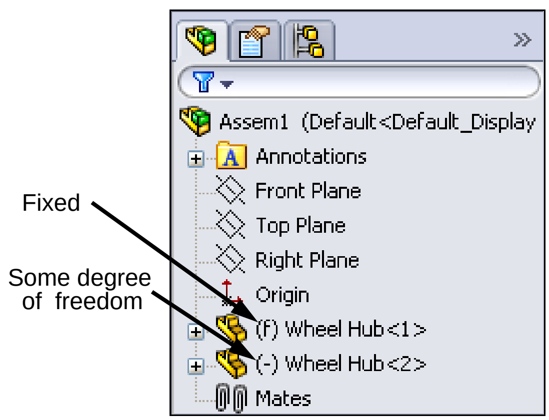
4 The Wheel Hub is now fixed in space with its Origin mated to the assembly Origin and its three principal planes mated to the three planes of the assembly.

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.

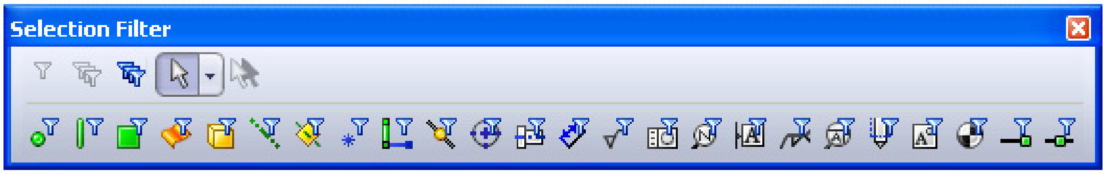
These mates use the same **Mate Pop-up** Toolbar as the **Mate** tool uses to set the type and other attributes. All mate types can be created with this method.

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.

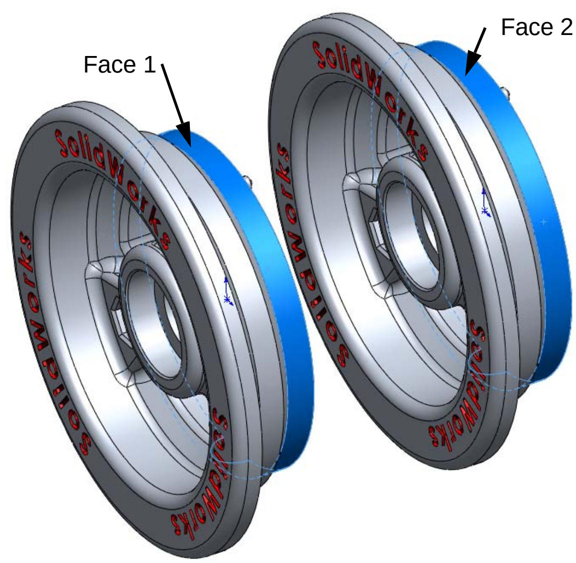


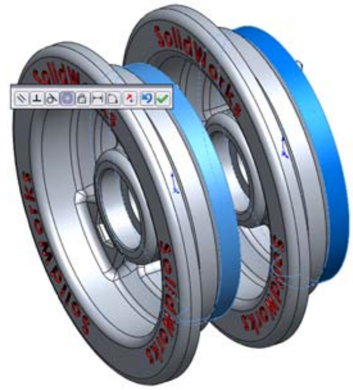
7 Most of the mating entities we will be choosing are faces. To make it easier to select faces, we will turn on a filter so that only faces can be selected. Click **Toggle Selection Filters Toolbars** on the Standard toolbar to open the Filter toolbar. Then click **Filter Faces** .

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 . When the cursor is over a face, the cursor will change to .

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 , indicating a concentric mate, release the **Alt** button.



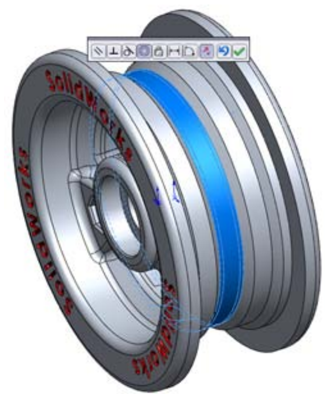


9 Reverse the alignment. The two cylindrical faces can be concentric in two orientations. The initial orientation is based on which of the two possibilities is closest. Press the **Tab** key. This will toggle the two possible alignments.

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.

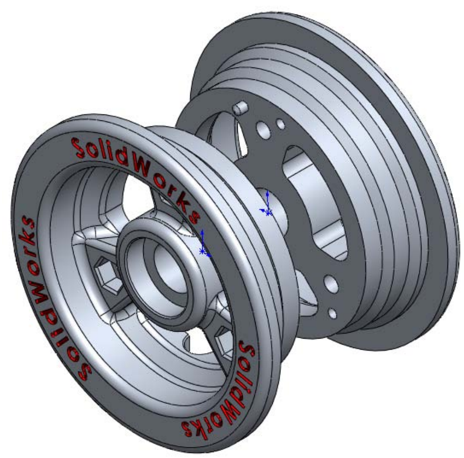
11 Click  to accept the mate.



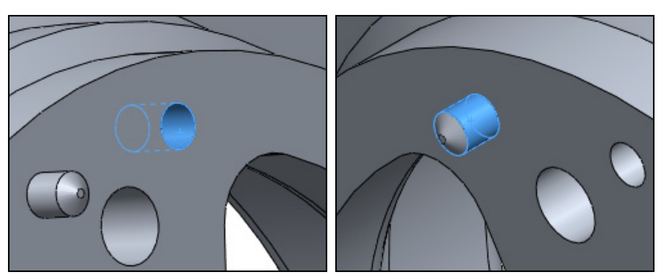
12 Move the second Wheel Hub. We need three mates to properly position the second Wheel Hub. First is the **Concentric** mate already added. Second is a **Concentric** mate between an Index Pin and the corresponding Index Hole. Finally a **Coincident** mate between the inside faces of the two Wheel Hubs 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**.

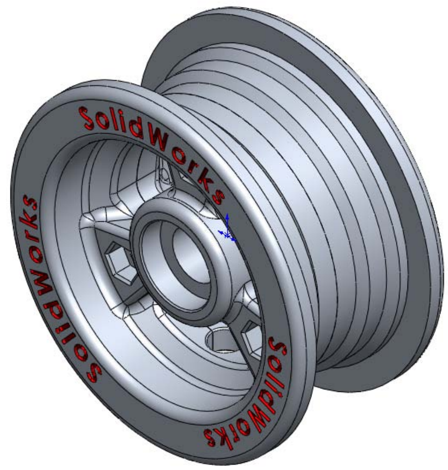


Note: Check to make sure that the hole for the Tube Stem lines up on the two Wheel Hubs. If it does not, edit the last mate and change the mate to the correct hole.

14 Mate the inside faces of the two Wheel Hubs together with a **Coincident** mate. Click 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

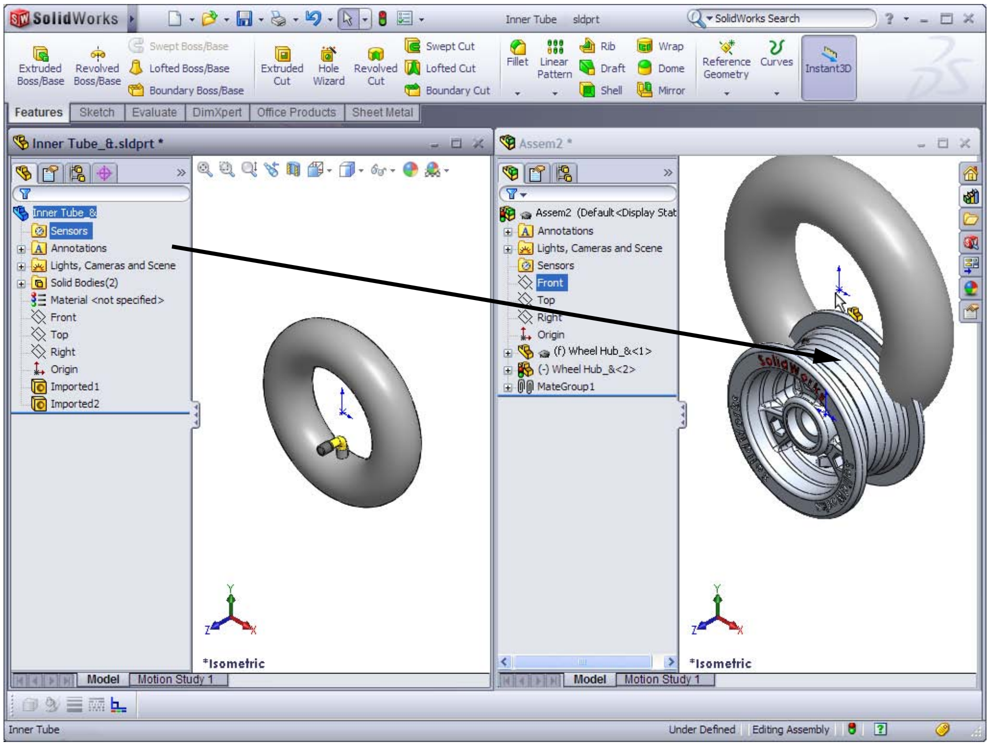
Part can be added to an assembly in several ways:

- 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.

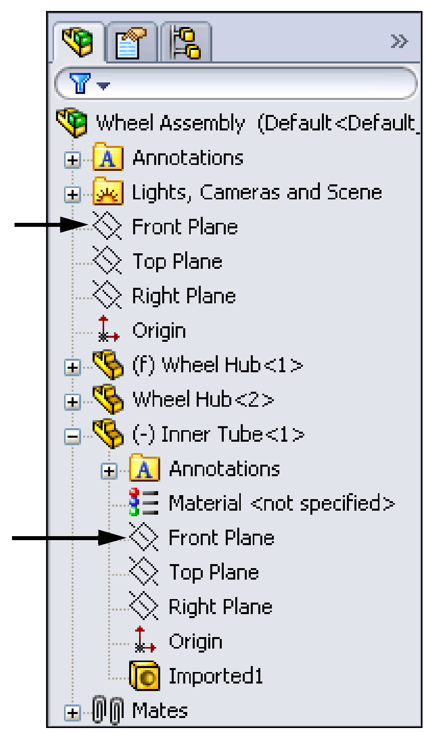


4 In the FeatureManager design tree, click the plus sign next to the part Inner Tube to expand the listing.

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`.

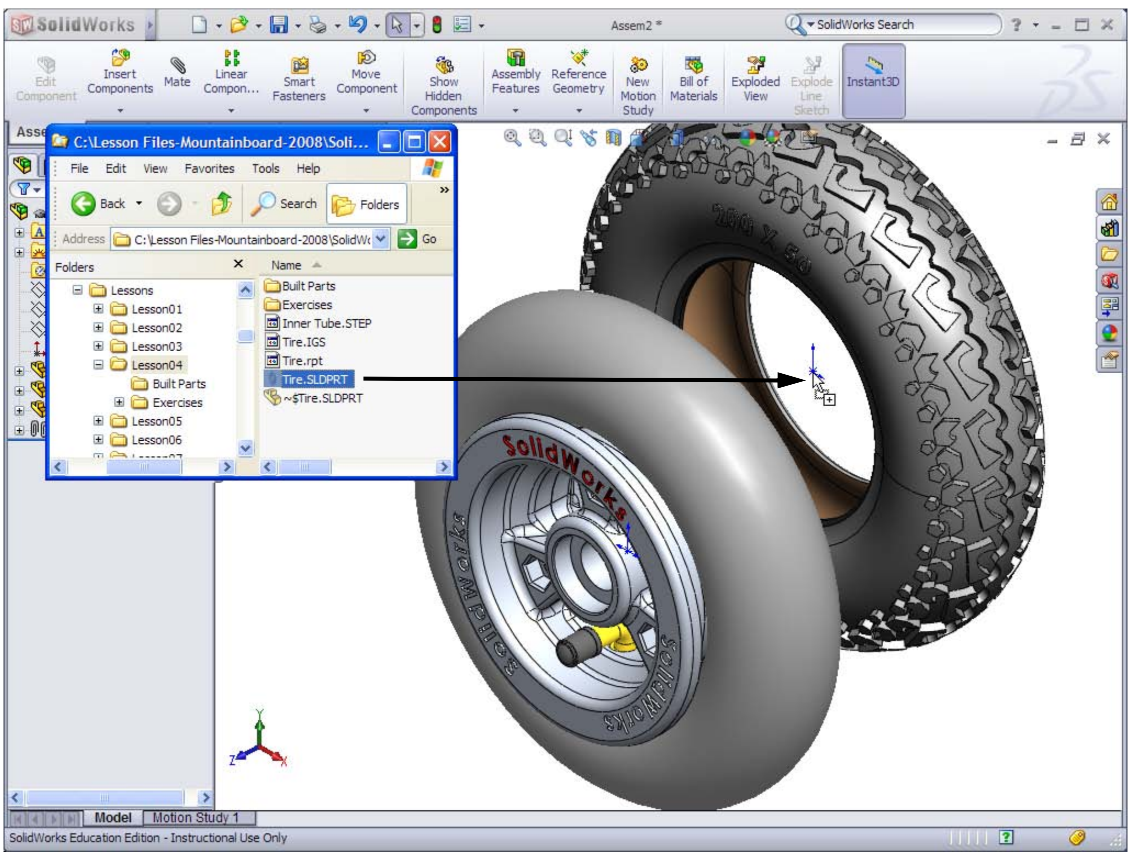
Task 3 — Add the Tire

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 don't want to have to create standard fasteners, we would rather just use pre-made models.

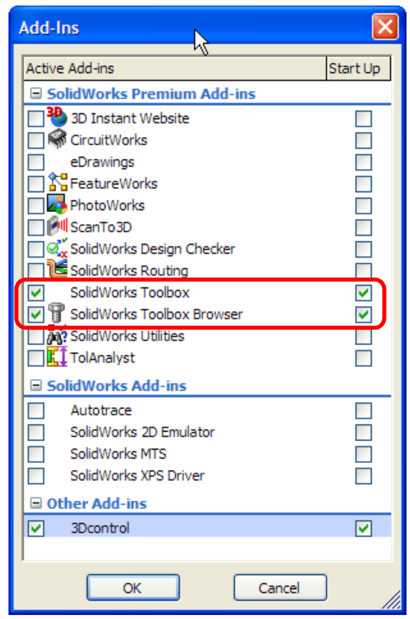
Toolbox

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**.

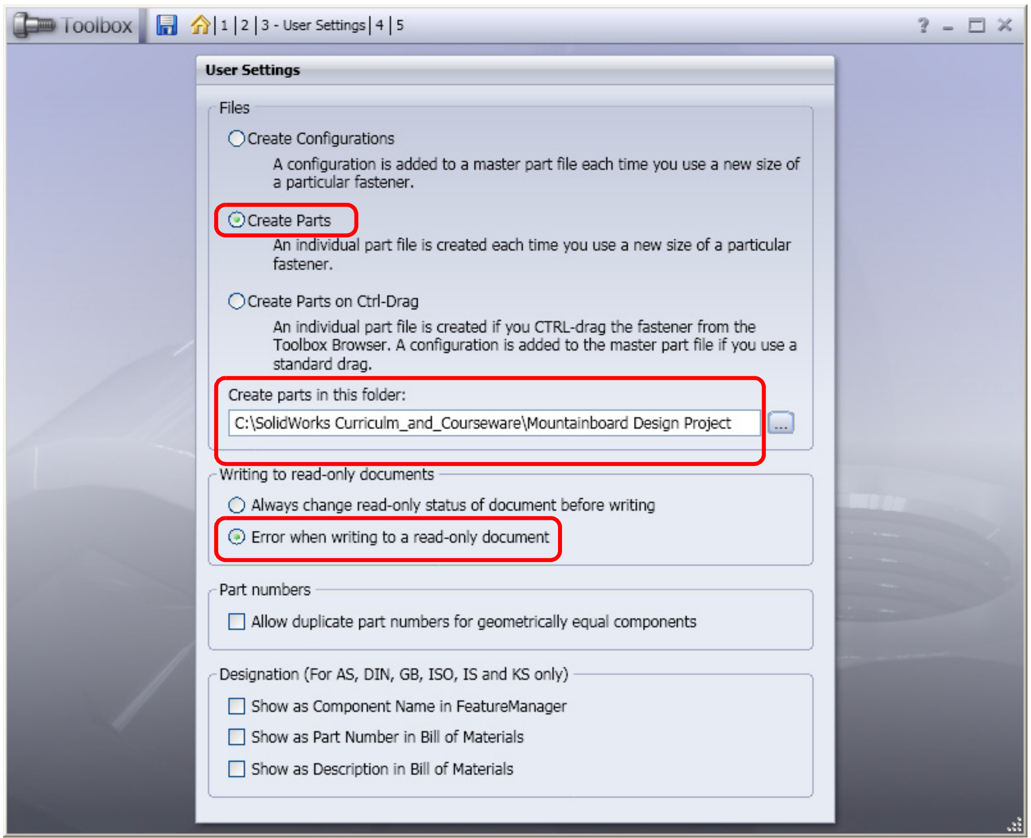
   Toolbox will be added to the Design Library.
3. Click **Toolbox, Configure** from the menu.



4 Select **Define user settings.**

Select **Create Parts**. Click **Browse** then navigate to the SolidWorks Curriculum and Courseware_2010\Mountainboard Design Project \Mountainboard\Hardware folder.

Select **Error when writing to a read-only document**.



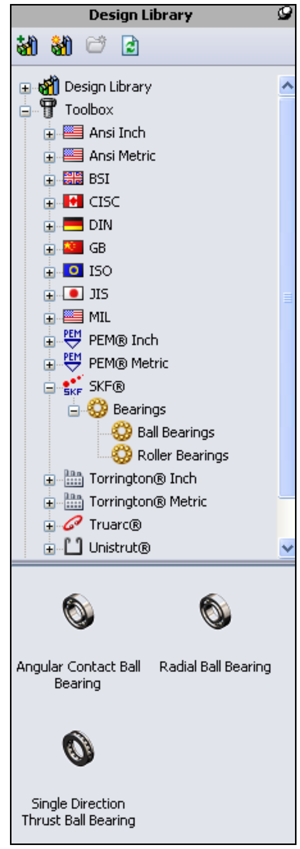
Click **Save**, then **Close**.

5 Open Toolbox Browser

Expand **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.

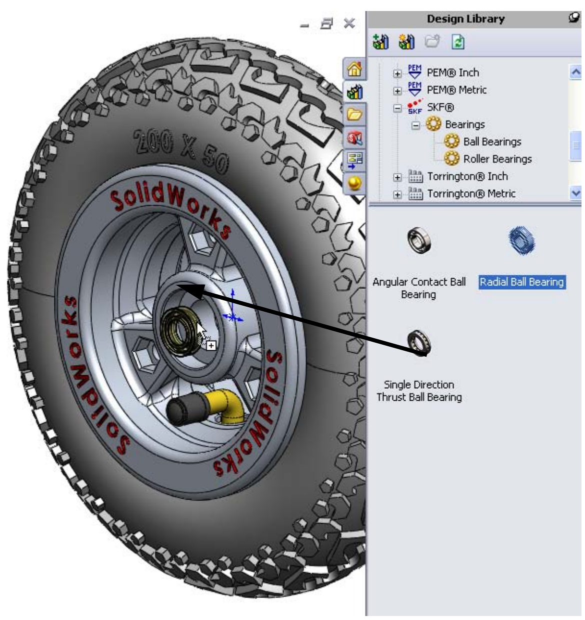


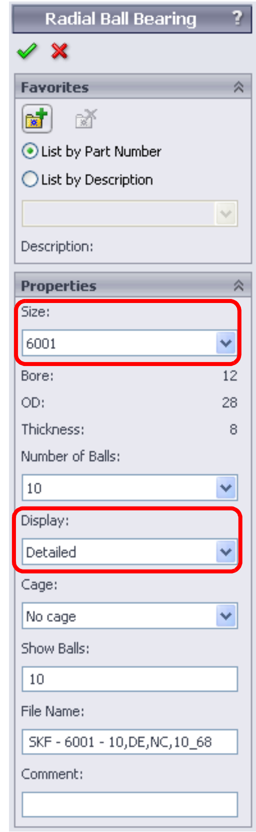
6 Click the pushpin to keep the Design Library open as we select other things.

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 12 mm and an Outside diameter of 28 mm 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** 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

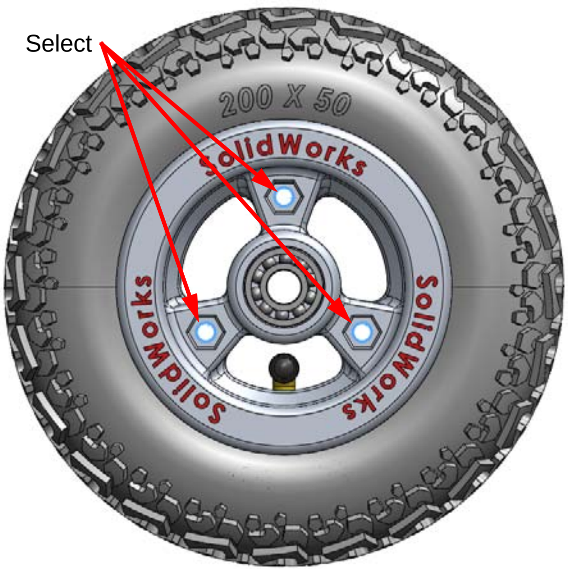
Toolbox parts can be added to multiple locations at the same time. Instead of dragging the part from the Toolbox Browser, the mating location or locations are selected first. To insert the Toolbox item, right-click the part in the Toolbox Browser and select **Insert into assembly**.

1 Change the view orientation to the Front view.

2 Select the **Ansi Metric**, **Bolts and Screws**, **Socket Head Screws** folder under Toolbox.

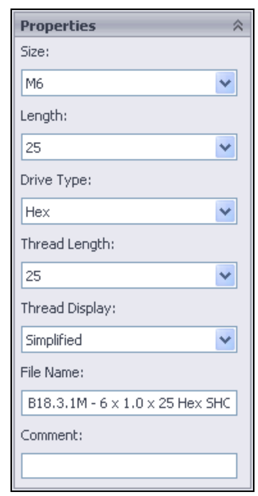
3 Press and hold **Control** and select the three edges of the bolt holes.

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.

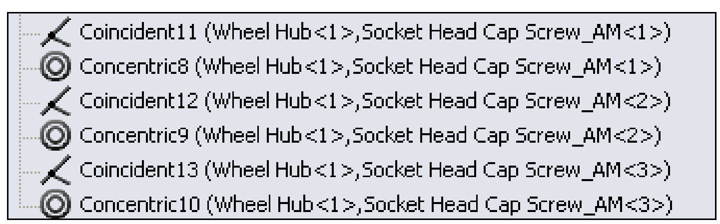
- Cosmetic — Represents some details of the hardware. Cosmetic display shows the barrel of the bolt or screw and represents the size of the threads as dashed lines.

- 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.



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.

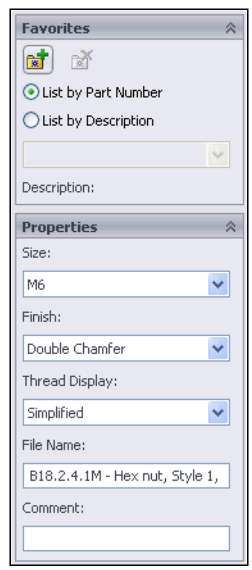
1 Change the view orientation to the Back view.

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**.

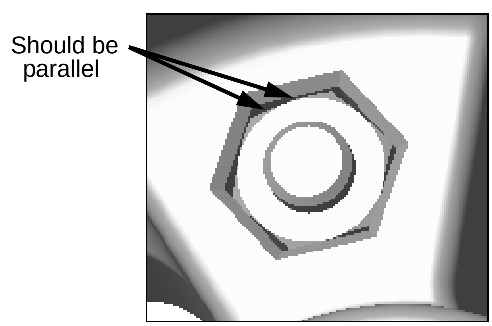


5 Select the sizes shown, then click .

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.



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

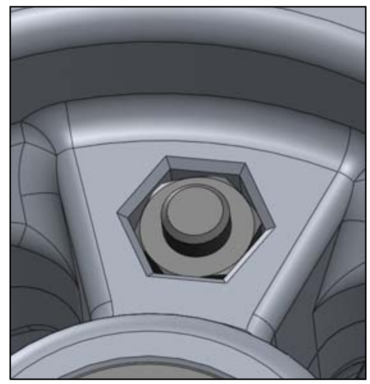
Select Other

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.

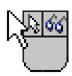
To select faces that are hidden or obscured, you use the **Select Other** option. When you position the cursor in the area of a face and press the right mouse button, **Select Other**  is available as an option on the shortcut menu. The face closest to the cursor is hidden and listed in the dialog under --*Hidden Faces*--. Other visible faces are numbered and listed in the dialog. Moving over them in the dialog highlights them on the screen.

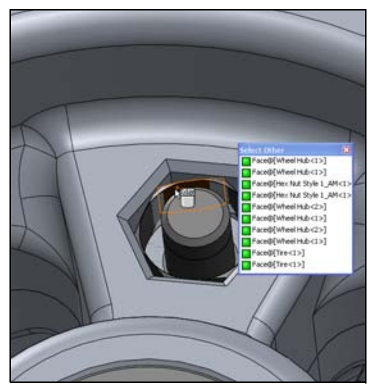
The reason the system hides the closest face is since that one was visible, if you wanted to select it you would have simply picked it with the left mouse button.



9 Click **Insert, Mate** to open the **Mate** PropertyManager.

10 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.

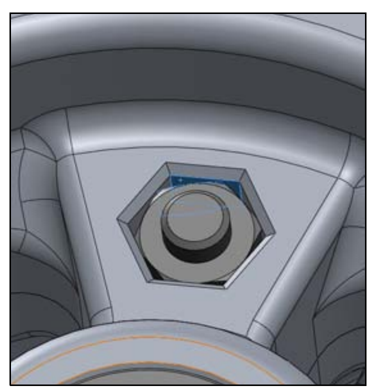
The cursor will change to . To select a face under the cursor, you can click the left mouse button. To remove a face so you can see deeper into the model, click the right mouse button.



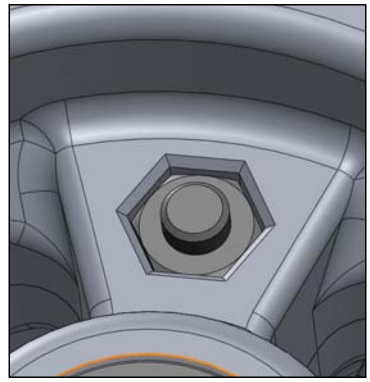
11 Move your cursor over the list. Each face will highlight when the cursor moves over it in the list.

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**.



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.

16 **Save** the assembly.

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.

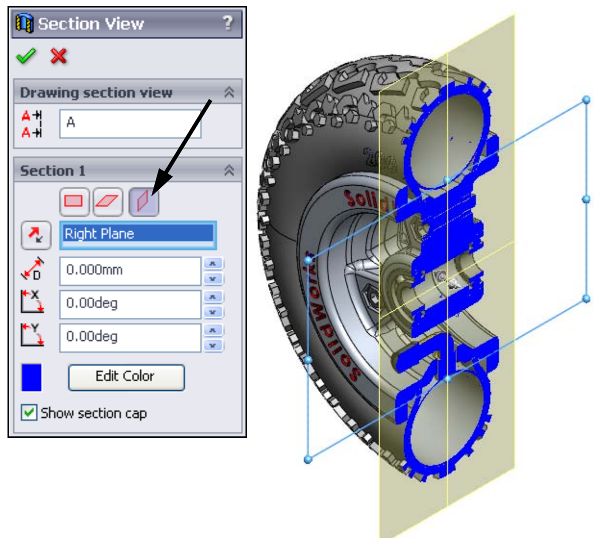
To check our work and see how the parts fit together, we can use the section view.

1 Orient the model to the Isometric view.

2 Click **Section View** on the **Heads-up View** toolbar or **View, Display, Section View** on the menu.

3 In the PropertyManager, click to select the Right plane as the section plane.

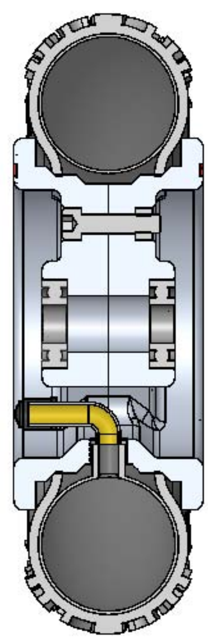
Click .



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** 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.

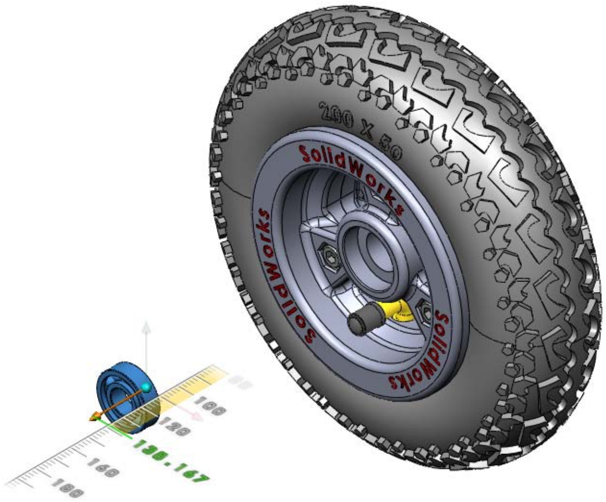
You can only create one exploded view for each configuration.

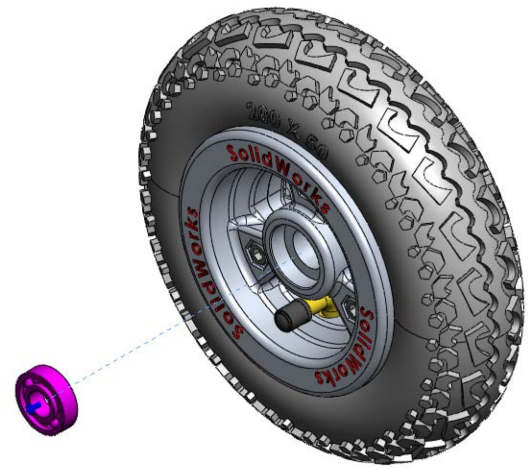
Task 1— Create An Exploded View

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.



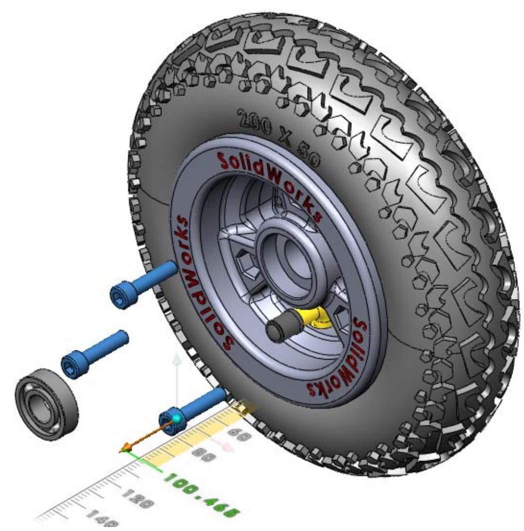


6 When you drop the part, it will change color to magenta and there will be a blue drag handle to further refine the position.

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 about **100mm** and all three bolts will move. When done, click in the graphics area to complete the step.

11 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.

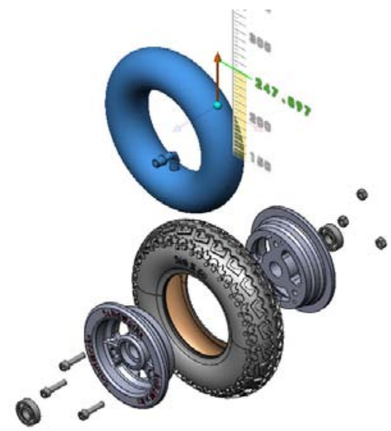
14 Select one of the `Wheel Hubs` and move it by the blue manipulator handle.

15 Repeat for the other `Wheel Hub`.

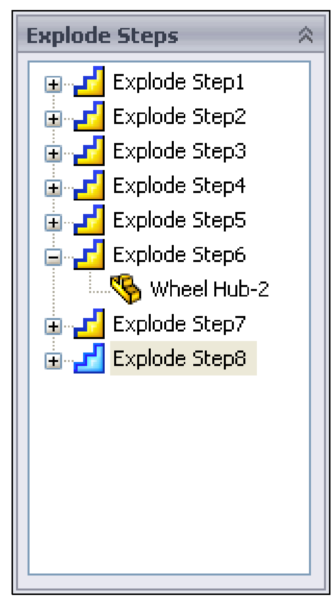
16 We want the `Inner Tube` to make two moves. First will be along the same direction as the other components, then we want it to move up along the Y direction.

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.



Adjusting the Steps

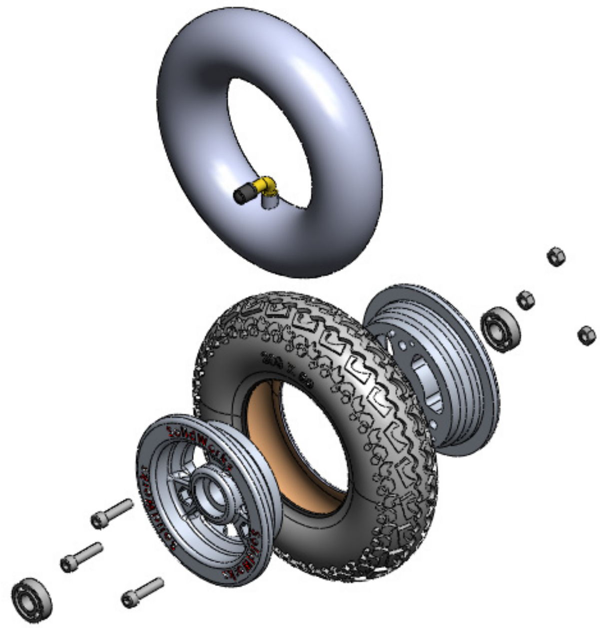


Now that all the parts have been exploded, we need to adjust their positions so that we can clearly see each component of the assembly.

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.
3 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**.

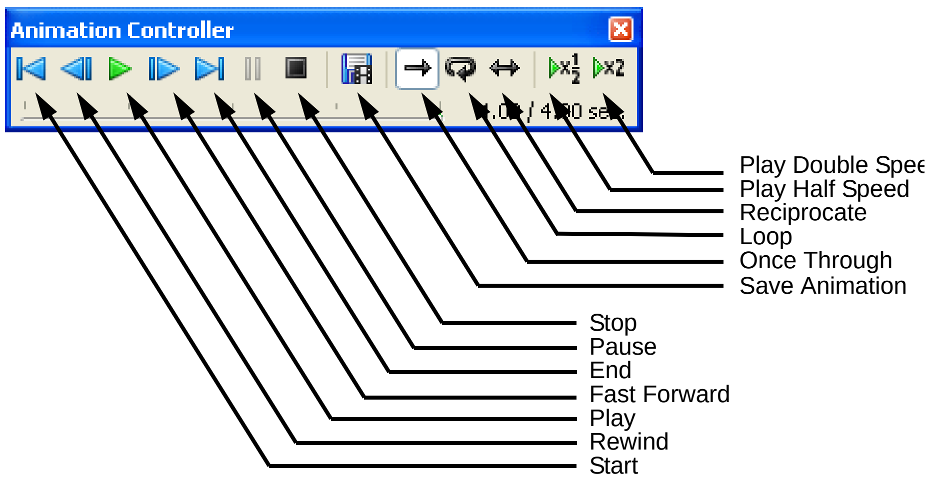
The exploded assembly will collapse to the assembled form. To explode the assembly, right-click `ExplView1` and select **Explode**.

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.



1 Right-click `ExplView1` and select **Animate Collapse**.
2 The assembly will collapse and the **Animation Controller** will appear.
3 Click the **Reciprocate**  on the **Animation Controller**. The animation will continue to explode and collapse.
4 To end the animation, click **Stop** .
5 **Close** the Animation Controller.

Note: When the Animation Controller is open, you cannot access other SolidWorks functions. You must close the Animation Controller first.

6 Collapse the assembly.
7 **Save** the assembly.

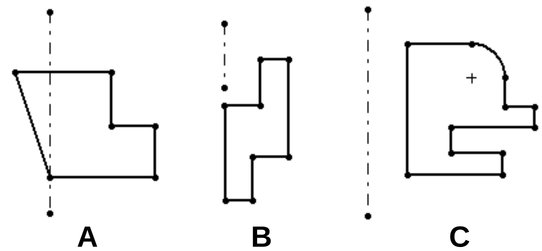
5 Minute Assessment – #4

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?

____

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?

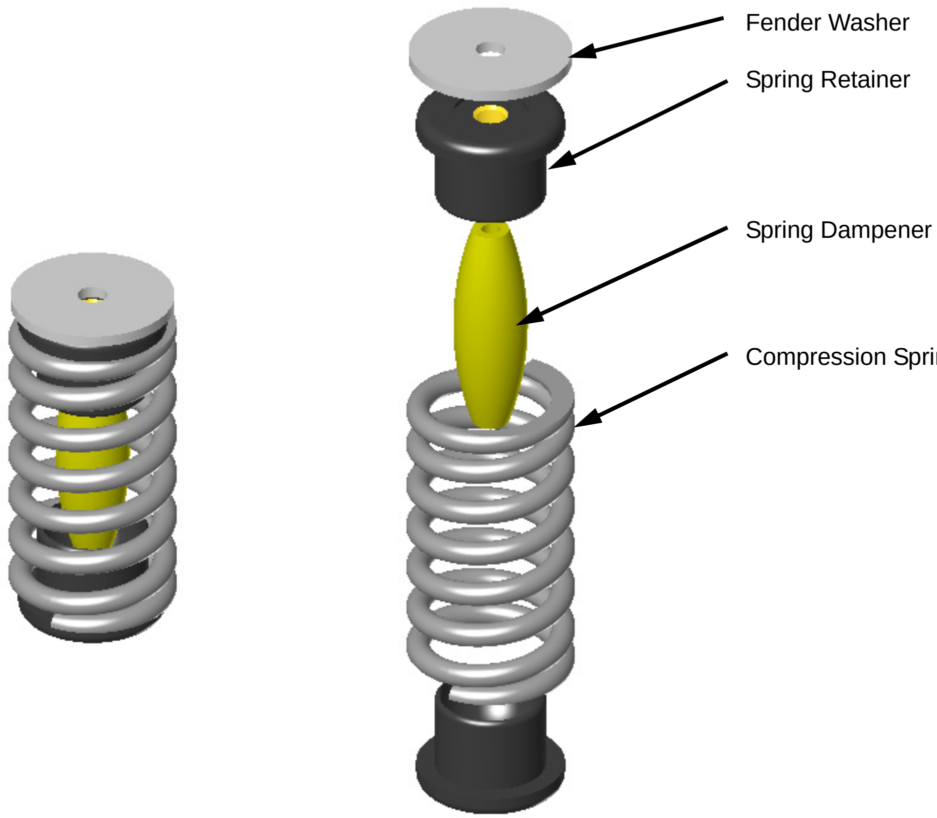
____

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.

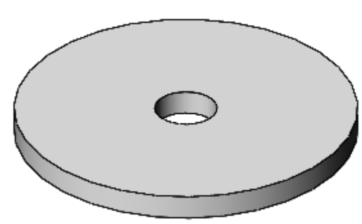
Exercises and Projects — Additional Mountainboard Parts

The following parts will be needed for the suspension of the mountainboard. The compression spring will be made in a latter lesson.

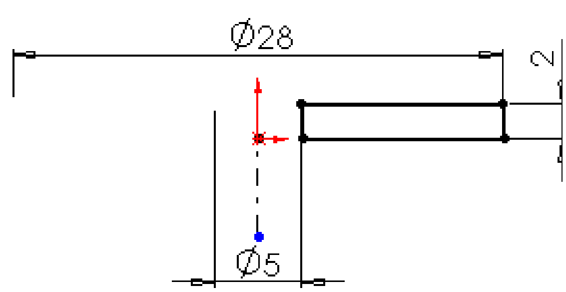


Exercise 11: Fender Washer

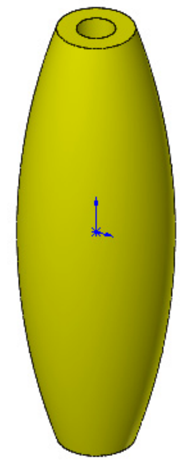
The Fender Washer is a simple revolved part.



1. Revolve the washer from the sketch shown.
2. Save the part as `Fender Washer` to the `Mountainboard\Parts` folder.



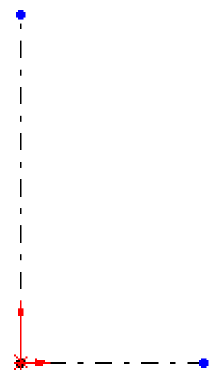
Exercise 12: Spring Dampener



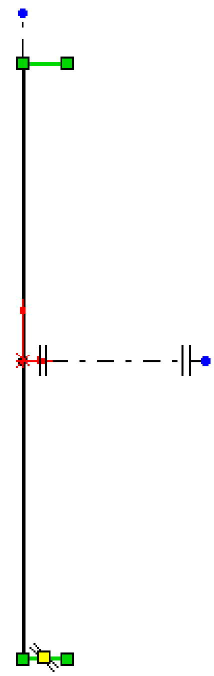
Create the Spring Dampener as a revolved part.

1 Create a sketch on the Front plane.

2 Create two centerlines from the Origin. Select the horizontal centerline and click **Dynamic Mirror Entities** .



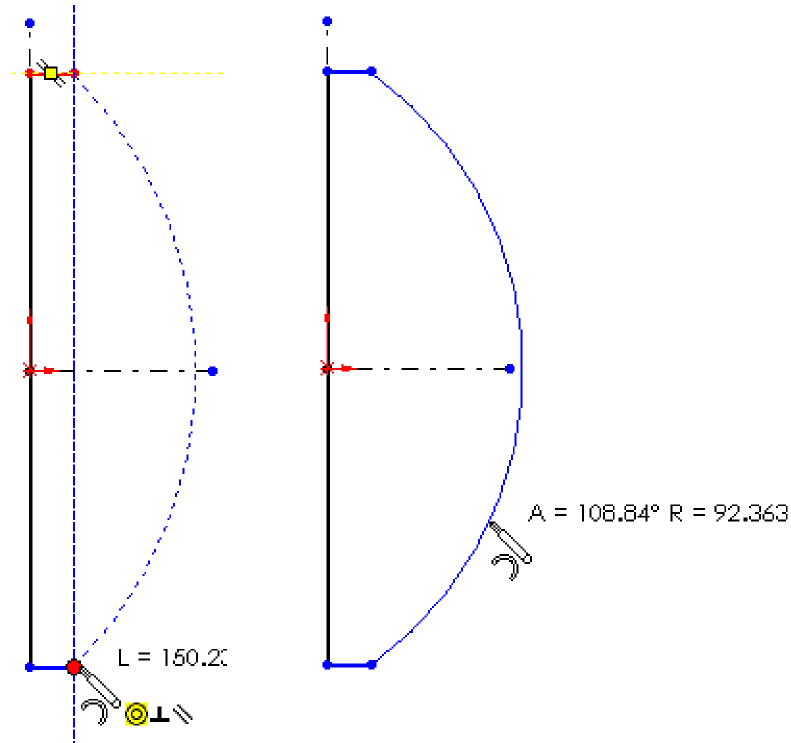
3 Sketch a vertical line from the Origin, then from its end sketch a horizontal line.



4 Turn off **Dynamic Mirror Entities**. Select **3 Point Arc** .

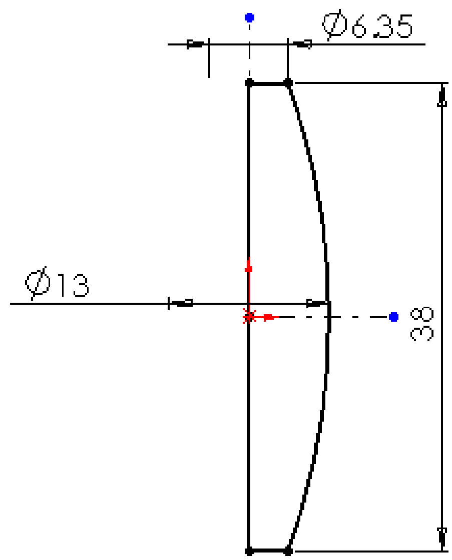
Creating a **3 Point Arc** is a two step process. Sketch from the end of one of the horizontal lines to the end of the other, then release the mouse button.

Place the cursor over the arc, then drag the arc to adjust its size.



5 Dimension the sketch as shown.

TIP: The 6.35mm and 13mm dimensions must be created to the centerline, NOT the vertical line in the sketch. After selecting, move the cursor to the left of the centerline to create the diameter dimensions. Also, remember to change the 13mm dimension to Minimum arc condition in the dimension properties.

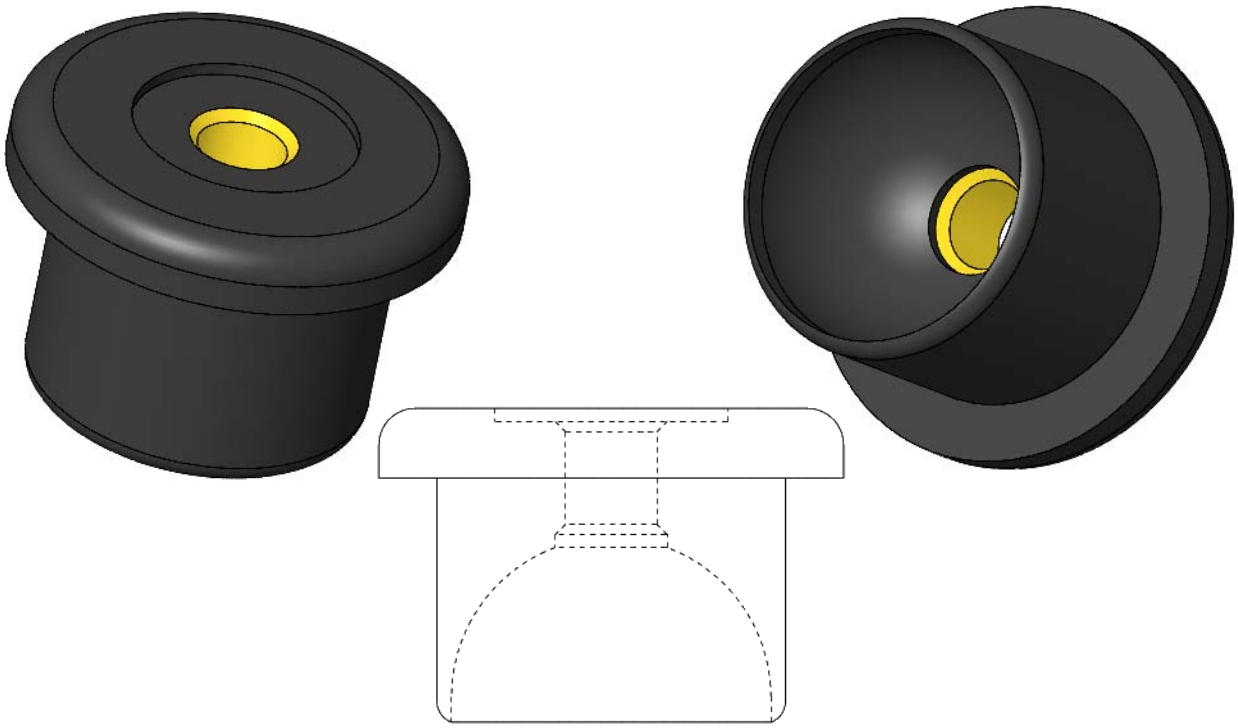


6 **Revolve** the part about the vertical centerline.

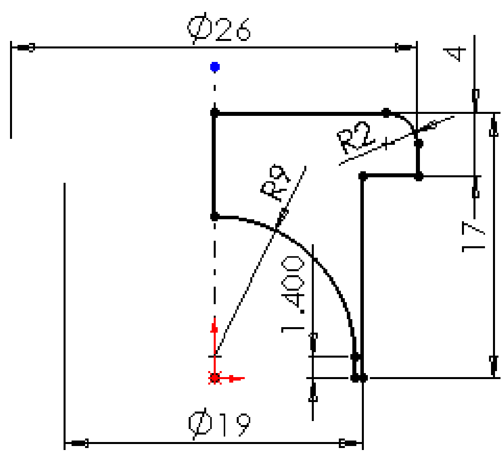
7 Use the Hole Wizard to create a **M3** clearance hole.

8 **Save** the part as Spring Dampener to the Mountainboard\Parts folder.

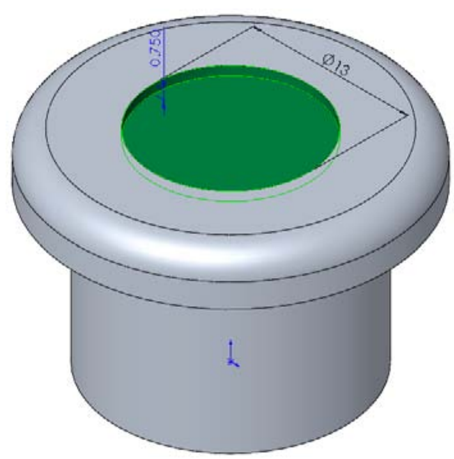
Exercise 13: Spring Retainer



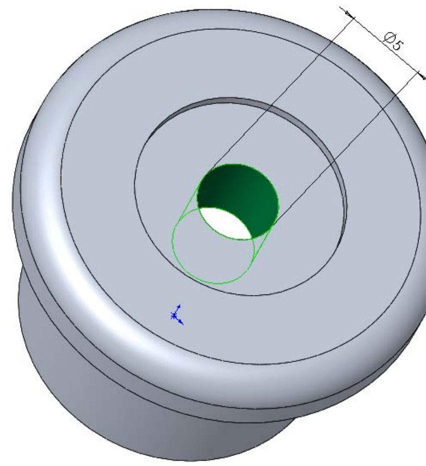
1 Create a base revolve using the dimensions shown.



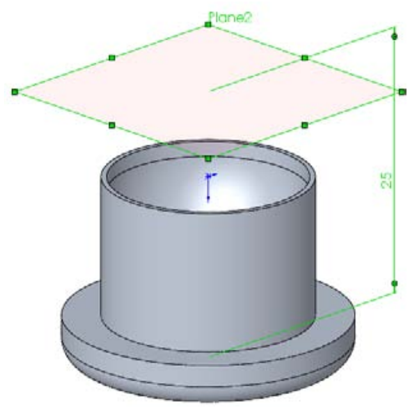
2 Create a recess cut **13mm** in diameter and **0.75mm** deep.



3 Create a **Through All** cut, **5mm** in diameter.

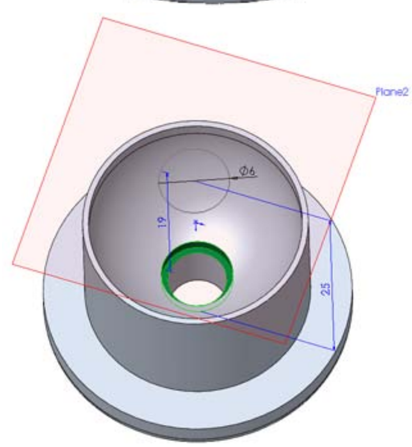


4 Create a new plane **25mm** from the top face of the model.

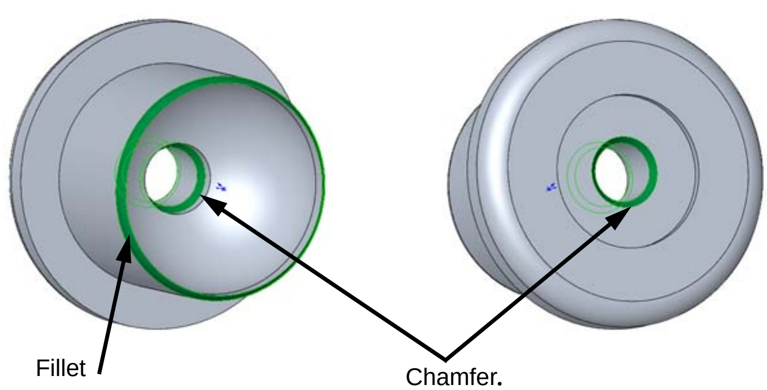


5 Extrude a Blind cut.

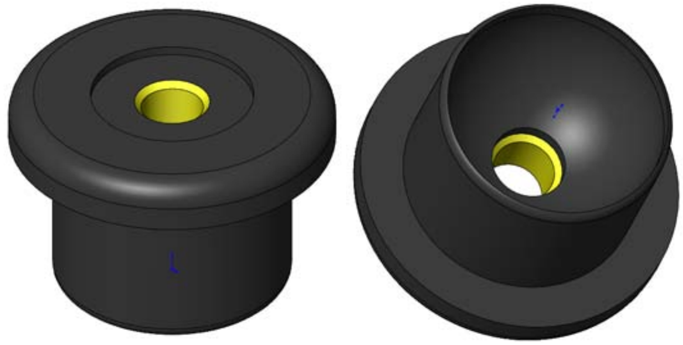
Sketch a circle **6mm** in diameter on the new plane. Extrude a cut to a Blind depth of **19mm**.



6 Add a **1 mm** fillet and a **0.5mm** by **45°** chamfer to the edges shown.



7 Add appearance to the part. Make the overall part black and the center hole and the chamfers on each end yellow.

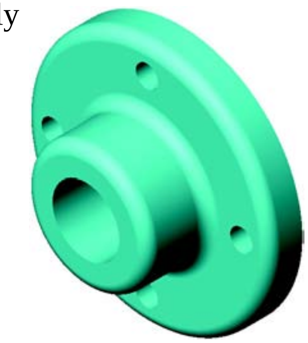


8 **Save** the part as `Spring Retainer` to the `Mountainboard\Parts` folder.

9 **Close** the part.

Exercises and Projects — Revolved Parts

Exercise 14: Flange



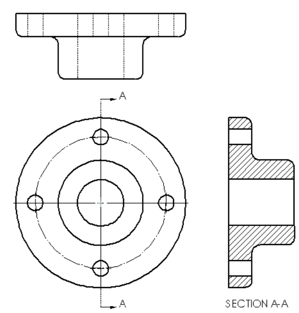
Create this part using the dimensions provided. Use relations wisely to maintain the design intent.

This lab uses the following skills:

- ❑ Revolved features.
- ❑ Circular patterning.

Units: **inches**

Design Intent



The design intent for this part is as follows:

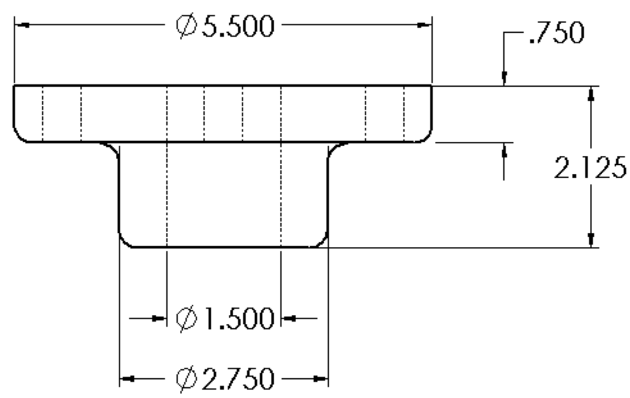
- ❑ Holes in the pattern are equally spaced.
- ❑ Holes are equal diameter.
- ❑ All fillets are equal and are **R0.25”**.

Note that construction circles can be created using the **Properties** of a circle.

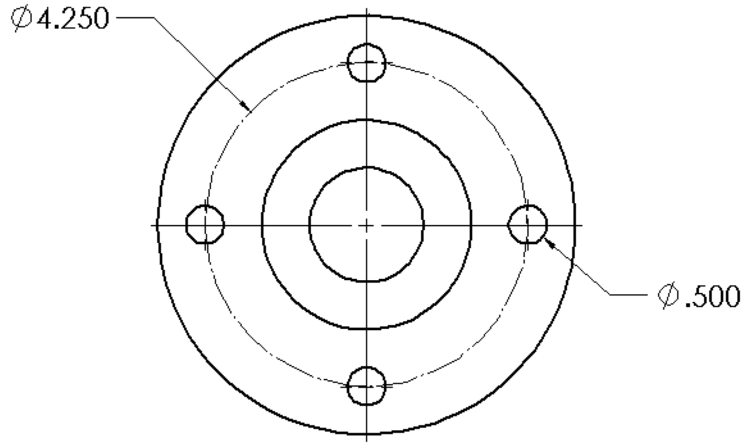
Dimensioned Views

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

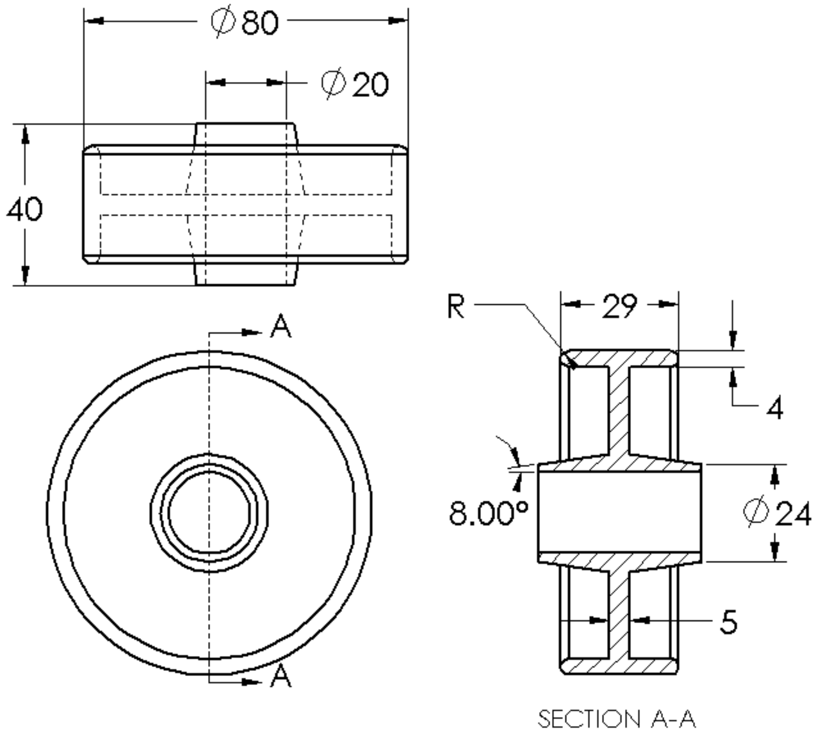
The design intent for this part is as follows:

- 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.

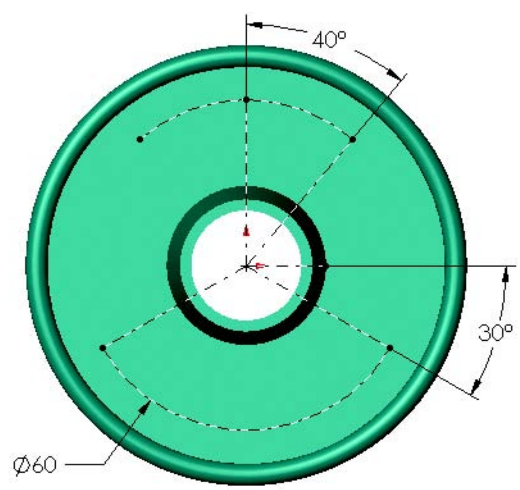


Optional: Text in a Sketch

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.

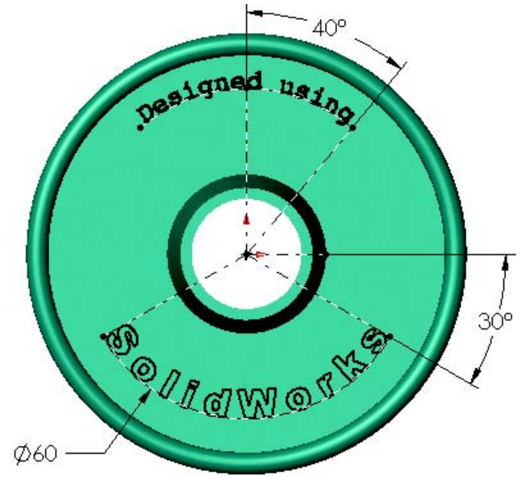
Sketch on the front face and add construction lines and arcs as shown.



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

1 Extrude.

Extrude a boss with a **Depth** of **1mm** and **Draft** of **1°**.

2 **Save** the part and close it.

Lesson 4 — Quiz

Name: ______________________________Class: _________ 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? ___________________________________

____

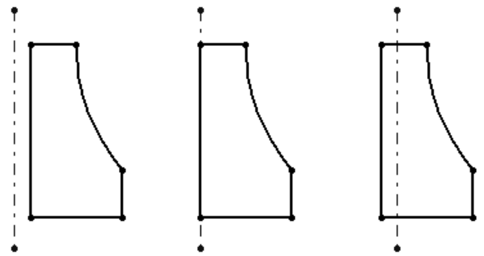
9 How do you specify the location of a Toolbox part? ____________________________

____

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.

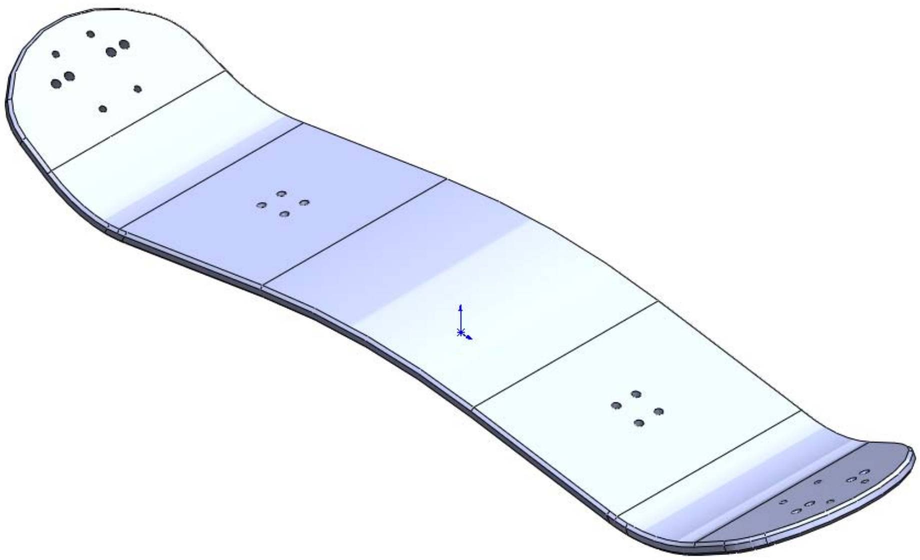


- Files can be imported from other CAD software using neutral file formats.
- Toolbox provides ready-to-use parts — such as bolts and screws.
- Toolbox parts are placed by dragging and dropping them into assemblies.
- An assembly contains two or more parts.
- In an assembly, parts are referred to as 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.

5

Lesson 5: Thin Features — The Deck

- 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

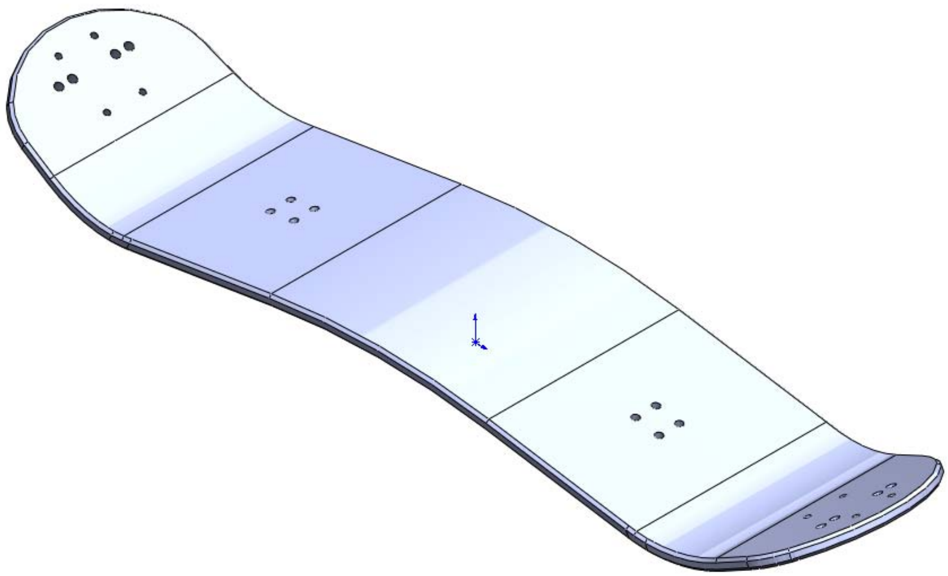
  The deformation resulting from totaling all the internal strain.

- 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.

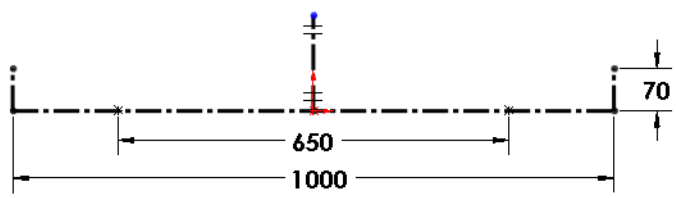
In the PropertyManager select to add the **Midpoint** relationship.

5 Add a dimension to the line of **1000mm**.

6 Mirror as we sketch. Click the vertical centerline, then click **Dynamic Mirror Entities** on the Sketch toolbar to begin mirroring the sketch.

7 From one end of the horizontal centerline draw a vertical centerline and dimension it **70mm**.

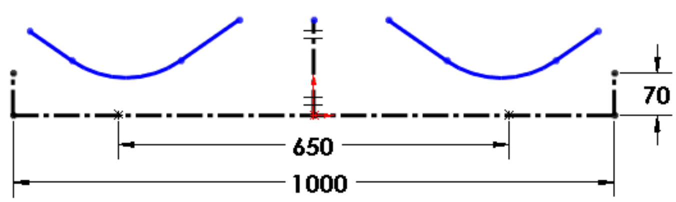
8 Click the **Point** tool on the Sketch toolbar and click on the horizontal centerline. The sketch mirror will create a second point on the other side of the `Origin`.



9 Dimension the distance between the two points **650mm**.

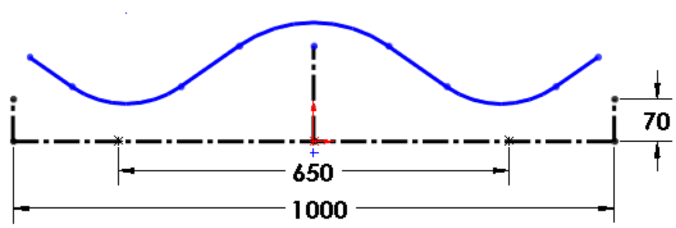
10 Sketch a line, tangent arc, and another line as shown. **Dynamic Mirror Entities** will create a mirror image.

11 Turn off **Dynamic Mirror Entities** by clicking in the **Sketch** toolbar.



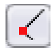
12 Sketch a tangent arc to connect the two halves of the sketch.

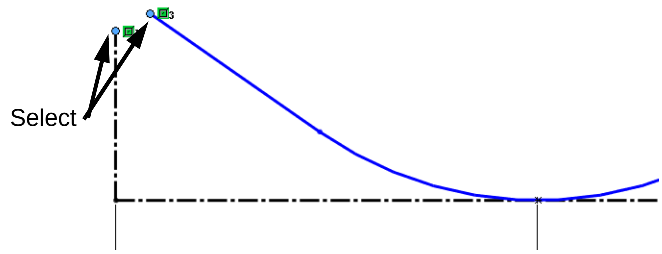
13 Add relationships. Select the left arc and **Control** select the horizontal construction line. In the PropertyManager select **Tangent** .



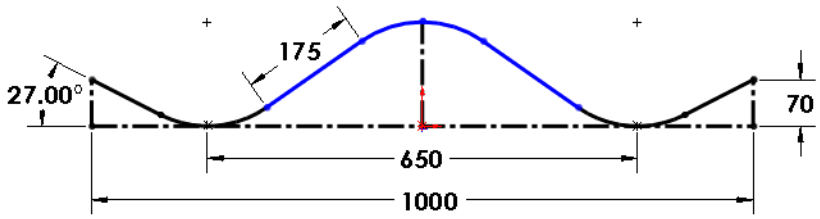
14 Select the left arc again and control select the left point we added. In the PropertyManager select **Coincident**.

15 This makes the arc tangent at this specific point.

16 Select the two points shown.In the PropertyManager select  **Merge**. Because the endpoint of the construction line is fully defined, the sketch line must move to it.

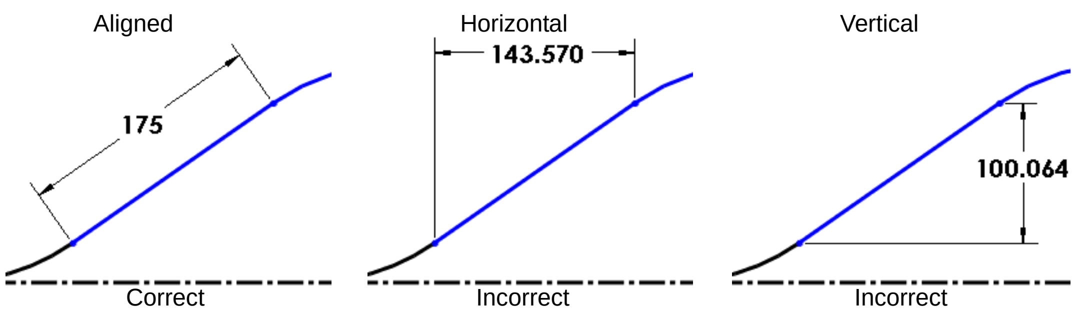


17 Add the two dimensions shown.



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.

.

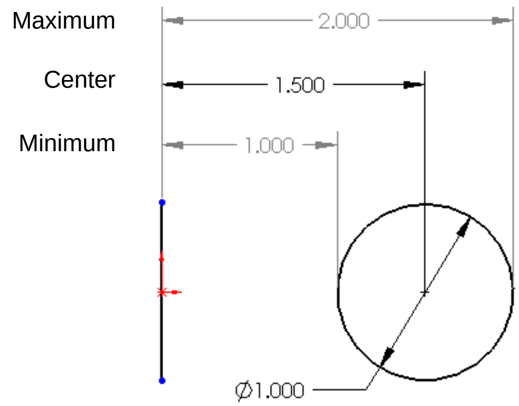


18 Add a dimension from the center arc to the horizontal centerline. Place the dimension below the horizontal centerline and accept the existing dimension.

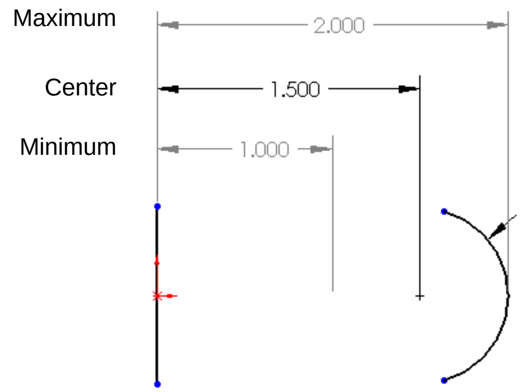
Arc Conditions

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:

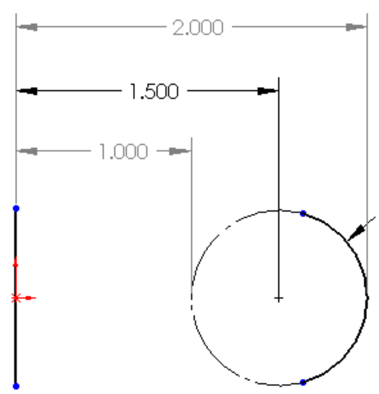
- Drag the dimension’s extension line to the arc or circle
- Edit the Properties of the dimension.



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?



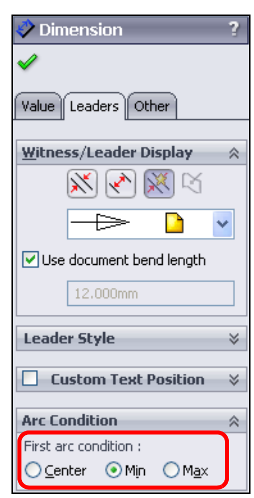
Even though we see an arc, the underlying geometry is a circle.



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.

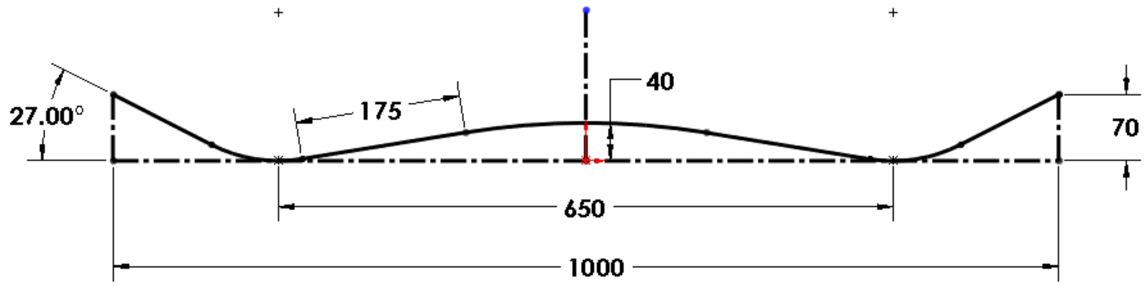


21 Click **OK**.

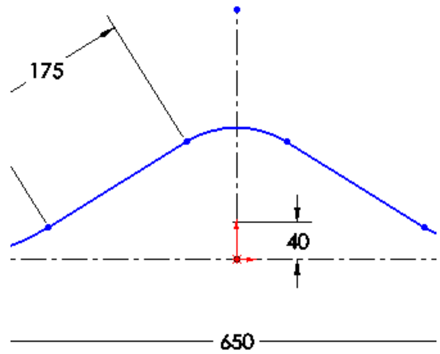
22 Double click the dimension and change it to **40mm**.

23 Click **OK**.

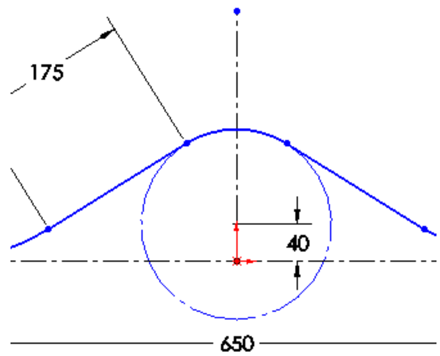
24 The sketch is now fully defined.



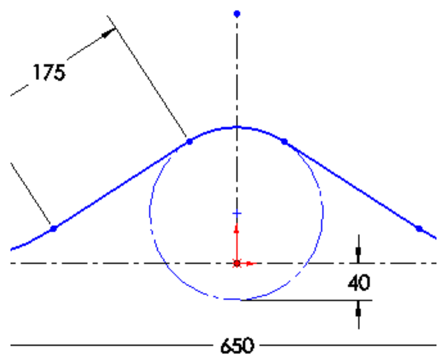
Note: If after placing the dimension between the arc and the horizontal centerline your sketch looks like the sketch at right, your arc is too far above the horizontal centerline.



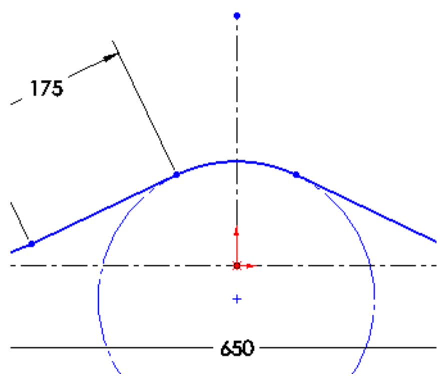
If you could visualize the underlying circle used to create the arc, you would see that the initial dimension actually goes to the center of the arc.

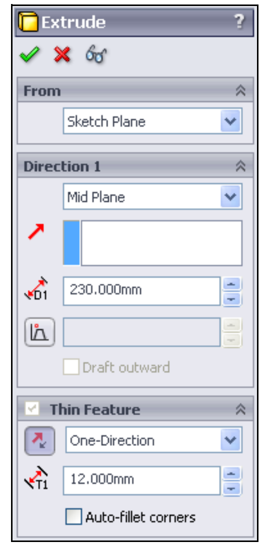


The minimum arc condition will be to the lower part of the circle because it is closer to the centerline.



To get the dimension we want, we must first drag the sketch closer to the correct position, BEFORE adding the dimension. We want the center of the arc to be below the horizontal centerline. This will mean that the upper part of the circle is closer (minimum condition) to the horizontal centerline than the lower part of the circle which will then be farther away (maximum arc condition).

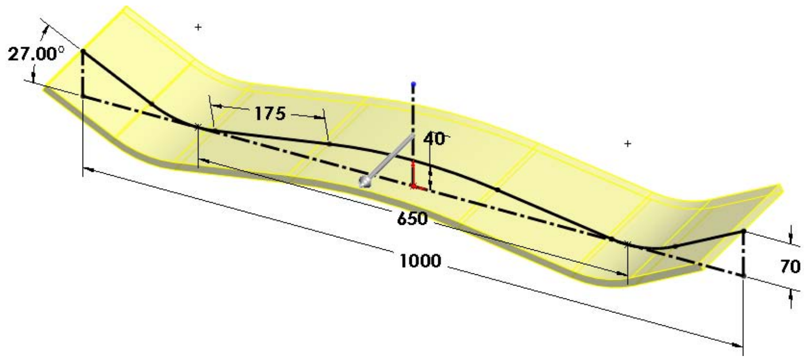




25 Create a thin extrusion. Click **Extrude Boss/Base**  on the **Features** toolbar.

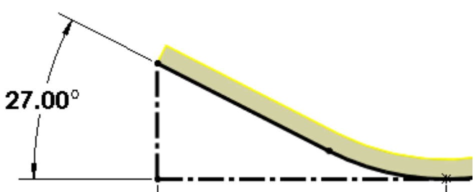
26 Select **Mid Plane** for Direction 1 and set the Depth to **230 mm**.

Mid plane depth refers to the total depth of the extrusion. In this case, the extrusion will be 115 mm to each side of the sketch.

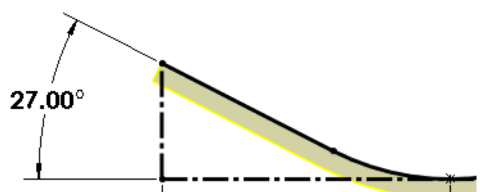


Because we are extruding an open sketch, **Thin Feature** is checked by default. For thin features, **T1** is the thickness of the extrusion in the sketch plane. Thickness can be added to either side of the sketch or both. Type **12mm** for the thickness.

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  in the PropertyManager.



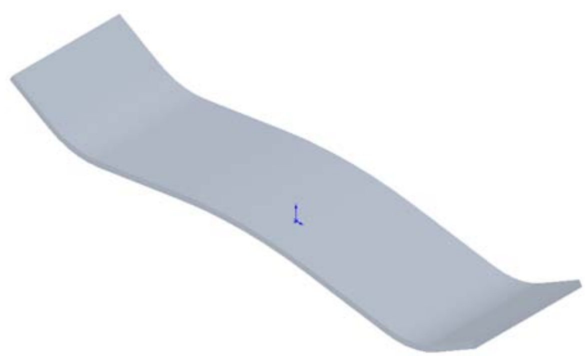
Correct



Incorrect

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"

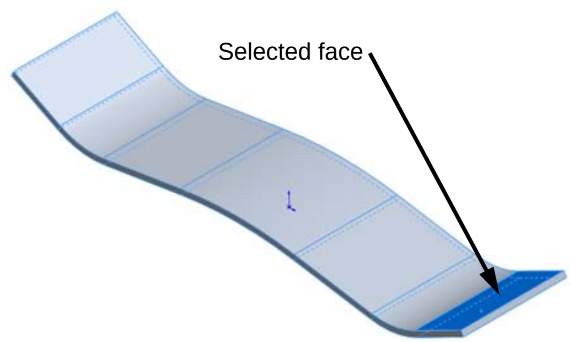
The **View Normal To** option is used to change the view orientation to a direction normal to a selected planar geometry. The geometry can be a reference plane, sketch, planar face or feature that contains a sketch.

Clicking the **Normal To** icon a second time will flip the orientation around to the opposite side of the plane.

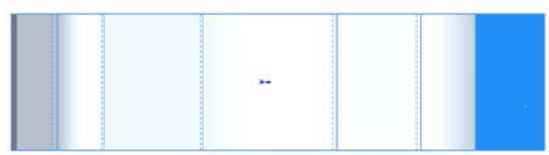
Zoom To Selection

Zoom to Selection zooms in on the selected entity. You can select the entity in either the Graphics Area or the FeatureManager design tree.

1 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.

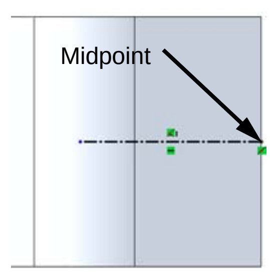


2 With the face still selected, click **Zoom to Selection** on the View toolbar. This will zoom in to the selected face.

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.

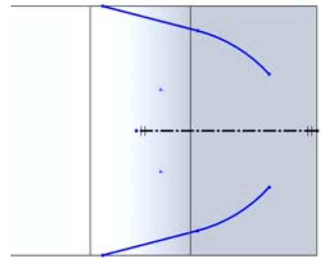
5 Click the **Centerline** tool and draw a centerline from the midpoint of the vertical edge.



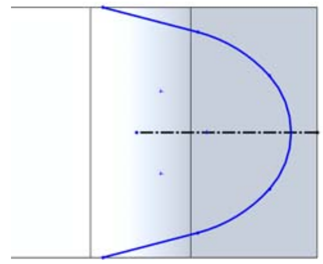
6 Click **Dynamic Mirror Entities** on the **Sketch** toolbar.

7 Sketch a **Line** and **Tangent Arc** as shown.

8 Turn off **Dynamic Mirror Entities**.



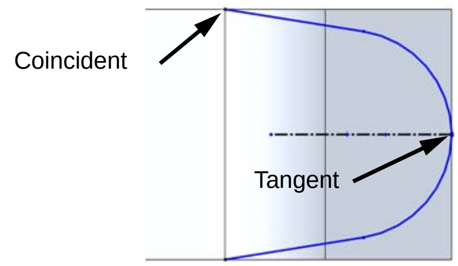
9 Sketch a **Tangent Arc** to connect the two existing arcs.



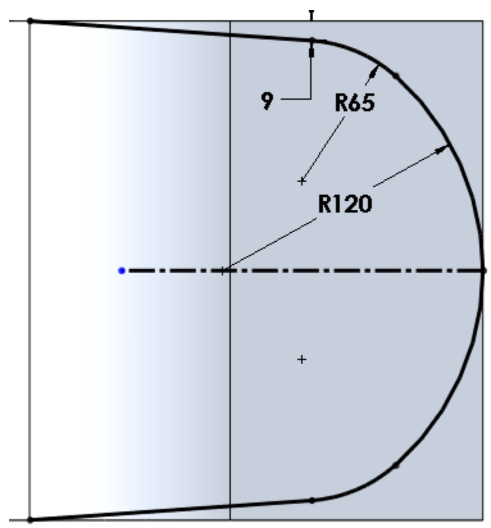
10 Add a **Tangent** relationship between the arc and the edge of the board as shown.

11 Add a **Coincident** relationship between the endpoint of the line and the vertex as shown.

12 The sketch should now be fully defined.

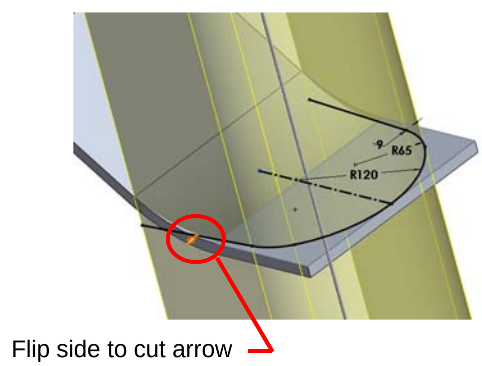


13 Add the three dimensions shown.

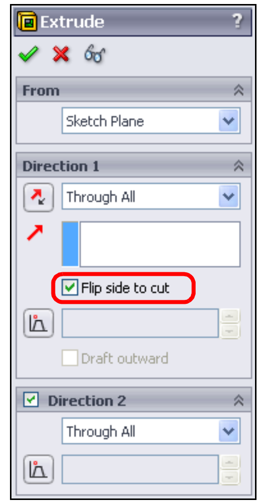


14 Use the open sketch to cut off the end of the Deck.Click to create a **Insert, Cut, Extrude** from the menu.

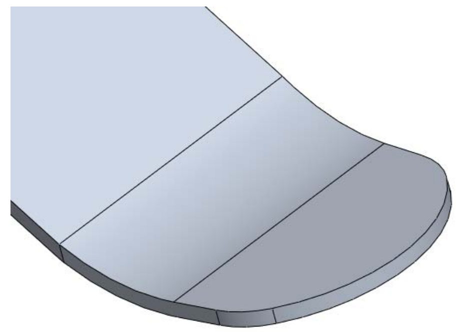
15 Reorient the model to the Isometric view and Zoom in. Using an open sketch will automatically choose to cut **Through All** in both directions. Locate the **Flip Side To Cut** arrow, it points to the side of the sketch that will be removed. Make sure it is pointing to the outside.



16 To change the side to remove, you can click on the **Flip Side To Cut** arrow or select **Flip side to cut** in the PropertyManager.



17 Click  to complete the cut.



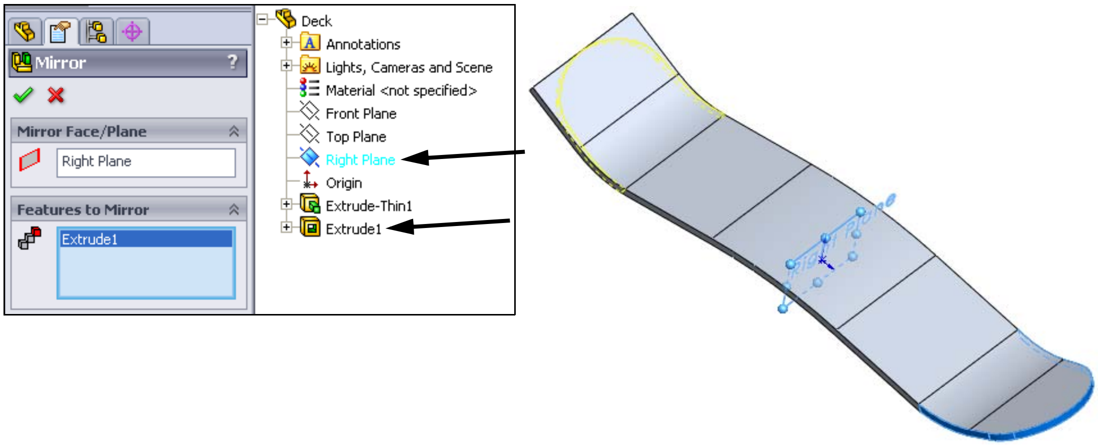
Mirroring Features

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

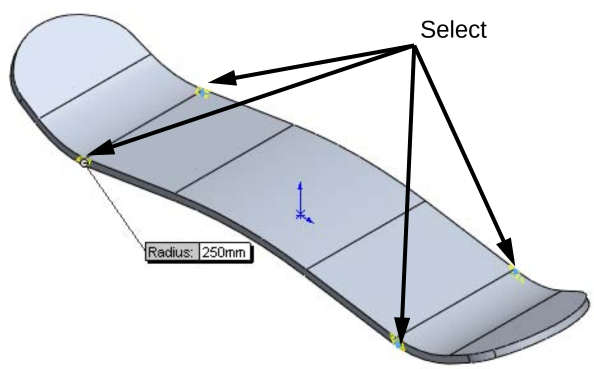
The transition between the cut we just made and the side of the Deck created a hard edge. Both for aesthetics and safety we would like a smoother transition. This can be accomplished is a large radius fillet.

1 Click **Fillet**  on the **Features** toolbar to create a fillet.

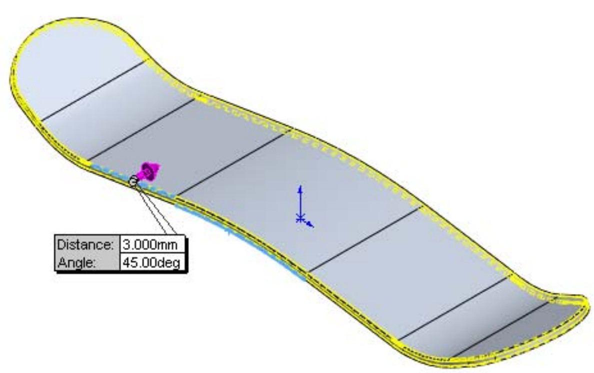
2 Select the four edges shown.

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.
- Type **Ctrl + C**

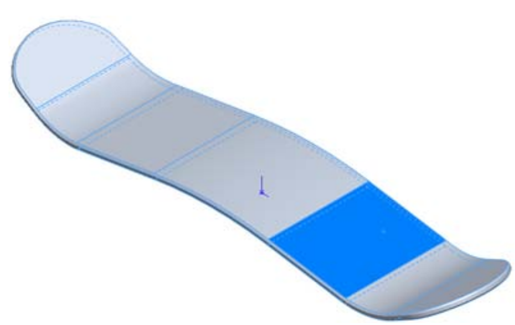
To past a sketch or feature, select the plane or face where you want to paste the sketch or feature, then:

- 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** on the View toolbar. Then, click **Zoom to Selection** .

3 Click to show the model **Shaded With Edges**.

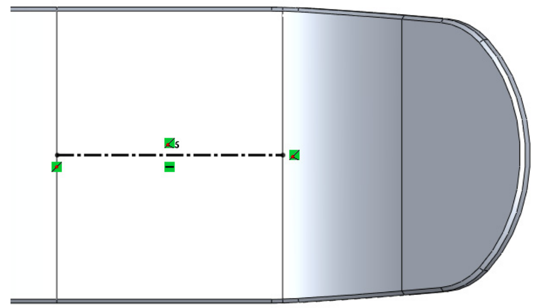


4 Sketch a centerline between the two midpoints as shown.

5 Select **Dynamic Mirror Entities** .

6 Sketch one circle.

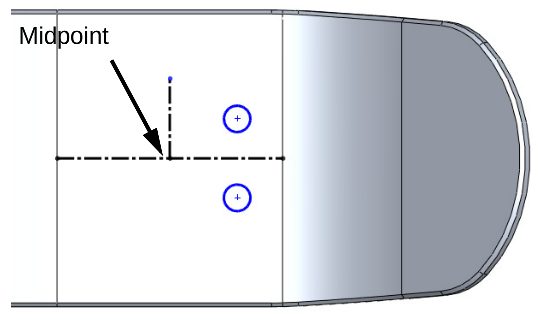
7 Turn off **Dynamic Mirror Entities**.



8 Sketch a second centerline, vertically from the midpoint of the first centerline.

9 Select this new centerline, then hold **Control** and select the two circles.

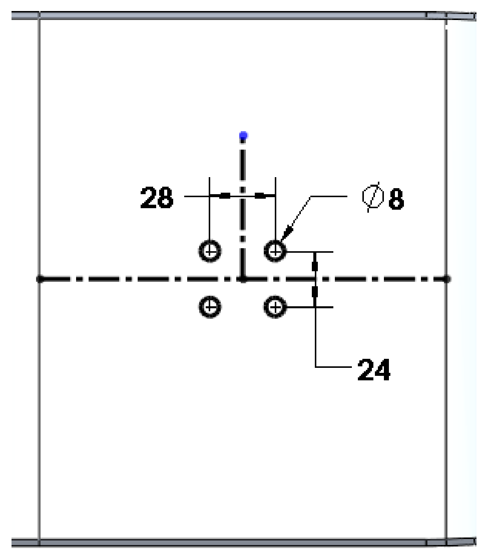
10 Click **Mirror Entities** to mirror the two circles.



11 Dimension the sketch as shown.

12 Create a cut, **Through All**.

13 Rename this feature Binding Holes.

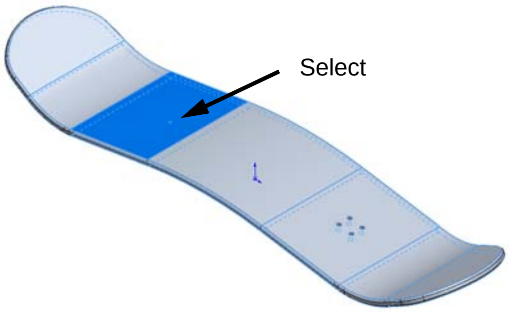


Task 5 — Copy The Binding Holes Sketch

1 Copy the sketch for the Binding Holes. Click the plus sign next to the Binding Holes feature and select the sketch.

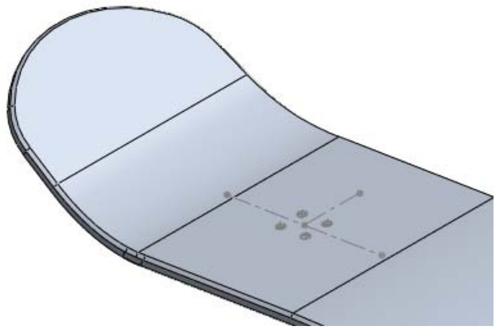
2 Click **Edit, Copy** from the menu. This places a copy of the sketch on the Windows clipboard.

3 Select the face shown. This is where we will paste the sketch.



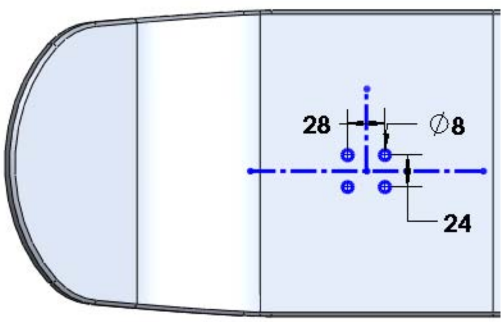
4 Click **Edit, Paste** from the menu. The sketch will appear on the selected face but its position will depend on where you picked the face to select it. A new, under defined, sketch appears in the FeatureManager design tree.

5 Edit the new sketch. Right-click the new sketch and select **Edit Sketch**.

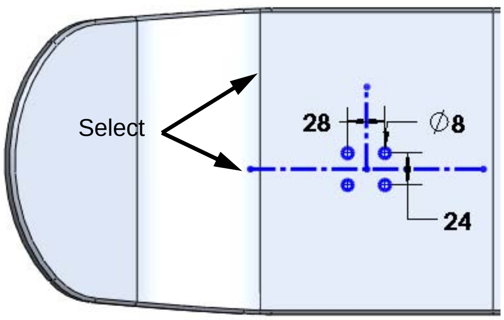


6 Change the view orientation to **Normal To** the sketch.

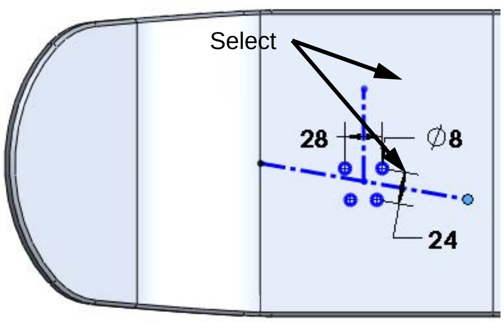
7 The sketch has been copied with all the dimensions. What could not be copied were the relationships to things outside the sketch. In this case, that would be the midpoint relationships between the ends of the horizontal centerline and edges of the sketch plane.



8 Click **Add Relation** on the **Sketch** toolbar. Select the endpoint of the centerline and the edge shown. Add a **Midpoint** relationship.



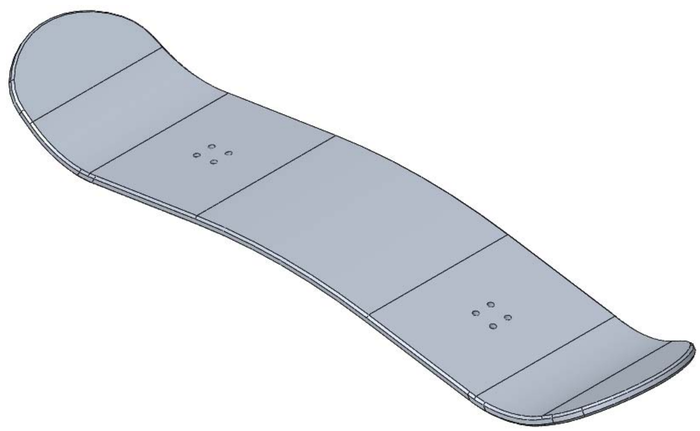
Notice that the Sketch is not fully defined. To determine what's wrong, try to drag the right end of the horizontal centerline.



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.

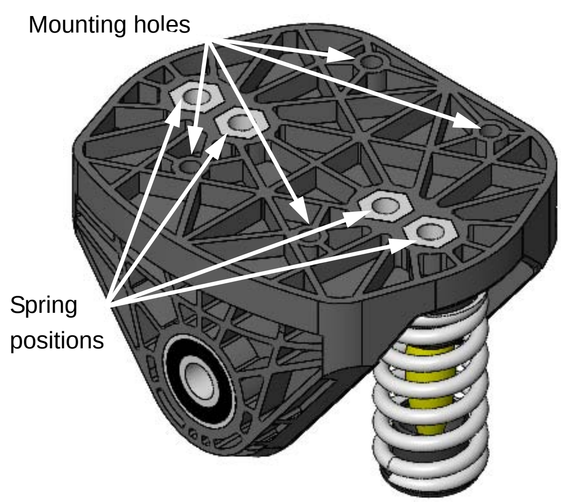


Task 6 — Add The Truck Mounting Holes

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.

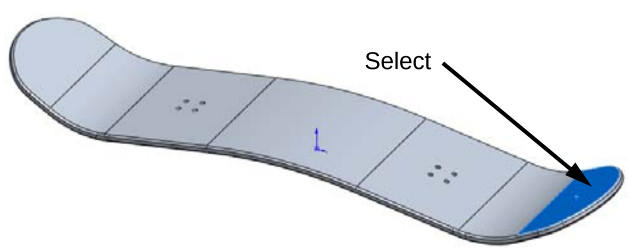
All holes in the `Deck` will be through hotels.



1 Select the face shown.

2 Reorient the view to **Normal To** and zoom in.

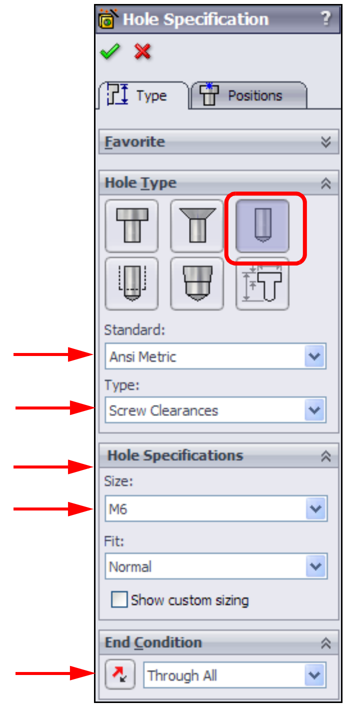
3 Start the Hole Wizard by clicking  on the **Features** toolbar.



4 Select the following settings:

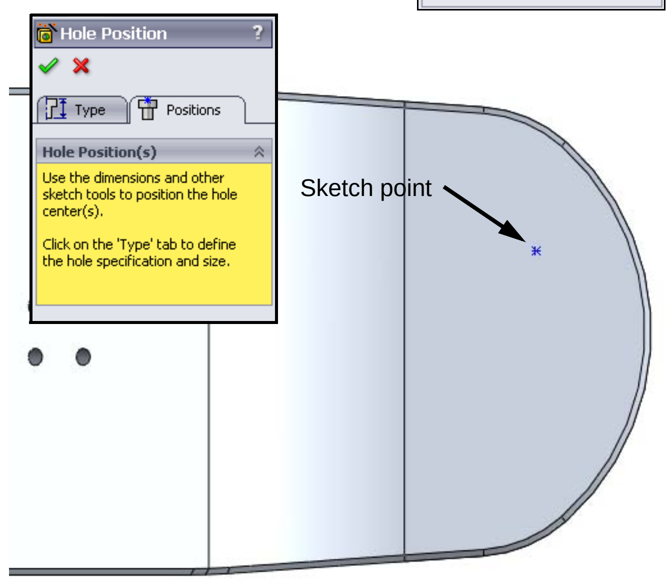
- Type - **Hole**
- Standard - **Ansi Metric**
- Screw Type - **Screw Clearances**
- Size - **M6**
- End Condition - **Through All**

Click on the **Positions** tab.



1 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**.



2 Clear the **Point** tool.

3 We need a total of four holes of this size that will be symmetric about the centerline of the deck.

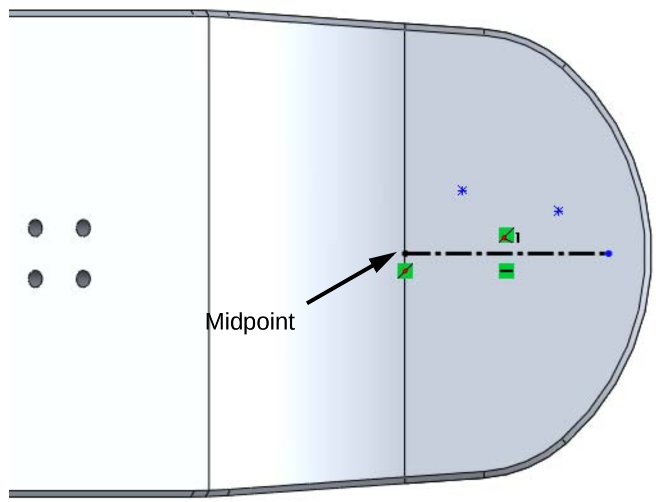
Draw a centerline from the midpoint of the edge shown.

4 Mirror the two sketch points.

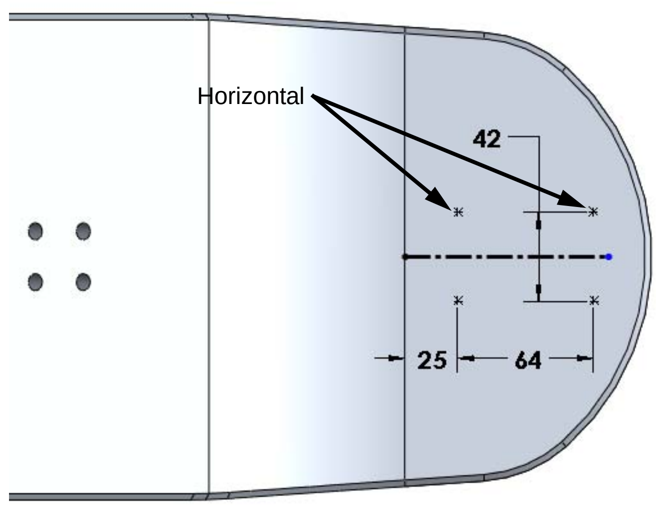
Clear the **Centerline** tool.

Press and hold **Control** and select the two points and the centerline.

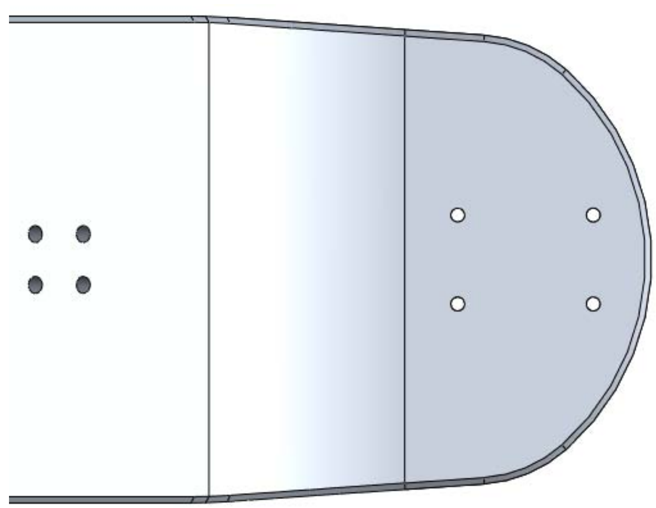
Click **Mirror Entities** to mirror the two points.



5 Dimension the points as shown. Add a **Horizontal** relationship between two points as shown.



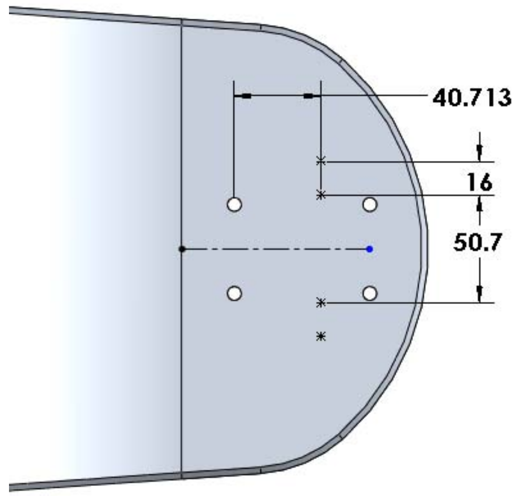
6 Click **OK** in the **Hole Wizard** dialog. These are the four holes that will be used to mount the Truck.



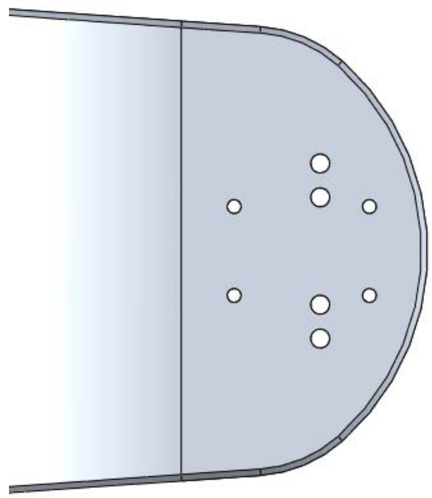
7 Repeat the above procedure to create four additional holes for the spring adjustment.

8 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**

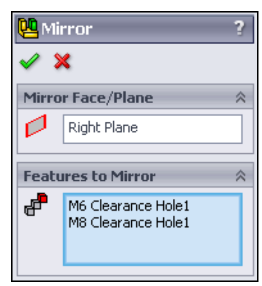


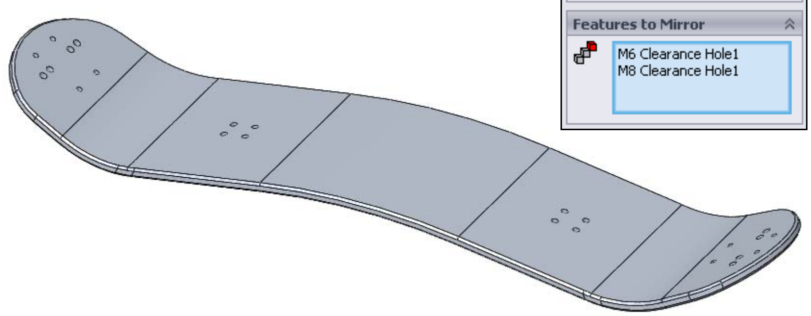
9 The finished pattern should look like this.



10 Mirror both hole patterns to the other end of the Deck. Mirror both hole patterns around the Right plane using the **Mirror** command only once.

11 **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.

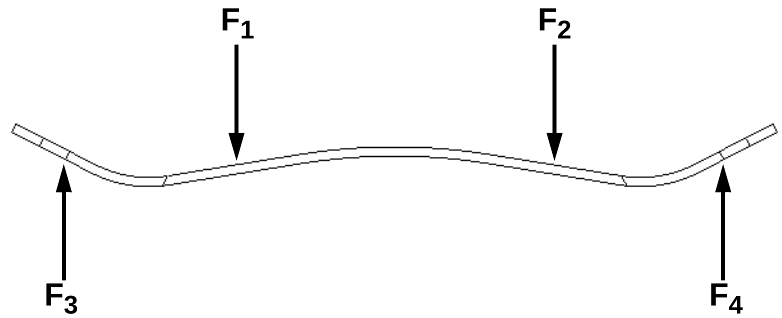
If you push on a body, like the mountainboard, it will continue to move faster and faster. If there was no friction or wind resistance, it would just keep accelerating. In the real world, as the mountainboard picks up speed, friction and wind resistance increase until they equal the force you are pushing 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.

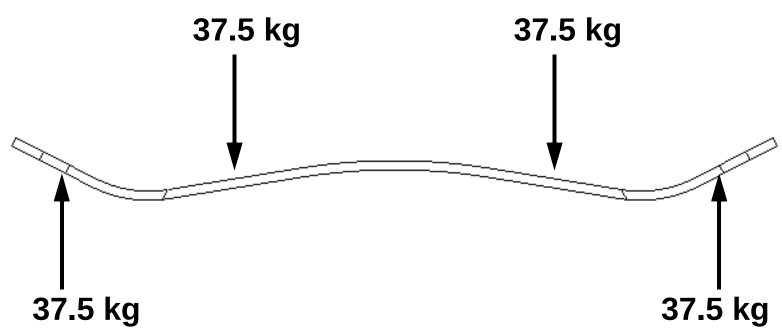
Free Body Diagrams

The Free Body Diagram isolates the part and shows all the external loads applied to it. For the Deck, we would have $\mathbf{F_1}$ and $\mathbf{F_2}$ which would be the weight of the rider transmitted to the Deck through the rider's feet. $\mathbf{F_3}$ and $\mathbf{F_4}$ 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, $\mathbf{F_1} + \mathbf{F_2} = 75$ kg. If the rider's weight is evenly distributed, then $\mathbf{F_1} = \mathbf{F_2} = 37.5$ kg. Because the Deck is symmetrical and the rider's weight is applied symmetrically it should be obvious that $\mathbf{F_3} = \mathbf{F_4} = 37.5$ 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 $\mathbf{F_1} = 75$ kg and $\mathbf{F_2} = 0$. How do we determine $\mathbf{F_3}$ and $\mathbf{F_4}$ 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 = $\Sigma \mathbf{F_y} = \mathbf{F_3} + \mathbf{F_4} - \mathbf{F_1} - \mathbf{F_2} = 0$

$\mathbf{F_3} + \mathbf{F_4} - 75 - 0 = 0$

$\mathbf{F_3} + \mathbf{F_4} = 75$

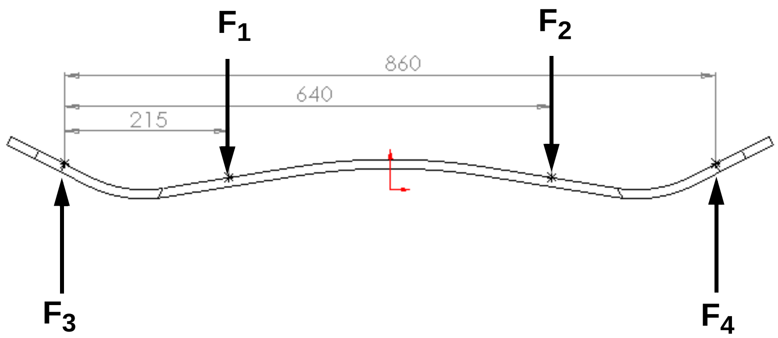
Moments

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 $\mathbf{F_3}$ and $\mathbf{F_4}$ 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 $\mathbf{F_3}$ acts on the Deck. Since $\mathbf{F_3}$ 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 $\mathbf{F_3}$ with counterclockwise as positive:



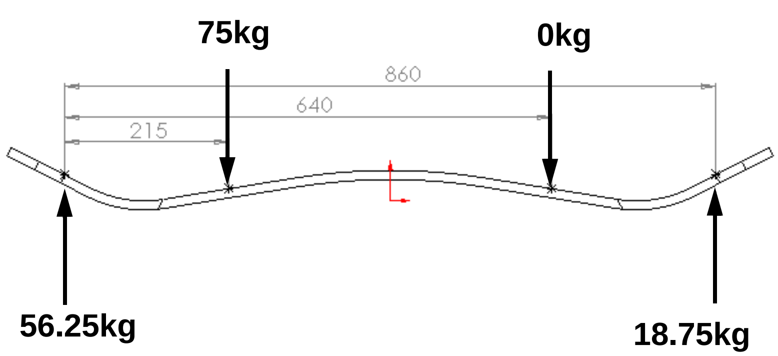
Sum of the moments about $\mathbf{F_3} = \Sigma M_{F3} = (-\mathbf{F_1} * .215) + (-\mathbf{F_2} * .640) + (\mathbf{F_4} * .860) = 0$

$(-75 * .215) + (0 * .640) + (\mathbf{F_4} * .860) = 0$

$-16.125 + 0 + (\mathbf{F_4} * .860) = 0$

$\mathbf{F_4} = 16.125 / .860 = 18.75$ kg

Therefore: $\mathbf{F_3} = 75 - \mathbf{F_4} = 75 - 18.75 =$ **56.25kg**

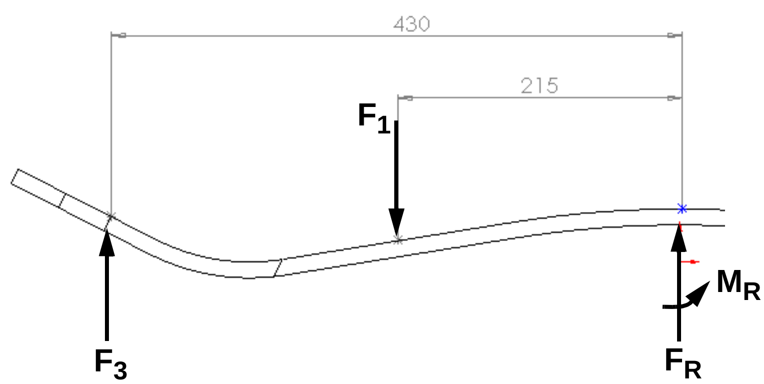


Stress

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 $\mathbf{F_1}$ and $\mathbf{F_3}$. These must be resisted by a resultant internal force and moment $\mathbf{F_R}$ and $\mathbf{M_R}$.



At this point in the model, $\mathbf{F_R} = \mathbf{F_1} - \mathbf{F_3} = 75 - 56.25 = 18.75$ kg.

If we consider counterclockwise rotation to be positive and clockwise rotation as negative:

$\mathbf{M_R} = (\mathbf{F_1} * .215) - (\mathbf{F_3} * .430) = (75 * .215) - (57.35 * .430) = -8.54$ kg-m.

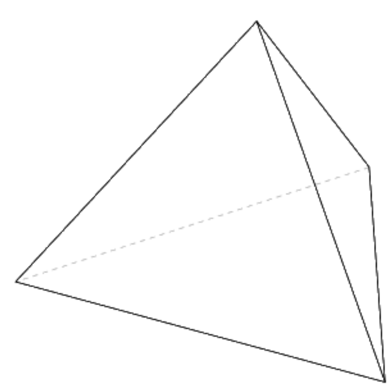
The negative value means that the moment acts in a clockwise direction.

Finite Element Analysis

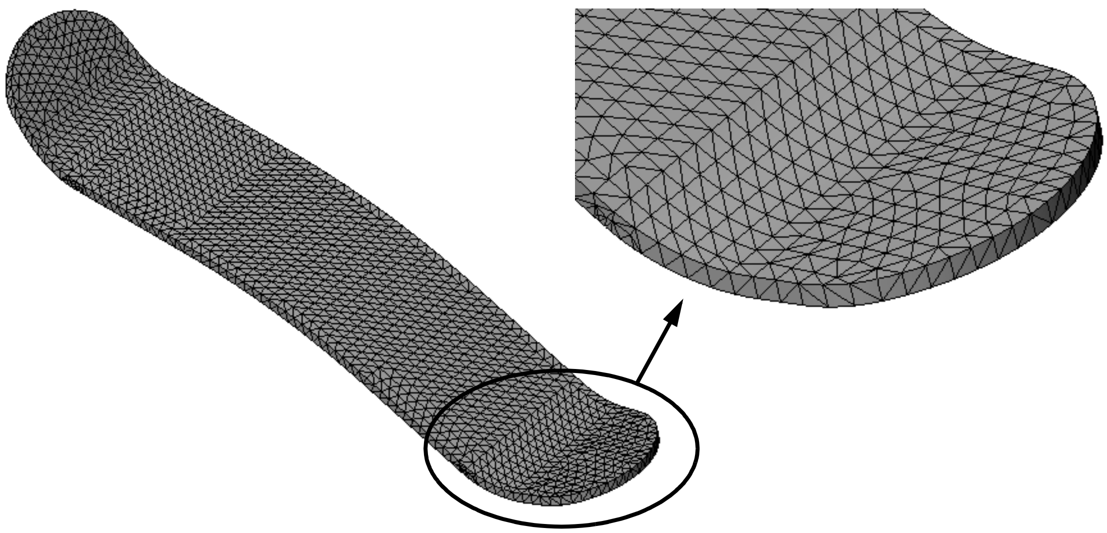
The basic concept behind Finite Element Analysis or FEA is to continue to divide the model into smaller elements so that we can determine the stress throughout the model. In addition to stress, FEA can determine a variety of other quantities such as deformation and strain.

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.

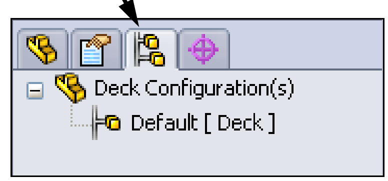
Task 1— Create A Configuration

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.

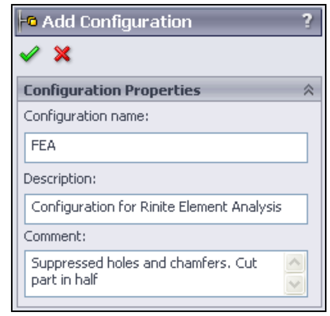
There is only a Default configuration until we add additional configurations.

ConfigurationManager



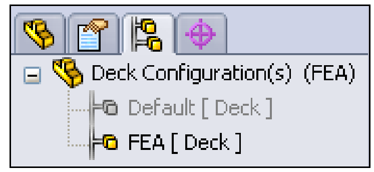
2 Right-click the top level icon 

and select **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.

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**
- ❑ Select the feature, and click **Suppress** on the **Context** toolbar.
- ❑ Click **Suppressed** in the Feature Properties dialog box.

1 Right-click the `Chamfer1` feature in the FeatureManager design tree and select **Suppress** from the **Context** toolbar.

2 The Chamfer feature will turn grey Chamfer1 to show that it is suppressed, and the Chamfer will disappear from the model.

3 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.

4 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 on the **Context** toolbar.

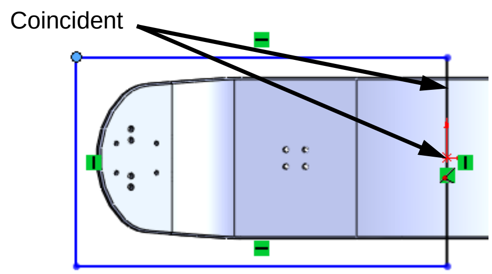
Mirror2 will also be suppressed because it is a child of the clearance holes.

Task 3 — Cut The Deck In Half

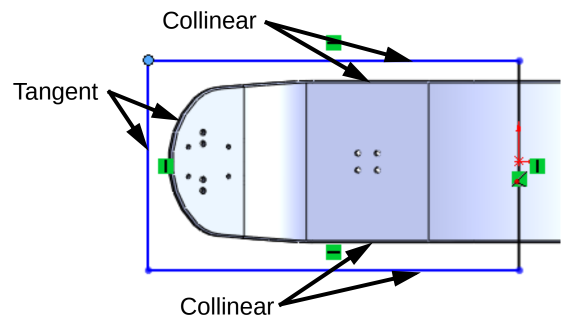
1 Open a sketch on the Top plane.

2 Change the Top view.

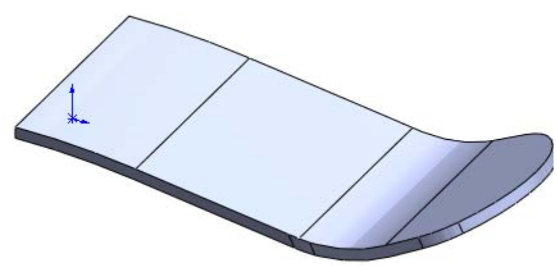
3 Sketch a rectangle as shown making the right side coincident with the Origin.



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.

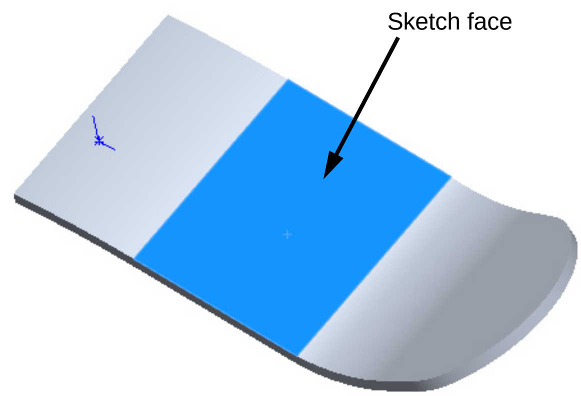
Split Lines

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.

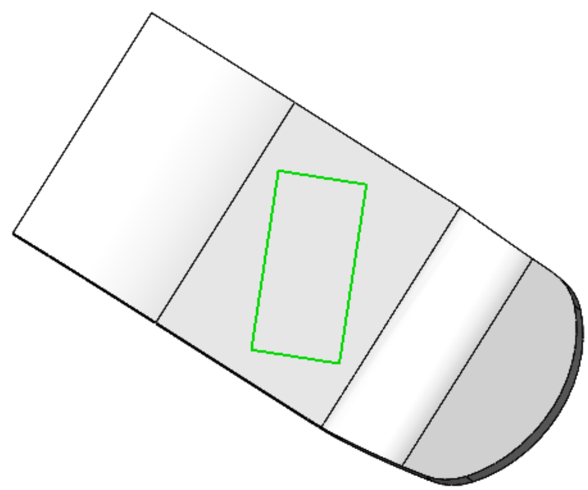
To insert a split line:

- ❑ Click **Insert Curve, Split Line**
- ❑ Click **Split Line** 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.

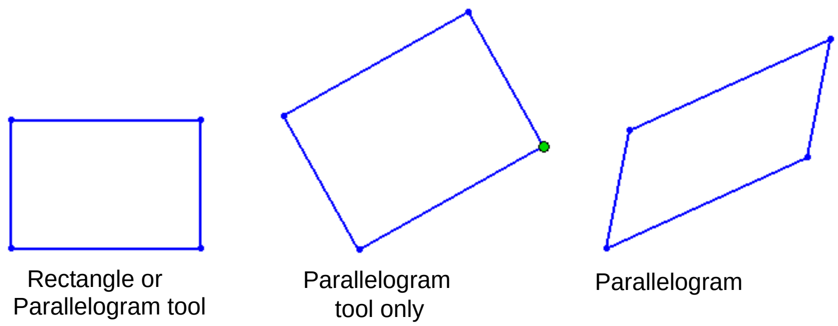


Parallelogram Tool

The **Parallelogram** tool is used to sketch four sided closed shapes that can be rectangles or parallelograms.

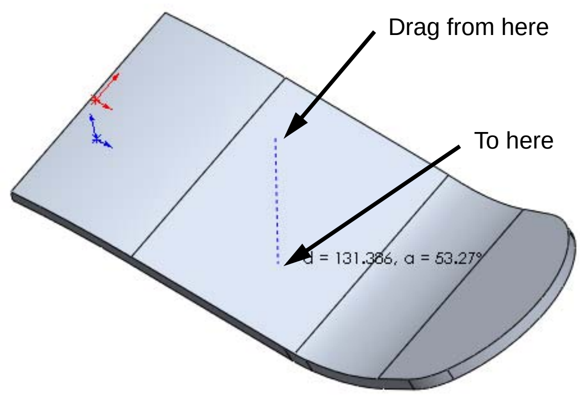
Both shapes have opposite sides parallel. The difference between the two is the angle between the sides. Rectangles can only have 90° corners. Parallelograms can be any angle less than 180°.

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.

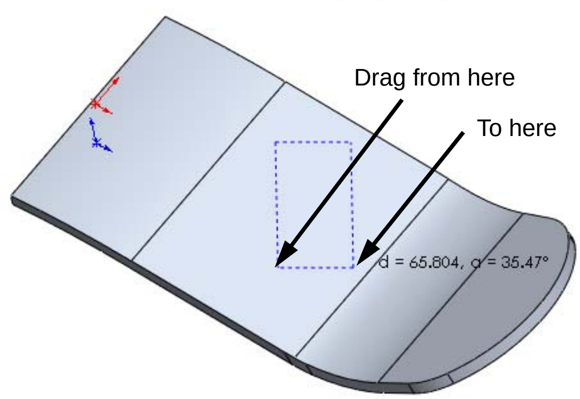


1 Click **Tools, Sketch Entities, Parallelogram** from the menu.

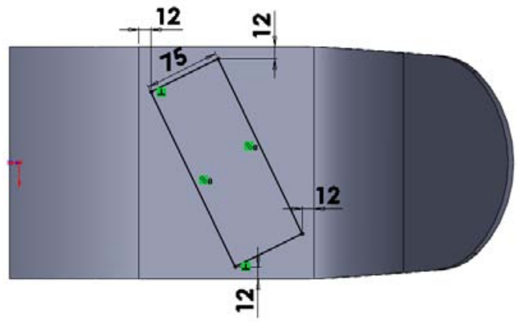
2 Click and drag to create one side of the Parallelogram.



3 Release the mouse button, then click again and drag sideways to sketch the adjacent side.



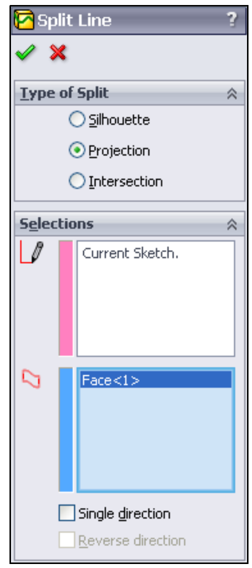
4 Dimension the sketch as shown. It should now be fully defined.



5 Click **Insert, Curve, Split Line** from the menus.

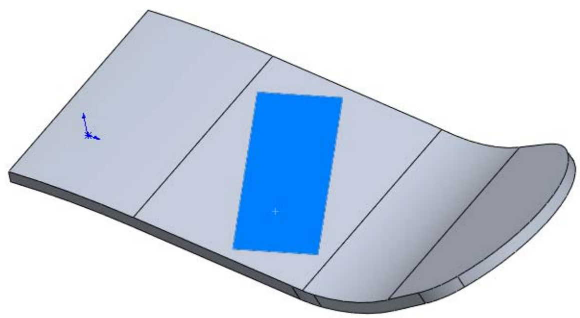
6 Select the sketch face.

7 **Projection** should be selected by default. This will project the sketch onto the selected face.



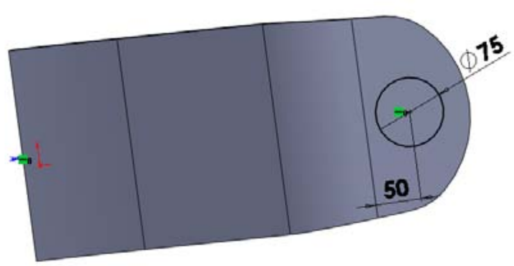
8 Click **OK**.

The parallelogram area is now a separate face. We will apply the rider's foot pressure to this area during analysis.



9 Create another split face where the `Truck` will attach to the `Deck`.

10 **Save** the part.

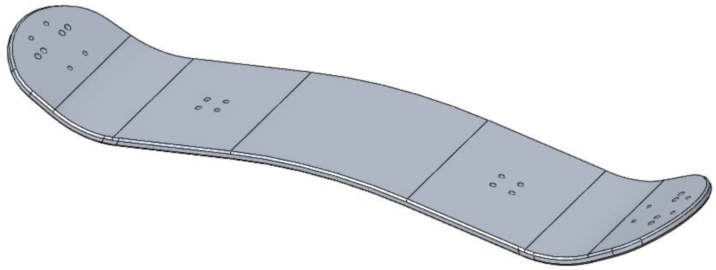


Task 5 — Review the Configurations

1 Click the  tab to change to the ConfigurationManager.

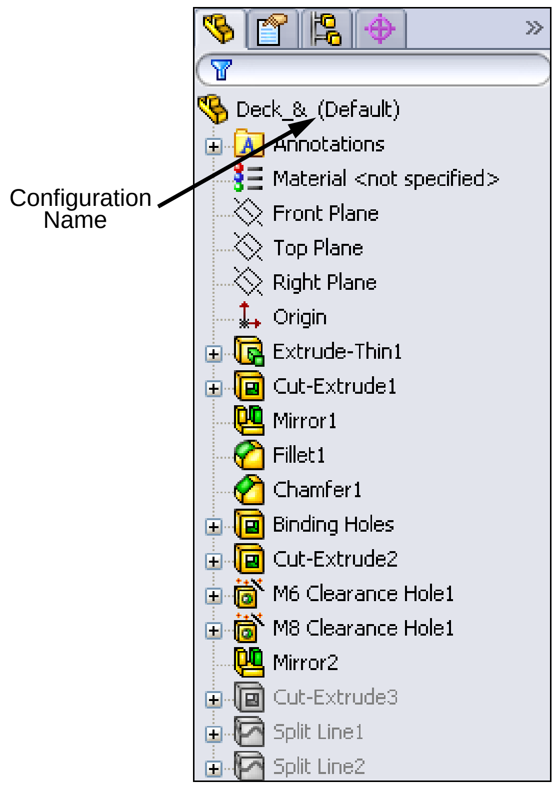
2 Double-click the Default configuration.

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.



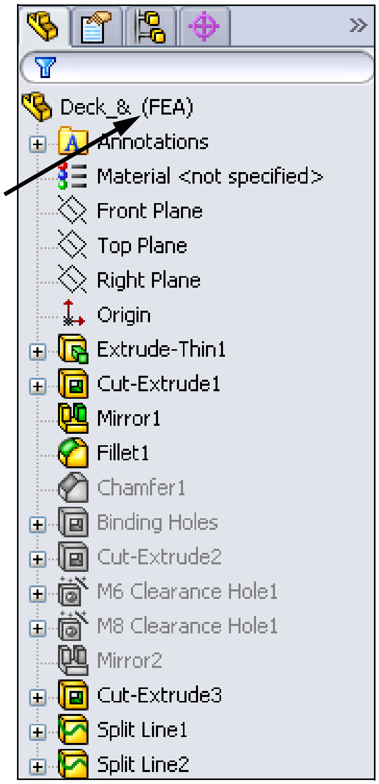
4 Click the tab to change to the ConfigurationManager, then double-click the `FEA` configuration.

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.

Configuration Name



Task 6 — Add A Material and Texture

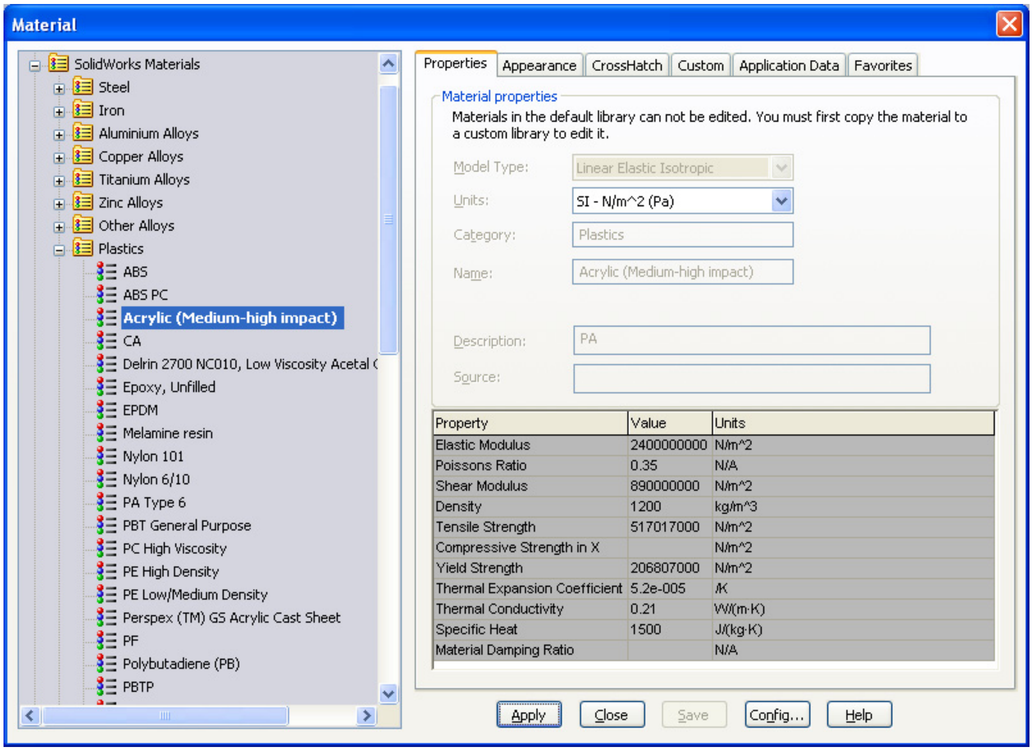
The next task is to add a material and texture to the model. The material will be used for weight calculations, the Bill of Materials that will be added to the drawings and to do the stress analysis. In the FeatureManager design tree, right-click **Material** and select **Edit Material**.

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.

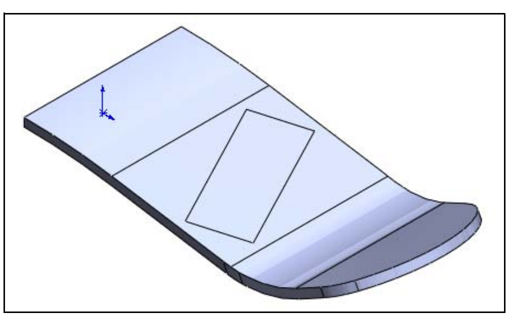
For a simple strength analysis we need the Elastic Modulus, Poissons Ratio, Tensile Strength and Yield Strength.

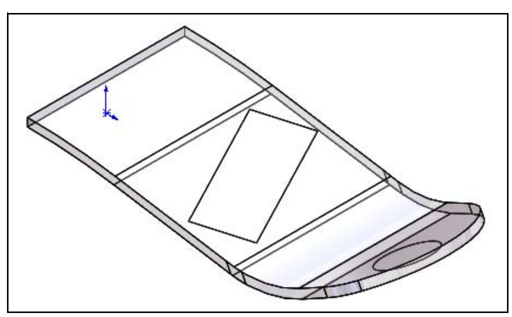


4 Click **Apply** and **Close** to accept this material.

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 Appearance 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 plastic


Acrylic (Medium-high impact)

Task 7 — Do A Simple Analysis

Using SolidWorks SimulationXpress, conduct a simple analysis of the Deck.

SolidWorks SimulationXpress

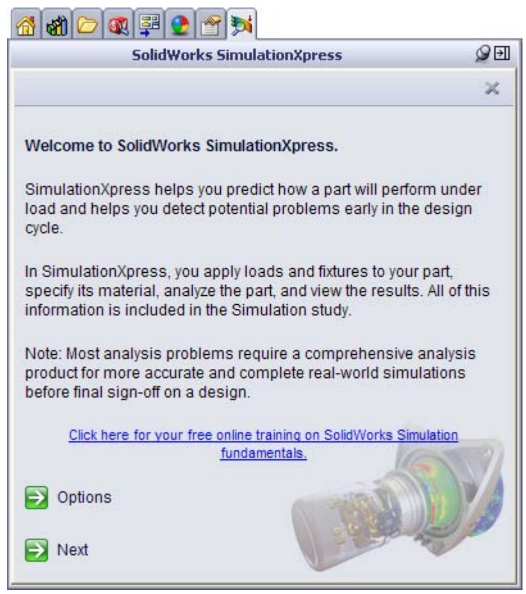
SolidWorks SimulationXpress is a first pass stress analysis tool for SolidWorks users. It helps you judge whether you part will withstand the loading it will receive under real-world conditions.

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.

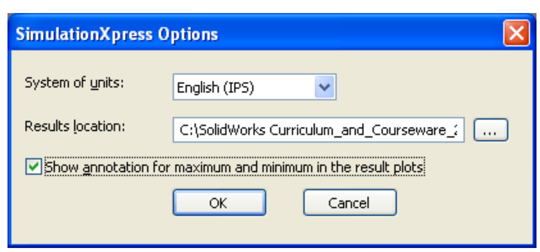
The wizard has several tabs showing the steps we will take to do the analysis. As each step is completed, a circle with a green check will appear on the tab indicating that the action for that step is complete.



Note: There is a link to additional training on SolidWorks SimulationXpress Welcome tab.

Options

The **Options** dialog contains settings for the System of units and Results location.

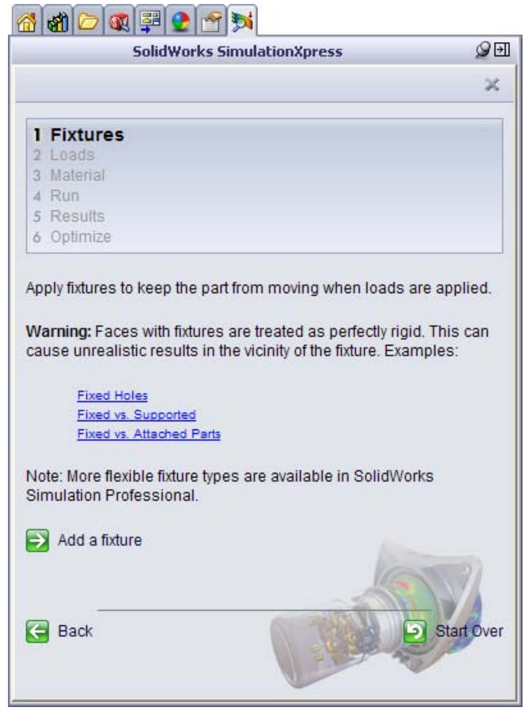


2 Click **Options.** Set the units to **English (IPS)** and select **Show annotation for maximum and minimum in the stress plots**.

3 Set the path to the **Results location** to C:\SolidWorks Curriculum and Courseware_2010\ Mountainboard Design Project \Lessons\Lesson05.

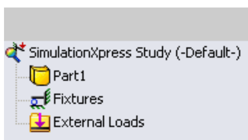
4 Click **OK** and then **Next**.

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.

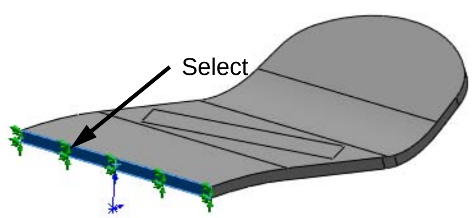


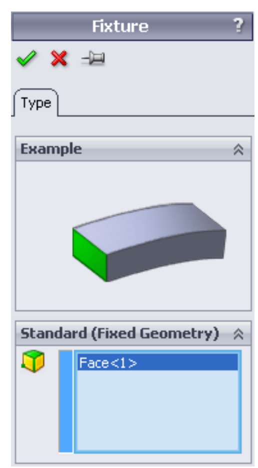
Fixtures

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.

1 Click **Add a fixture**.

2 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.

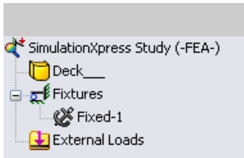




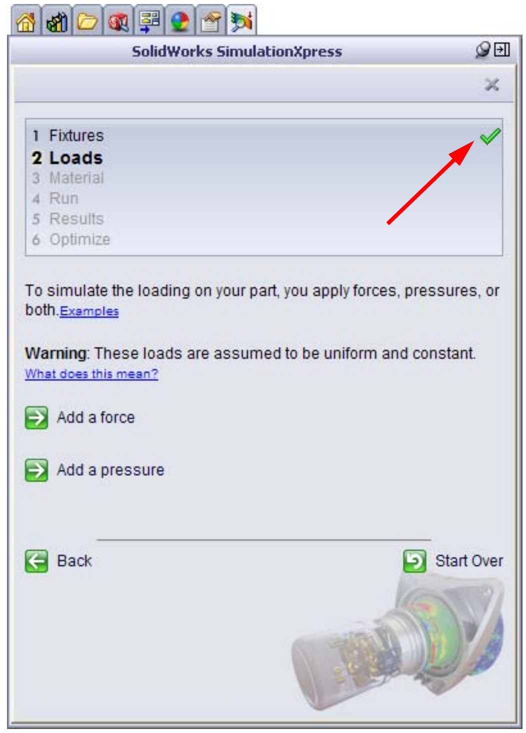
3 Click .

4 Click **Next** in the wizard.

The restraint is added as `Fixed-1`. We could add 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.

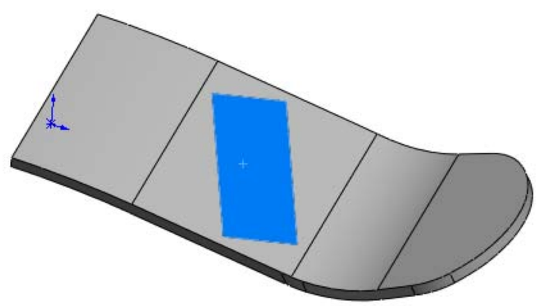


Loads

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.

1 Click **Add a force**.

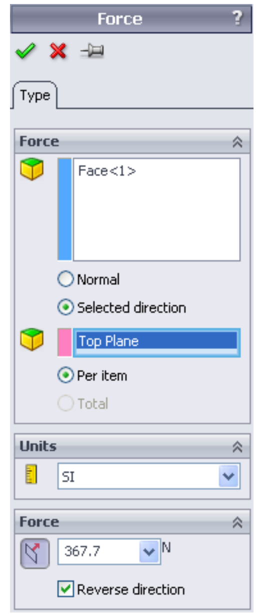
2 .Select the split face representing the rider's foot



3 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.

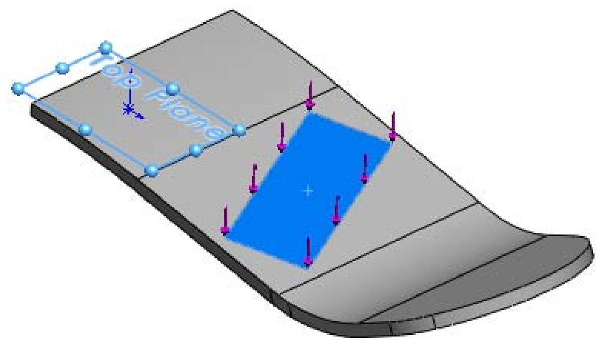
4 Select **N** from the pull down list and type **367.7** for the value of the force



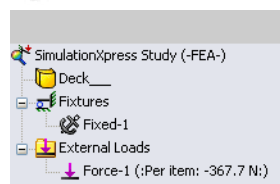
.

Note: We had to input the force in Newtons. One kilogram force is the equivalent of 9.807 Newtons.

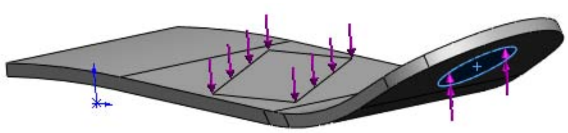
5 Examine the model. Make sure the arrows are pointing toward the board. If they point away from the board, select **Reverse direction**.

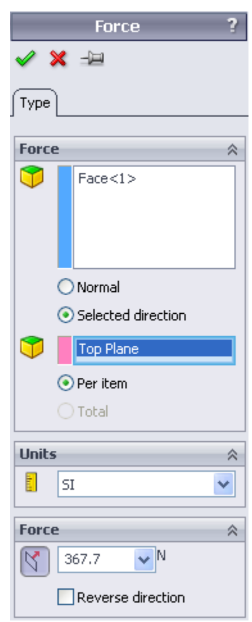


6 Click  . The load representing the weight of the rider has been added as Load1.



7 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.





Note: There are additional force components to both the force applied by the rider and the truck that act along the centerline of the Deck. We will ignore those for this preliminary check of the model.

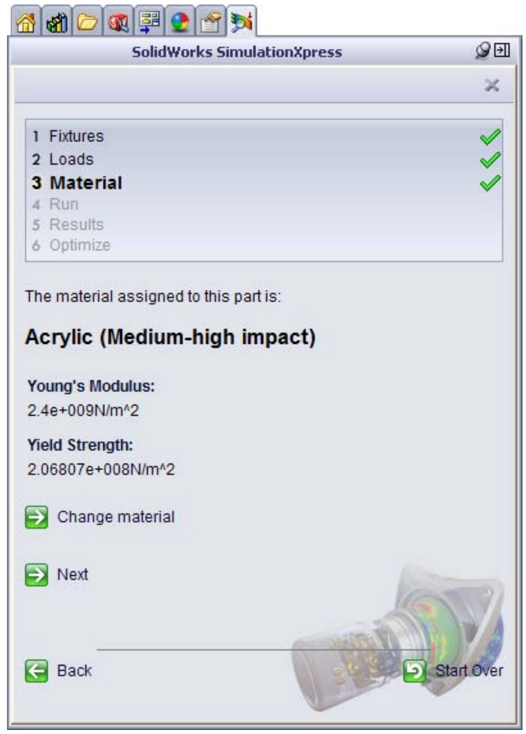
8 After applying the second force, click **Next**.

Material

The next phase is selecting the Material. You can choose from libraries of standard materials or add your own.

1 Because we had applied a material to this part in SolidWorks, the material will already be select in the wizard. Since we have a material applied, there is already a green check mark on the Material tab.

2 Click **Next**.

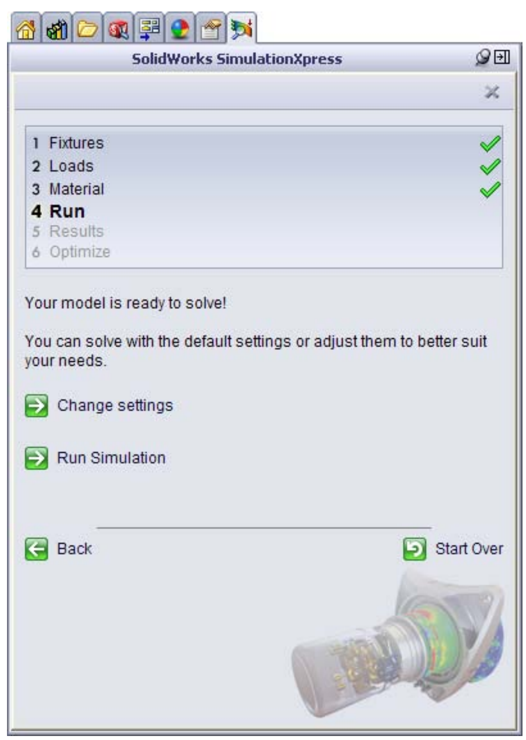


Run

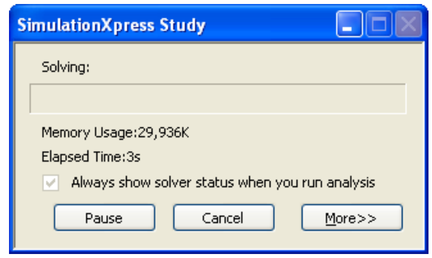
SolidWorks SimulationXpress prepares the model for analysis and then it calculates displacements, strains, and stresses.

We could change the mesh size by selecting Change settings. As this is our first analysis, we will use the default settings.

1 Click **Run Simulation** to begin the solution.



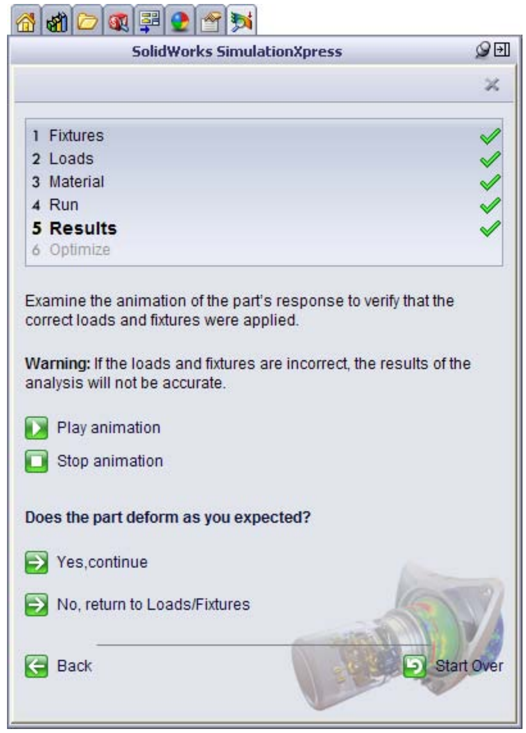
2 Click **Run** to begin the analysis. A status window appears. The stages of the analysis process are displayed with elapsed time.



Note: As this is a simple analysis, the progress window may not be seen as the solution takes very little time.

Once the solution completes, the part will be ani am tat ed to show how it deforms.

3 Click **Stop animation**.

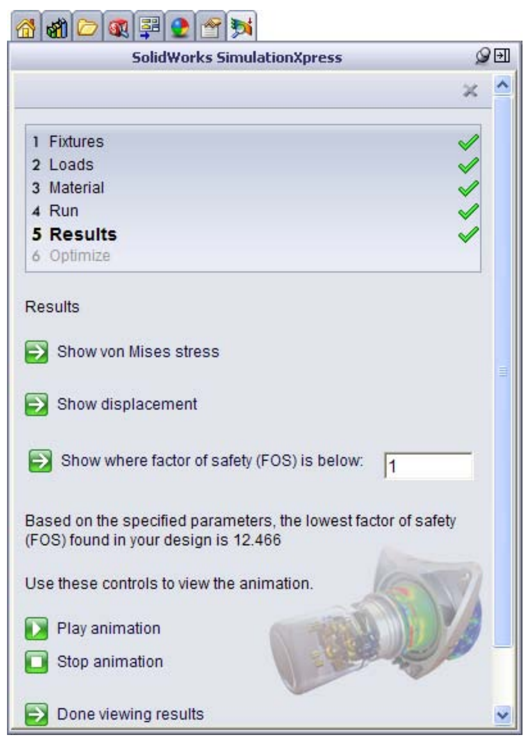


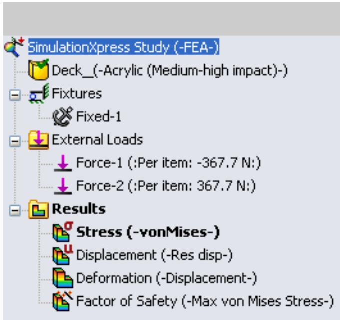
Results

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 double-click the result in the Results folder or select it in the SimulationXpress wizard.





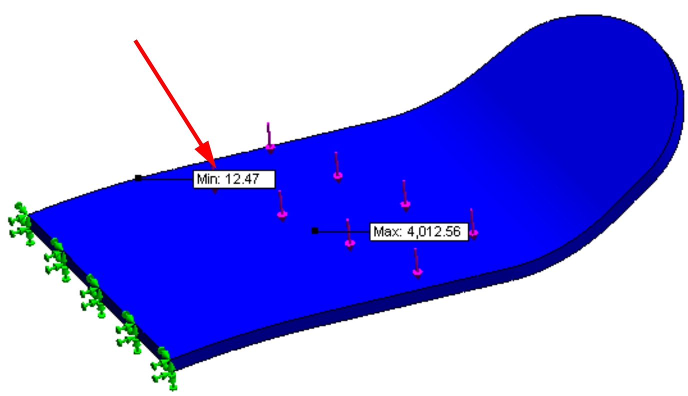
Factor of Safety

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.

1 Select **Show where factor of safety (FOS) is below: 1** in the wizard.



2 The initial analysis shows a Factor of Safety of **12.47** as shown in the callout. This indicates that for the deck is over 12 times stronger than it needed to carry the applied load.

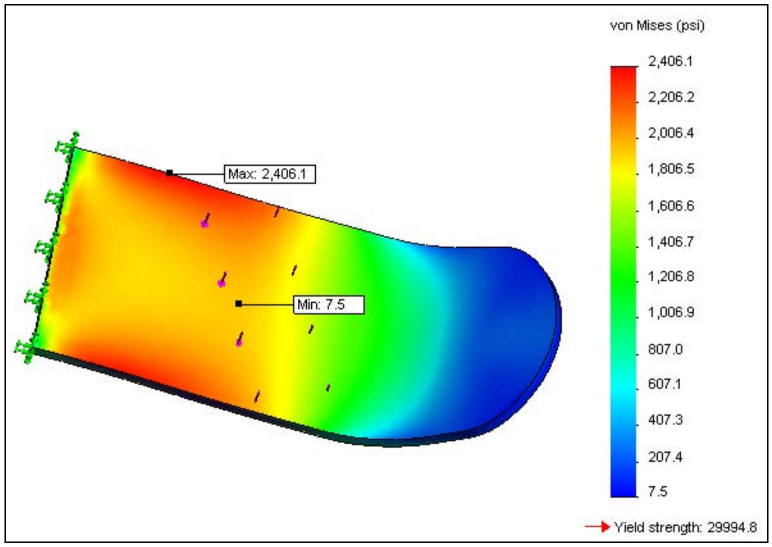
Does this mean that we can reduce the strength of the deck by making it thinner or changing material? No, not yet. Remember that this was just a preliminary analysis for the very simple loading condition of a rider standing on the deck with weight equally distributed. Before making design changes, we 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 2,406 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.

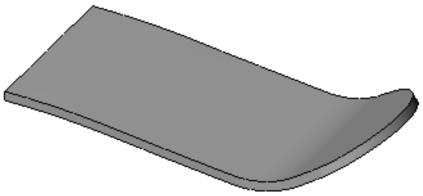
Engineering Notation

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:

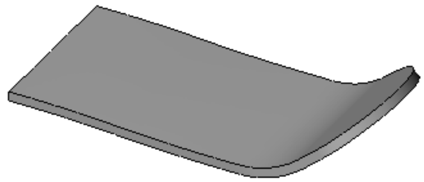
Number	Scientific Notation	Engineering Notation
1,234	1.234e+003	1.234e+003
12,345	1.2345e+004	12.345e+003
123,456	1.23456e+005	123.456e+003
1,234,567	1.234567e+006	1.234567e+006

❑ Deformation

This is best seen by viewing the animation.



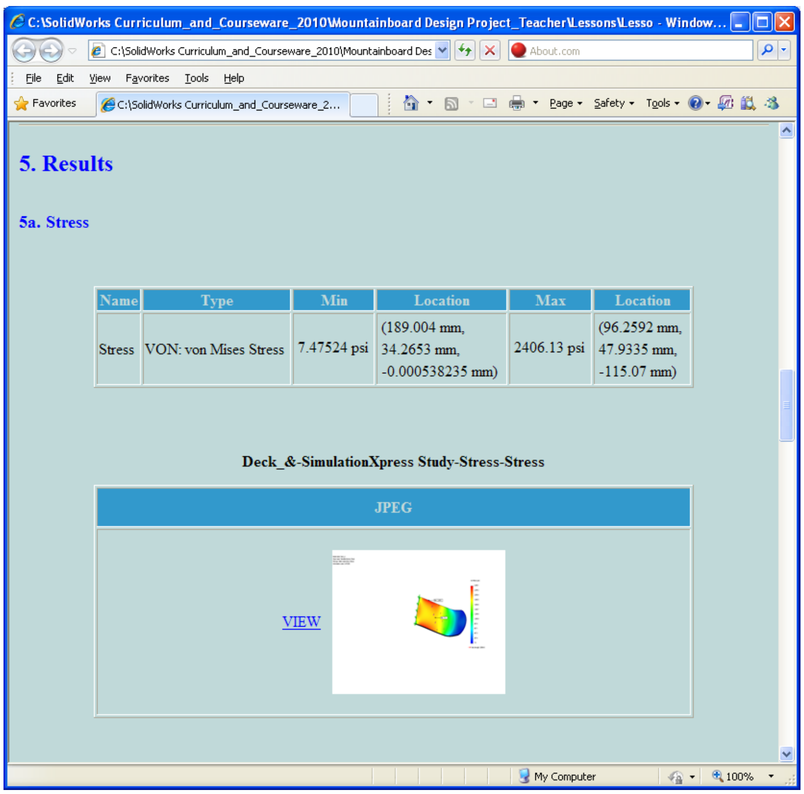
Deformed



Undeformed

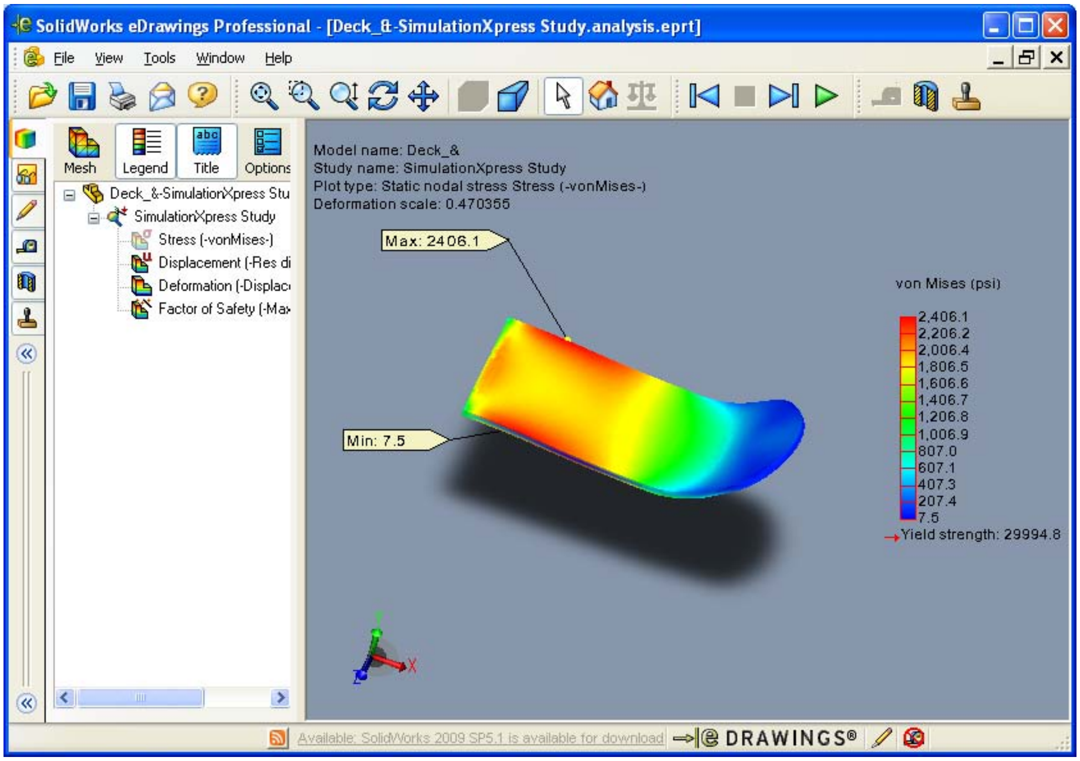
❑ HTML Report

HTML reports can be viewed in any web browser.



- ❑ **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 double-click 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

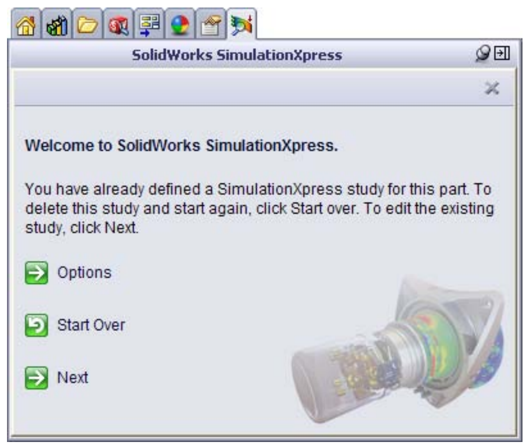
TIP: To select more than one item in the FeatureManager design tree you can either hold down the Control key as you select multiple items or if the things you want to select are sequential you can select the first item, then hold down the Shift key and select the last item. These are both Windows techniques 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**.



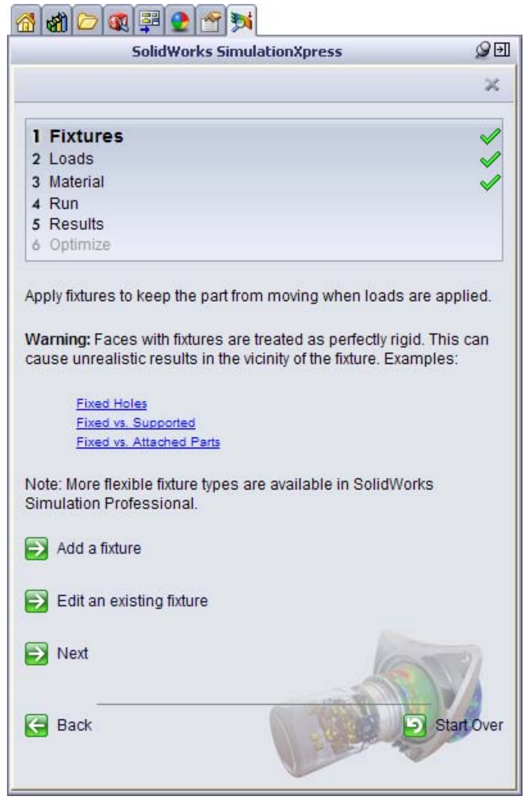
6 Several items; **Fixtures**, **Loads** and **Material** still have green check mars as these items have not changes.

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.

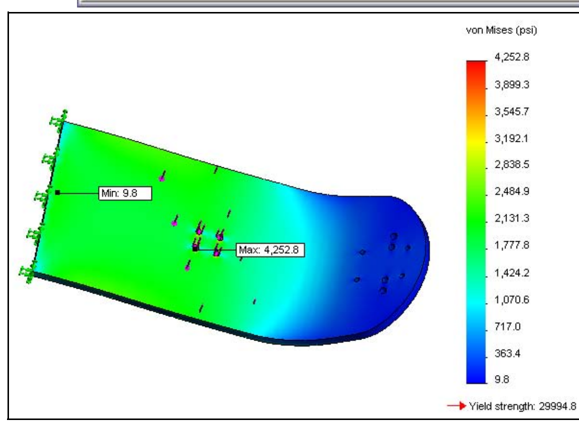
9 Stop the animation and double-click the Factor of Safety plot.

The holes have reduced the Factor of Safety from 12.47 to 7.05.



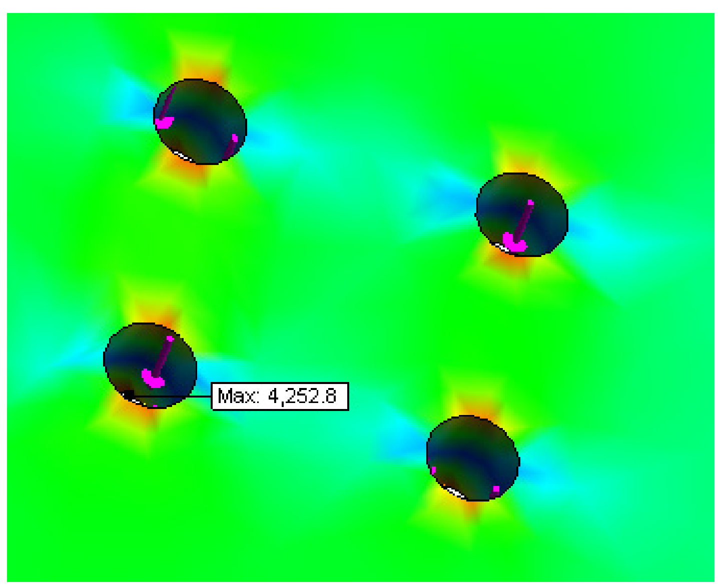
10 Double-click the Stress results.

The maximum stress in now4,253psi.



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?

Answer: 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.

Exercises and Projects — Thin Features

Exercise 16: Thin Bracket

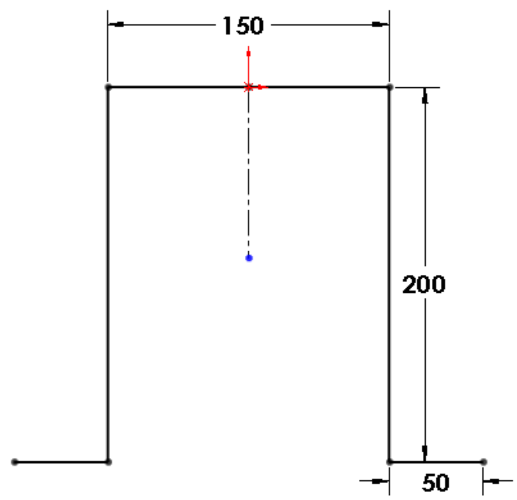
Create the part shown.

The exercise reinforces:

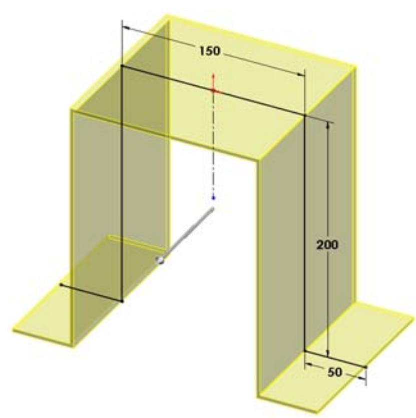
- Thin features
- Mirror features
- Apply material to a part.

1 Create a new part using the Part_MM template.

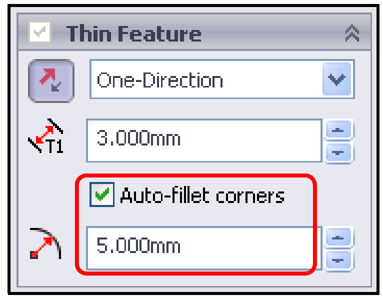
2 Create a sketch on the front plane.
The sketch is symmetrical.



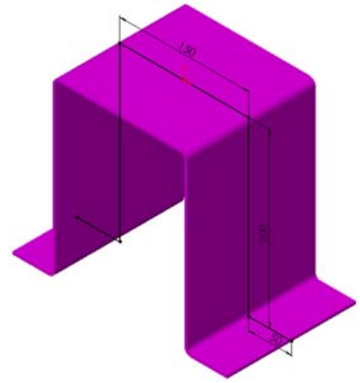
3 Extrude the sketch as a thin feature.
Extrusion depth is **150mm**, using **Mid Plane**.
The material thickness is **3mm** to the outside of the sketch.



4 Add fillets to the extrusion.
Right-click the base feature `Extrude-Thin1` and select **Edit Feature**.
Select **Auto-fillet corners** and set the radius to **5mm**.



5 Click **Detailed Preview** to see the effect of the fillet. Click **OK** to finish the edit.

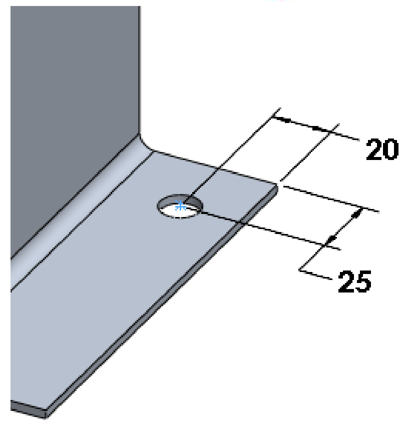


6 Use the **Hole Wizard** to add a M12 Clearance Hole.

7 Mirror the hole around the Front plane.

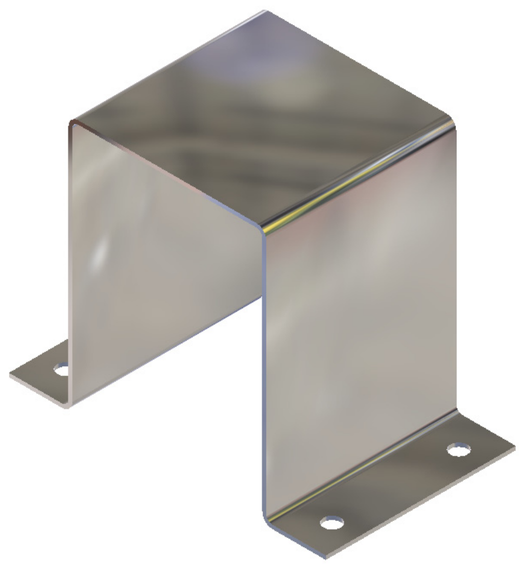
8 Mirror the two holes about the Right plane.

9 Add the material Chrome Stainless Steel to the part.



10 If your computer supports RealView graphics, click 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.

____

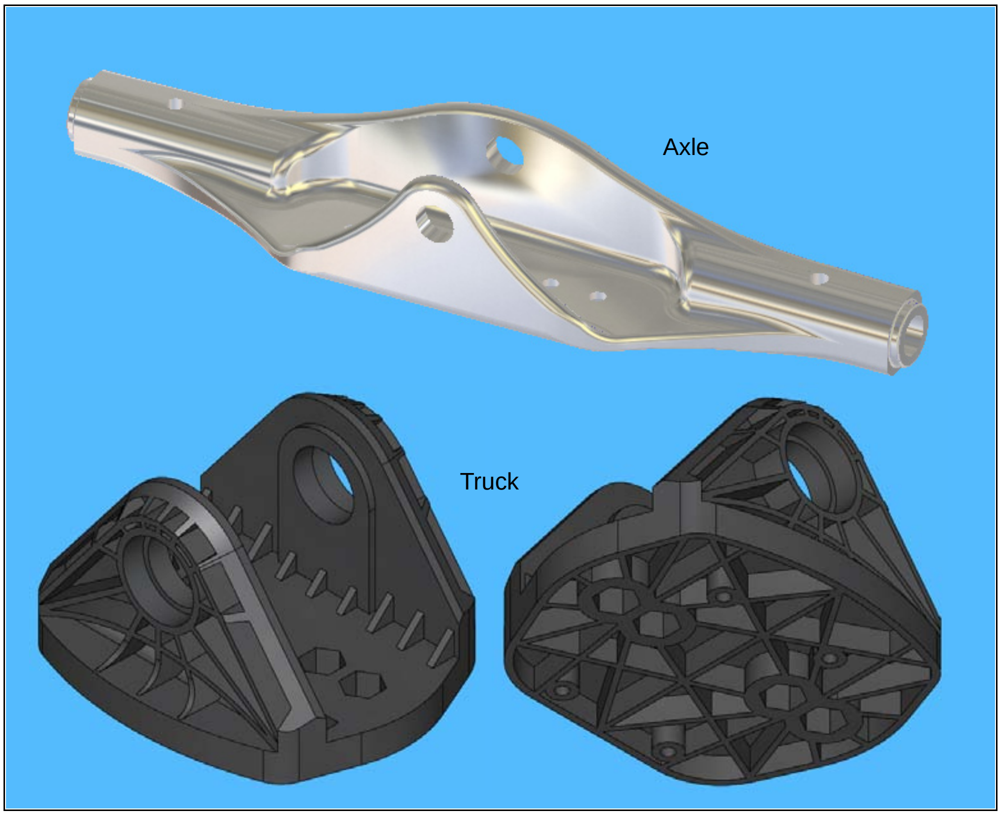
Lesson Summary

- 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

6

Lesson 6: Multibody Parts — The Axle and Truck

- ❑ 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

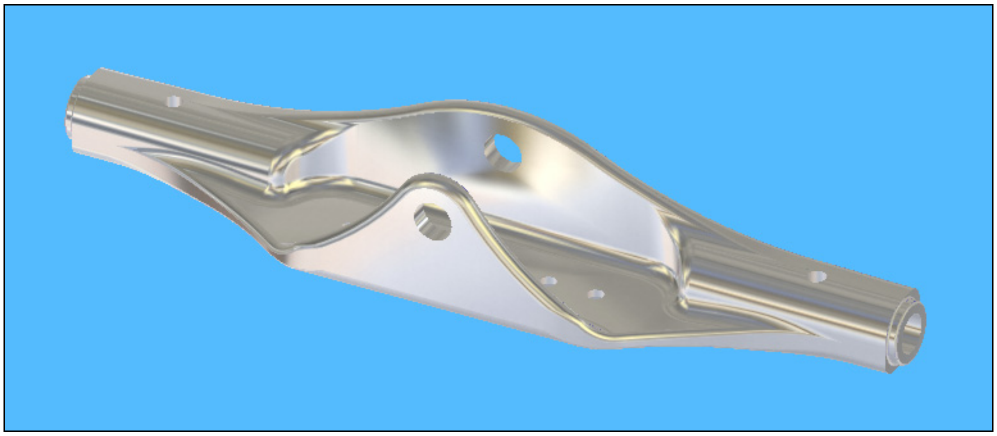
The most direct way to create a multibody solid is by clearing the **Merge result** check box for specific boss and cut features.

- 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.



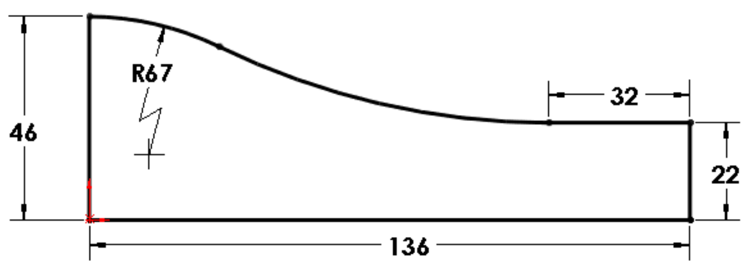
Design Intent

- 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).

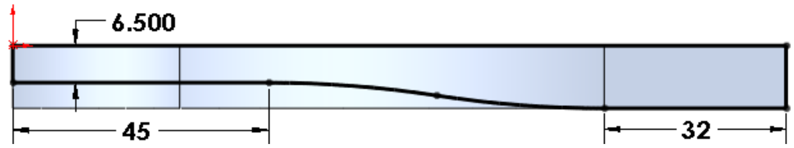
1. Create a new part using the `Part_MM` template.
2. **Save** the new part with the name `Axle`.
3. Create the following sketch on the `Front` plane.



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.

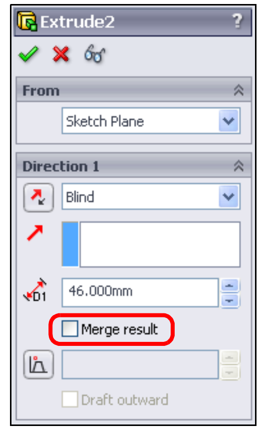


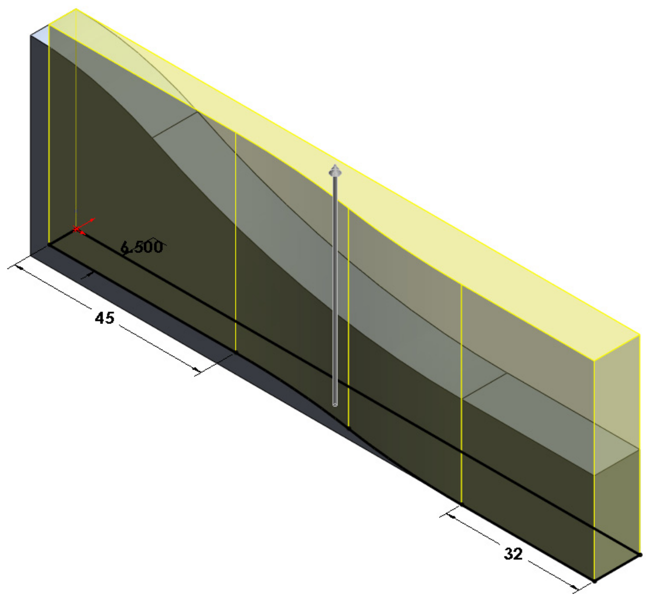
6 Both arcs are **Tangent** to each other and the lines they are attached to. The arcs also have an **Equal** relationship.

7 Click **Extrude** .

8 **Extrude** to a **Blind** depth of **46mm**.

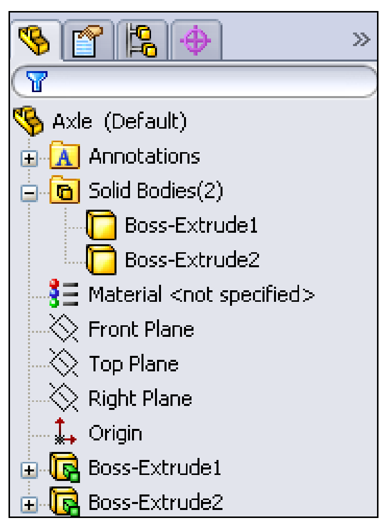
Clear **Merge result**. Click **OK**.



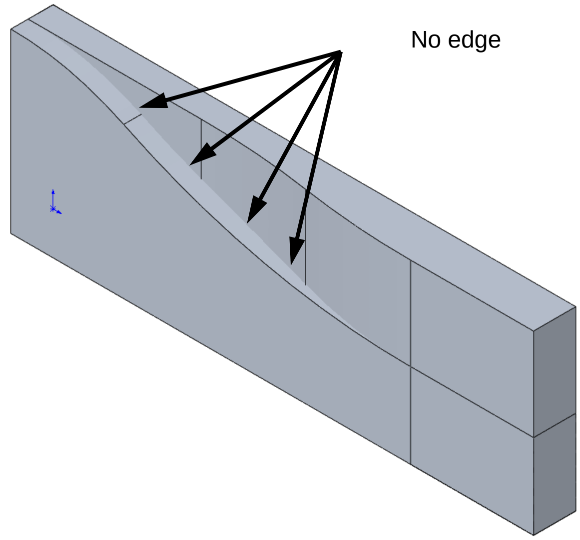


9 Examine the results. The FeatureManager design tree shows two solid bodies. Because the results of the two extrusions were not merged, each remains separate.

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.

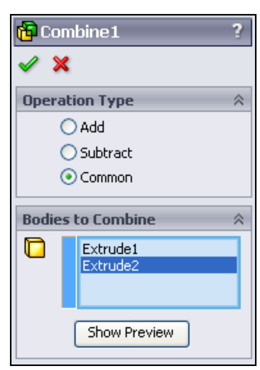
To **Combine** solid bodies:

- Click **Combine** on the Features toolbar
- Click **Insert, Features, Combine**

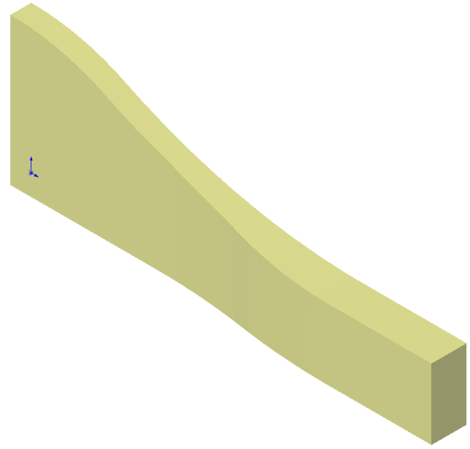
1 Click **Combine** on the Features toolbar.

2 Select the two solid bodies.

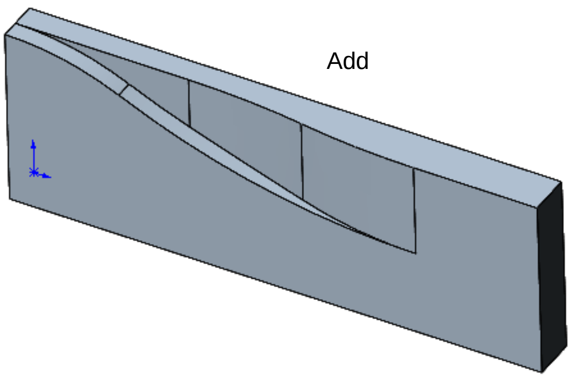
3 Select **Common**. This will create a single solid body from the volume that is common to the two bodies.

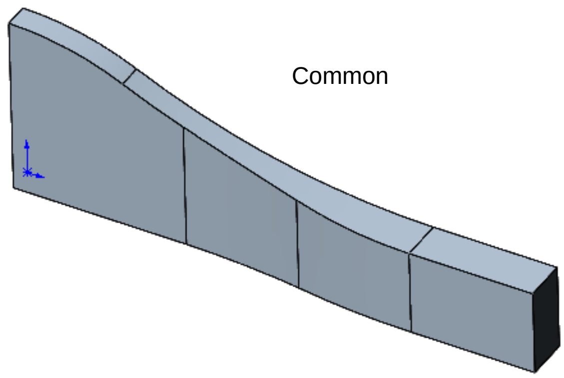


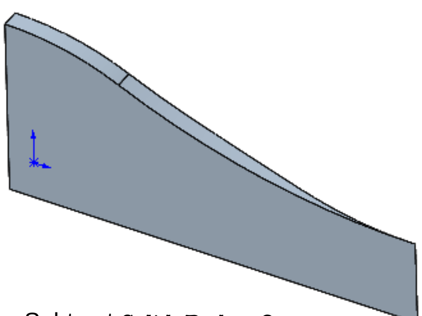
4 Click **Show Preview**. The preview shows the common volume



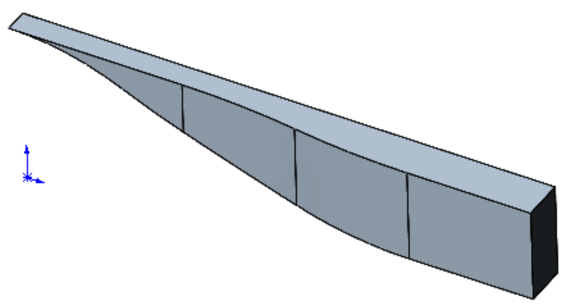
5 There are four possible choice for combining the two solidbodies.







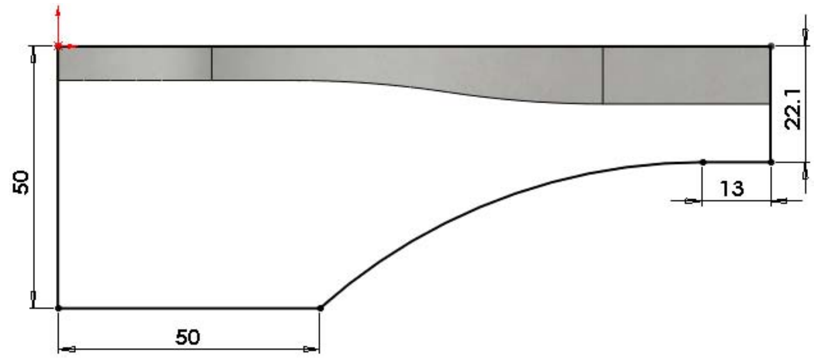
Subtract Solid Body <2>
from Solid Body <1>



Subtract Solid Body <1>
from Solid Body <2>

6 Click .

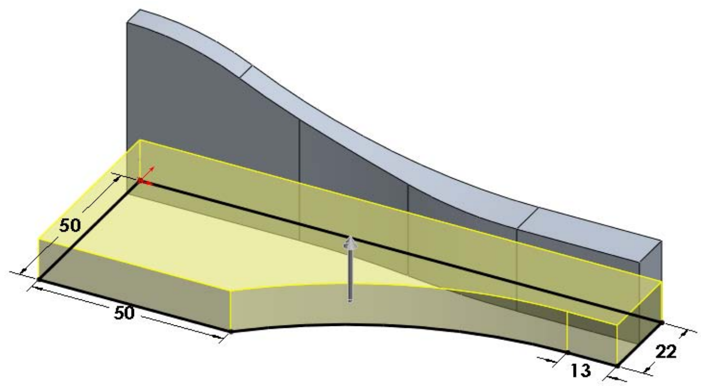
7 Create the following sketch on the `Top` plane.



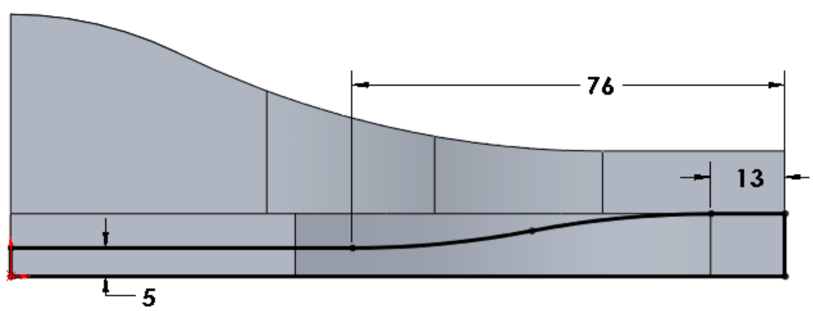
8 **Extrude** the sketch to a depth of **11mm**.

Clear **Merge results**.

Click **OK**.



9 Create the following sketch on the `Front` plane. Both arcs are of equal radius.

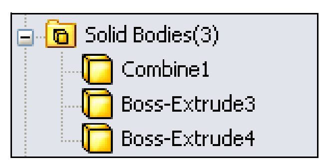


10 **Extrude** the sketch to a depth of **50mm**.

Clear **Merge results**.

Click .

11 Examine the Solid Bodies folder. There are now three solid bodies. We will combine all the bodies to make a single solid, however we need to do it in two steps. First, `Boss-Extrude3` and `Boss-Extrude4` will be combined to get the common volume. This is the same as was done with `Extrude1` and `Extrude2`. Next, the two combined volumes will be added together.



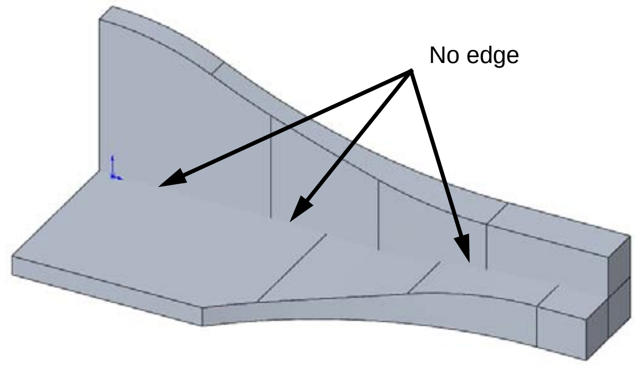
12 Click **Insert, Features, Combine**.

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.

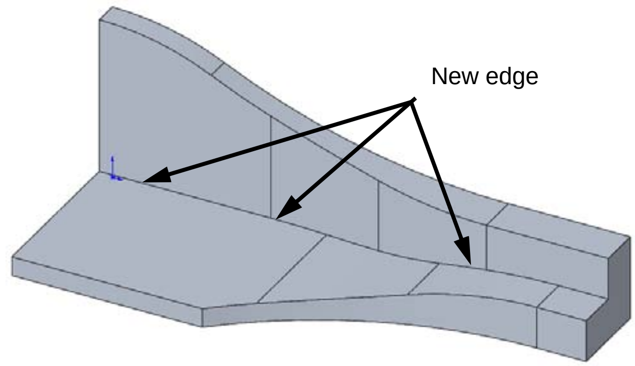


17 Click **Combine** .

18 Select **Add**, then select the two solid bodies `Combine1` and `Combine2`.

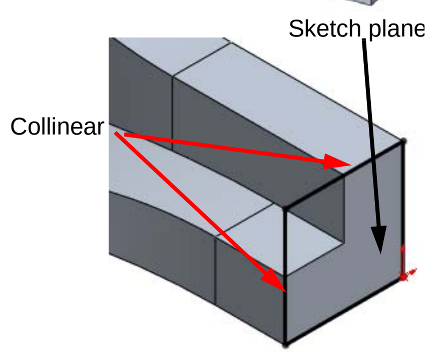
19 Click .

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`.

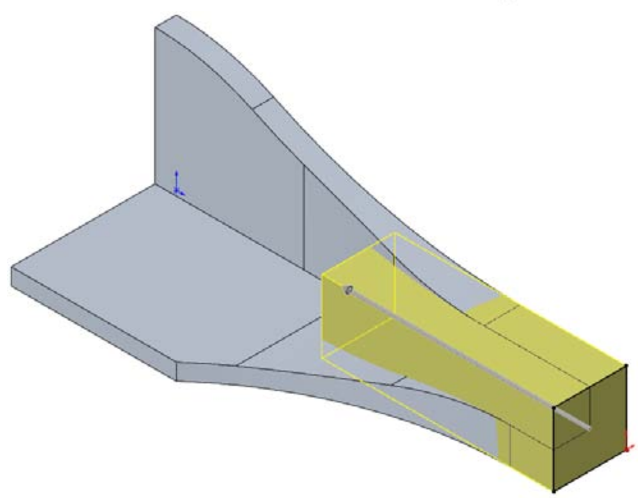


21 Create a new sketch on the end face shown.

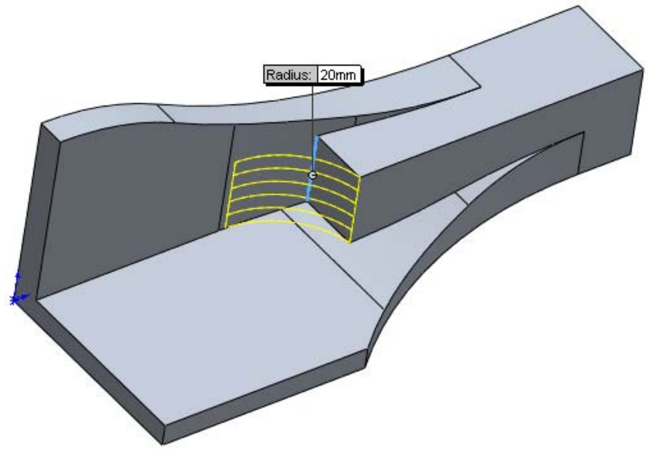
22 Sketch a rectangle and relate it to the existing edges of the model with **Collinear** relationships.



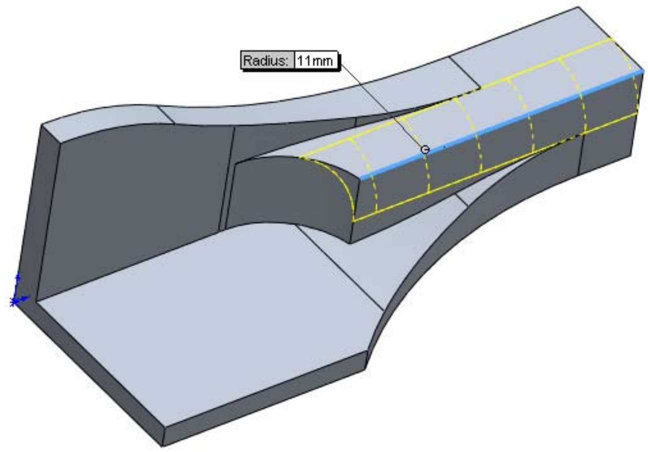
23 Extrude the sketch to a depth of **70mm**, into the model.
Select **Merge results**.



24 Create a **20mm** fillet to fill in the area behind the last extrusion.

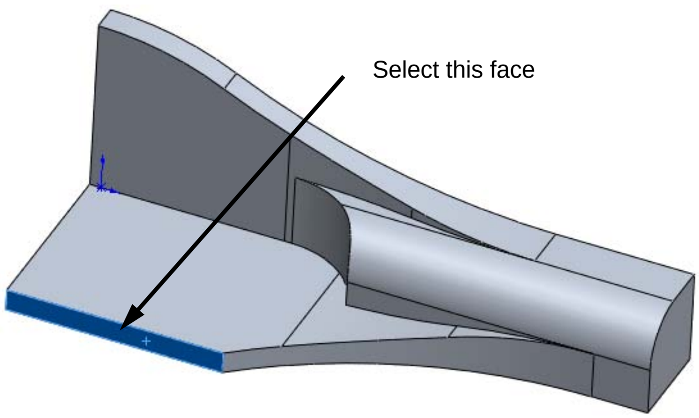


25 Add an **11mm** fillet to the edge shown.

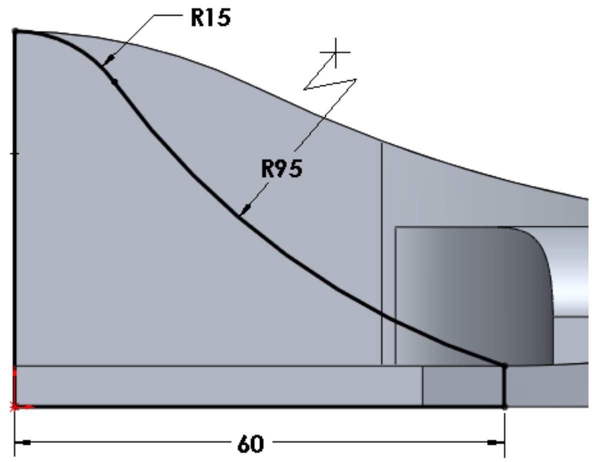


26 Create a sketch on the face shown.

27 Reorient the model to the `Front` view.



28 Create the sketch as shown.

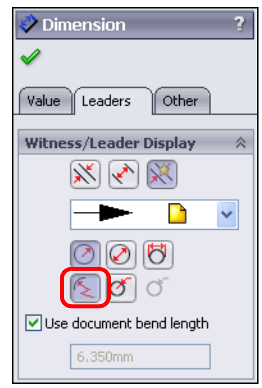


Foreshorten Radius

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.

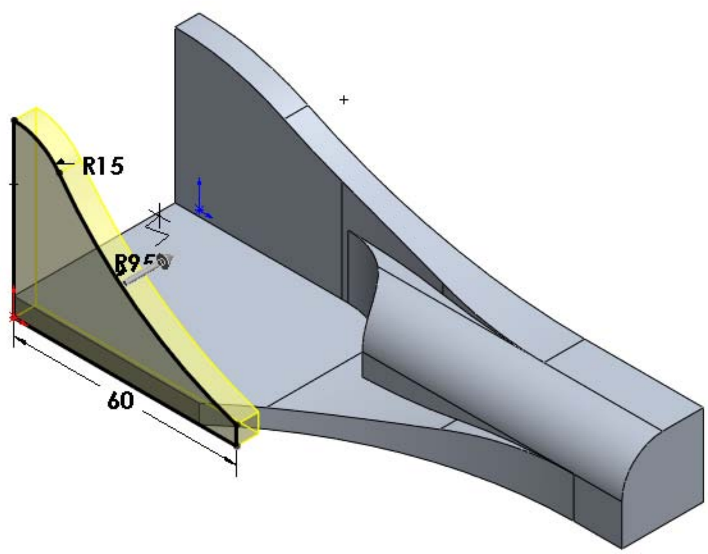
1 Select the 95mm radius dimension.
Select the Leader tab in the PropertyManager.



2 Select **Foreshorten radius**, then click  .

3 The radius dimension is now displayed with a broken leader

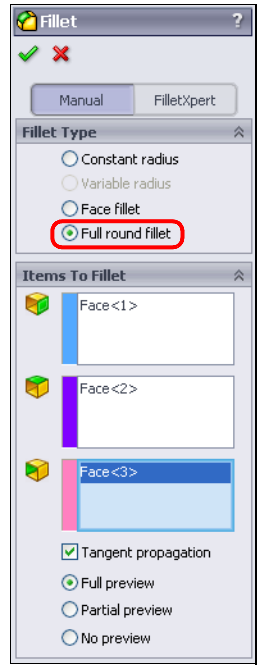
4 **Extrude** the sketch *into* the model **6mm**.



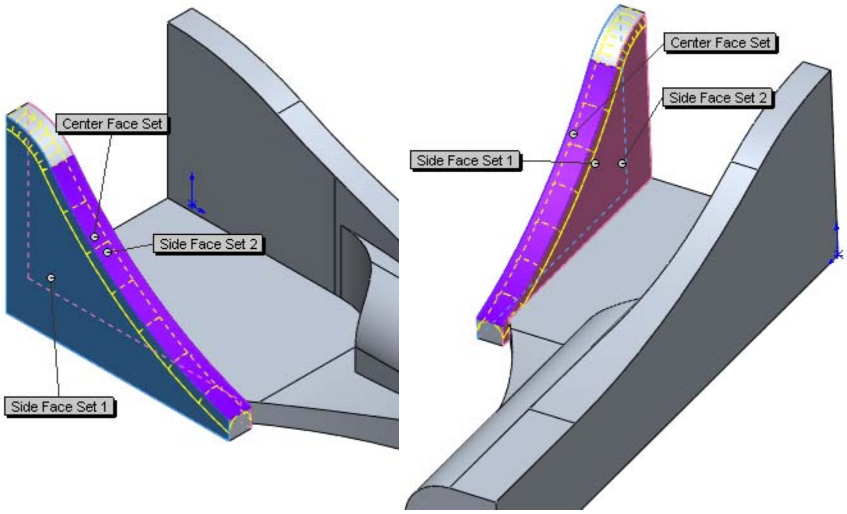
Full Round Fillets

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

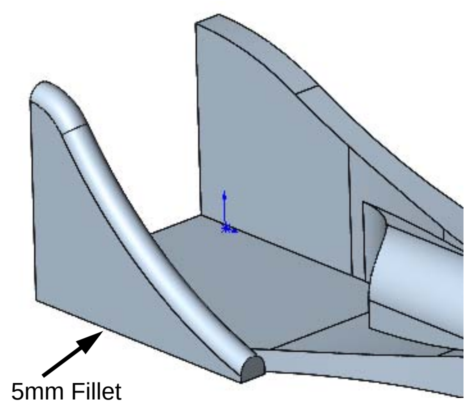
Face <2> (Center Face) Indigo

Face <3> Magenta

---

6 Click .

Task 2 — Trim the Edge



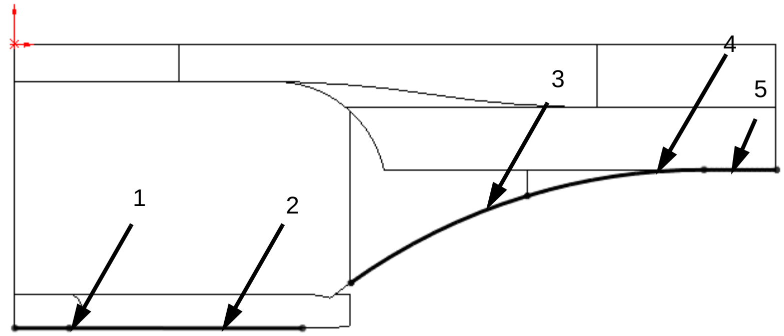
The last extruded feature we created is too long and needs to be trimmed back to the existing edge of the previous extrusion.

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.

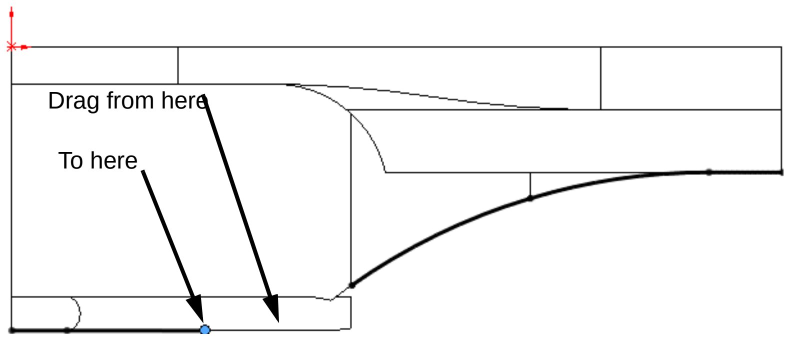
5 Click **Convert Entities**  on the **Sketch** toolbar.

6 Select the five edges shown.

7 Click .

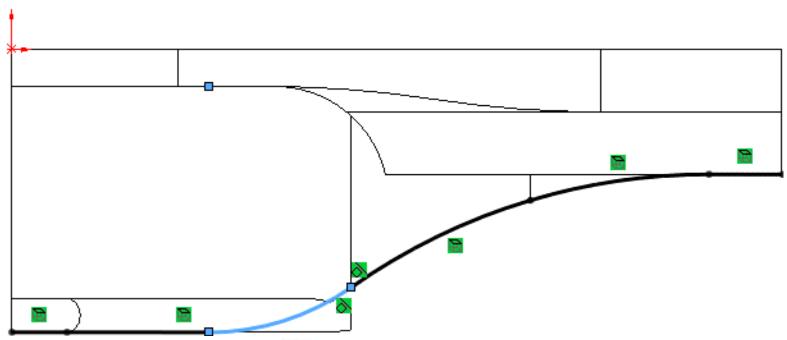


8 Drag the endpoint of the line shown to shorten it. The exact length is not important, we are just leaving enough room to draw the next arc.



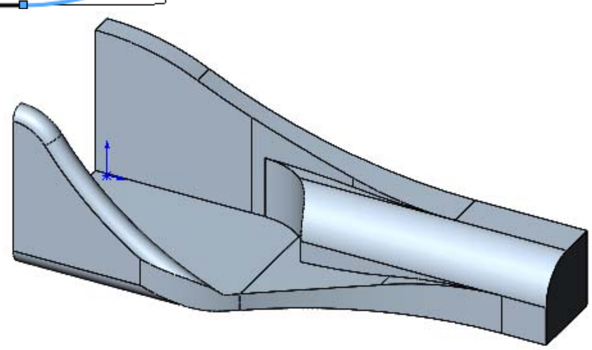
Note: Even though the endpoint is black, indicating fully defined, it can be moved.

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.

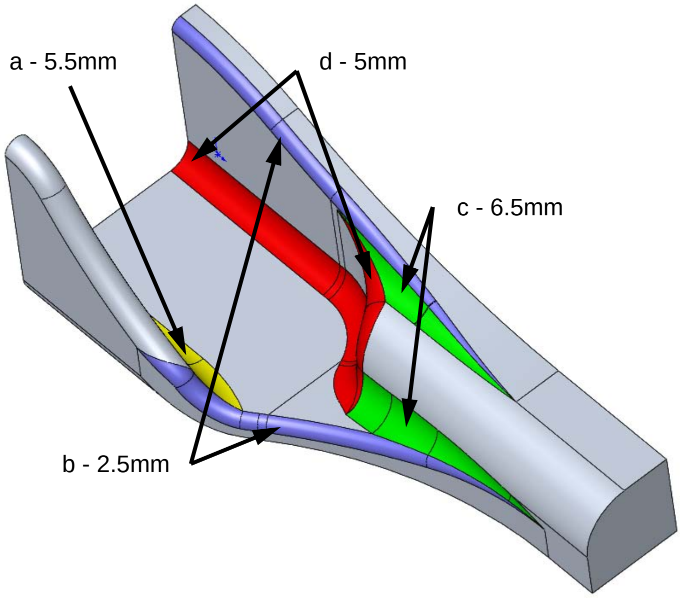


Task 3 — Filleting The Edges

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.

1 Add constant radius fillets to the edges shown. Do them in the following order:

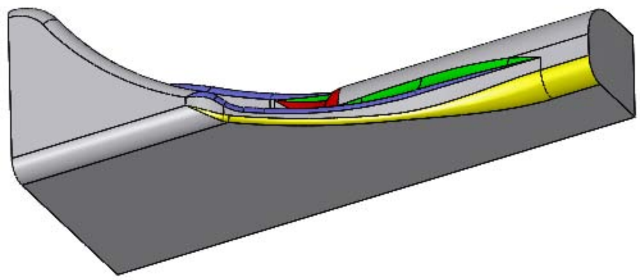
a) 5.5mm
b) 2.5mm
c) 6.5mm
d) 5mm



Note: The fillets are shown in color for easier identification.

Variable Radius Filleting

The bottom edge of the axle will need a fillet that changes radius along the length of the edge.

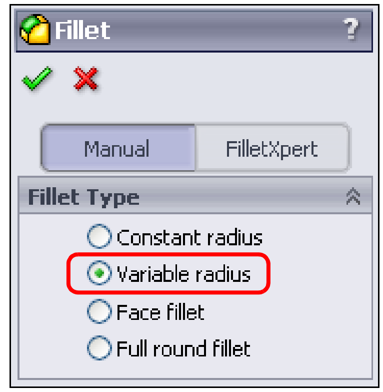


Variable radius fillets are defined by specifying a radius value for each vertex along the filleted edges and optionally, at additional control points along the edges. Variable radius control points operate as follows:

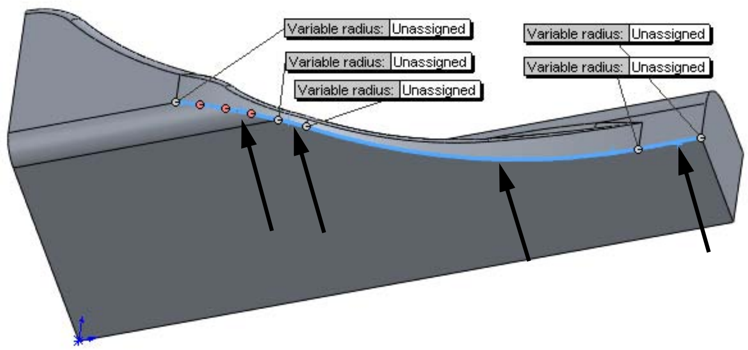
- The system defaults to three control points, located at equidistant increments of 25%, 50%, and 75% along the edge between the vertices. You can increase or decrease the number of control points.
- You can change the position of any control point by changing the percentage assigned to that control point. You can also drag any control point, and its assigned percentage will update accordingly.
- 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.



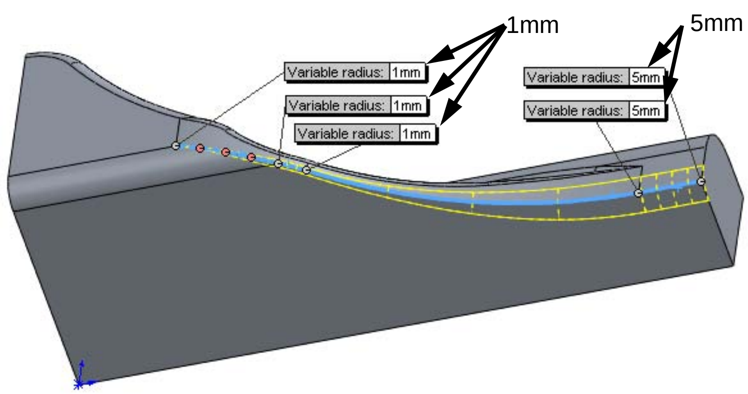
4 Select the four edges shown.



5 Callouts will appear at each end of the individual line segments. The last line segment selected will also show the intermediate points.

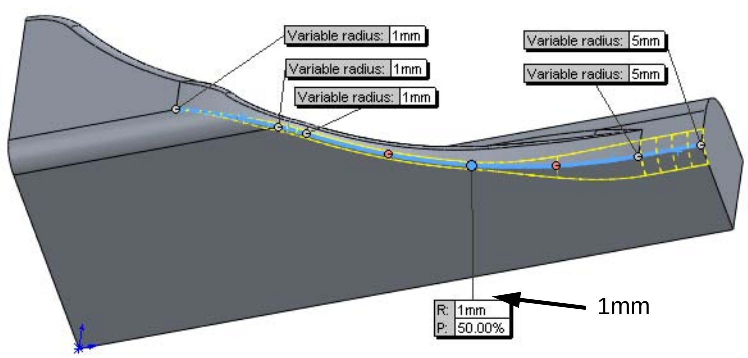
6 Individual radius values can be entered using the callouts.

7 Double-click each callout and enter the radius values shown. There are only two radius values, **5mm** and **1mm**.



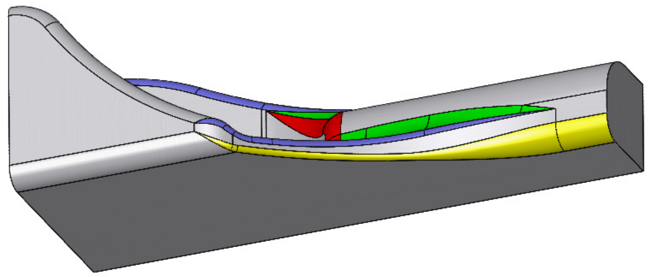
8 In the **Items to Fillet** box, select each edge in turn. Notice that the intermediate points will appear on the edge in the graphics area. Select the longest edge. If you picked edges in order from right to left, it will be Edge3. If you picked from left to right it will be Edge2.

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.

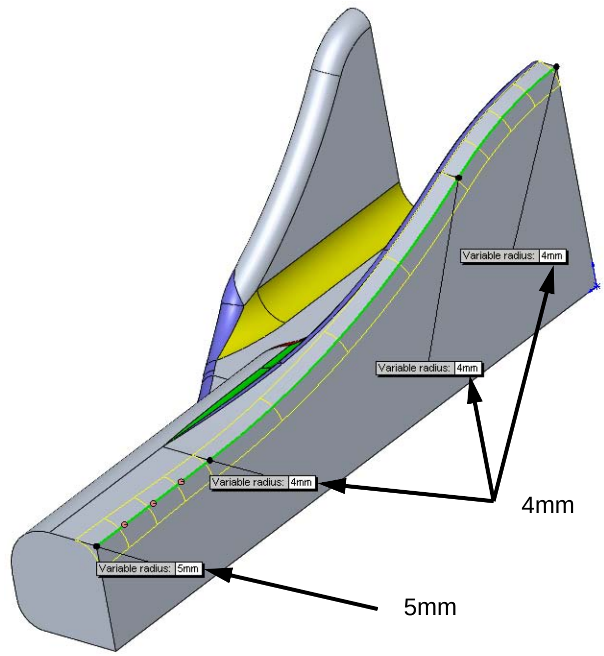


10 Click .

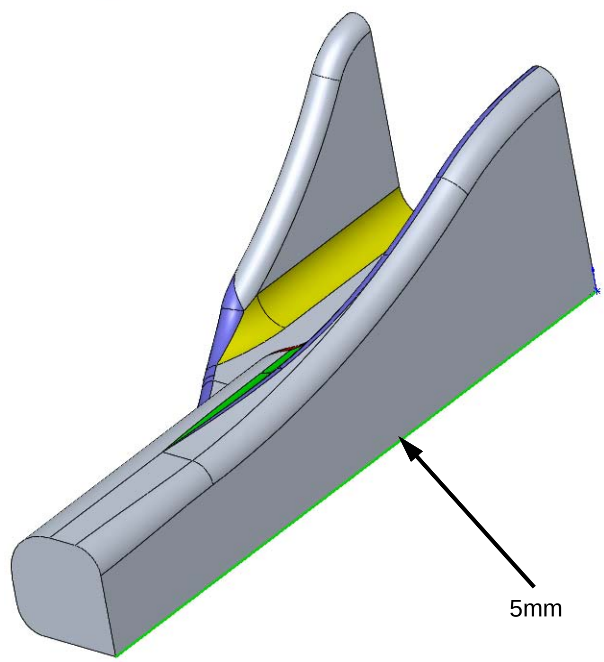
We now have a radius that blends smoothly from 5mm at one end to 1mm at the other end.



11 Add another **Variable Radius fillet**. There are only two radius values, **4mm** and **5mm**.



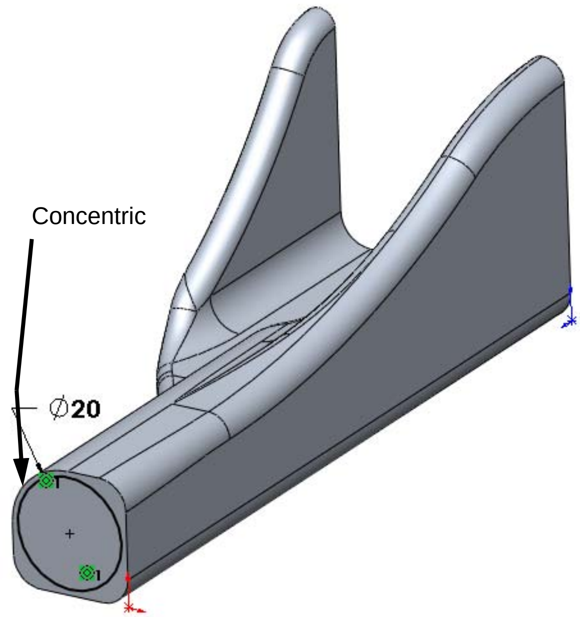
12 Add a constant radius fillet of **5 mm** to the edge shown.



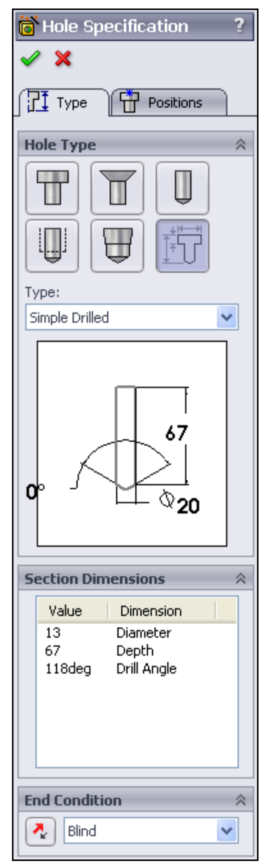
Task 4 — Create The Connections

The final steps are to create the connections to the other components and the second half of the model.

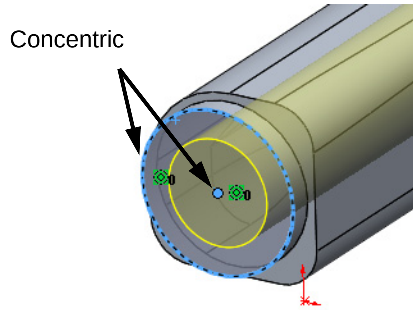
1. Create a sketch on the end face shown.
2. Sketch a circle with a diameter of **20mm** and make it **Concentric** to the fillet edge shown.
3. **Extrude** a boss to a blind depth of **3mm**.
4. Select the end face of the boss and click **Hole Wizard** on the Features toolbar.



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.

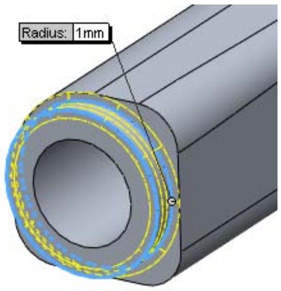


11 Add a **Concentric** relationship between the point and the edge of the boss.



12 Click ✓.

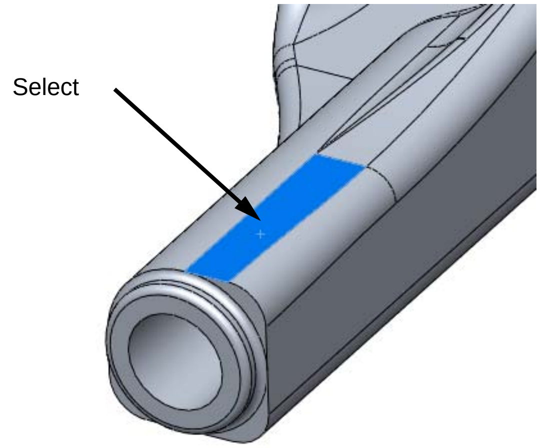
13 Add **1mm** fillets to the edges of the boss.

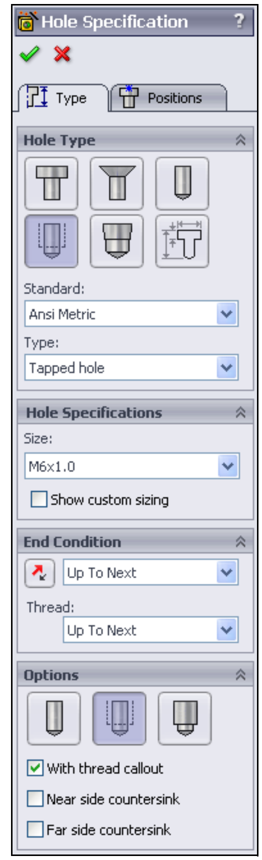


Task 5 — Add Tapped Hole

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 **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**

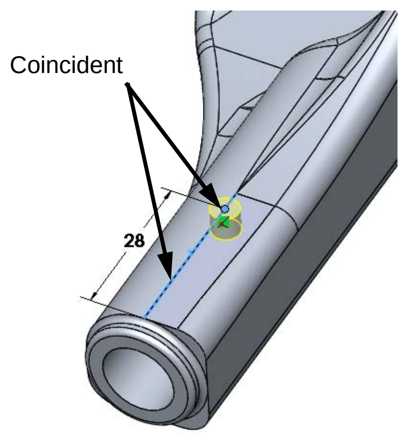
5 Click on the **positions** tab.

6 Clear the **Point** tool.

7 Add a **Coincident** relationship between the point and the edge shown.

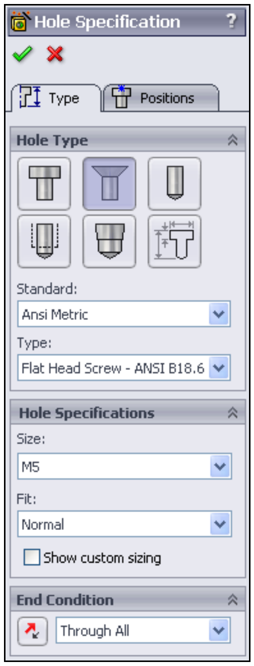
8 Dimension the point **28mm** from the end face.

9 Click .

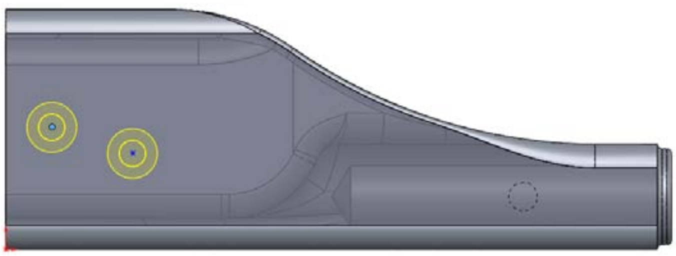


Task 6 — Add Countersunk Holes

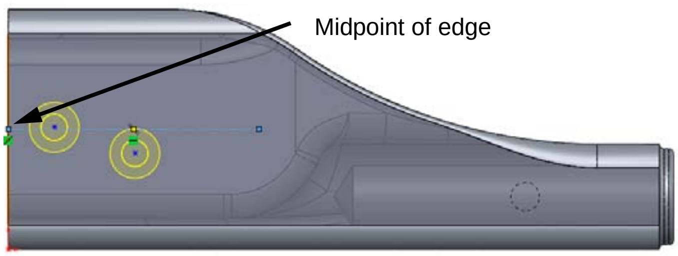
In the final assembly, the spring and dampener assembly will be mounted between the Truck and the Axle. There are two springs and two possible positions for each.



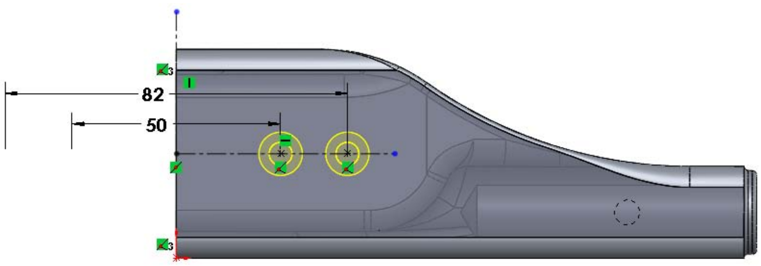
1. Reorient the model to the Bottom view.
2. Select the bottom face and click **Insert, Feature, Hole, Wizard**.
3. Select the **Countersink** button.
4. Make the following selections:
   - Standard: **Ansi Metric**
   - Screw type: **Flat Head Screw - ANSI B18.6.7.M**
   - Size: **M5**
   - End Condition: **Through All**
5. Click on 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.



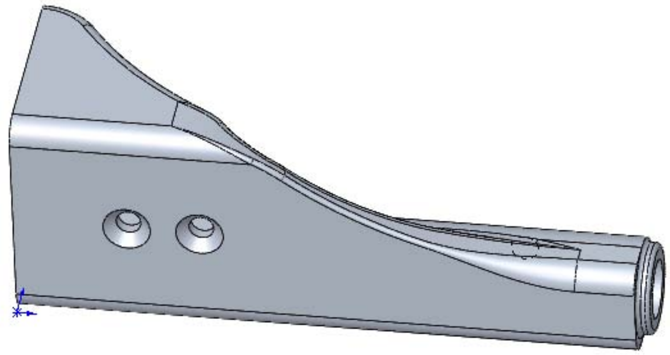
8. Sketch a horizontal **Centerline** from the midpoint of the left edge.
9. Add **Coincident** relationships between the points and the centerline.
10. Draw a vertical centerline from the Origin.



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`.

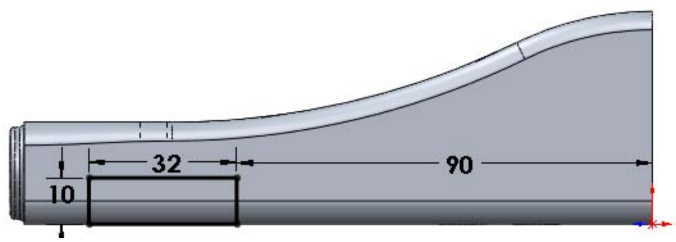
1 Reorient the model to the `Back` view.

2 Create a sketch on the back face.

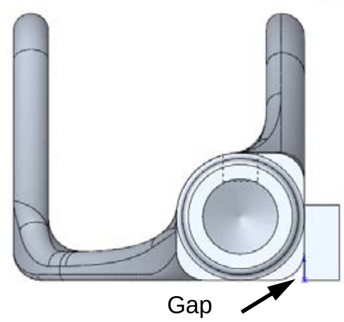
3 Sketch a rectangle.

4 Dimension the sketch as shown.

5 **Extrude** the sketch to a **Blind** depth of **6mm**.



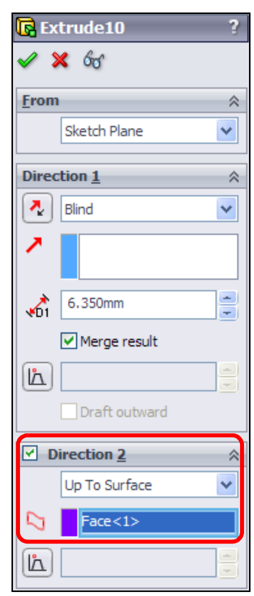
6 Examine the model. Reorient the model to the `Right` view. There is a gap between the extrusion and the rest of the axle. We could have avoided this by creating the brake pad before the fillet. We can also fix this by extruding the brake pad in two directions.



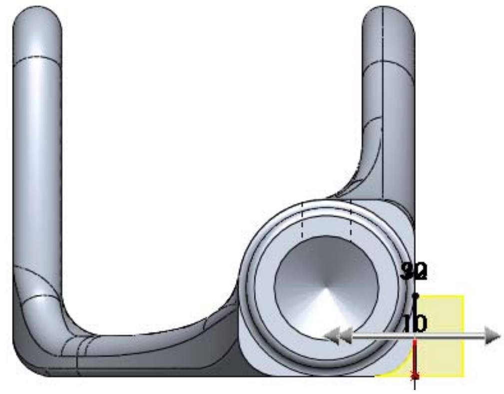
7 Right-click the brake pad extrusion and select **Edit Feature**.

8 Select **Direction 2**.

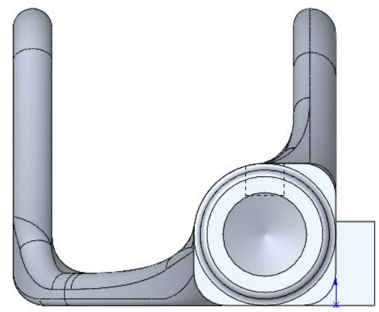
9 For **Direction 2**, select **Up To Surface**. Select the surface of the fillet.



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** on the Features toolbar.

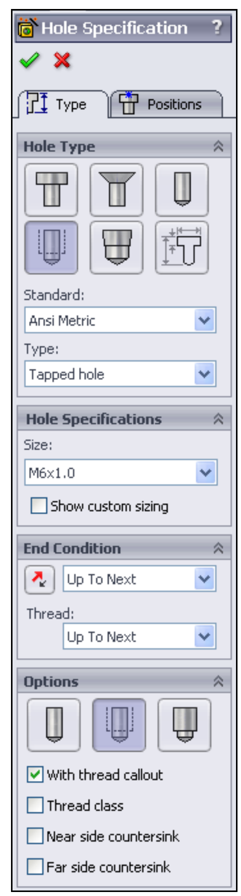
3 Select the **Tap** hole.

4 Make the following selections:

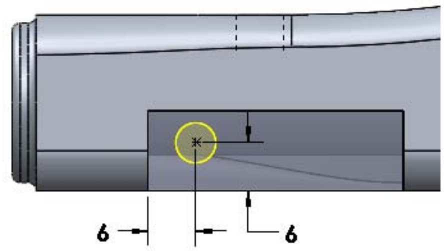
- Standard: **Ansi Metric**
- Screw type: **Tapped hole**
- Size: **M6x1.0**
- End Condition: **Up To Next**
- **Cosmetic thread without thread callout**

5 Click on the **Positions** tab.

6 Turn off the **Point** tool as we only need one hole.

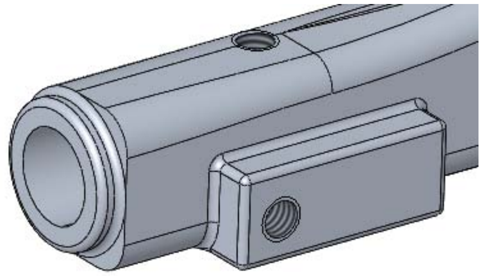


7 Dimension the **Point** as shown.



8 Click .

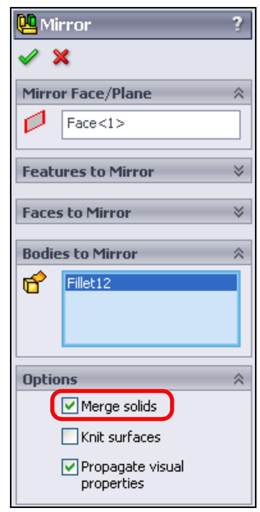
9 Add **1mm** fillets all around.



Task 9 — Mirror The Body

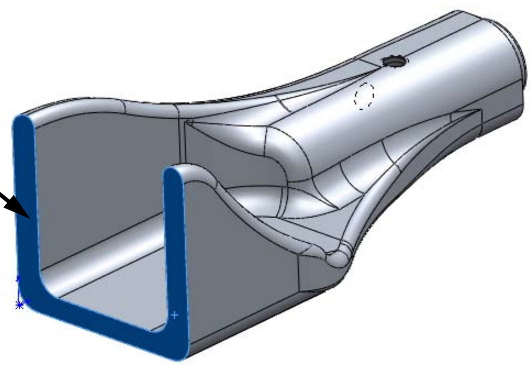
To this point, we have only created half of the Axle. Using the **Mirror** command, we can create the other half of the Axle in one simple step.

1 Click **Insert, Pattern/Mirror, Mirror**.



2 Select the face shown. The is the face we will mirror the part about.

Mirror face

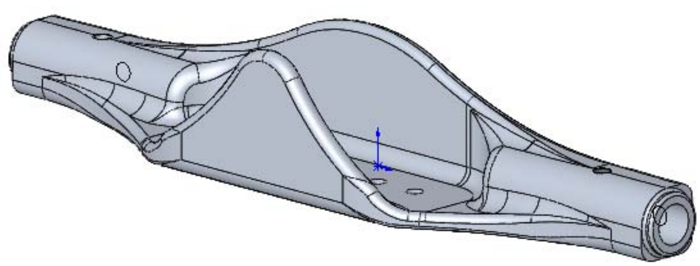


3 Make the **Bodies to Mirror** box active.

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.

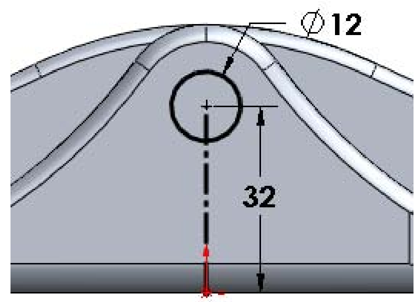
2 Sketch a vertical centerline from the Origin.

3 Dimension the centerline to **32mm**.

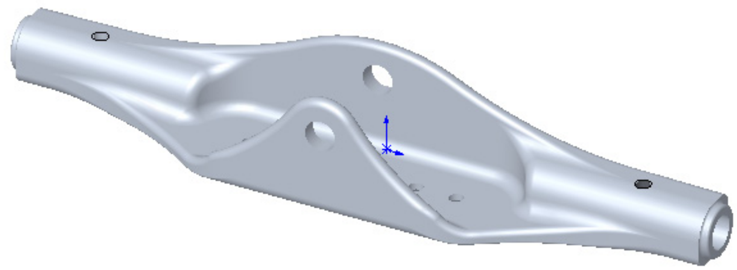
4 Sketch a circle and create a **Coincident** relationship with the end of the centerline.

5 Dimension the circle to **12mm**.

6 Extrude a cut **Through All**.



7 **Save** the part



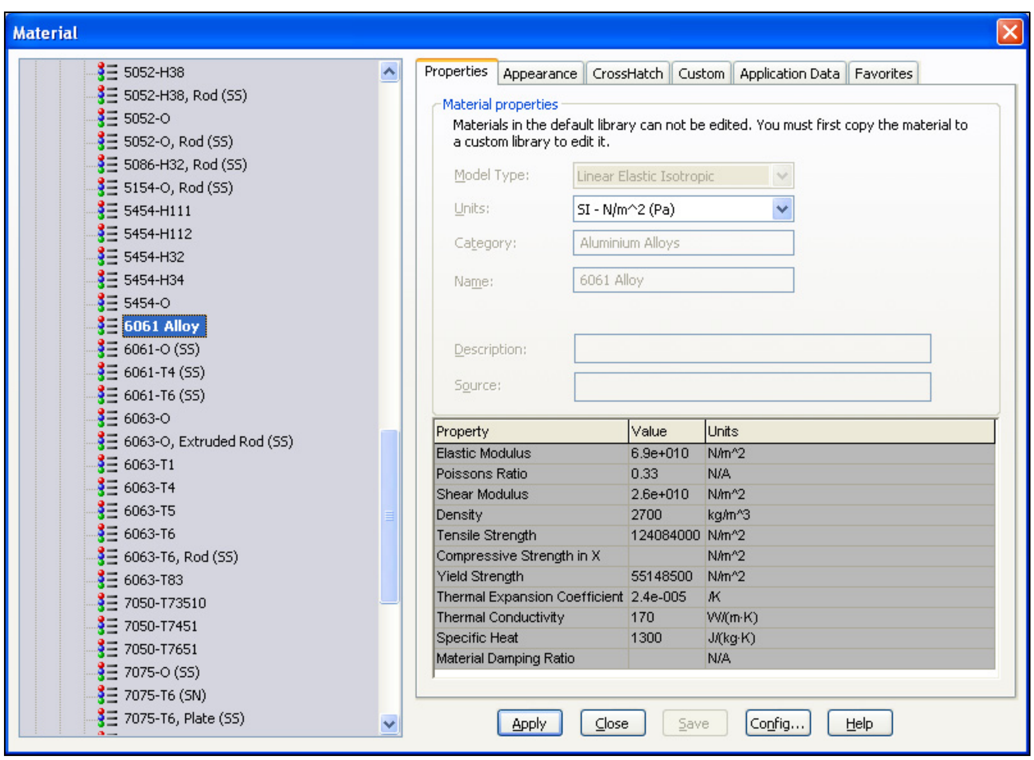
Task 11 — Apply Material

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**.

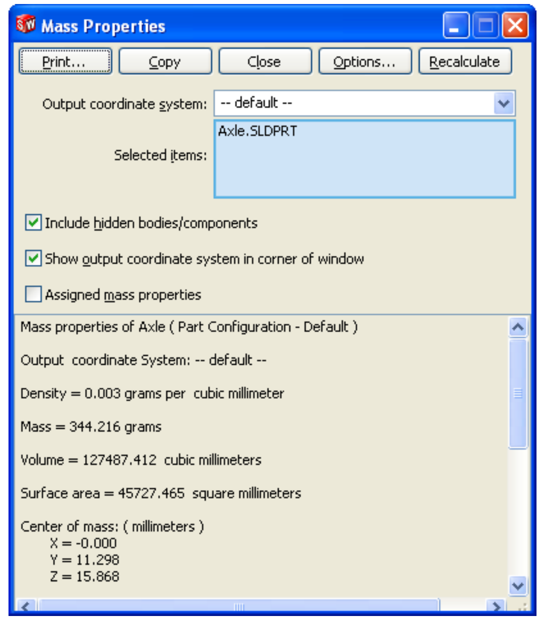


5 Check the weight of the part. Click **Tools, Mass Properties**.

The axle weighs 344.216 grams (0.755 pounds).

6 Click **Close**.

7 **Save** the part.



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?

____

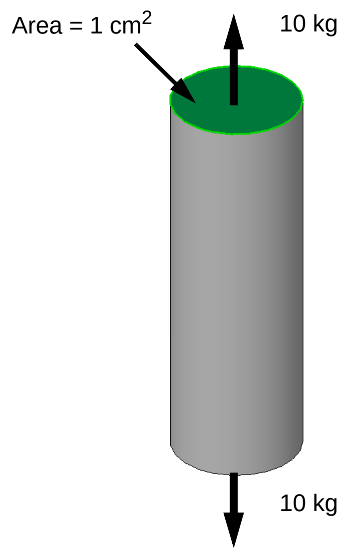
Active Learning Exercise, Part 2 — Stress Analysis

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.

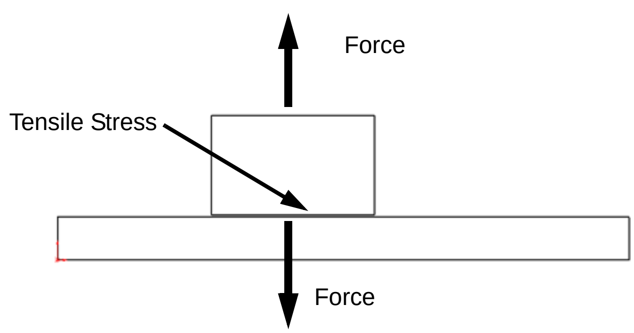
$$Stress = \frac{Force}{Area}$$

Applying a 10 kilogram force to a rod with a 1 square centimeter cross section would yield a stress of 10 kg/ cm^2.

Stress can be either tensile or shear and is a vector quantity having both a magnitude and direction.

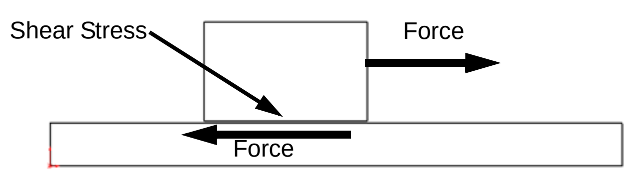
Tensile Stress

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

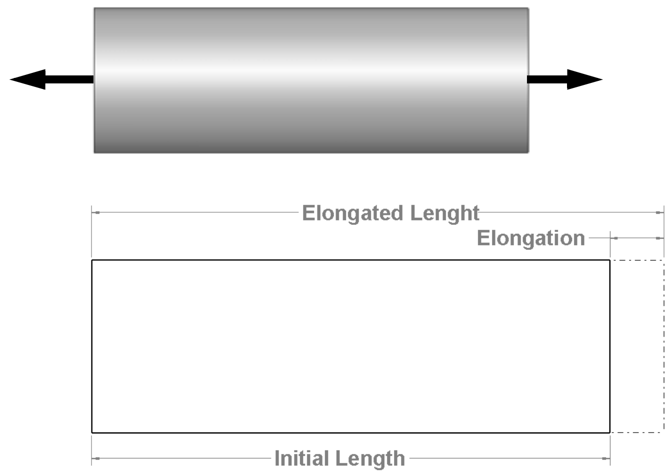
Shear stress is like trying to slide a block on a surface. If we tried to pull the block to the side, the glue joint would be in shear.



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



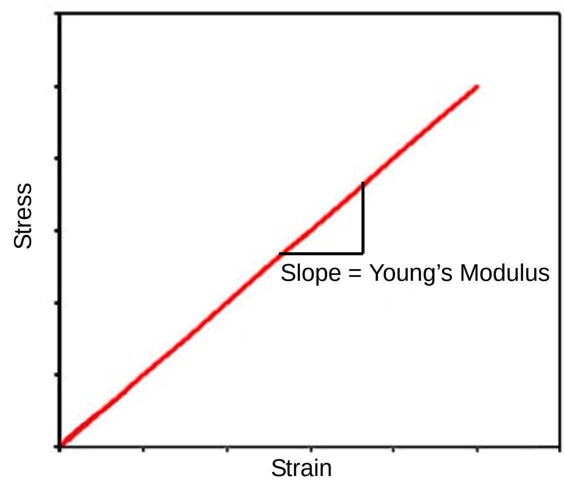
Strain is defined as the intensity of deformation. This is a measure of the change in length of material as a force is applied, measured in units of length per units of length such as inches per inch or centimeter per centimeter.

$$\textbf{Strain} = \frac{\textbf{Elongation}}{\textbf{InitialLength}}$$

If the bar at right was 30cm long before the load was applied and 33cm after the load was applied:

$$\textbf{Strain} = \frac{33 - 30}{30} = \frac{3}{30} = 0.1$$

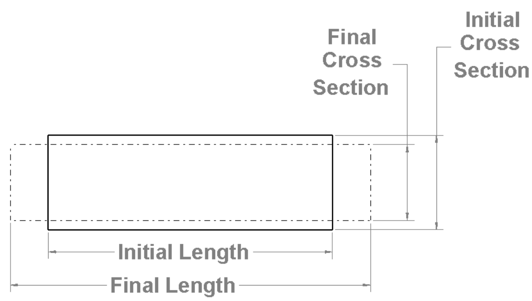
Modulus of Elasticity or Young's Modulus



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.

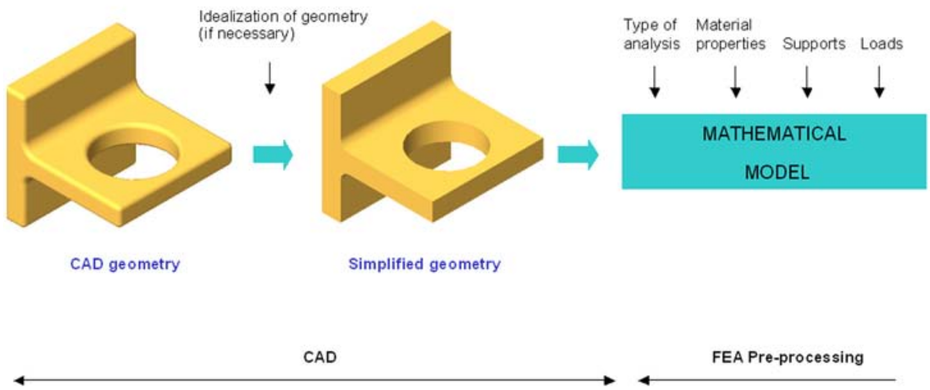
Clean-up

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.

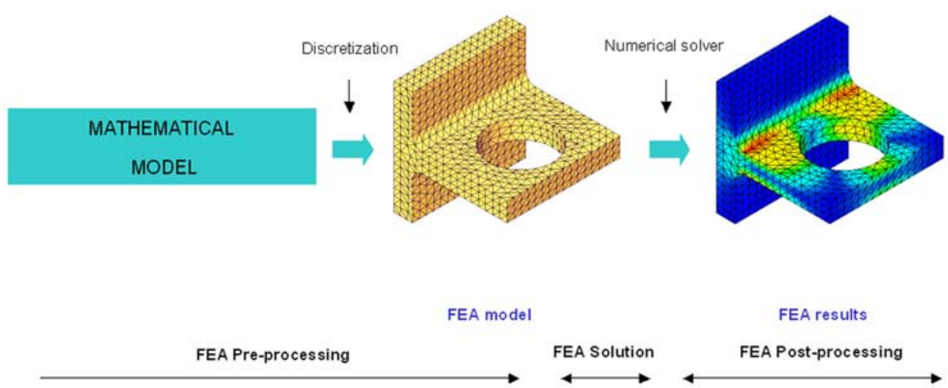
Having prepared a meshable geometry, we define material properties, loads, supports and restraints, and provide information on the type of analysis that we wish to perform.

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

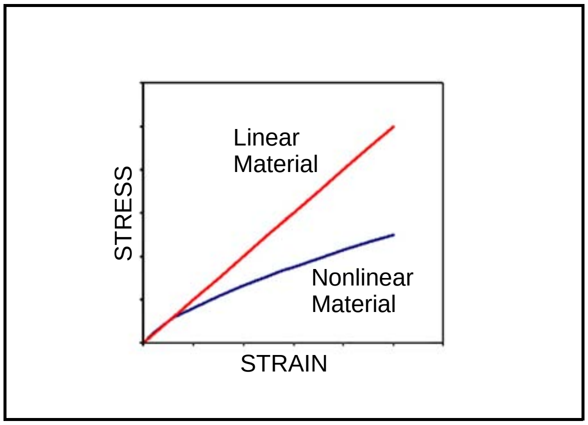
With any FEA software, we need to take advantage of its strengths as well as work within its limitations. Analysis with SolidWorks Simulation is conducted under the following assumptions:

- 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.

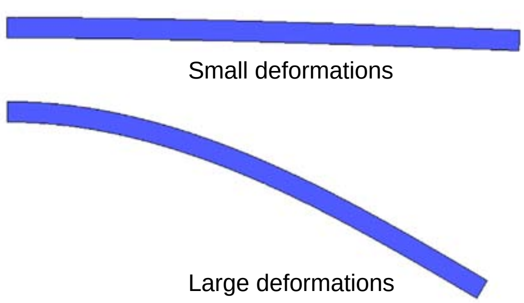
Material yielding is not modeled. Whether or not yield, in fact, takes place can only be interpreted based on the stress magnitudes reported in results.

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

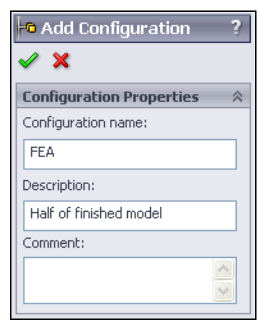
All loads, as well as fixtures, are assumed not to change with time. This limitation implies that loads are applied slowly enough to ignore inertial effects. Dynamic loading conditions can not be analyzed with SolidWorks Simulation.

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.

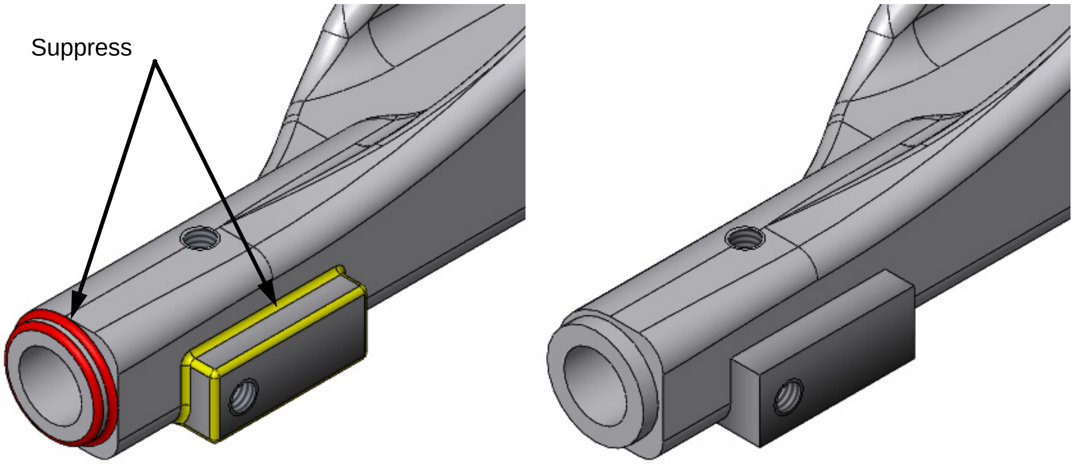
Task 1— Prepare the model for analysis

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.



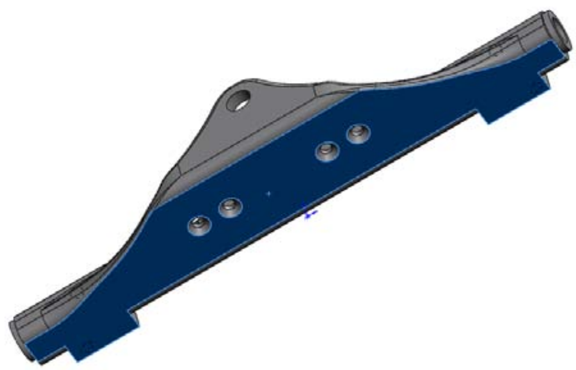
1 In the FeatureManager design tree, select the ConfigurationManager tab.
2 Right-click the top level icon and select **Add Configuration**.
3 Name the new configuration FEA.
4 Type `Half of finished model` for the description.
5 Click  to create the configuration.

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.

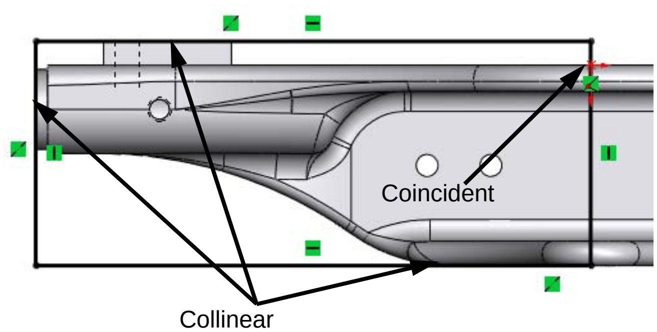


7 Select the bottom face of the model and open a sketch.

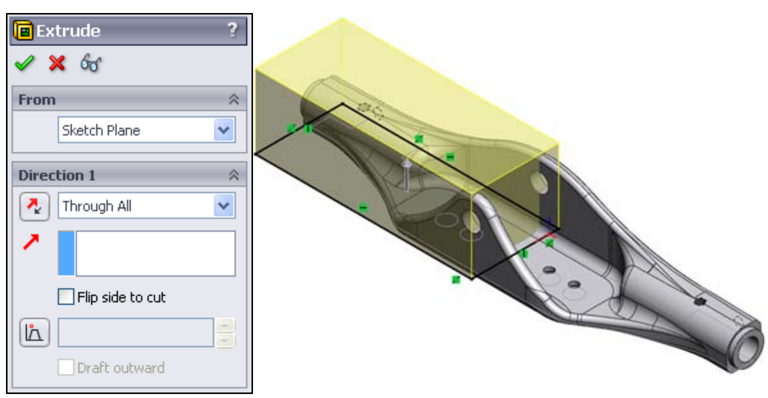
8 Reorient the model to the Top view.



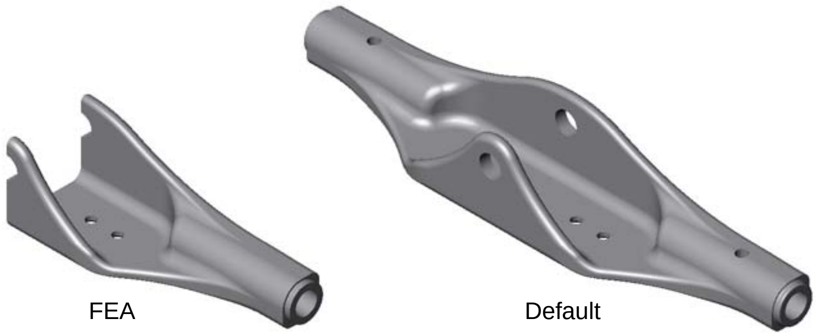
9 Sketch a rectangle and add relationships to the origin and three edges of the model so that the rectangle covers half of the model.



10 Extrude a **Cut** through the entire mode. Click ✓.



11 We now have two configuration of the part, `Default` and `FEA`.



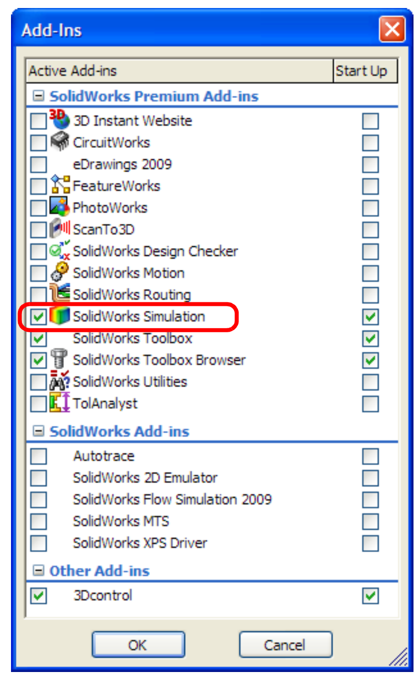
Task 2 — Setup the analysis

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**.



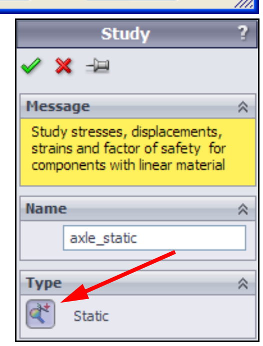
3 Examine the CommandManager, there will be a new tab for SolidWorks Simulation. There will also be a new menu titled **Simulation**.

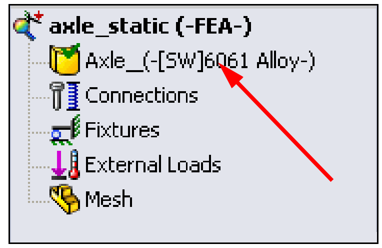
4 To do an analysis, we must create a Study. On the menu, select **Simulation, Study**.

5 In the PropertyManager, type `axle_static` for the study name.

6 Under **Type**, select **Static**.

7 Click .

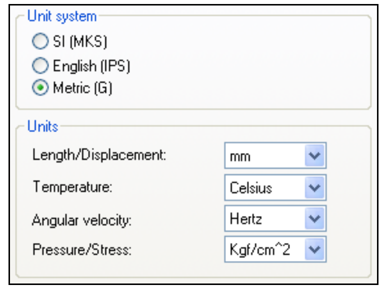




8 The SolidWorks Simulation Manager will appear below the FeatureManager design tree and shows the study with folders for the different parameters used in the study.

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.

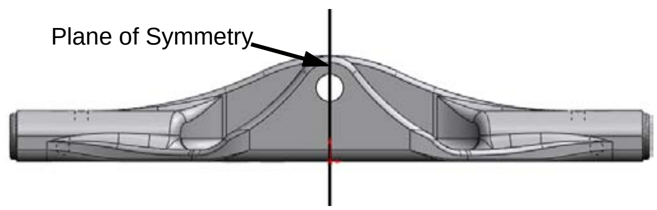


11 On the menu, click **Simulation, Options**.

12 Select the **Default Options** tab.

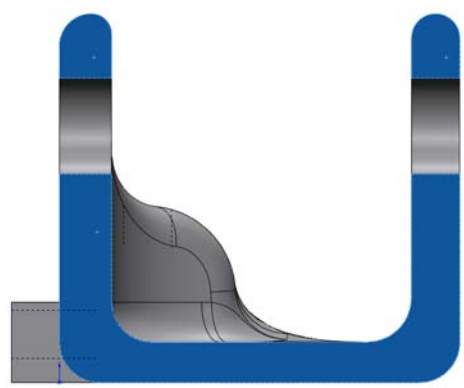
13 Select **Metric** for the Unit system, **mm** for the Length/ Displacement units and **Kgf/cm^2** for Stress. Click **OK**.

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`.

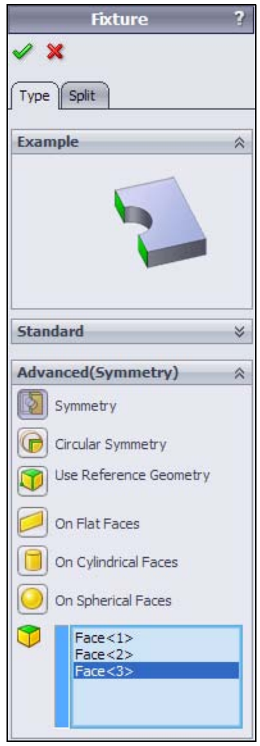
The third restraint will be fixed, just to keep the model from moving. We will only apply the fixed restraint to a single vertex to prevent the fixture from causing the model to deform unnaturally.

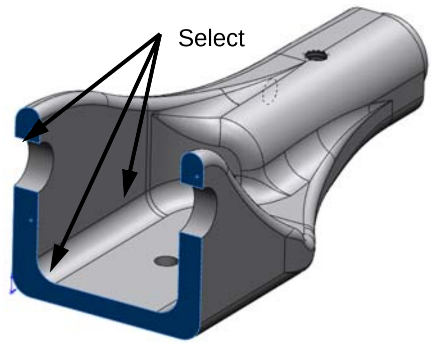
1 In the SimulationManager, right-click the `Fixture` item. Select **Advanced Fixtures**.

2 Under **Advanced**, select **Symmetry**.

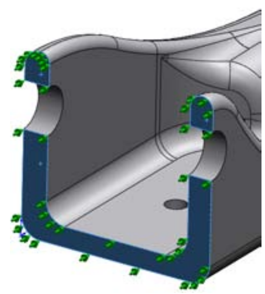
3 Select the three faces shown.

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.



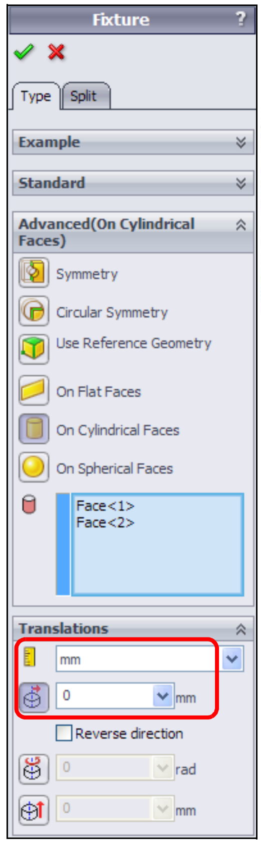
5 Right-click the Fixtures item again, and select **Advanced Fixtures**.

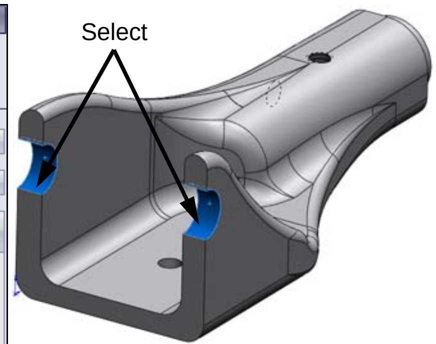
6 Under **Advanced**, select **On Cylindrical Face.**

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 .



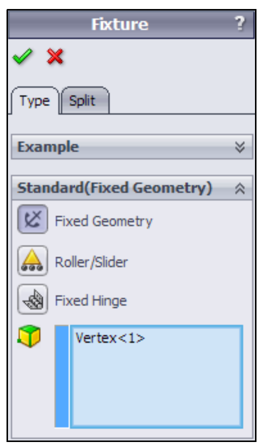


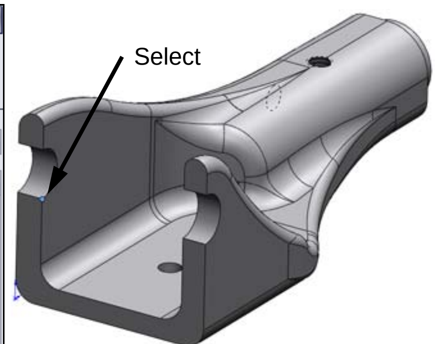
10 Right-click the Fixtures item again, and select **Fixed Geometry**.

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**.

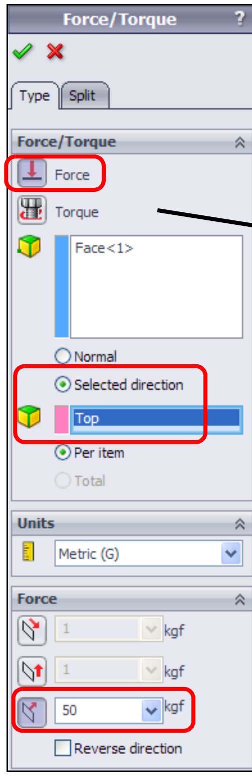
15 Select the face shown.

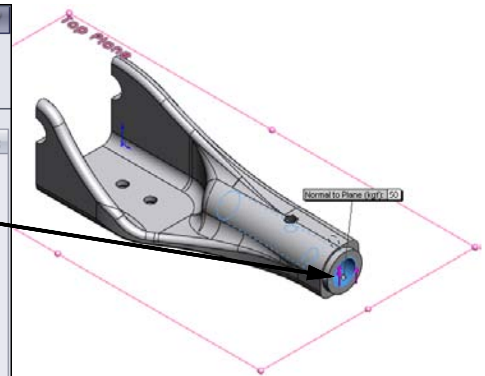
16 Under **Force/Torque**, select **Selected direction**.

17 Click in the **Direction** box to make it active, then select the Top plane from the Fly-out FeatureManager design tree.

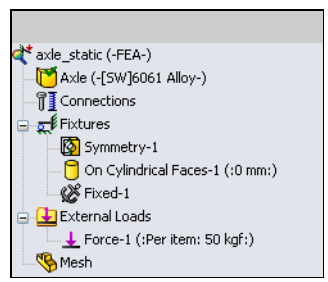
18 Under **Force**, click **Normal to plane**. Type **50** to apply 50 kgf normal to the Top plane. *If necessary,* **Reverse direction** for the force to be applied upwards.

19 Click ✓.





20 Examine the SimulationManager. The material, fixtures and force that we applied are all listed.

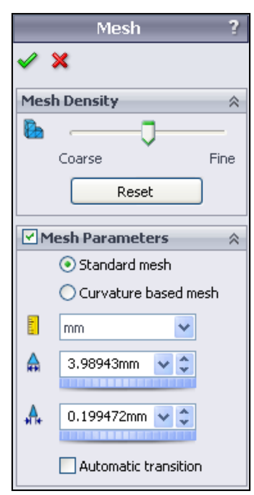


Task 4 — Run the analysis

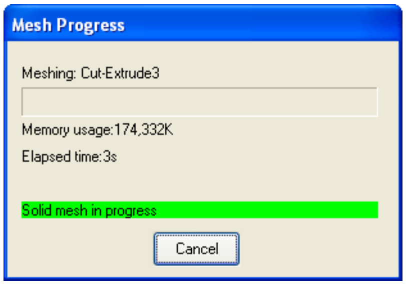
Before the analysis can be run, the model must be meshed.

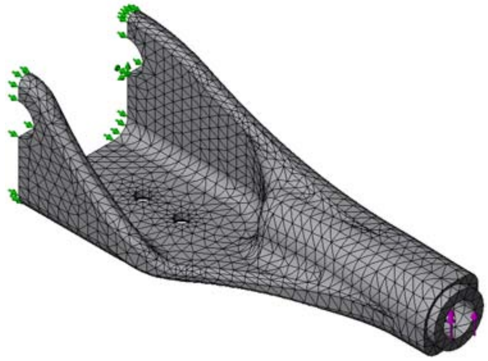
1 Right-click the **Mesh** item and select **Create Mesh**.

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.

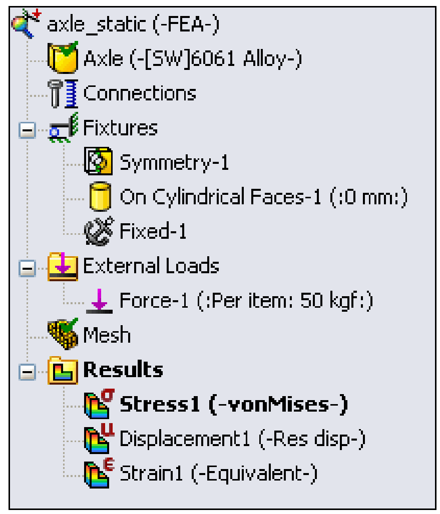




4 We now have everything needed to run the analysis. In the SimulationManager, right-click the study `axle_static` and select **Run**.

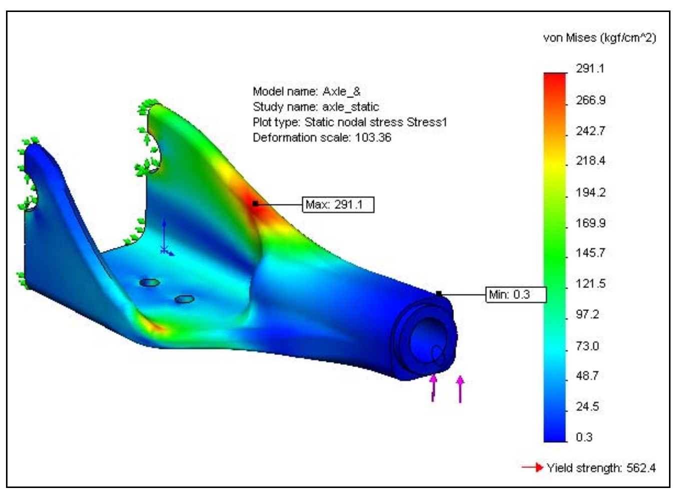
Task 5 — Analyze the results

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.



1 Click the plus sign next to the `Results` folder.

Double-click `Stress1`.

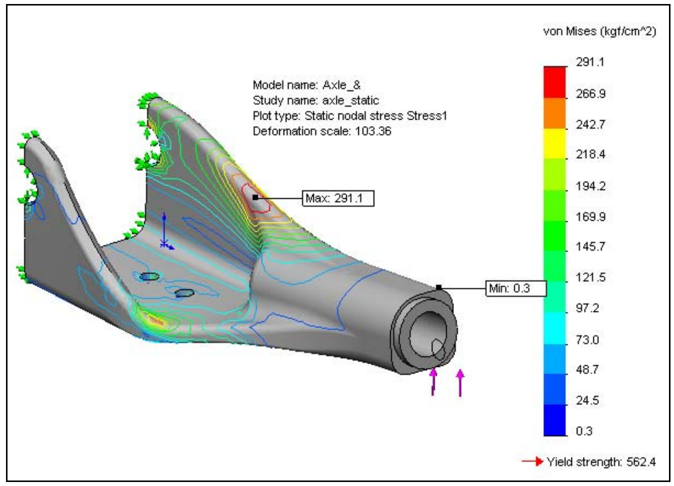


2 Examine the plot. The color of the model represents the stress and corresponds to the scale on the right. Our default settings show von Mises stress measured in kg/cm^2.

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.

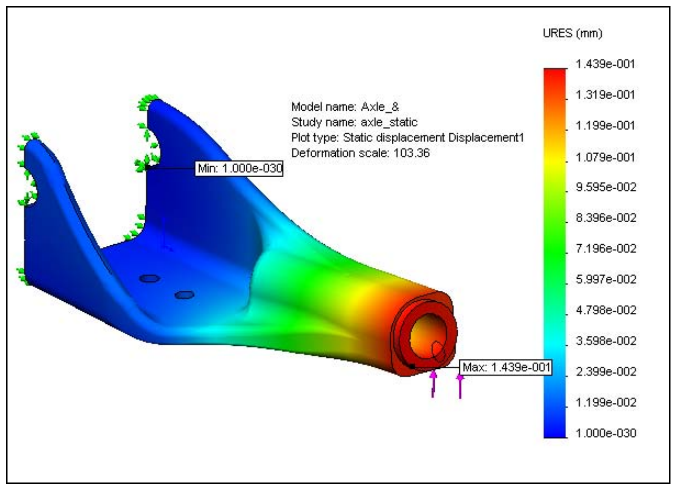
4 There is a plot title in the upper left corner of the graphics area. Notice that the deformation scale is 103.087. This means that we are seeing slightly more than 100 times the actual deformation of the part. This is done just to make it easier to visualize the result.

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.

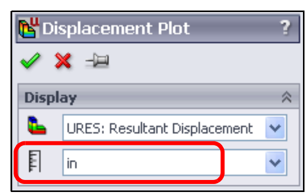


6 Double-click `Displacement1` under the `Results` folder. The colors now represent the amount of displacement of each element. The scale is in mm and we are looking at just about 100 times the actual deformation.

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.



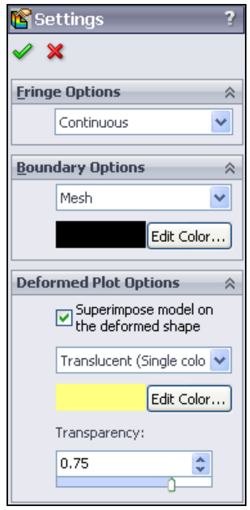
Click .

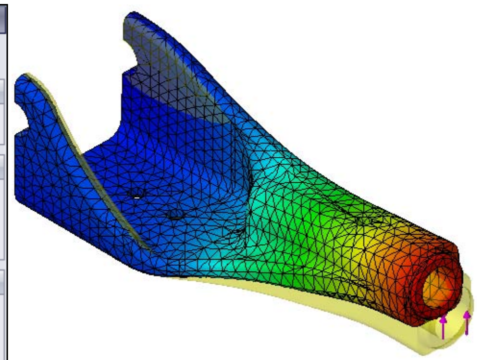
8 We can see from the scale that the maximum displacement is 5.683e-003 or about 0.005 inches.

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.



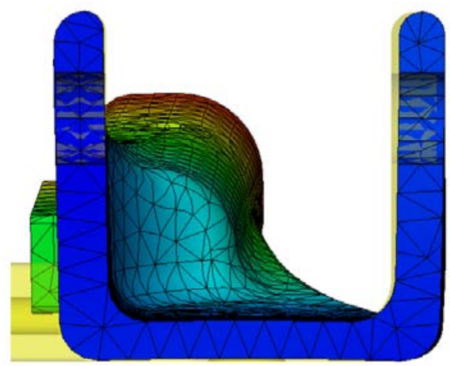


Adjust the transparency slider to be able to see the undeformed shape.

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

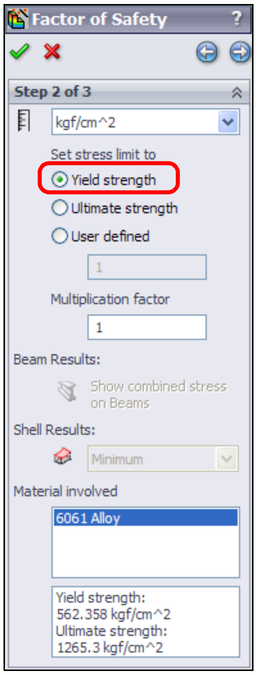
When we used SolidWorks SimulationXpress, one of the result we got was the Factor of Safety. We can create a similar plot in SolidWorks Simulation.

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.

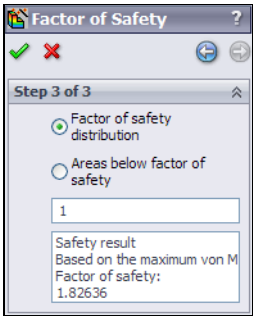
3 Use the default Max von Mises stress for the first step. We will create a Factor of Safety plot by having the maximum von Mises stress divided by the yield stress.

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** .



7 Select Factor of safety distribution.

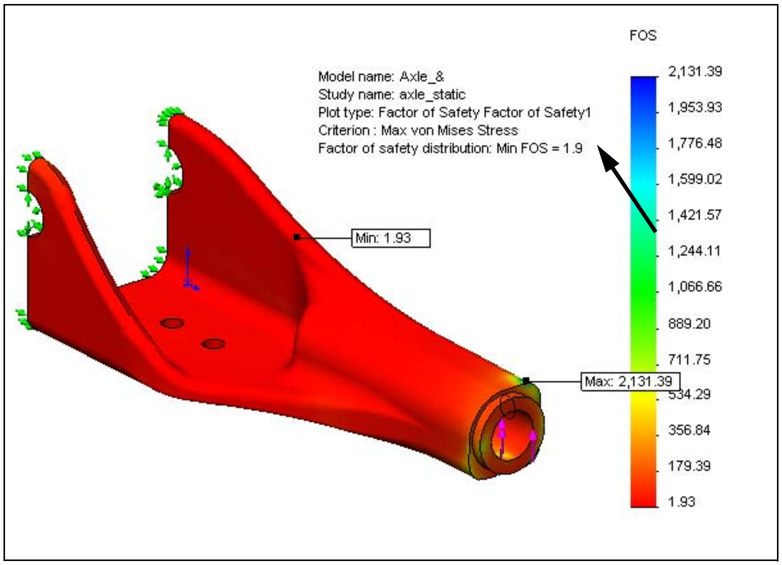
8 Click .

9 This plot shows us that the minimum **Factor of Safety** is **1.9**.

10 Select the ConfigurationManager tab.

11 Double-click the `Default` configuration.

12 **Save** and **Close** the `Axle` part.



5 Minute Assessment — #6-2

1 What are the three steps of the FEA process?

____

____

____

2 What happens during discretization or meshing.

____

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

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

____

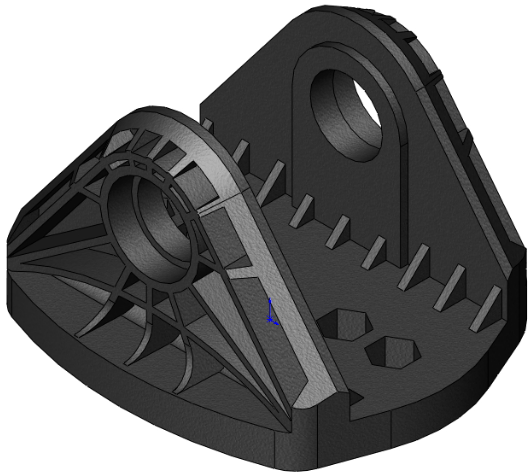
____

____

5 If you apply a material in SolidWorks, do you have to apply it again in 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.

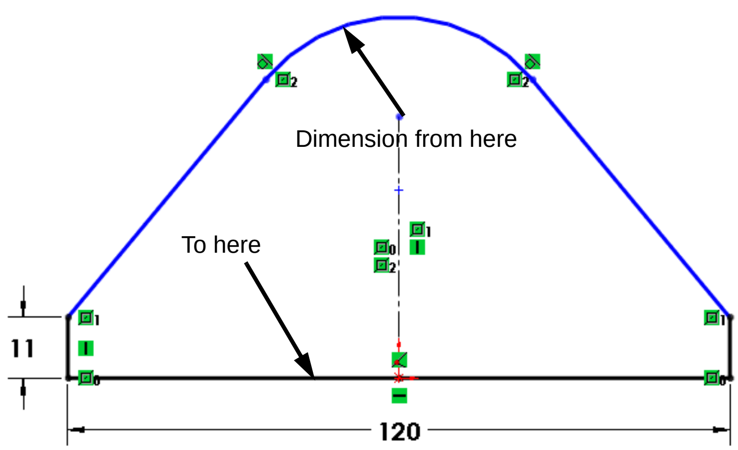
Task 1— Create the basic shape.

1 Create a new part using the Part_MM.slddot template.

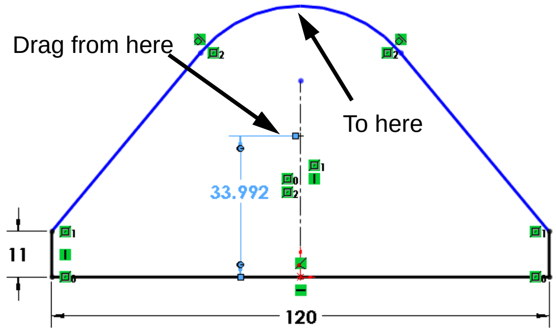
2 Create a sketch on the Front plane.

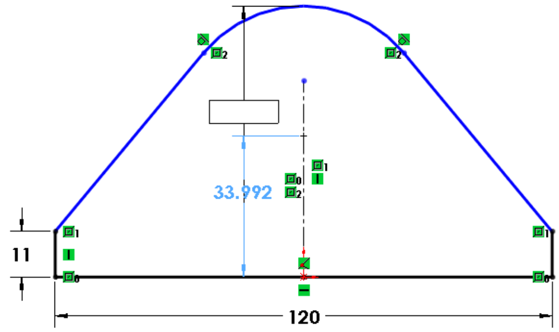
3 Use the centerline and sketch mirroring to create the sketch.

4 Add a dimension between the arc and the base line. By default, the dimension will be placed at the center of the arc.

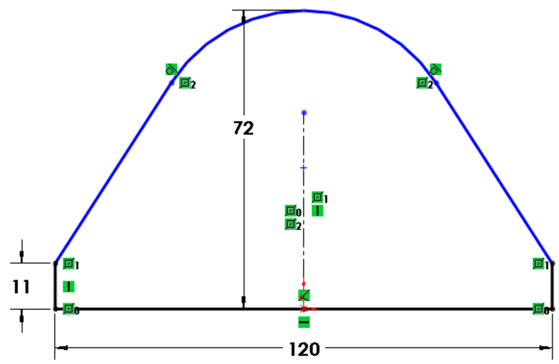


5 With the dimension selected, drag the end of the extension line from the center of the arc to the arc itself.



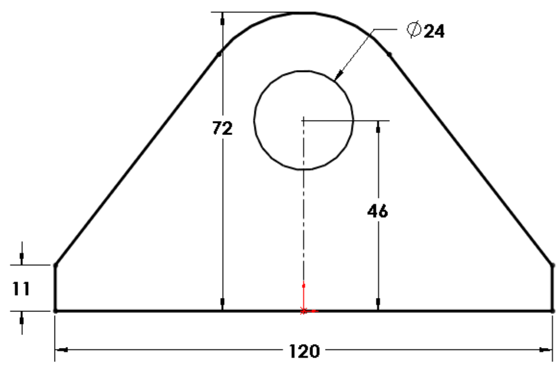


6 Once the dimension goes to the top of the arc, it can be changed to the correct value of **72mm**.



7 Sketch a circle and add the two dimensions shown. The center of the circle should be coincident to the centerline.

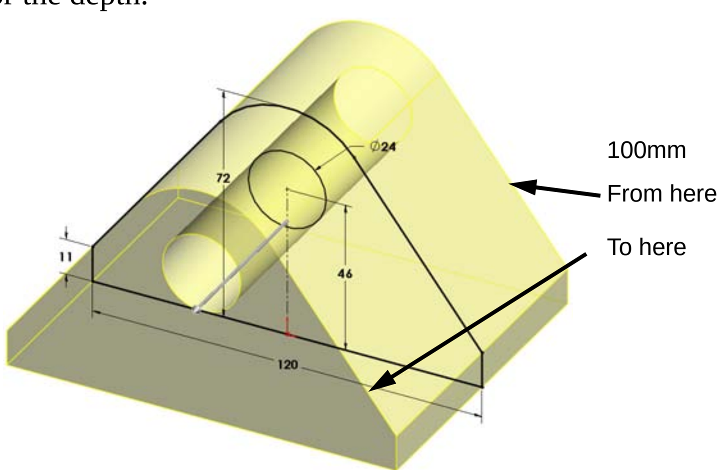
8 To fully define the sketch, add a **Concentric** relationship between the circle and the arc.

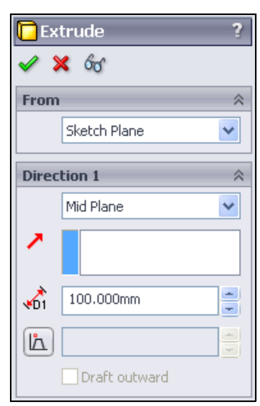


Mid Plane Extrusion

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 type **100mm** for the depth.

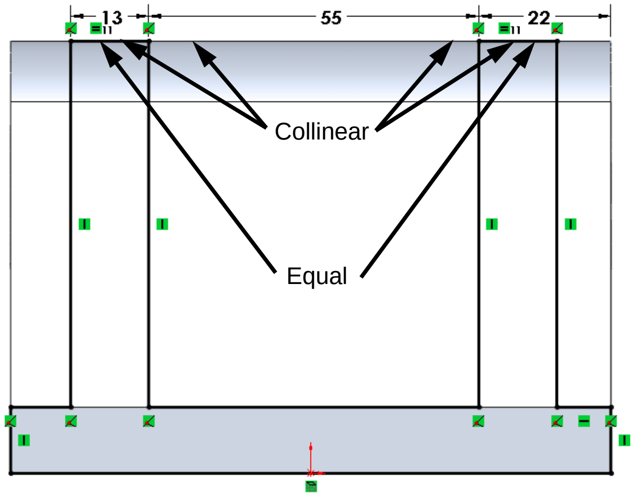




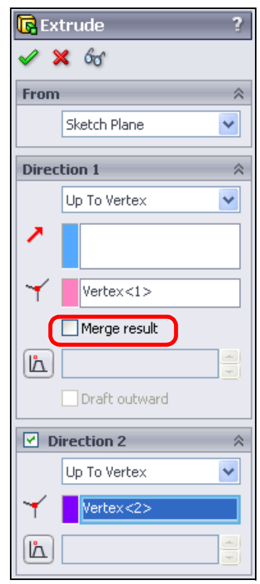
10 Click **OK**.

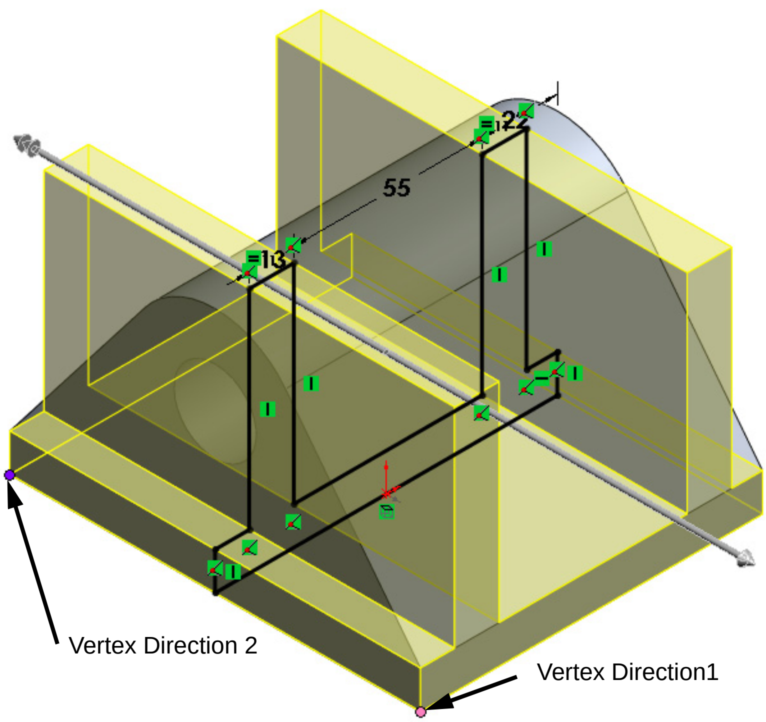
11 Create the following sketch on the Right plane.

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.



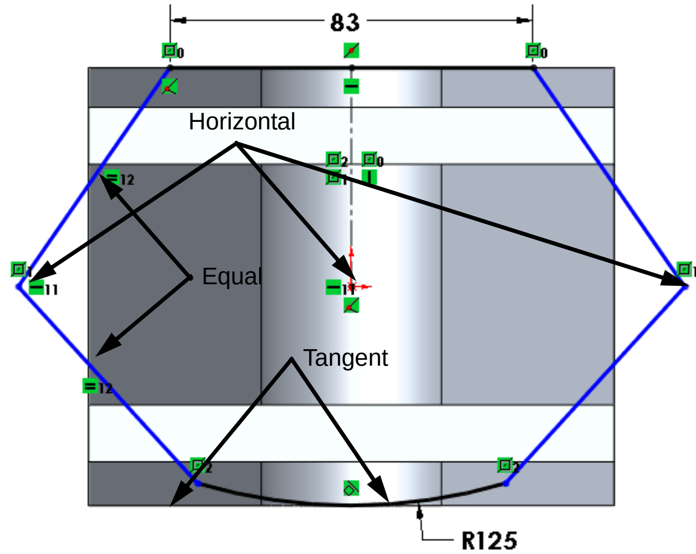


14 Clear **Merge result** and click **OK**.

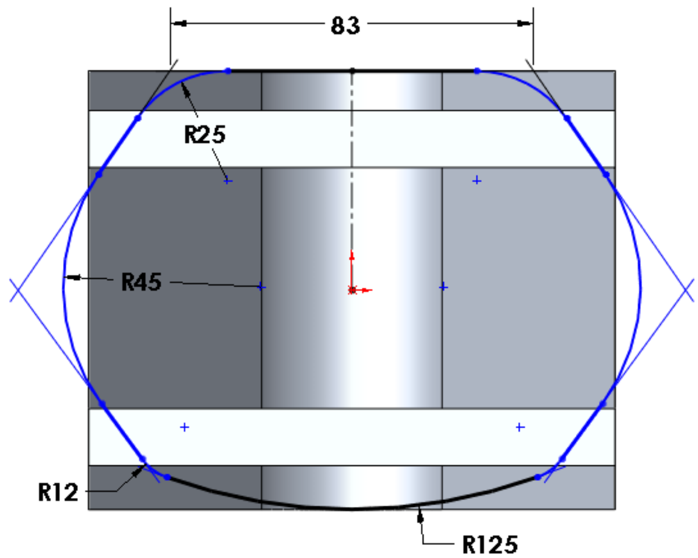
15 Create a new Sketch on the **Top** plane.

16 Sketch the following geometry.

17 The sketch is symmetrical. Use the centerline and **Dynamic Mirror Entities**.



18 Add **45mm**, **25mm** and **12mm** sketch fillets.



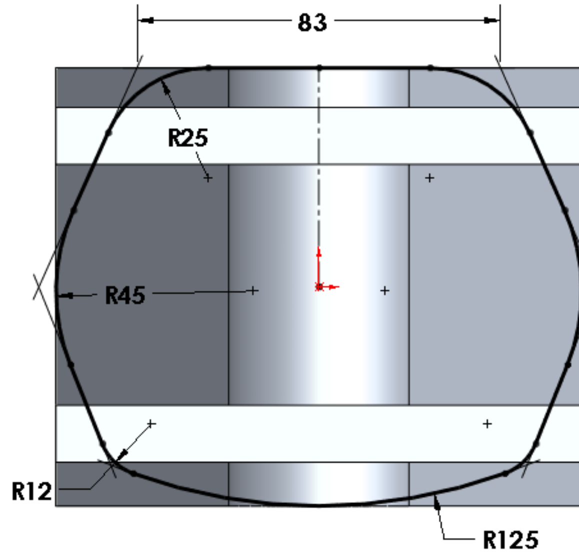
Note: When you add the first of each pair of fillets, you will get a warning that "At least one segment being filleted has a midpoint or equal length relation. Geometry may have to move to satisfy this relation when the fillet is created. Do you want to continue?"

Click **Yes**. Once you add the second fillet of the pair, the midpoint relationship will be restored and solved.

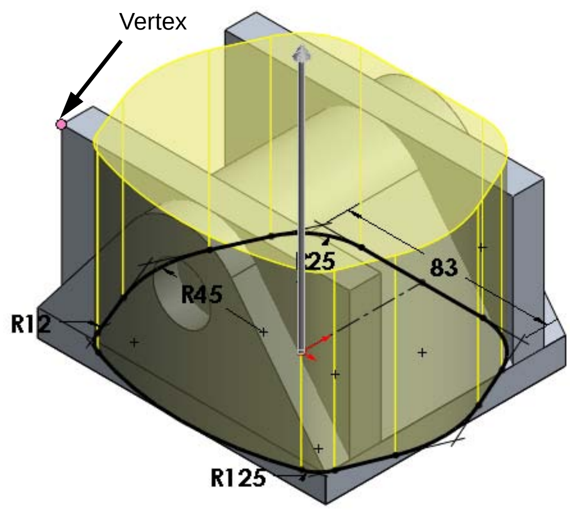
---

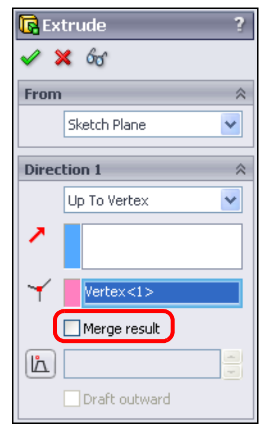
19 Add a **Tangent** relationship.

The sketch should now be fully defined.



20 Clear **Merge result** and **Extrude** the sketch **Up To Vertex**.





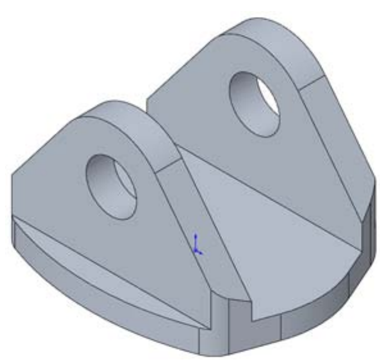
Question: Why didn't we use **Blind** for the end condition?

Answer: We wanted to make sure that this extrusion always goes to the top of the part, even if the other two extrusions change.

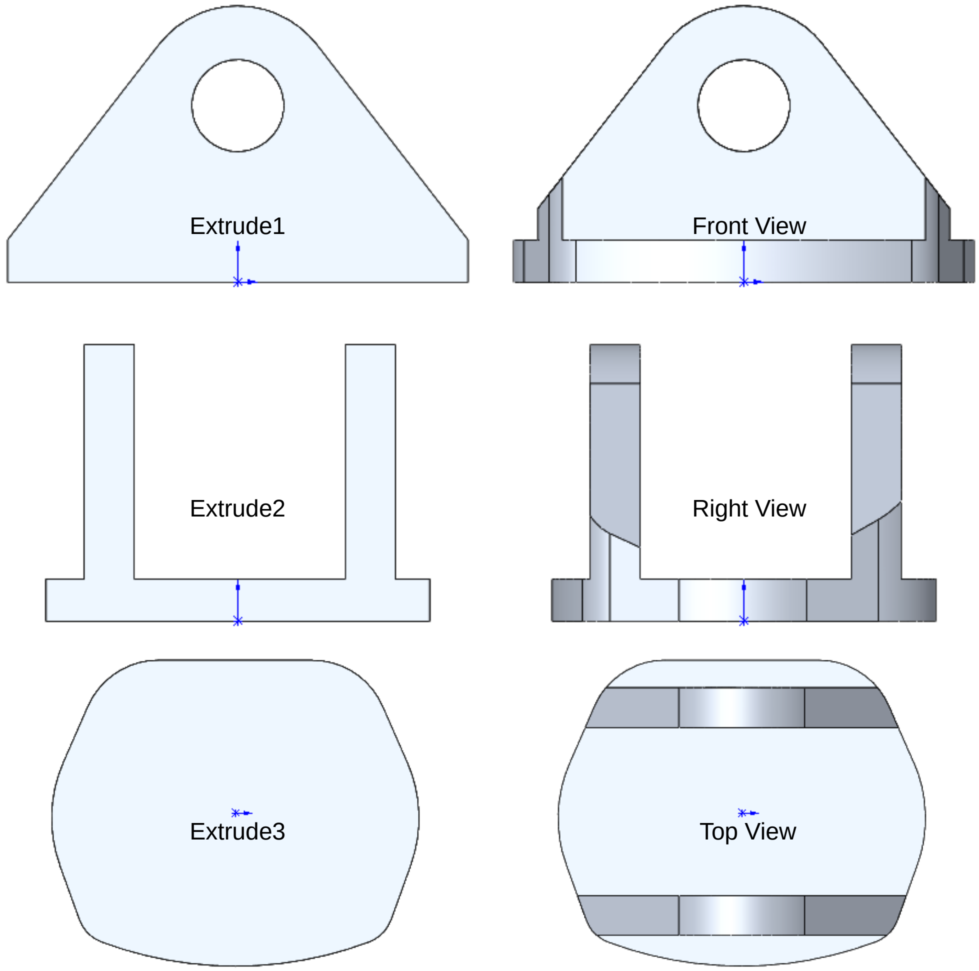
21 Combine the three bodies. Click **Combine** and select **Common**. Select the three solid bodies from the Solid Bodies folder.

22 Click **OK**.

23 **Save** this part as Truck to the Mountainboard\Axle-Truck folder.

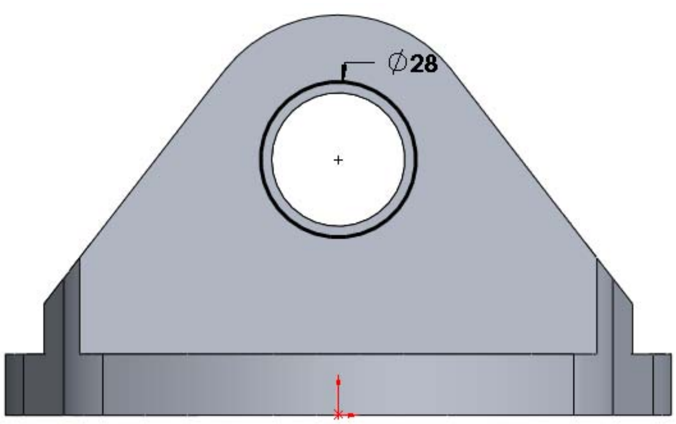


Examine the part. The technique we used was to create three solid bodies that represented the Front, Right and Top view of the combined body.



Task 2 — Create the Bearing Holes

1. Orient the model to the Front view.
2. Sketch a circle as shown, centered on the existing hole in the Truck.
3. Dimension the hole to **28mm**.

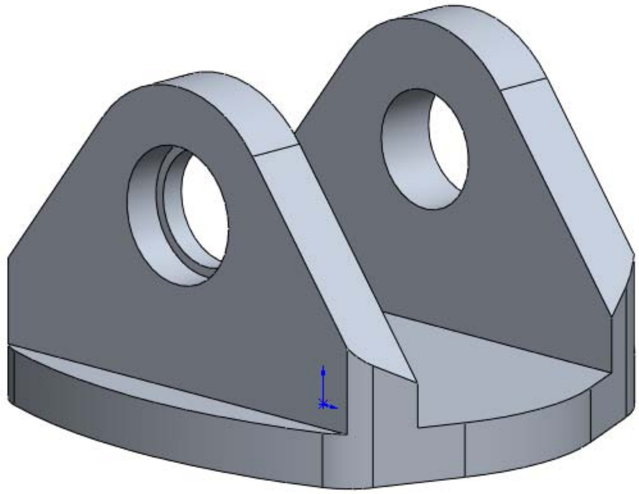


4 Create a **Blind** cut to a depth of **8mm**.

5 Reorient the model to the `Back` view and repeat the cut.

<u>Question:</u> 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.

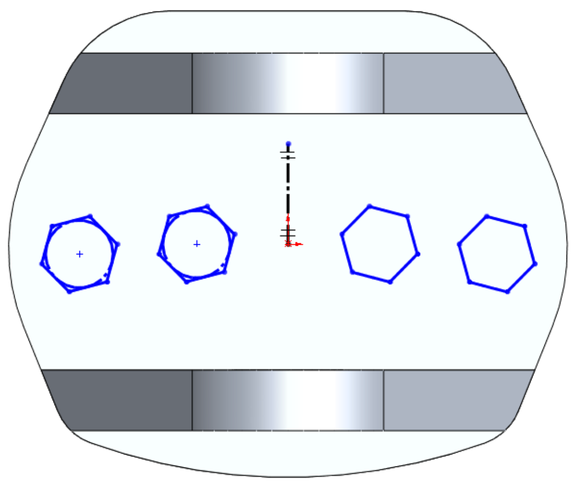
1 Orient the model to the `Top` view.

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.



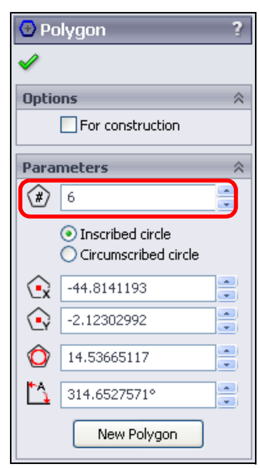
6 The number of sides to the polygon is adjusted in the PropertyManager. Select **6**.

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.



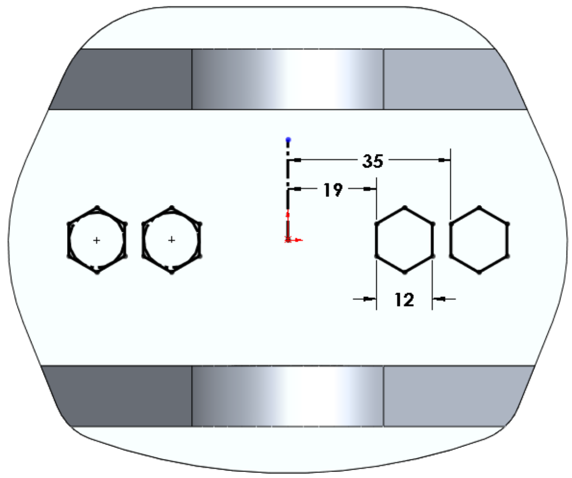
10 Select the to circles that are inscribed in the polygons and add an equal relationship. With the circles equal, we only need to add size dimensions to one of the polygons.

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`.



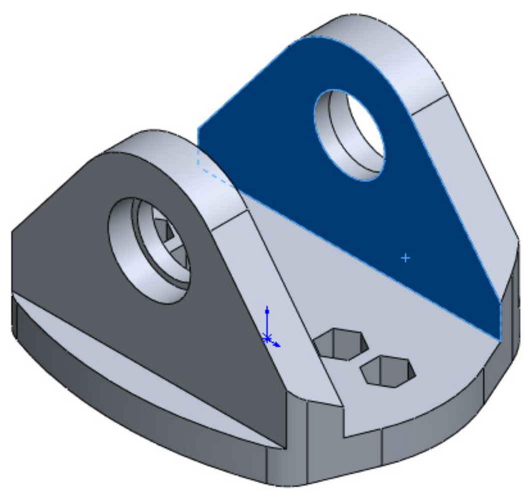
Task 4 — Create Standoffs

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** 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.



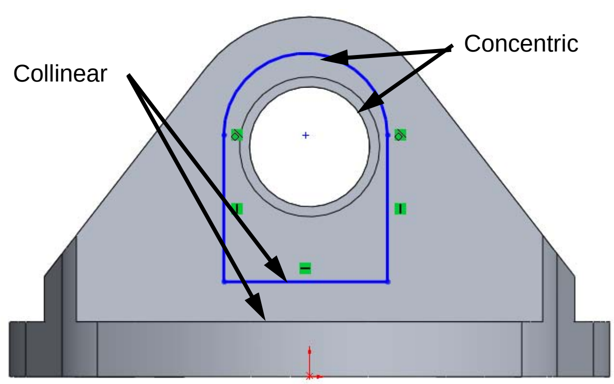
4 Create a “headstone” sketch.

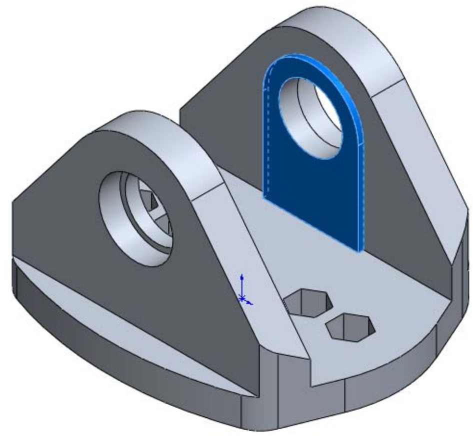
5 Add a **Concentric** and **Collinear** relationship.

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 click **Convert Entities** .





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

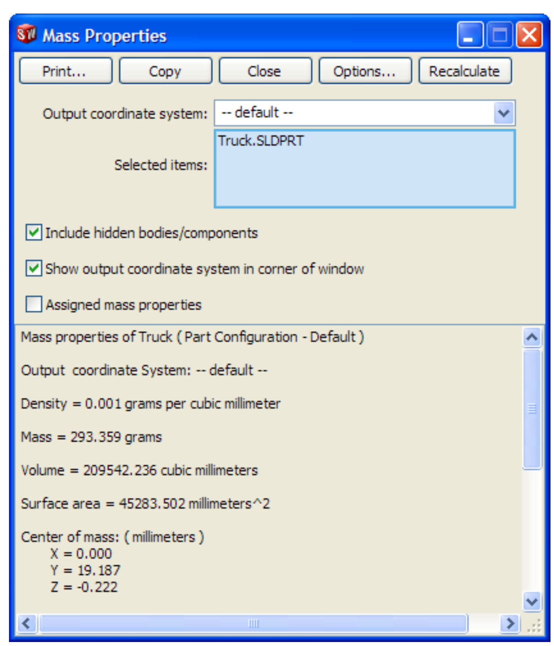
We have everything in this version of the Truck except the holes that will be used to mount it to the Deck. We will check the weight of the part, then do a static analysis to make sure the part is strong enough.

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 **209,542 cubic millimeters** (12.787 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 **293.359 grams** (0.647 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.

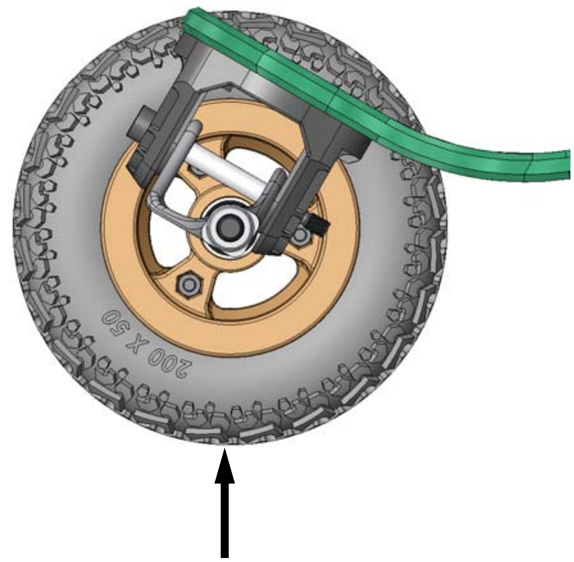
Task 6 — First stress analysis.

To determine if the part has sufficient strength, we will use Finite Element Analysis to examine the stress distribution and deformation of the model.

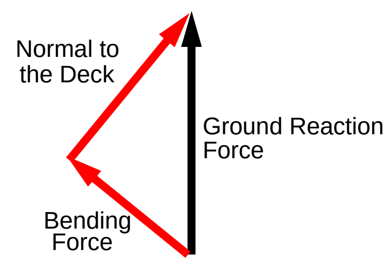
When in use, the `Truck` has loads applied from the weight of the rider plus numerous impact loads from running over objects, taking jumps and cornering. The actual computation of the magnitude and direction of these forces is beyond the scope of this course, so we will use a set of loads that were determined 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`.

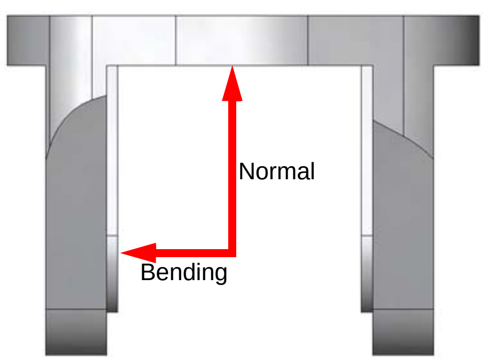


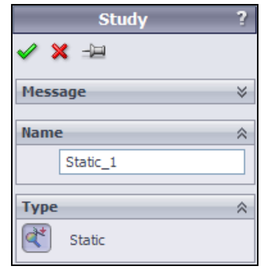
Using vector addition, this force can be broken into two forces, one which pushes the `Truck` into the `Deck`, and a second force that tries to bend the vertical bosses of the `Truck`.



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.

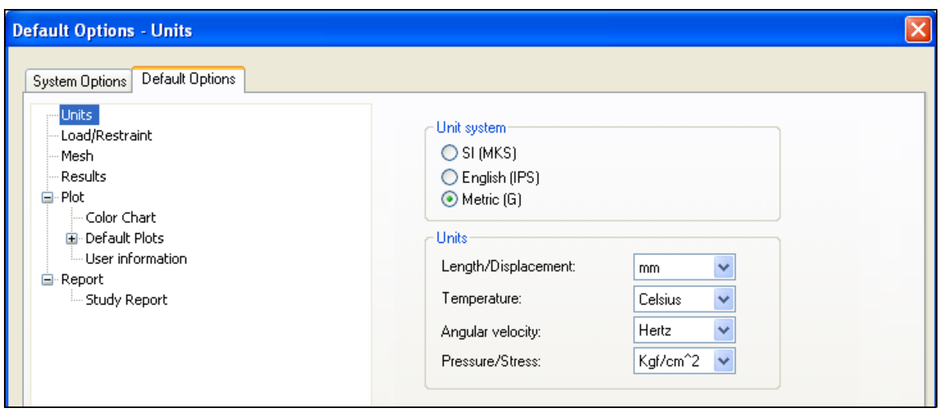
In the menu, select **Simulation**, **Options**.

4 Select the **Default Options** tab.

Select **Units** in the left pane.

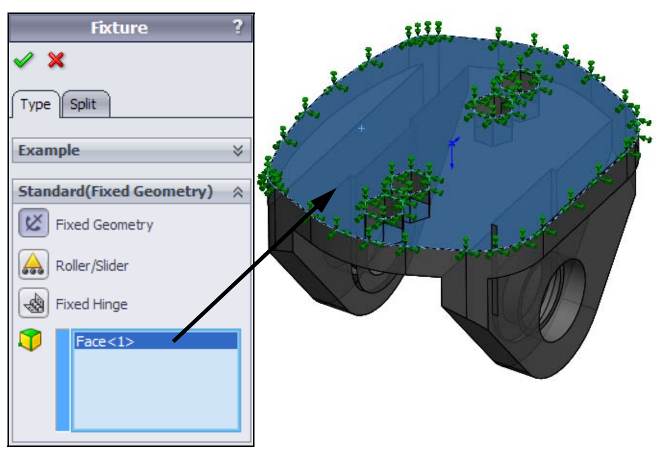
Set the **System of units** to **Metric (G),** the **Length units** to **mm** and **Pressure/Stress** to **Kgf/cm^2**.

Click **OK**.



5 In the SimulationManager, right-click `Fixtures` and select **Fixed Geometry**.

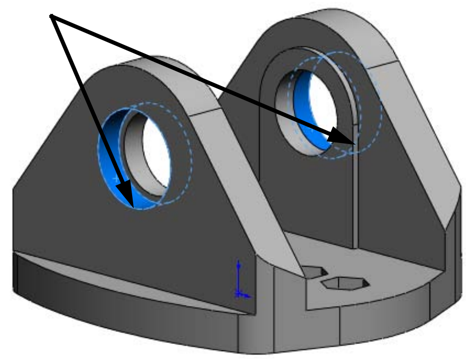
6 Select the bottom face of the **Truck**. Under Standard, select **Fixed** Geometry. Click **OK**.



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)**.

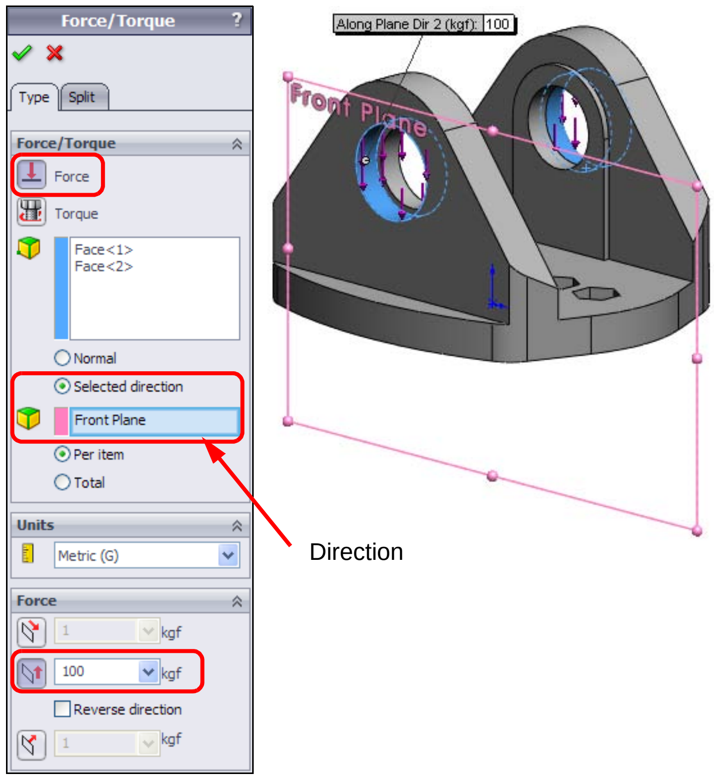
10 Select **Force**.

11 Select **Selected direction**, then select the `Front Plane` in the FeatureManager design tree.

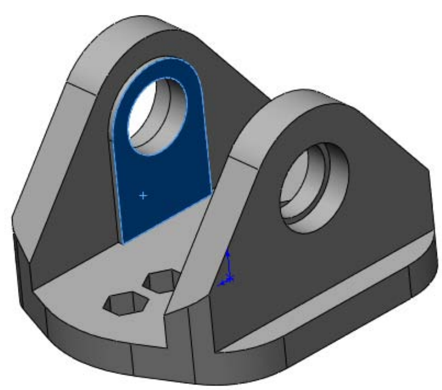
This will set the force the force direction relative to the `Front Plane`.

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**.

Click ✓.



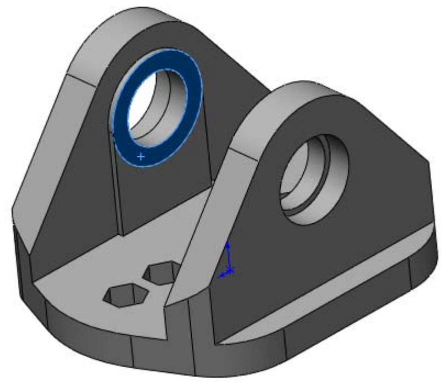
13 To apply the bending force, we must split a surface to limit the area where the force applies.



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.

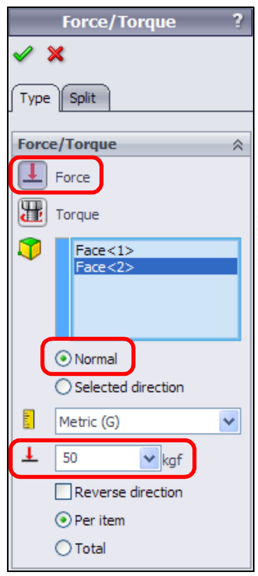


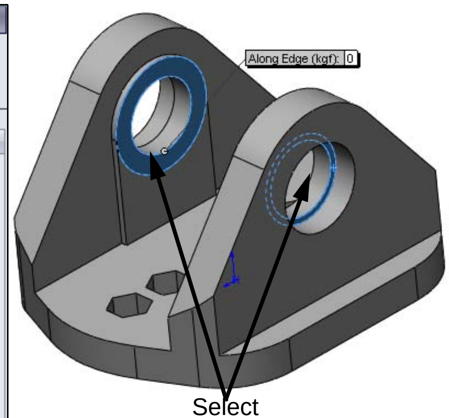
17 In the SimulationManager, right-click `External Loads` and select **Force**.

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.

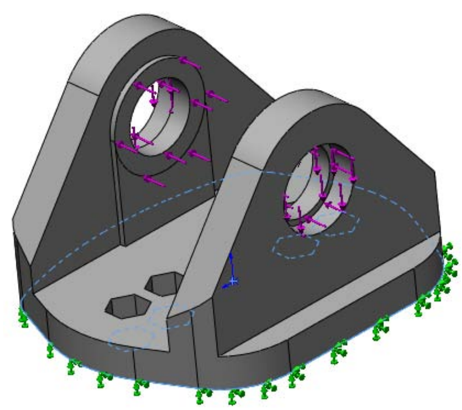
20 Click .





21 We now have all the forces and restraints to run the first analysis.

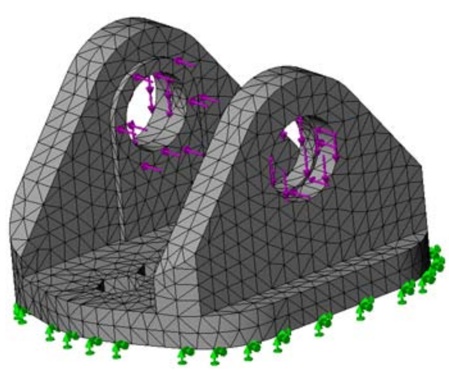
22 We applied a material (Nylon 6/10) to the part in SolidWorks so it is already applied to the part. We do not have to add the material again.



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**.

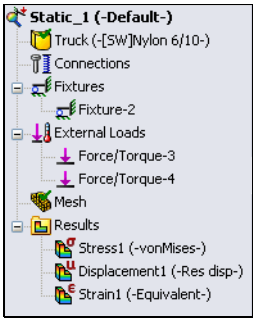
25 Run the analysis by right-clicking the study `Static_1` in the SimulationManager and selecting **Run**.



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.

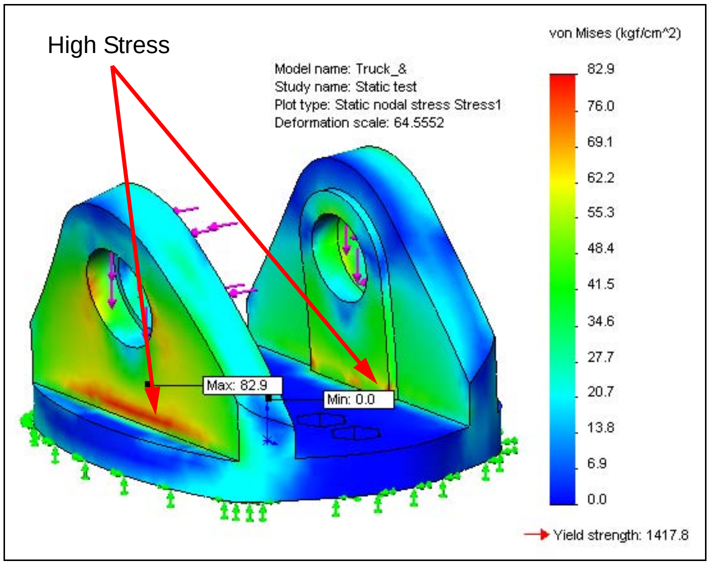
To display a plot:

- 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.

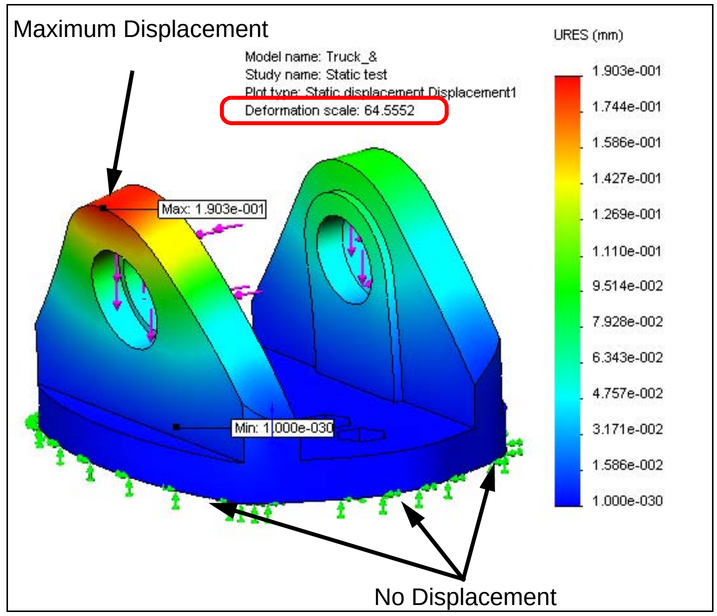
By looking at the color code, we can see that the stress is very small on the base plate. On the two vertical bosses, the stress increases as we move from the holes where the load is applied toward the base plate. We have the highest stresses where the vertical plates connect to the base plate. Much of this load is from bending.



Displacement Plot

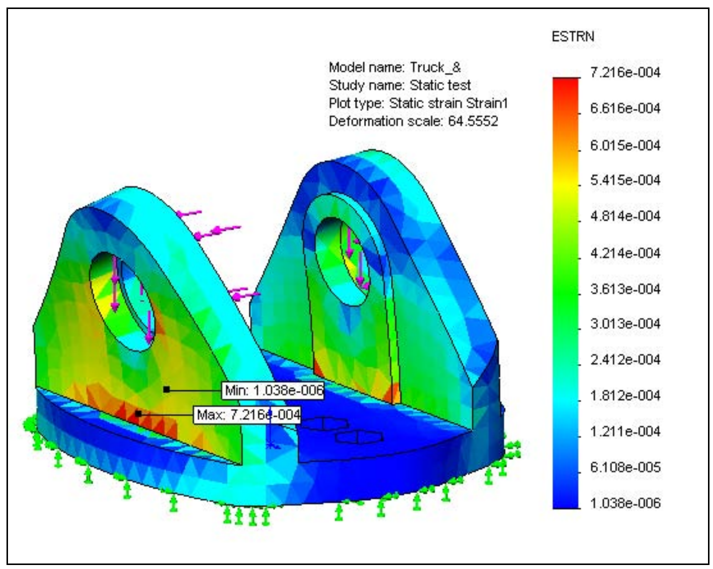
The Displacement Plot shows how much each element of the model moves because of the applied loads. This plot is to an exaggerated scale. Look at the information in the upper left, it shows that the displacement is shown almost 65 times the actual displacement.

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

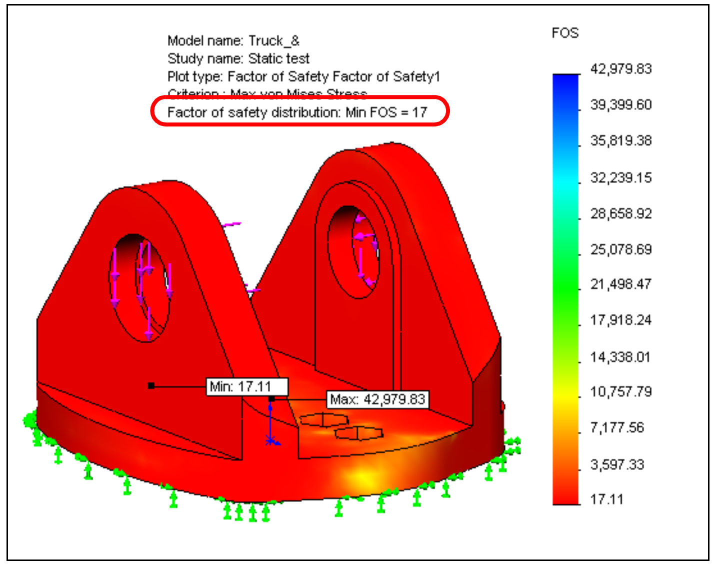
The Strain Plot shows the strain for each individual finite element.



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.

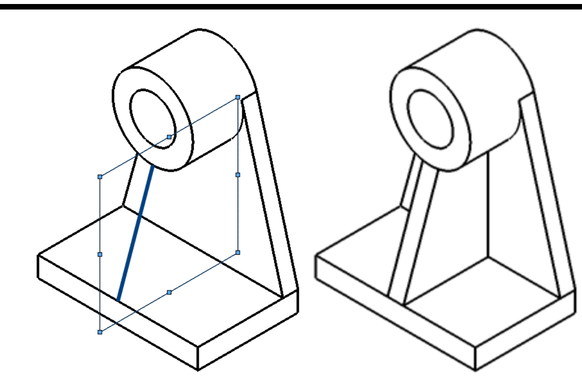
Ribs

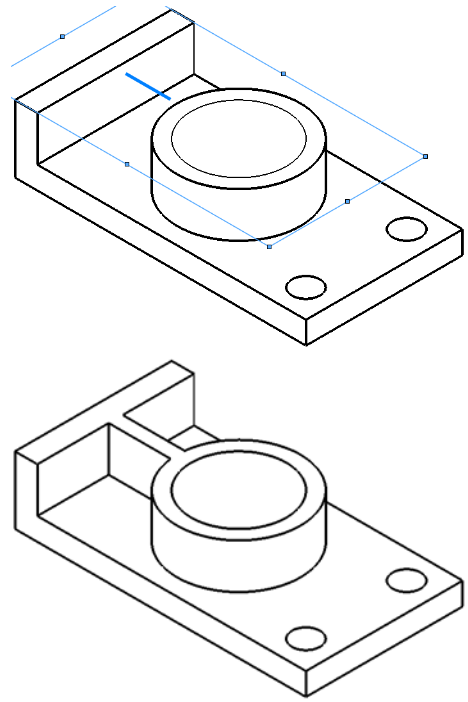
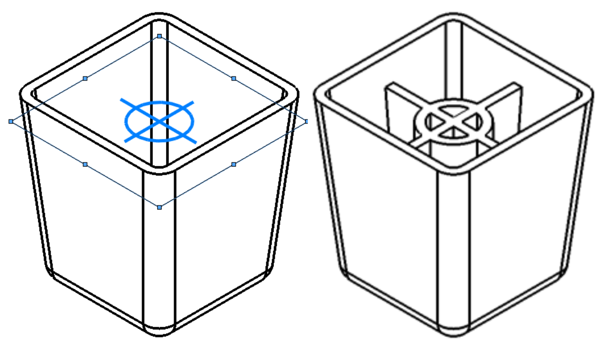
The rib tool, **Insert, Features, Rib…**, allows you to create ribs using minimal sketch geometry. The tool prompts you for the thickness, direction of the rib material, how you want to extend the sketch if necessary, and whether you want draft.

Rib Sketch

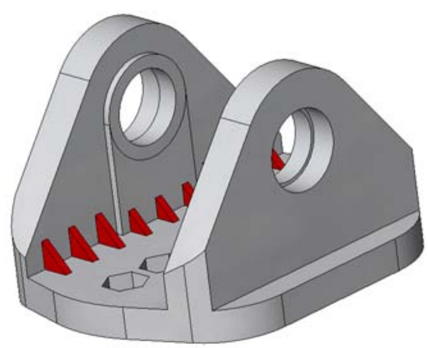
The rib sketch can be simple or complex. It can be as simple as a single sketched line that forms the rib centerline, or it can be more elaborate. Depending on the nature of the rib sketch, the rib can be extruded parallel or normal to the sketch plane. Simple sketches can be extruded either parallel to or normal to the sketch plane. Complex sketches can only be extruded normal to the sketch plane. Here are some examples

:

Simple sketch extruded parallel to the sketch plane.	

Simple sketch extruded normal to the sketch plane.	
Complex sketch extruded normal to the sketch plane.	

Task 9 — Add ribs to vertical bosses.

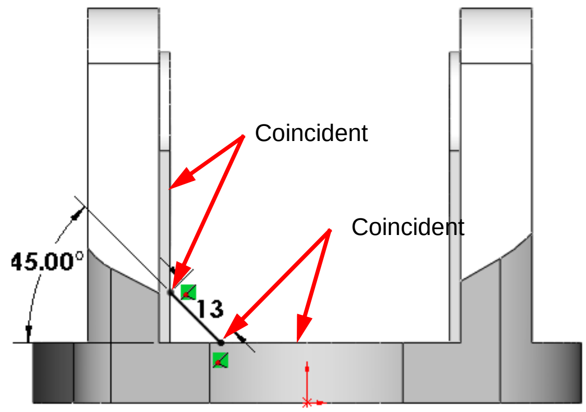


To help support the load transmission from the vertical bosses to the base plate, we will add some ribs between the two.

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. Create a sketch on the Right plane.
2. Change the model view to the Right view.

3 Sketch a single line making each end point coincident to the edges shown.

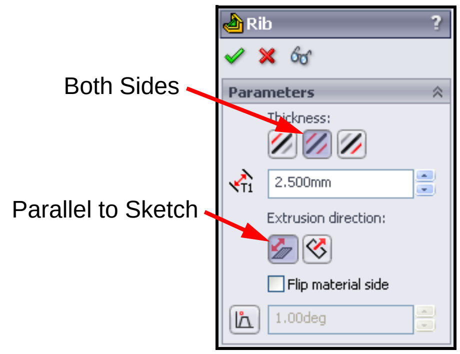


4 Click the **Rib** tool

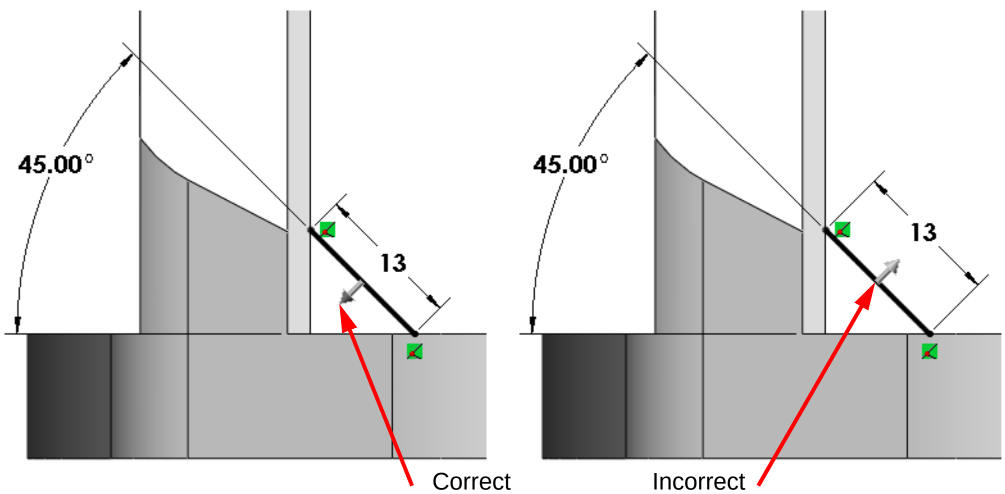
5 Select **Both Sides**. This will add material to both sides of the sketch line.

6 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.

7 Type **2.5mm** for the rib thickness.

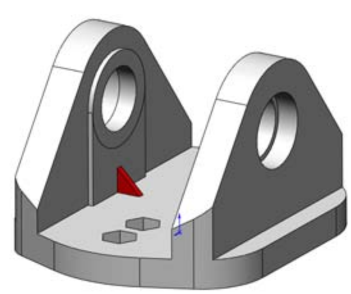


8 Examine the sketch in the graphics area. Make sure the gray arrow is pointing towards the intersection of the vertical boss and base plate. If it is not, right click on the arrow or select **Flip material side** in the PropertyManager.



9 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.

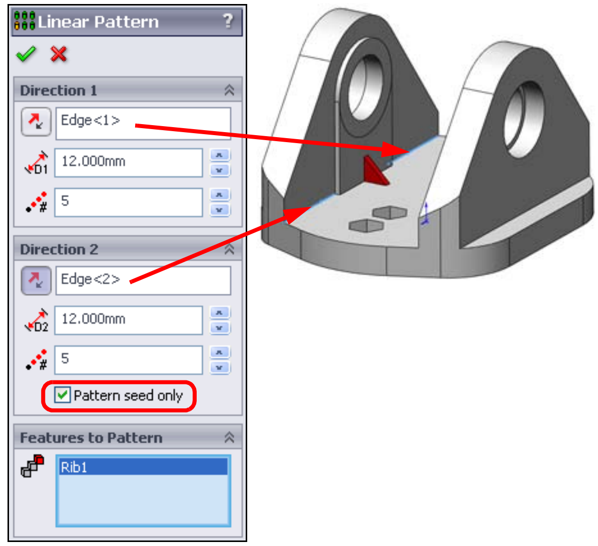


10 Now that we have one rib, the remaining ribs can be made using a pattern. Click **Insert, Pattern/Mirror, Linear Pattern**.

11 Select the two edges shown for direction 1 and 2.

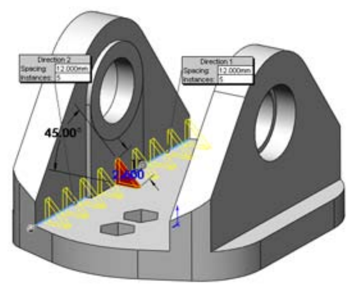
12 Set the spacing to **12mm** for each direction and the number of instances to **5**.

13 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.

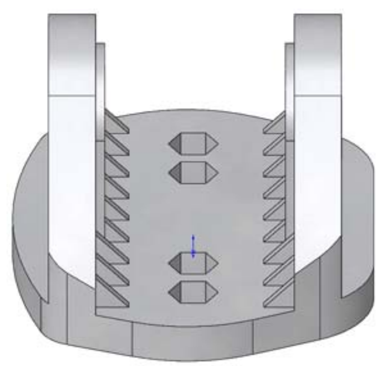


14 Click in the box **Features to Pattern** to make it active, then select the Rib either in the graphics area or the FeatureManager design tree. You should now have a preview of the ribs.

Click **OK**.



15 Repeat the above procedure to create ribs for the other vertical boss.



Task 10 — Remove material from the base.

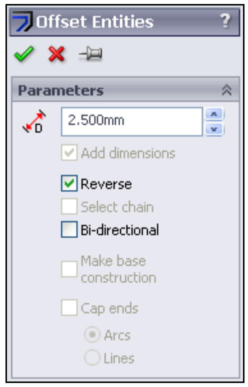
We can reduce the weight of the Truck by removing material from the three thick areas and then adding ribs to maintain the stiffness of the truck and restore some of the strength.

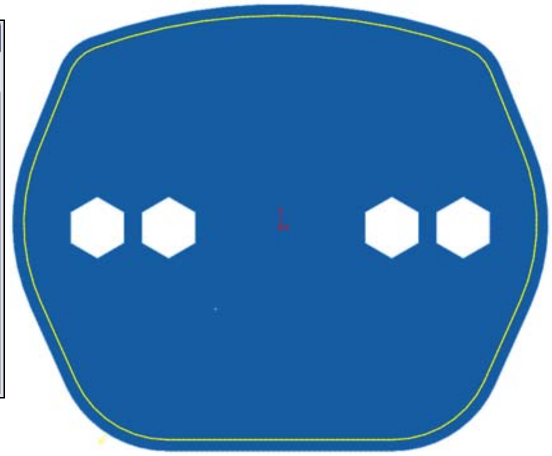
The first part of the task is to remove material from the base plate.

1 Create a sketch on the bottom face of the Truck.

2 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.





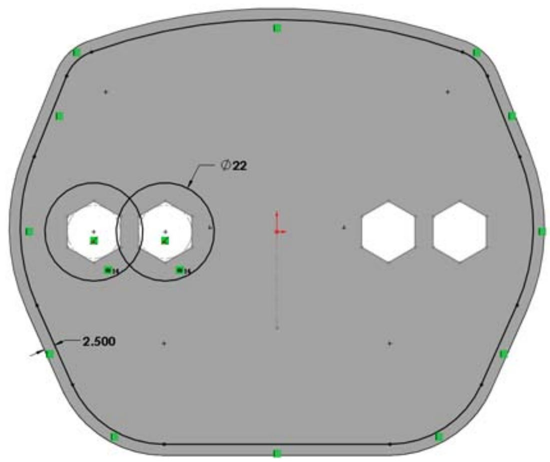
4 Show the sketch used to create the hex holes. Right-click the sketch under the feature `Hex Cuts` and select **Show**.

5 Sketch two circles centered on the two hex holes on the right.

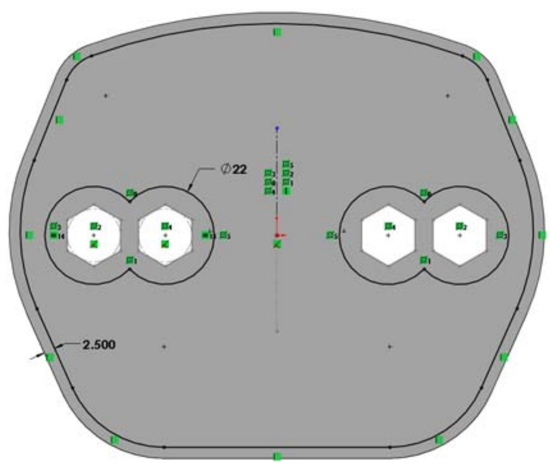
6 Add an equal relationship. Dimension one of the circles to **22 mm**.

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.

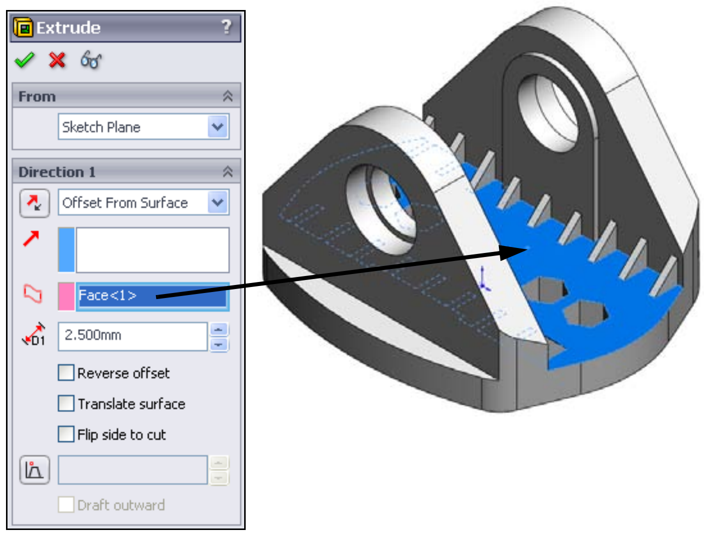
This allows a feature to terminate at a set distance from the selected surface. This can be used to create a cut that will always leave a specified thickness of material after the cutting operation.

10 Click **Extruded Cut**.

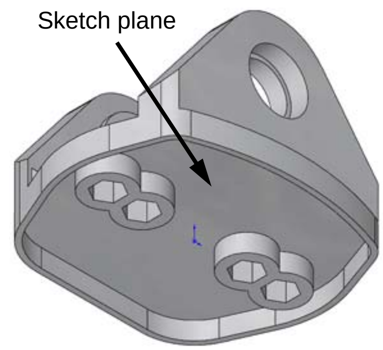
11 Select **Offset From Surface** for the end condition, then select the top face of the truck.

12 Type **2.5mm** for the distance.

13 Check the preview.



14 Click **OK**.



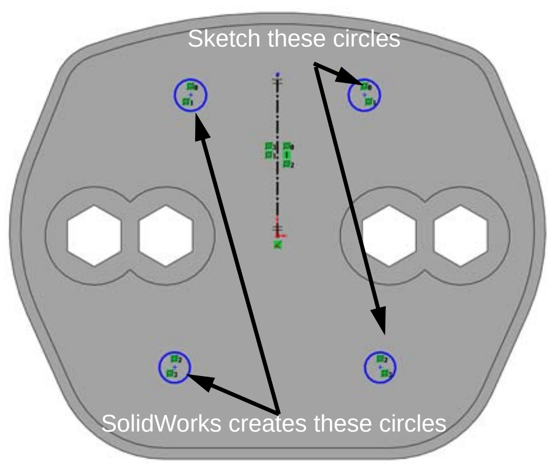
15 Hide the sketch under Hex Cut feature.

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** .



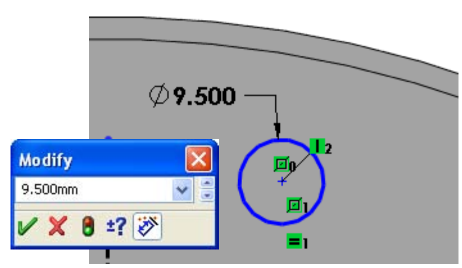
19 Sketch two circles to the right of the centerline, SolidWorks will create two mirror images of the circles.

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.

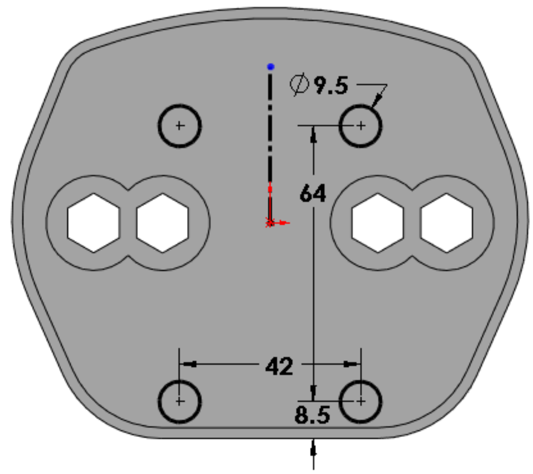
21 Add a **Vertical** relationship between the centers of the two circles you drew.

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.



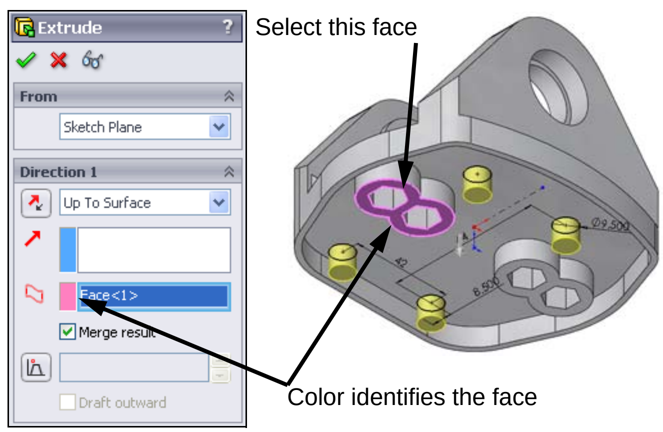
Up To Surface

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** . 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.

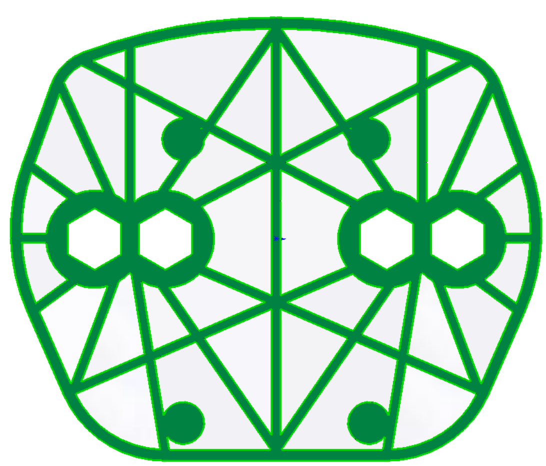
Click .



Task 11 — Create strenghtening ribs.

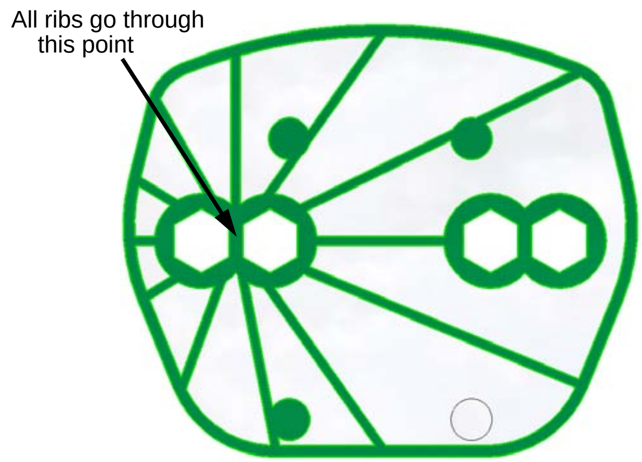
We can’t just leave the base plate hollowed out because it will not have enough stiffness to keep the vertical bosses upright. To strengthen it, we will add a web of ribs that look like this.

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.



Construction Geometry

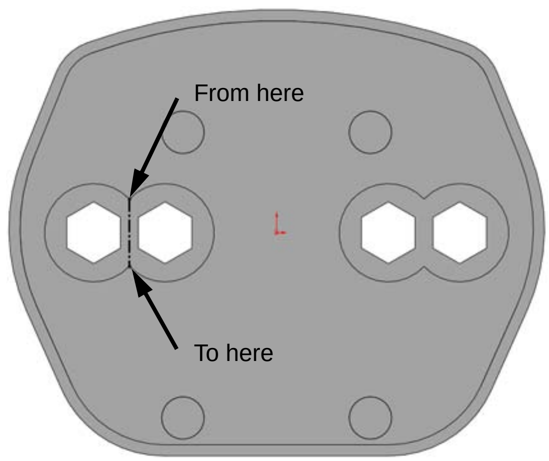
Construction geometry can be created to capture different relationships. This construction geometry can be very useful as it doesn't actually create anything. A simple example in the physical world would be the case of laying tile on a floor. To get the first row of tiles straight, we could use of a 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.

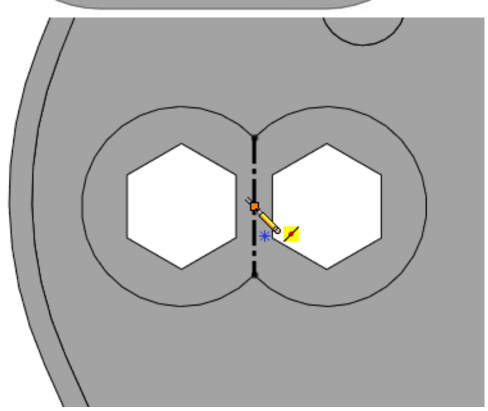
2 Click to start a centerline. A centerline is a line used for construction.

3 Sketch a centerline between the two vertices as shown.

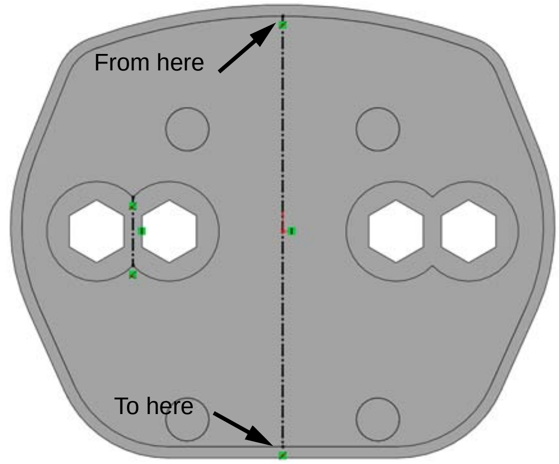
4 Click **Point** on the Sketch toolbar.



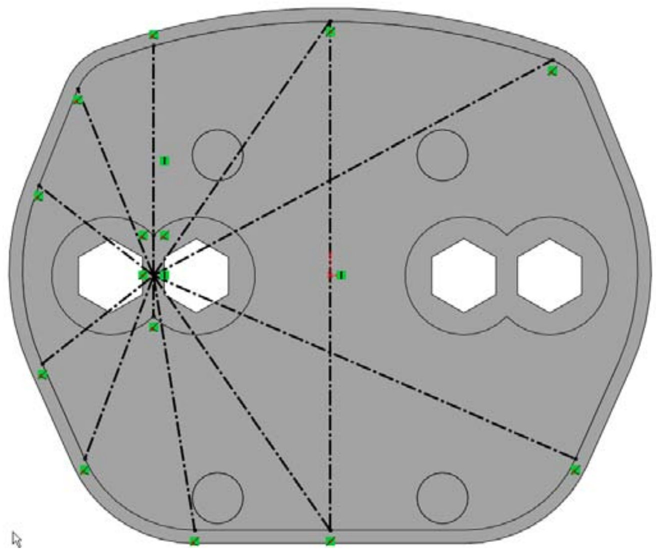
5 Move the cursor over the centerline until the midpoint symbol appears. Click on the midpoint symbol to create a point at the midpoint of the centerline.



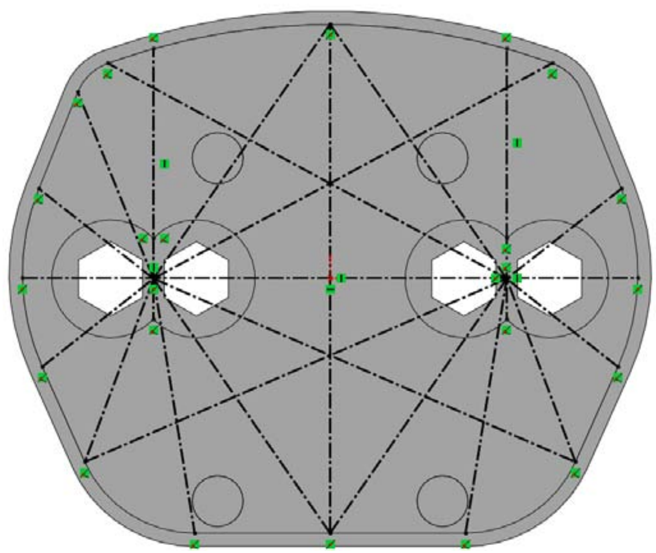
6 Sketch a centerline from the midpoint of the top arc to the midpoint of the bottom inside edge. This represents the center of the part.



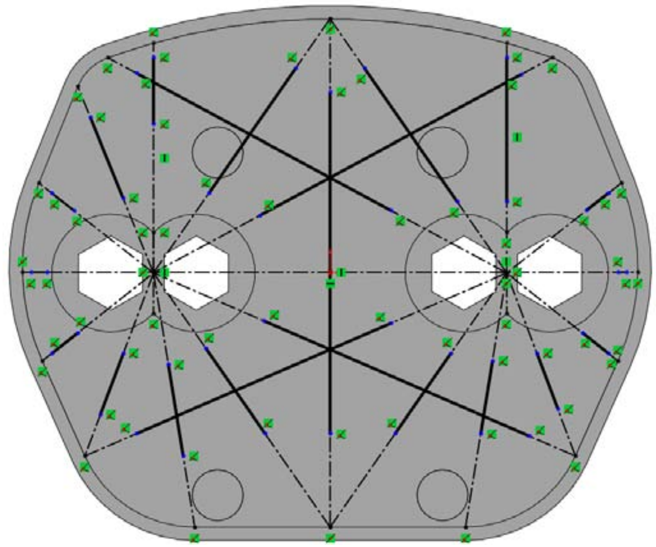
7 Sketch the centerlines shown. Each centerline starts at the point. Except for the horizontal and vertical centerlines, the other end of each centerline is coincident to the endpoint of an arc on the inner edge of the base.



8 Create the centerline and point on the right side of the model and create the remaining centerlines.

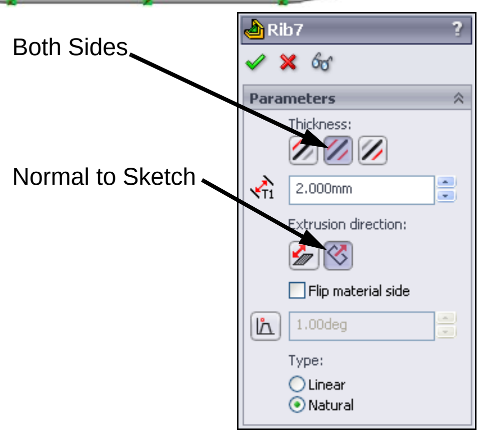


9 Sketch lines on top of the centerlines. These lines do not have to extend all the way to the edges because the rib tool will automatically extend the rib up to the next geometry. Each line however must cross all the other lines in the sketch. If they do not, the rib will stop at the next line rather than extending all the way to the existing part geometry.

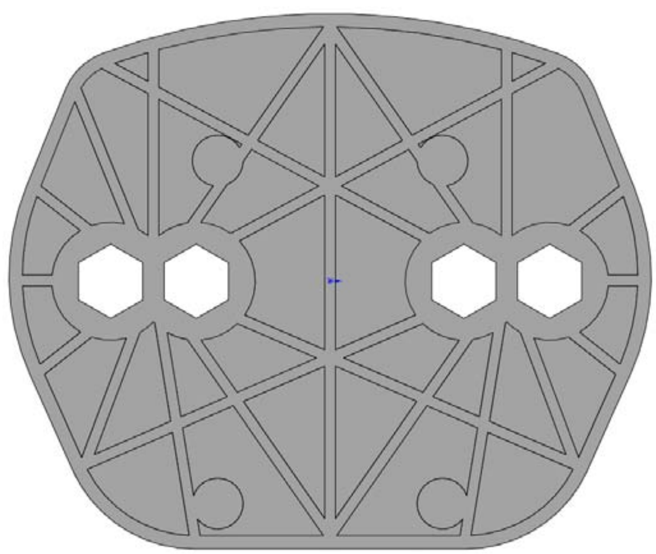


10 Click **Rib** and set the rib width to **2mm**.

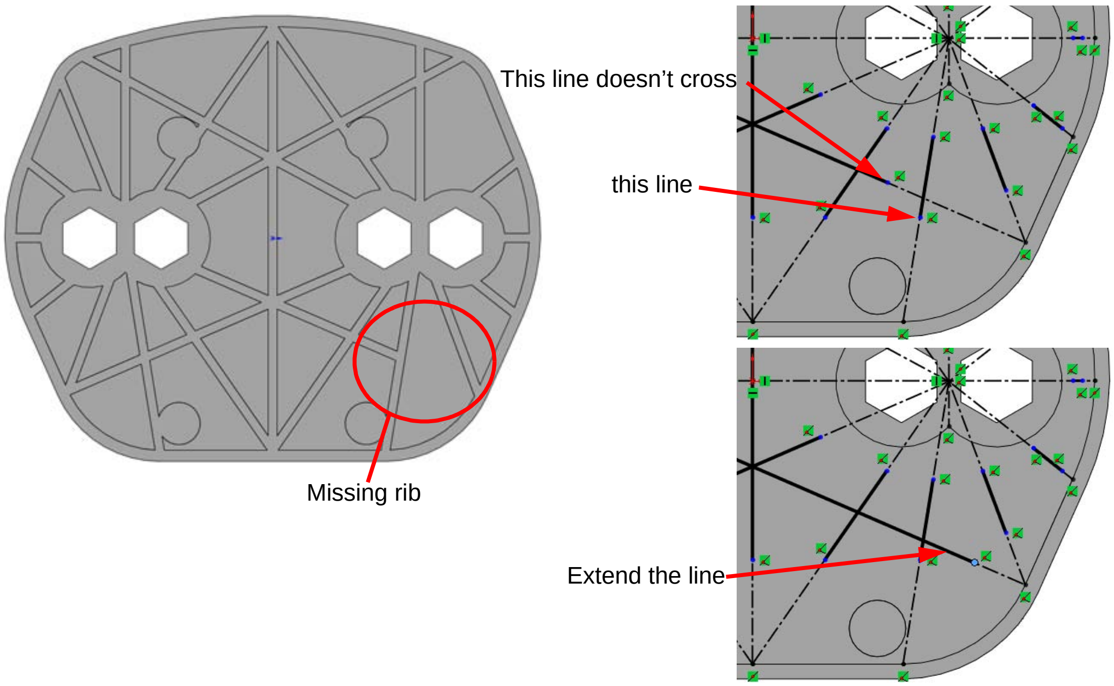
Select **Both Sides** and **Normal to Sketch**. This will center the ribs on the sketch lines and extrude the ribs in a direction normal to the sketch plane instead of in the sketch plane as was done in the previous rib.



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:

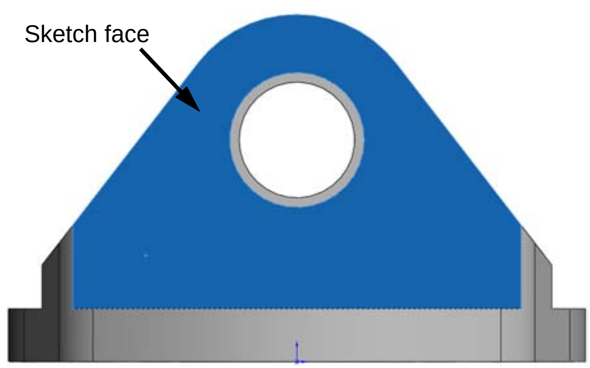


Task 12 — Add ribs to vertical boss.

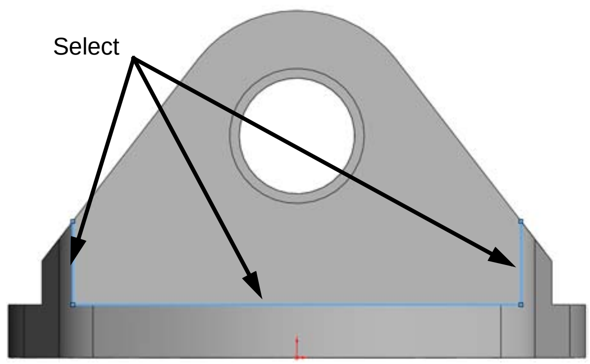
In this task, we will need to remove material from each of the vertical bosses, then add ribs to stiffen the part.

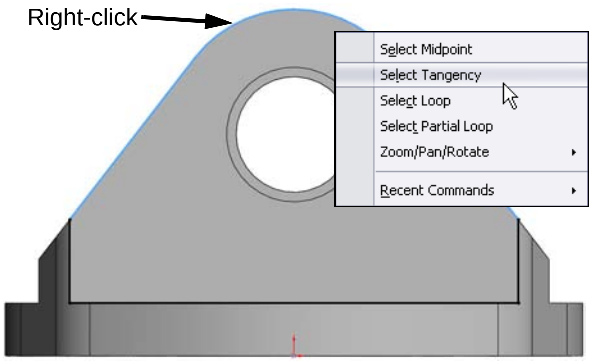
1 Orient the model to the `Front` view.

2 Create a sketch on the face shown.



3 Select the three edges shown, then click **Convert Entities** . 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.

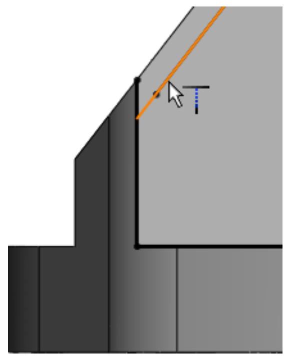
5 Click **Offset Entities** . Set the offset to **2.5mm** to the inside and click **OK**.

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.

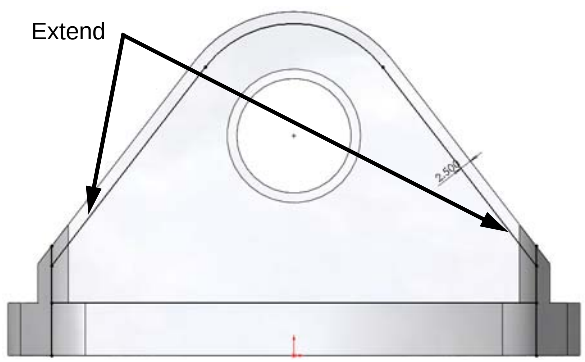
Extend Tool

Extend can be used to lengthen sketch geometry.

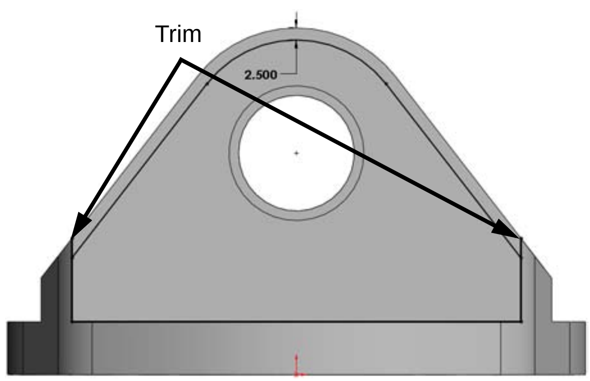
7 Click **Extend Entities** on the Sketch toolbar or **Tools, Sketch Tools, Extend** from the menu. When you move the cursor over a line, the extended line will be previewed. **Extend Entities** will extend the line until it intersects the next sketch entity.



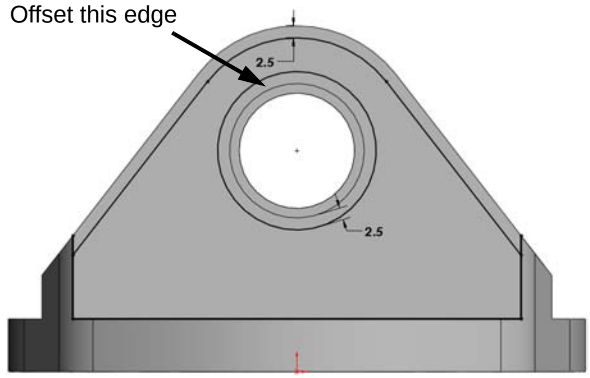
8 Extend the two lines as shown.



9 Trim the sketch to obtain a single closed profile.

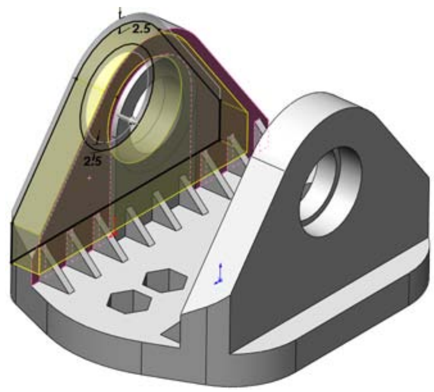


10 Create an additional 2.5mm offset from the edge of the circular cutout. This will be used to hold the bearing.



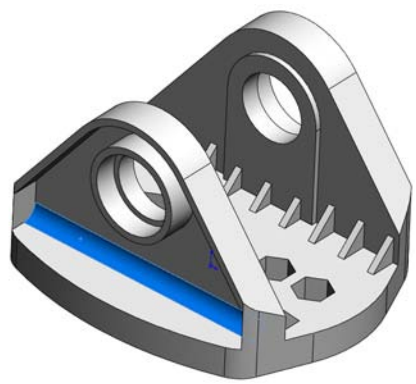
11 Extrude a cut using **Offset From Surface**. Type **2.5mm** for the offset distance.

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.

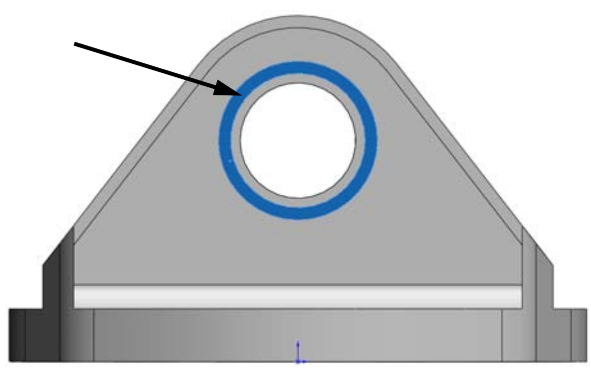


Task 13 — Add ribs.

1 Orient the model to the `Front` view.

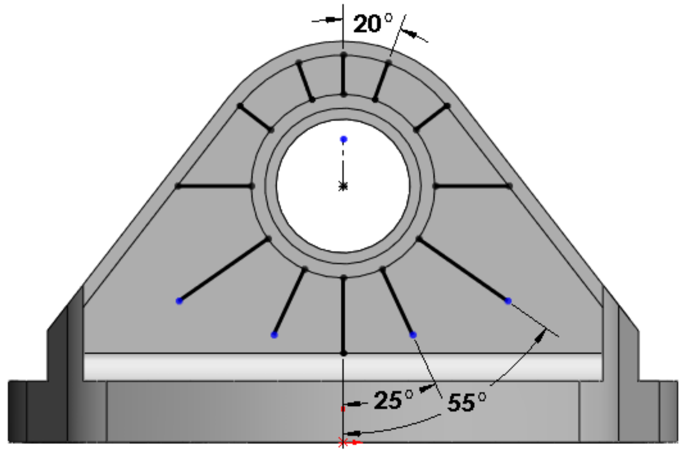
2 Create a sketch on face shown.

3 Click **Point** and add a point **Coincident** to the center of the hole.



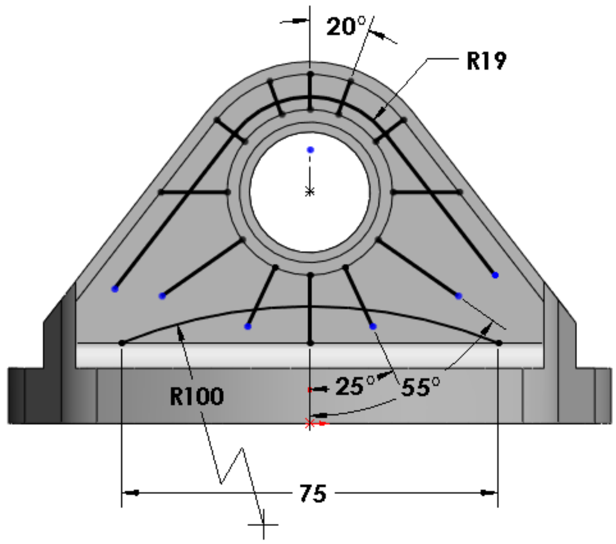
4 Create the following sketch.

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.

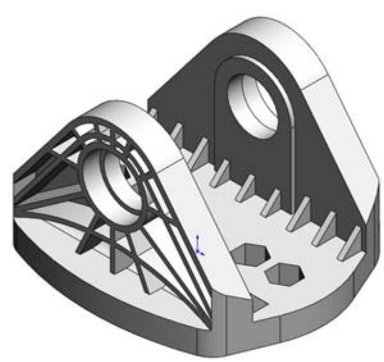


TIP: Use **Mirror Entities** and **Symmetric** relationships to reduce the amount of sketching.

5 Add the additional arcs and lines to complete the sketch.

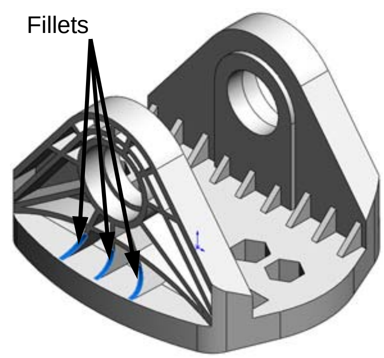


6 Create ribs. Use the rib tool to create ribs that are **2mm** thick and have **2°** of draft.



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.

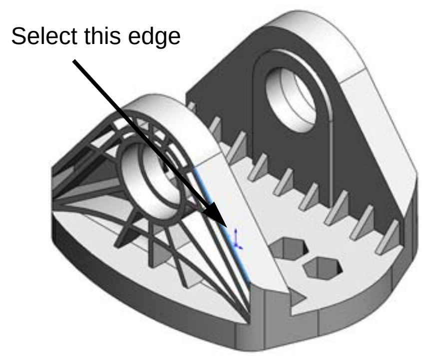


Task 14 — Remove more material

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** on the Features toolbar.

2 Select the edge shown.



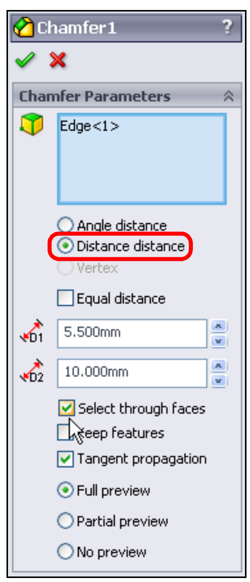
3 Select **Distance distance** for the type of chamfer.

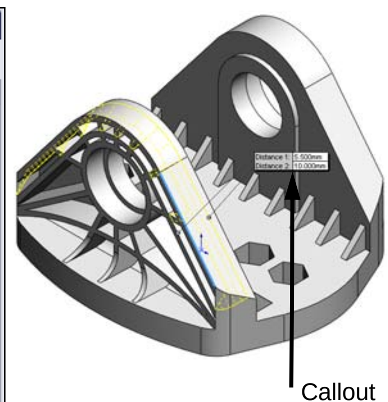
4 Type **5.5mm** for the first distance and **10mm** for the second distance.

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.





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.

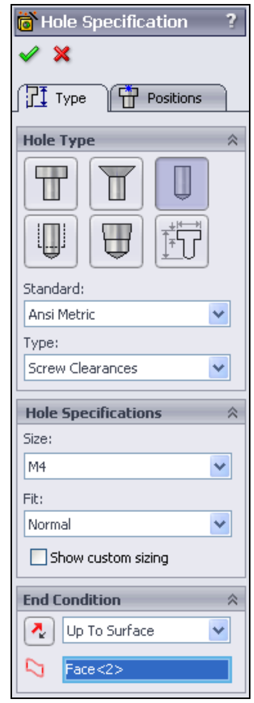
2 Select the bottom face of the model.

3 Click the **Hole Wizard**  on the Features toolbar.

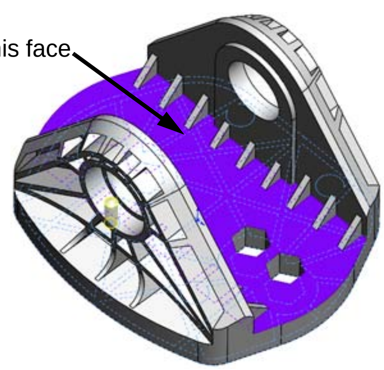
4 Click the **Hole** button.

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.

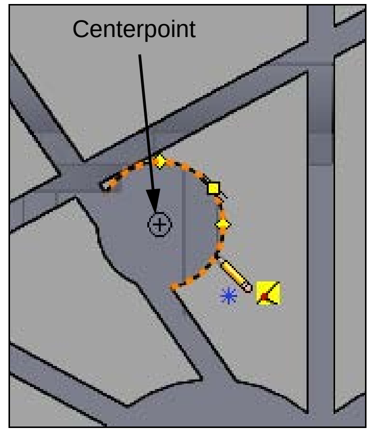


6 Click the **Positions** tab.

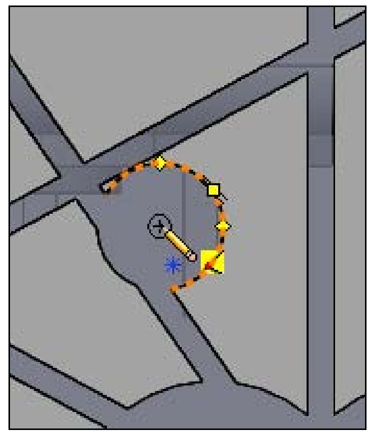
7 Place the holes. This portion of the wizard is used to locate and fully define the center point of the holes. A sketch point is added as the hole center point.

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.



10 Move the cursor to the centerpoint and select it. This will add a concentric relationship between the point and the circular edge.

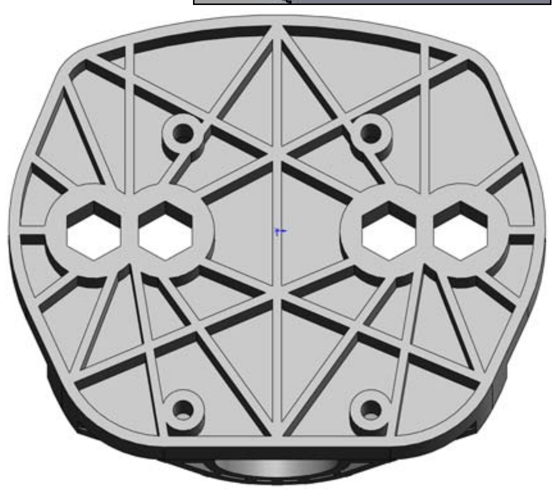


11 Repeat this procedure for the remaining three bosses.

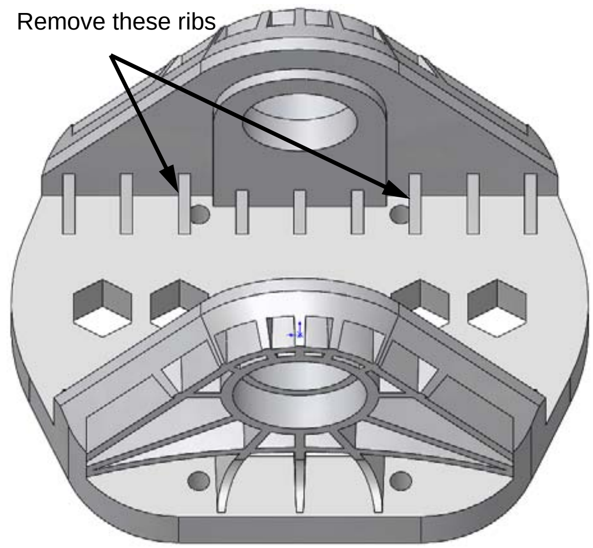
12 Click **Point** to turn it *off*.

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**.



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.

1 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.

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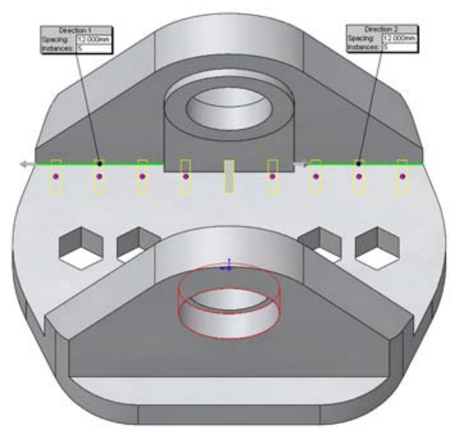
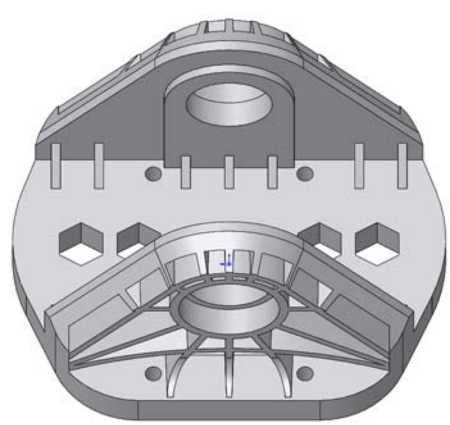
2 Right-click the pattern feature and select **Edit Feature**.



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.

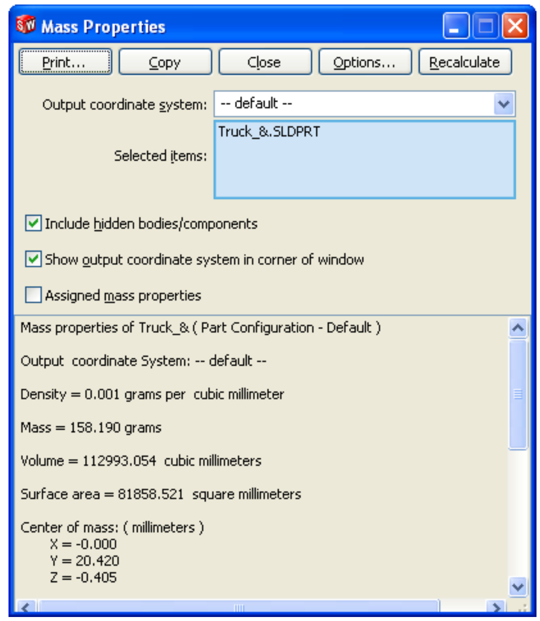
Now that the model is complete, we analyze it again to make sure it is still strong enough with the material removed.

1 Check the weight of the part. Click **Tools, Mass Properties**.

2 The part now weights 158.190 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.

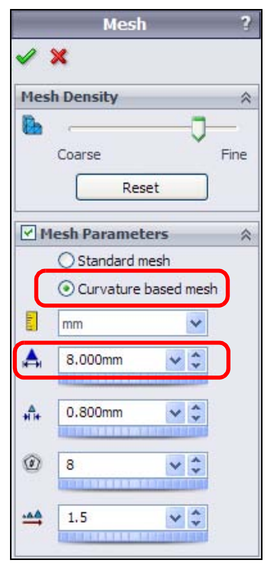


5 Re-mesh the model. Right-click the `Mesh` icon and select **Create Mesh**.

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 .

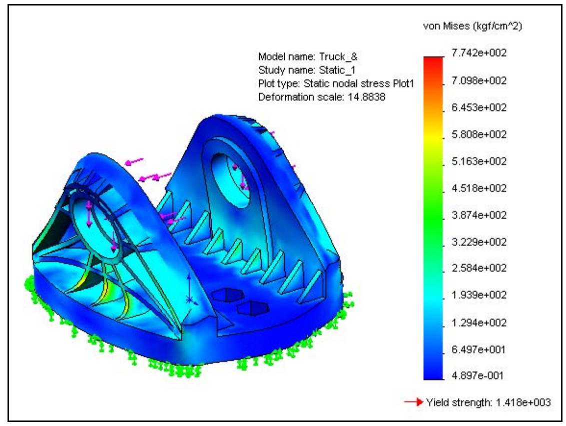


Note: Meshing techniques and settings are a topic in themselves and are beyond the scope of this project. For now we will just accept that these settings are needed to obtain a proper mesh of this model.

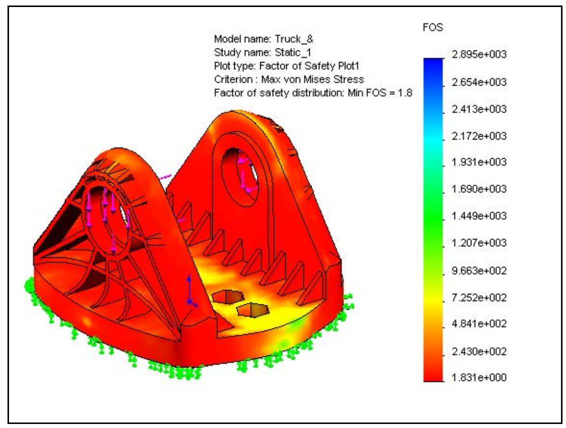
---

8 Rerun the analysis. Right-click the study by right-clicking `Static_1` in the SimulationManager and selecting **Run**.

9 Examine the plots. The stress plot shows that the maximum stress of 7.742 x 10^2 kgf/cm^2 is still below the Yield Strength of the material.



10 The Design Check plot shows a minimum factor of safety of 1.8.



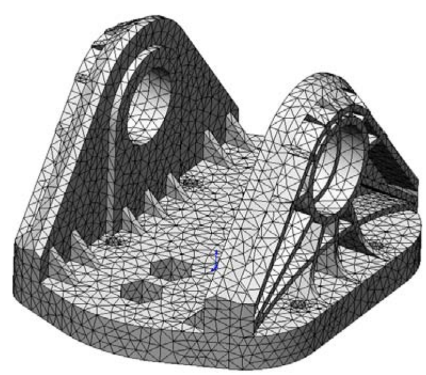
Continued refinement.

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.

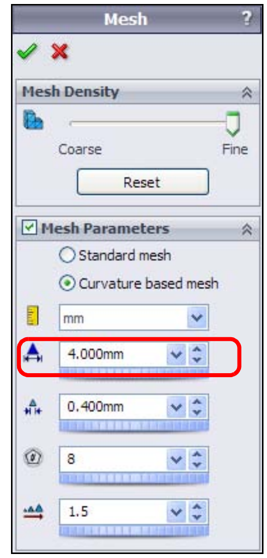
Task 18 — Refine the analysis

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**.

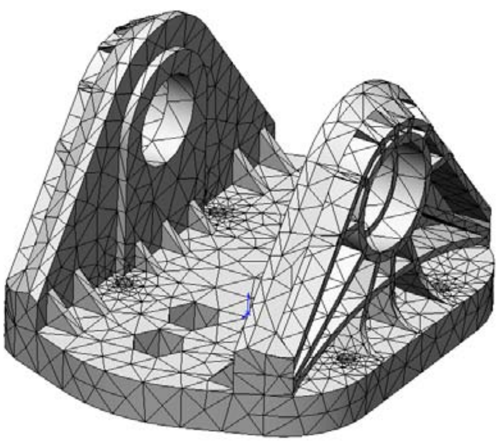


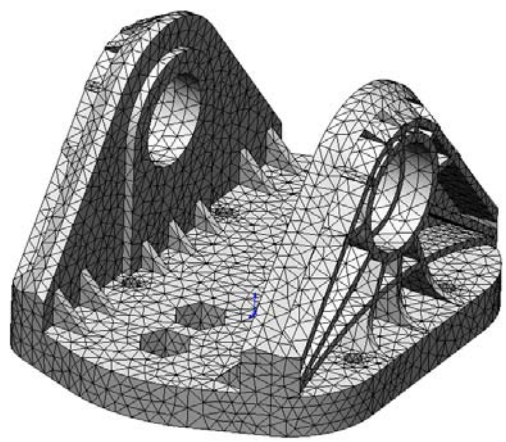
2 Type **4.0mm** for the mesh element size. This will create smaller elements.



3 Click **OK** to create the mesh.

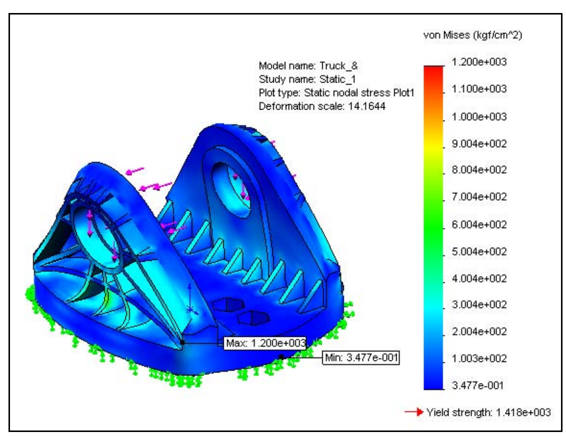
This mesh uses elements half the size of the previous mesh. The result is about four times the total elements than used in the first run. Also note that the mesh has adjusted its size around some areas such as the holes.


First mesh

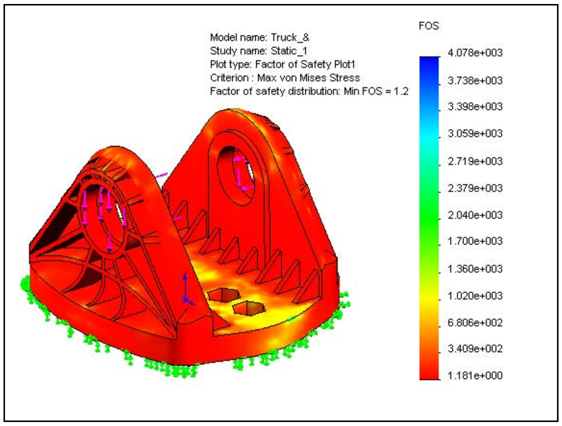

Second mesh

4 Run the analysis. This will take longer than the previous analysis due to the higher number of elements.

5 Examine the results. The Maximum stress is 1.200 x 10^3 kgf/cm^2, this is higher than the previous result.



6 The FOS is 1.2 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.

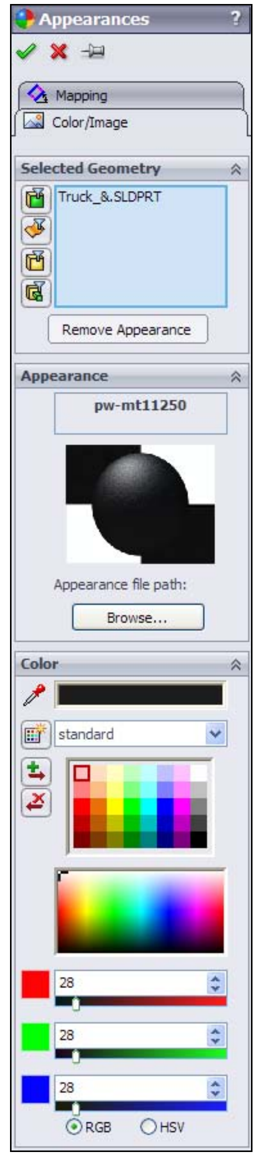
- Small Deformations

  The material will flex considerably under impact loads, so we have probably exceeded the small deformations limit.

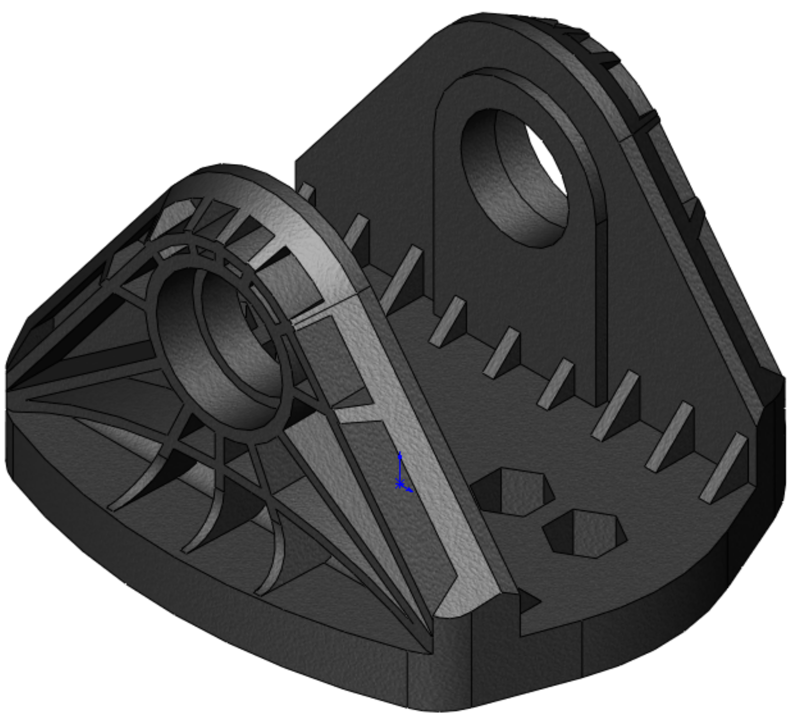
We also made some assumptions about the size of the load. While these load assumptions and limitations of SolidWorks Simulation were acceptable during the early development phase of this part, we would need to use additional tools to get us closer to the loads this part will see in use. The calculations 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 Appearance tab on the Task Pane.
2. Expand the **Appearances** and **Plastic** folders and select **Textured**.
3. 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.
4. Click .
5. **Save** and **Close** the part.



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?

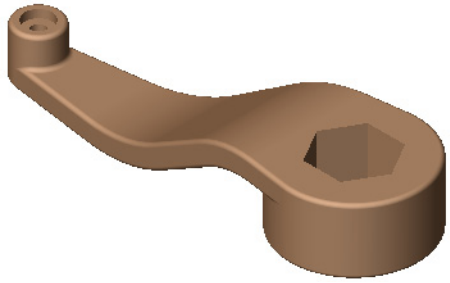
   ____

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**

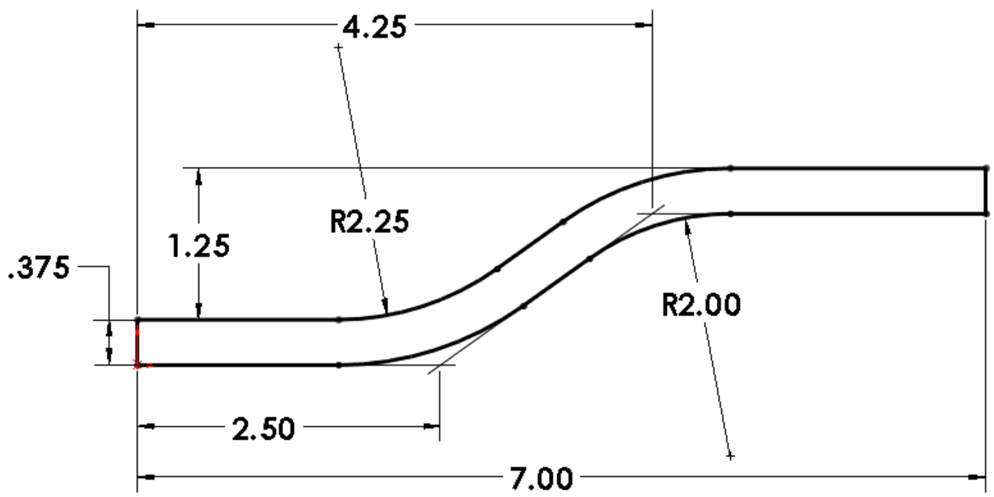
Procedure

Open a new part using the `Part_IN` template and name it `Mbody1`.

1 Sketch first profile.

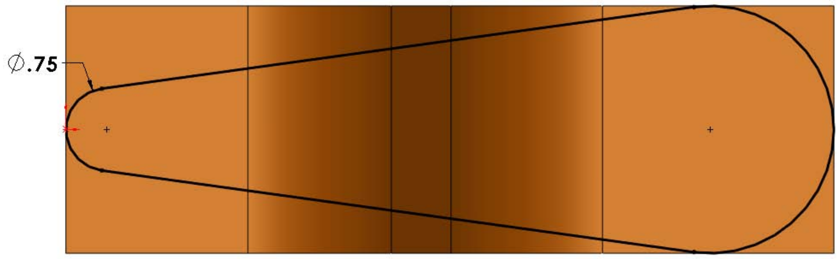
Use lines, fillets and offsets.

Extrude the profile **2.25in** using a mid-plane end condition.



2 Sketch second profile.

Extrude as required.



3 Combine bodies.

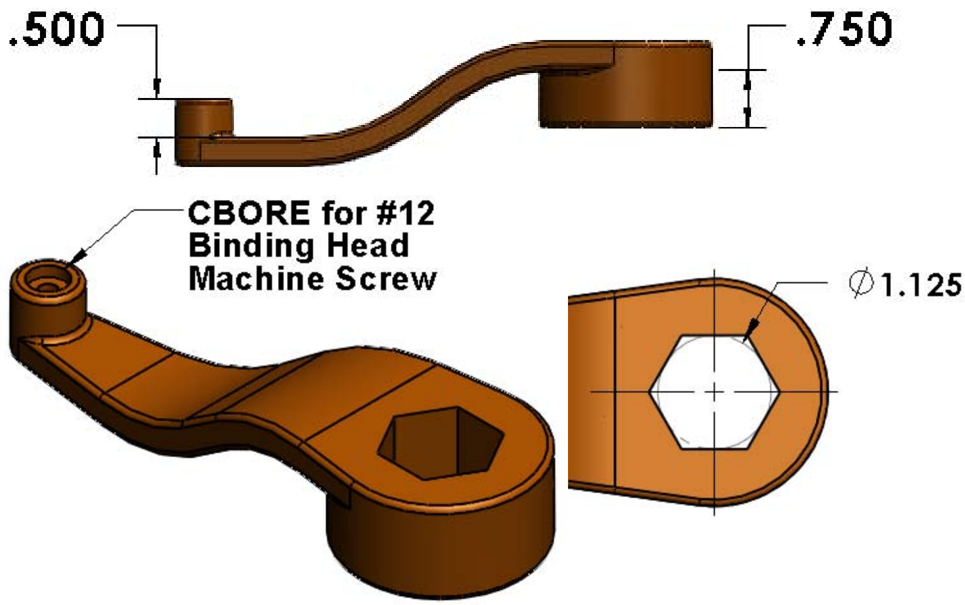
Combine the two solid bodies into one.

4 Add features

Add boss, cut, hole wizard and fillet features.

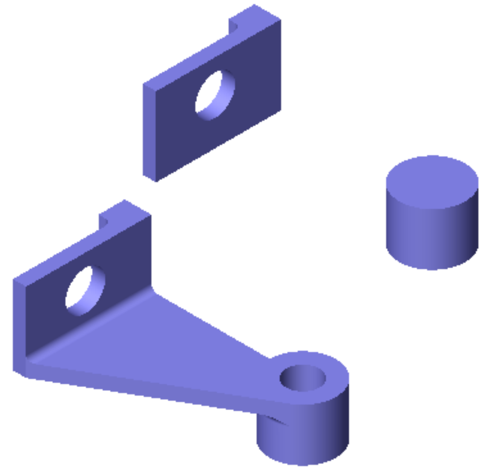
5 Finish part with **0.0625”** radius fillets and rounds.

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

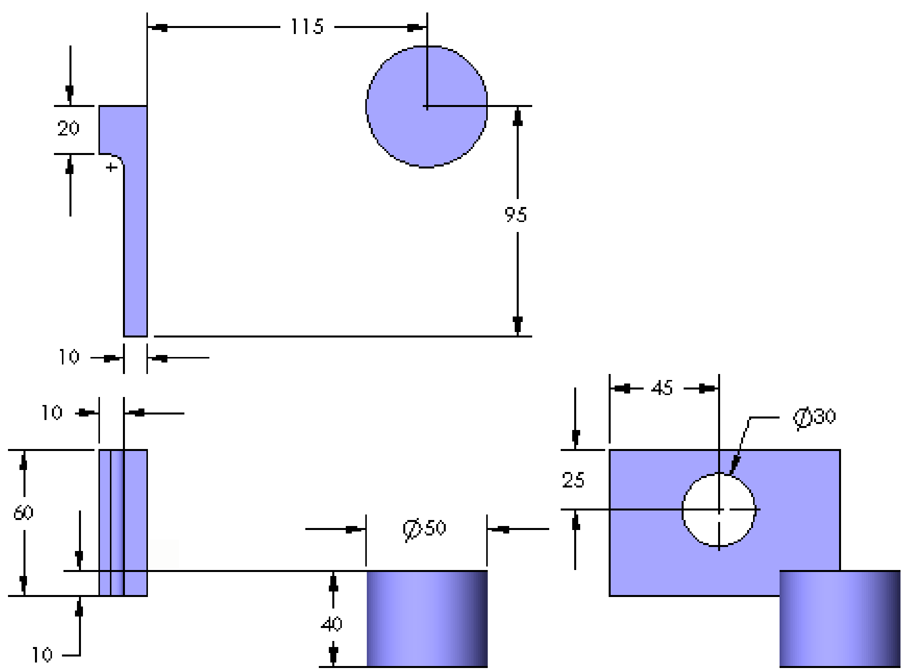
The design intent for this part is as follows:

- Part is *not* symmetrical.
- Holes are though all.
- All fillets and rounds are **5mm** radius.

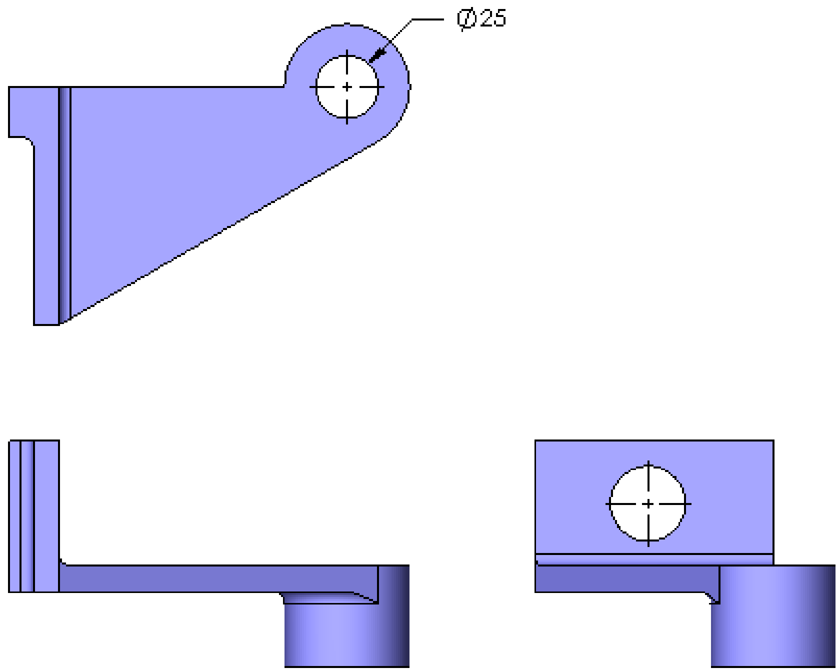
Procedure

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: _________ 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

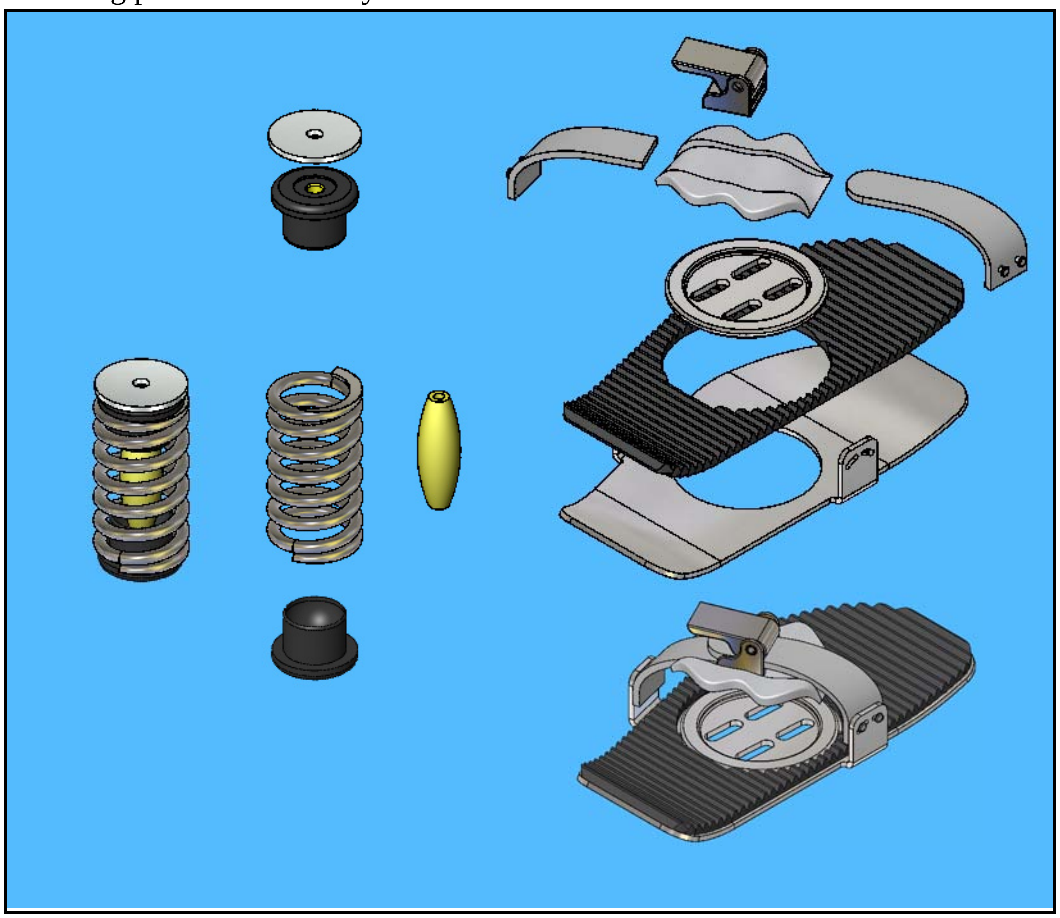
- Using multibody techniques, we can create a part or feature by creating extrusions based on the standard views of the final part, then combine them into a single body.
- SolidWorks Simulation is used to analyze parts and assemblies to determine the internal stress, strain, deformation and factor of 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.

7

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.

- ❑ *Revolves and Sweeps*
- ❑ *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

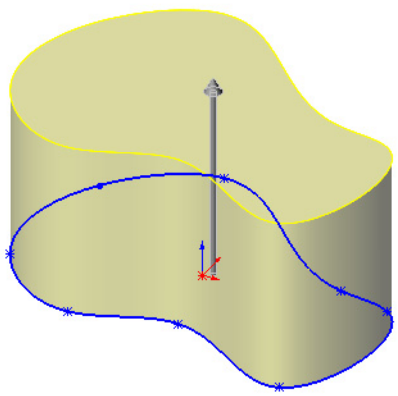
- ❑ In Class Discussion
- ❑ Active Learning Exercises, Part 1 — Creating a Spring
  - Creating a sweep
  - Composite curves
  - Create a user defined plane
  - Create an axis
- ❑ Active Learning Exercises, Part 2 — Create an Assembly
  - Create an exploded view
  - Animate the exploded view
- ❑ Active 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

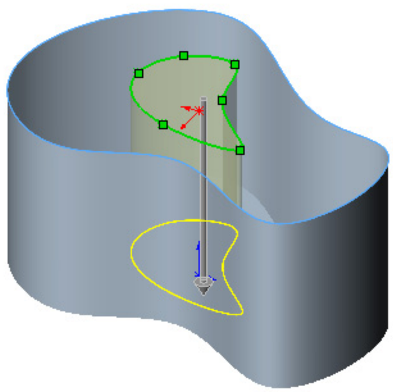
Extrusions and Revolves can be used to create a large number of features, however they have limitations when it comes to complex shapes.

Quick Review:

- ❑ Extrusions: A sketch is moved along a straight path to add material (Extruded Boss) or remove material (Extruded Cut).

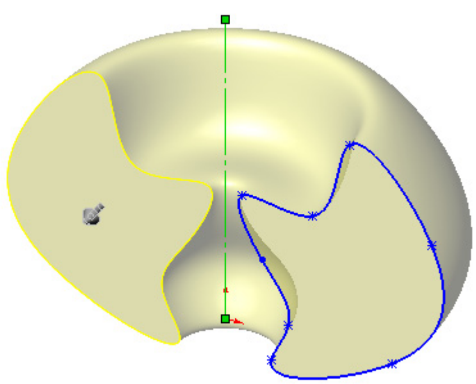


Extruded Boss

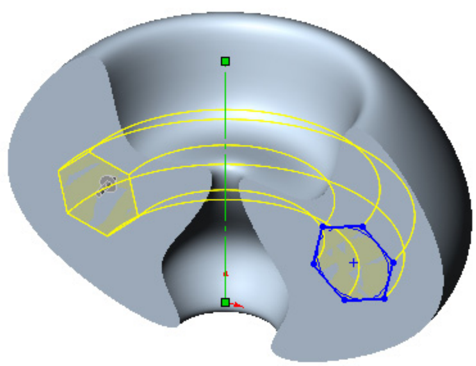


Cut-Extrude

- ❑ Revolves: A sketch is rotated about a centerline or edge to add material (Revolved Boss) or remove material (Revolved Cut).



Revolve

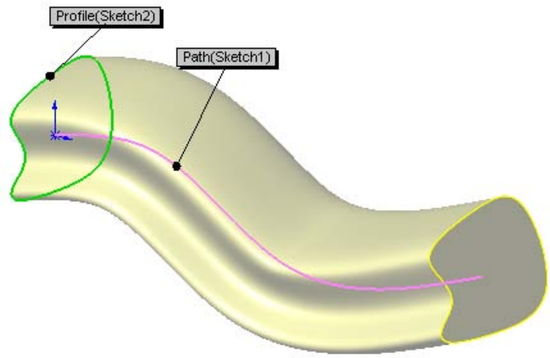


Cut-Revolve

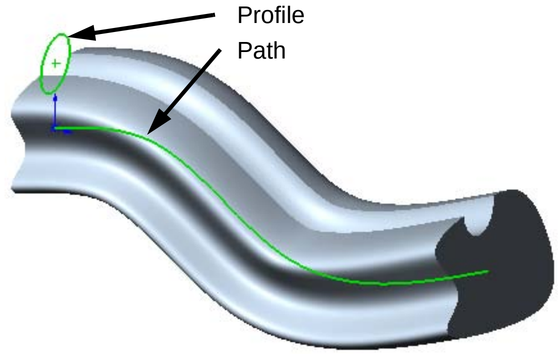
More complicated shapes can be created with Sweeps and Lofts.

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.

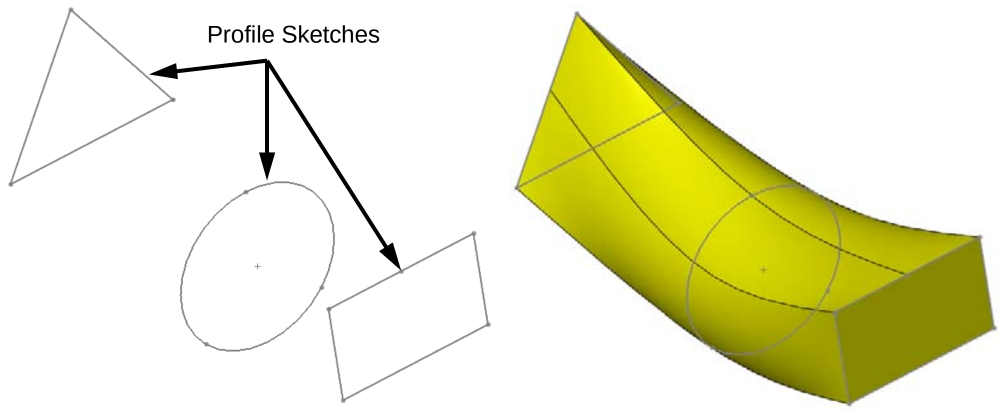


Sweeps can also be used to remove material as a **Swept Cut**.



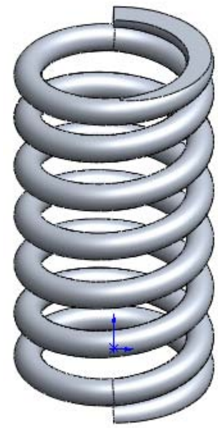
Lofts

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

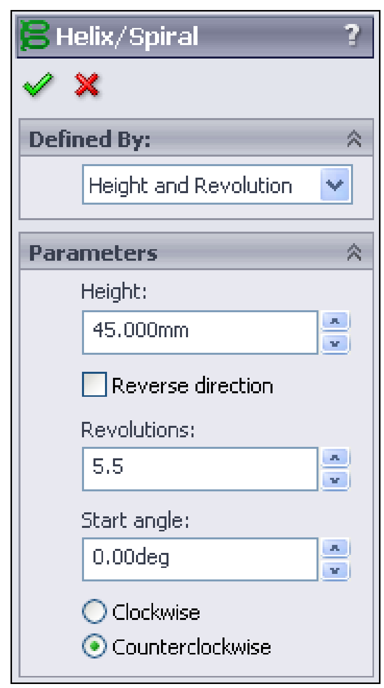
A spring is created by forming a rod into the shape of a helix. To create a spring, we will sweep a circle along a helix.

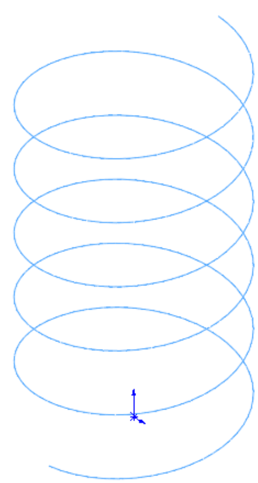
1 Open a new part using the `Part_MM` template.
2 Create a sketch on the `Top` plane.
3 Sketch a circle and dimension its diameter as **25mm**.

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 **0deg** for the **Start angle**.
8 Click **OK**. The helix will be the sweep path for the spring.

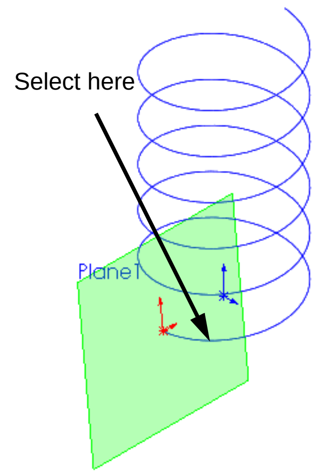




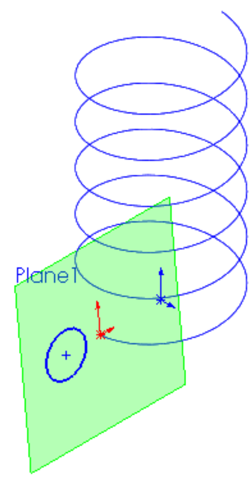
Task 2 — Create a Profile

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** relationship.

4 Dimension the circle to a diameter of **4mm**. The sketch should be fully defined.

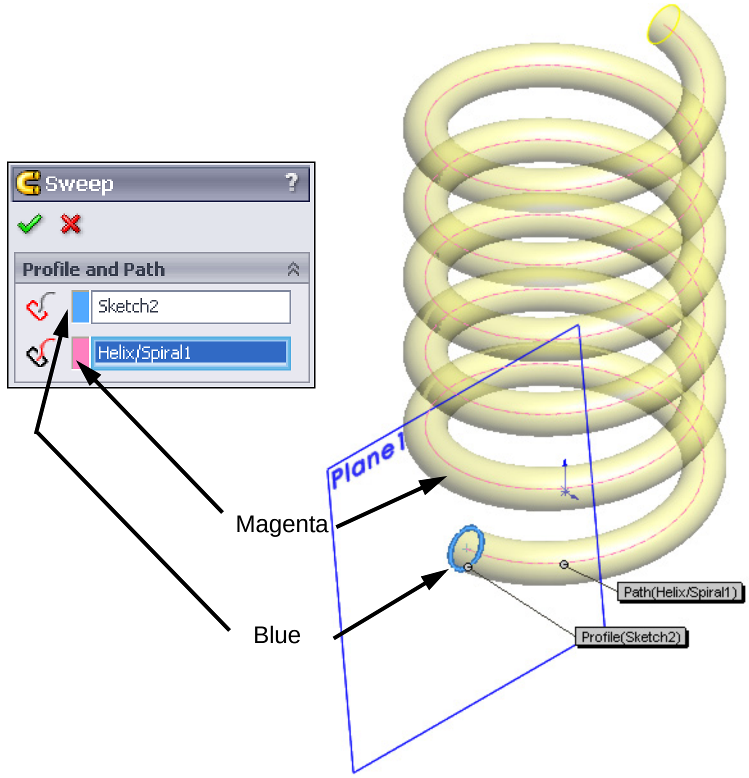
5 **Exit** the sketch.

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.

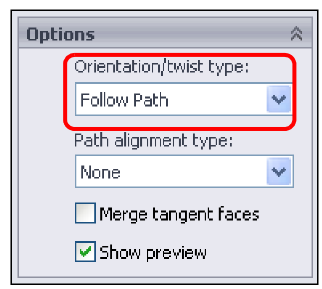
6 Create the sweep. Click **Swept Boss/Base** on the **Features** toolbar.

7 Select the circle for the profile and the helix for the path.

The callouts and color help to identify which sketch is which.

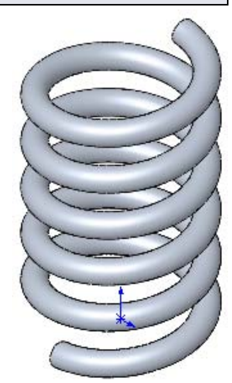


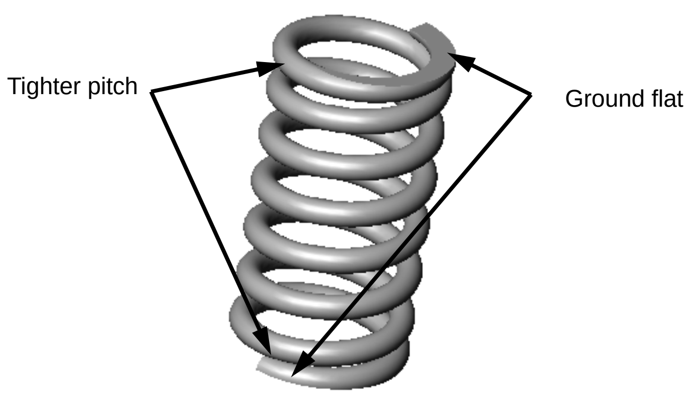
8 Set Options to **Follow Path**. This will keep the profile sketch normal to the path curve as it sweeps.



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.

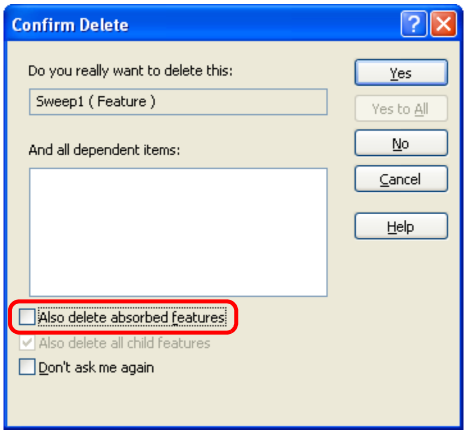


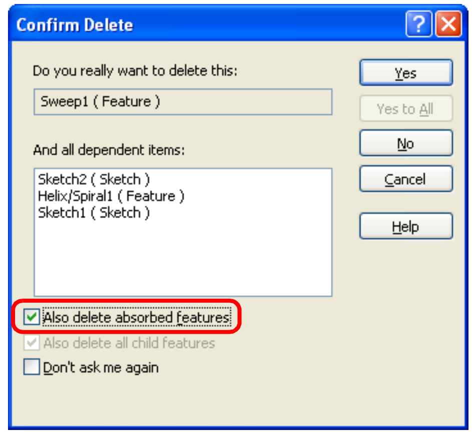


Task 3 — Create the Lower Helix

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).

   Click **Yes**.

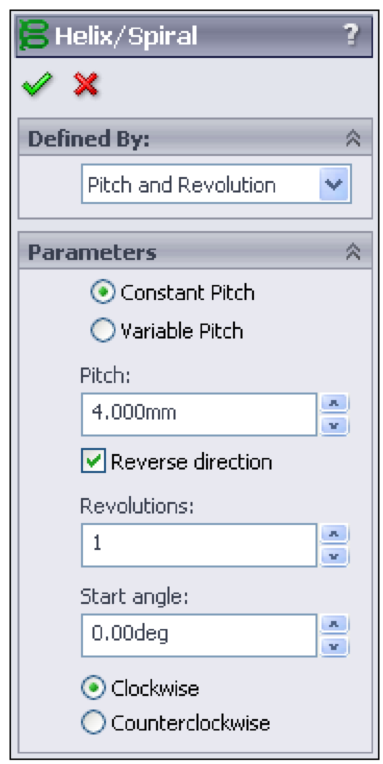


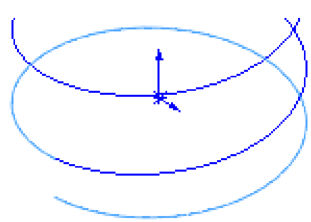


2. **Delete** the profile sketch (Sketch2).

   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.



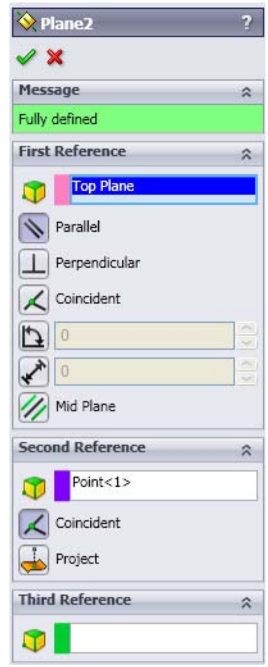


Task 4 — Create the Upper Helix

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 part icon. Select the Top plane in the FeatureManager design tree and the top end of the helix.



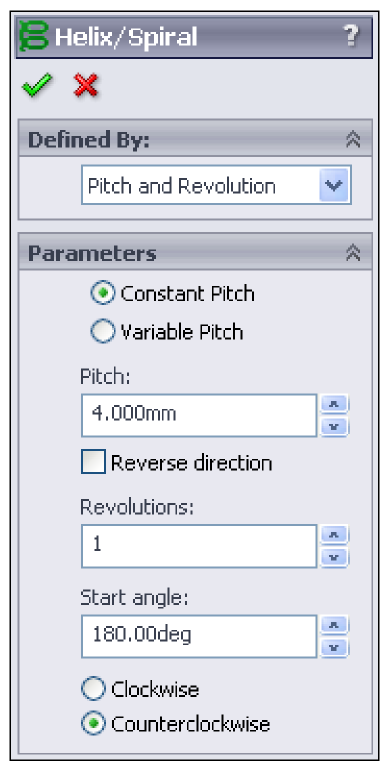
3 Because we selected a plane and a point, SolidWorks will default the type of plane to **Parallel**.

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.

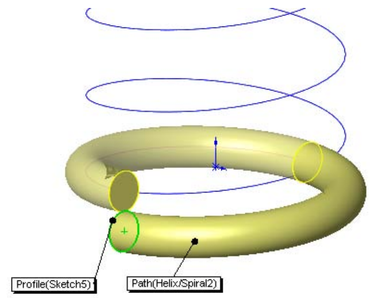
Notice that we have to start at 180° because the original helix was 5.5 turns.

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 .

15 **Delete** the sketch with the circle profile.

Task 5 — Create a Composite Curve

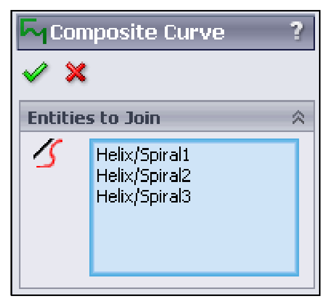
We could not use the three helixes for the sweep path because the path must be a single curve. To make a single curve from the three, we will create a combination called a Composite Curve.

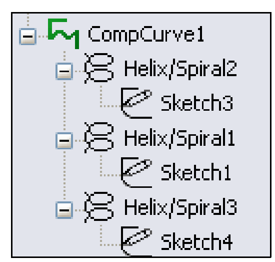
Composite Curve

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.





3 Click .

The three Helix/Spiral features and their sketches will now be absorbed by the `CompCurve1` feature. If you select `CompCurve1` 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.

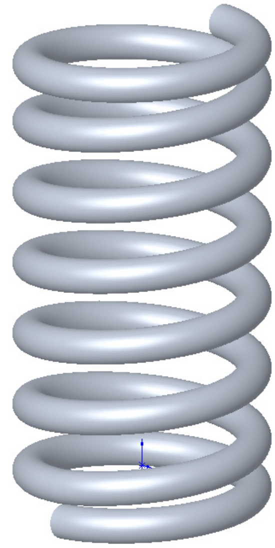
2 Sketch a circle, dimension it as **3.9mm** in diameter.

3 Add a **Pierce** relationship between the center of the circle and the composite curve.

4 **Exit** the sketch.

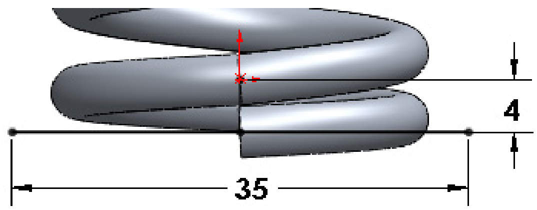
5 Create a **Sweep** using `CompCurve1` as the path.

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.

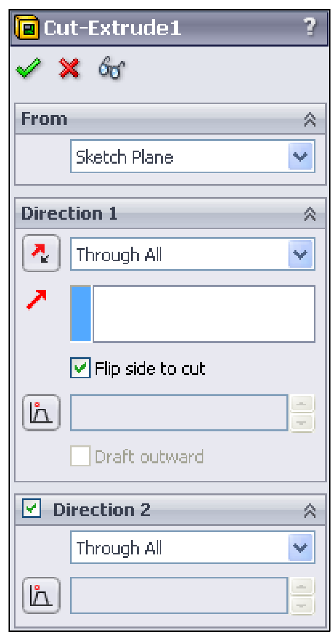
4 Draw a horizontal line.

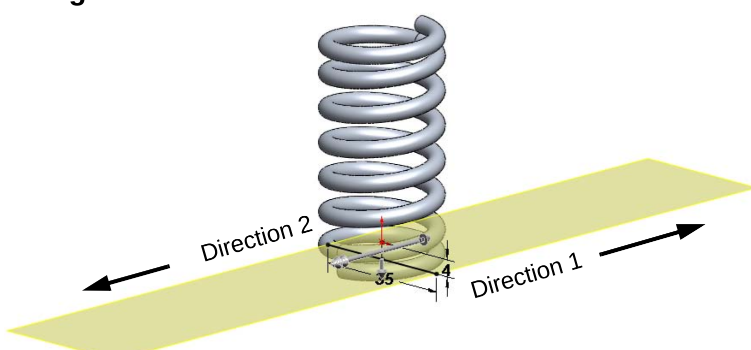
5 Add a **Midpoint** relationship between the end of the centerline and horizontal line.

6 Dimension the sketch as shown.

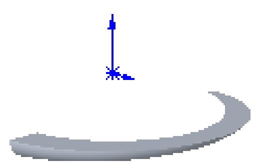
7 Click **Insert, Cut, Extrude** on the menu.

8 Because this is an open sketch, the default end condition will be **Through All**. Notice that **Direction 2** is also checked and **Through All** is the end condition.





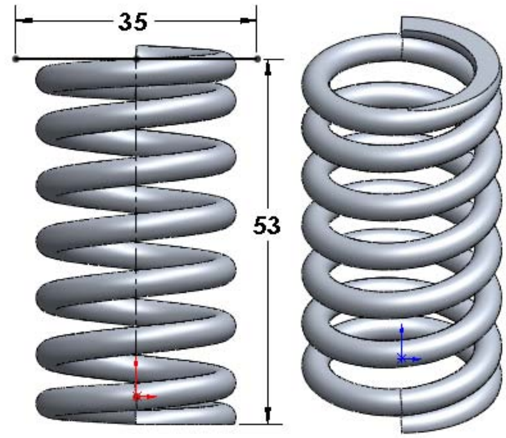
Note: If your cut results in just a little piece of the spring being left, edit the feature and select **Flip side to cut**.



9 Click .

10 Repeat this procedure to the other end of the spring. The total length of the spring should be **53mm**.

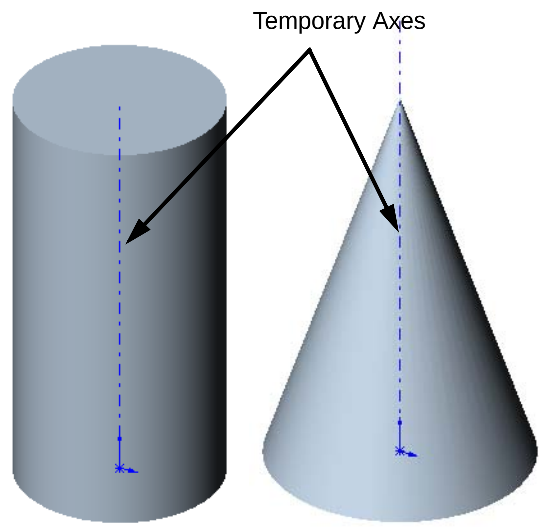
11 **Save** this part as Spring in the ...\Mountainboard\Spring Assembly folder.



Task 8 — Create Reference Geometry

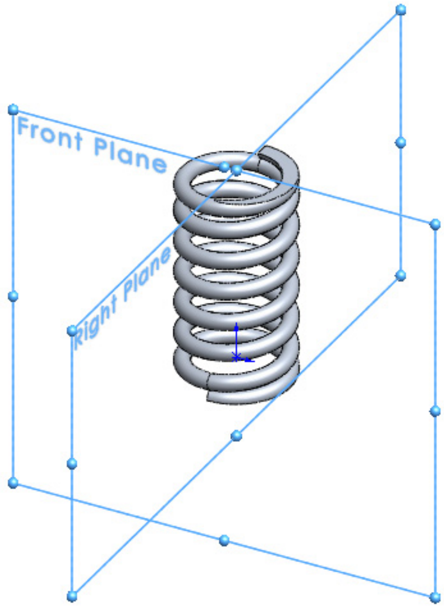
Thinking ahead to the task of creating an assembly using this spring, we will need to create some reference geometry to make it easier to mate the spring in the assembly.

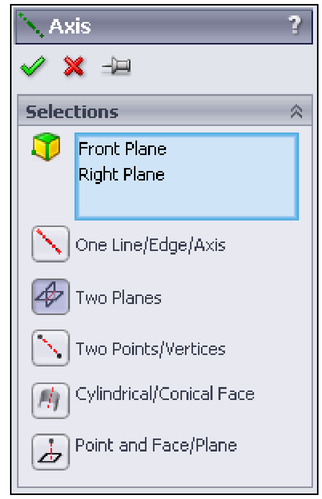
Cylinders and cones have Temporary Axes created by SolidWorks. The helix does not have a Temporary Axis so we must create an axis manually. The axis will make it easier to mate the spring concentric with dampener and retainers.



1 Click **Insert, Reference Geometry, Axis** from the menu

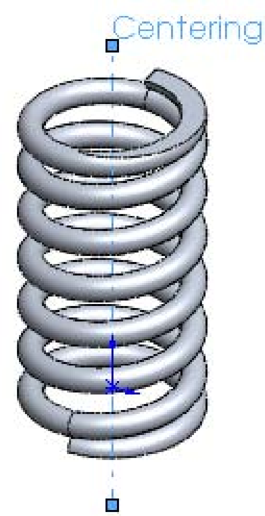
2 Select the Front and Right planes. By selecting two planes, the axis will be created at the intersection of the planes.





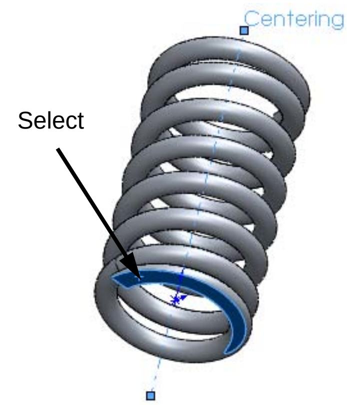
3 Click .

4 Rename the Axis **Centering**.



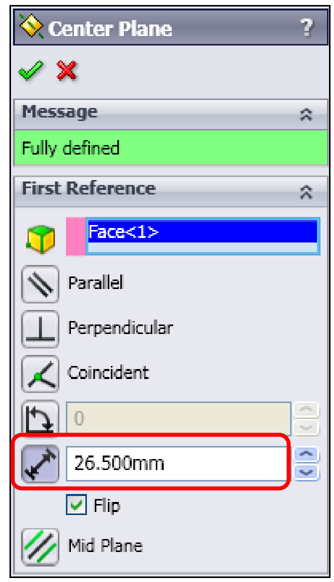
5 Click **Insert, Reference Geometry, Plane**.

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.

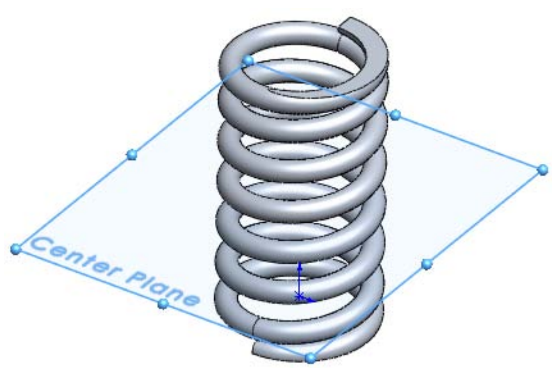


7 Check the preview to insure that the direction places the new plane in the middle of the spring. If not, select **Reverse direction**.

8 Click .

Rename the plane Center Plane.

9 **Save** the part.



Active Learning Experience, Part 2— Create An Assembly

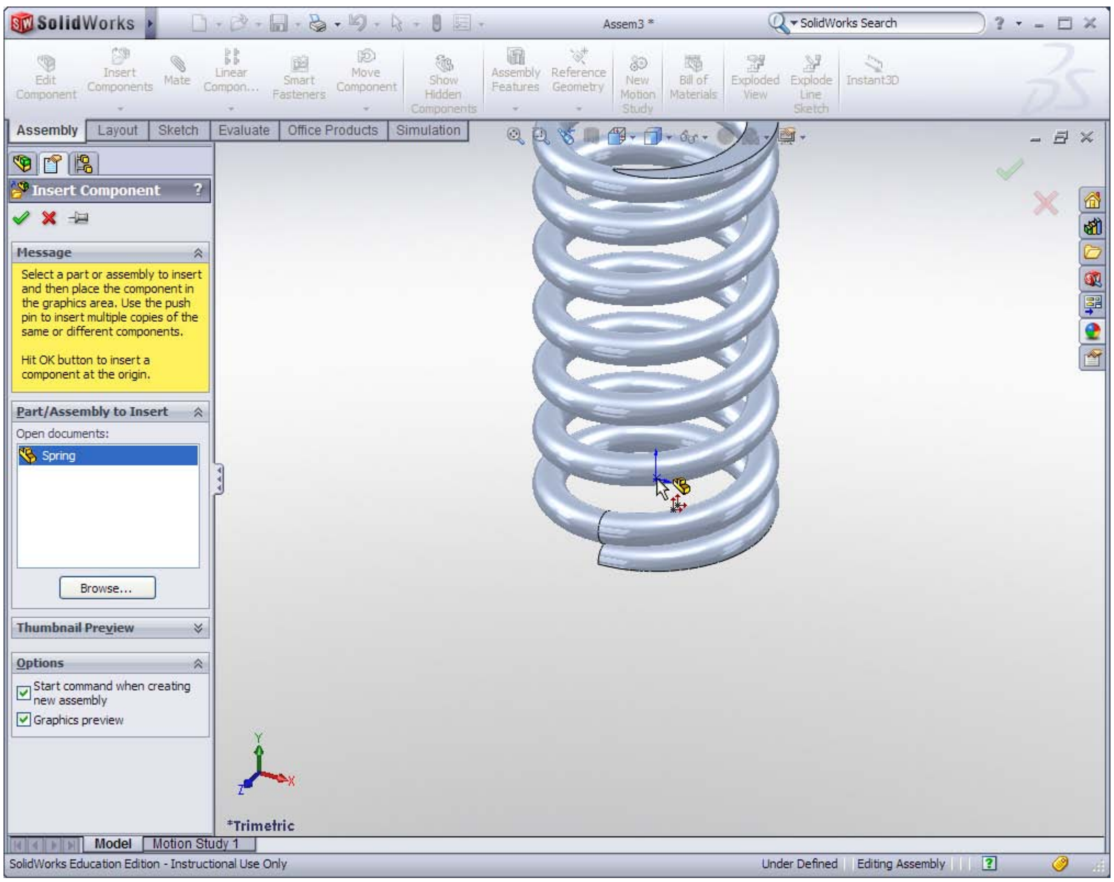
Creating the final assembly of the mountainboard will be easier if we create the sub-assemblies as we go along.

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.

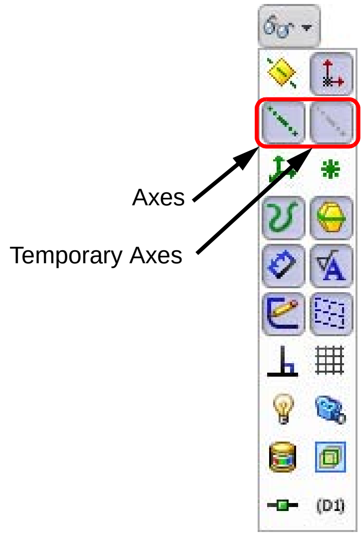
This will fix the Origin of the spring to the Origin of the assembly with corresponding planes coincident (Front to Front, Top to Top, etc.)



3 Add the Spring Dampener. Click **Insert Components**  on 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 ...\Lesson04\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, Axes** and **View, Temporary Axes** from the menu or selecting them in the **Heads-up** toolbar.

5 Click **Mate**  on the Assembly toolbar.

6 Select the axis of the Spring and the temporary axis of the Spring Dampener. The default mate will be **Coincident**.

Click .

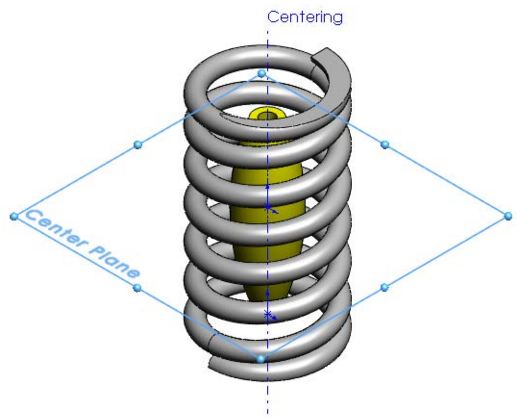
This positions the Spring Dampener inside the Spring.

7 Click **Mate**  on the Assembly toolbar.

8 Select the Centering Plane in the Spring 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.

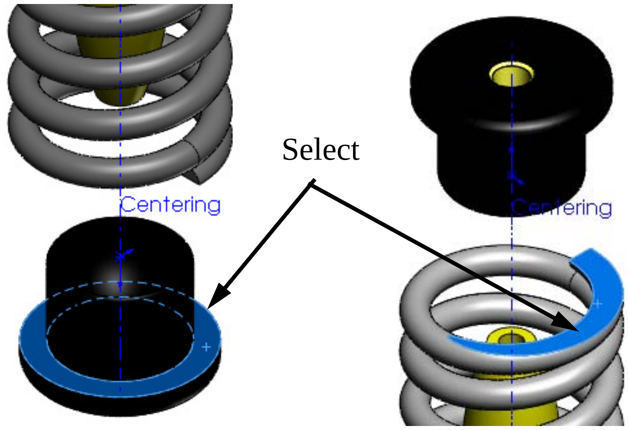
11 Mate the Temporary Axis of the Spring Retainer to the Centering axis of the Spring.

12 Select the two faces shown, then click **Mate** . Add a **Coincident** mate.

13 Hide the Centering Plane.

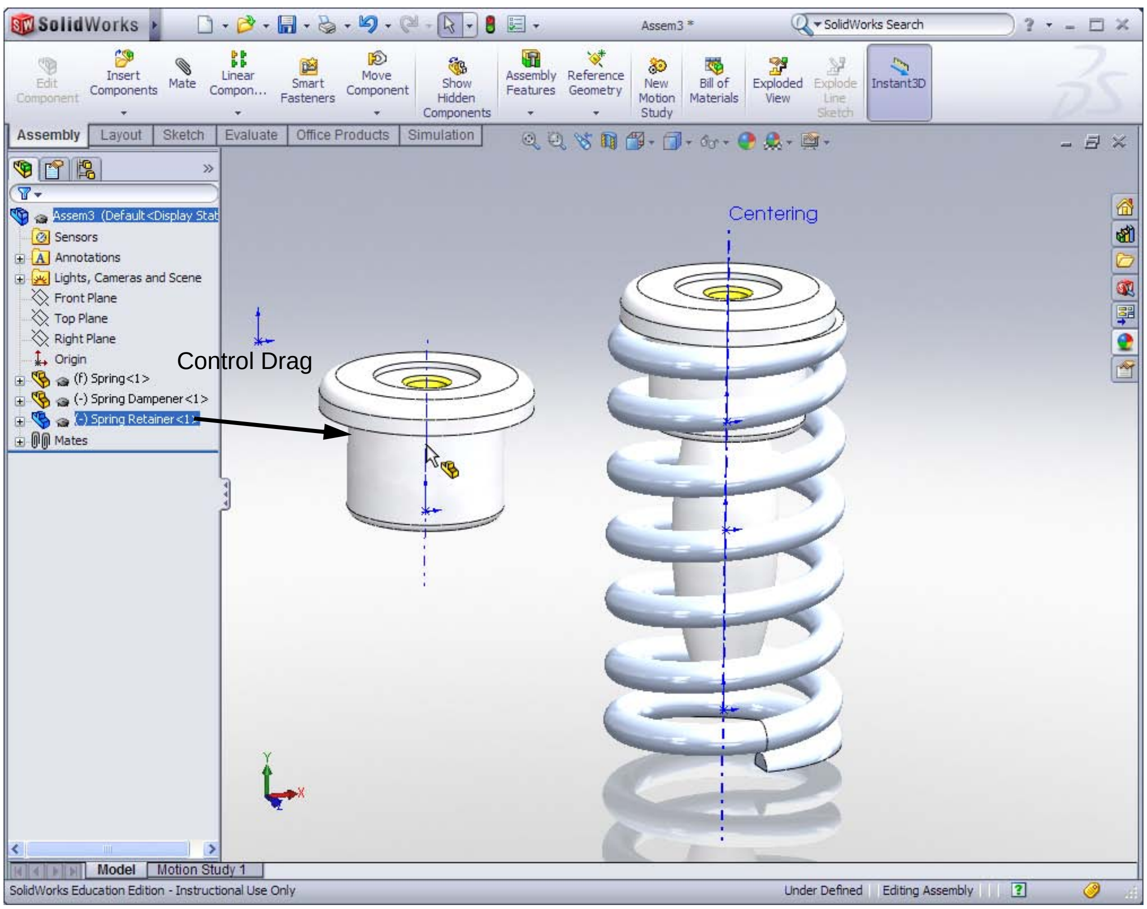
In the FeatureManager design tree, expand the Spring features by clicking the plus sign next to the Spring part.

14 Right-click the Center Plane and select **Hide** from the menu.

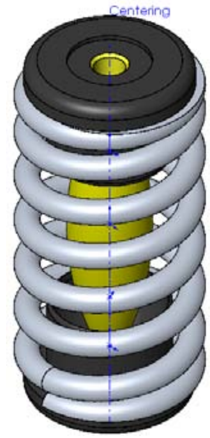


15 Add another Spring Retainer to the assembly. Hold the **Control** key and drag the Spring Retainer from the FeatureManager design tree into the graphics area

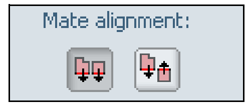
.



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.

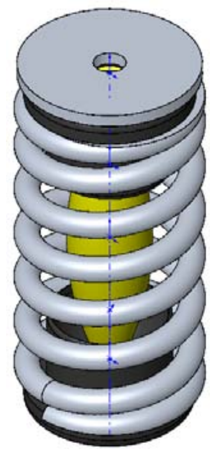


Note: .To reverse the direction of the Spring Retainer you have to change the alignment. Click either **Align** or **Anti-Align** to reverse the direction



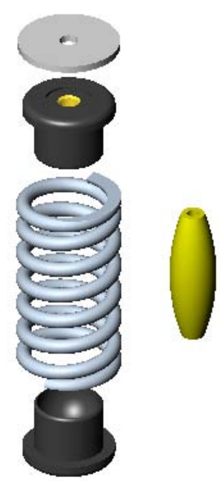
17 Add a Fender Washer to the assembly and mate it to the top Spring Retainer.

18 **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.



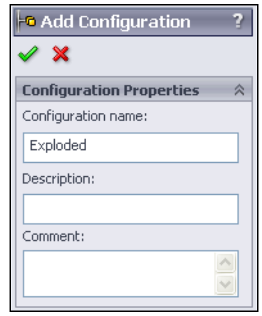
Exploded Assemblies - Review

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.

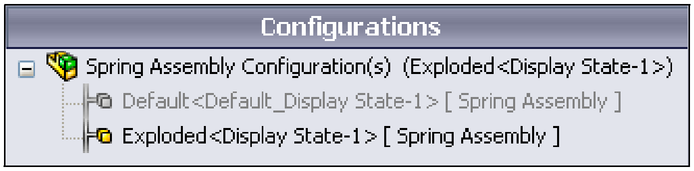
You can only create one exploded view per configuration of the assembly.

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.

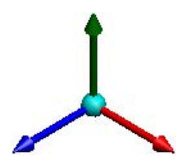
Assembly Exploder - Review

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.

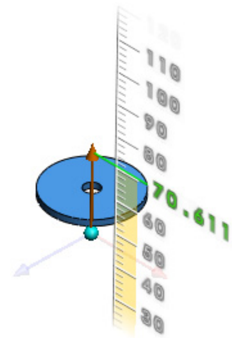
1. Orient the model to the `Isometric` view, then zoom out. We need to zoom out because the assembly will take up more room on the screen as we move the components away from their mated position.
2. Click **Insert, Exploded View**.
3. Select the `Fender Washer`. A **Triad** will appear on the part. The **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.

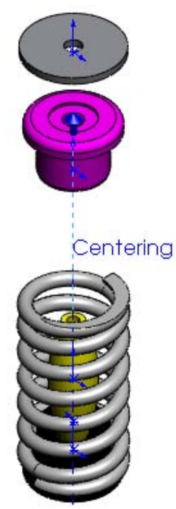
8 Click on the lower Fender Washer. Drag it by the green arrow until it is below the Spring

9 Click on the lower Spring Retainer. Drag it by the green arrow until it is below the Spring.

10 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.

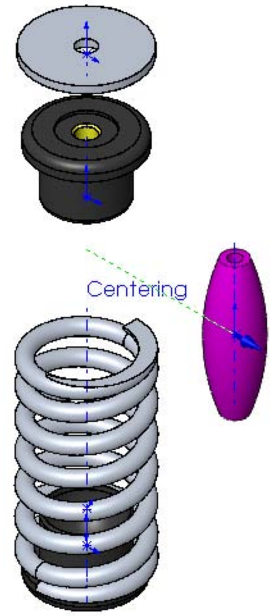
11 Click on the Spring Dampener, then drag it by the green arrow of the **Triad** until it is above the Spring.

12 Click in an empty area of the graphics window to finish the step.

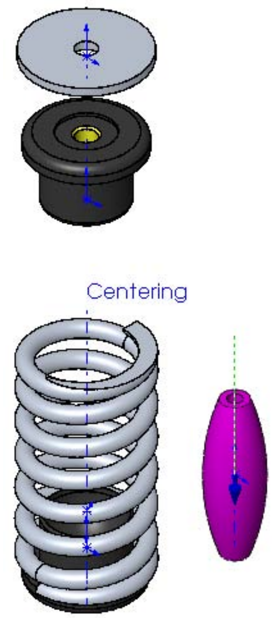


13 Click on the Spring Dampener, then drag it by the red arrow of the **Triad** until it is outside of the Spring.

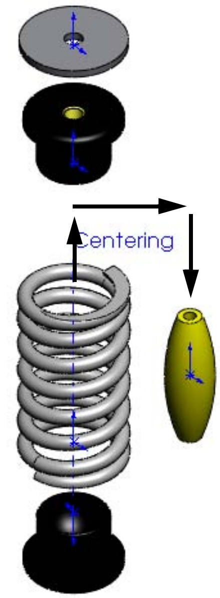
14 Click in an empty area of the graphics window to finish the step.



15 Click on the Spring Dampener, then drag it by the green arrow of the **Triad** until it is alongside the Spring.



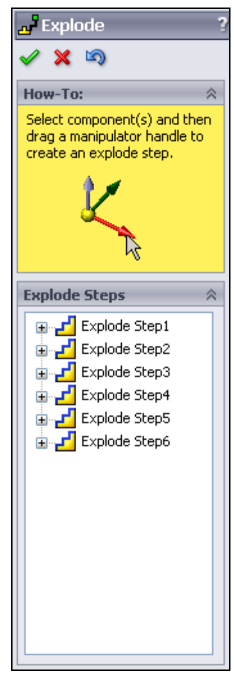
When complete the exploded assembly should look like this.



16 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**.

17 Select `Explode Step1`. Drag the `Fender Washer` further from the `Spring`.

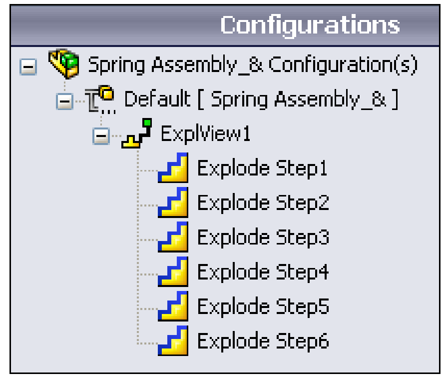
18 Select `Explode Step2`. Drag the upper `Spring Retainer` further from the `Spring`.

19 When all the components are where you want them, click .

20 Turn off the `Origin`, `Axes` and `Temporary Axes` by clearing them on the **View** menu.

21 **Hide** any planes that are showing.

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 ExplodeView and select either **Animate explode** or **Animate collapse**.

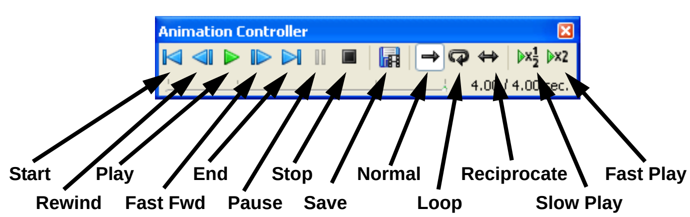
Task 10 — Explode and Collapse the assembly

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.

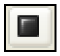
Animation Controller

The **Animation Controller** proves simple controls, similar to a CD or DVD player, to control the playback of animations.



If PhotoWorks is installed, the animations can be recorded with photorealistic rendering.

1 Click **Reciprocate** ↔, then **Play** ▷. The assembly will continue to explode and collapse.

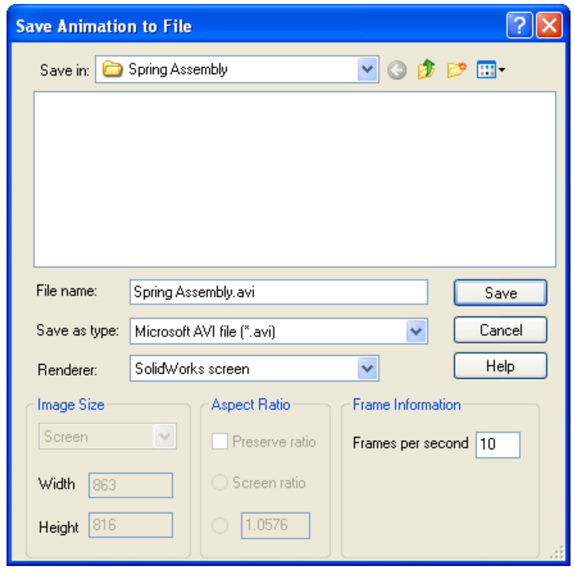
2 Click **Stop** .

3 Click **Normal**  to reset the **Animation Controller**.

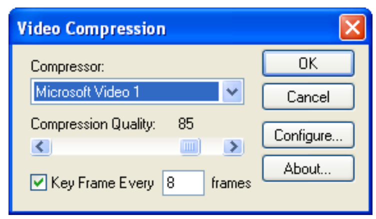
4 Click **Save Animation**  in the **Animation Controller**.

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 Render to **SolidWorks screen**.

Type **10** for **Frames per second**.



6 Click **Save**. Select **Microsoft Video 1** as the Compressor.



7 Click **OK**.

8 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.

9 Use Windows Explorer to locate the AVI file.

10 Double-click the file. The default media player should start and play the animation.

11 **Close** the media player.

12 **Save** and **Close** the assembly.

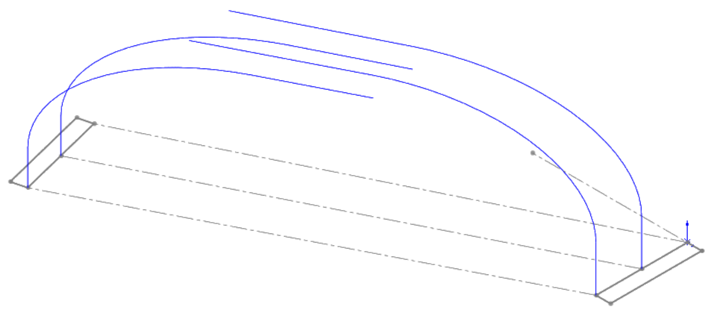
Active Learning Experience, Part 3 — Binding Straps

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.

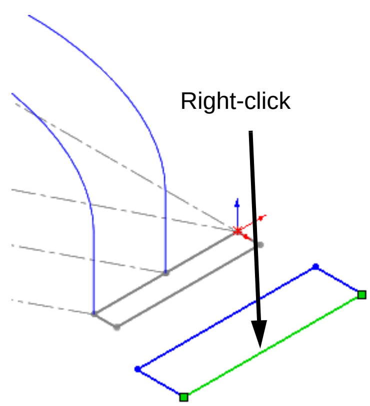
Task 11 — Create the Binding Straps

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.

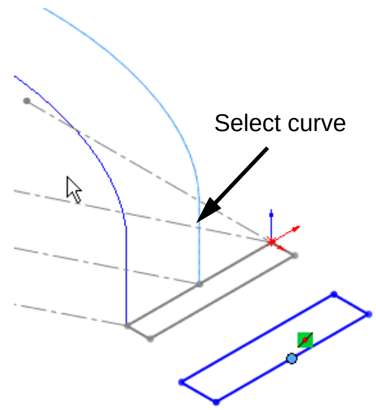
   The rectangles and centerlines represent the positions of the posts of the Binding Base Plate. The straps will later attach to these posts.

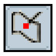
   The four blue curves will become the paths and guide curves when we sweep the straps.
2. Create a sketch on the Top plane.
3. Sketch a rectangle as shown. The size and position are not critical.
4. We now have to position the sketch using relationships to the path and guide curve.
5. Right-click on the line shown and choose **Select Midpoint** from the menu.

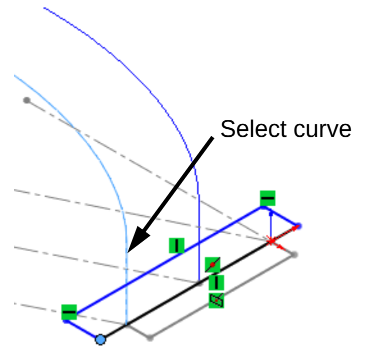
6. Click **Add Relationship** .

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.

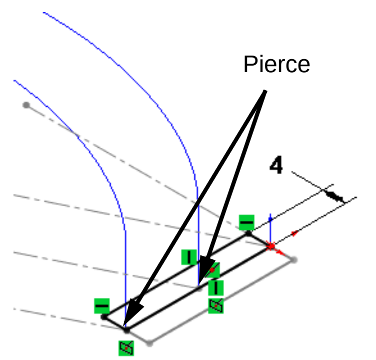


10 Dimension the width of the rectangle to **4mm**.

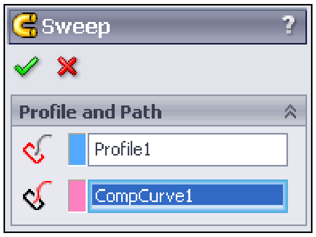
11 **Exit** the sketch.

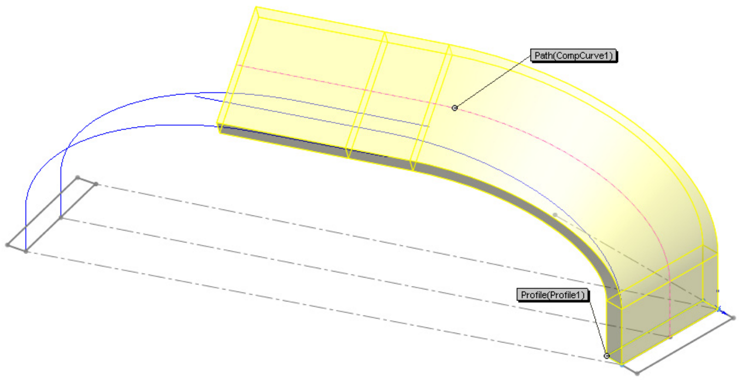
12 Rename the sketch to `Profile1`.

13 Create a sweep. Click **Insert, Base/Boss, Sweep**.



14 Select `Profile1` as the profile sketch and `CompCurve1` as the path.





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.

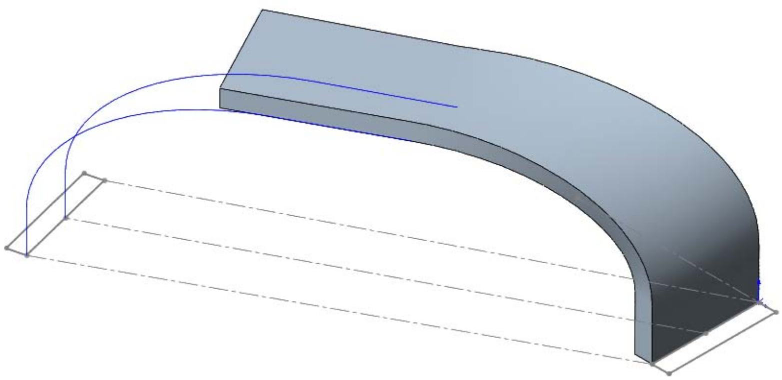
17 Right-click the sweep and selecting **Edit Feature** from the menu.

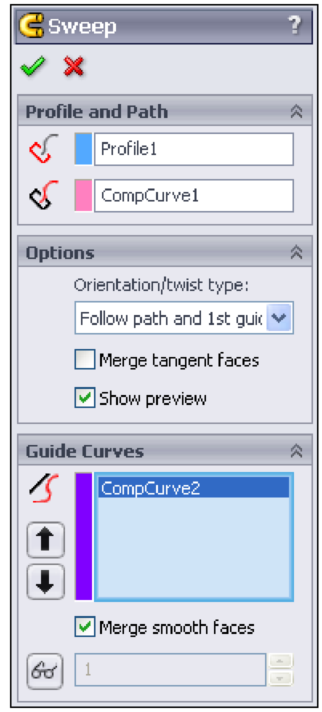
18 Expand the sections **Options** and **Guide Curves** by clicking the arrow .

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

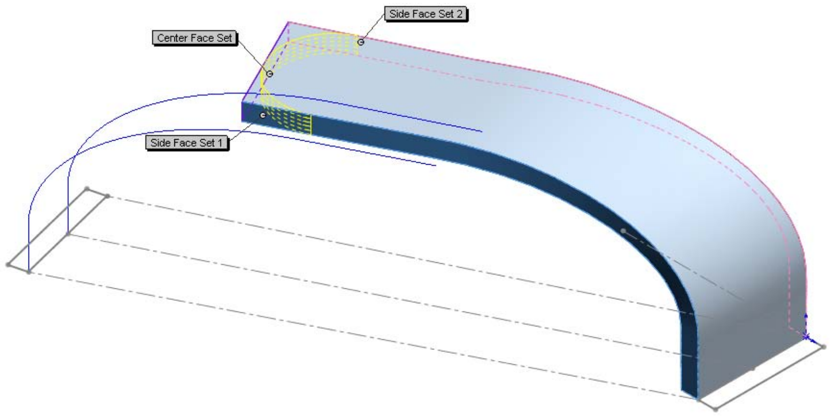
The **Full Round Fillet** option creates a fillet that is tangent to three adjacent sets of faces. Each face set can contain more than one face. However, within each face set, the faces must be tangent continuously.

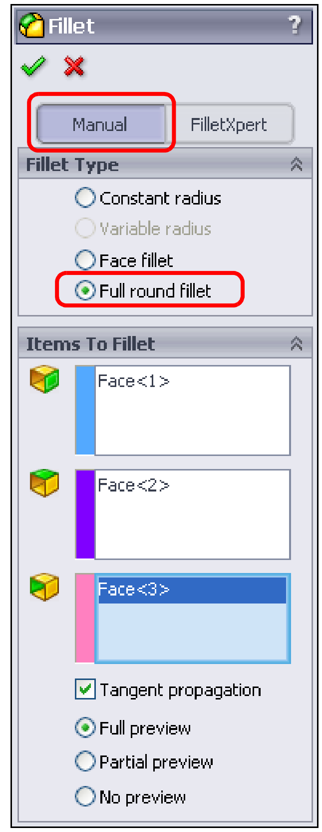
A **Full round Fillet** does not need a radius value. The radius is determined by the shape of the faces you select.

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.

24 Select the three faces shown.





Note: After selecting the first face, you must make the next box active in the PropertyManager before selecting the second face. The same will be required before selecting the third face.

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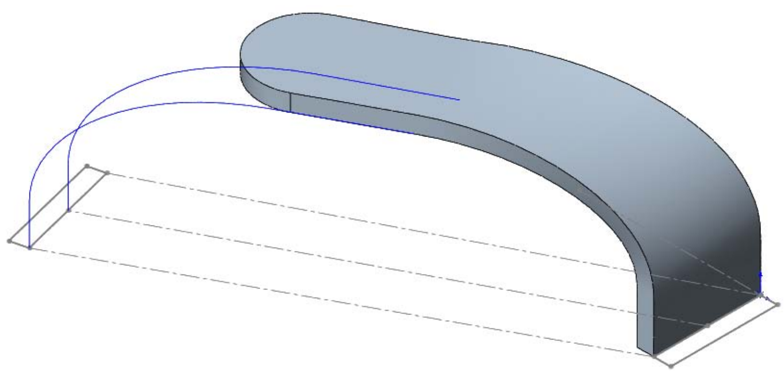
25 Click ✔. The strap now has a full round on the end.



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.

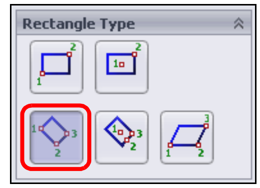
Parallelogram - Review

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.

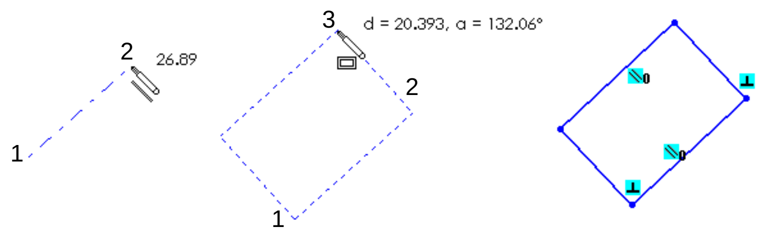
2 Use the **Parallelogram** tool to create a rectangle. Click the Rectangle tool and then select the **3 Point Center Rectangle** in the PropertyManager.



Note: To sketch a parallelogram:

Drag from point 1 to point 2 and then release the mouse button.

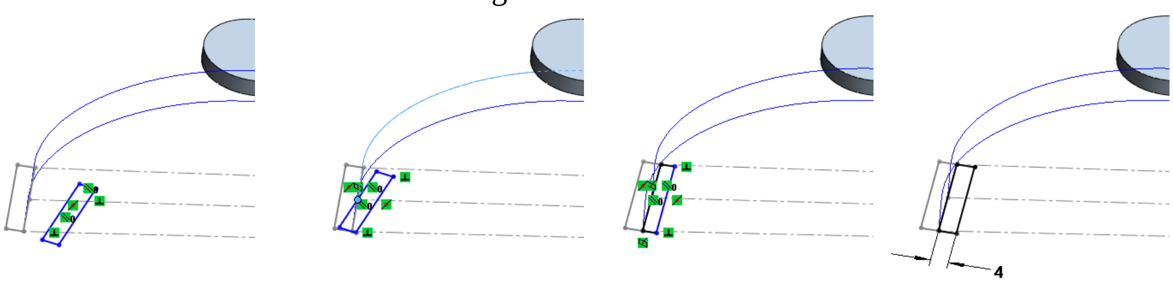
Place the cursor at point 2 and drag to point 3 and release the mouse button.



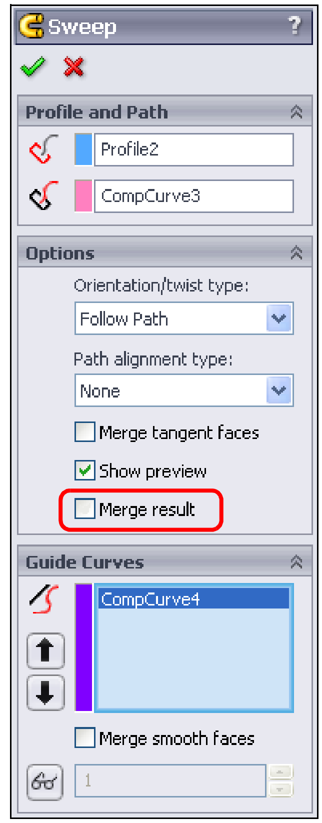
Step 1 Step 2 Result

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**.



Sketch Pierce Pierce Dimension

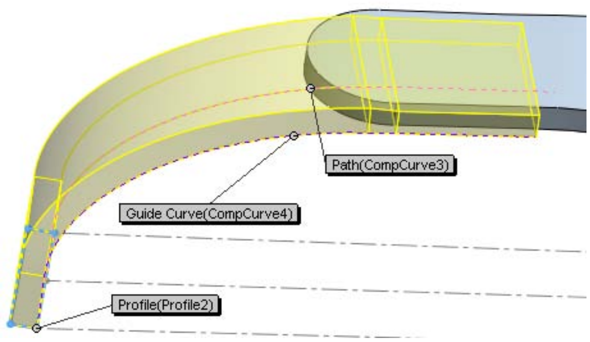


5 **Exit** the sketch.

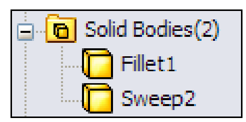
6 Rename the sketch Profile2.

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.

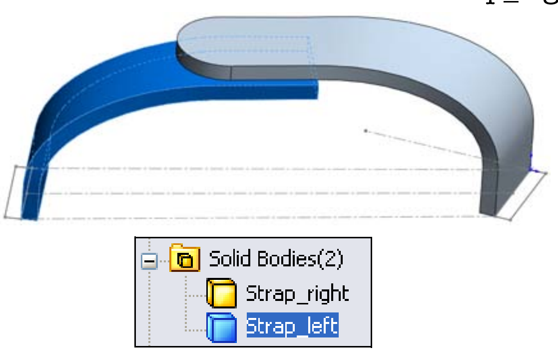


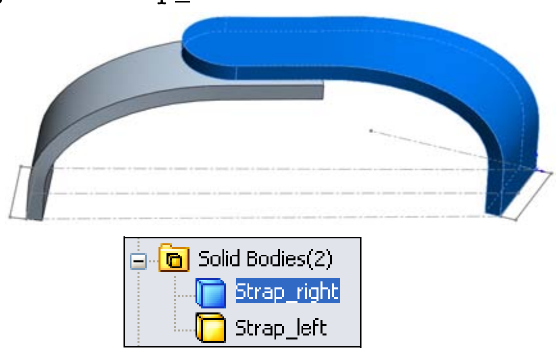
9 Click .



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.



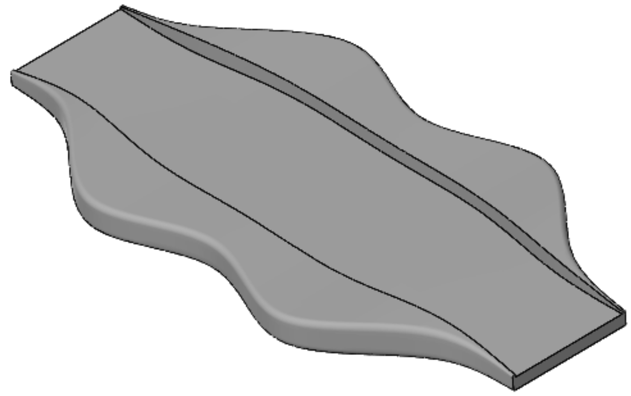


12 **Save** the part.

Task 13 — Create the Foam Pad

The binding needs a pad to spread the load out on the top of the rider's foot.

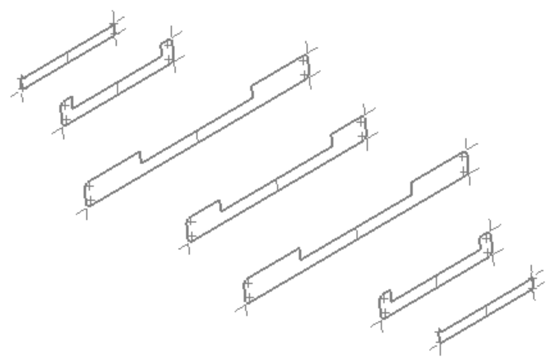
This part will be molded in the flat state shown at right. When installed in the binding, it will take a shape defined by the straps and the rider's foot. We will create this pad two different ways. First we will create it in the flat state as this is what will be manufactured. We will also create the part in its bent state in order to show the pad in our product literature.



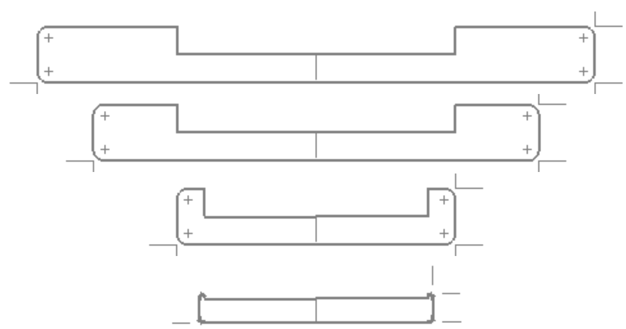
Lofting

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.

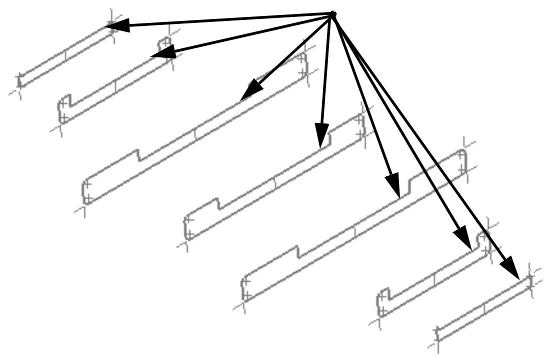


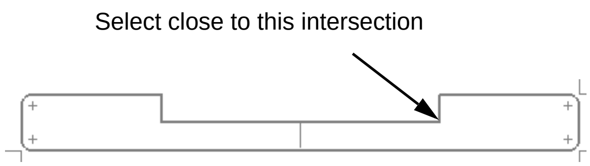
Each profile is a different size but is similar in shape. When lofting, the individual sections can be different shapes, this is what makes lofting so powerful. The one important consideration is that each section have the same number of sketch segments.



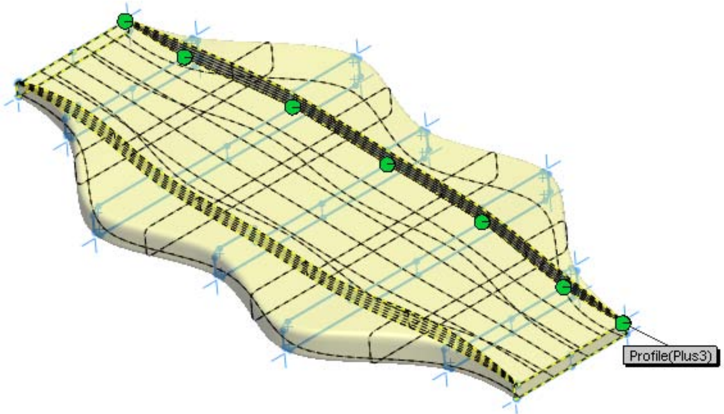
2 Create a loft. Click **Insert, Boss/ Base, Loft**.

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.





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?

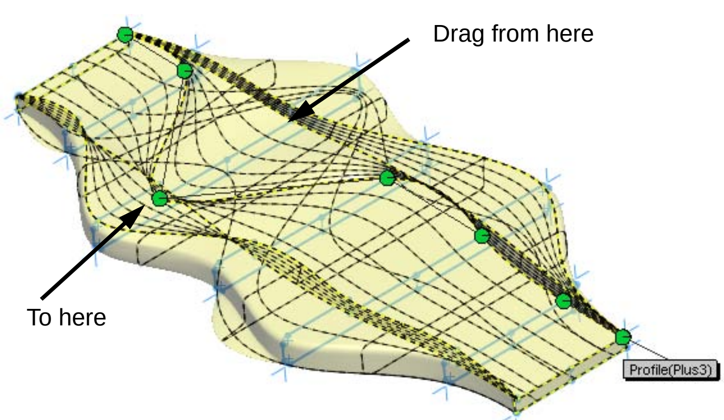
Answer

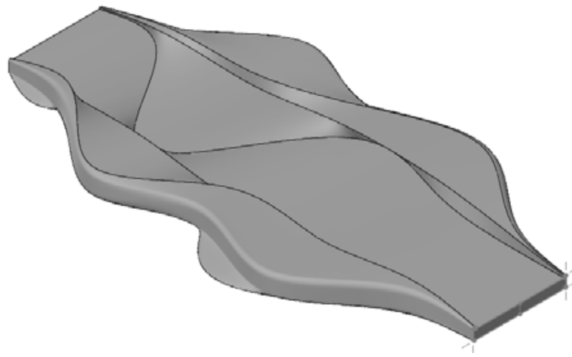
The preview shows the selected points with the cyan control handles. These can be dragged to different positions.

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.





9 Click . The loft 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. Click **Edit Material** on the Standard toolbar.

12 Expand the material category **Other Non-metals** and select **Rubber**.

13 Click .

14 **Save** and **Close** the part.

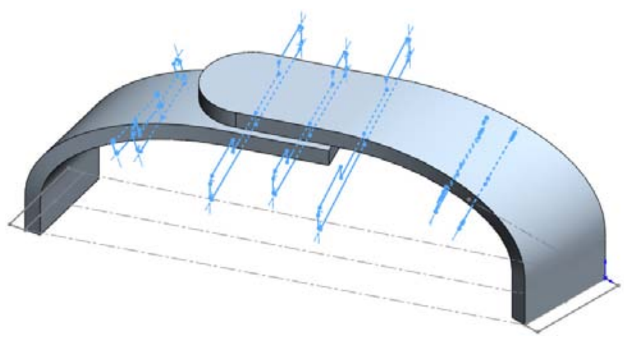
Active Learning Experience, Part 4 — Multibody parts

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.

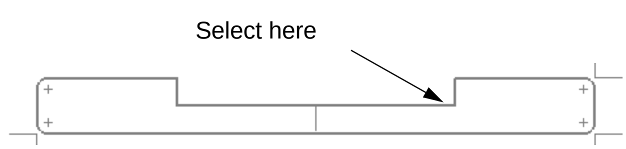


2 Select the sketches `Loft Section 1` through `Loft Section 7`. Right-click any of the selected sketches and select **Show**.

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.



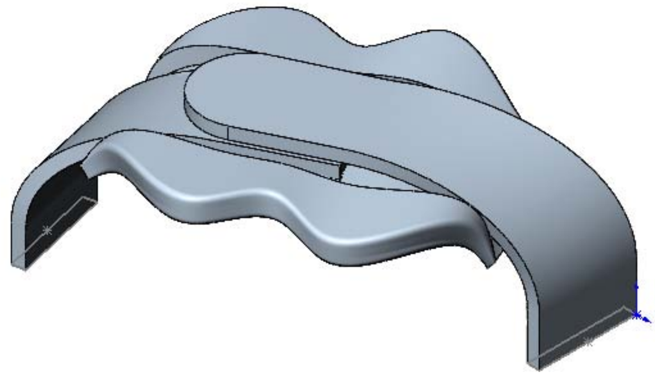
6 Clear **Merge results**.

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`.

10 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.

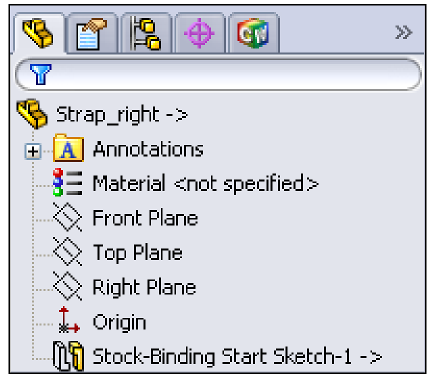
Note: We could also set the default assembly and drawing templates through this same procedure.

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.



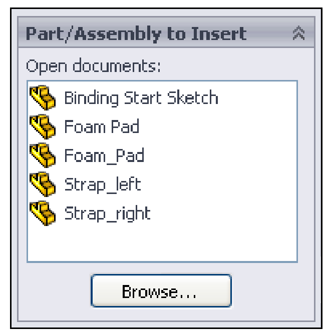
8 Press **Control-Tab** to shift back to Binding Start Sketch.

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.

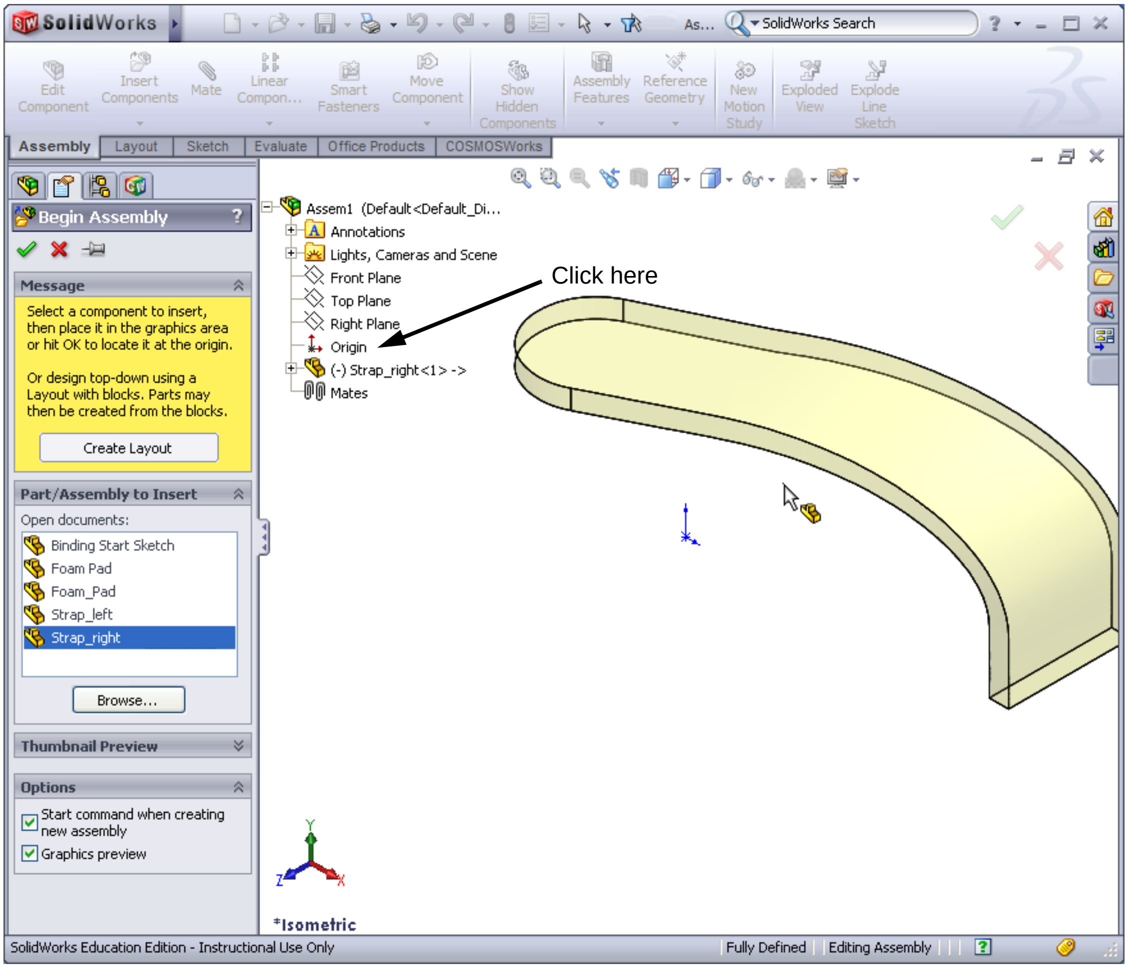
2 All the open parts will be listed in the Open documents section of the PropertyManager.



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.

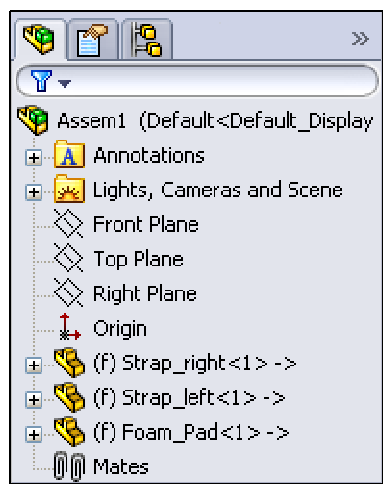
5 Click on the Origin. This will insert the part Strap_right on the assembly Origin with its three planes aligned to the corresponding planes in the assembly.



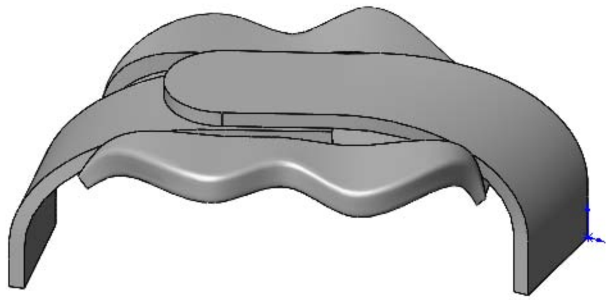
6 Click **Insert, Component, Existing Part/Assembly** from the menu and insert the Strap_left and Foam_Pad_curved part using the same method.

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.

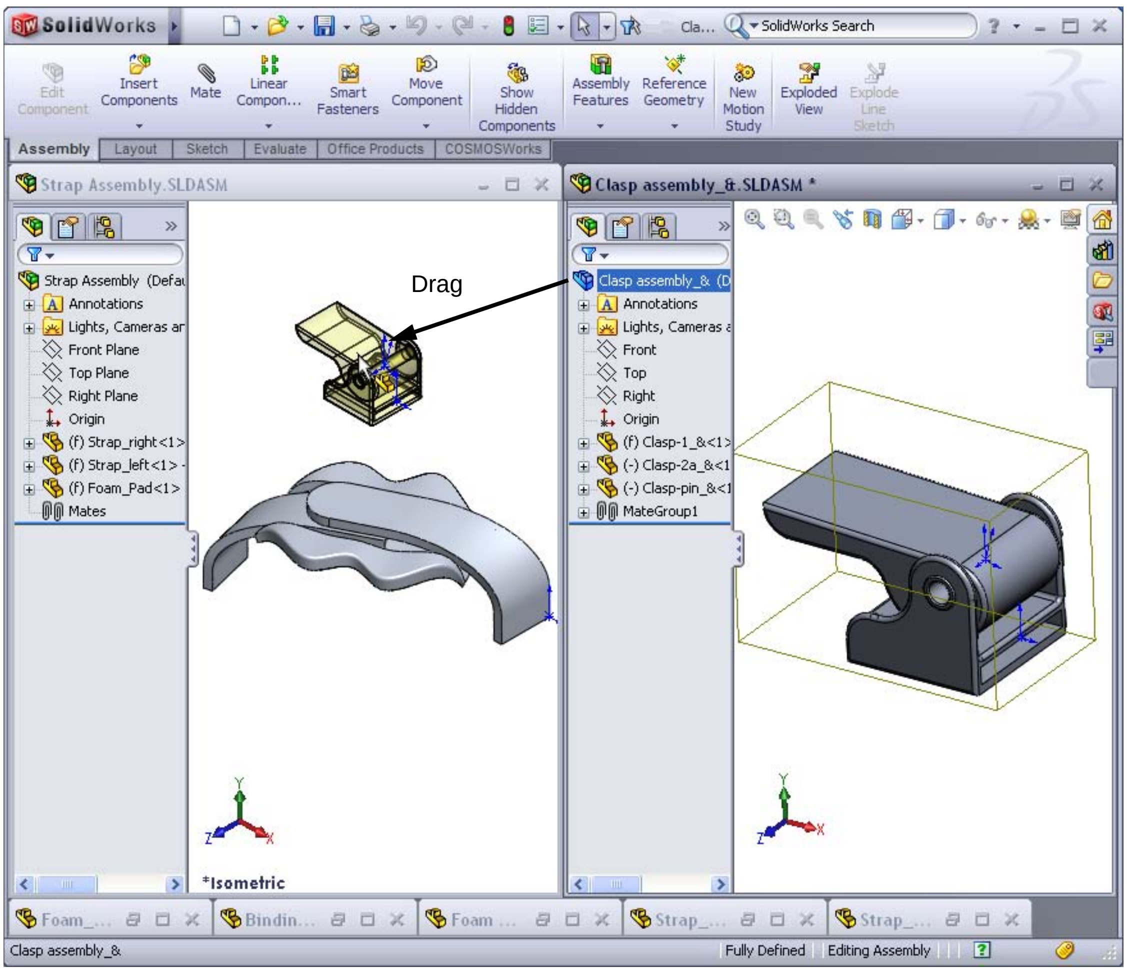


8 **Save** the assembly as Strap Assembly.



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**  on the Assembly toolbar. This acts as a toggle. If the 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**  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**.

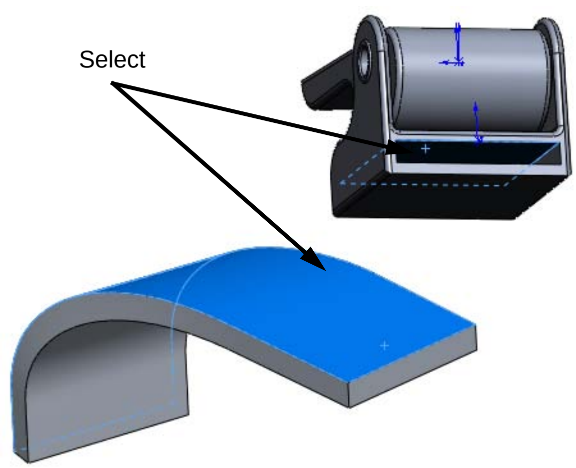
4 Hide the parts Strap_Right and Foam_Pad-curved. Select the two parts, either in the graphics area or the FeatureManager design tree, then click  on the Assembly toolbar.

Mate Considerations

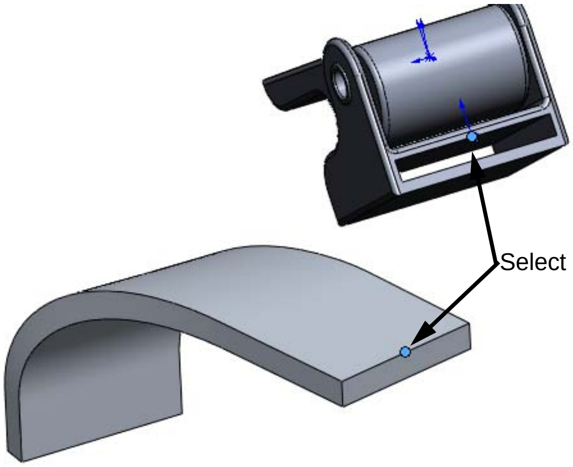
The sweep provides some additional challenges for mating as there are few planar surfaces or linear edges. To mate the Clasp Assembly we will have to mate some edges and points.

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.

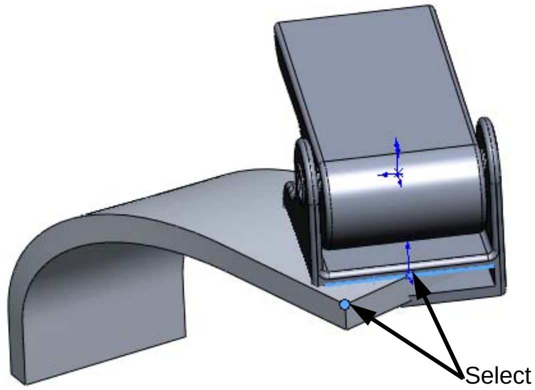


6 Add a **Coincident** mate between the midpoint of the strap and the midpoint of the edge of the clasp.

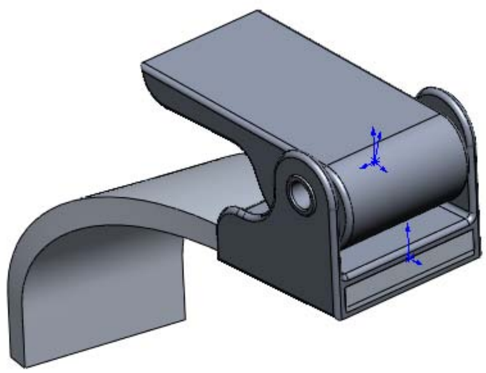


7 While it may look like we have enough mates, if you try to drag the clasp assembly, it can still rotate.

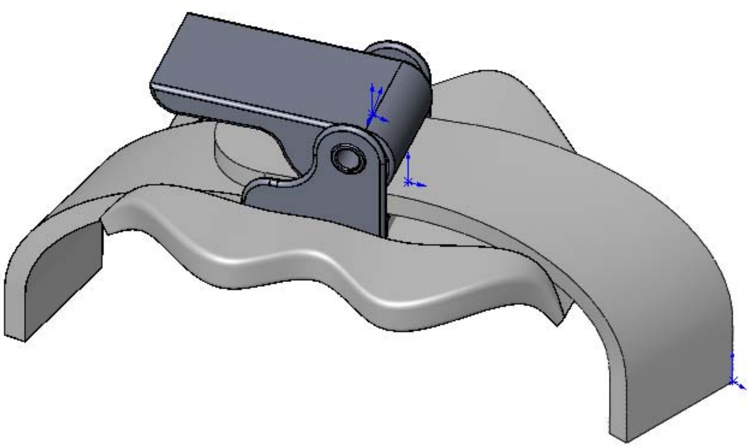
8 Add a **Coincident** mate between the vertex of the strap and the edge of the clasp.



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** .

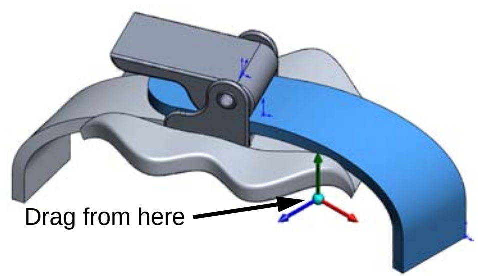


Task 5 — Add an exploded view

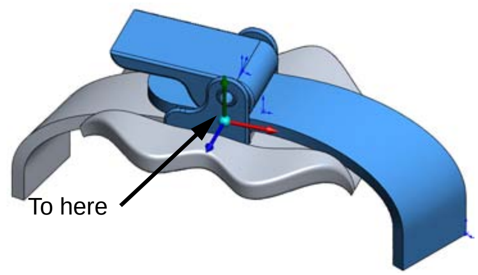
Exploding the straps will be a little different than with previous assemblies because we want the straps to move as if they are being retracted from the clasp assembly. These directions are not along the assembly X, Y, or Z axes.

1 Click **Insert, Exploded View** from the menu.

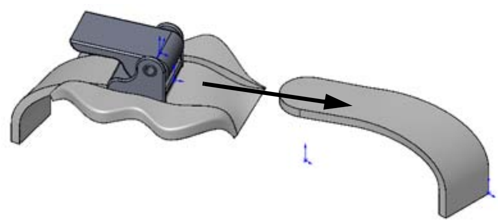
2 Select the component `Strap_right`, the **Triad** will appear and be aligned with the assembly axes.



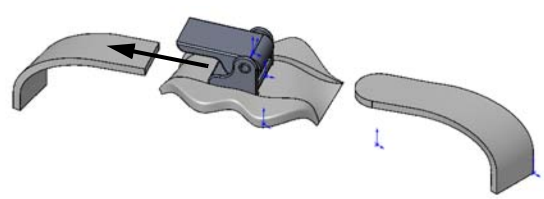
3 Right-click the center of the **Triad** and select **Align to**. Select the face shown. The Triad will align itself to this face.



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.

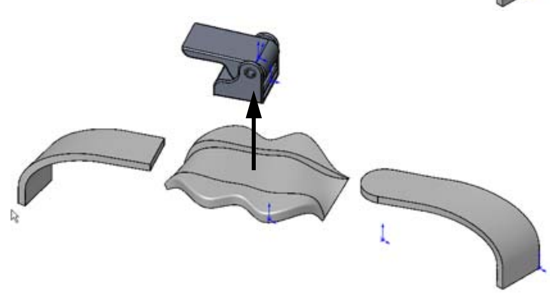


5 Repeat this procedure to move the `Strap_left` to the left and clear of the other components.



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.

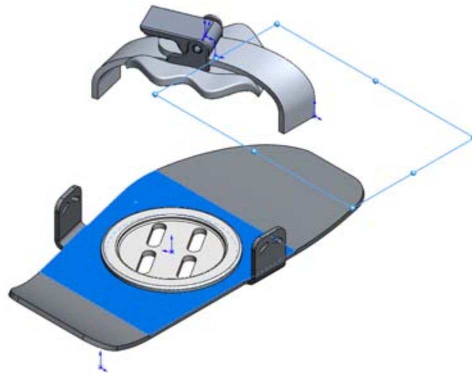
1 **Open** the `Binding` assembly.

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** on the Window Title Bar.

5 Click to add a **Mate**.



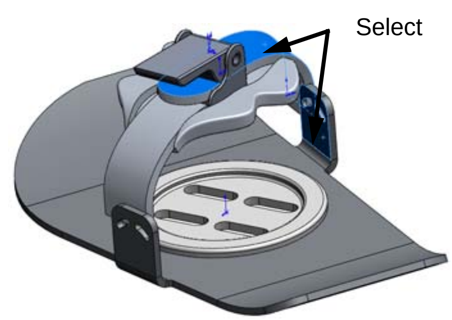
6 Expand the Strap Assembly in the FeatureManager design tree and select the Top plane of the Strap Assembly.

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.

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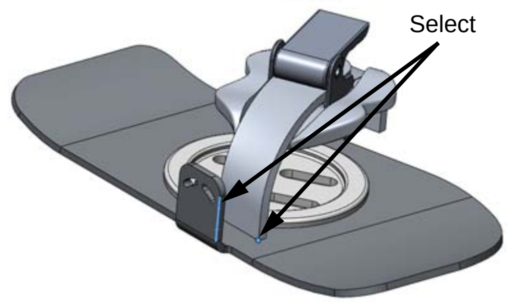
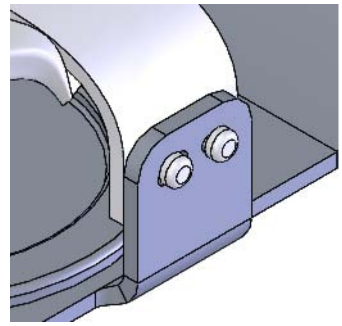
10 Add a **Coincident** mate between the vertex and face shown. The Strap Assembly should now be fully mated into the Binding assembly.



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.

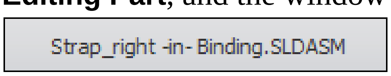
To edit a part in an assembly, select the part you wish to edit, then either:

- 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** tool.

TIP: The **Edit Component** 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 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:



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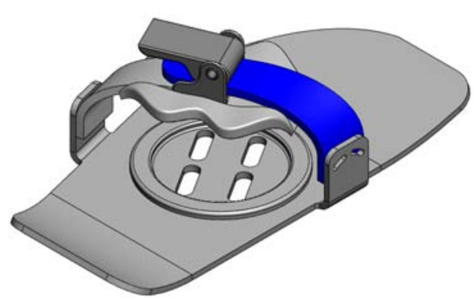
1 Select the part Strap_right, then click to the **Edit Part** tool.

The strap we are editing turns pink 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).

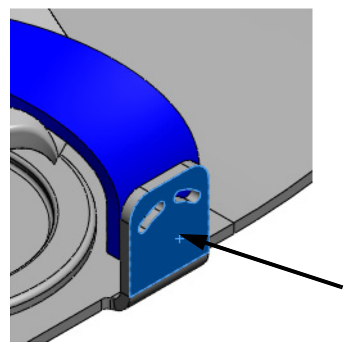
Select **Use specified colors when editing parts in assemblies**.

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.

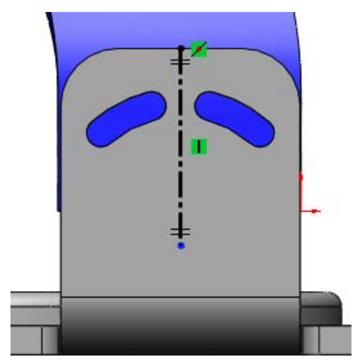


3 Click **Normal To** on the Standard Views toolbar. This will change our viewpoint so we are looking directly at the selected face.

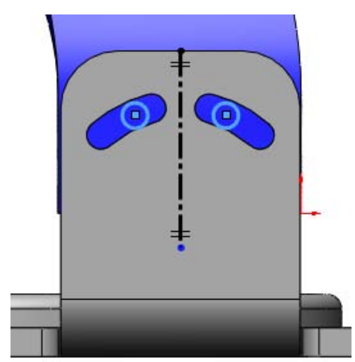
4 Click **Zoom To Selection** on the View toolbar. This will make the selected face fill the screen.

5 Sketch a vertical centerline from the midpoint of tab.

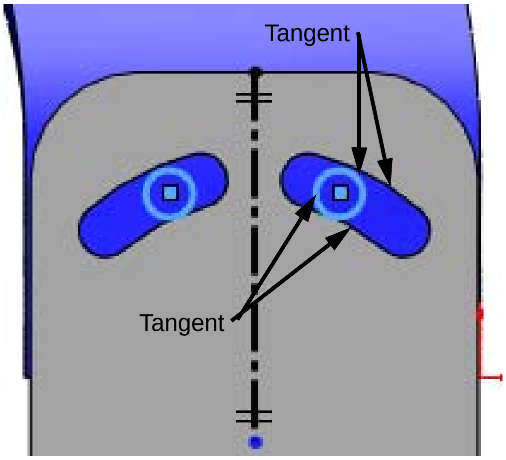
6 Turn on **Dynamic Mirror Entities** by clicking on the Sketch toolbar.



7 Sketch a circle in one of the slots. The second circle will be drawn by the mirror command.



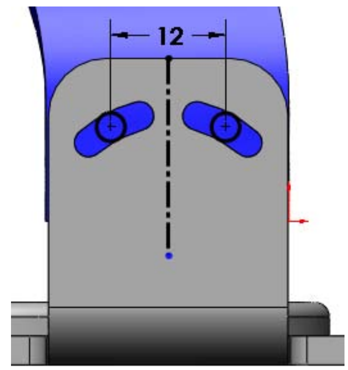
8 Add two Tangent relationships between one of the circles and the sides of the slot.



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.

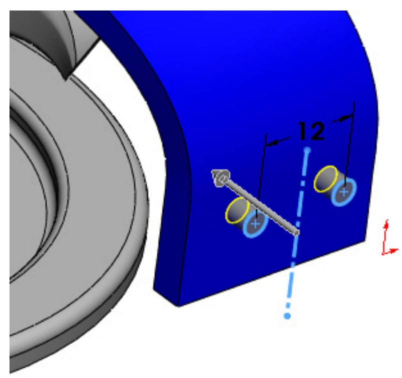


12 Click **Extrude**.

13 Click  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.)

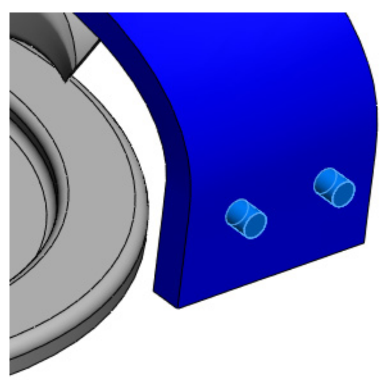


15 Click .

16 Right-click the Strap_right part and select **Open Part** from the menu.

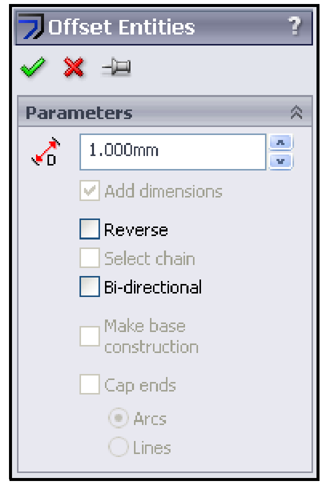
We do not need the surrounding geometry to create the tops on the two pins, so it is easier to work in the part instead of the assembly.

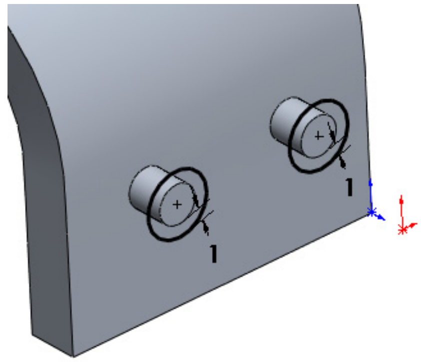
17 Select the top face of one of the pins and open a sketch.



18 Click the **Offset Entities** tool. Type **1mm** for the distance. Click **OK**.

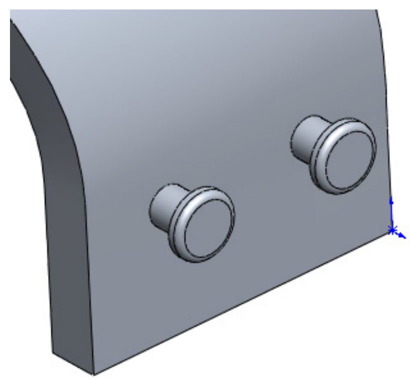
Select the top of the other pin, click **Offset Entities**, then click .





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.



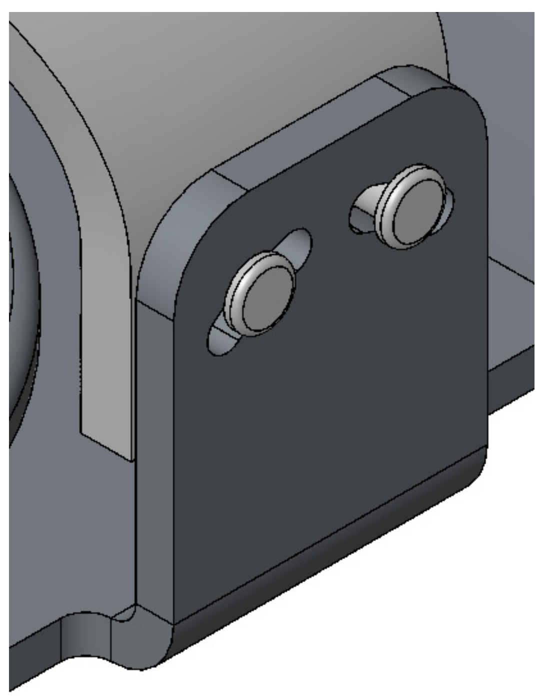
20 Return to the Binding assembly by clicking **Window** from the menu and selecting Strap_right -in- Binding.SLDASM from the list.

21 We are still in the **Edit Part** mode. Click **Edit Component** to return to the **Edit Assembly** mode.

22 In the FeatureManager design tree, select the Binding Base_Plate and click **Hide/Show Components** 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.

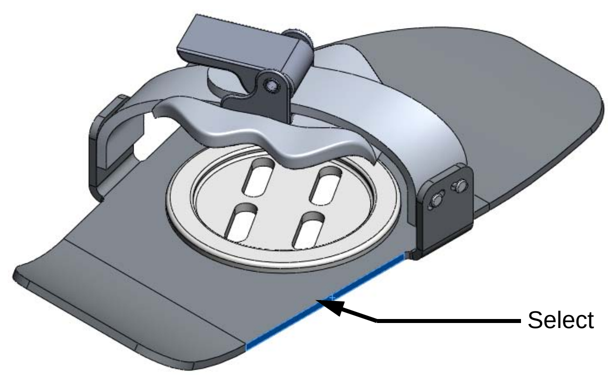
We will create the curved version of the part in the context of the Binding assembly.

1 Click **Insert, Component, New Part** from the menu.

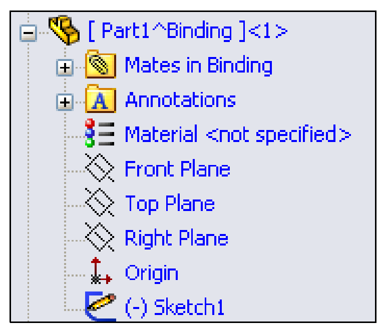
2 The cursor changes to indicating that we need to select a plane for the first sketch. Whichever plane or face we select will become the Front plane of the new part. When the cursor is over a valid face, it will change to .

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.



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 in-context 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.

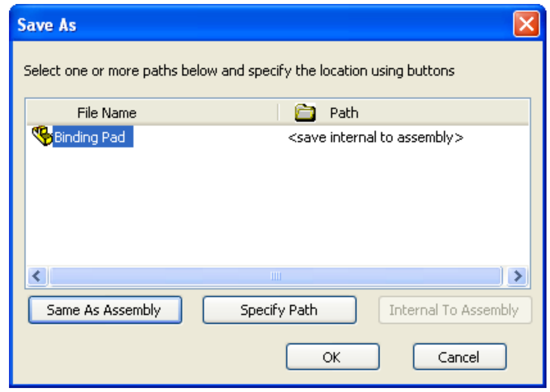
1 In the FeatureManager design tree, right-click [Part1^Binding] and select **Rename Part**. Type Binding Pad as the new file name.

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 Sketch **Tools, Convert Entities** from the menu.

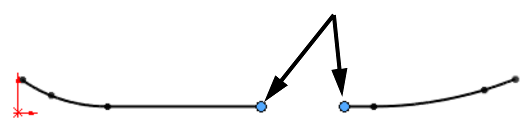
1 Reorient the view to the Right View.

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.

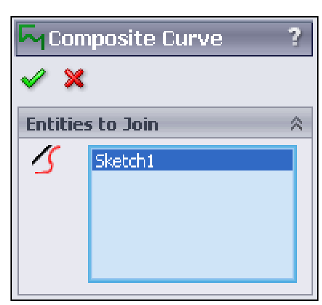


Note: Even though the endpoints were black, indicating that they were fully defined, they can be dragged to new locations. This is only true when the sketch entities were converted from edges.

6 **Exit** the sketch.

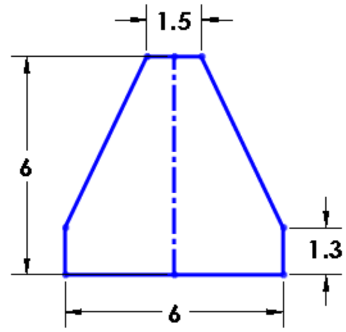
7 In the FeatureManager design tree, select the sketch.

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.

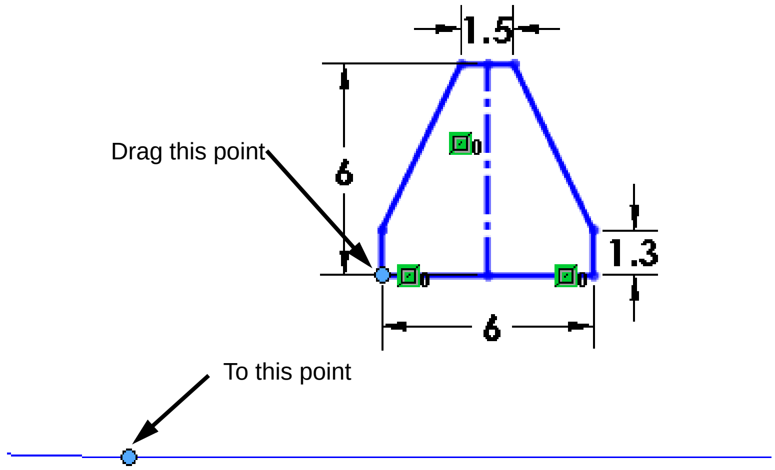


9 Click .

10 Create the following sketch on the Front plane.

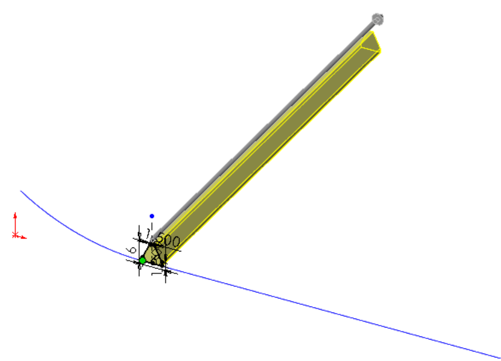


11 Drag the corner of the sketch onto the point in the composite curve where the left end of the straight line meets the tangent arc. The sketch should be fully defined.



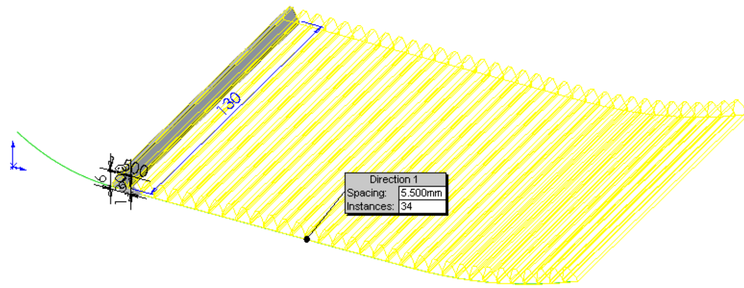
12 **Extrude** the sketch **130mm**.

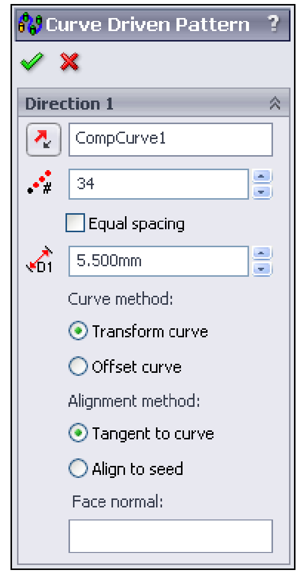
We will later cut away the extra material. For now we just need to make sure the extrusion will extend past the end of the Binding Base Plate.



13 Pattern the feature. Click **Insert, Pattern/Mirror, Curve Driven Pattern**.

Select CompCurve1 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**.



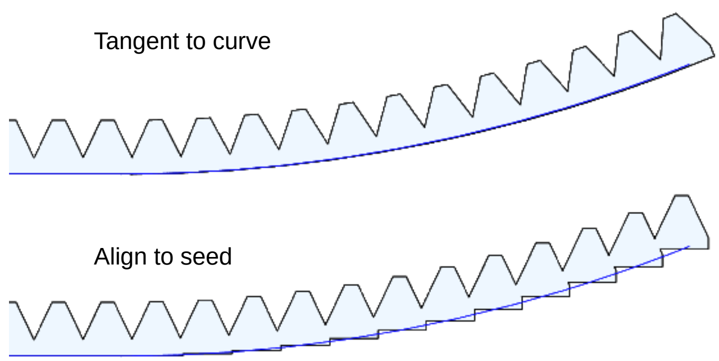


Note: Each instance of the extrusion was 6 mm wide but we patterned it with a spacing of 5.5 mm. This makes sure that the final pattern does not have gaps when it makes the turn up the slope at the end of the Binding Base Plate.

Pattern Alignment

Alignment method controls the orientation of the patterned instances. Selecting **Tangent to curve** will cause the pattern instances to rotate as the pattern changes direction.

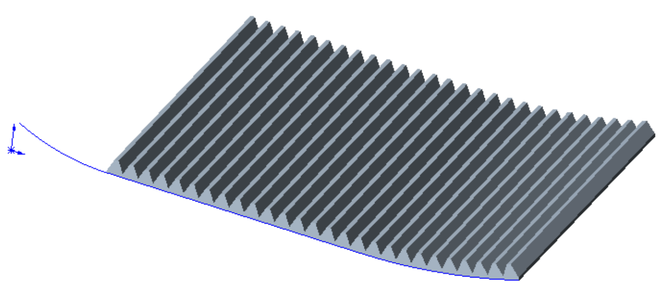
If **Align to seed** is selected, the pattern instances will maintain the same orientation as the seed feature.



14 Click .

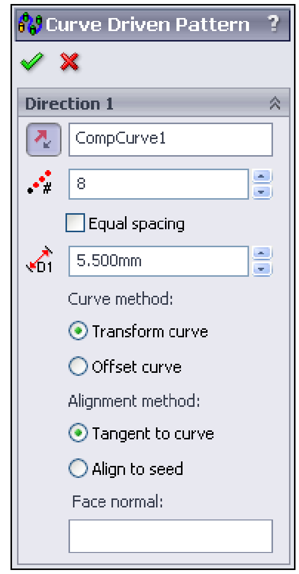
This creates the pattern in just one direction.

To create the rest of the pattern, we will create another pattern from the same extrude feature and pattern it in the other direction.

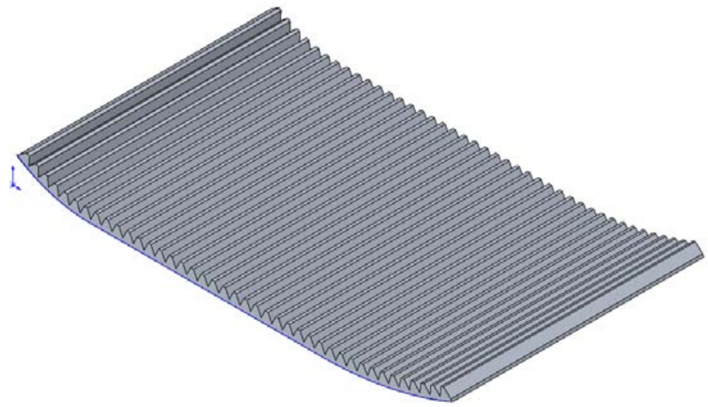


15 Create another pattern. Click **Insert, Pattern/Mirror, Curve Driven Pattern**.

16 Select CompCurve1 in the FeatureManager design tree for the direction and Extrude1 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 . This 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.

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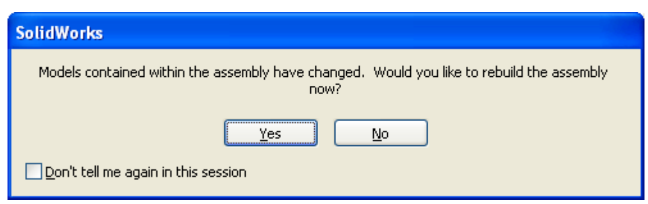
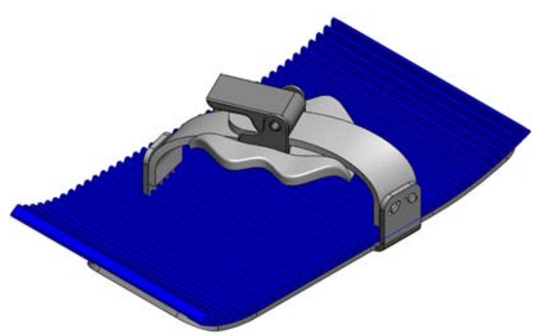
19 Because the Binding Pad is different from when we left the Binding assembly, the assembly needs to be rebuilt. Click **Yes**.



The Binding Pad should completely cover the top of the Binding Base Plate.



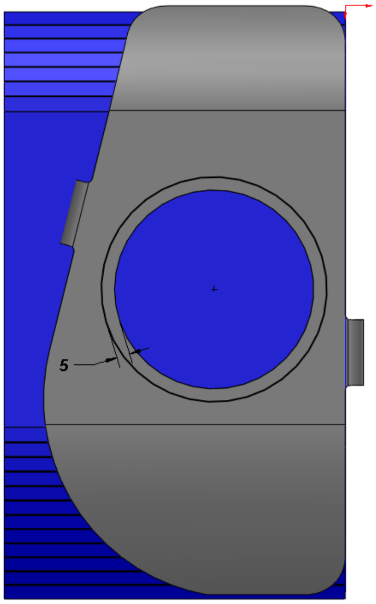
20 The only part we need to use to trim the Binding Pad is the Binding Base Plate. **Hide** all the other parts.

21 Reorient the model to the Bottom view.

22 Create a sketch on the bottom face of the pad.

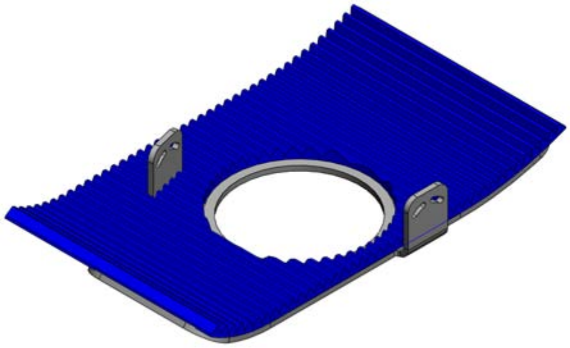
23 Select the edge of the hole in the Binding Base Plate.

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.



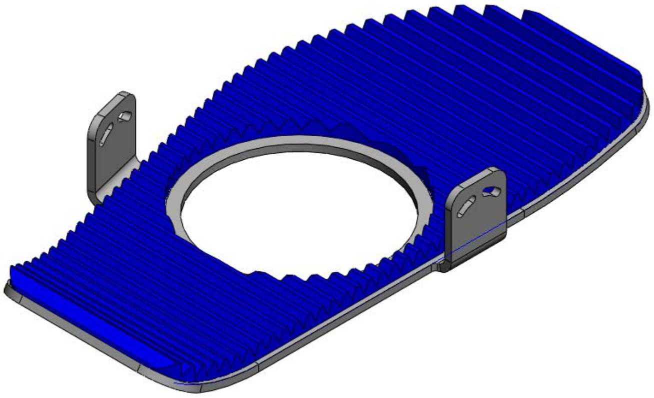
26 Create another sketch on the bottom face of the Binding Pad.

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**.

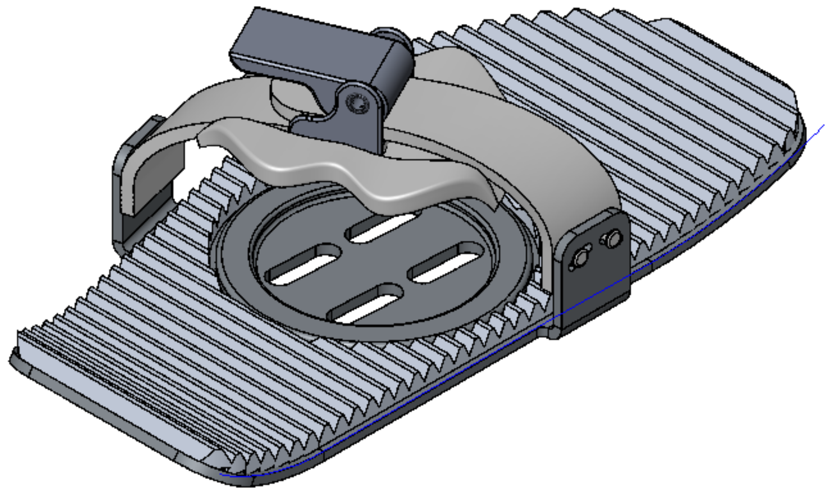
29 Return to the **Binding** assembly. Click  to exit **Edit Part**.



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

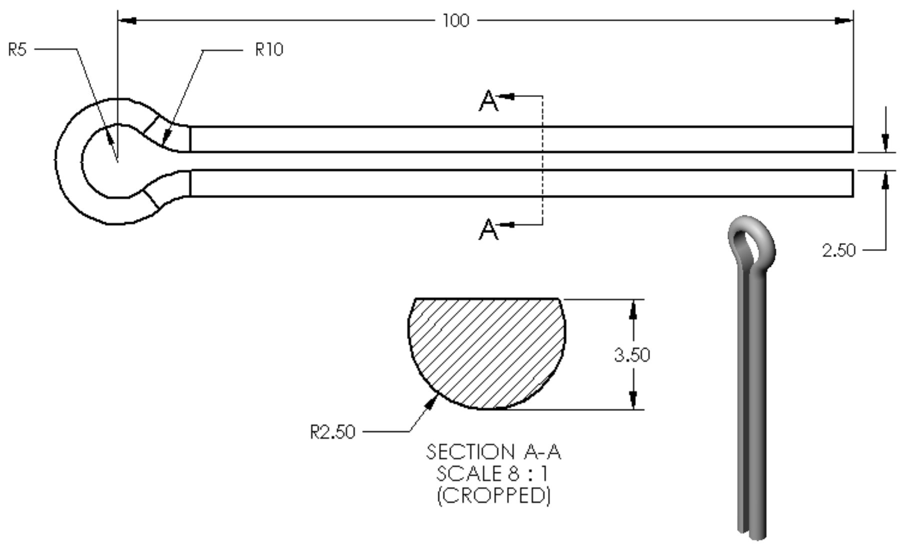
Exercise 19: Sweeps without Guides

Create these two parts using swept features. These require only a path and a section, no guide curves.

Units: **millimeters**

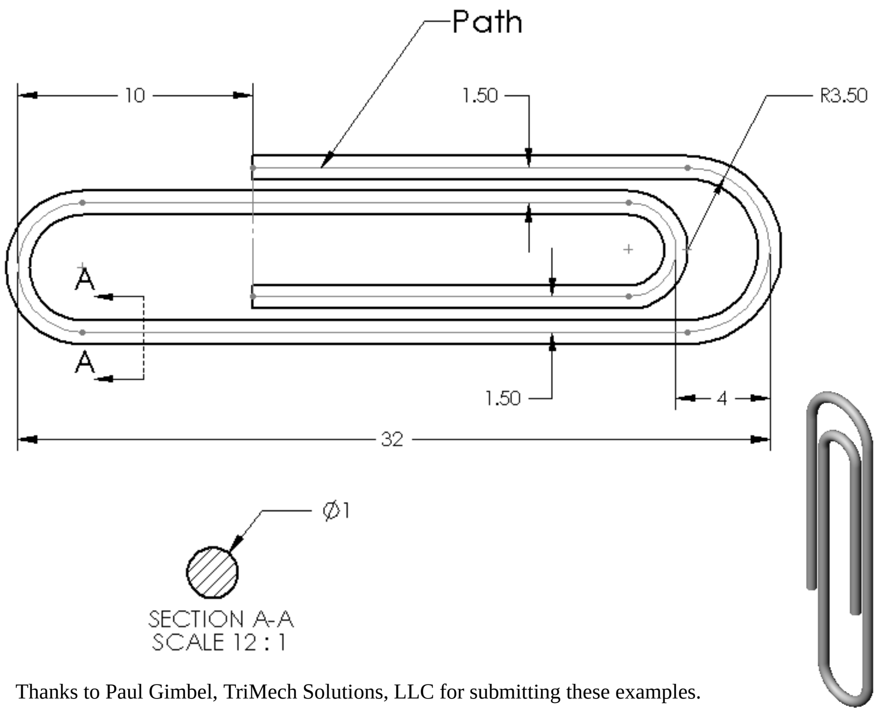
Cotter Pin

The `Cotter Pin` uses a path that describes the inner edge of the sweep.



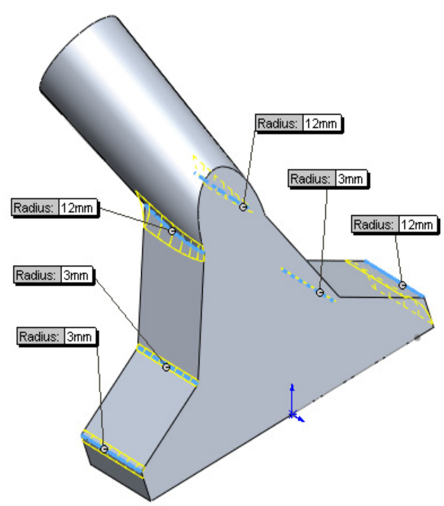
Paper Clip

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

Design Intent

The design intent for this part is as follows:

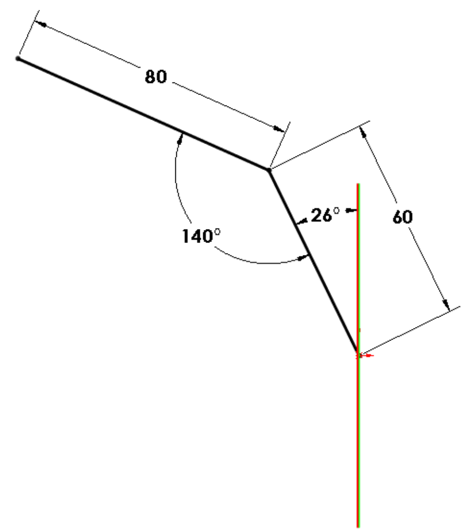
- ❑ Part is symmetrical.
- ❑ Wall thickness is uniform.

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.

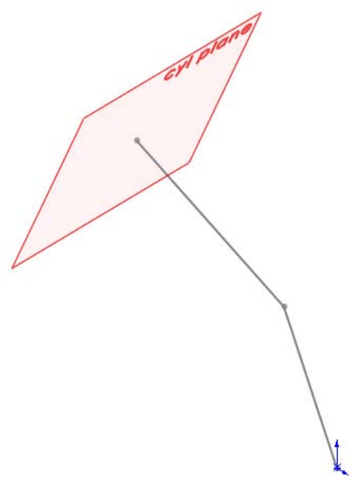
Note: The **26°** angle is dimensioned to the Right reference plane.

---

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.



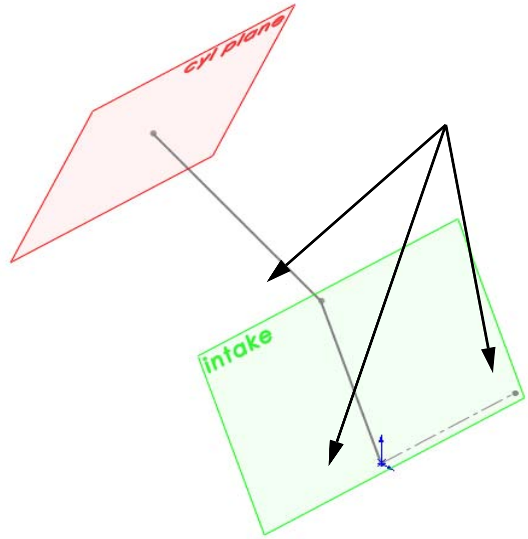
4 Plane through 3 points.

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.

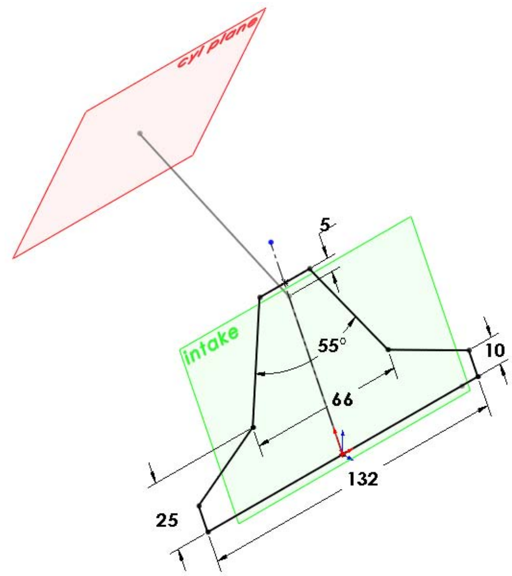
Name this plane intake.



5 Sketch the profile.

Sketch on the intake plane to create the profile of the nozzle.

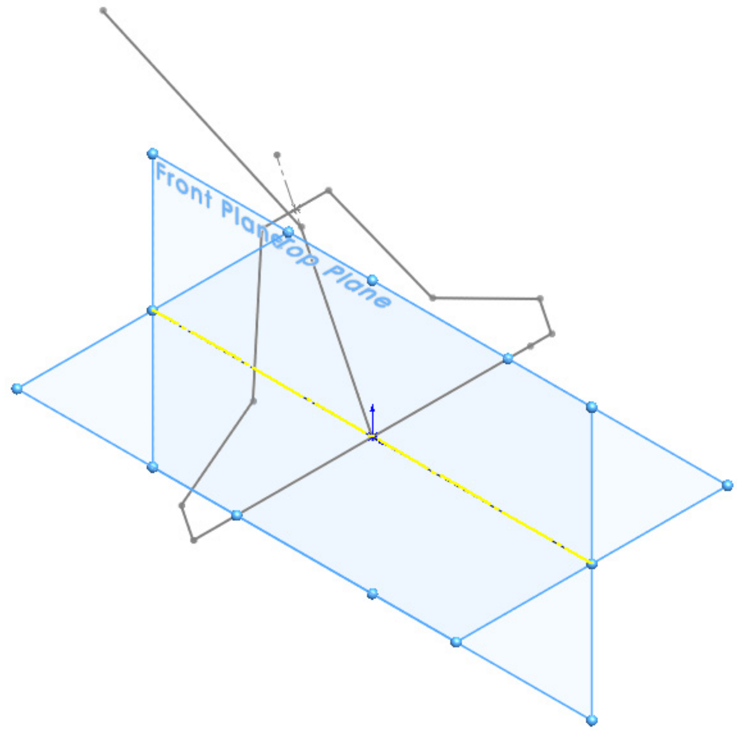
Use symmetry to create the sketch and tie it to the Layout sketch.



6 Axis.

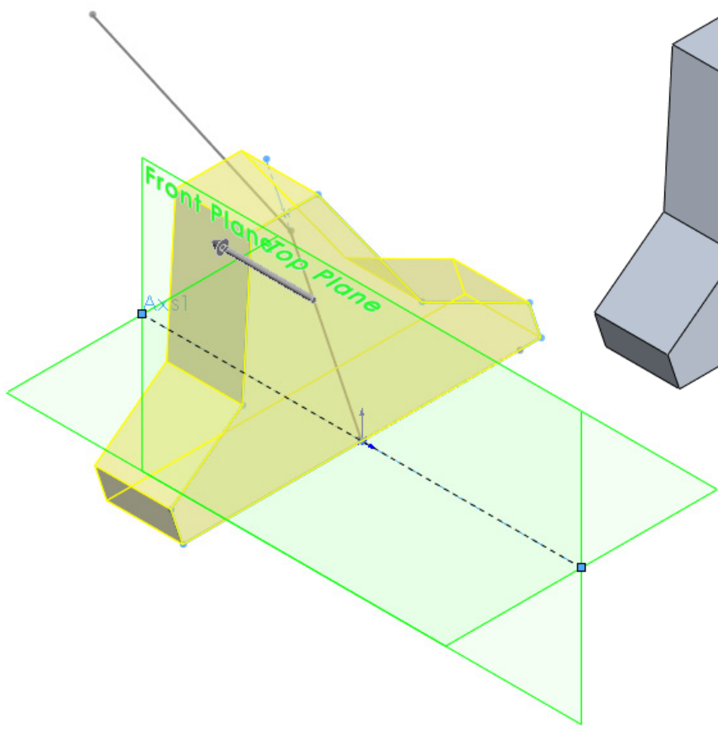
Create an axis defined by the intersection of the Front and Top reference planes.

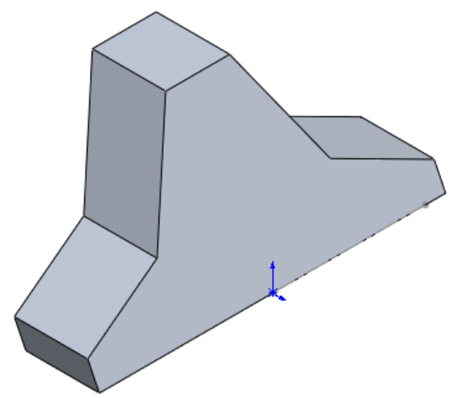
This will be the vector for the extrude direction.



7 Extrude.

Extrude the profile sketch using the **Blind** end condition. Select the axis for the **Direction of Extrusion**. Set the **Depth** to **28mm**.

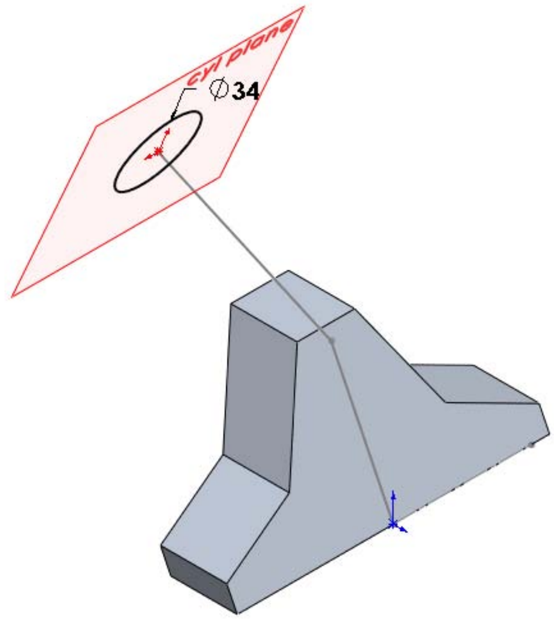




8 Cylinder.

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.

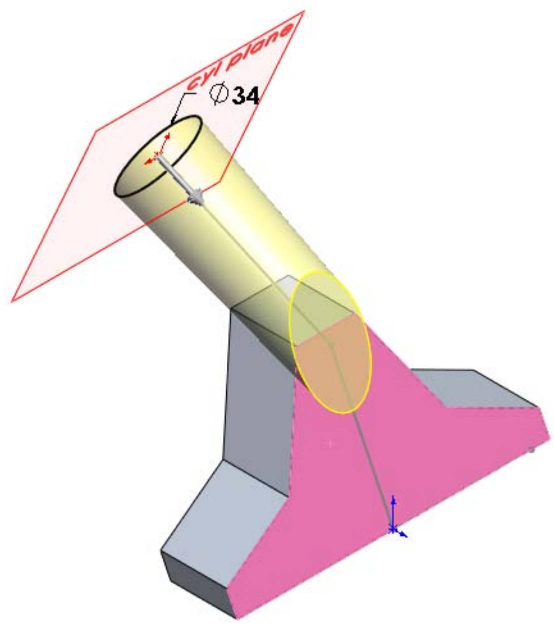
Click **Insert, Boss, Extrude**. Verify from the preview that the boss is extruding in the correct direction. If it is not, click **Reverse Direction**.

From the **End Condition:** list, select **Up to Surface**.

Select the front face of the swept first feature.

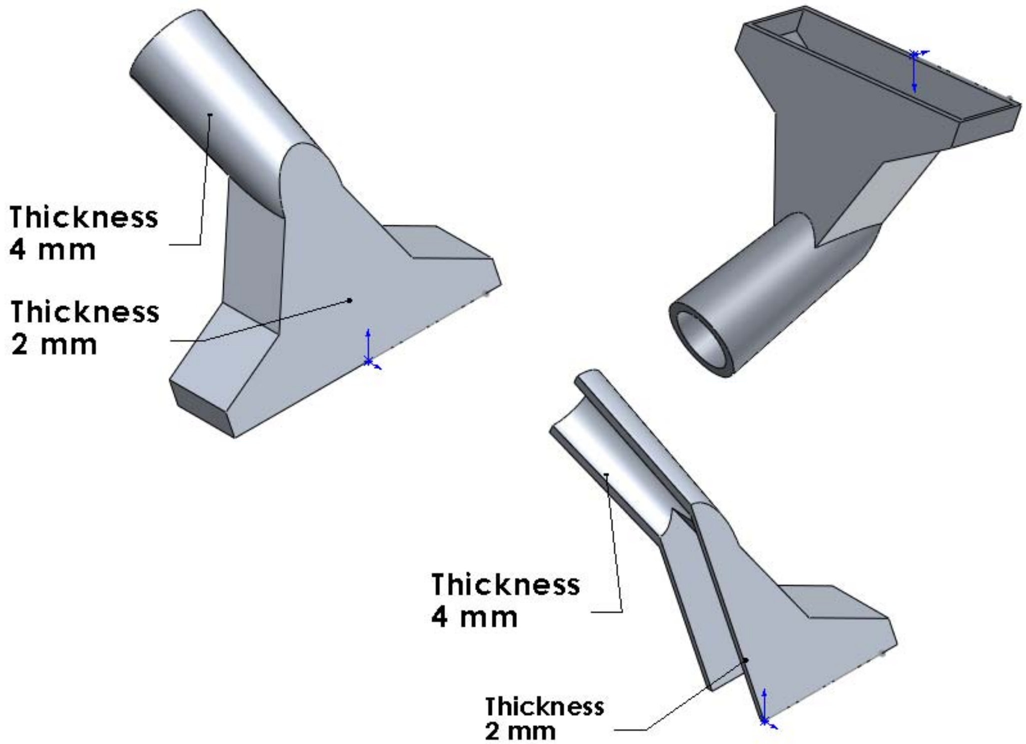
Click **Draft On**, set the angle to **2°**, and check **Merge result**.

Click ✓.



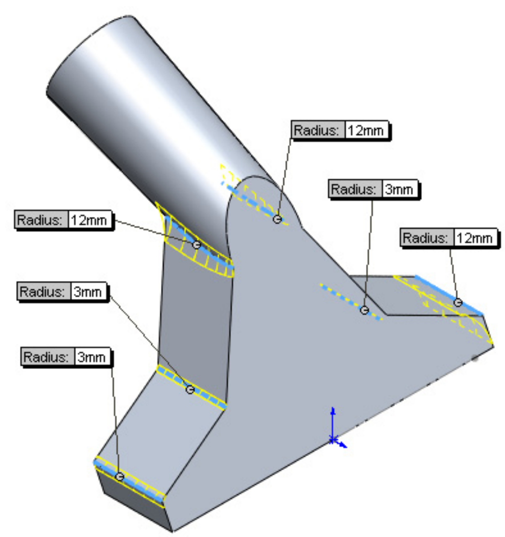
10 Multiple-thickness Shell.

Shell the solid **2mm** to the inside, selecting the end faces for removal. Select the cylindrical face and set it to **4mm**.



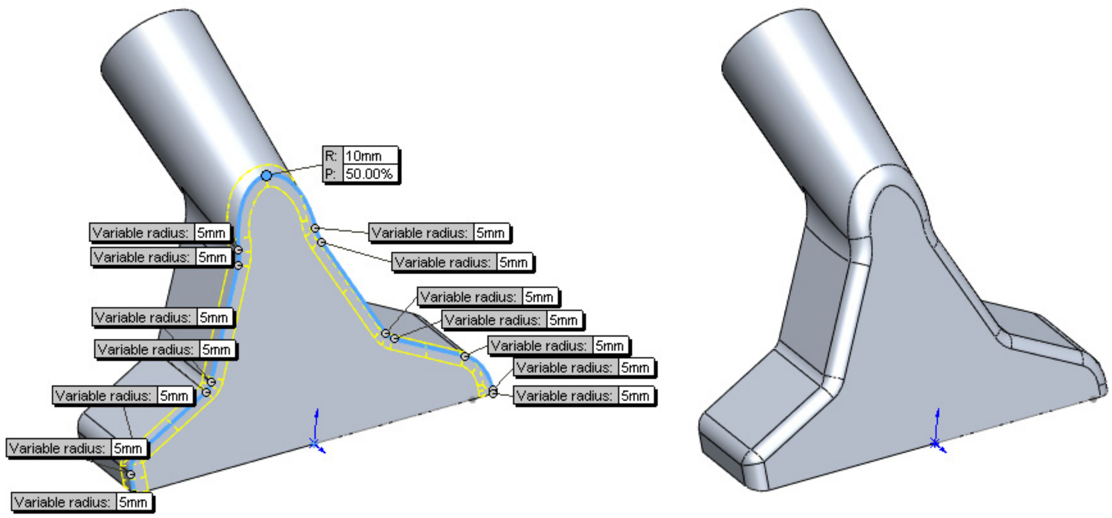
11 Fillets and rounds.

Add fillets and rounds to the outside of the solid body as shown.



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**.

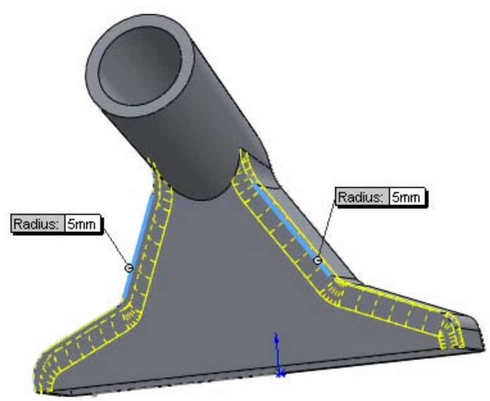


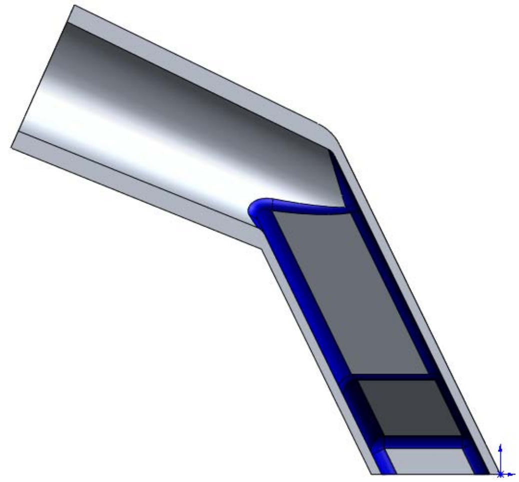
TIP: This technique simplifies assigning the values to the vertices:

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.



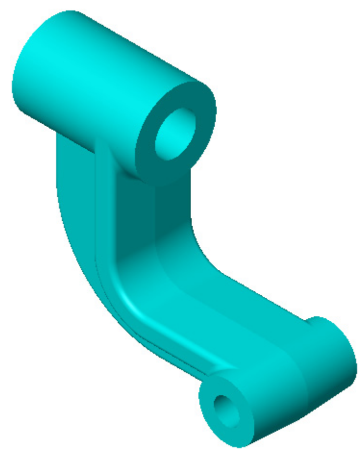


14 Inner fillets and rounds.

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



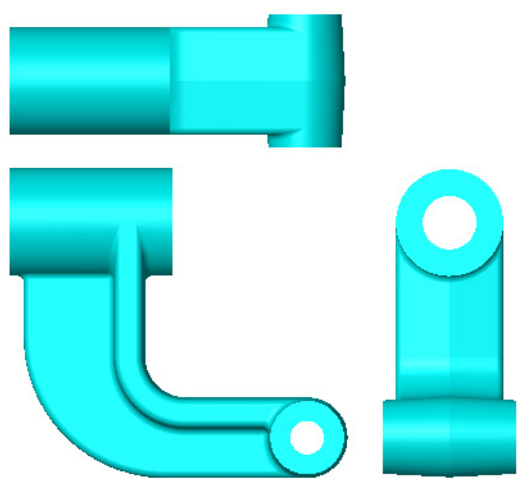
Create this by following the steps as shown.

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:

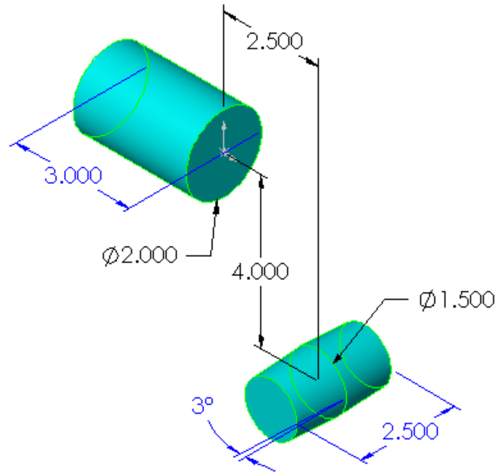
- ❑ All fillets and rounds are **0.125”**.
- ❑ Part is symmetrical with respect to the parting line.
- ❑ Draft is **3°**.

Procedure

1 Open a new part using the `Part_IN` template and name it `Hanger Bracket`.

2 Create sweep ends.

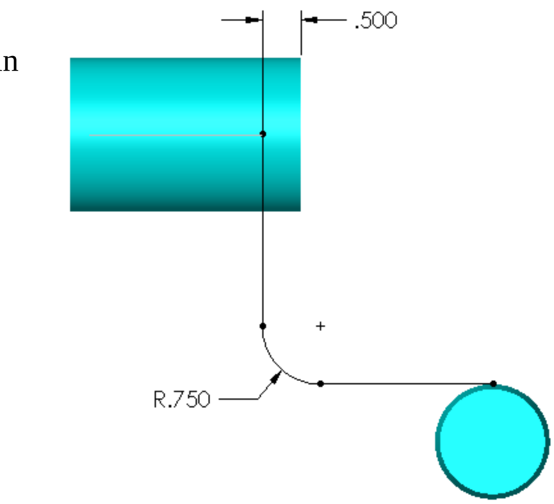
Create two extruded solid bodies to represent the ends of the sweep.



3 Create sweep path.

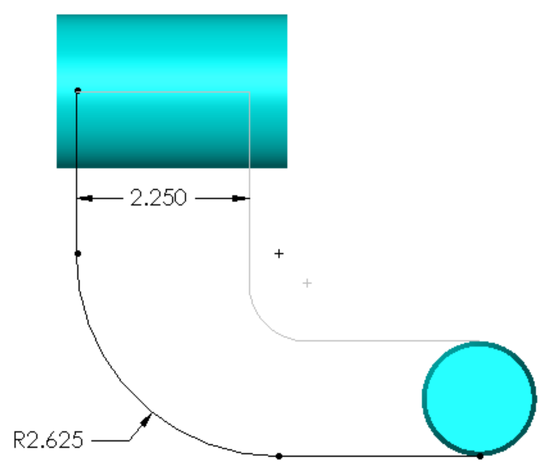
The path and the guide curve must each be in separate sketches.

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.

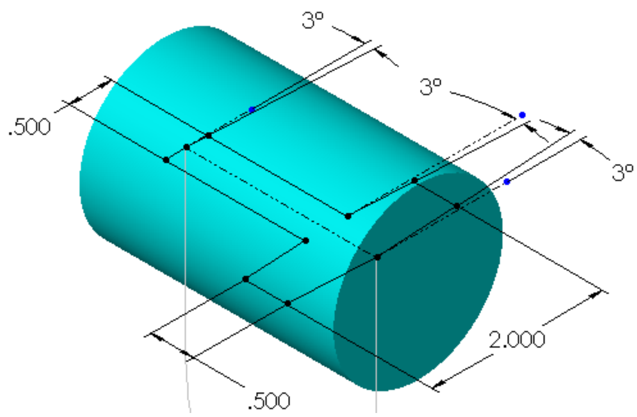


TIP: If you sketched *all* the geometry in one sketch, it can still be used. Change the two lines and the arc that form the guide curve to construction geometry. Open a new sketch for the guide curve.

Use **Convert Entities** to copy the guide geometry into the new sketch.

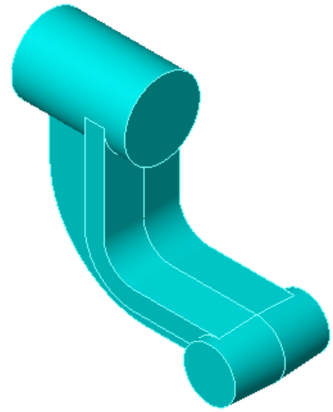
5 Create sweep section.

Create the sweep section as a sketch using the dimensions shown at the right.



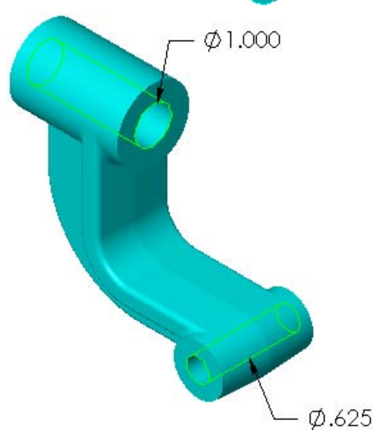
6 Insert sweep.

Using the sketches, sweep the feature. Use the **Merge result** option to combine all the solid bodies.



7 Create through holes.

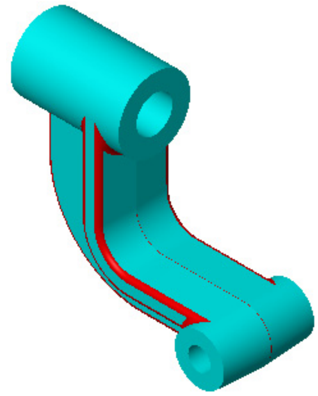
Add two through holes cuts to the model.



8 Insert fillets and rounds.

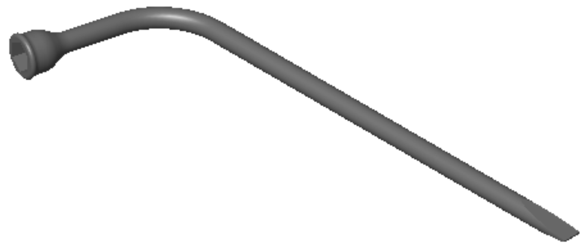
Add **0.125”** fillets and rounds, shown here in red, to complete the model.

TIP:	Filleting by feature works best.



9 Save and close the part.

Exercise 22: Tire Iron



Create this by following the steps as shown.

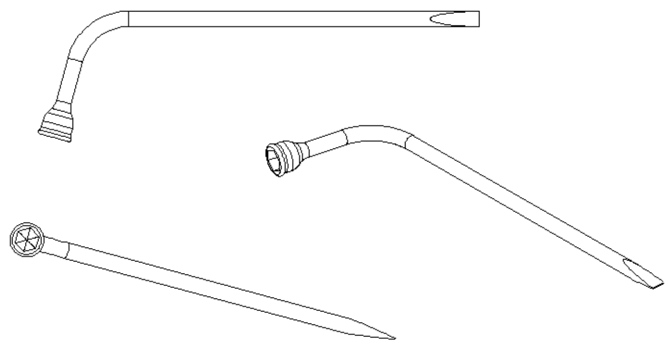
This lab uses the following features:

- 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.

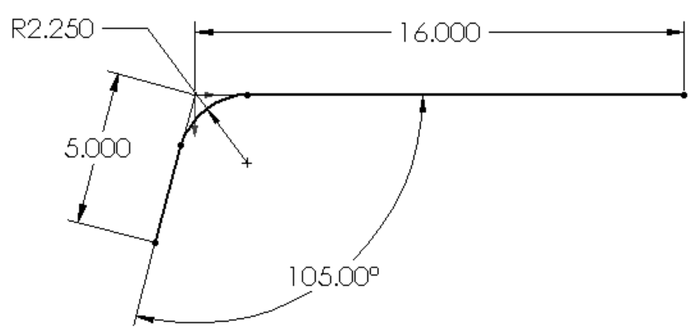


Procedure

1 Open a new part using the `Part_IN` template and name it `Tire Iron`.

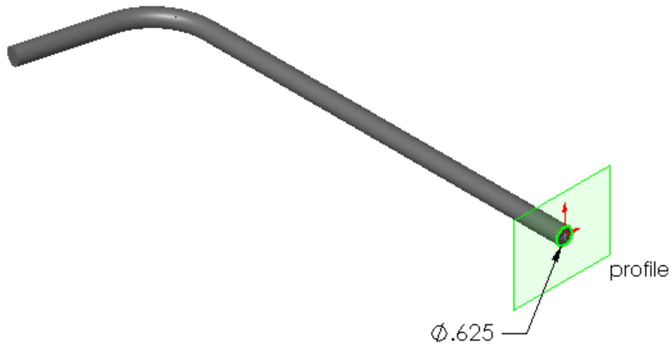
2 Create the sweep path.

Create the sketched lines then add the fillet.



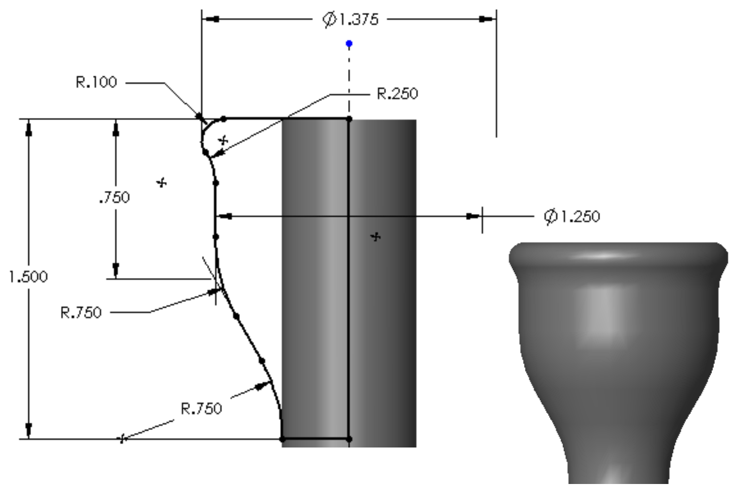
3 Insert sweep.

Create a new reference plane and use it to sketch the sweep section sketch. Sweep the profile along the path.



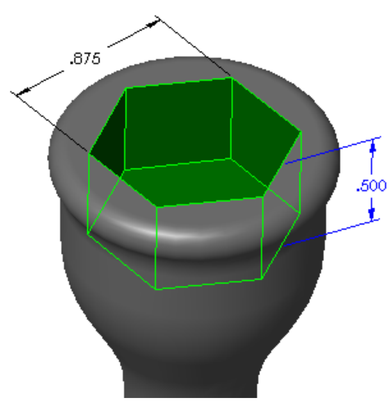
4 Revolved feature.

Create a revolved feature on the angled end of the sweep feature. This boss will hold the hexagon cut.



5 Hexagonal cut.

Create a hexagonal cut using the **Polygon** tool .



Lesson 7 Quiz

Name: ______________________________Class: _________ Date:_______________

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

1 Describe the steps required to create a swept feature.

____

____

____

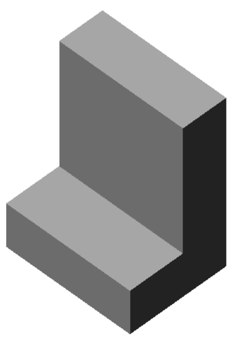
____

____

____

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.



Part 1

Part 2

Part 3

Part 1: ____

Part 2: ____

Part 3: ____

3 Describe the steps required to create a Loft feature.

____

____

____

____

____

____

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

____

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

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

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.

8

Lesson 8: Final Assembly

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 sub-assemblies 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.

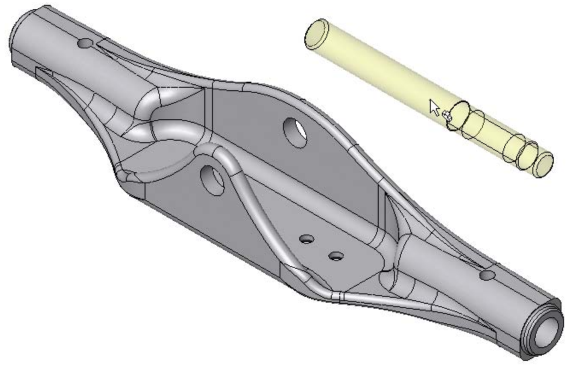
3 When the preview image appears with your cursor, move the cursor over the assembly origin until the icon appears, then click the origin.

4 Save the new file as Axle Assembly to the Mountainboard folder.

5 Click **Insert Component** 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.

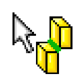
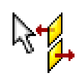
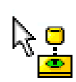
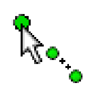


Smart Mates

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.

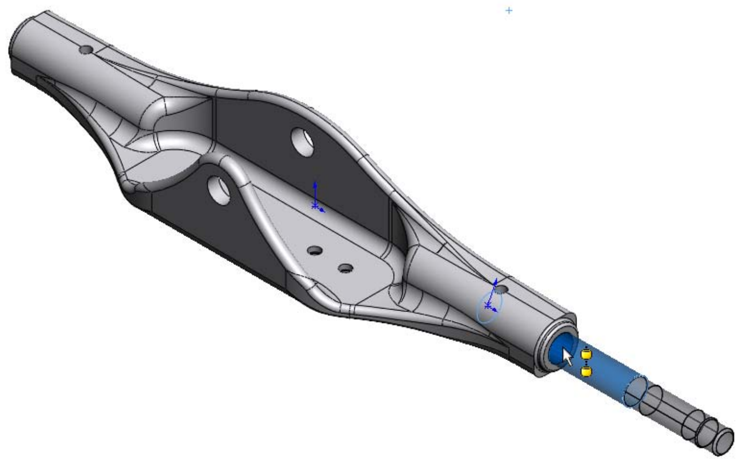
Certain techniques generate multiple mates and do not use the toolbar. These require the use of the **Tab** key to switch mate alignment.

Mating Entities	Type of Mate	Pointer
2 linear edges	Coincident	
2 planar faces	Coincident	
2 vertices	Coincident	
2 conical, or 2 axes, or 1 conical face and 1 axis	Concentric	
2 circular edges	Concentric and Coincident	

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.

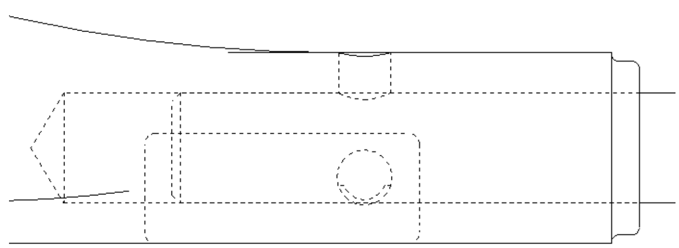


- Drop the part when the **Concentric** pointer appears, indicating a concentric mate.
- Confirm the **Concentric** type from the **Mate Pop-up** Toolbar.

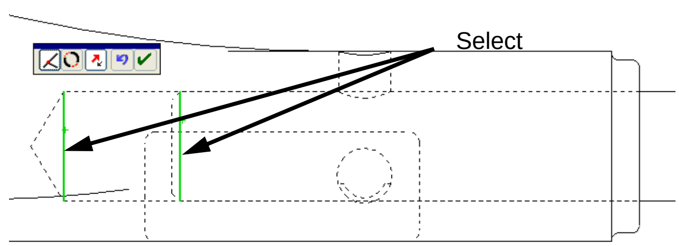
A **Concentric** mate is added between the `Axle Shaft` and the `Axle` parts.

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 on the **View** toolbar.



4 Click **Mate** on the Assembly toolbar and select the edge of the `Axle Shaft` and the conical face shown. Click to accept the **Coincident** mate.



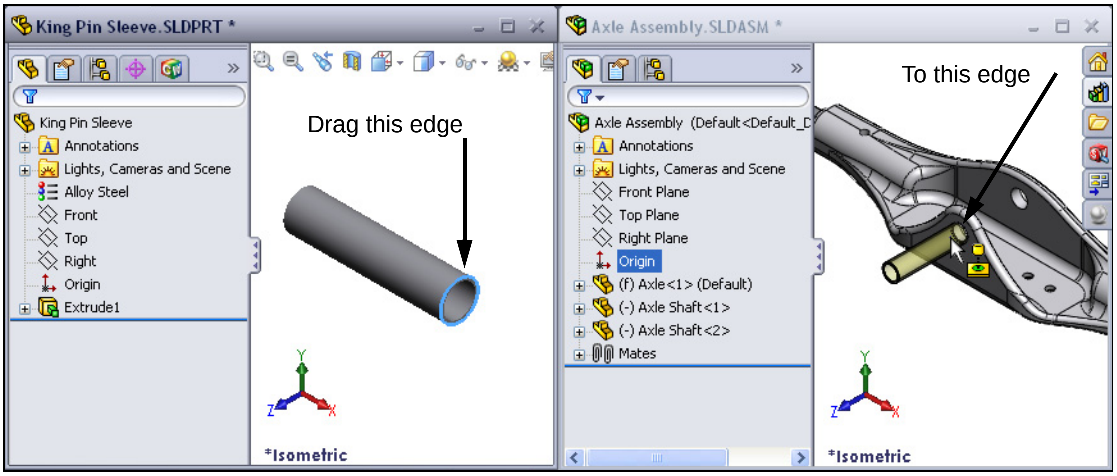
5 Hold down the **Control** key and drag another copy of the `Axle Shaft` from the FeatureManager design tree into the graphics area. This will insert another instance of the `Axle Shaft` part into the assembly.

6 Apply the same **Concentric** and **Coincident** mates this shaft.

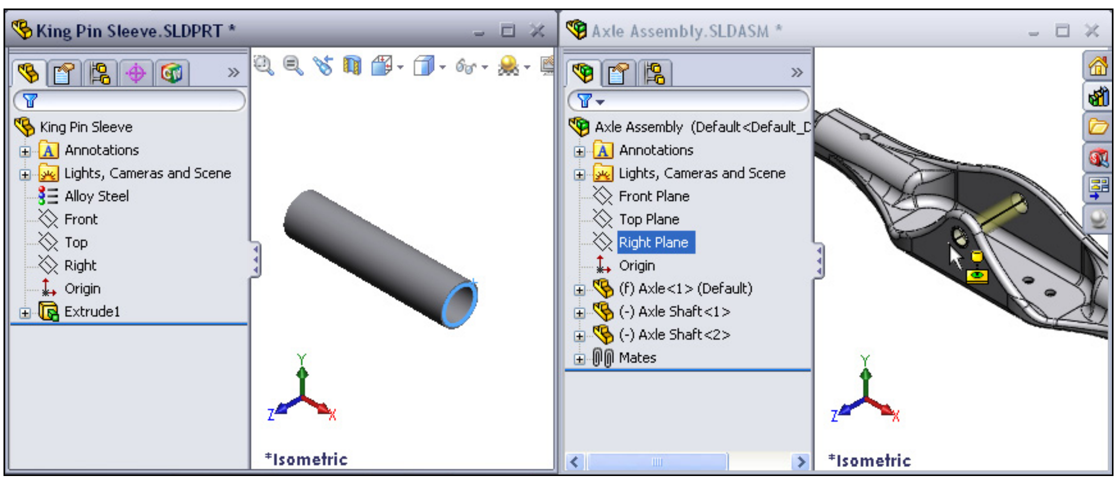
7 **Open** the part `King Pin Sleeve` from the `Mountainboard\Hardware` folder.

8 Tile the two windows vertically by clicking **Windows, Tile Vertically** from the menu.

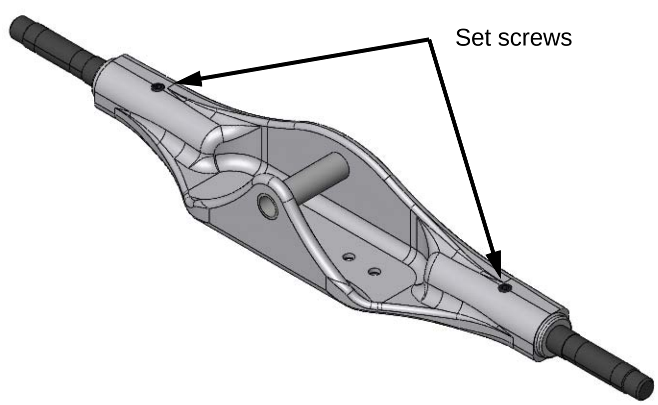
9 Drag the edge of the King Pin Sleeve shown to the edge of the Axle indicated. When you are on the correct edge the cursor will change to the **Pin in Hole**  .



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.

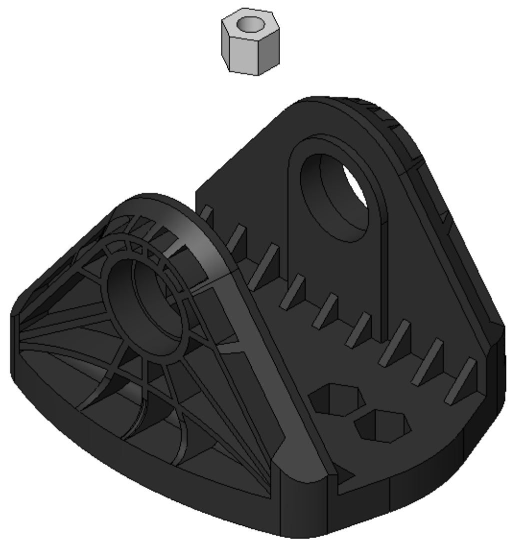


Mate each set screw using one **Concentric** and one **Coincident** mate.

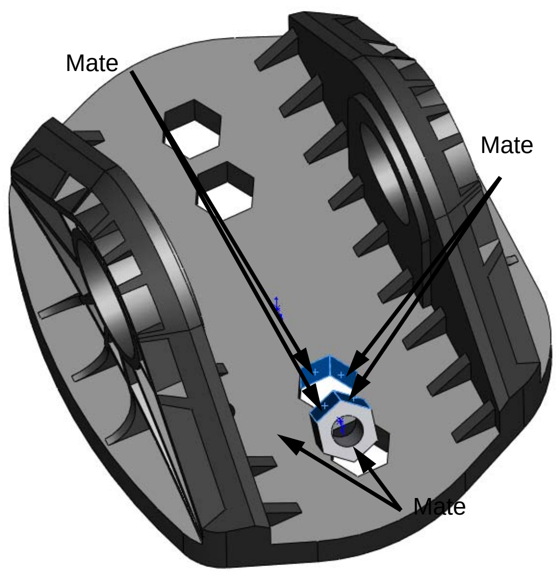
12 **Save** the assembly.

Task 3 — Create the Truck Assembly

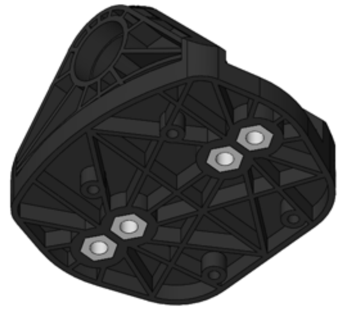
1. Click **File, New** from the menu and select the Assembly_MM template.
2. Click **Browse** and locate the part Truck in the Mountainboard folder.
3. When the preview image appears with your cursor, move the cursor over the assembly origin until the  pointer appears, then click the origin.
4. Save the new file as Truck Assembly to the Mountainboard folder.
5. Click **Insert Component**  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 `8mm Threaded 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.

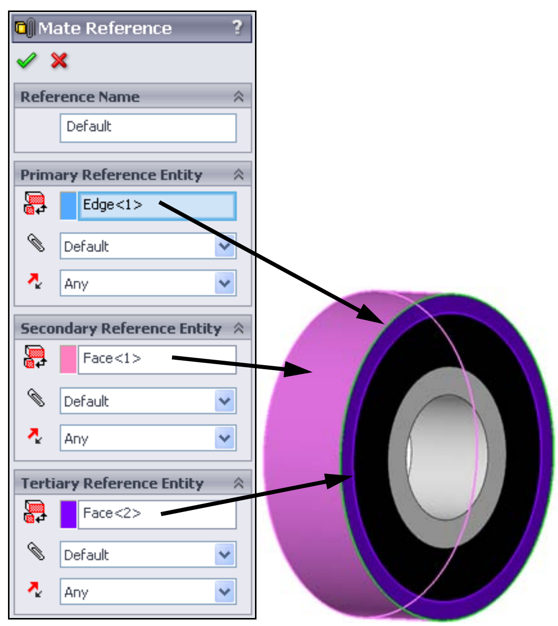
Primary, Secondary, and Tertiary References

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.

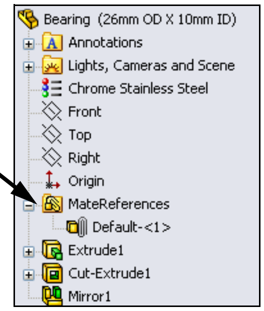
Task 4 — Add Mate References to the `Truck` bearing.

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  .

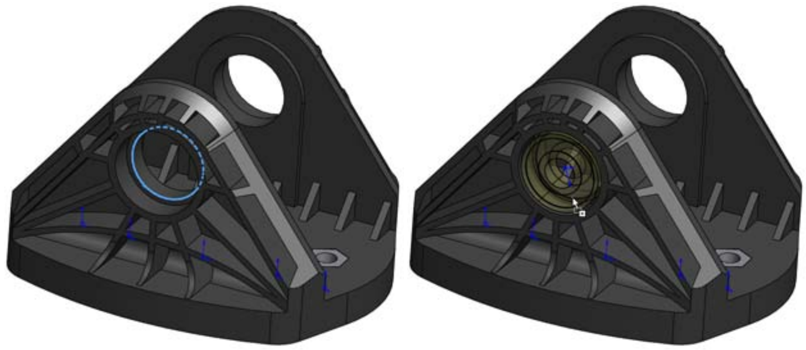


5 The MateReferences will be listed in the FeatureManager design tree.

6 **Save** and **Close** the part.



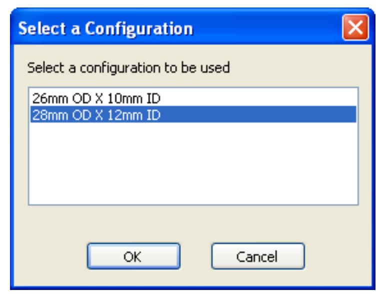
7 Drag a Bearing from the Mountainboard \Parts folder in the File Explorer to the edge shown The Bearing will snap into position and both a **Coincident** and **Concentric** mate will be created. If the Bearing is sticking out of the hole, press the **Tab** key to reverse the alignment.



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**.

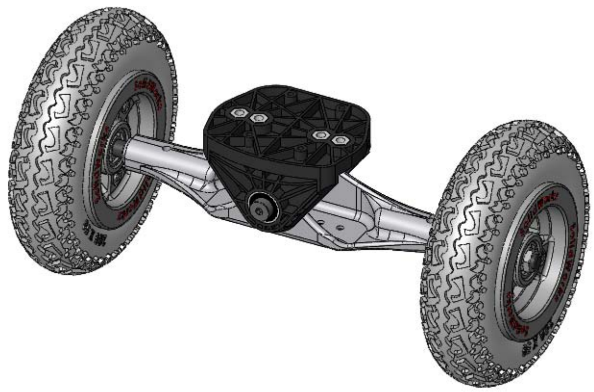
10 **Save** the assembly as Truck Assembly.



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.

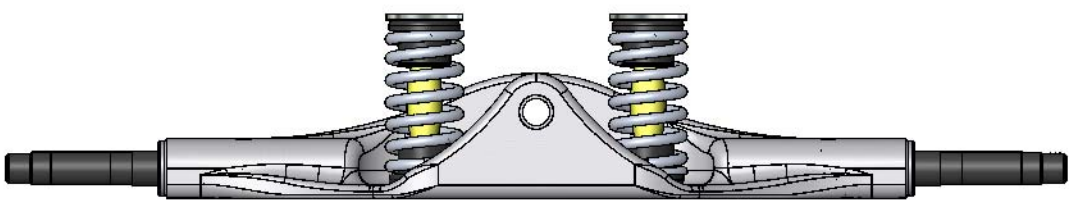
3 Drop the Axle Assembly on the origin of the new assembly.

4 **Save** the new assembly to the Mountainboard folder as Truck_Axle_Wheel.sldasm.

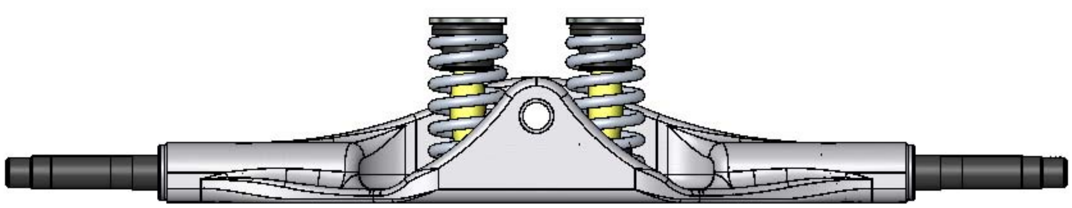
5 Insert two instances of the Spring Assembly into the Truck_Axle_Wheel assembly.

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.

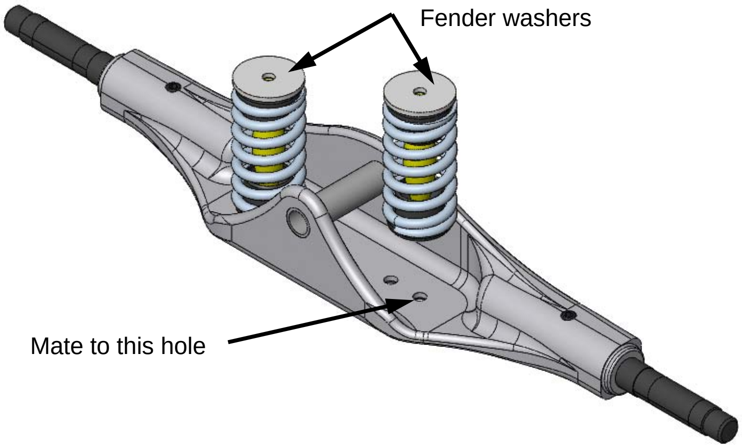


Stiff Ride



Soft Ride

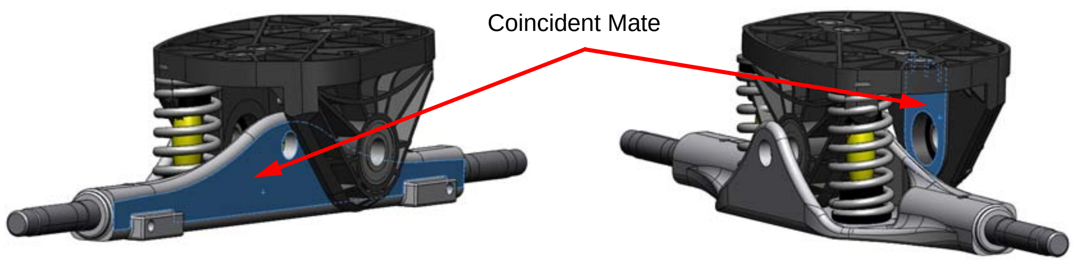
6 Mate each of the Spring Assemblies to the outer holes in the Axle.
The end of the Spring Assembly with the Fender Washer should be up.



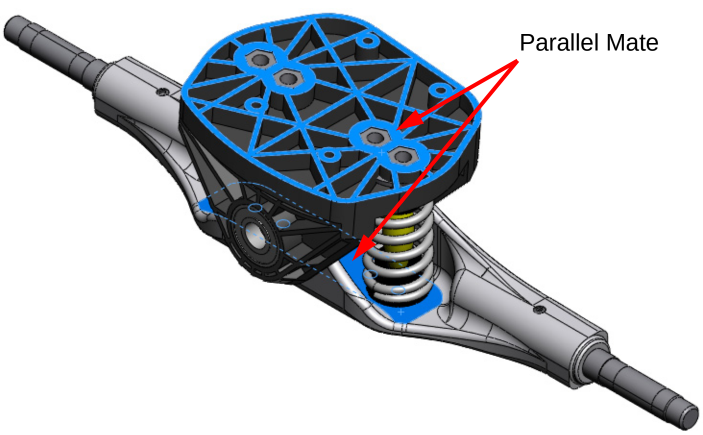
7 Drag the Truck Assembly into the Truck_Axle_Wheel assembly.

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.



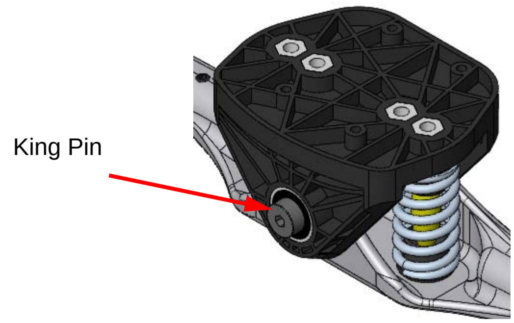
10 Add a **Parallel** mate between the two faces shown.

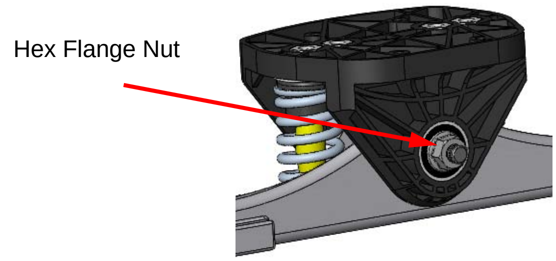


Task 6 — Add the remaining hardware

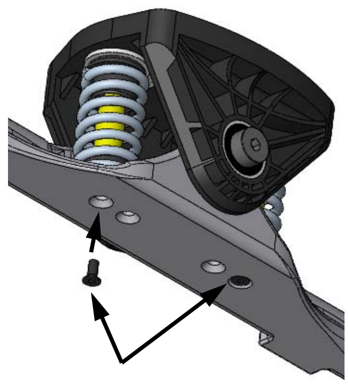
While our CAD model is held together with mating relationships, we need to add the remaining fasteners that will hold the real model together. The parts provided have mate references so that we can drag and drop them from the Design Library to our assembly.

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.



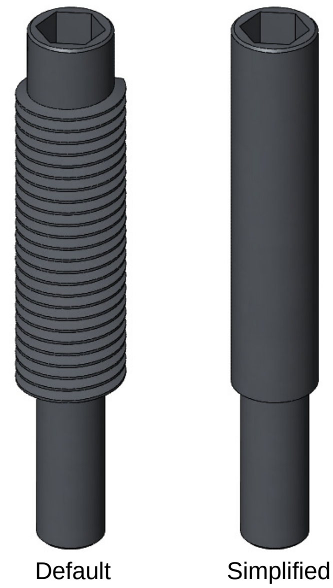


2 Add, from Toolbox, two B18.3.5M - 4x0.7x10 Socket FCHS-10N screws to hold the bottom of the spring assemblies to the axle.

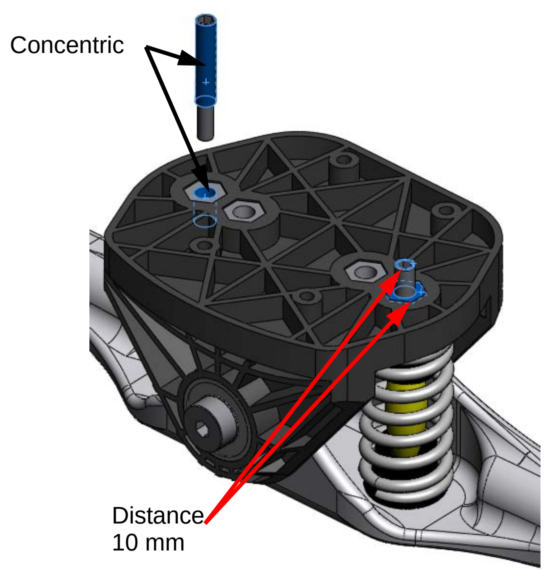


3 **Open** the part Adjusting Screw from the Mountainboard\Parts folder. This part is used to pre-load the springs to further adjust the ride of the mountainboard. There are two configurations of this part, the Default configuration that has all the threads modeled and a Simplified configuration without 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.



5 Mate each Adjusting Screw with a **Concentric** 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.

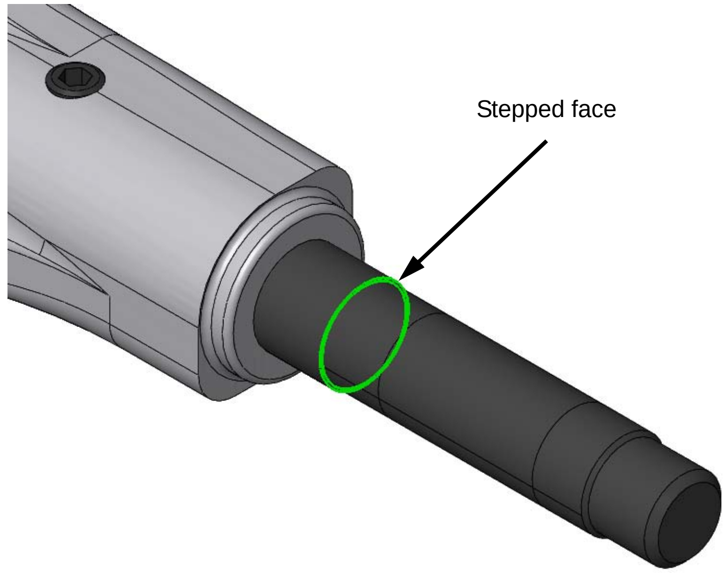
2 Click **Browse** and locate the Wheel Assembly. Insert two instances of the Wheel Assembly into the Truck_Axle_Wheel assembly.

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.

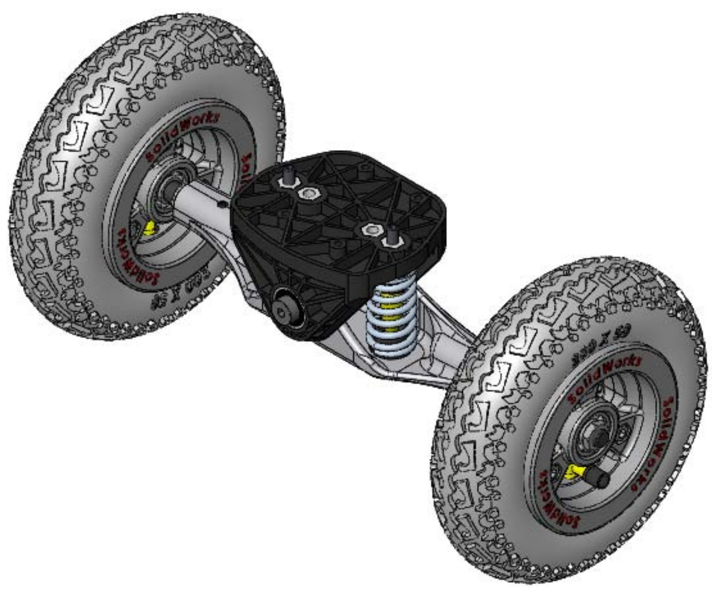
5 Add a **Coincident** mate between this stepped face and the side of the bearing.



TIP: To select the stepped face, you will have to zoom in close as the face is only .5 mm wide.

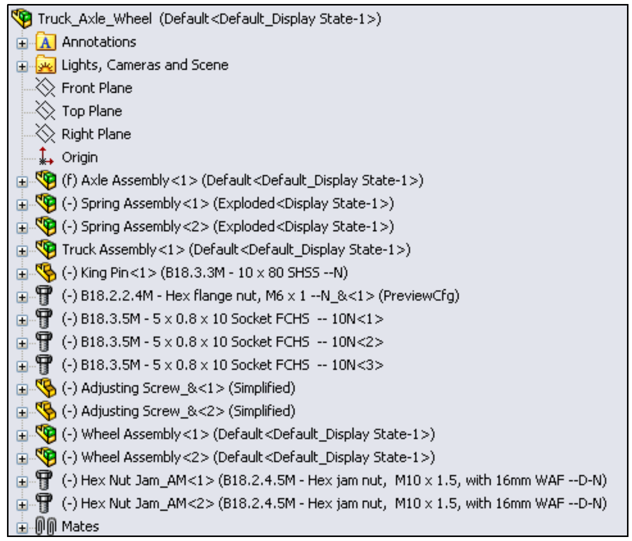
6 Finish the assembly by adding a `B18.2.4.5M-Hex jam nut, M10 x 1.5` to each `axle Shaft` to hold on the wheels.

7 **Save** the assembly.



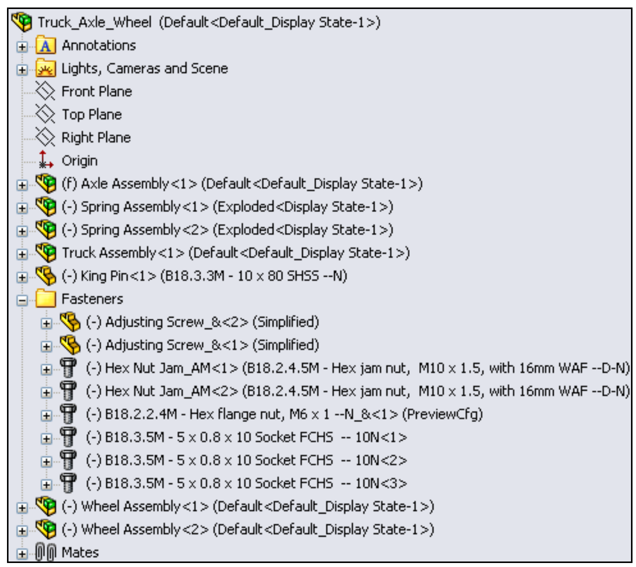
Task 8 — Clean up the FeatureManager design tree

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.

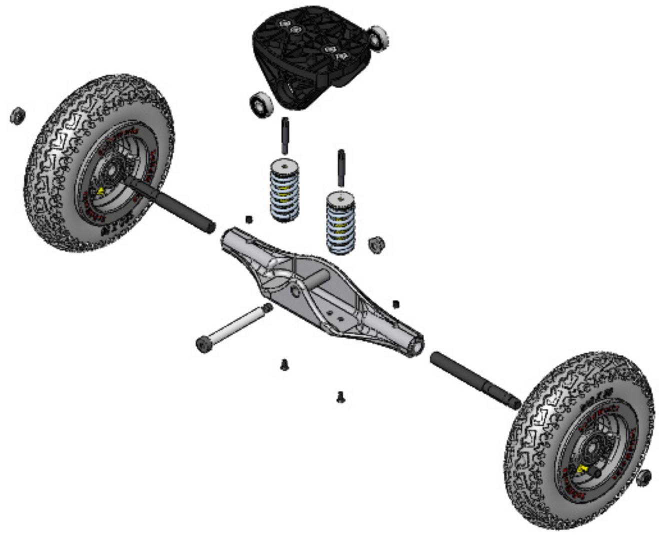


1. Right-click any of the fasteners in the FeatureManager design tree and select **Add to New Folder**.
2. Name the folder `Fasteners`.
3. Drag each of the fasteners into this folder. Drag the part to the folder.

   When the cursor changes to ⏎ 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.



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 Lesson08\Built Parts folder.

Task 10 — Create a drawing of the Wheel Assembly

The procedure to create an assembly drawing are the same as creating a drawing of a part.

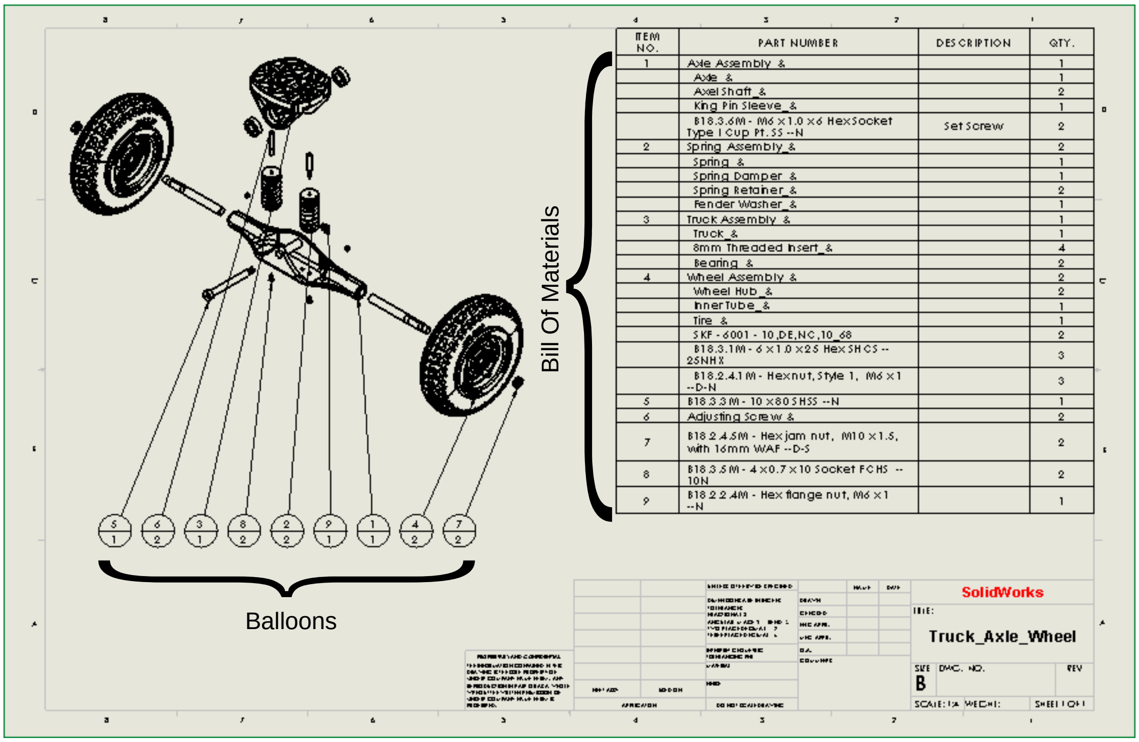
Bill Of Materials

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.

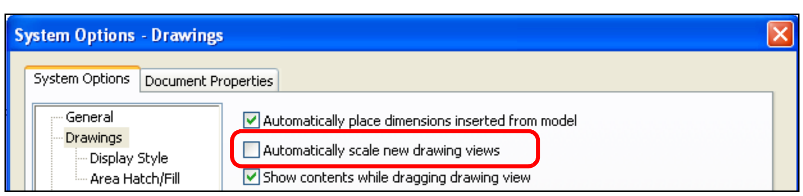
Balloons can be formatted to show the item number, quantity or a custom property.



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**.

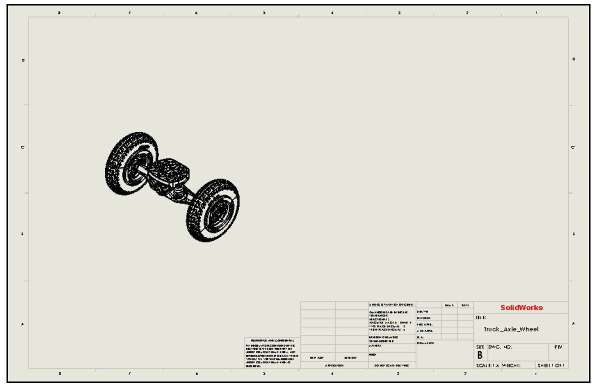
If **Automatically scale new drawing views** is selected, SolidWorks will override the scale of the drawing template to make the drawing view fit. In our case, we want to have the drawing views use the scale of the drawing template.



2 Click **OK** to close the **Options**.

3 With the `Truck_Axle_Wheel` assembly still open, click **Make Drawing from Part/ Assembly** on the Standard toolbar.

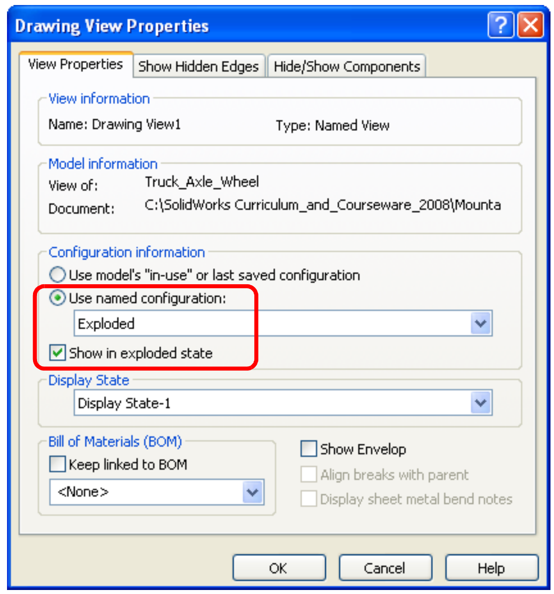
4 Select the Drawing template `B-Scale1to4`.



5 In the Task Pane, select the `Isometric` view, then drag it onto the drawing.

Click .

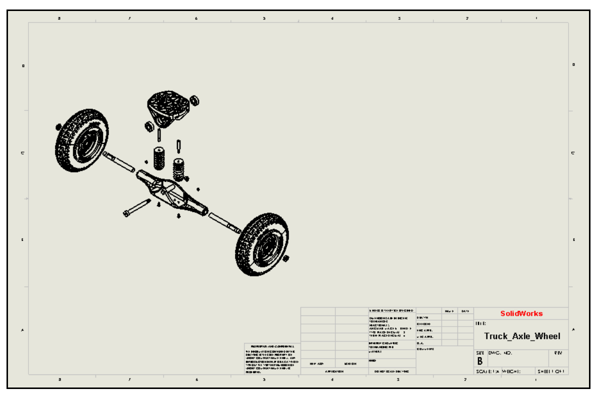
6 We want to show the exploded view instead of the collapsed view. Right-click `Drawing View1` in the FeatureManager design tree and select **Properties**.



7 Select **Use named configuration:** and select **Exploded** from the list. Click **OK**.

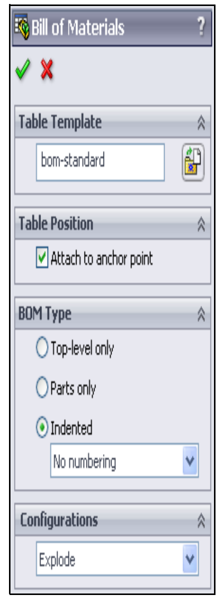
8 Select **Show in exploded state**. This will change the view from collapsed to exploded.

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.



9 Position the view as shown by dragging its border.

10 Select the view and then click **Insert, Tables, Bill of Materials**.

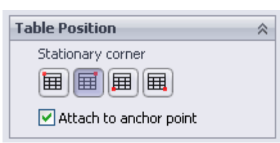


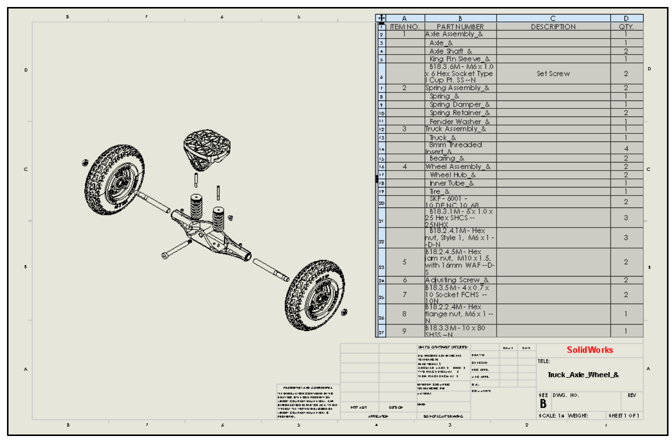
11 Select the following choices:

- Table Template: **bom-standard**
- Table Anchor: **Attach to anchor**
- BOM Type: **Indented, No numbering**
- Configurations: **Explode**

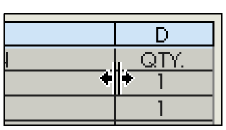
Click .

12 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.



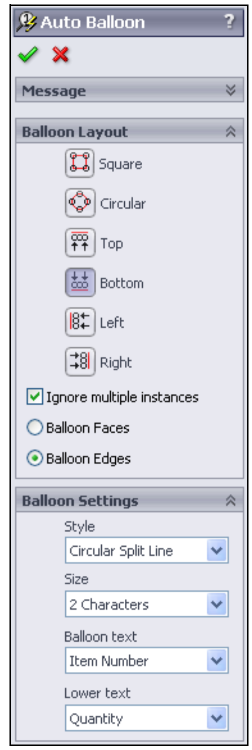


If it is too wide it may cover part of the drawing view. Adjust the column widths by dragging the column borders. When the cursor is over a column border it will change to the pointer shown below.



13 Examine the BOM. Each sub-assembly and independent part has its own item number.

14 Select the drawing view and click **AutoBalloon**  on the Annotations toolbar.



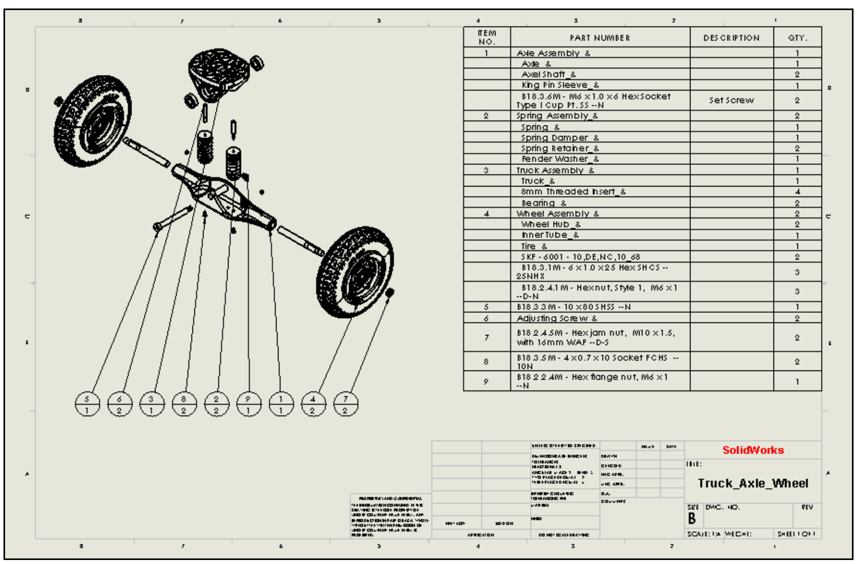
15 Select **Bottom** for Balloon Layout.

16 From the pull-down lists in the PropertyManager, select:

- Style: **Circular Split Line**
- Size: **2 Characters**
- Balloon text: **Item Number**
- Lower: **Quantity**

17 Drag the balloons to a position that is centered under the drawing view.

18 Click .



19 **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.

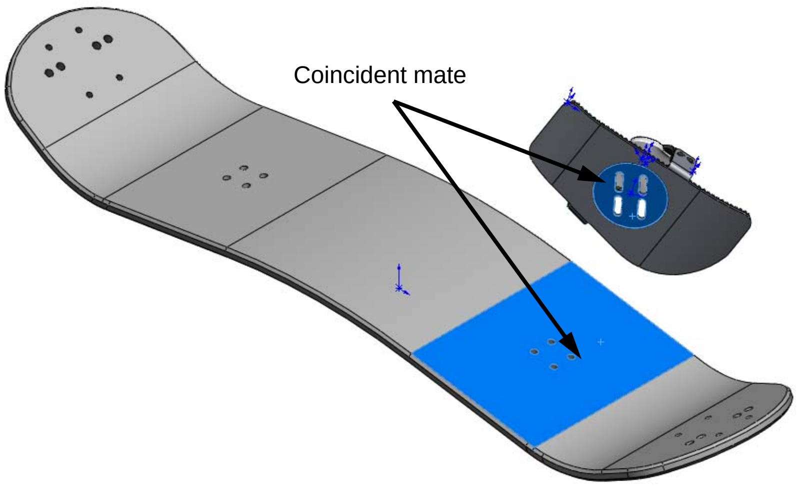
3 **Save** the assembly as `Mountainboard.sldasm`. This will be our top level assembly.

4 Add the Binding assembly. Click **Insert Components**  and then **Browse**. Locate the Binding assembly in the Mountainboard\Binding folder and click **Open**.

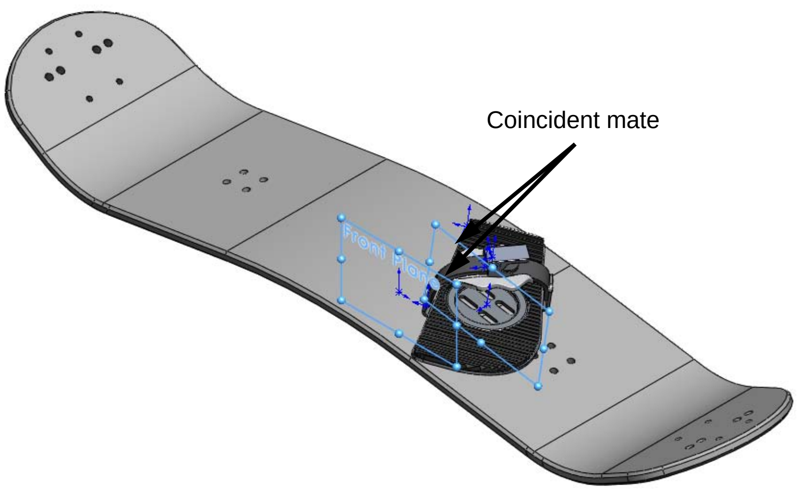
5 Drop the Binding assembly into the Mountainboard assembly. There are no **SmartMates**, so you can place the Binding anywhere.

6 We will create all the mates between the Deck and the Binding Anchor. This is consistent with the way the actual binding works.

7 Add a **Coincident** mate between the bottom of the Binding Anchor and the planar face where the binding will rest.

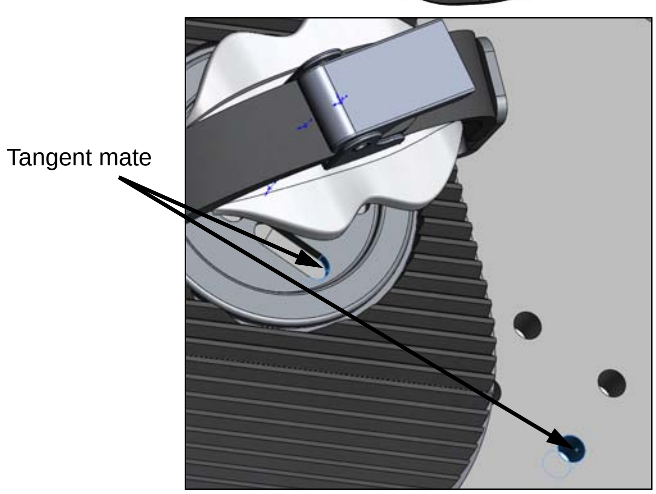


8 Add a **Coincident** mate between the Front plane of the Deck and the Right plane of the Binding Anchor. Reverse the Mate Alignment if necessary to get the Binding to point in the correct direction.

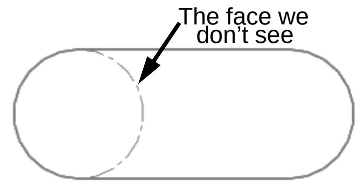


9 Add a **Tangent** mate between the end surface of one of the racetrack shaped holes in the Binding Anchor and the appropriate mounting hole in the Deck.

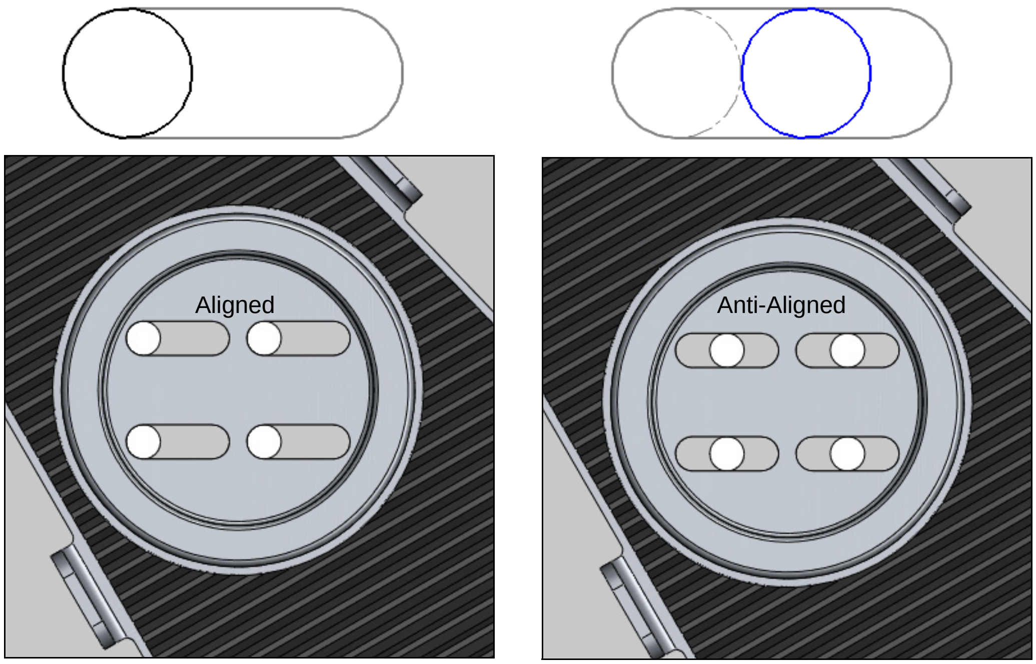
Switch between **Aligned** and **Anti-Aligned** to center the slot over the hole.



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:

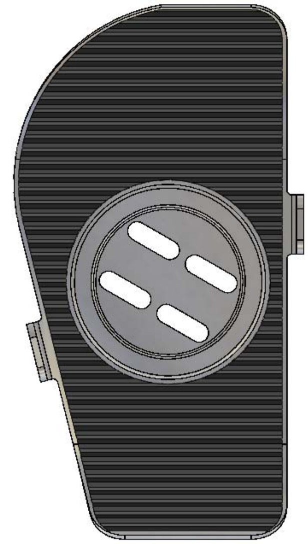
- Change the angle in the Binding assembly
- Make the Binding assembly Flexible in the Mountainboard assembly

For the first binding, we will change the angle in the Binding assembly.

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 sub-assemblies 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 sub-assembly 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

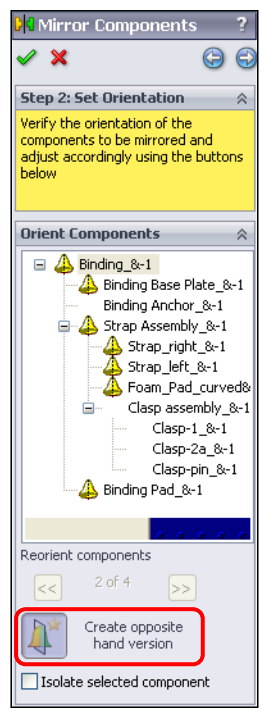
1 Click **Insert, Mirror Components** from the menu.

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.

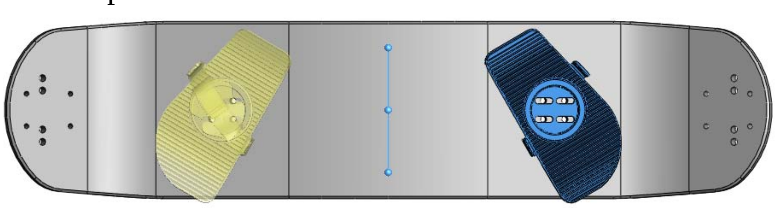


6 Select everything that gets mirrored.

Select each of the following components in the **Orient Components** box and then click **Create opposite hand version**:

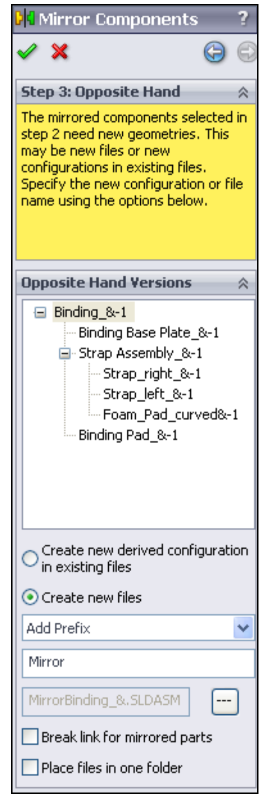
- Binding
- Binding Base Plate
- Strap Assembly
- Strap_right
- Strap_left
- Foam_Pad
- Binding Pad

7 As the components are selected we can see a preview to determine if the parts are correct.



8 Click **Next** .

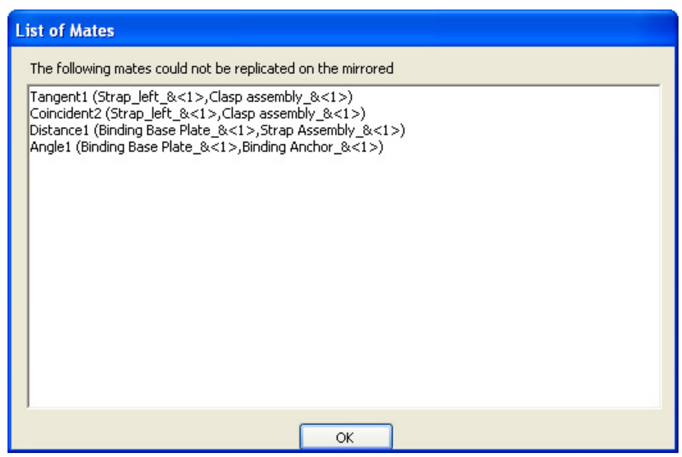
9 The mirrored components are new parts that must be named. The default is to add Mirror as a prefix to the existing file names so that the mirrored assembly of Binding will be MirrorBinding.



10 Click .

11 Some of the mates could not be created automatically. We will have to fix this manually.

Click **OK**.



12 The mirrored binding has been created.

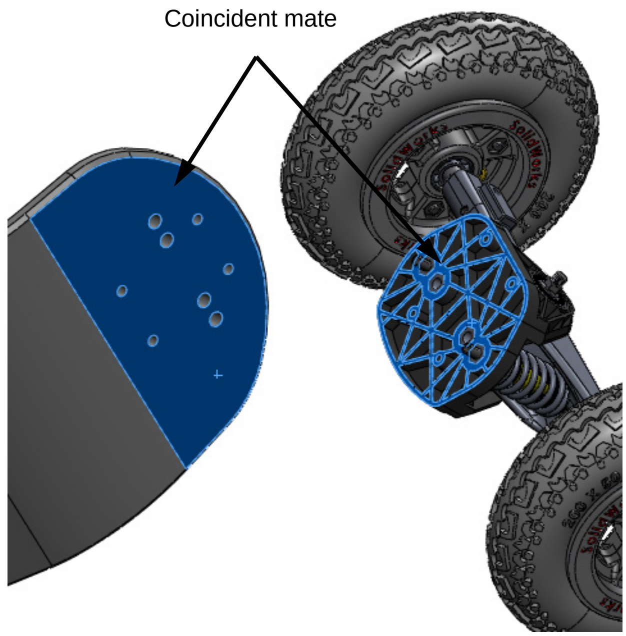
13 Try to move the new binding. It is not mated to the Deck, so we will have to do it manually.

14 Add the three mates between the Binding Anchor and the Deck that were used to mate the original Binding.

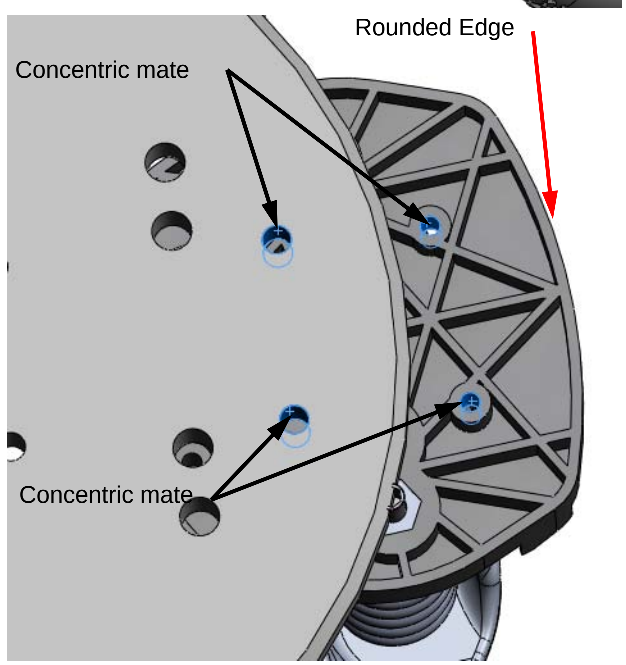
Task 14 — Add the wheel assemblies

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.

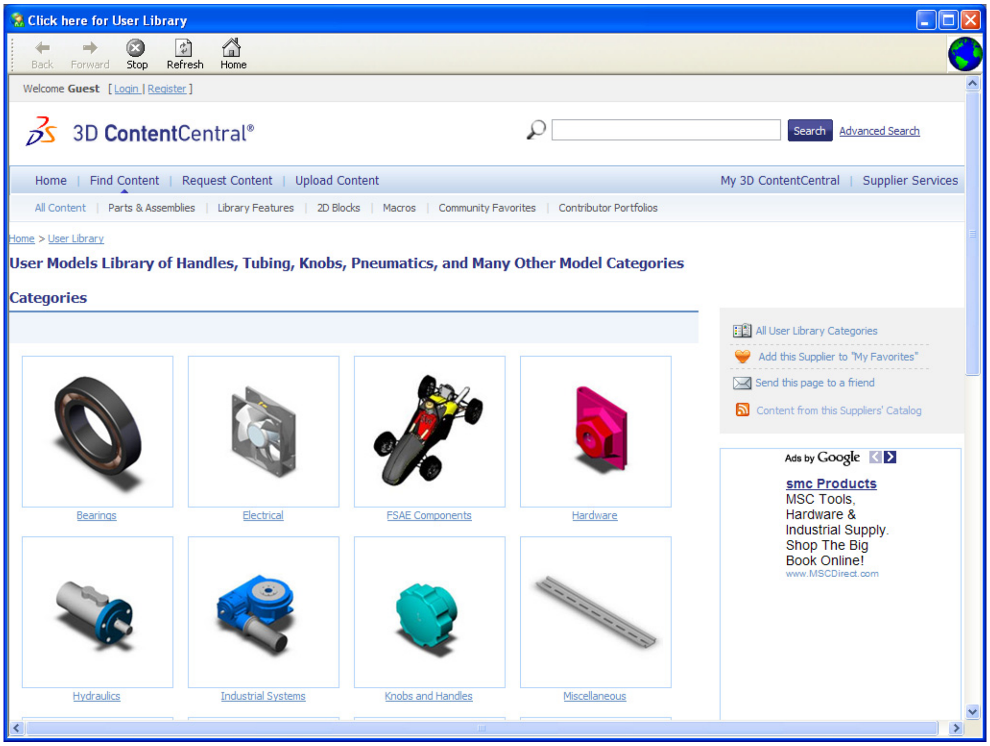
We can access 3D ContentCentral through the Design Library in the Task Pane. You must have internet access to use 3D ContentCentral.

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.

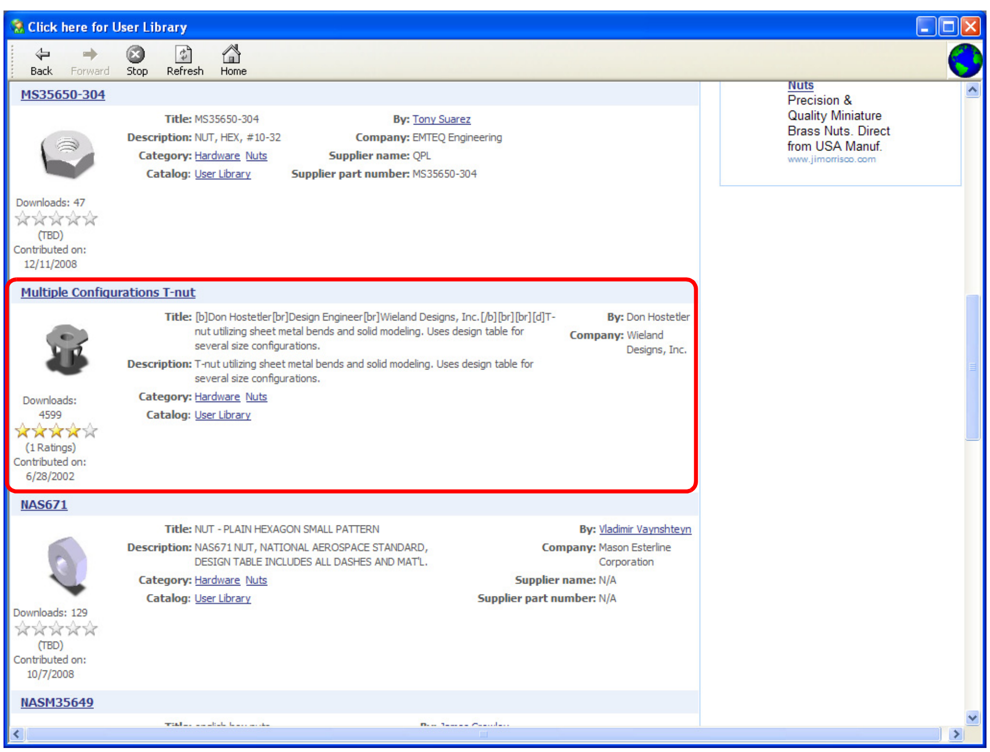
The first time you logon to 3D ContentCentral you will be asked to provide an email address and accept the licensing agreement.

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.

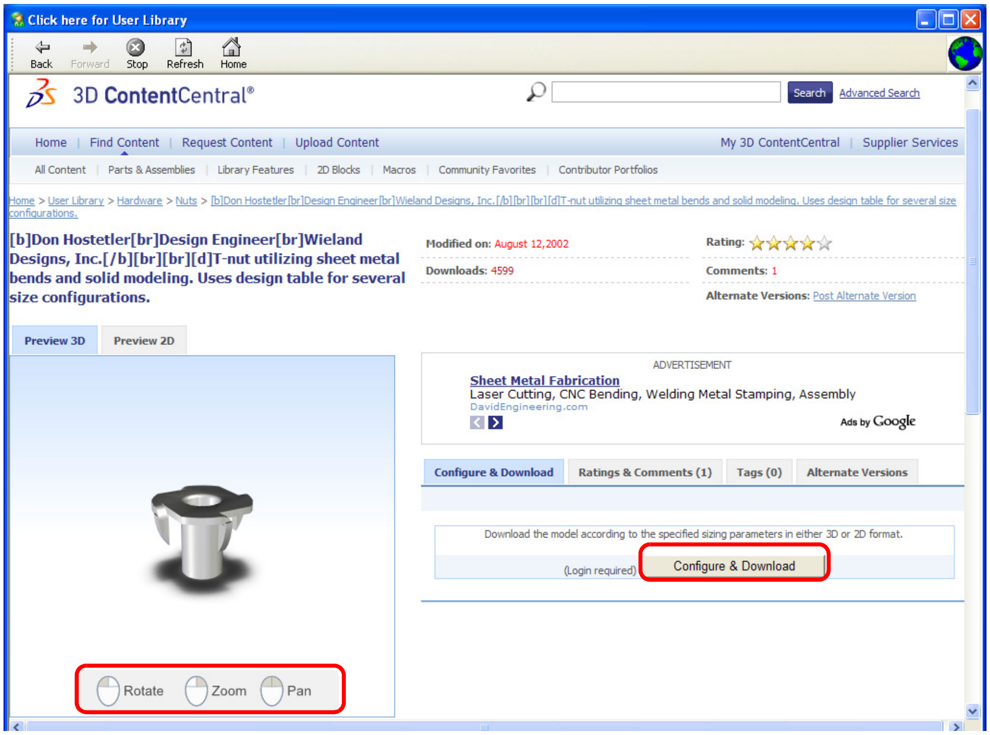


Note: 3D ContentCentral is continually being expanded so you may have to search the listing for this specific T-nut.

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.

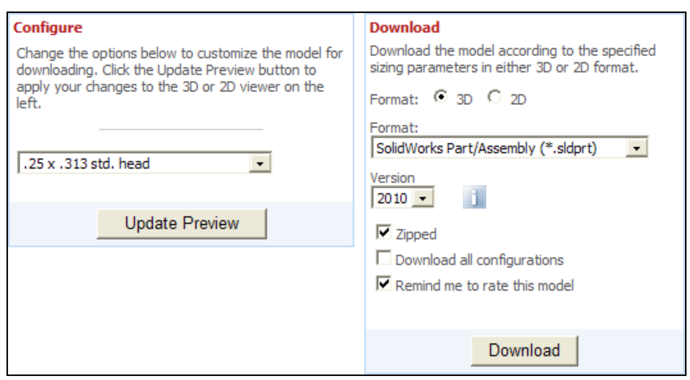


6 Select **Configure & Download**.

7 Select **.25 x .313 std. head** for the size.

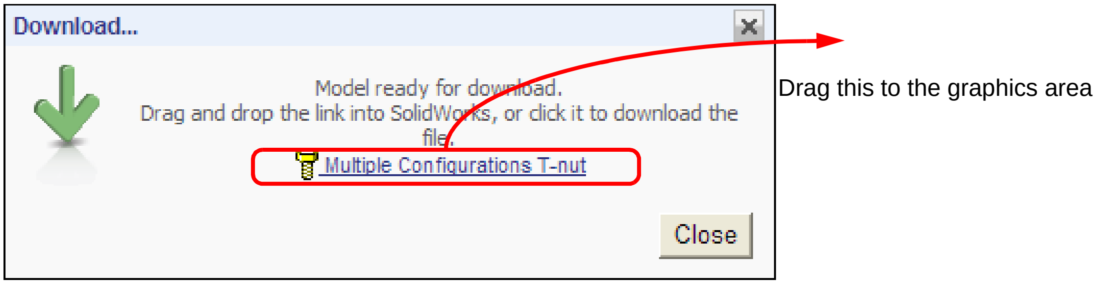
8 Select **SolidWorks Part/ Assembly** for the format and **2010** for the Version.

9 Clear the options to download as a Zipped file and to download all configurations.



10 Click **Download**.

11 Drag the Toolbox icon to the SolidWorks graphics area.

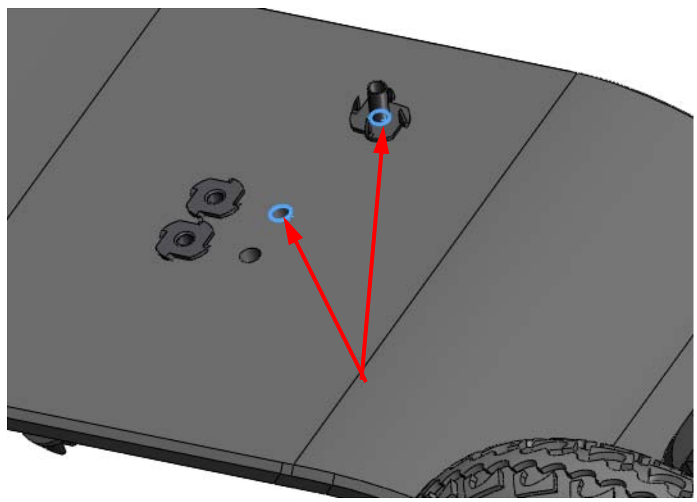


12 **Save** the part to the Mountainboard\Hardware folder.

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.

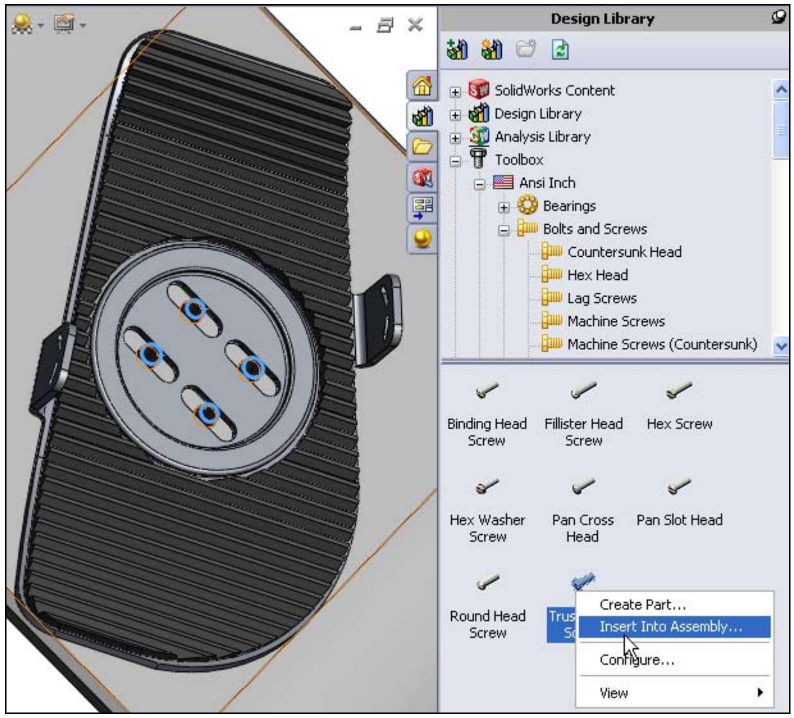


16 Reorient the model so you can see the tops of the four T-nuts.

Note: It will be easier to insert the screws in the next step if you hide the Binding Strap assembly.

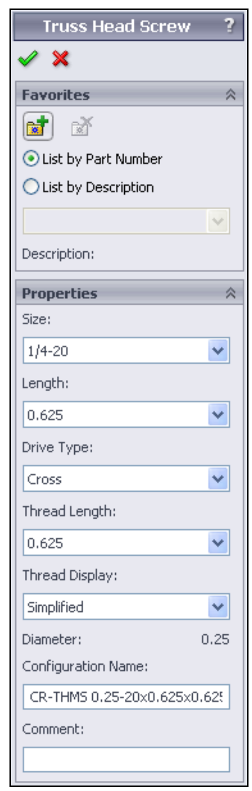
17 Click the pushpin in the Design Library to keep it open. Expand Toolbox, then Ansi Inch, then Bolts and Screws by clicking the plus sign next to each. Select Machine Screws.

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**



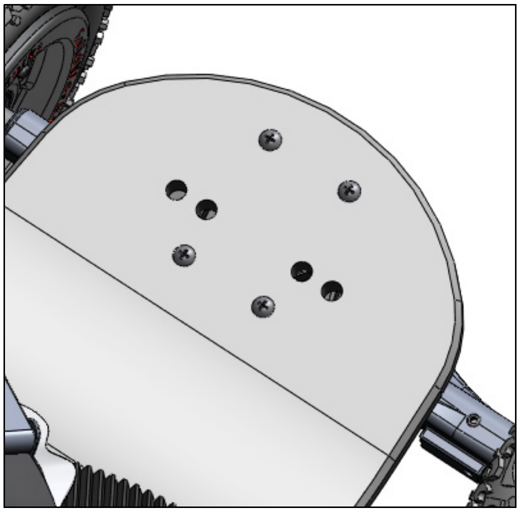
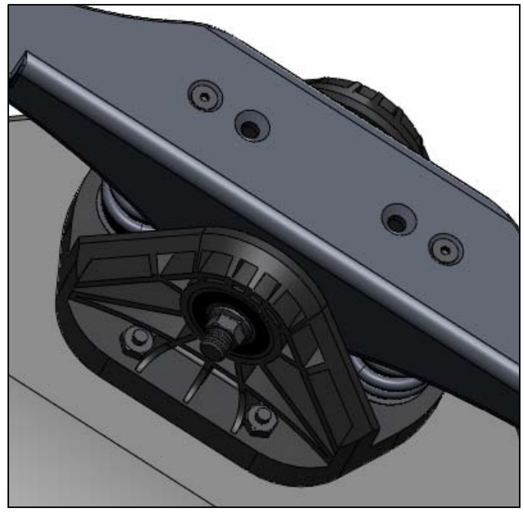
Click .

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

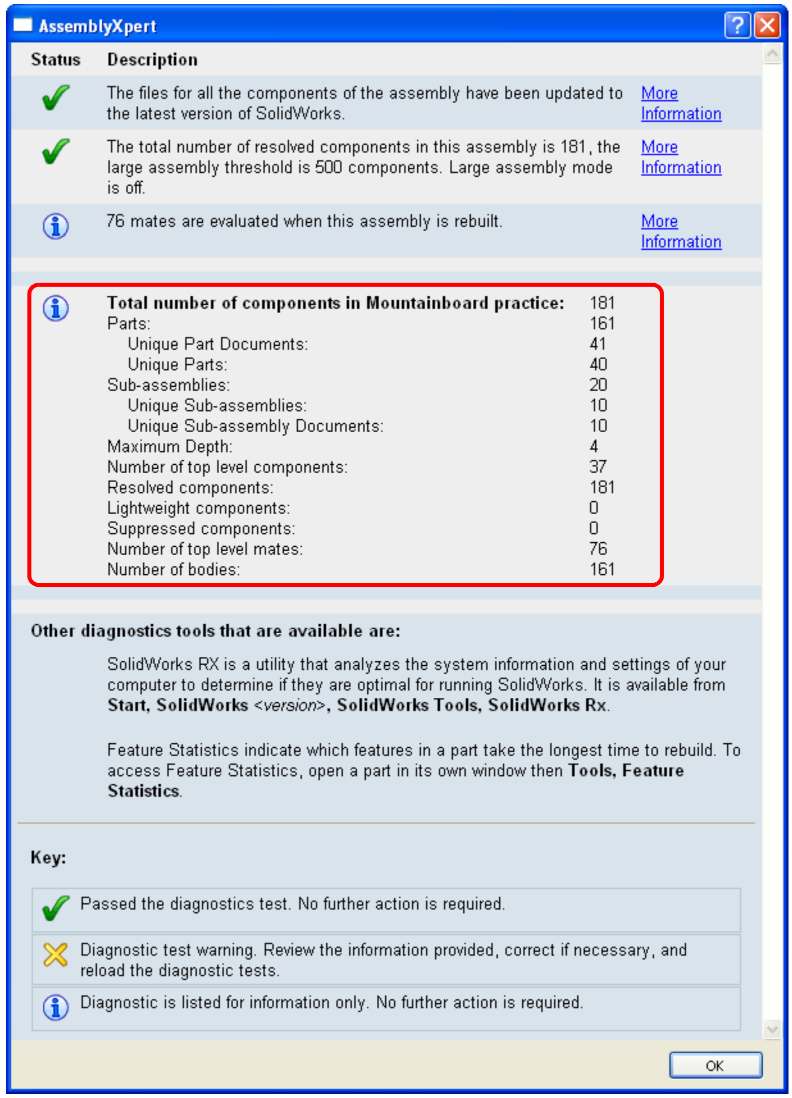
1 Examine the size of the Mountainboard assembly. Click **Tool, AssemblyXpert** from the menu.

2 **Assembly Statistics** show us that there are 181 total components in our assembly. This is the total of all parts and sub-assemblies

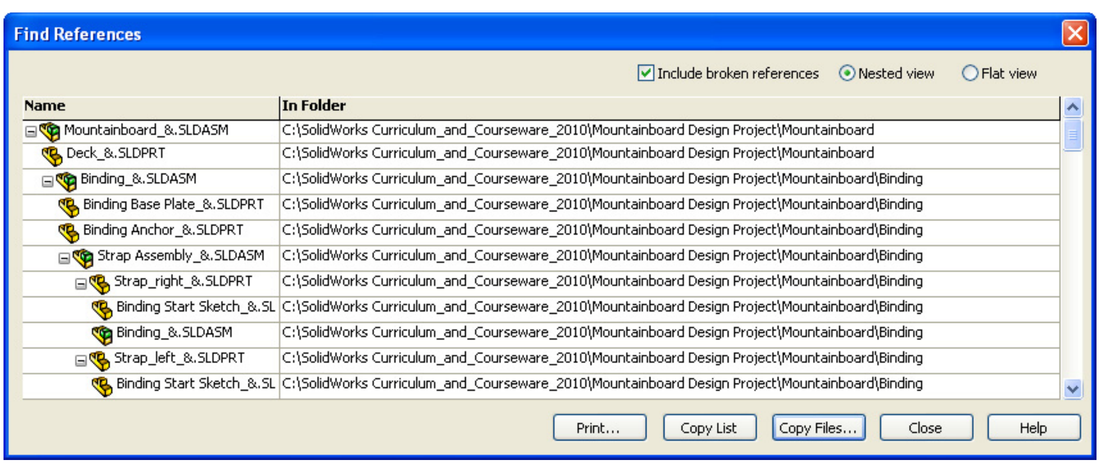
3 There are 161 parts which means that there must be 181 - 161 = 20 sub-assemblies. This is shown in the fifth line.

4 There are 41 unique parts. The difference between this and total number of parts (181) 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.



7. Examine the list. All the files should be in the `SolidWorks Curriculum and Courseware_2010\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** .

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.

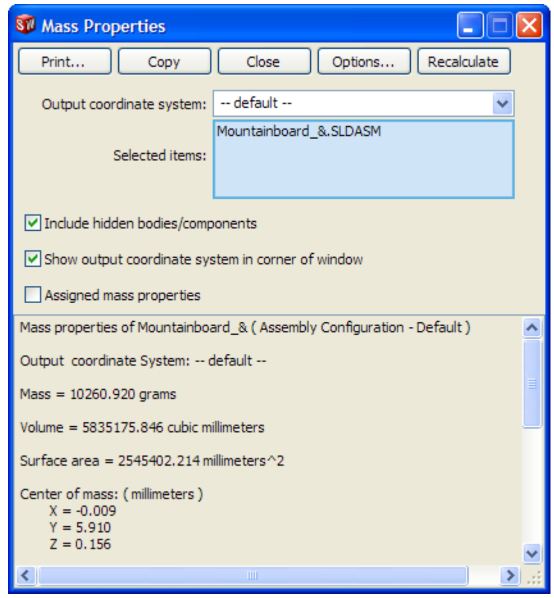
3 After editing the material, return to the **Edit Assembly** mode by selecting the part and clicking the **Edit Component** tool.

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 sub-assemblies, 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 10,260.920 grams or 10.26 kg which meets our original design intent.



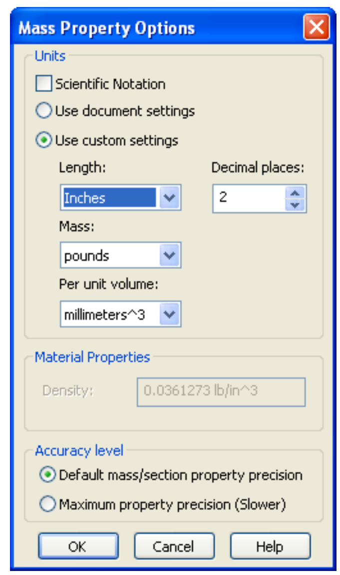
6 To check the mass in other units, click **Options**.

Select **Use custom settings** and choose **Inches** and **Pounds** from the lists.

Click **OK**.

The weight of the finished mountainboard is 22.621 pounds.

Click **Close**.



Active Learning Exercises, Part 3 — Create an Exploded View

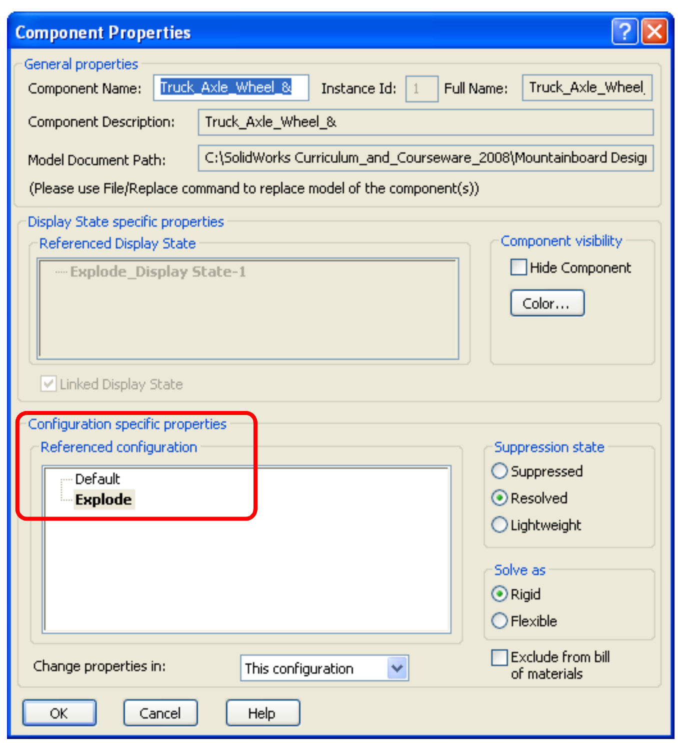
Task 3 — Create an exploded view

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.

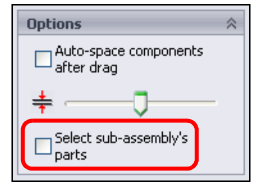
2 Change the sub-assemblies for the front Truck_Axle_Wheel and the Binding to their Explode configurations. In the FeatureManager design tree, right-click the assembly instance of the Truck_Axle_Wheel that is positioned at the front of the Mountainboard and select **Properties**. Select the Exploded configuration and then **OK**.



3 Repeat this procedure for the Binding.

4 Insert an Exploded view into the Mountainboard assembly. Click **Insert, Exploded View** from the menu.

5 Make sure that **Select sub-assembly's parts** is cleared.



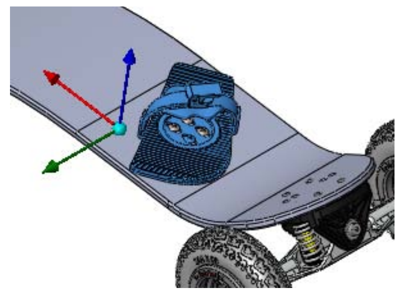
6 We only need to move two sub-assemblies, the Binding and the front Truck_Axle_Wheel assembly. Select the Binding assembly

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.

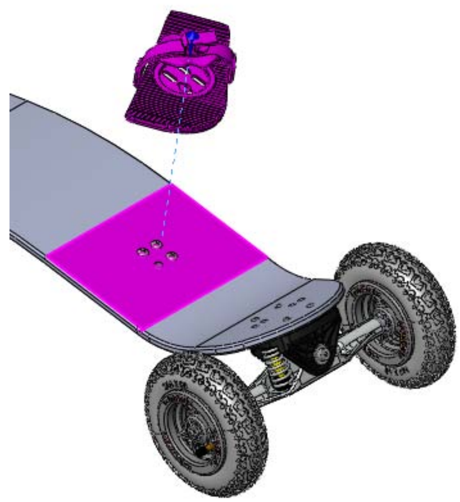
Aligning The Move Triad - Review

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.

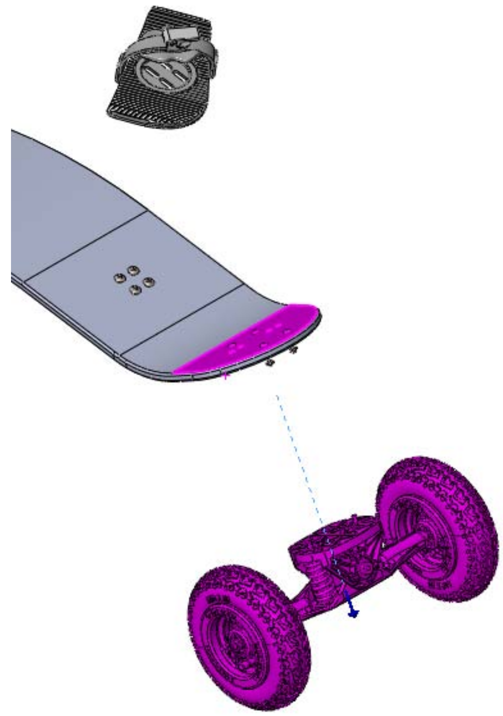


2 Use the triad to drag the Binding away from the Deck. The exact position is very subjective, but we want to make sure that it is clear that the Binding has moved away from the Deck. Click anywhere in the graphics area to complete the step.



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.

4 Use the Triad to drag the sub-assembly away from the `Deck`. We need to move this sub-assembly further from the `Deck` than the `Binding` because we will also explode this sub-assembly. When we explode the sub-assembly, 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

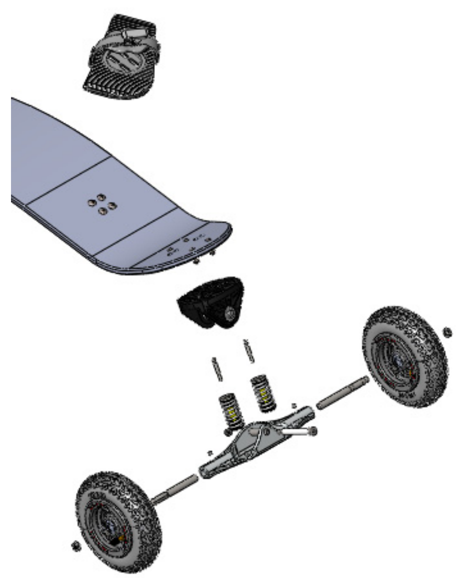
When a sub-assembly has had exploded steps created, they can be re-used in a higher level assembly. This can save a considerable amount of work.

To re-use a sub-assembly's explode steps, while creating explode steps, select the sub-assembly and click **Re-use Sub-assembly Explode**.

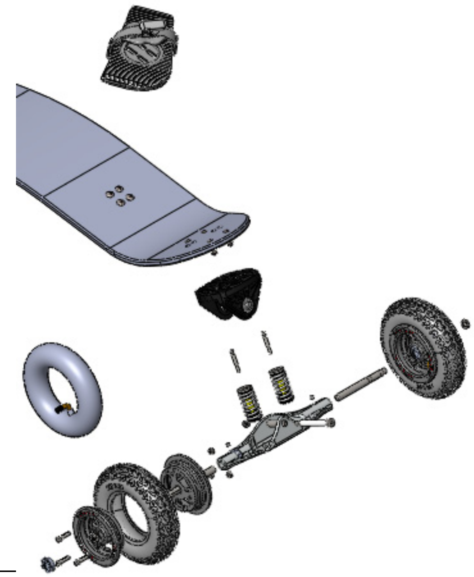
5 Select the front `Truck_Axle_Wheel` assembly.

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.

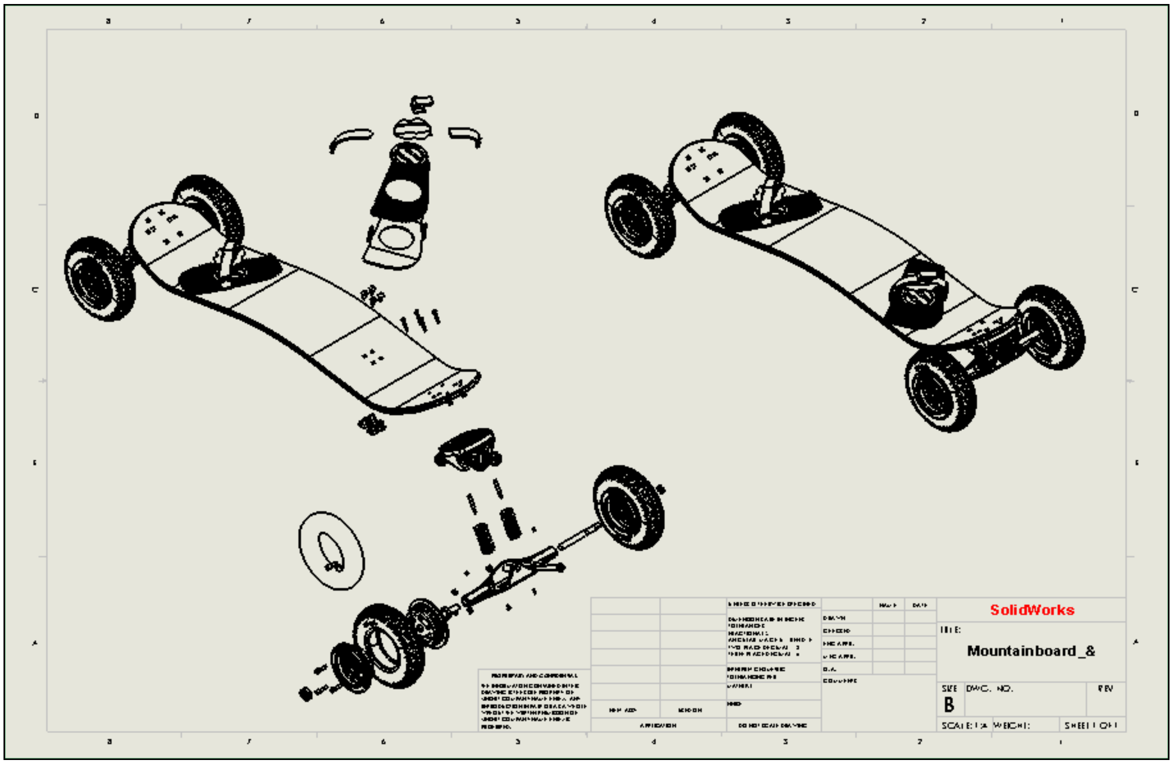
9 Complete the exploded view of the Mountainboard by exploding the front Binding and the fasteners that hold the front Binding and Truck.

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?

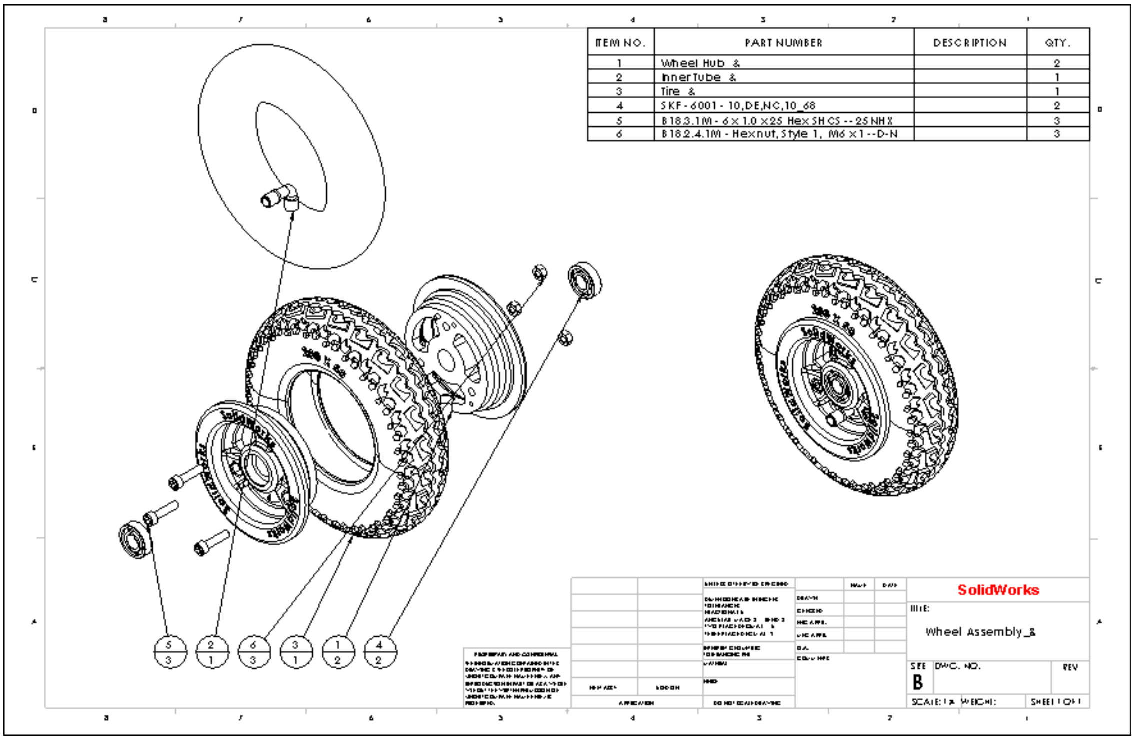
____

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.

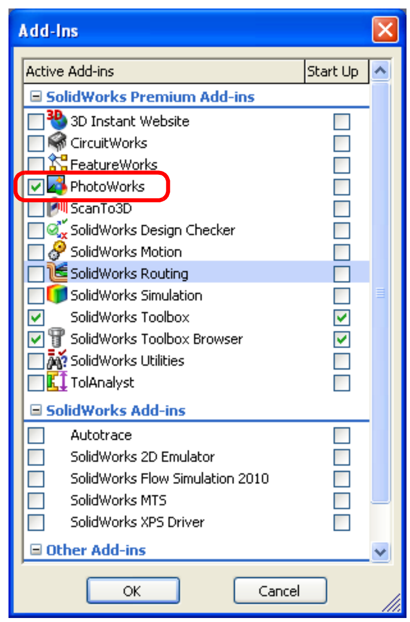
9

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 has 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 eDrawings2010 is set up and running on your computer.
- Verify that PhotoWorks is set up and running.
- Verify that Adobe Acrobat Reader is installed.

Resources for This Lesson

This lesson plan corresponds to the *eDrawings* and *PhotoWorks* 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 sub-assemblies 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 PhotoWorks 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 PhotoWorks.
- 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.
- 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 receiver 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, Mental Ray, EPS and others can be created when using PhotoWorks.

PhotoWorks 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. PhotoWorks lets you use realistic surface materials, lighting and advanced visual effects to display you models. SolidWorks MotionManager lets you capture and replay motion. Together, PhotoWorks and SolidWorks MotionManager display a model close to real life.

PhotoWorks uses advanced graphics to create photorealistic images of SolidWorks models. You can select materials 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 material.

In addition to advanced materials, PhotoWorks 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.

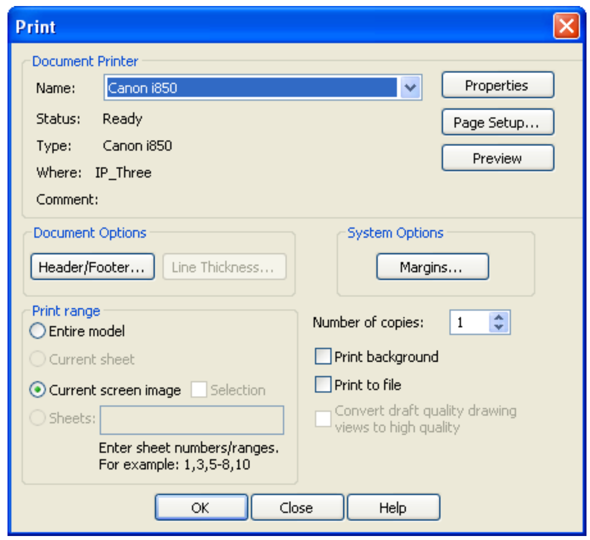
SolidWorks MotionManager generates Windows-based animations (`*.avi` files). The `*.avi` file uses a Windows-based Media Player to playback the animation. You can use these animation files for product illustrations, design reviews, and so forth.

Active Learning Experience, Part 1 — Screen Outputs

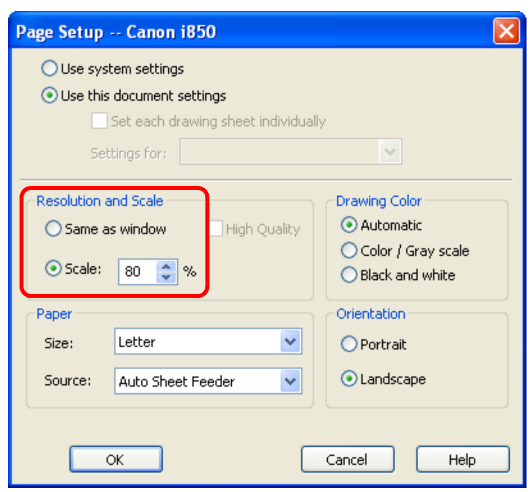
Task 1— Create printed versions of the Axle.

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.

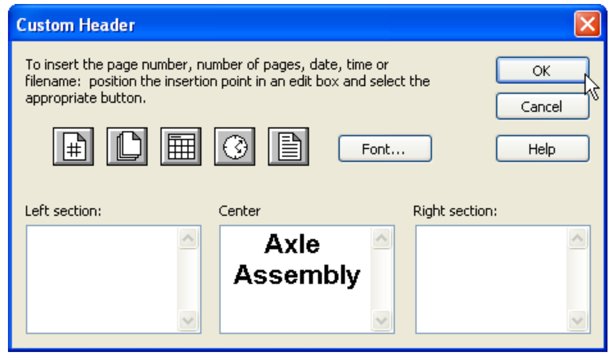
1. Open the Axle Assembly.
2. Orient the model in the Isometric view.
3. Click **View, Display, Shaded** so that we do not see the edges of model.
4. Turn off RealView graphics. Click **View, Display**, and make sure that **RealView Graphics** is not selected.
5. Click **File, Print** from the menu.
6. Select your printer.
7. Click **Page Setup**.



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**.

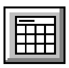


13 Type `Axle Assembly` in the **Center** box. Click **OK**.

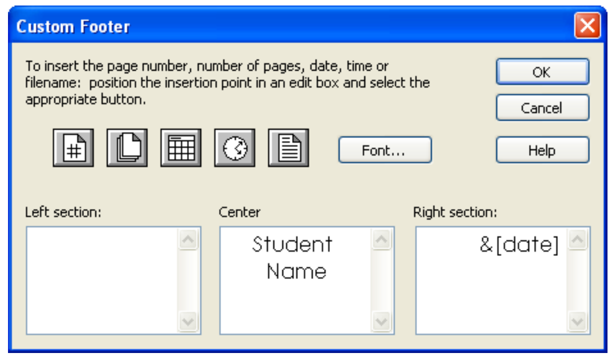


14 Click **Custom Footer**.

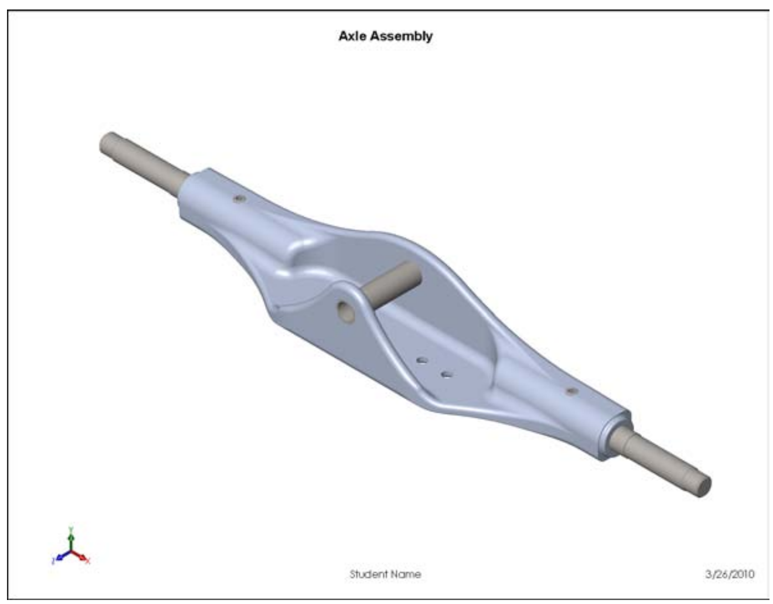
15 Type your name in the center box.

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.



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  will appear on the View toolbar. If your graphics card does not support RealView, the icon will be grayed out.

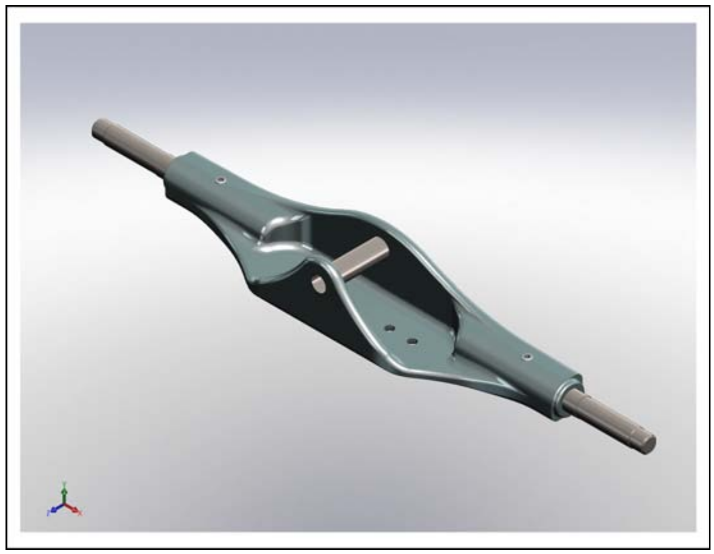
To view parts and assemblies with Realview graphics, materials must first be added to the parts.

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 material applied to the Axle to Chrome Stainless Steel.

3 Rotate the view and notice how the reflections move as if created by a scene behind the viewer



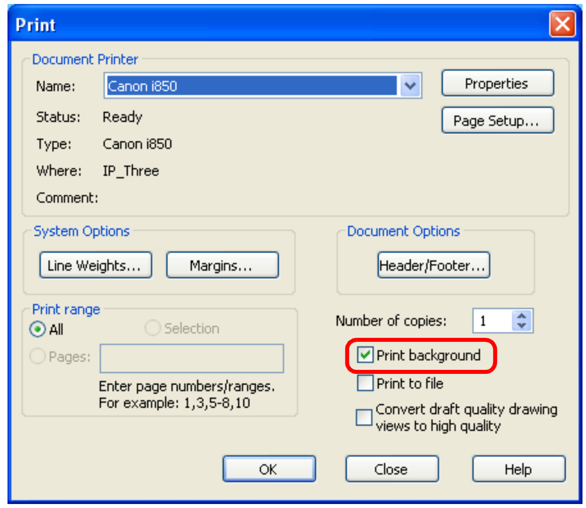
4 Change the scene. Click **Apply Scene** from the Heads-up View toolbar and select **Factory Background** from the list.

5 Click **File, Print**.

6 Select **Print background**.

7 Click **Custom Header**. Add the text "RealView Graphics" to the Right section.

8 Click **OK** three times to exit all the open dialogs and print the image.



9 Compare this image to the first image and notice the improvement.

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

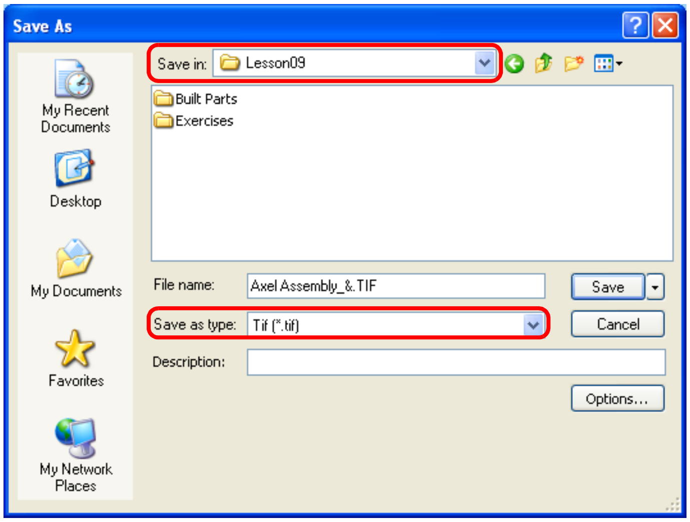
You can save a SolidWorks and PhotoWorks images to an image file format that can be used for design proposals, technical documentation and product presentations.

SolidWorks images can be saved as:

- JPEG (*.jpg)
- TIFF (*.tif)
- Adobe Illustrator (*.ai)
- Adobe PhotoShop (*.psd)

PhotoWorks Images can be rendered to the following file types:

- Windows Bitmap (*.bmp)
- TIFF (*.tif)
- TARGA (*.tga)
- Mental Ray Scene file (*.mi)
- JPEG (*.jpg)
- PostScript (*.ps)
- Encapsulated PostScript (*.eps)
- Silicon Graphics 8-bit RGBA (*.rgb)
- Portable pixmap (*.ppm)
- Utah/Wavefront color, type A (*.rla)
- Utah/Wavefront color, type B(*.rlb)
- Softimage color (*.pic)
- Alias color (*.alias)
- Abekas/Quantel, PAL (720x576) (*.qntpal)
- Abekas/Quantel, NTSC (720x486) (*.qntntsc)
- Mental images, 8-bit color (*.ct)
- High Dynamic Range (*.hdr)
- Portable Network Graphics (*.png)
- Adobe PhotoShop (*.psd)



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.

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 Lesson09 folder. This is a blank presentation with a a title slide and body slide.

11 Select the body slide.

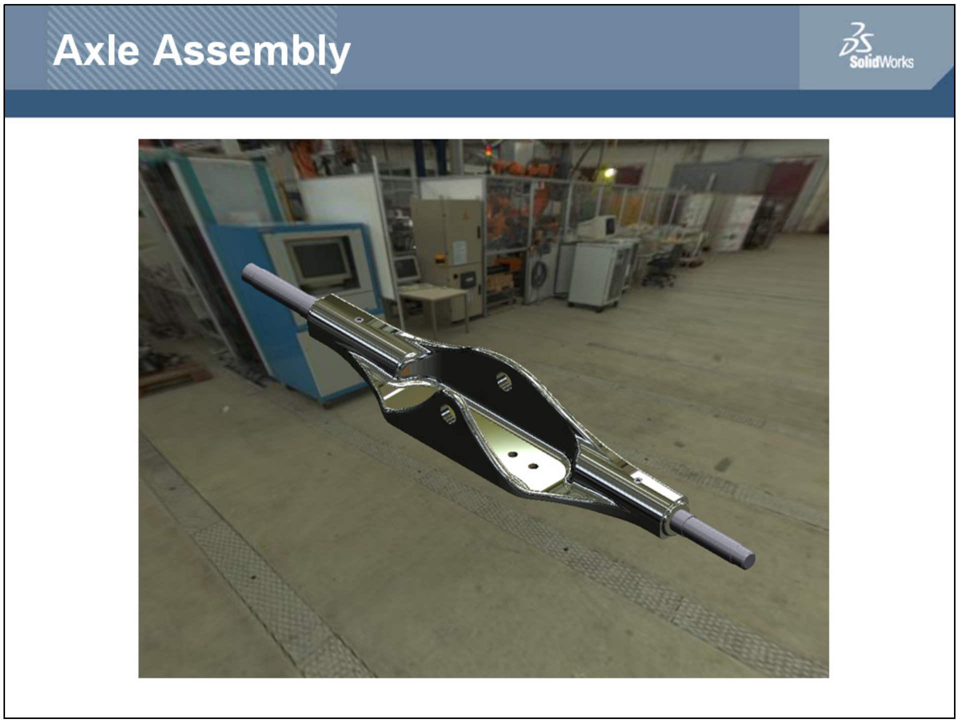
12 Click “Click to add title” and type Axle Assembly.

13 Click in the center of the slide. Click **Insert, Picture, From File** 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 **Insert, New Slide** from the menu.

17 In SolidWorks click **Image Capture** 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.

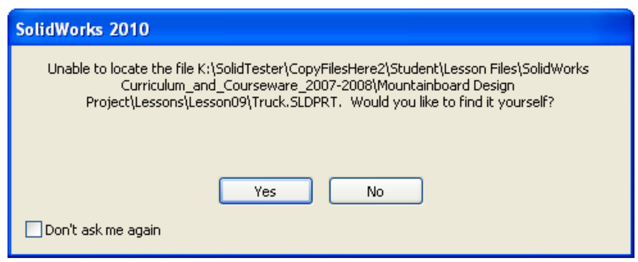
Some of the choices are:

- ❑ 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.

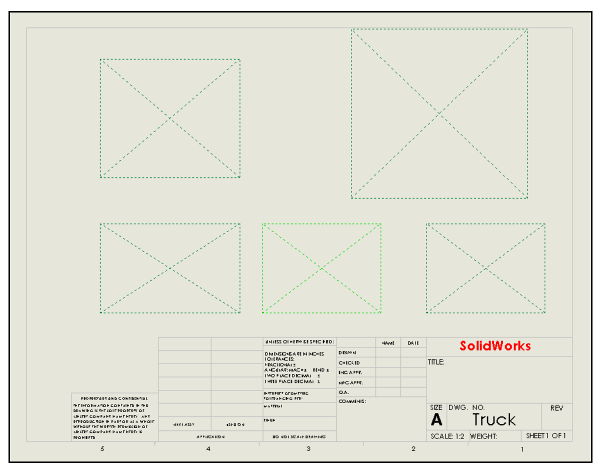
Click **No**.



3 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.

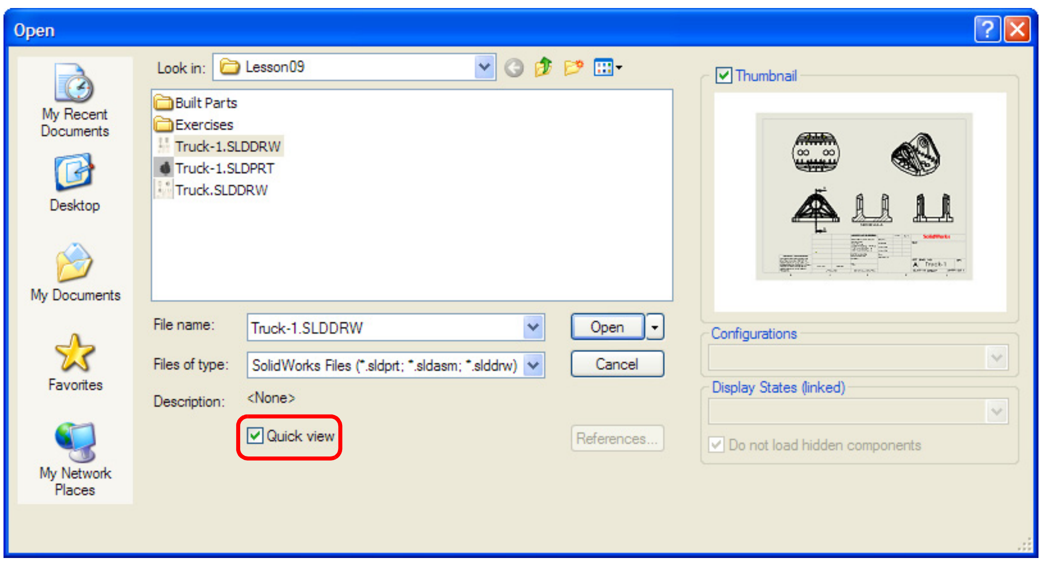
When a file is loaded as view-only, the FeatureManager design tree will be empty.

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, www.solidworks.com.

1 Click **File, Open**.
2 Locate the file `Truck.slddrw` in the `Lesson09` folder but *do not open it*.
3 Select **Quick View**.



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.

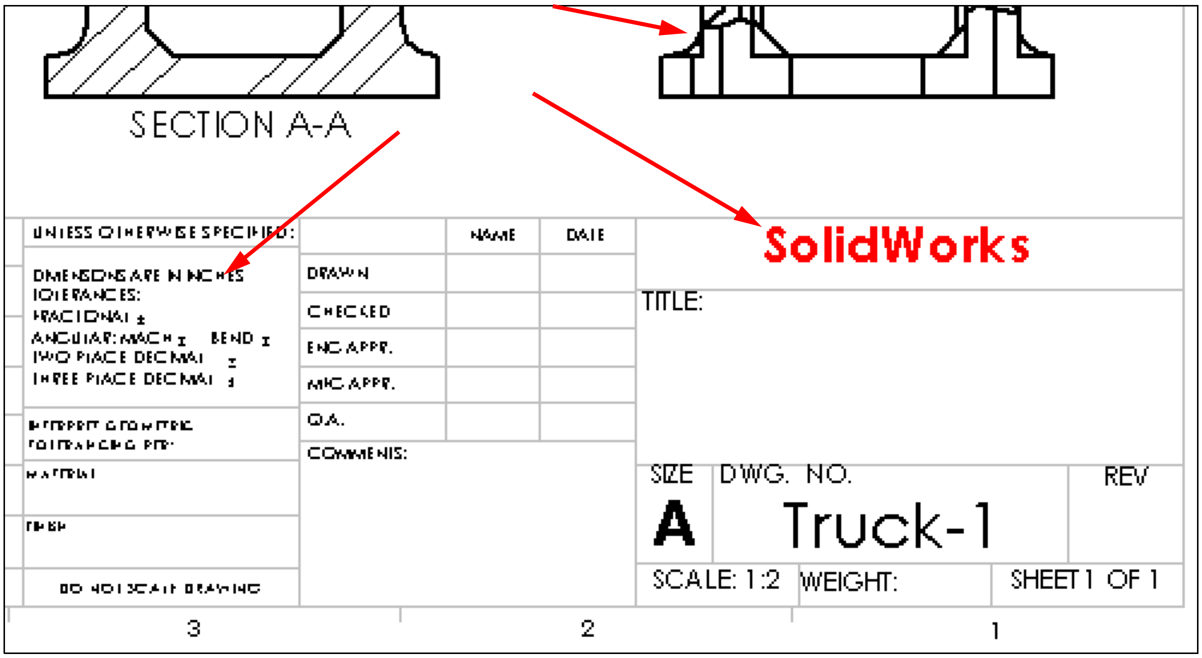
Vector Files

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.



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.

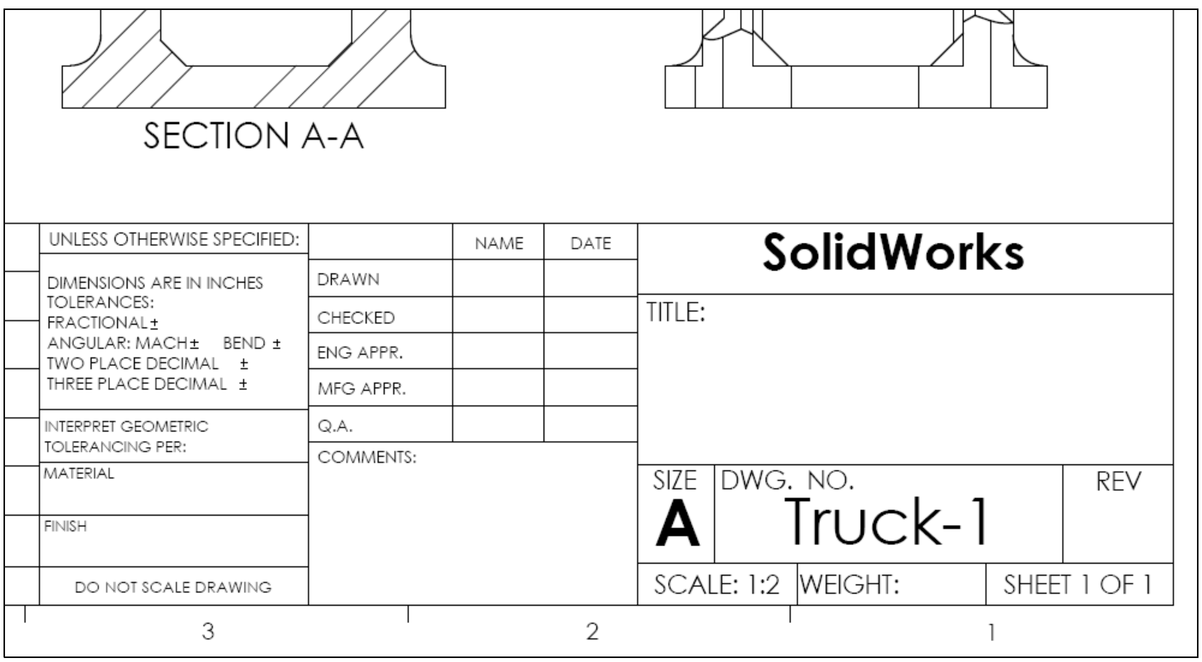
Acrobat Reader

Acrobat Reader is a free program crated by Adobe. If not already loaded on your computer, it may be downloaded from www.adobe.com.

1 Use Windows Explorer to locate the file `Truck-1.pdf` in the `Lesson09` 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 eDrawing 2010 File**.
- Click **File, Save As** and select **eDrawings (*.edrw)**.

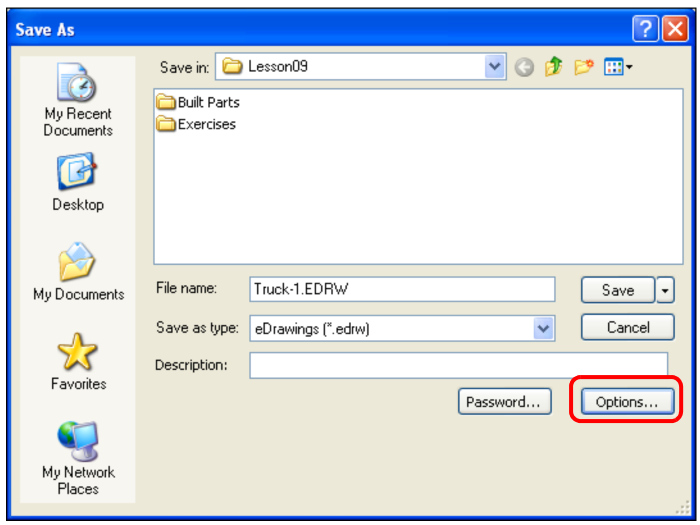
1 Click **File, Save As**.

2 Select **eDrawings (*.edrw)** from the **Save as type** list.

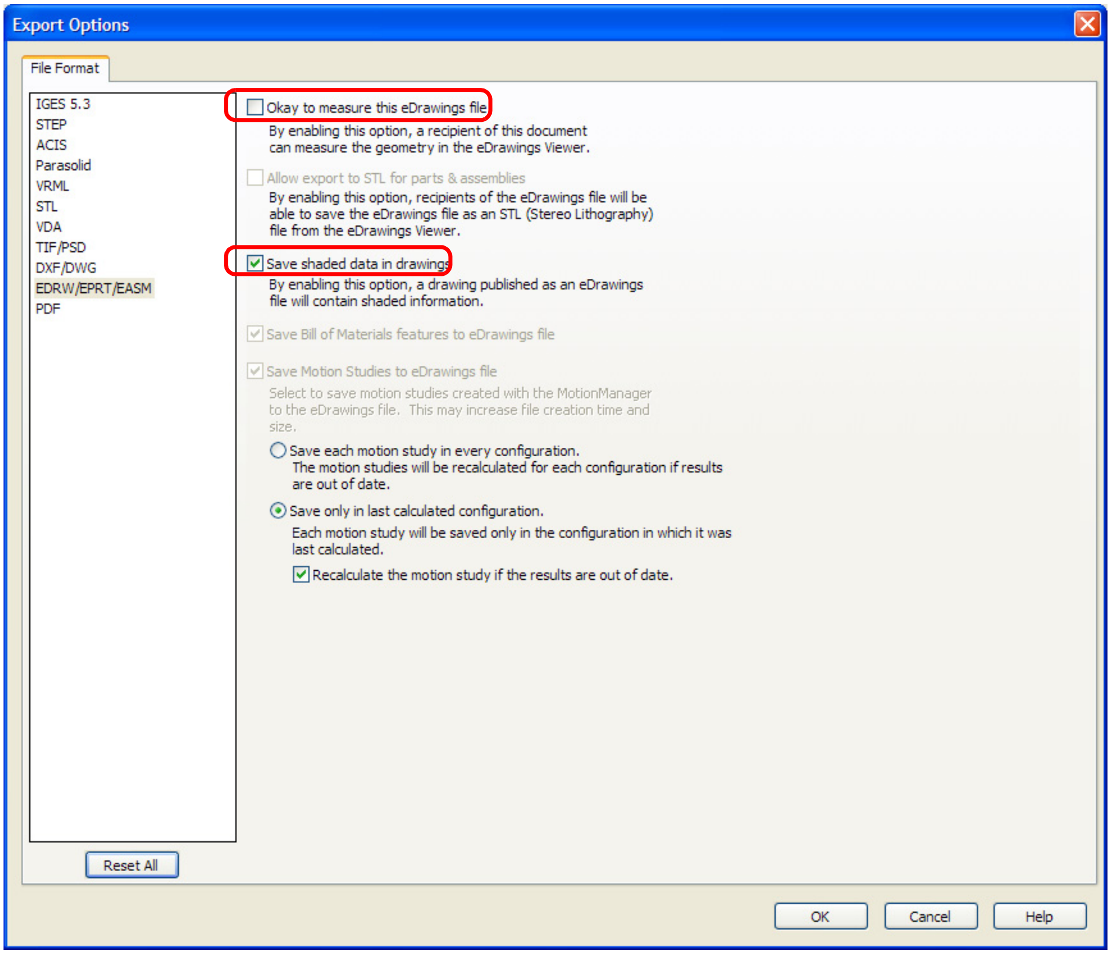
3 The default file name should be Truck-1.edrw.

4 Navigate to the Lesson09 folder.

5 Click **Options**.



6 Select **Okay to measure this eDrawings file** and **Save shaded data in drawings**.

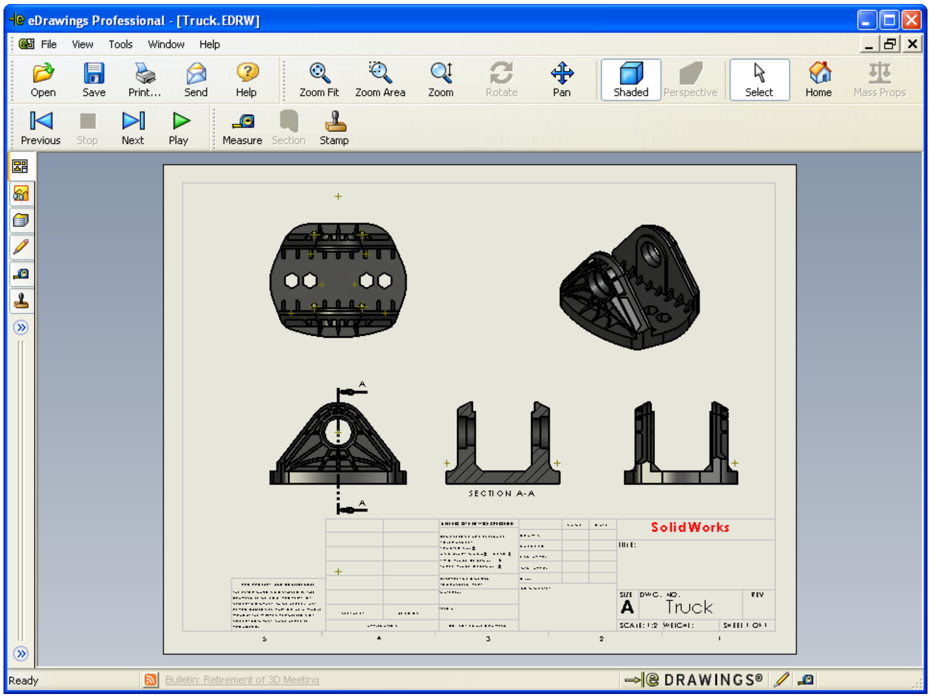


7 Click **OK** then **Save**.

The eDrawing will be created.

8 Locate the file `Truck-1.EDRW` in the `Lesson09` 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** 

.

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** 

.

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** 

.

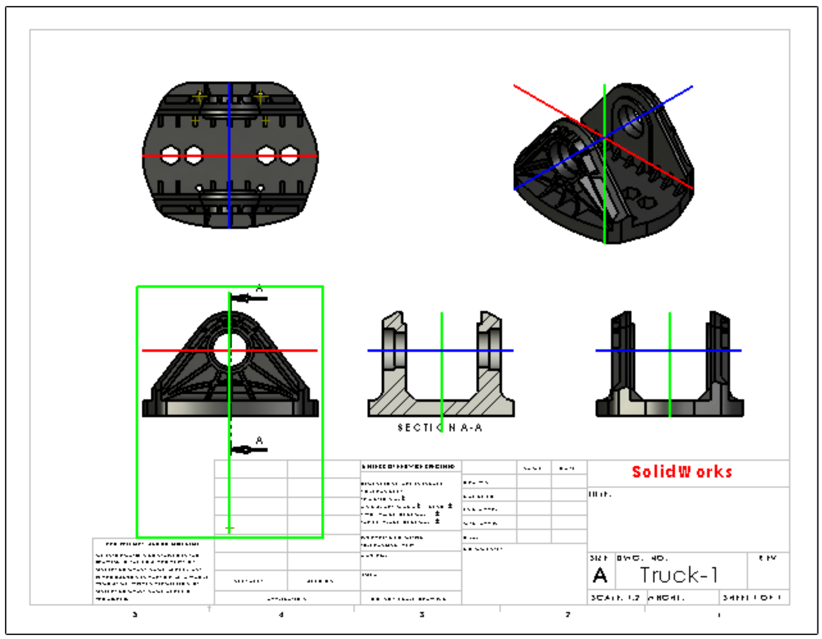
The 3D Pointer

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** 

. 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** 

to turn off the 3D pointer.

Sending eDrawings

eDrawings can be sent as email in various forms.

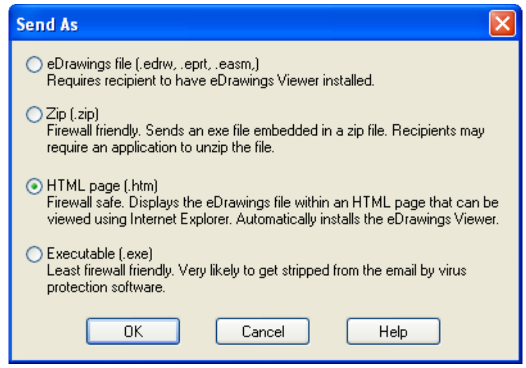
If the receiving party has the eDrawing viewer, the eDrawing can be sent as an eDrawing file (.edrw [drawing], .eprt [part], .easm [assembly]).

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).

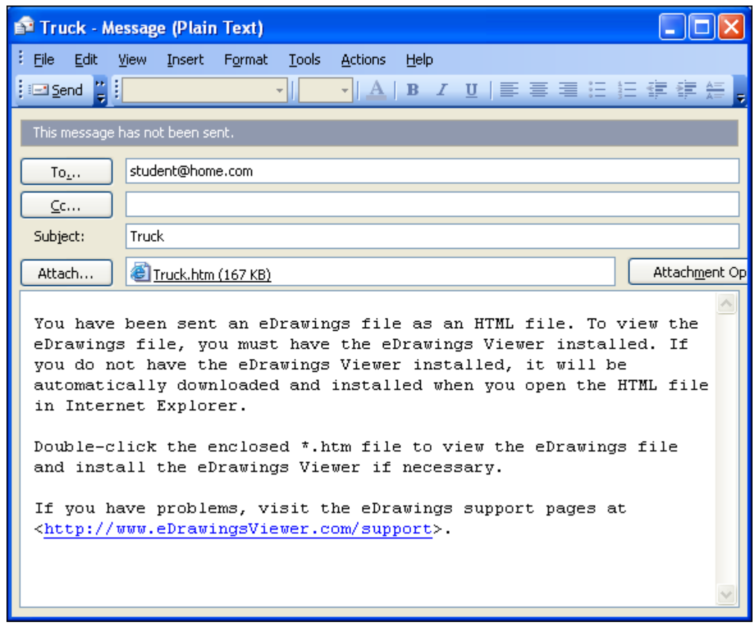
Task 4 — Send the eDrawing as email

1. Click **File, Send** from the menu or just click **Send**  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`.



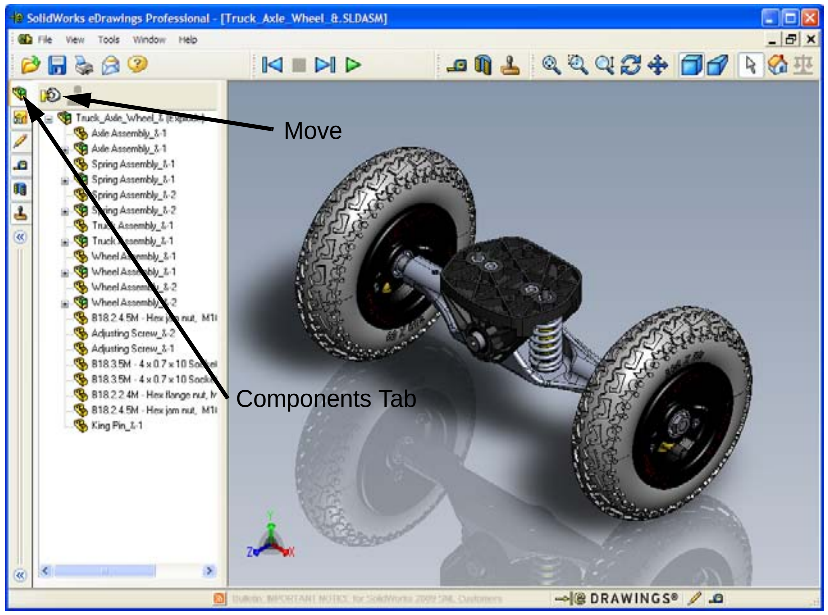
eDrawings As A Viewer

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**.



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.

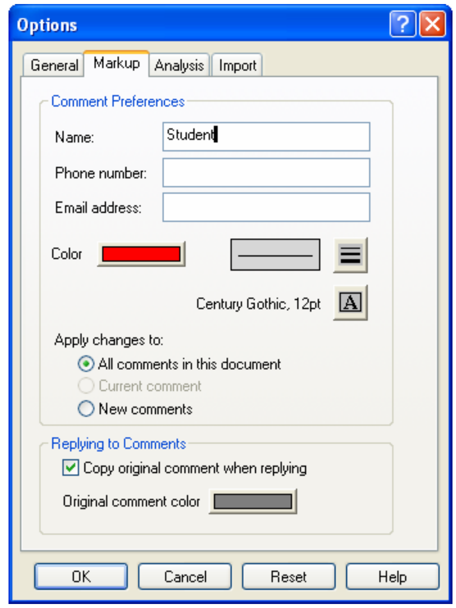
Markup allows the reviewers to add comments with arrows or clouds. The markup may be saved as a separate file so that only the markup comments need be sent back to the originator rather than the entire eFile.

Markup Options

Markup options allow you to establish the name of each reviewer and the color that will be used to display the comments.

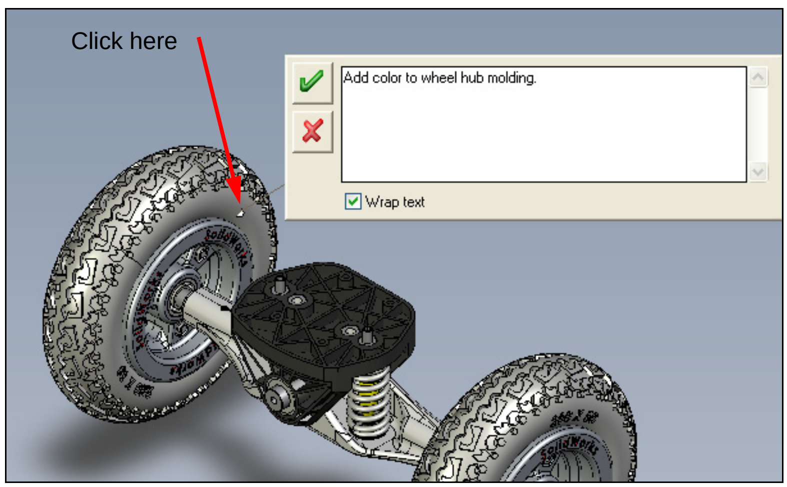
1. Click **Tools, Options**. Select the **Markup** tab.
2. Type your name, or `Student`, for **Name**.
3. Select the color to be used for your comments by clicking in the color box.

   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.

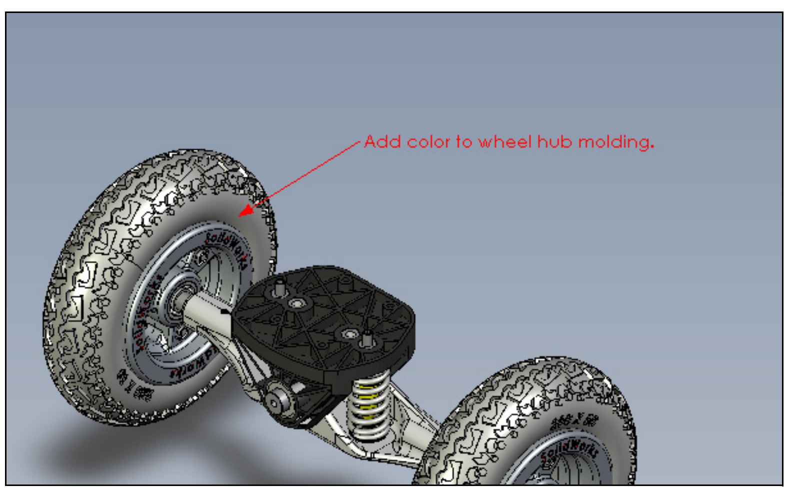


5. Select the **Markup** tab , then click **Text with Leader** .

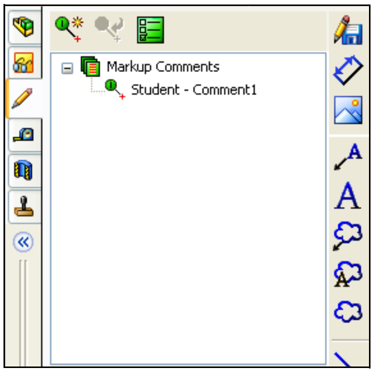
6. Click on one of the `Wheel Hub` parts near the SolidWorks text, then move the cursor to the position where you would like to drop the text and click.
7. Type "Add color to wheel hub molding".



8. Click .

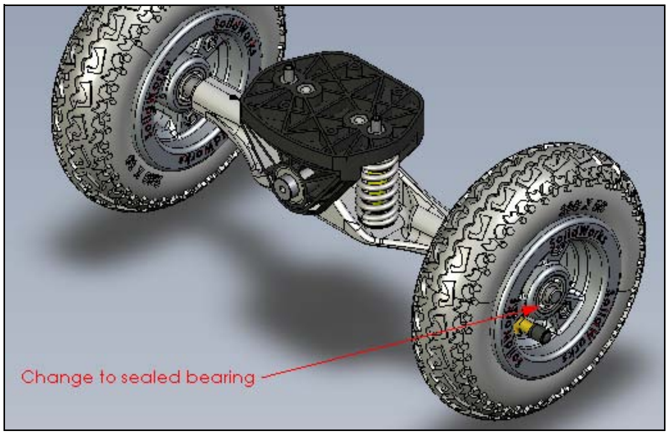


9 Examine the **Markup** tab in the eDrawingManager. All **Markup** comments will be listed on this tab.



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.**

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 PhotoWorks Rendering

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 PhotoWorks software

PhotoWorks 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. PhotoWorks can produce photorealistic images to add visual impact to presentations and documents.

Some of the key features of PhotoWorks are:

Photorealistic images directly from SolidWorks models

PhotoWorks interacts with the 3D geometry created with the SolidWorks software. All changes to SolidWorks models are accurately represented in the PhotoWorks images.

Fully integrated into SolidWorks

PhotoWorks software is supplied as a SolidWorks dynamic link library (`*.dll`) add-in. You access all the controls for the PhotoWorks rendering interface from PhotoWorks items on the main SolidWorks menu bar, the PhotoWorks toolbar or the Task Pane. The menu bar is displayed whenever a SolidWorks part or assembly document is open.

Appearances

Appearances are used in both SolidWorks and PhotoWorks to specify model surface properties such as color, texture, reflectance and transparency. PhotoWorks is supplied with numerous predefined appearances. Others can be downloaded from various web sites, created using image creation software, or by scanning. Additionally, appearances added in SolidWorks can be used directly in PhotoWorks to avoid duplicating effort.

Lighting

Lights may be added in the same way a photographer adds lights when taking photographs. PhotoWorks uses the same lights as SolidWorks but also contains numerous predefined lighting schemes to simplify and speed up the rendering process. PhotoWorks has the sophistication to trace light rays and reflections.

Scenes

Each SolidWorks model is associated with a PhotoWorks scene, for which you can specify properties such as rooms, environments and backgrounds. Scenes help to put products in context.

Decals

Images, such as company logos, can be applied to models.

Output

The PhotoWorks software can output to the screen, a printer, or a graphics file.

Starting PhotoWorks

When PhotoWorks is installed, the menu and toolbar do not automatically appear as part of the SolidWorks screen. They must be turned on.

To turn on PhotoWorks:

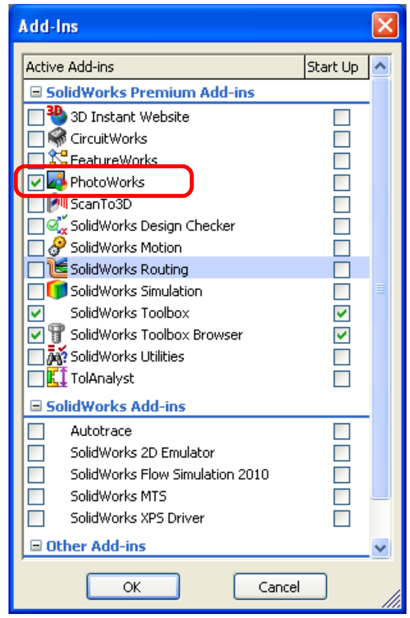
❑ From the **Tools** menu, select **Add-Ins…**, select **PhotoWorks.**

Task 1— Start PhotoWorks

1 Click **Tools, Add-Ins.**

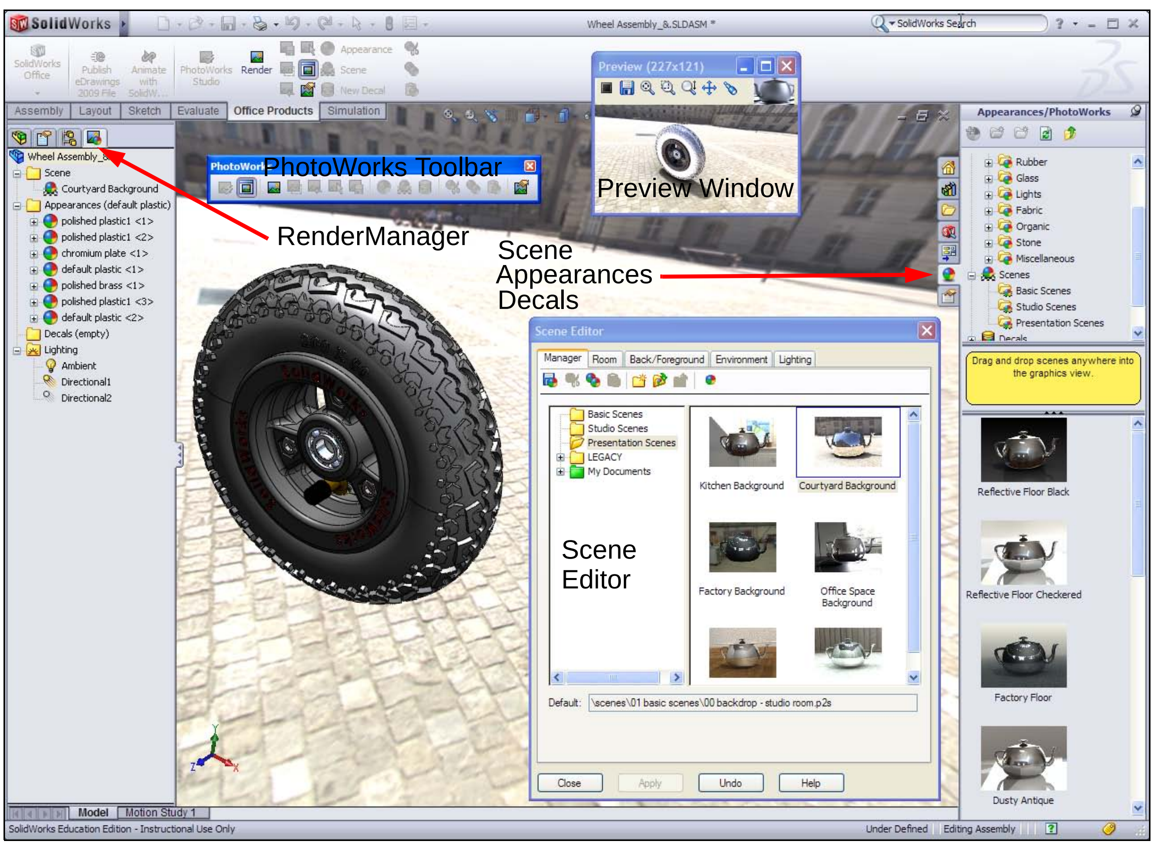
In the **Add-Ins** dialog box, select **PhotoWorks**.

Click **OK**.

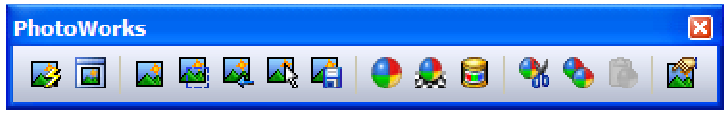


PhotoWorks User Interface

The PhotoWorks software uses the same user interface as the Solidworks software. No new interface techniques are required.

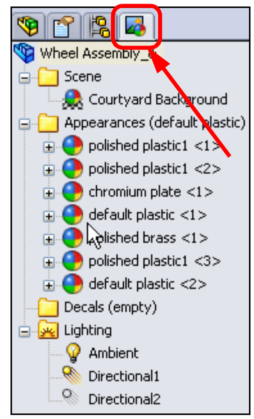


PhotoWorks Toolbar



The PhotoWorks 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 **PhotoWorks** or by using the **View, Toolbars** menu.

PhotoWorks RenderManager



PhotoWorks creates an additional tab  on the FeatureManager design tree. This tab is called the RenderManager. The RenderManager provides an outline view of the PhotoWorks materials, decals, and scenery associated with the active SolidWorks part or assembly.

The RenderManager indicates which items of geometry are attached to which PhotoWorks materials and decals.

The RenderManager also makes it easy to:

- Understand the way in which material and decal inheritance works.
- Select and edit materials and decals associated with the model.
- Transfer materials and decals between components, features, and faces.

Getting Help

The PhotoWorks help system is part of the standard SolidWorks help system.

To Get Help

- Select **SolidWorks Help** from the **Help** menu, then select **PhotoWorks** in the **Contents** tab.

Wizards

Although not usually listed under **Help**, wizards are designed to help you through steps that may be unfamiliar. PhotoWorks has one wizard, the Render Wizard. It helps to ensure that the necessary steps to complete a basic rendering are completed to achieve the desired result.

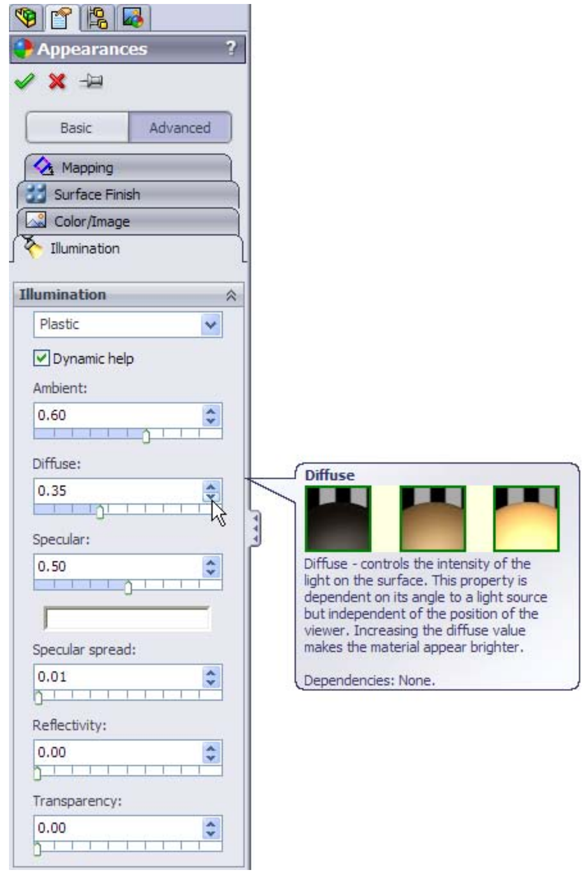
Dynamic Help

Dynamic help is provided to assist you in understanding the effects of various PhotoWorks controls. Dynamic help is enabled as a PhotoWorks System Options.

Whenever you select, or hover over, an item on the Illumination tab of either the Appearance or Decal PropertyManager, 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 **Diffuse** property, dynamic help shows the way the model will reflect light as the slider is moved.



Options

PhotoWorks has its own options dialog box. Options allow you to customize the PhotoWorks software to reflect your preferences of default settings. Options are divided into System Options, Document Properties, Advanced, Illumination and File Locations.

For a complete listing of all the settings available though the PhotoWorks Options dialog refer to the Help menu.

To set PhotoWorks Options:

- ❑ Click **PhotoWorks, Options**
- ❑ Or, on the PhotoWorks toolbar, click **Options** .

Task 2 — Set PhotoWorks Options

Before beginning a project in PhotoWorks, system options need to be set to make sure everyone sees the same results.

1 **Open** the part `Axle`.

Note: PhotoWorks options can only be set if a part or assembly document is open.

---

2 Turn off RealView. RealView must be off to access one of the settings in the following step.

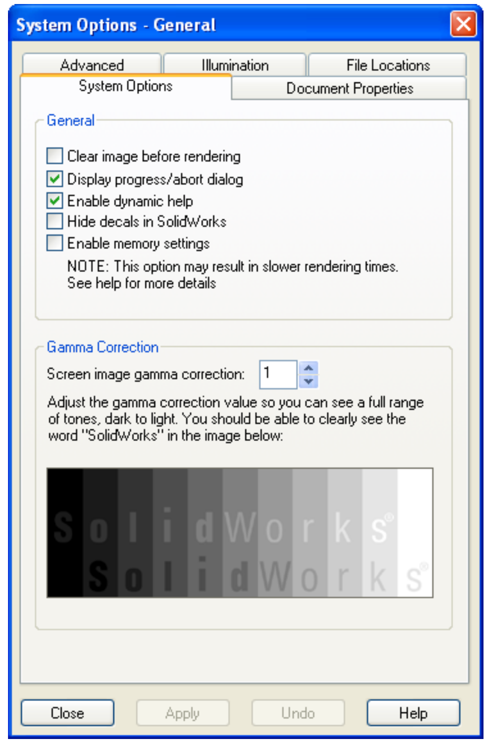
3 Set the PhotoWorks Options.

Click **Options** on the PhotoWorks toolbar.

Select the **System Options** tab and set the options as follows:

- Clear image before rendering: **Cleared**
- Display progress/abort dialog: **Selected**
- Enable dynamic help: **Selected**
- Hide decals in SolidWorks: **Cleared**
- Enable memory settings: **Cleared**
- Screen image gamma correction: **1**

Click **Apply.**



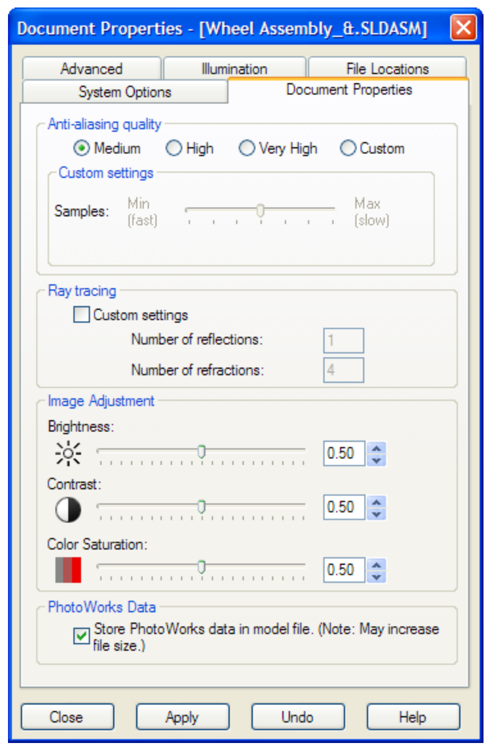
4 Set the **Document Properties**.

Select the **Document Properties** tab.

Set the **Document Properties** options as follows:

- Anti-aliasing quality: **Medium**
- Ray tracing, Custom settings: **Cleared**
- Brightness: **0.50**
- Contrast: **0.50**
- Color Saturation: **0.50**
- PhotoWorks Data: **Selected**

Click **Apply.**



5 Set the **File Locations**.

Select the **File Locations** tab.

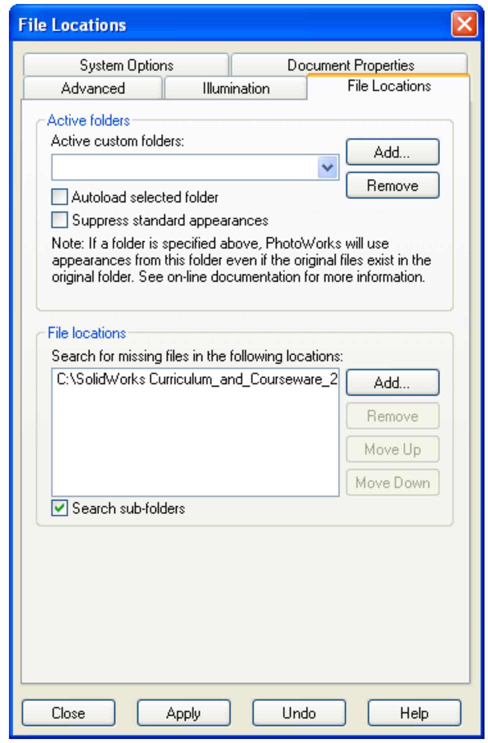
- Autoload selected folder: **Cleared**
- Suppress standard appearances: **Cleared**

For File locations, click **Add**.

Browse to the folder `SolidWorks Curriculum and Courseware_2010` and click **OK**.

Select **Search sub-folders**.

Click **Apply**.



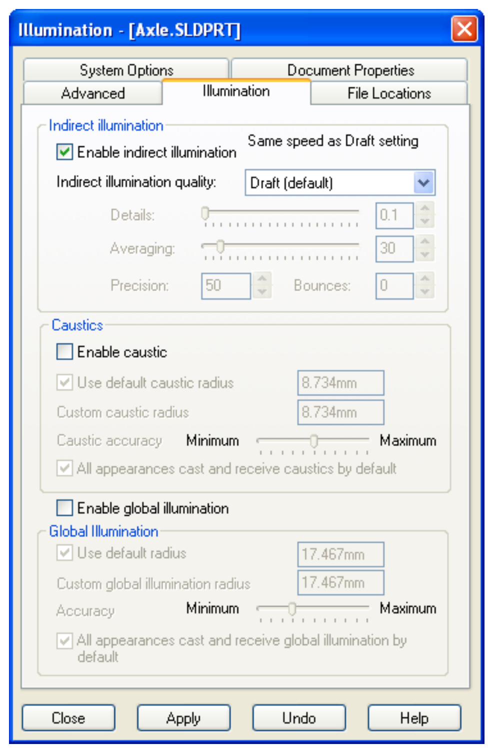
6 Set the **Illumination** options.

Select the **Illumination** tab.

Set the **Illumination** options as follows:

- Enable indirect illumination: **Selected**
- Enable caustic: **Cleared**
- Enable global illumination: **Cleared**

Click **Apply**.



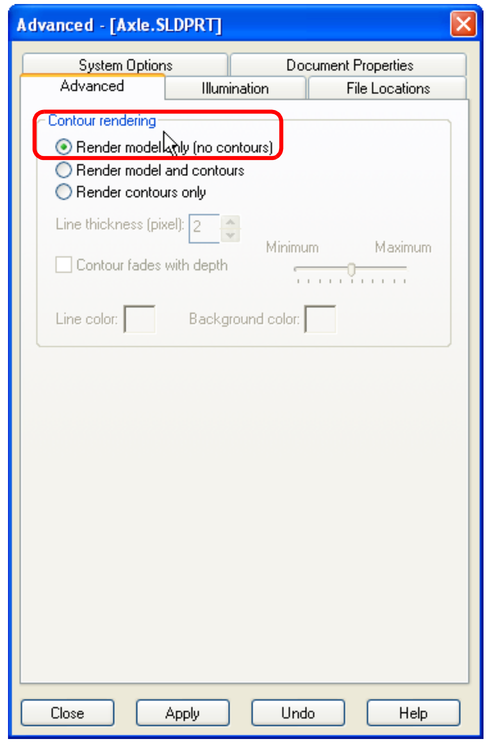
7 Set the **Advanced** Options

Select the **Advanced** tab.

Select: **Render model only (no contours)**

Click **Apply.**

8 **Close** the **Options** dialog box.



PhotoWorks Rendering Procedure

For each SolidWorks model to be rendered, the process requires the same steps within PhotoWorks. 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

  Choose from the various preset lights, or create your own.

- ❑ Render the Model

  The model, in whole or in part, is rendered to the screen.

- ❑ Choose Output

  PhotoWorks can produce both electronic and printed output. Decisions are made to determine type of output, the size, and resolution.

- ❑ Post Processing

  The PhotoWorks output is not always the final product. Often the PhotoWorks output is used with other programs for additional effects.

Render Wizard

When you are unsure of the proper steps to render a part, the Render Wizard can be used to lead us through the basic steps. It will show us how to apply material and set a basic scene.

To use the **Render Wizard**:

❑ Click **PhotoWorks, Render Wizard**.

Task 3 — Use the Render Wizard

1 Open part Axle.

2 Orient the model to the Isometric view.

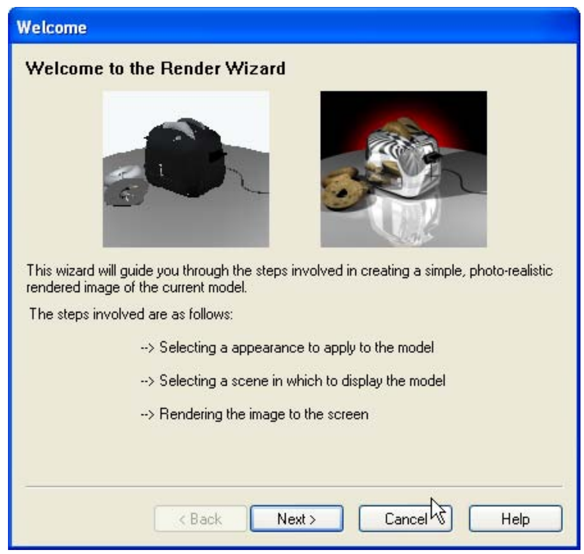
3 Render Wizard.

Click **PhotoWorks, Render Wizard**.

The **Render Wizard** lists three basic steps to create the first rendering:

- Apply material
- Select a scene
- Render the image

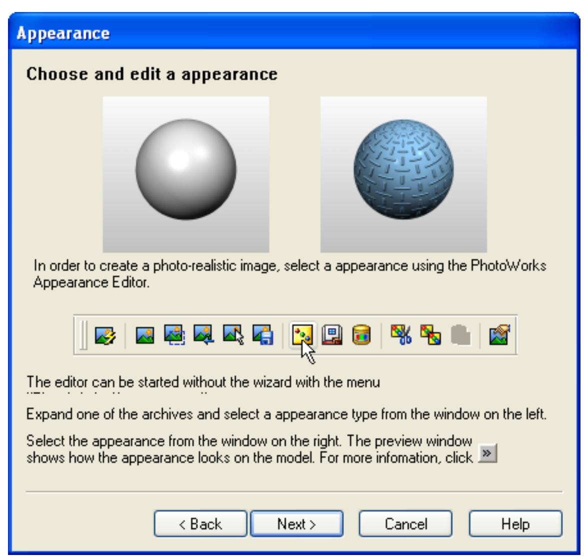
Click **Next**.



4 Select an appearance.

The wizard instructs us to choose an appearance from the **Appearances Editor**.

The **Appearances Editor** will open in the PropertyManager and the Task Pane will open to the Appearances/ PhotoWorks tab.



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 material, select the object to which the material will be applied. Then, double-click the material in the material selection area of the task pane, or drag the material onto the selected part, feature or face.

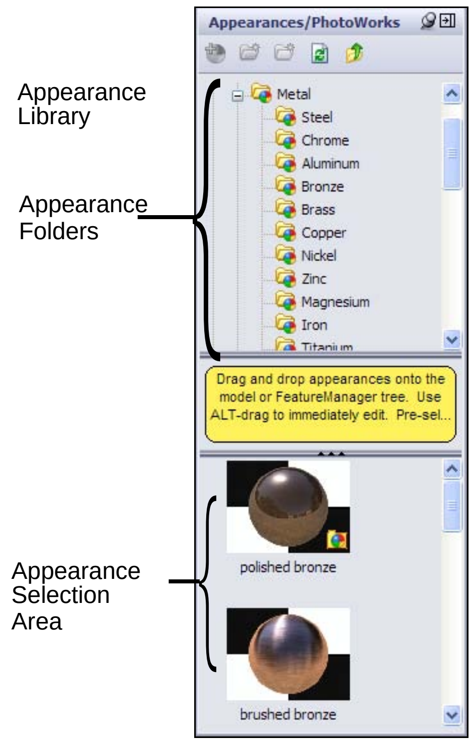
To apply materials:

- ❑ Click **PhotoWorks, Appearance**.
- ❑ Or, click **Appearance**  on the PhotoWorks toolbar.

Appearance/PhotoWorks Pane

The **Appearance/PhotoWorks Pane** lists all the materials 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

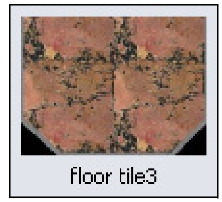
Procedural appearances are defined by some procedure and consist of one or more colors and the way the appearance reacts to light. Procedural appearances can be thought of as 3D, that is they go all the way through the part. This is like adding dye to a plastic to be injected; the color will go all the way through the finished part. Procedural appearances are shown in the material selection area as a sphere on a checkered background.

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.

Appearances Display

Texture appearances and procedural appearances are included together in the appearances folders. Procedural appearances have a preview image that is a sphere. Texture appearances have a 2D image of the texture.

Procedural		Texture	
Procedural materials are shown on a rendered sphere in a showroom.		Texture materials are show as rectangular images	

Appearance Hierarchy

PhotoWorks appearances follow the same hierarchy as do appearances in SolidWorks. Appearances applied to a face override appearances applied to a feature, which override appearances applied to the part.

5 Apply appearances to the part.

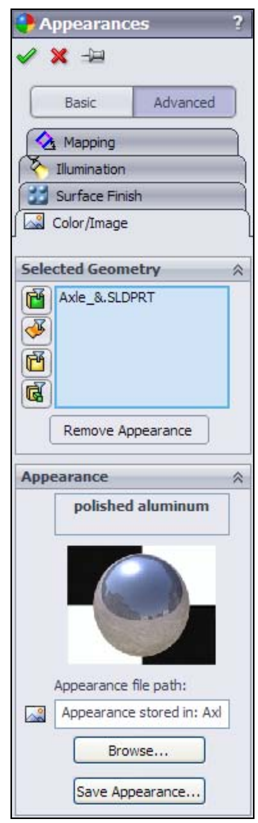
Click the plus sign to the left of the **Metal** folder, then select the sub-folder **Aluminum**.

In the material selection area, scroll until you find **polished aluminum**. 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 .

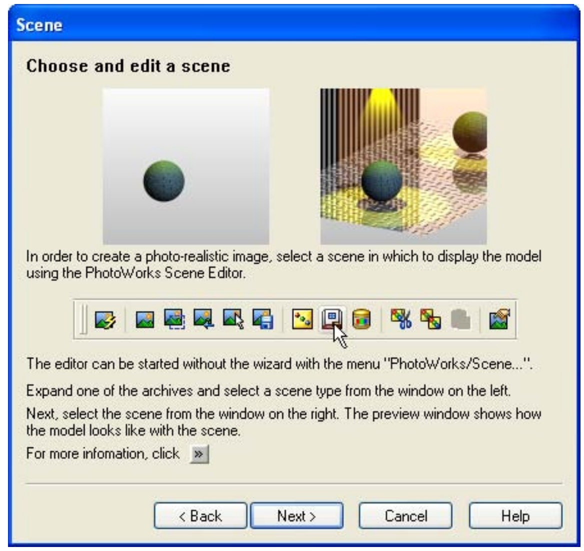
Click **Next** in the Wizard.



6 Select a scene.

The next task is to apply a scene.

The **Scene Editor** will open in its own window.



Non-Modal Dialog Boxes

All dialog boxes in PhotoWorks are non-modal, meaning that they do not lock up SolidWorks or PhotoWorks when they are open.

PhotoWorks Scenes

PhotoWorks 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 box or sphere around the model. Scenes are composed of backgrounds, foreground effects, and scenery. PhotoWorks has numerous predefined scenes to make initial renderings quick and easy.

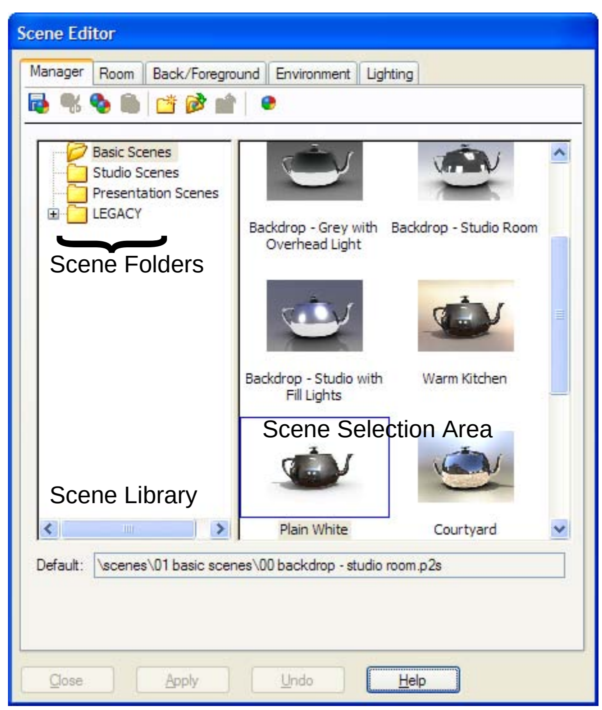
Scene Editor

The **Scene Editor** lists all the pre-defined scenes that are available to be applied to the model.

The left 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 right panel is the **Scene Selection** area.

To apply a scene:

- ❑ Select the scene in the scene selection area, then click **Apply.**



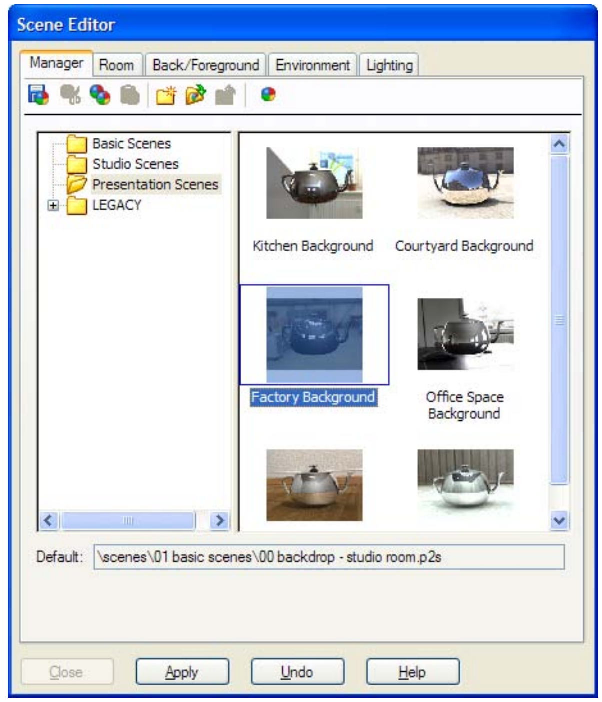
7 Select a scene.

Click the plus sign next to the **Presentation Scenes** folder.

Select the scene **Factory Background**.

Click **Apply**.

Click **Next** on the Render Wizard.

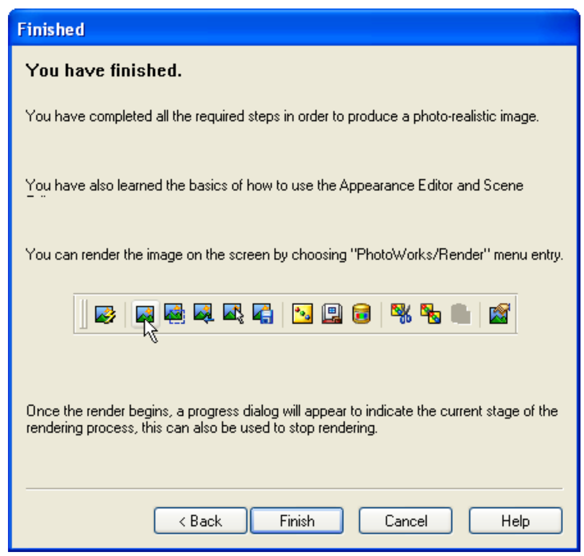


8 Render.

The final instruction summarizes the process.

The Render Wizard will close the **Scene Editor** and render the model.

Click **Finish** to close the Render Wizard.



The `Axle` will be rendered. Examine the rendering, you should be able to see the reflections of the holes on the flat surfaces of the model.

The Render Wizard led us through the steps to add material and a scene to the model, but it didn't change the default lighting. We can improve the rendering by adding some additional light.

Lighting

Proper lighting can greatly enhance the quality of the rendering. The same principles used by photographers work well in PhotoWorks.

Lights are created and positioned in SolidWorks. PhotoWorks has a few additional controls to refine the quality of the light and shadows.

Types of Lights

SolidWorks and PhotoWorks 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.

Creating Lights

SolidWorks creates multiple lights with each new part, depending on the RealView scene selected. To create additional lights, right-click either the `Lights, Cameras and Scene` folder or any light in the lighting folder. 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.

Task 4 — Add a Fill Light

The default lighting has two Directional lights. As we are looking at the computer monitor, the default lights are both on the right side of the model which makes some of the faces dark and hard to see. We will move one of the lights over our left shoulder to act as a Key light, then use the remaining light as a fill light to make the shadow areas easier to see.

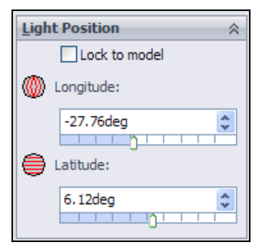
1. Clear the rendering. The model will only stay rendered if it is not moved. To clear the rendering, move the model, then change the view back to `Isometric`.

2 In the FeatureManager design tree, right-click the Lights, Cameras and Scene folder and select **Show Lights**.

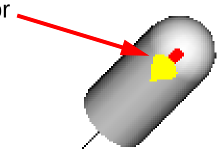
3 Double-click light Directional2. The position of the light can be changed by dragging the manipulator or by the sliders and entry boxes in the PropertyManager.

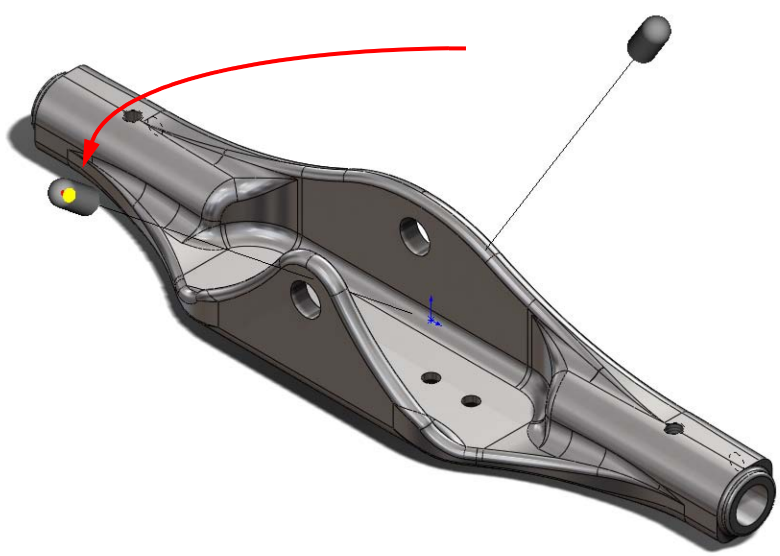
For this rendering, drag the position of Directional2 to the approximate position shown.

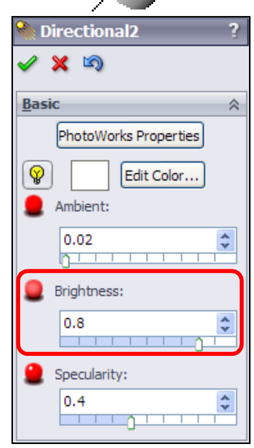
Click  to close the Directional2 PropertyManager.



Manipulator







4 Because this is a Key light, we are using it as the primary illumination, so it should be brighter than the Fill light.

Adjust the brightness by either moving the slider or typing the value **.8** in the box.

5 Render the model again by clicking **Render** on the PhotoWorks toolbar.

6 Examine the rendering. The model will look brighter and the rounded edges will standout because they have two lights on them.

7 **Save** and **Close** the Axle.

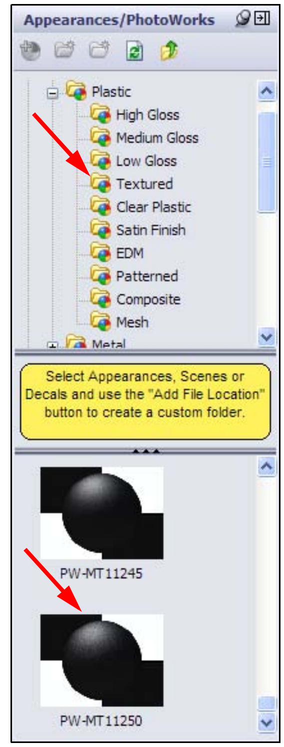
Active Learning Exercise, Part 4 — Rendering the Truck

In the last task, we let the **Render Wizard** lead us thought the steps. This time we will do the steps manually to gain some additional control.

To review, we must do the following steps:

- ❑ Position the model
- ❑ Apply Appearance
- ❑ Set the Scene
- ❑ Set the Lights
- ❑ 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. Click **Appearance** on the PhotoWorks toolbar or click the **Appearances/PhotoWorks** 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. Render the model by clicking **Render** on the PhotoWorks toolbar.

   Because we have not specified a scene, PhotoWorks will render the model with a default scene and lights. 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.

Task 2 — PhotoWorks Studio

PhotoWorks Studio provides a quick method to apply and adjust a stock scene and lighting. With just two adjustments, you control many variables to create a rendering.

To start PhotoWorks Studio:

- ❑ Click **PhotoWorks, PhotoWorks Studio** from the menu.
- ❑ Or, click **PhotoWorks Studio** on the toolbar.

1. Click **PhotoWorks Studio** on the toolbar.

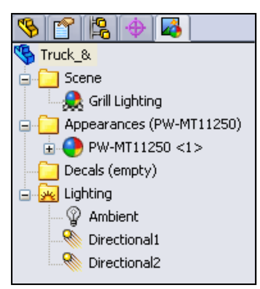
2 You can choose different scenery by either using the pull-down list or by clicking the forward or back arrows.

There are 28 different studios to select from. Using either the forward/back arrows or the pull-down list, select **Grill Lighting**. *Do Not* click **OK**.

3 Render the model.

We did not click , so the settings for this studio have not been added to the `Truck` file. Experiment with different studios by selecting the studio and then rendering the image. When you are done, return to the **Grill Lighting** studio and click **OK**.

4 Examine the results. 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.



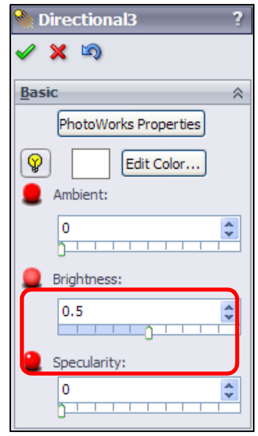
Some things we can see in the rendering:

- We can get a good idea of the light positions from the shadows, but there are too many shadows. The image may look better if we don't see shadows from all the lights.
- 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.

Task 3 — Changing the light intensity

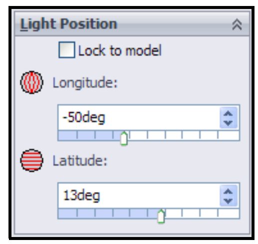
We are going to do two things to refine our rendering. First is to add a directional light. We will use this light to shine into some of the dark areas. Second, we will turn off some of the shadows.

1. In the FeatureManager design tree, right-click the **Lights, Camera and Scene** folder and select **Add Directional Light**.
2. In the FeatureManager design tree, double-click the light Directional3. This will open the properties of the light and show the manipulator.



Note: We could also right-click the light and select **Properties** to access the properties of the light.

3. Adjust the **Brightness** slider to a value of **0.5**.
4. 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.



5. Click  and render again.
6. Examine the rendering. 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 so we will eliminate the shadow from the directional light.

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 in two places, globally through the Scene Manager and individually in the properties for each light.

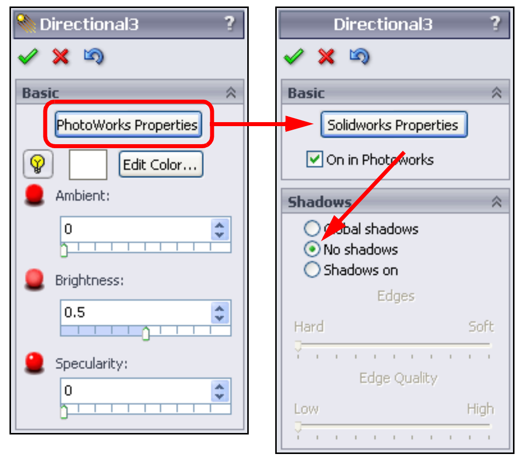
Task 4 — Shadow Control

There are too many shadows in the rendering which causes shadow clutter. We will remove the shadow from the light Directional3, then soften the edges of the remaining shadows.

1. In the FeatureManager design tree, right-click on the light Directional3 and select **Properties**.

   Select **PhotoWorks Properties**.

   Select **No shadows**. This will turn off the shadow for just this one light. Click **OK**.



2. **Render** the model.

   The shadow on the back surface of the part and the floor, caused by the directional light is now gone.

3. In the RenderManager, right-click **Scene** and select **Edit Scenery**.
4. Click the **Lighting** tab.
5. Move the **Edges** slider to mid-scale.
6. Move the **Edge quality** slider to **8**.

Caution: Do not move these sliders too far to the right as rendering time goes up rapidly as the sliders move to the right. Notice that the **Edge quality** slider has binary units (1, 2, 4, 8, 16,...). Think of rendering time doubling for every tick mark you move the slider to the right.

7 Click **Apply** and **Close**.

8 **Render** the model.

This rendering is acceptable for our needs.

9 **Save** the part.

Render to a Printer

Rendering directly to a printer is useful for creating a hard copy image of a project. It is limited because you cannot add captions, put multiple images on a page or manipulate the image. They are not useful for illustrations in manuals as they would have to be converted into a graphics file. Some common uses of printer renderings might be for:

- Lobby displays of products before production begins.
- Display boards at conferences.
- Project reports.

To obtain rendered output from a printer, you must use the PhotoWorks print command, *not* the SolidWorks print command.

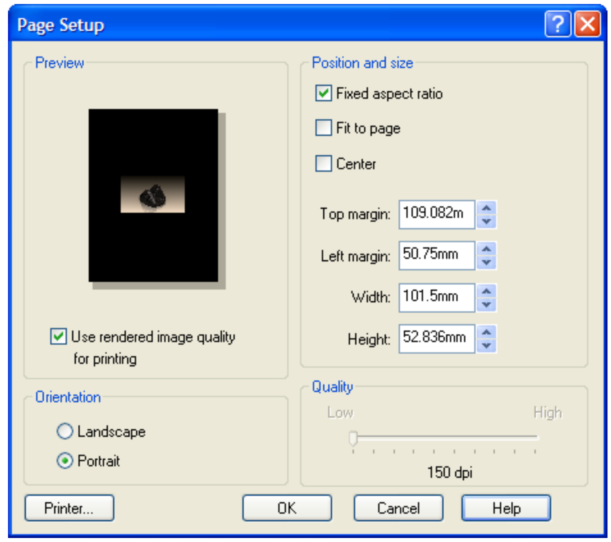
To print a PhotoWorks rendering:

❑ Click **PhotoWorks, Print**.

Task 5 — Print a rendering

Now that the `Truck` is setup to render on our screen, we will send a copy to the printer.

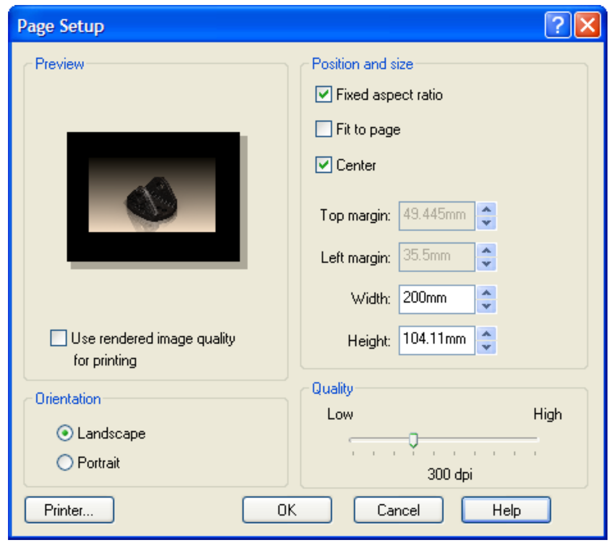
Note: If you do not have access to a color printer, you will not be able to do this task.



1 Click **PhotoWorks, Page Setup**.

2 The PhotoWorks **Page Setup** allows us to size and position the image that will be printed.

Note: If you do not have the image rendered on the screen, the **Page Setup** will show an empty frame with crosshatch instead of a rendered preview.



3 Select **Landscape** orientation.

4 Make sure **Fixed aspect ratio** is selected so we do not distort the image when we change its size.

5 Select **Center**. This will automatically center the image on the printed page.

6 Type **200mm** for the **Width**, the **Height** will automatically adjust to maintain the aspect ratio.

The actual value of the height is determined by the proportions of your graphics area.

7 Clear **Use rendered image quality for printing**. The rendered image on our screen is only 72 or 96 dpi (dots per inch), if we use this quality for our print, it will be very grainy.

8 Adjust the **Quality** slider to **300 dpi**.

Note: There is generally no need to print above 300 dpi. If you have a higher quality printer, you can try a higher setting for print quality, but it will also take longer to render.

9 Click **OK.**

10 Click **PhotoWorks, Print** to render the image and send it to the printer. Note that this will take much longer than when we render to the screen as PhotoWorks must render more than nine times more pixels.

11 **Save** and **Close** the `Truck`.

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. PhotoWorks installs a variety of texture appearances, however, it is easy to create additional texture materials. We can do this from any image type that PhotoWorks recognizes.

Task 6 — Add A Texture Appearance

We will add a texture appearance to the Deck, then customize it.

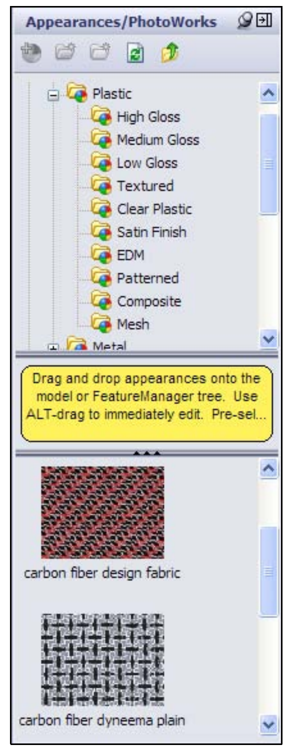
1 **Open** the part Deck.

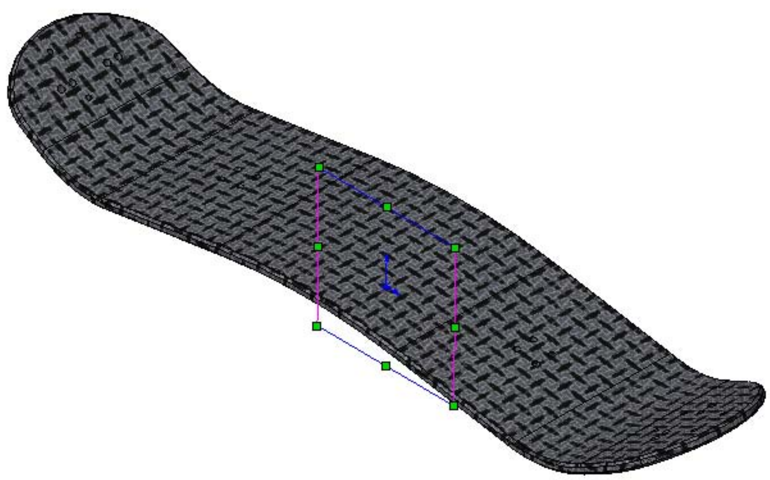
2 Click **Appearance** on the PhotoWorks toolbar.

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.

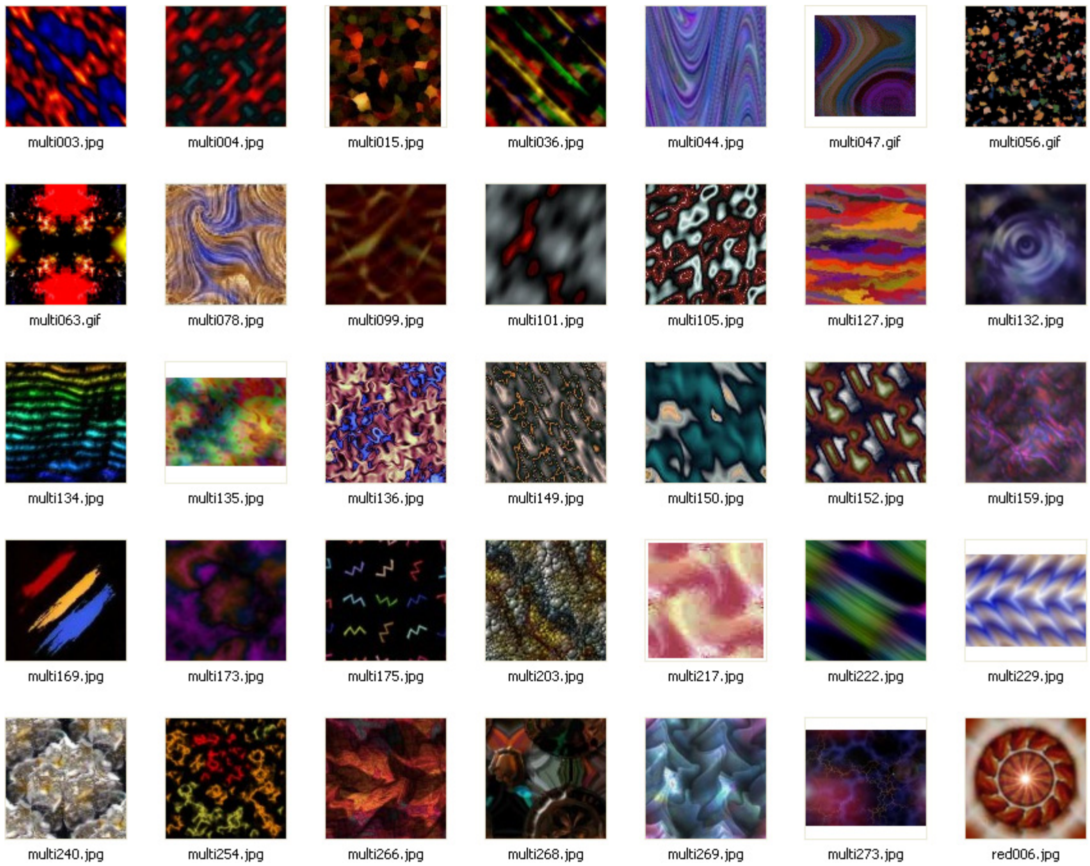
4 Click **OK**.

5 The material 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.





6 If we can't find something we like, we can make our own material from available graphic images. A variety of images are provided in the folder `SolidWorks Curriculum and Courseware2010\Mountainboard Design Project\Images\Appearance Images`.



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

PhotoWorks 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.
- 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:

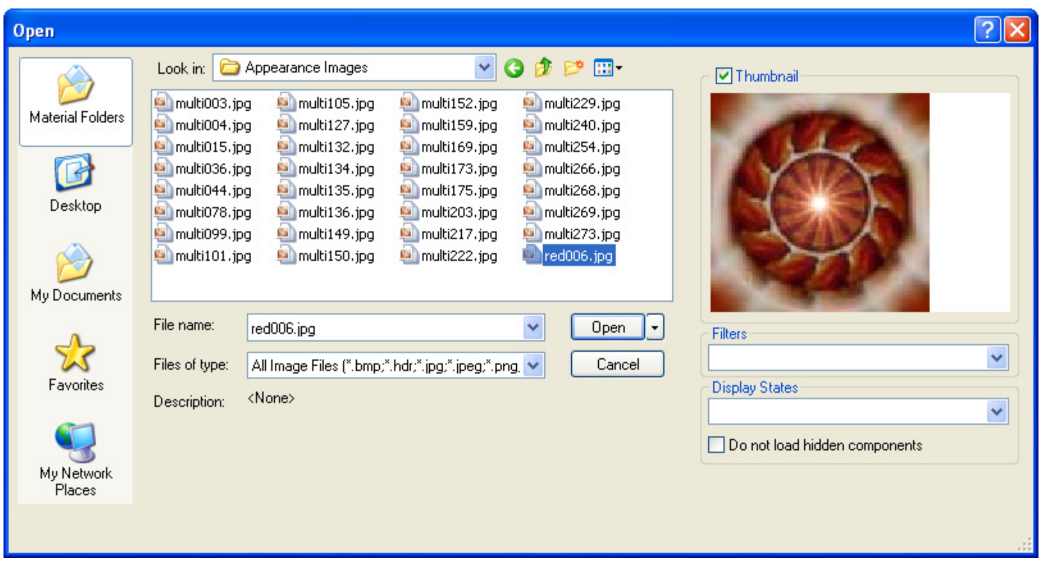
- ❑ Editing the appearance and selecting a different appearance
  Right-click the appearance and select **Edit**
- ❑ Detaching the current appearance
  Right-click the appearance and select **Detach**
- ❑ Cutting the appearance
  Right-click the appearance and select **Cut**

Task 7 — 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 material.

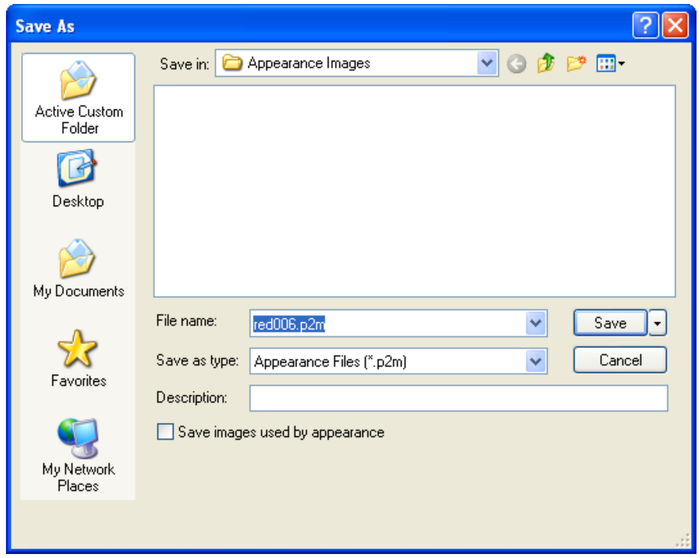
1 In the Appearances PropertyManager, select the **Advanced** button. Under **Appearance**, click **Browse** and navigate to the SolidWorks Curriculum and Courseware_2010\Mountainboard Design Project\Images\Appearance Images folder.

2 Select the image red006.jpg or any other image you would like to use.



3 Click **Open**.

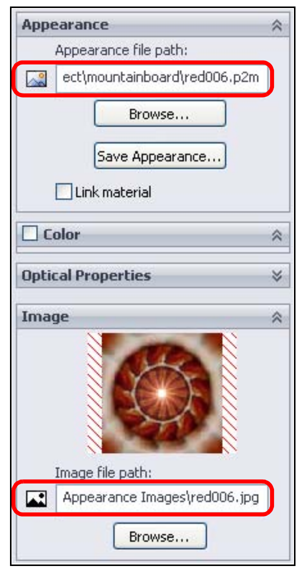
4 Because this is a new Appearance, we must save the new Appearance file. It will have a file extension of *.p2m. Save the file to the same folder as the image with the same name.



5 PhotoWorks will ask if you want to open this new folder in the **Appearance folder**, click **Yes**.

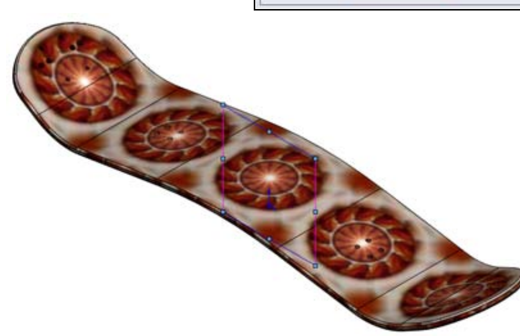
6 The new folder will appear in the RealView/PhotoWorks section of the Task Pane with the new material listed.

7 In 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.

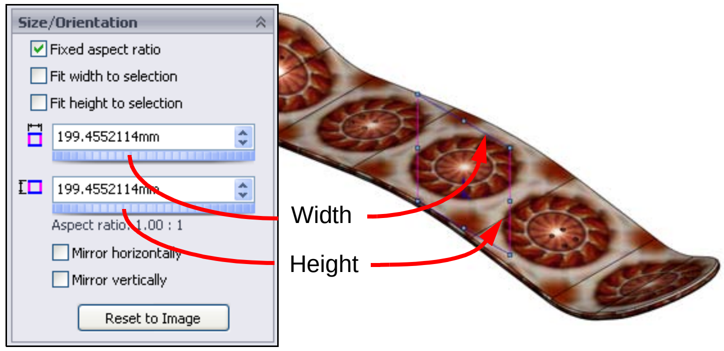


8 The image is interesting, but we would only like to see one pattern instance stretched over the top surface.

9 Select the **Mapping** tab in the Appearance PropertyManager.



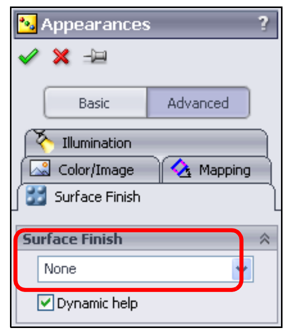
10 Clear **Fixed aspect ratio**, this will allow us to control the width and height separately.



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 ✔.

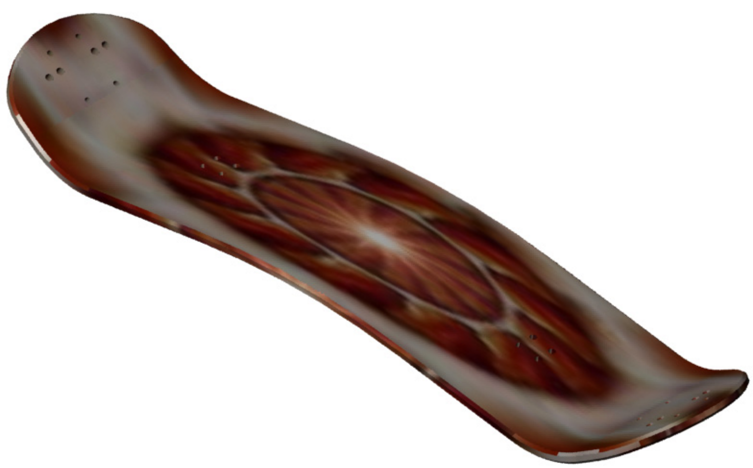
13 Select the **Surface Finish** tab. Select **None** from the list. Click ✔.



14 **Render** the model.

We have stretched a single instance of the pattern to cover the entire top face of the Mountainboard.

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 will 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 material to the chamfer feature and the side faces.

16 In the FeatureManager design tree select the feature `Chamfer1`. Click **Appearances**  on the PhotoWorks toolbar.

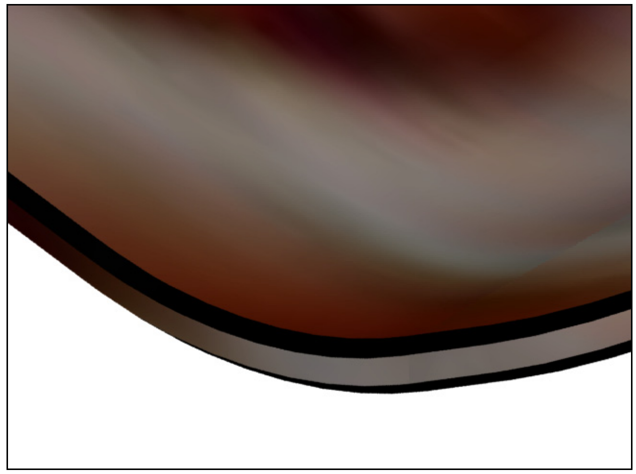
17 Double-click the material **blue polished ABS plastic** from the **Plastics, High Gloss** folder. This will apply the material to just the chamfer feature.

18 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 **Render** the model.



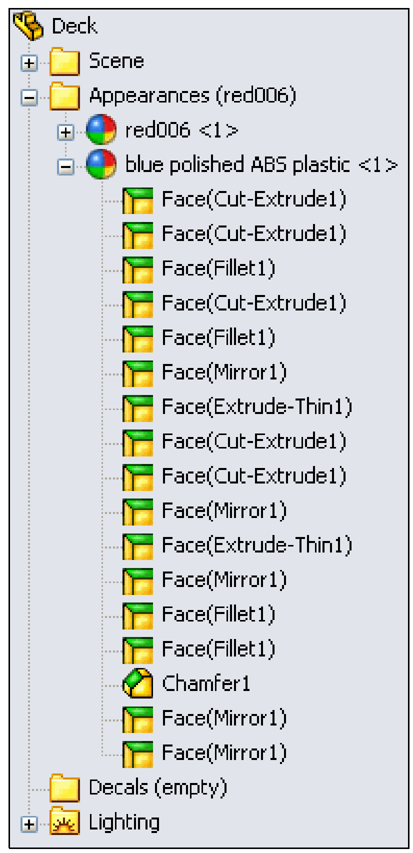
22 The chamfers are black, but we still have to do the thin faces between the chamfers.

23 Select the RenderManager tab. Click the plus sign to expand the Appearances folder so you can see the two materials listed.

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.

24 In the RenderManager, right-click the material **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 materials.

Task 1— Create a Decal file

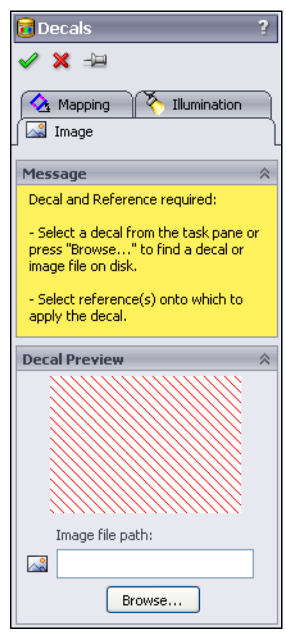
We have a JPG image file with our logo that we want to add to the top face of the Deck.

1 We will create a new Decal using essentially the same method we used to create a new texture material in that we will use an existing decal and modify it to use our own image.

2 Click **New Decal** on the PhotoWorks toolbar.

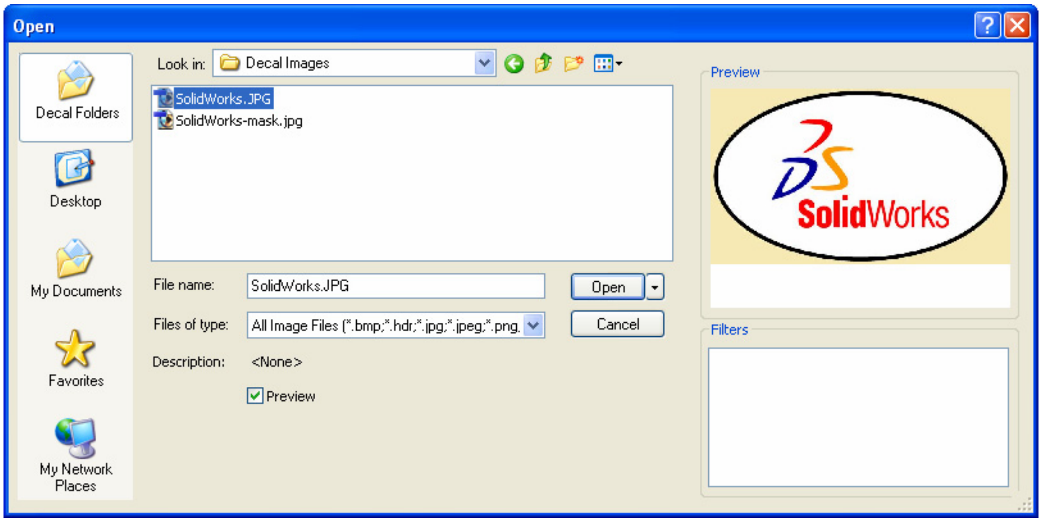
3 The **Decal Editor** opens. We will make a decal from the file provided.

4 Under **Image file path**, Click **Browse...**.



5 Navigate to the SolidWorks Curriculum and Courseware_2010\Mountainboard Design Project\Images\Decal Images folder.

6 Select the file SolidWorks.jpg, then click **Open**.



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.

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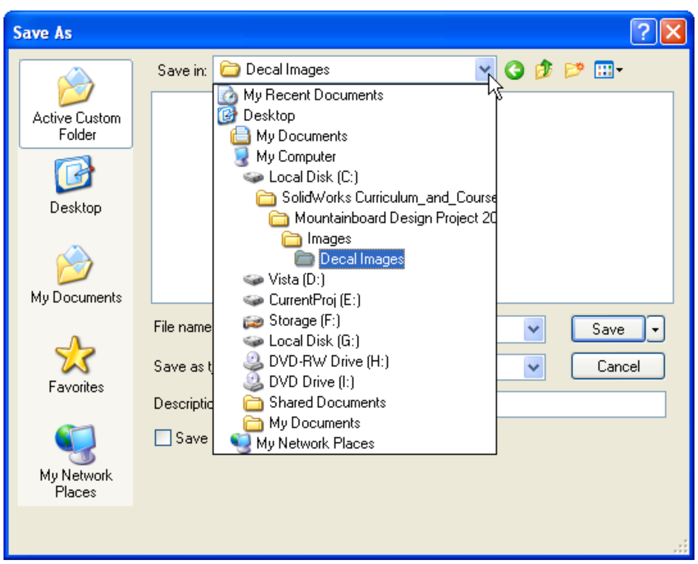
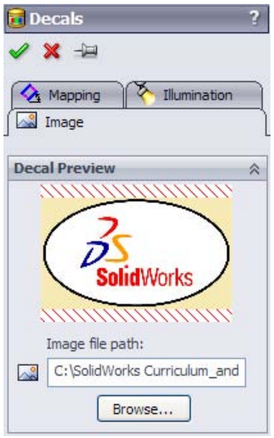
7 Click **Save Decal...**.
Save the decal file as SolidWorks.p2d to the ...\SolidWorks Curriculum and Courseware_2010\Mountainboard Design Project\Images\Decal Images folder.
Click **Save**.

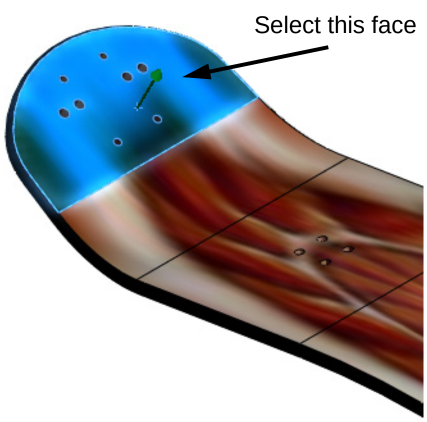


8 Click **Yes** to have PhotoWorks open the Decals folder in the **Decal Editor**.

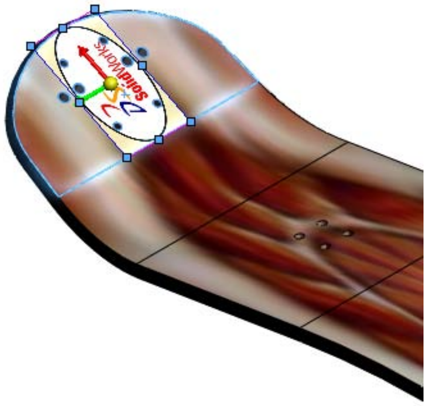
9 The **Decal Editor** will show the image.



10 In the graphics area, zoom in on the area shown and select the face indicated.



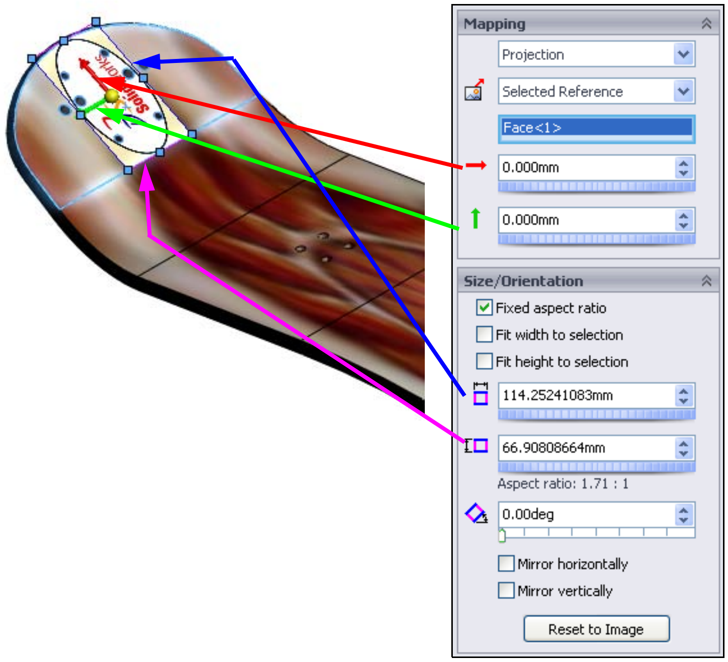
11 The Preview shows the decal on the selected face.



12 Select the **Mapping** tab in the **Decal Editor**.

Notice that the colors used in the Preview correspond to the color shown in the **Mapping** tab indicating:

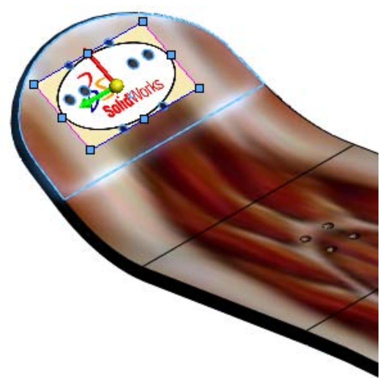
- Width - Blue
- Height - Magenta
- Horizontal Location - Red
- Vertical Location - Green



15 Make sure **Fixed aspect ratio** is selected.

16 Type **270.00deg** for **Rotation Angle** the press **Enter**.

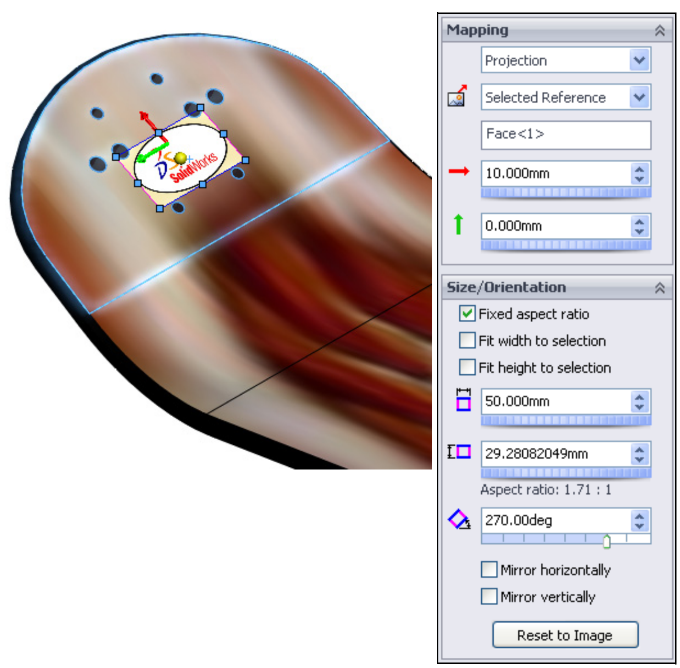
17 The preview will show that the decal has been rotated to the correct orientation. If the decal is upside down, use 90 deg instead of 270 deg.



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.

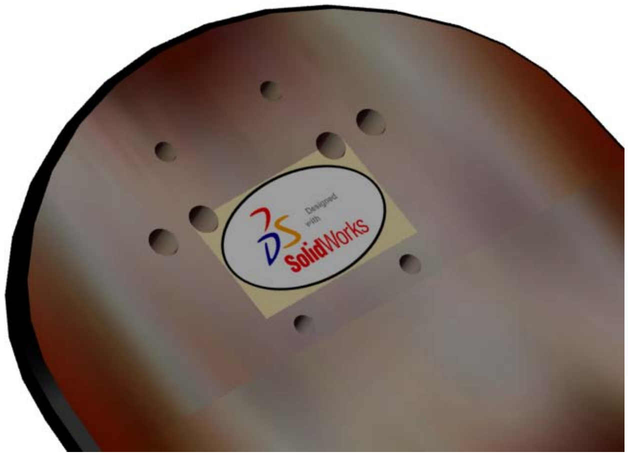
- Horizontal Location:**10 mm**
- Width: **50mm**

Click ✓.



19 **Render** 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.



20 Right-click the decal in the RenderManager and select **Edit**.

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.

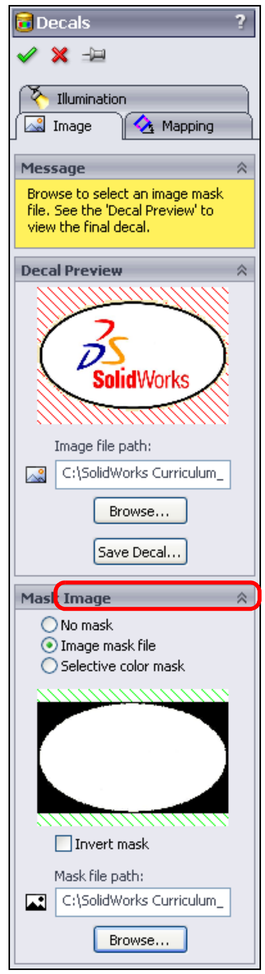


Image Mask

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.

Click ✔.

24 **Render** the model.

The tan area has now been masked out leaving only the ellipse and the image inside.

The decal is a little dull as it takes on the reflective properties of the material on which it is applied.

25 In the RenderManager, right-click the decal `SolidWorks` and select **Edit**.

26 Select the **Illumination** tab.

27 Select **Constant** from the list.

28 Click **OK**.

29 **Render** the model. The decal now stands out as its reflective characteristics are now different from the `Deck`.

30 **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 or face. At the assembly level we can add the appearance to the entire assembly or individual parts. We can not add appearances to features or faces while at the assembly level.

Where we apply the appearance depends on the different renderings we intend to create. If we apply a 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.

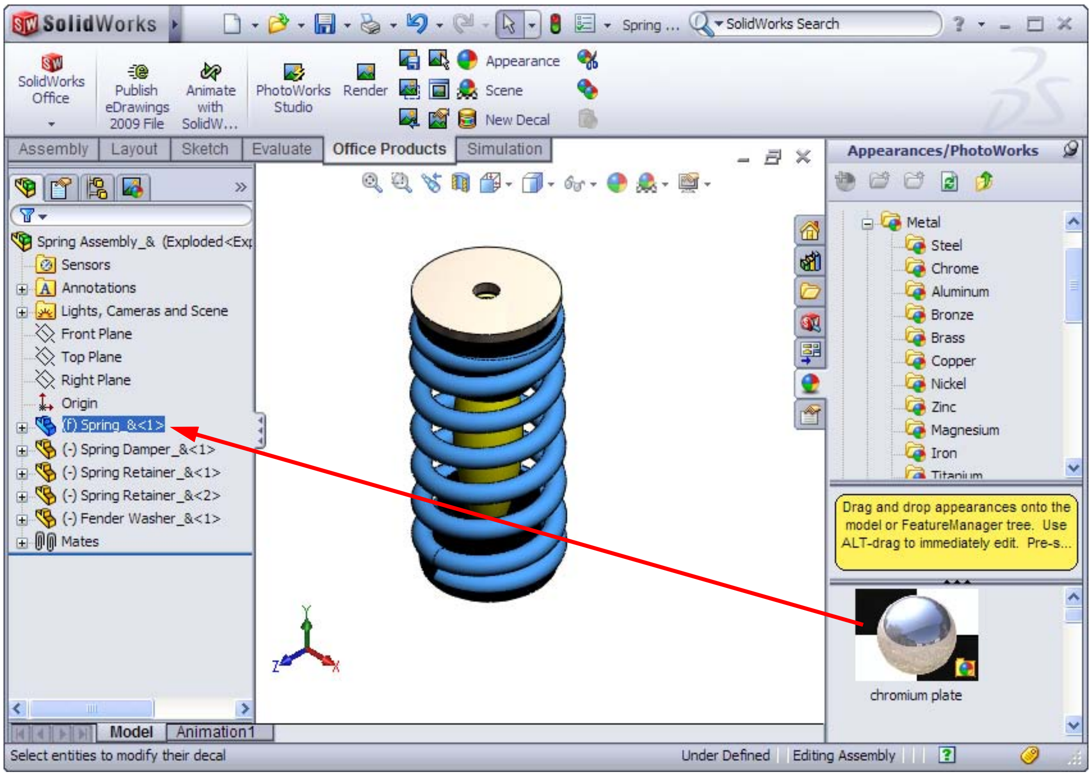
Task 1— Add appearances to the Spring Assembly

1 **Open** the Spring Assembly.

2 Select the **PhotoWorks** tab in the Task Pane then click the push pin to keep the pane open .

3 Locate the material **chromium plate** in the **Metals, Chrome** folder. Drag the appearance onto the Spring part in the FeatureManager design tree.

Click . This will apply the material to just the Spring part.



4 Use the same procedure to apply the other materials to the parts in this assembly:

- Fender Washer - Metal, Steel, brushed steel
- Spring Dampener - Plastics, High Gloss, white high gloss plastic (change color to yellow)
- Spring Retainer - Plastics, High Gloss, white high gloss plastic (change color to black)

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 materials 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 material to the surfaces that will show. because most of the inner tube is unseen because it is inside the tire. We will only add material 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 **polished plastic1** from the **Plastics, High 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.

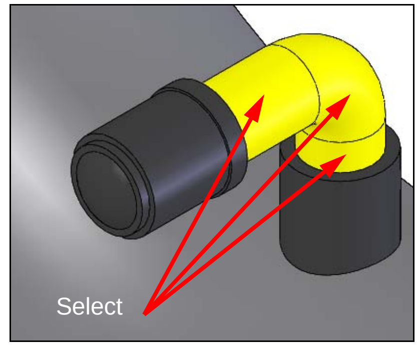
6 In the Task Pane, double-click the material **polished plastic1.** This will apply the material to just these two features. Select the **Color/Image** tab and change the color to red.

7 **Render** 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 **polished plastic1** from the plastics, High Gloss folder. Change the color to black.

16 **Save** the part.

17 Return to the Wheel Assembly by clicking Window and selecting Wheel Assembly from the list.

18 Click **Scene** on the PhotoWorks toolbar.

19 Select the **Studio Scenes** folder and the **Light Cards** studio.

Click **Apply** and **Close**.

20 Change the view to **Trimetric** by clicking **Trimetric** on the Standard Views toolbar.

21 **Render** the model to check your work.

22 **Save** the assembly.

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 - **blue polished ABS plastic (change color to white)**
- Binding Anchor - **polished aluminum**

3 **Save** and **Close** the assembly.

4 Apply the same materials to the MirrorBinding assembly.

Active Learning Exercises, Part 8 — Final Rendering

The final step is to apply materials 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

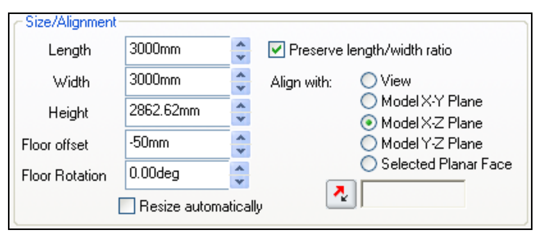
1. **Open** the Mountainboard assembly.
2. Select all the fasteners and hardware. Because we put all these files in their own folder they are easier to find and select. Click the plus sign next to the Hardware folder to expand the listing. Select the first fastener in the list, then press and hold the **Shift** key and select the last fastener in the list. This will select the two fasteners you selected plus everything in between.
3. Click **Appearance** on the PhotoWorks toolbar.
4. Double-click the material **polished steel** from the **Metal, Steel** folder.
5. Click to apply the material and close the Appearance Editor.

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 to add realism to the final output.

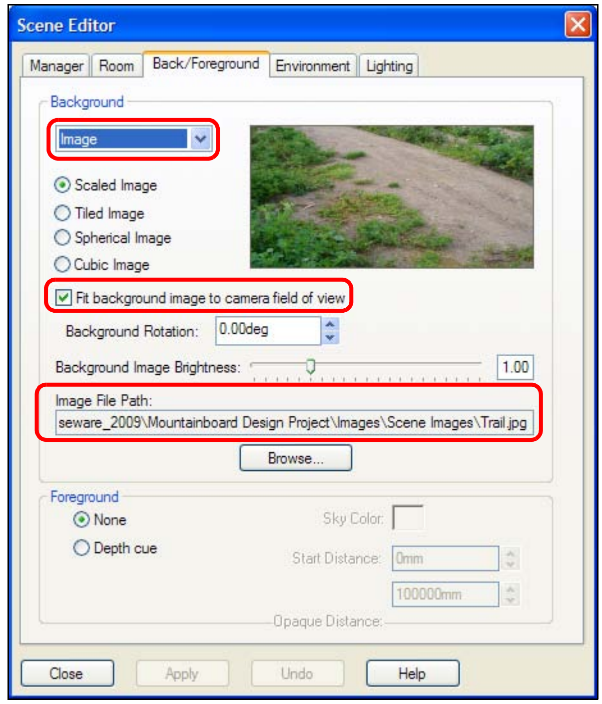
1. Reorient the Mountainboard to the Isometric view.
2. Click **Scene** in the PhotoWorks toolbar.
3. Select **Basic Scenes** and look at the available choices.
4. None of these studios are really what we want, so we can pick anyone of the existing bases then modify it.
5. Select the **Plain White**, then click **Apply**.
6. Select the **Room** tab.
7. Type **3000mm** for the **Length**. If you have **Preserve length/width ratio** selected, the **Width** will also be 3000mm.
8. Select **Align with: Model X-Z Plane**.
9. Clear **Resize automatically** to prevent the size of the base from changing if we make any changes to the model.



10. Type **-50 mm** for the **Floor offset**.
11. The default appearance for the floor is called shadow floor and is also visible. This appearance will be basically invisible, but will show where the shadows fall.

Digital Images

To apply a digital image to a scenery element, we can do it in a number of different ways. The easiest is to use it as a background.



12 Select the **Back/Foreground** tab.

13 Select **Image** from the list.

12 Click **Browse**. Navigate to the `Images\Scene Images` folder and select the image `Trail.jpg`.

13 Click **Open**.

14 We could adjust the size and rotation if necessary, but for this image, just select **Fit background image to camera field of view.**

15 Click **Apply**.

16 Select the **Lighting** tab.

17 Turn on **Opaque** shadows.

18 Adjust the **Edges** to mid-scale and **Edge quality** to **8**.

19 Click **Apply** and **Close**.

20 Adjust the position of the Mountainboard to an appropriate position over the trail.

21 **Render** the model.

Task 3 — Additional PhotoWorks 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  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**  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**  in the box.

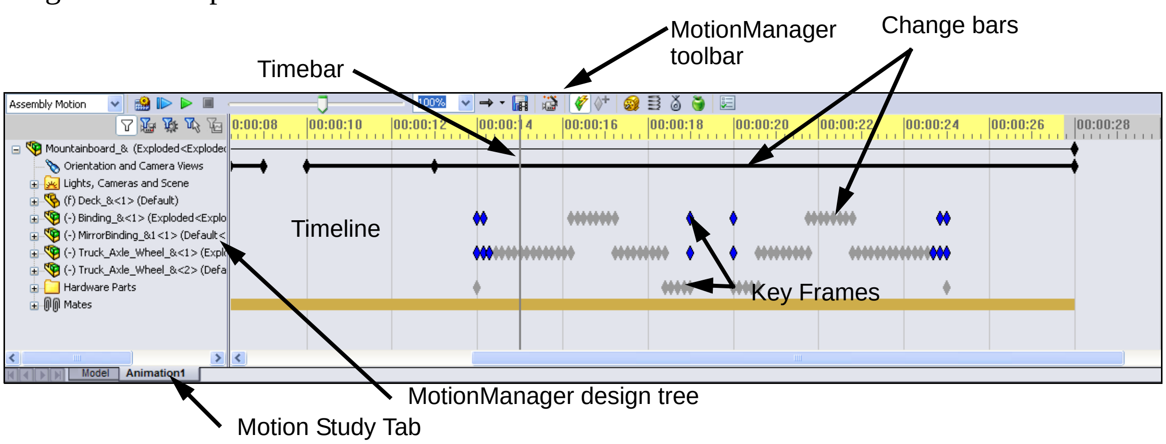
16 Type **Explode** for the name of the view and click **OK**.

17 Collapse the assembly.

18 Use **View, Orientation** to zoom to the **Start** view.

MotionManager Interface

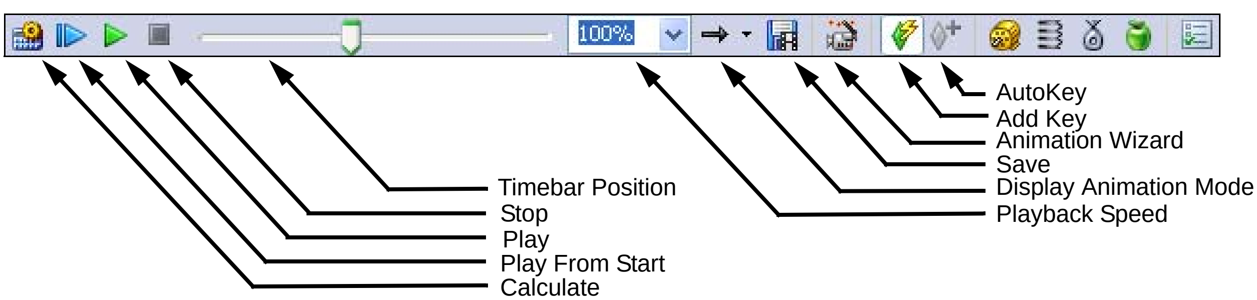
The animation process is based on a key frame-based interface. You decide how your assembly should look at various times, and then Animator computes the sequences needed to go from one position to the next.



Change bars and key frames are color coded to show their function.

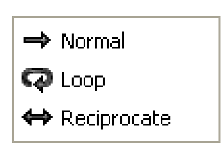
MotionManager Toolbar

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)

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.

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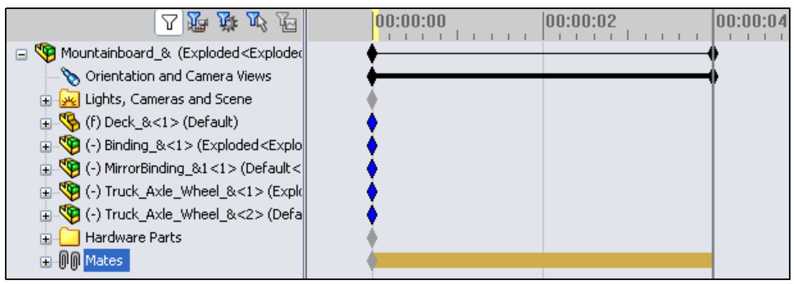
5 Double-click the named view **Zoomed In**.



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 `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.

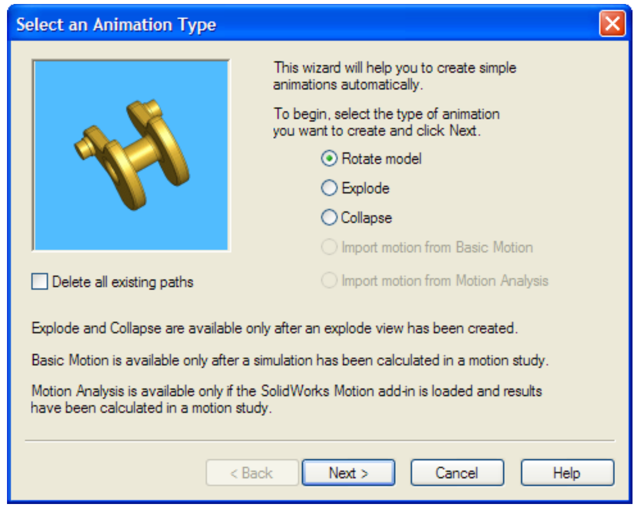
Task 3 — Use the Animation Wizard to create a view rotation

The Animation Wizard makes it easy to rotate the model abut the three axis of our screen.

1 Click **Animation Wizard**  on the Animator toolbar.

2 Select **Rotate Model** for the type of animation.

3 Click **Next.**

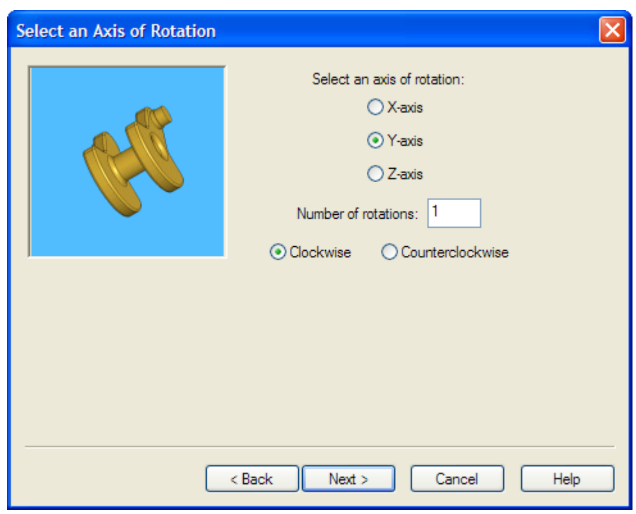


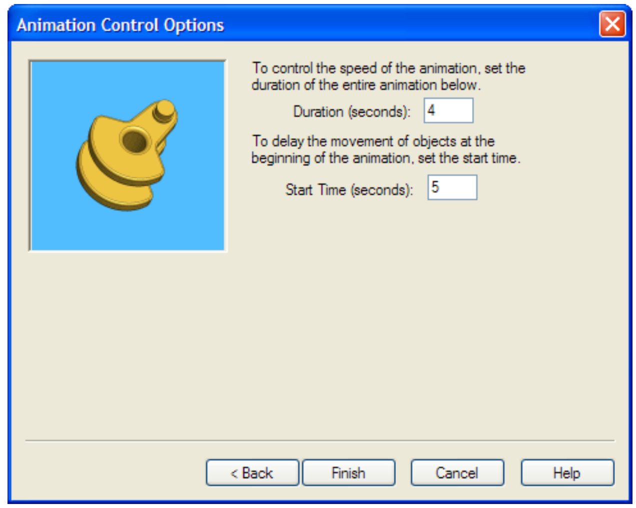
4 Select the **Y-axis** for the axis of rotation.

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.

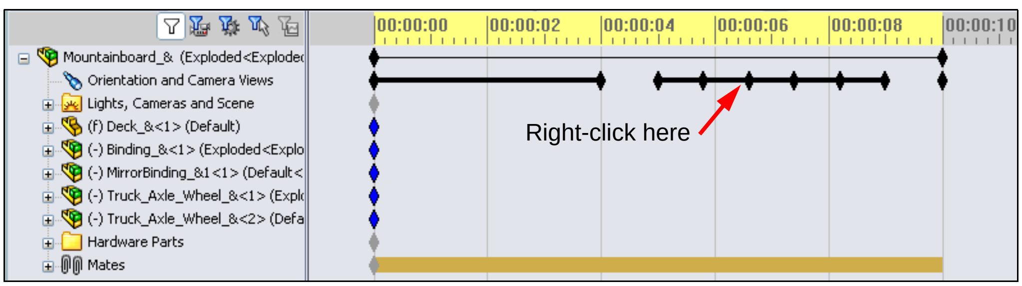
Click **Finished**.

8 Review the animation. Click **Play from Start** on the **MotionManager** toolbar.

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**.

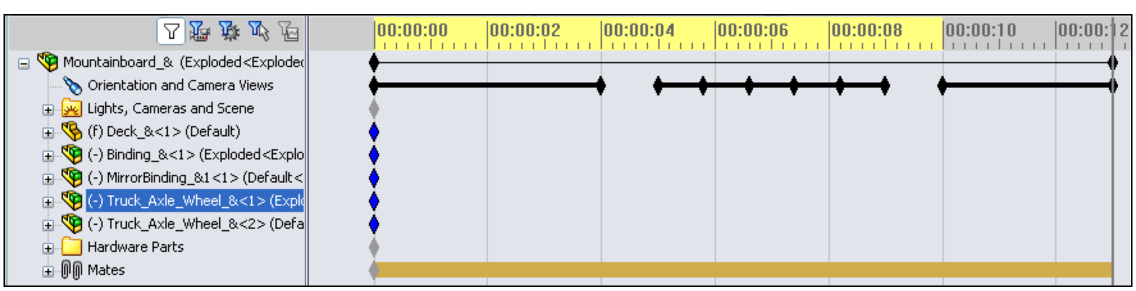


11 Drag the **Timebar** to **13** seconds.

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.

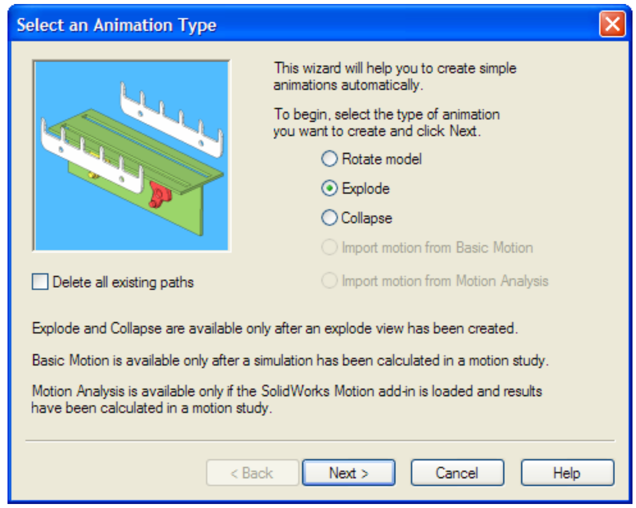


14 Review the animation. Click **Play from Start** on the **Animation** toolbar.

Task 4 — Animated Explode

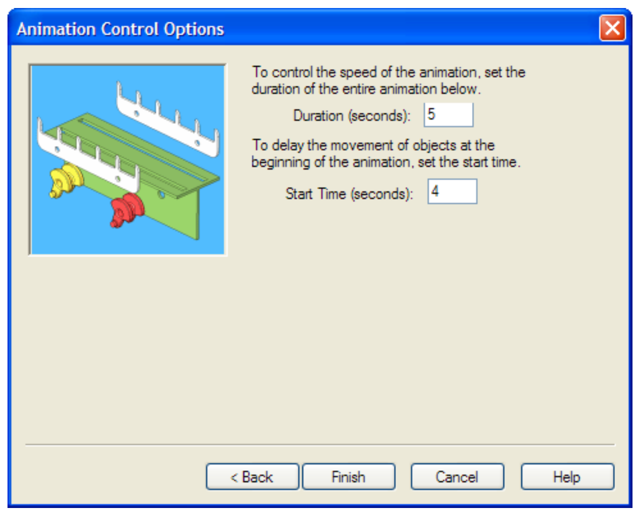
1 Click **Animation Wizard** on the **Animator** toolbar.

2 Select **Explode** and then **Next**.

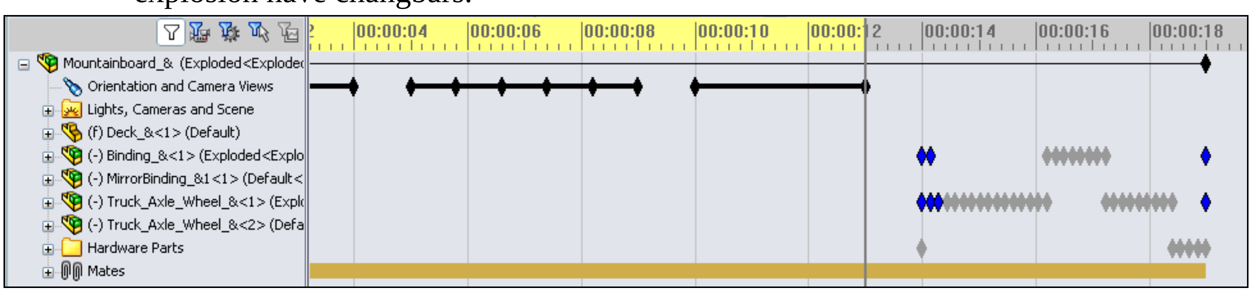


3 Type **5** for **Duration** and **14** for **Start Time**.

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.

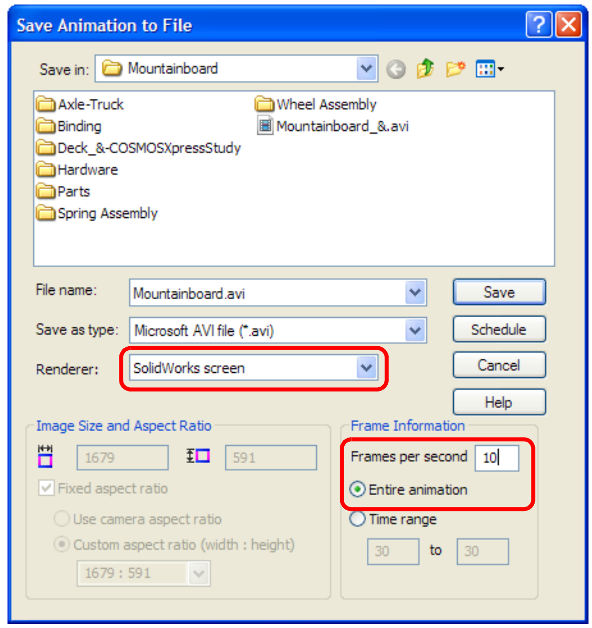
7 Move the **timebar** to 28 seconds and change the view to **Zoomed in**.

8 Use the Animation Wizard to rotate the assembly from **19** to **22** seconds.

9 Review the animation. Click **Play from Start** 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.

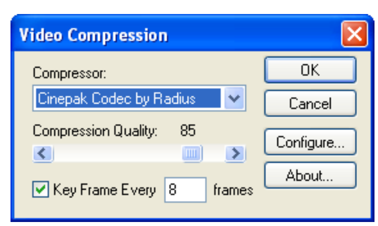
4 Type **10** for **Frames per second** and select **Entire animation**.

5 Click **Save**.

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 you video files because the destination computers must also have the same CODEC loaded.



6 Select **Cinepak Codec by Radius** 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.

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 and 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 PhotoWorks

PhotoWorks 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 PhotoWorks?

____

2 List the rendering effects that are used in PhotoWorks?

____

3 The PhotoWorks__________ ____________ 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: PhotoWorks

Create a marketing brochure using photorealistic images of the Mountainboard and its components.

Exercise 31: PhotoWorks and Animator

Create a web page showing the rendered Mountainboard and an animation of the explode and collapse steps.

Lesson 9 Vocabulary Worksheet

Name: ______________________________Class: _________ Date:_______________

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

1 The ability to dynamically view an eDrawing: ________________________________

2 Halting a continuous play of an eDrawing animation:___________________________

3 Command that allows you to step backwards one step at a time through an eDrawing animation:____

4 Non-stop replay of eDrawing animation:_____________________________________

5 Rendering of 3D parts with realistic colors and textures: _________________________

6 Go forward one step in an eDrawing animation: ______________________________

7 Command used to create an eDrawing: _____________________________________

8 Graphic aid that allows you to see the model orientation in an eDrawing created from a SolidWorks drawing: ____

9 Quickly return to the default view:____

10 Command that allows you to use email eDrawings with others: __________________

Lesson 9 Quiz

Name: ________________________________Class: _________ Date:________________

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

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

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

3 How do you create an eDrawing? ____

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

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

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

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

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

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

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

11 What is PhotoWorks?
____

12 What is SolidWorks MotionManager?
____

13 Where do you modify the scene background?
____

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

____

15 True of False. PhotoWorks 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.
- PhotoWorks creates photorealistic renderings of parts and assemblies.
- PhotoWorks allows you to add materials, scenes, decals and lights to create images that look like photographs.
- PhotoWorks 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.

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Glossary

animate View a model or eDrawing in a dynamic manner. Animation simulates motion or displays different views.

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.
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.

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).

PropertyManager 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.