

# SolidWorks®

## Engineering Design Project The Mountainboard



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

Outside the U.S.: +1-978-371-5011  
Fax: +1-978-371-7303  
Email: [info@solidworks.com](mailto:info@solidworks.com)  
Web: <http://www.solidworks.com/education>

© 1995-2006, SolidWorks Corporation

300 Baker Avenue  
Concord, Massachusetts 01742 USA  
All Rights Reserved

U.S. Patents 5,815,154; 6,219,049; 6,219,055;  
6,603,486; 6,611,725; and 6,844,877 and certain  
other foreign patents, including EP 1,116,190 and JP  
3,517,643. U.S. and foreign patents pending.

SolidWorks Corporation is a Dassault Systemes  
S.A. (Nasdaq:DASTY) company.

The information and the software discussed in this  
document are subject to change without notice and  
should not be considered commitments by  
SolidWorks Corporation.

No material may be reproduced or transmitted in any  
form or by any means, electronic or mechanical, for  
any purpose without the express written permission  
of SolidWorks Corporation.

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

SolidWorks, PDMWorks, and 3D PartStream.NET,  
and the eDrawings logo are registered trademarks of  
SolidWorks Corporation.

SolidWorks 2006 is a product name of SolidWorks  
Corporation.

COSMOSXpress, DWGeditor, DWGgateway,  
eDrawings, Feature Palette, PhotoWorks, and  
XchangeWorks are trademarks, 3D ContentCentral  
is a service mark, and FeatureManager is a jointly  
owned registered trademark of SolidWorks  
Corporation.

COSMOS, COSMOSWorks, COSMOSMotion, and  
COSMOSFloWorks are trademarks of Structural  
Research and Analysis Corporation.

FeatureWorks is a registered trademark of  
Geometric Software Solutions Co. Limited.

ACIS is a registered trademark of Spatial  
Corporation.

GLOBEtrrotter and FLEXIm are registered  
trademarks of Globetrotter Software, Inc.

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

## COMMERCIAL COMPUTER SOFTWARE - PROPRIETARY

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

Contractor/Manufacturer:  
SolidWorks Corporation, 300 Baker Avenue,  
Concord, Massachusetts 01742 USA

Portions of this software © 1988, 2000 Aladdin  
Enterprises.

Portions of this software © 1996, 2001 Artifex  
Software, Inc.

Portions of this software © 2001 artofcode LLC.

Portions of this software © 2005 Bluebeam  
Software, Inc.

Portions of this software © 1999, 2002-2006  
ComponentOne

Portions of this software © 1990-2006 D-Cubed  
Limited.

Portions of this product are distributed under license  
from DC Micro Development, Copyright © 1994-  
2002 DC Micro Development, Inc. All rights  
reserved

Portions © eHelp Corporation. All rights reserved.

Portions of this software © 1998-2006 Geometric  
Software Solutions Co. Limited.

Portions of this software © 1986-2006 mental  
images GmbH & Co. KG

Portions of this software © 1996 Microsoft  
Corporation. All Rights Reserved.

Portions of this software © 2005 Priware Limited

Portions of this software © 2001, SIMULOG

Portions of this software © 1995-2006 Spatial  
Corporation.

Portions of this software © 2003-2006, Structural  
Research & Analysis Corp.

Portions of this software © 1997-2006 Tech Soft  
America.

Portions of this software are copyrighted by and are  
the property of UGS Corp. © 2006.

Portions of this software © 1999-2006 Viewpoint  
Corporation.

Portions of this software © 1994-2006, Visual  
Kinematics, Inc.

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

All Rights Reserved

# Contents

---

<b>Introduction</b>	<b>1</b>
<b>Lesson 1: Using the Interface</b>	<b>5</b>
<b>Lesson 2: Basic Functionality</b>	<b>19</b>
<b>Lesson 3: Basic Parts — The Binding</b>	<b>71</b>
<b>Lesson 4: Revolved Features — The Wheel Hub</b>	<b>109</b>
<b>Lesson 5: Thin Features — The Deck</b>	<b>175</b>
<b>Lesson 6: Multibody Parts — The Axle and Truck</b>	<b>225</b>
<b>Lesson 7: Sweeps and Lofts — Springs and Binding</b>	<b>315</b>
<b>Lesson 8: Final Assembly</b>	<b>383</b>
<b>Lesson 9: Presenting Results</b>	<b>429</b>
<b>Glossary</b>	<b>511</b>





# 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.

## Introducing SolidWorks

The printed version of *Introducing SolidWorks* is provided with the Education Edition of the SolidWorks software. It provides additional material that can be used to supplement this manual.

## 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, Online Tutorial**. The SolidWorks window is resized and a second window will appear 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:

 Moves to the next screen in the tutorial.

 Represents a note or tip. It is not a link; the information is below 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.
-  **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.



## 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 from the *Companion Files* CD.

### Resources for This Lesson

---

- ❑ *Introducing SolidWorks*, Chapter 1.

### 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.

---

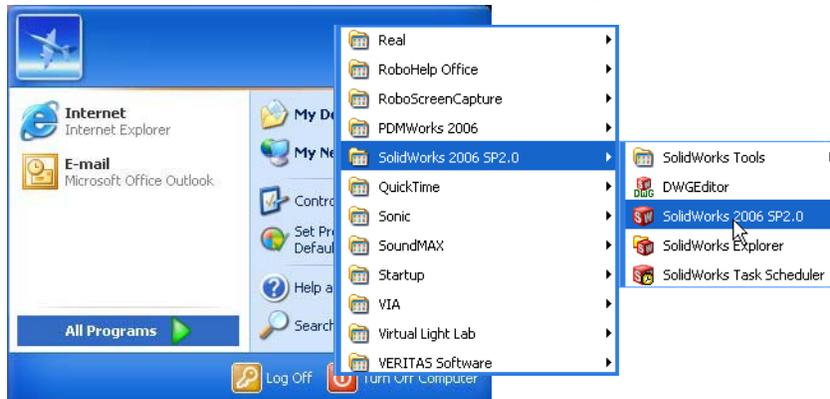
- 2 From the **Start** menu, click **All Programs, SolidWorks 2006, SolidWorks 2006** as shown below.

---

**Note:** Depending on how SolidWorks was installed on your computer, the Service Pack number, SP2.0 in this case, may not be 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.

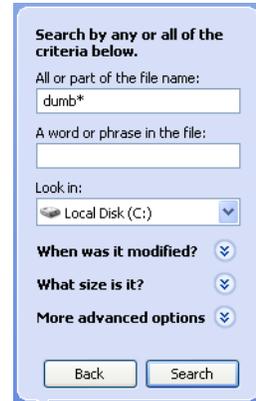
- 3 Click **Start, Search**. Search for the SolidWorks part `dumbell`. To do this click **All files and folders**, then enter `dumb*` in the **All or part of the file name:** field.

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

- 5 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

- 6 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.

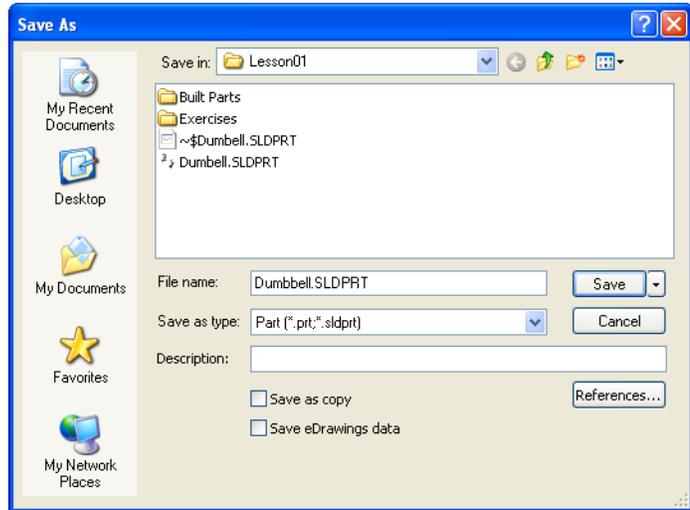
## Copying a File

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.



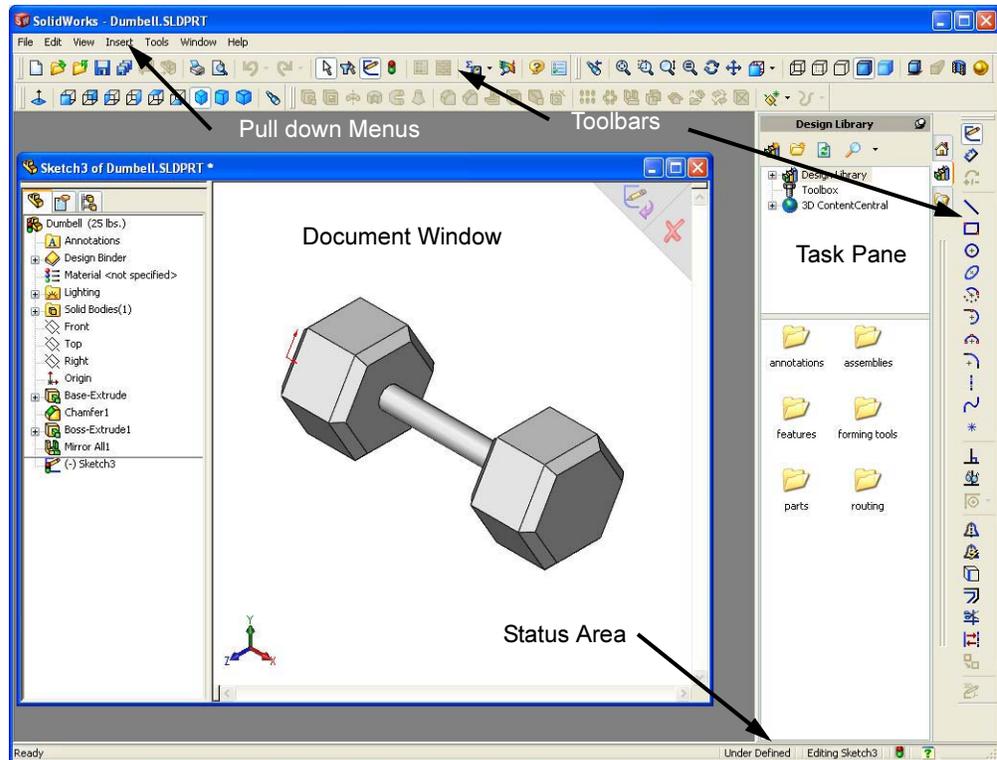
Windows can have multiple panels. You can resize these panels relative to each other.

- 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.

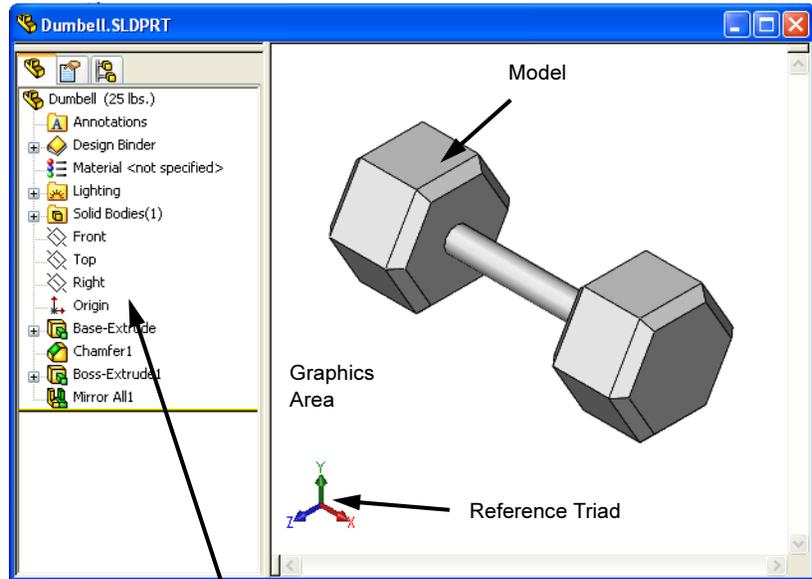
The leftmost panel of the window contains the FeatureManager® design tree, PropertyManager, and ConfigurationManager.

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

## Lesson 1: Using the Interface

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

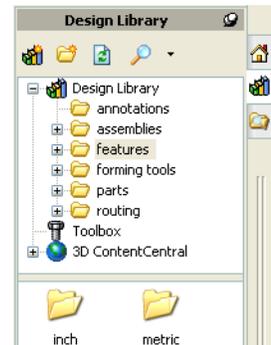
- 2 Look at the Graphics Area. See how the dumbbell is represented. It appears shaded, in color, and in an isometric view. These are some of the ways in which the model can be represented very realistically.



Left panel displaying the FeatureManager design tree

### Taskpane

The SolidWorks Taskpane is a window menu that contains three panels: SolidWorks Resources, the Design Library and the File Explorer. 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.

### 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.

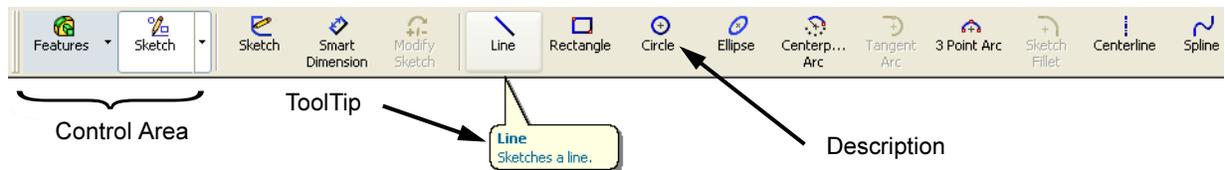


- Click the toolbar name to turn its display on or off. If it is not already on, click **View** to turn the **View** toolbar on.
- Turn several toolbars on and off to see the commands.

## Command Manager

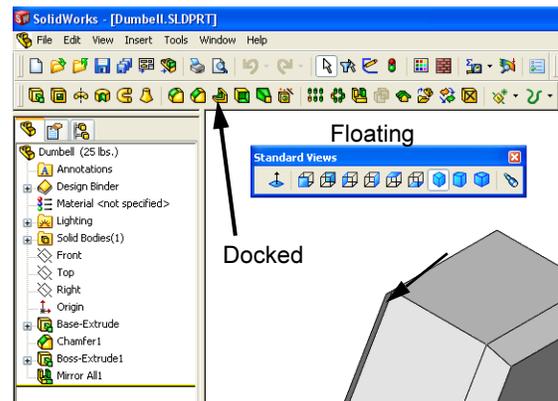
The Command Manager is a multifunction toolbar. Its contents can be adjusted quickly so that it may function in place of several toolbars. We will not use the Command Manager for the first several lessons. This is to make it easier to learn the various functions controlled by the different toolbars.

When you click a button in the control area, the CommandManager updates to show that toolbar. For example, if you click **Sketch** in the control area, the **Sketch** toolbar appears in the CommandManager.



## Arranging Toolbars

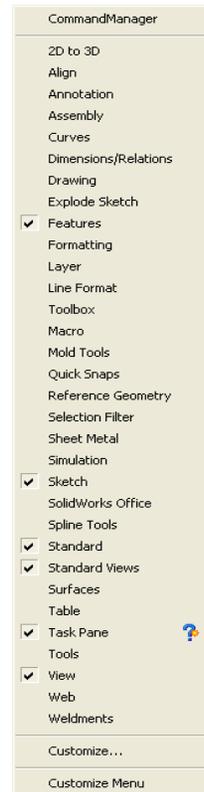
Toolbars may be positioned anywhere on the screen. If a toolbar displays its name, then it is floating and can be positioned anywhere on the screen. If a toolbar is positioned around the edge of the screen and is not displaying its name, it is docked.

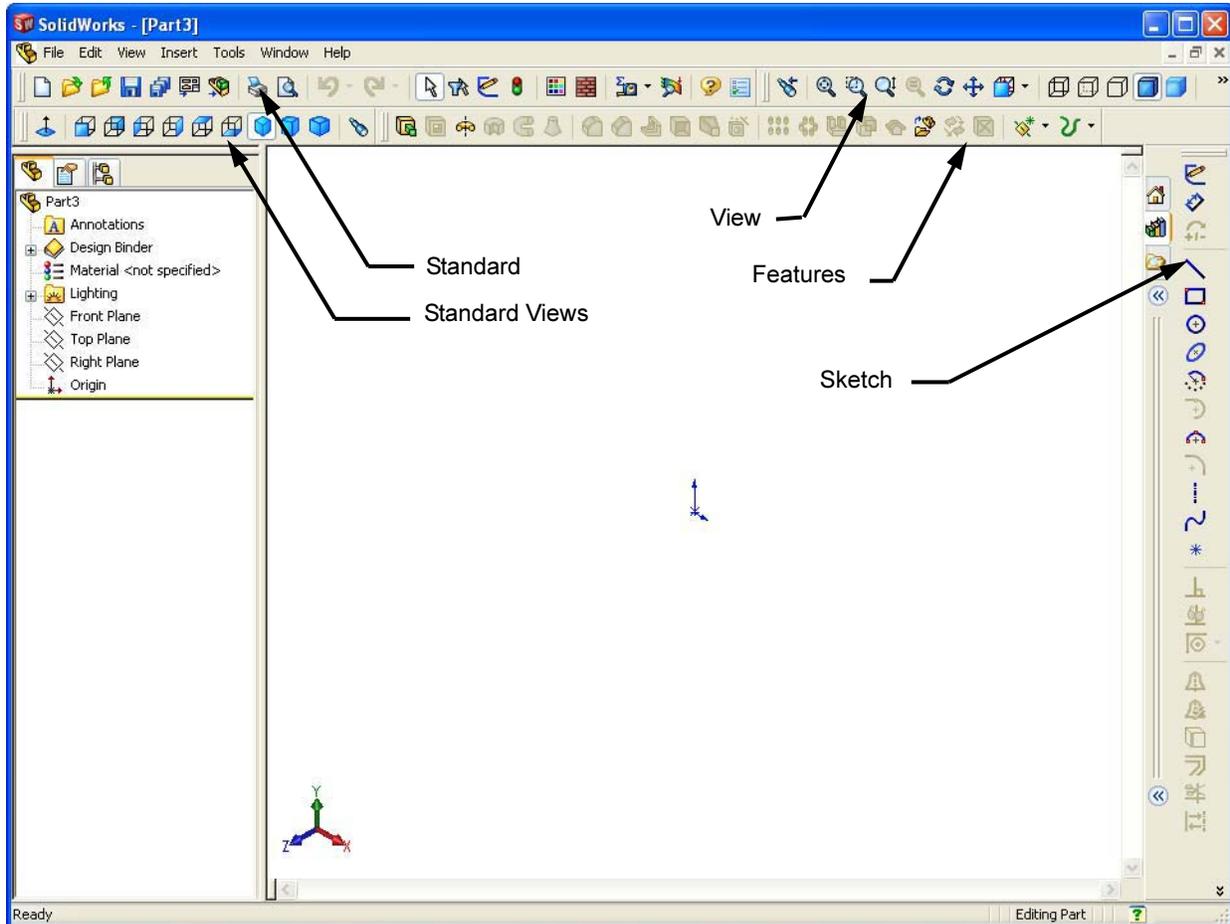


## Position the Toolbars

To make sure everyone's view of SolidWorks is the same, we will set the initial toolbars and their locations.

- 1 Click **View, Toolbars**.
- 2 Select the following toolbars:
  - **Features**
  - **Sketch**
  - **Standard**
  - **Standard Views**
  - **Task Pane**
  - **View**
- 3 Clear the **Command Manager**.
- 4 Drag each toolbar and dock them as shown below. To drag a toolbar that is docked, you drag it by the left end with the vertical bars.





## 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 shortcut menu of commands that are appropriate for wherever you clicked.

You can access the "more commands menu" by selecting the double-down arrows  in the menu. When you select the double-down arrows or pause the pointer over 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.

## 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.

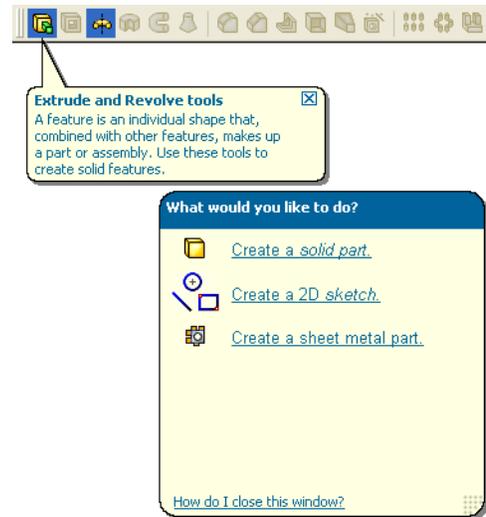
---

- 1 Click  or **Help, SolidWorks Help Topics** in the menu bar.  
The online help appears.
- 2 Click  on the Standard toolbar, then click a toolbar icon or a FeatureManager item.  
**What's This? help** appears in a new window.

## 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.

## 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 Mountain Board 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 Animator, 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.



## Lesson 2: Basic Functionality

### 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.

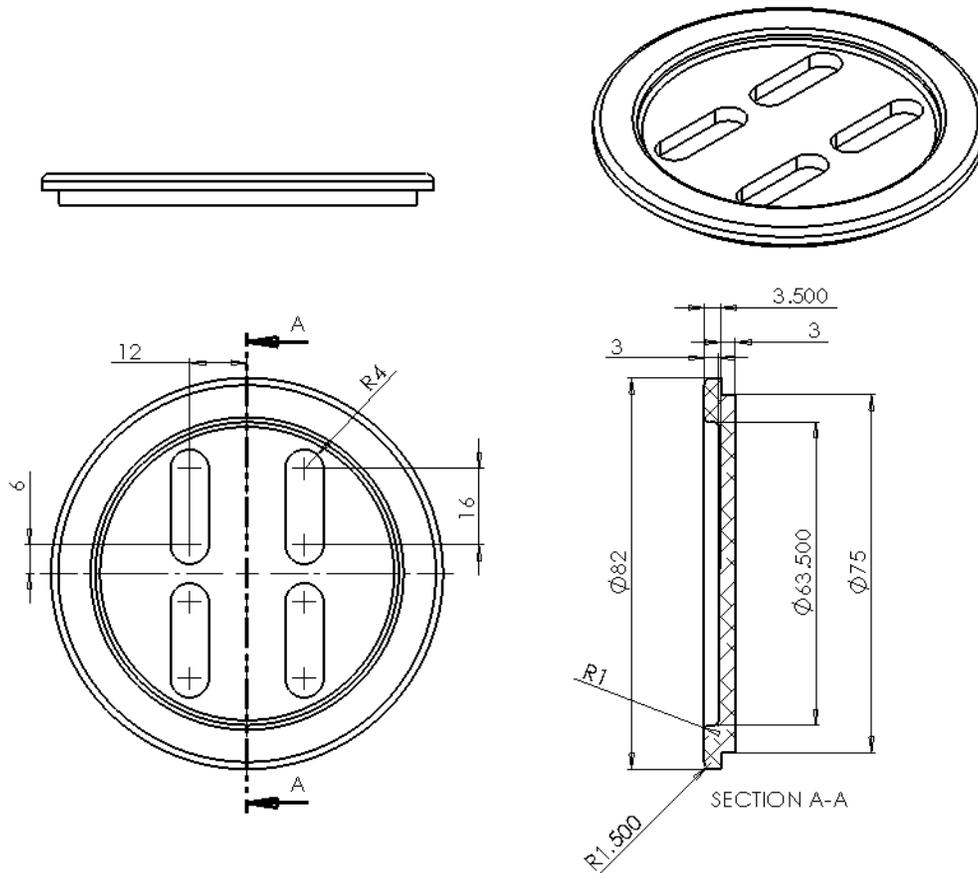


### Design Intent

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.



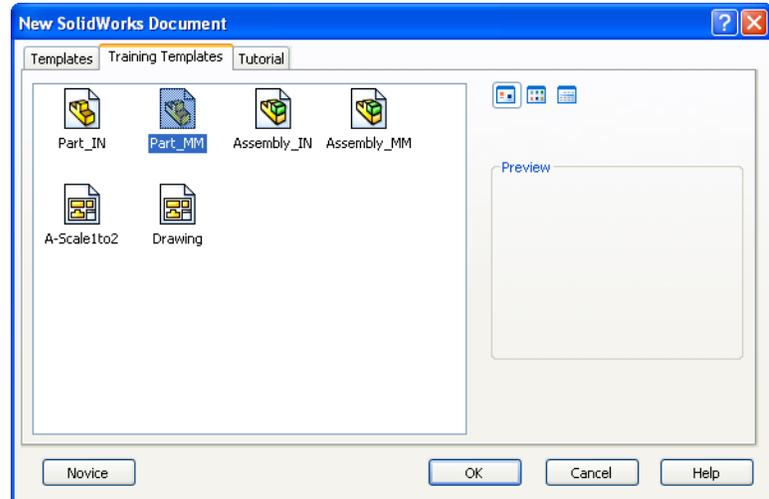
### Task 1— Create a New Part Document

- 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.

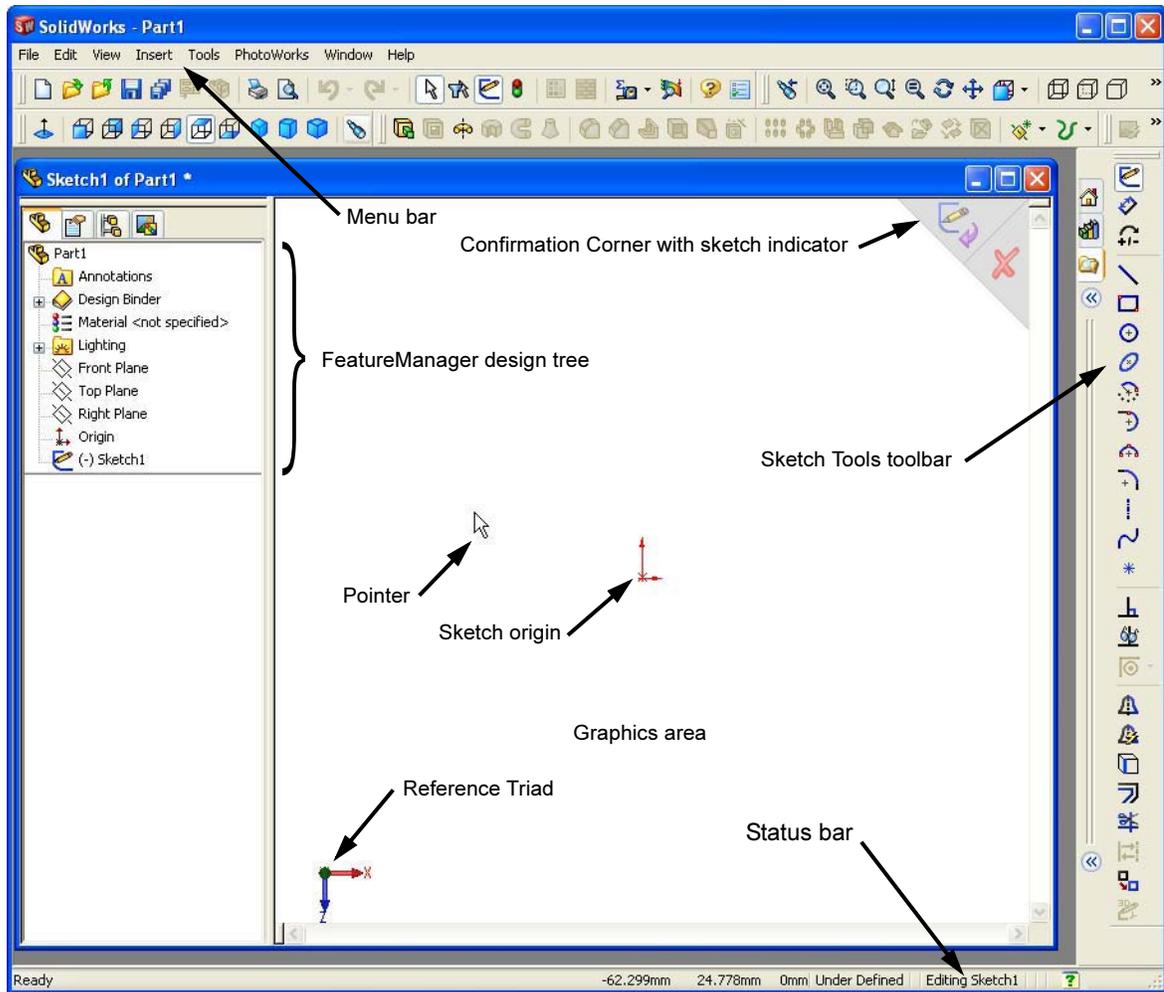


### Overview of the SolidWorks Window

When you create a new sketch:

- A sketch origin appears in the center of the graphics area.
- The Sketch toolbar is displayed.
- “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.

## Lesson 2: Basic Functionality



### First Feature

The first feature requires:

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

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

The sketch opens on the Top plane.

## View Orientation

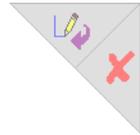
When we open a sketch for the first feature, SolidWorks will automatically change the view orientation to be normal to the sketch plane. This makes it easier to see the sketch. It is like looking straight down on a piece of paper.

## Confirmation Corner

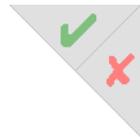
When many SolidWorks commands are active, a symbol or a set of symbols appears in the upper right corner of the graphics area. This area is called the **Confirmation Corner**.

## 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

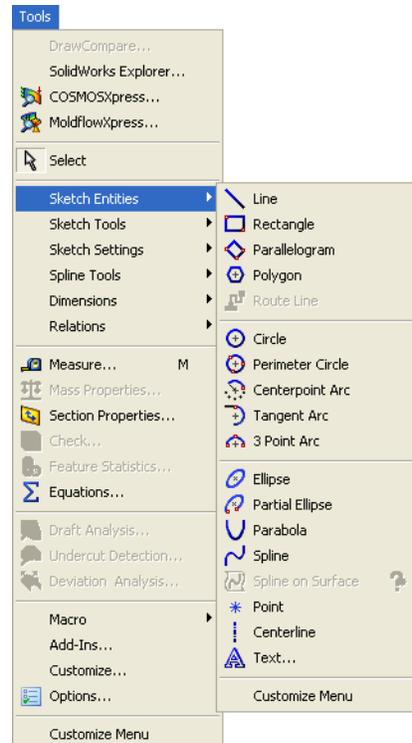
SolidWorks provides a variety of tools to create sketches. They can be found on both the **Sketch Entity** menu and most can also be found on the **Sketch** toolbar.

## 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.



## Task 2 — Create the first sketch

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

### The Circle

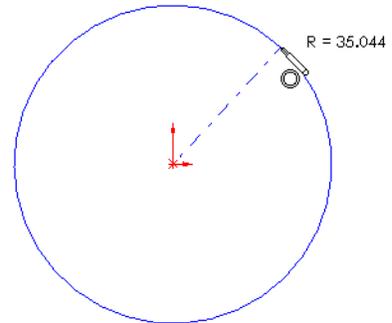
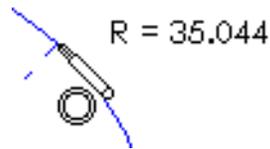
The circle tool  creates 2D circles. Using the mouse, press the left mouse button at the location for the center of the circle, then (holding down the left mouse button) drag until the circle is approximately the correct size. Release the left mouse button.

1 Click the **Circle** tool  from the Sketch toolbar. The cursor 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 32 mm. The cursor feedback will show the radius of the circle.

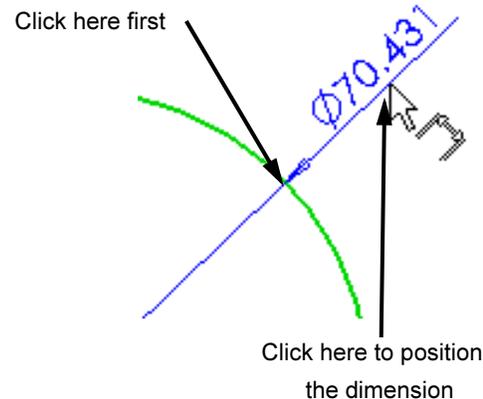


### Dimension the sketch

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

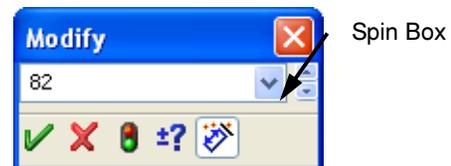
1 Click **Smart Dimension**  on the **Sketch** toolbar. The cursor will look like this,  indicating that the dimension tool is active.

2 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.



3 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 to the sketch plane.

### Task 3 — Extrude the first feature

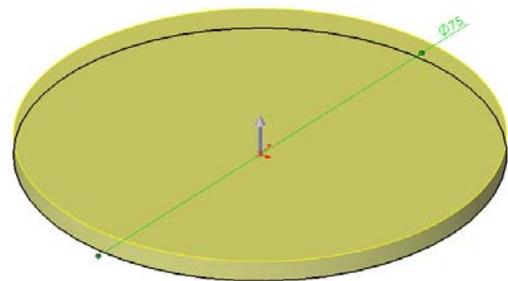
Extruding the 2D sketch will produce a 3D solid. In this case, we will make a short cylinder.

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

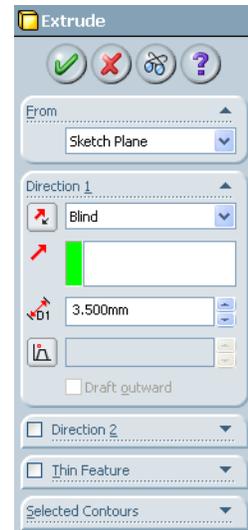
- 2 Preview graphics.

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.



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



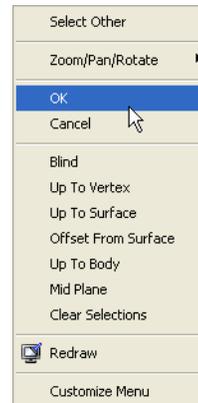
**TIP:**

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.



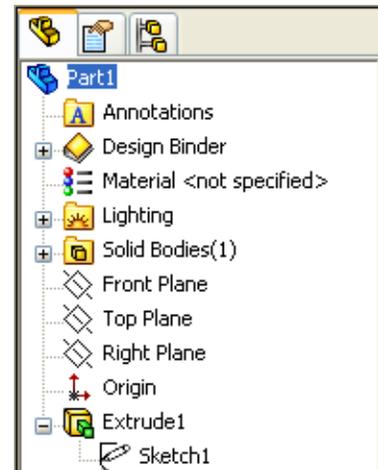
### Blind Extrusions

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 `Extrude1`.

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



## View Display

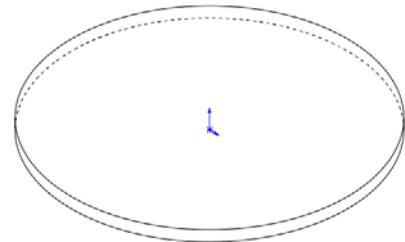
The View toolbar provides a quick method to change the way the model is displayed on the screen. It provides one set of tools to Zoom, Pan and Rotate the model view and another to change the way the model is displayed. In most cases, models are created in Shaded view because it most closely resembles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



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 lose everything you did since the last time you saved your work.

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

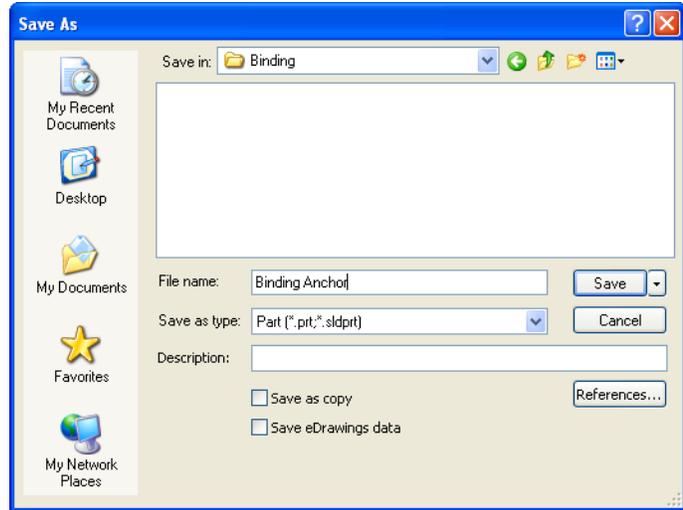
The **Save As** dialog box appears.

## Lesson 2: Basic Functionality

- 2 Type **Binding Anchor** for the filename.
- 3 Save the file to the directory **Binding** found under ...\**SolidWorks Curriculum and Courseware\_2006-2007\Mountainboard Design Project\Mountainboard**.
- 4 Click **Save**.

The **.sldprt** extension is automatically added to the filename.

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



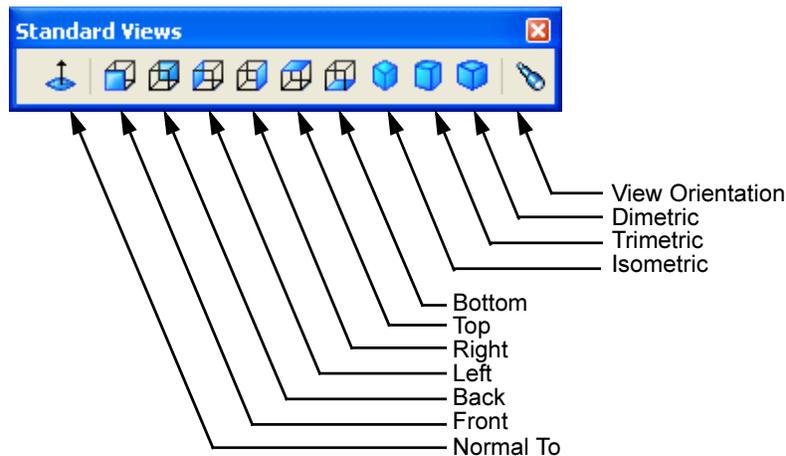
---

**Note:** All the files we create of the Mountainboard project should be saved in the appropriate folder under the folder ...\**SolidWorks Curriculum and Courseware\_2006-2007\Mountainboard Design Project**.

---

## Changing views

The Standard Views toolbar makes it easy to change your view of the model by simply clicking on the view you would like to see.



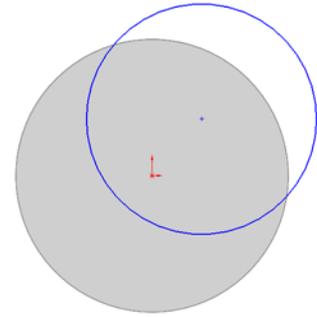
- 5 Change the view of the model to the **Bottom** view. Click  on the **Standard Views** toolbar.

## Task 4 — Add a second feature

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

- 1 Select the bottom face of the cylinder. It will turn green to show that it is selected.
- 2 Start a new sketch by clicking the **Sketch**  on the Sketch toolbar.

- 3 Select **Circle**  on the Sketch toolbar.
- 4 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.

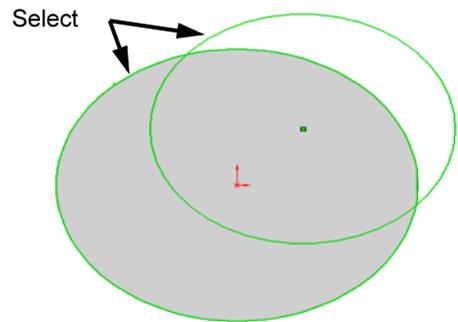


### Task 5 — Adding sketch relationships

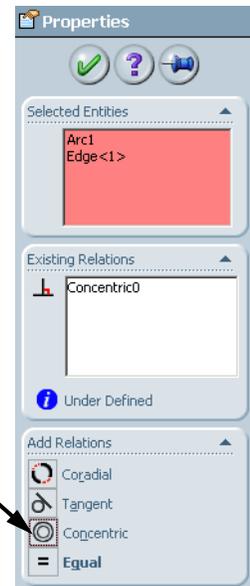
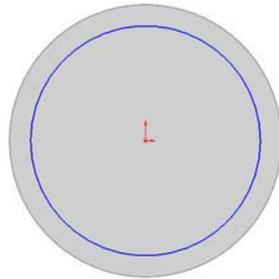
Sketch relationships are used to force a behavior on a sketch element to capture design intent. Some are automatic, others can be added as needed.

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.
- 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.



### Task 6 — Dimension the circle

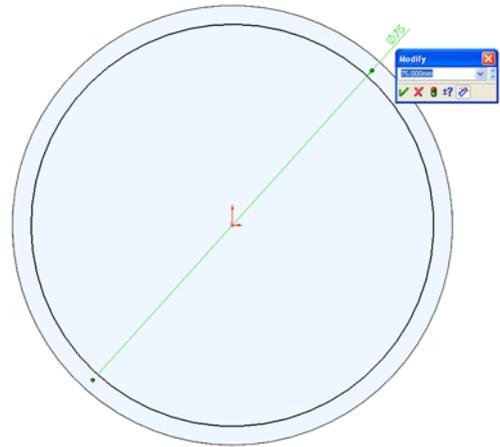
The concentric relationship defines the position of the circle, but it is still blue (under defined) because it doesn't have a dimension for its diameter.

- 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 **OK**. 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**

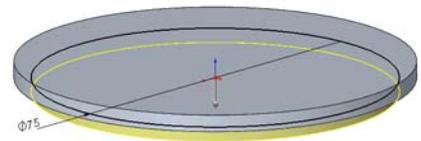


### Change the viewpoint

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 Standard Views toolbar.
- 2 Click **Extrude** on the Features toolbar.
- 3 Select **Blind** for the type of extrusion.
- 4 Type **3 mm** 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 **OK**.



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

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

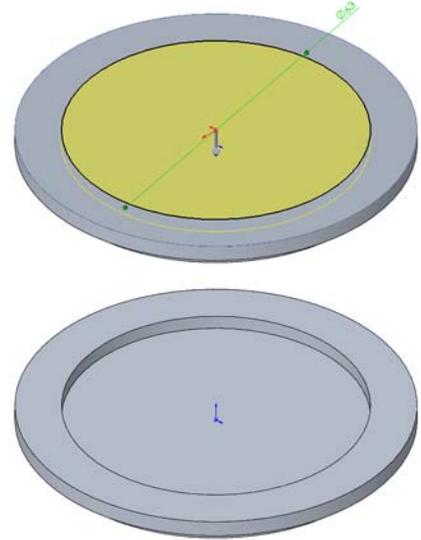
- Reduce weight. Each part must be designed to be as light as possible so that the assembled mountainboard is not too heavy to be carried.
- Lower the tops of the screws used to bolt this part to the deck. This will reduce the chance of anything (pants leg, shoe laces, etc.) getting caught on the screw 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 **Standard Views** 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 **30 mm**.
- 5 Dimension the circle to be **63 mm** 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 **3 mm** for the depth.



- 10 Click **OK**. 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.

$$\square \text{ Weight} = \text{Volume} \times \text{Density}$$

### Calculate the volume

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

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

### 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:

$$2.8 \text{ g/cc} \times .001 \text{ cc/mm}^3 = \mathbf{.0028 \text{ g/mm}^3}$$

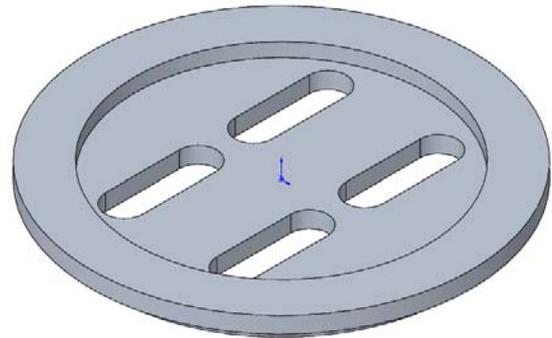
### Calculate the weight

$$\text{Weight} = \text{Volume} \times \text{Density} = 22,385.42 \text{ mm}^3 \times .0028 \text{ g/mm}^3 = \mathbf{62.68 \text{ grams (2.21 oz)}}.$$

### Task 9 — Create the screw slots

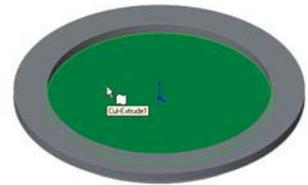
To make the position of the binding adjustable, the binding anchor will have four slots. These will allow the position of the bindings to be moved along the centerline of the deck.

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 **Standard Views** 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.

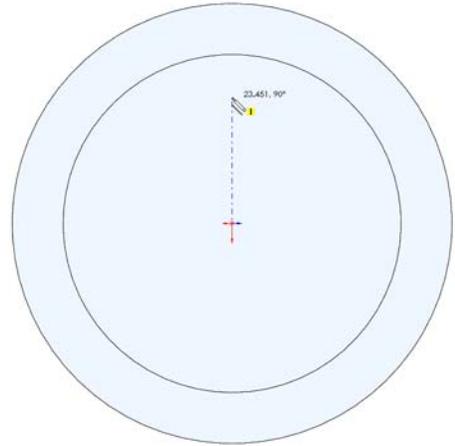
#### Lines and Centerlines

The Line tool draws straight lines. If the line is vertical, the cursor will show a  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.

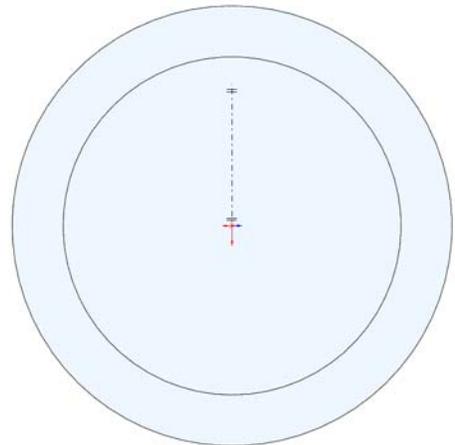
Centerlines are construction geometry. They are used to position other entities but do not result in features.

### Create a centerline

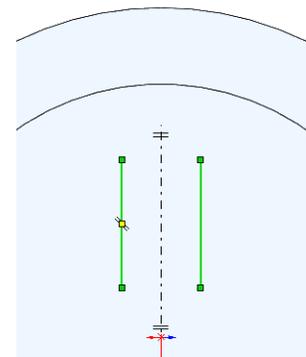
- 1 Click **Centerline**  on the Sketch toolbar.
- 2 Sketch a vertical **Centerline**. The length is not important. Make sure the cursor displays a , indicating a **Vertical** relationship will be added.



- 3 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.



- 4 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.



### 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.

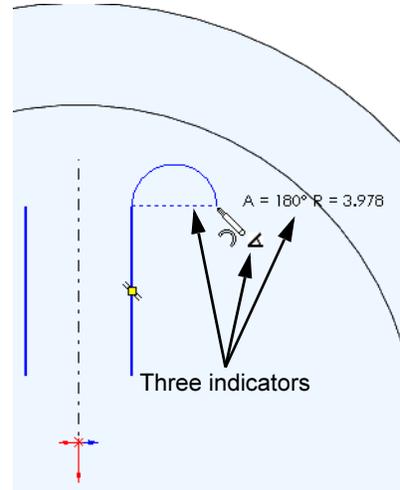
### Add an Arc

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

- 1 Click **Tangent Arc**  the **Sketch** toolbar.
- 2 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.

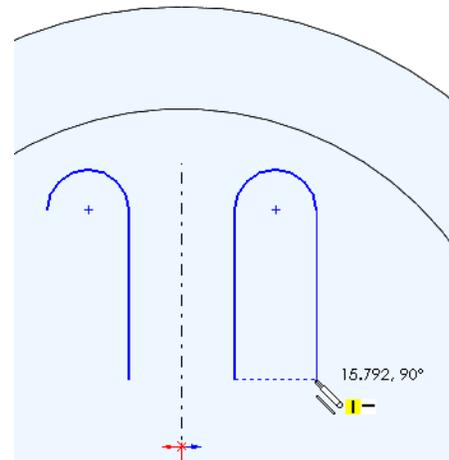
There will be three indicators that you have gone 180 degrees:

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

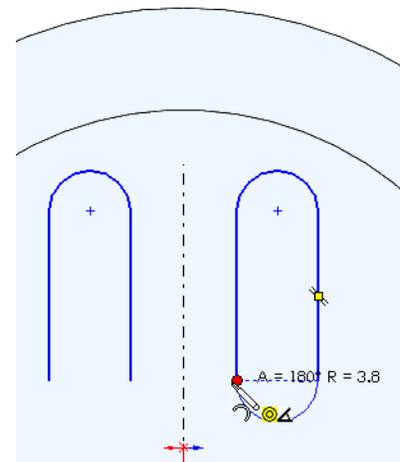


When you release the mouse button, a second symmetric arc will be drawn automatically.

- 3 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.



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



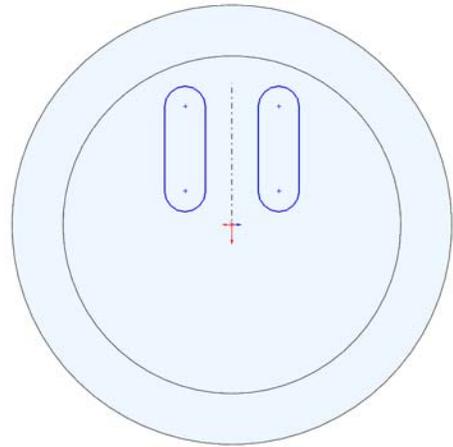
### Review the progress

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 by the mirror tool will make these sketch elements retain the symmetry we desire.

### 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. You will need to hold down the **Control** key to select multiple objects.

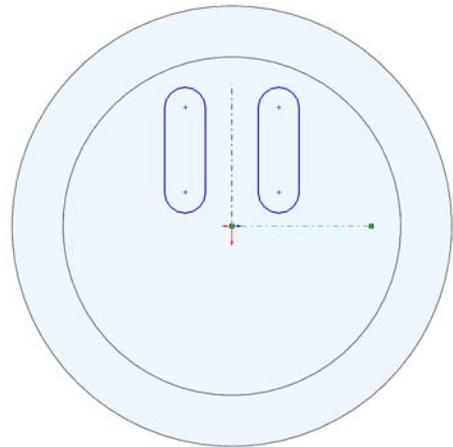
Once everything is selected, click **Mirror Entities** .



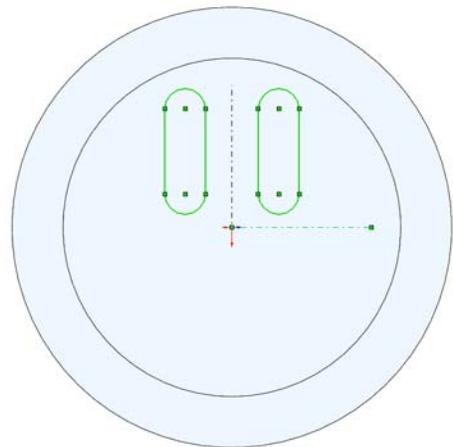
### Task 10 — Mirror the two slot sketches

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

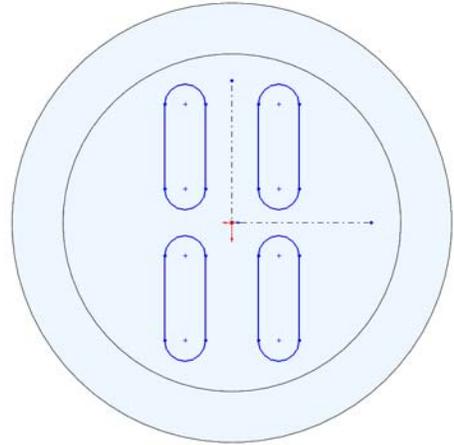


- 3 Hold down the **Control** key and select the four lines and four arcs plus the horizontal centerline. Make sure you *do not* have the vertical centerline selected.



4 Mirror the geometry by clicking **Mirror Entities**

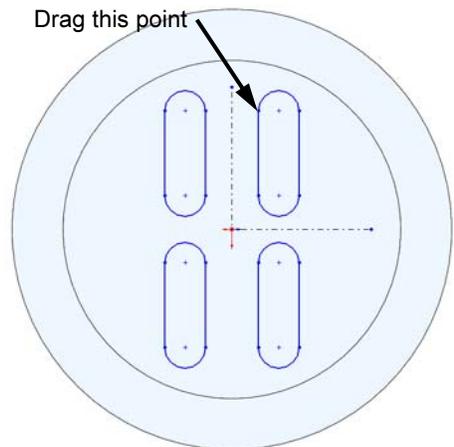
 on the Sketch toolbar.



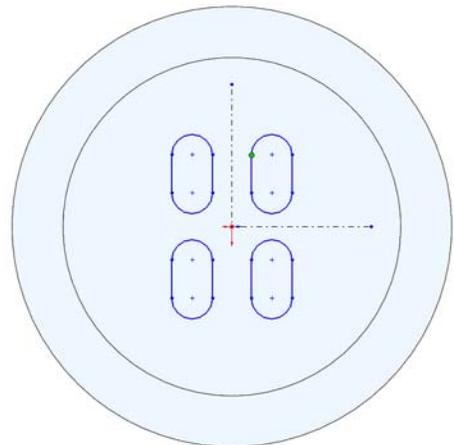
**Test the relationships**

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

Drag this point



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



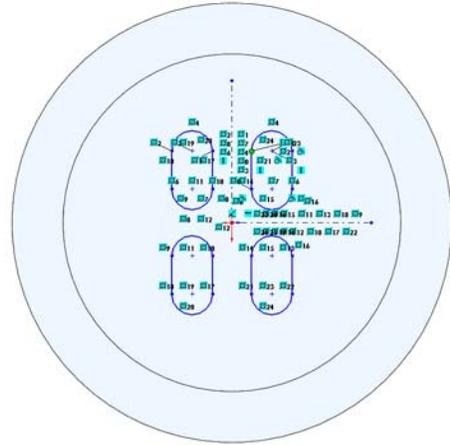
## View Relationships

Existing relationships can be displayed by selecting **View, Sketch Relations** from the menu. All existing relationships will be shown by callouts.

### 1 Click **View, Sketch Relations**.

Callouts are displayed showing that the arc is 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.

- 2 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**.

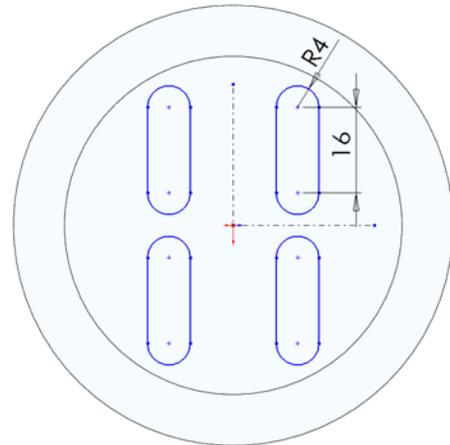


## Task 11 — Add dimensions

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.

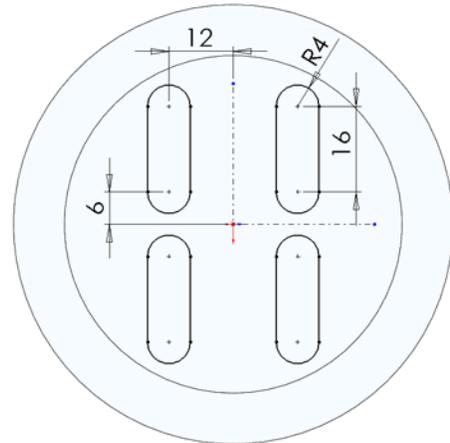
To add the **16 mm** dimension, select the two arcs, not the vertical line.



### 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

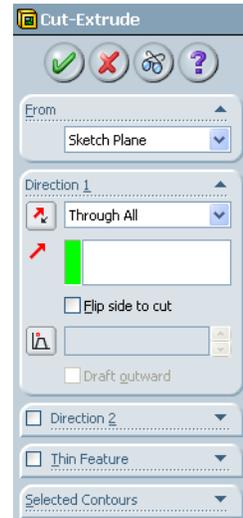
The four slots must cut completely through the Binding Anchor. When we create the cut, it must be done so that if we need to change the thickness of the material later in the design process, the slots do not have to be redone.

- 1 Click **Insert, Cut, Extrude** from the menu.

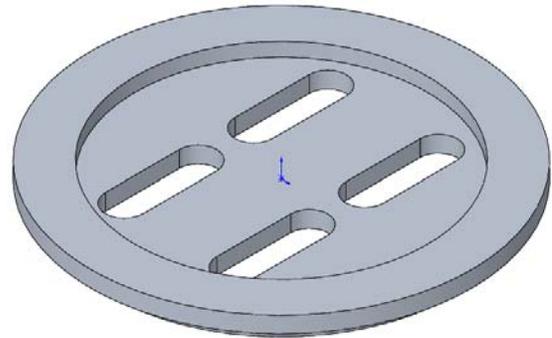
- 2 Click  on the **View** toolbar to change the view to **Isometric**.
- 3 From the list in the PropertyManager select **Through All**.

### 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 be 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 **OK**.



### 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 `CutExtrude2`, 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.

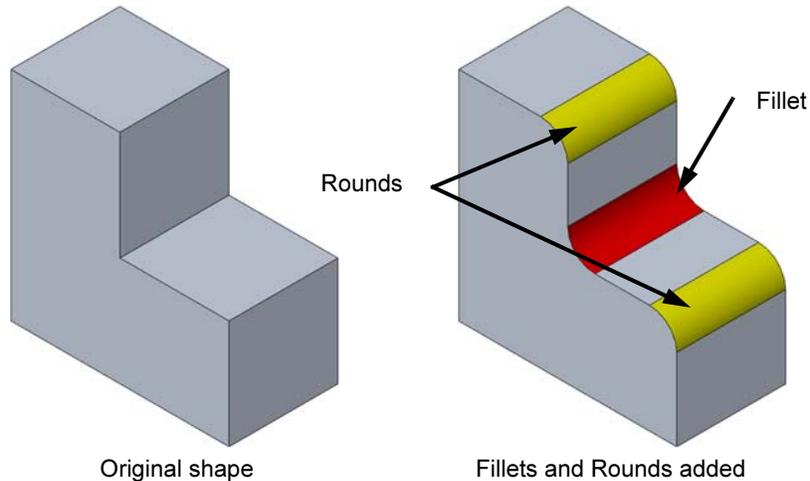


Notice that the feature's icon does not change. The  shows that this feature is a **Cut Extrude**.

## 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.

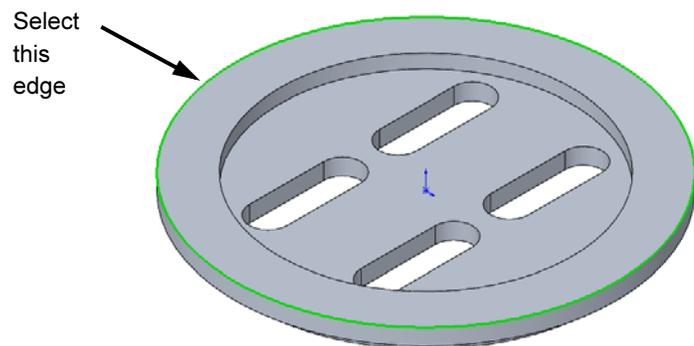
Both fillets (adding volume), shown in red, and rounds (removing volume), shown in yellow, are created with this command. The orientation of the edge or face determines which is used.



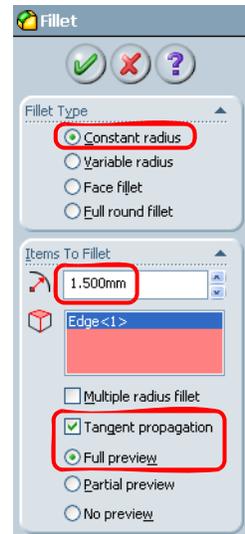
### Task 14 — Round an outside corner

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

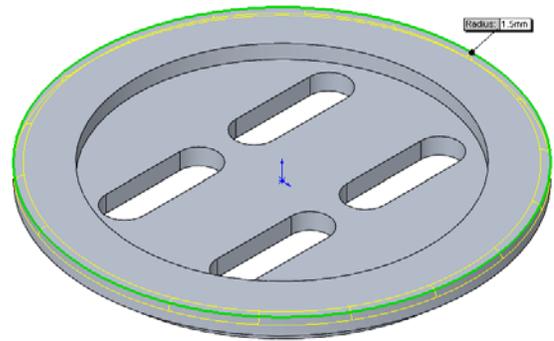
- 1 Click **Fillet**  on the **Features** toolbar.
- 2 Select the edge shown.



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



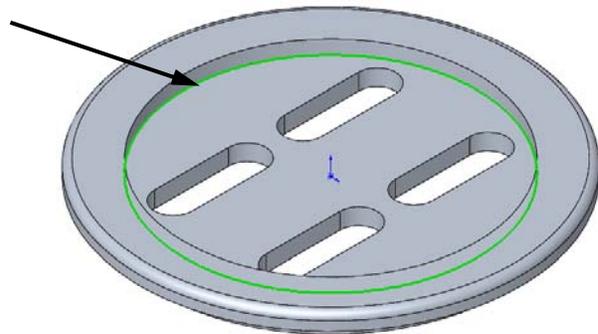
- 5 Preview. Once **Full preview** is selected, the outline of the fillet will be shown in yellow. The fillet radius will be shown in a callout, **Radius: 1.5mm** attached to the edge. Click **OK**.



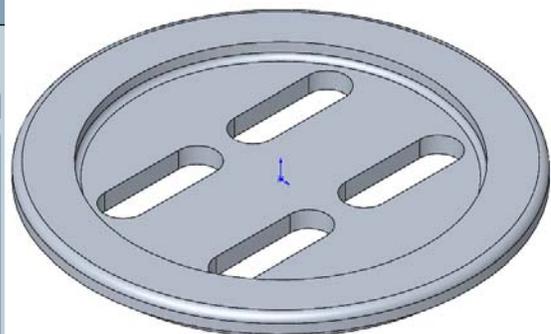
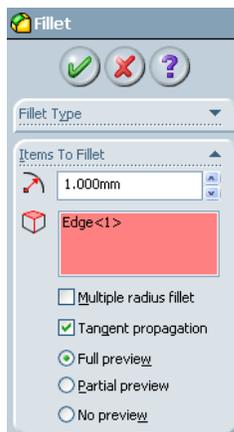
**Task 15 — Add a fillet to the inside edge**

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

Select this edge



- 3 Type **1 mm** for the fillet radius.
- 4 Click **OK**. 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.

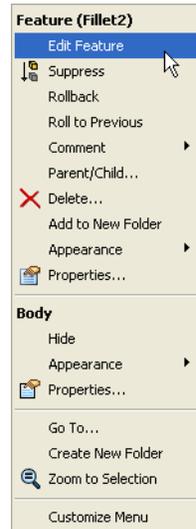
For the fillet just created, we could add additional edges to the feature or change the fillet radius.

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

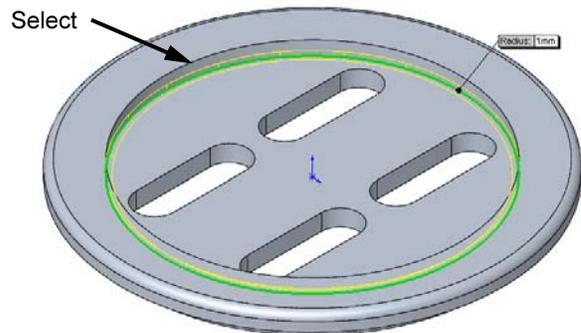
### Task 16 — Edit the Fillet

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

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



- 2 Select the edge shown.



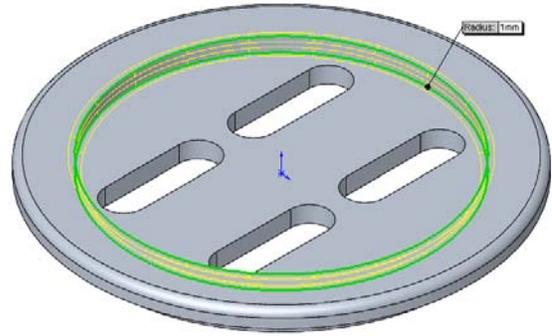

---

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

---

## Lesson 2: Basic Functionality

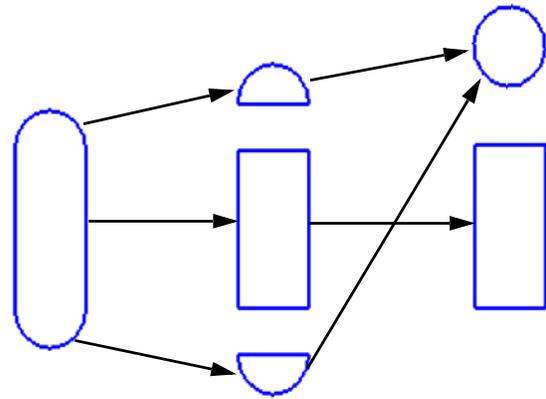
- 3 Once selected, the preview will show that the edge will be rounded as part of the `Fillet2` feature. Click **OK**.
- 4 **Save** the part.



### How much does it weight now?

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

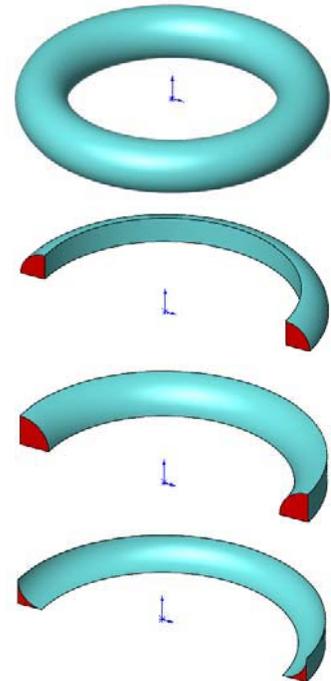
.The volume removed by the slots is not too difficult as the area of each slot can be thought of as a rectangle and circle



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.

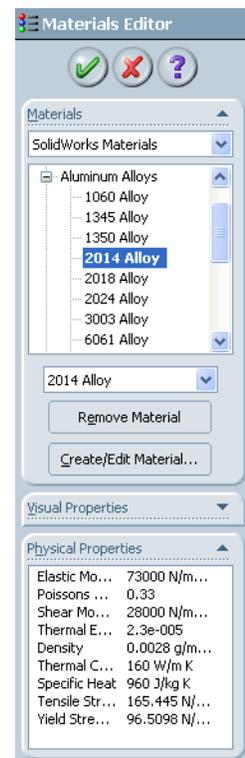
- 1 Click **Edit, Appearance, Material** from the menu.  
The **Materials Editor** will open in the PropertyManager.
- 2 Click the Plus sign next to **Aluminum Alloys** to expand the list.
- 3 Select **2014 Alloy**.

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

---

**Note:** Certain graphics cards support RealView advanced graphics visual properties, if installed, but not all. To find out if yours does, consult the **Help** documentation inside SolidWorks.

---



- 4 Click **OK** 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.

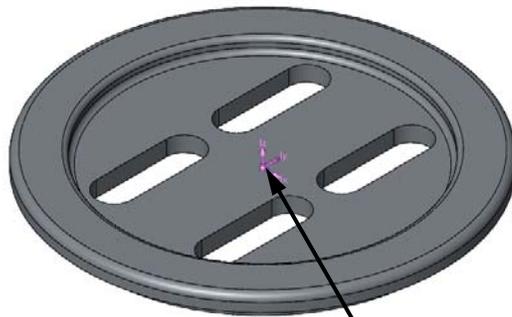
The Mass Properties box will appear.

The Volume is calculated to be **19,765.727 mm<sup>3</sup>** and the Mass is **55.344 grams**.

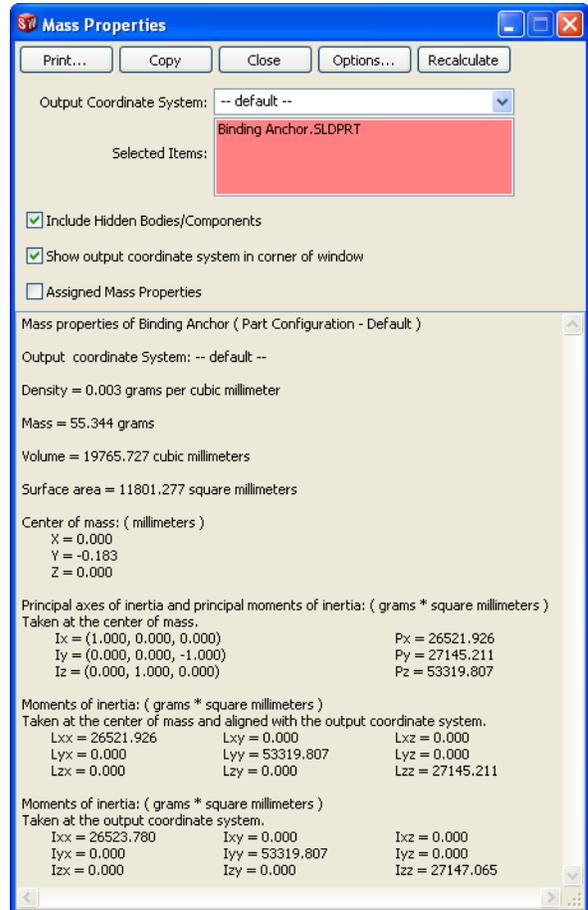
When we calculated the weight earlier it was 62.68 grams. This was before we removed material with the four slots and two rounds.

- 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.



Center of Mass



**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.

- 1 Click the **Options** button.
- 2 Select **Use custom settings**. Select:
  - Length - **Inches**
  - Mass - **Pounds**
  - Per unit volume - **inches<sup>3</sup>**

- 3 Click **OK**.

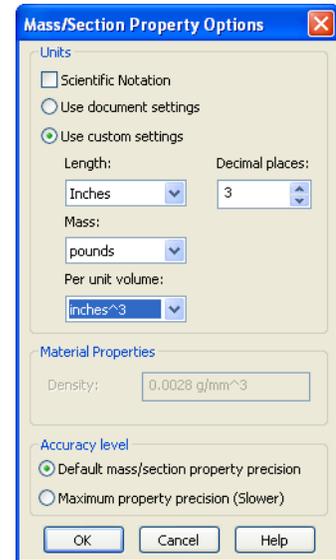
.

---

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

---

- 4 Click **Close** to close the Mass Properties.
- 5 **Save** the part.



```
Output coordinate System: -- default --
Density = 0.101 pounds per cubic inch
Mass = 0.122 pounds
Volume = 1.206 cubic inches
Surface area = 18.292 square inches
Center of mass: ( inches )
X = 0.000
Y = -0.007
Z = 0.000
```