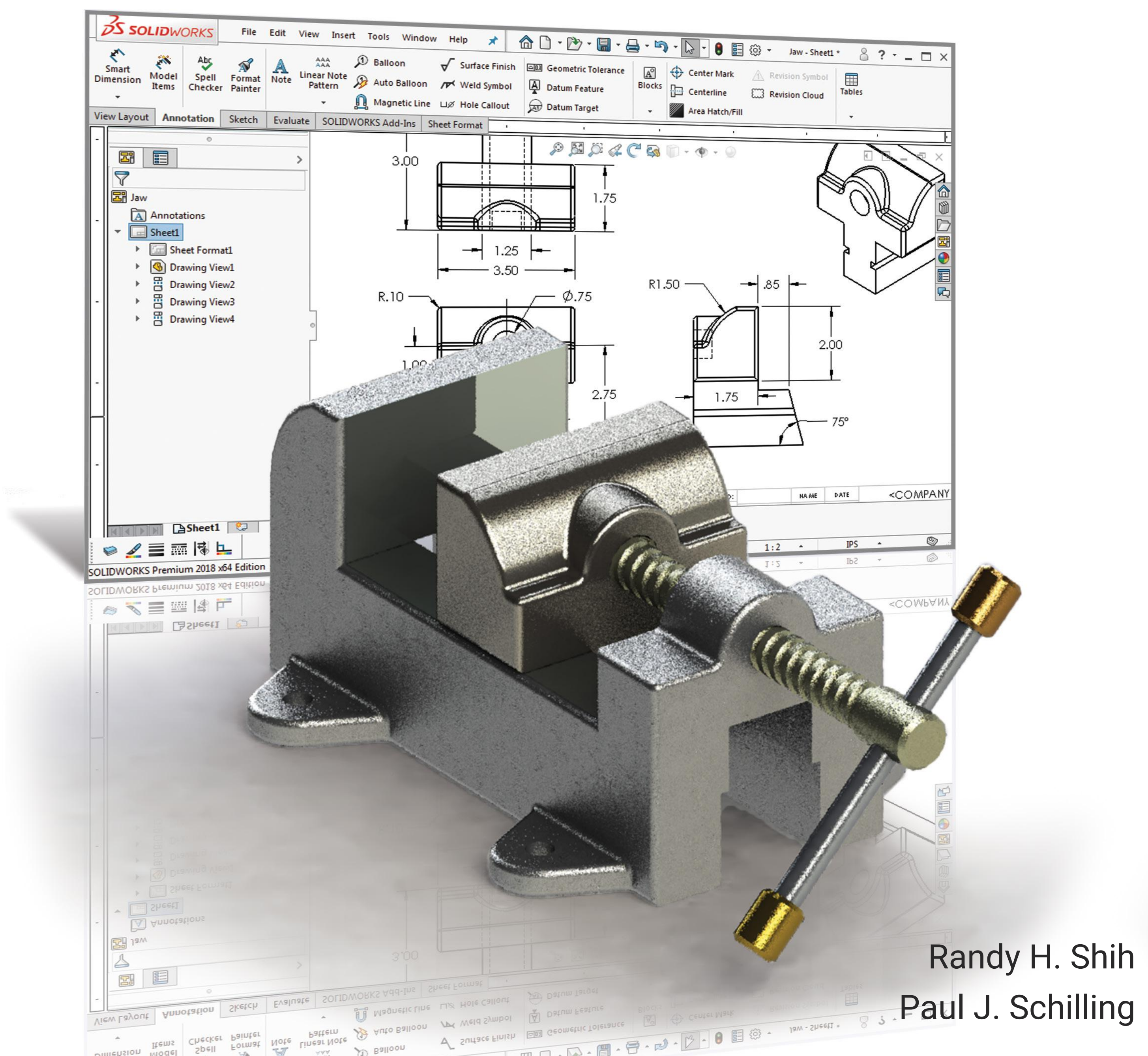
Parametric Modeling with SOLIDWORKS® 2019

Covers materials found on the CSWA exam



Randy H. Shih

Paul J. Schilling

SDC PUBLICATIONS

Better Textbooks. Lower Prices.

www.SDCpublications.com

Parametric Modeling with SOLIDWORKS 2019

Covers material found on the CSWA exam

Randy H. Shih
Oregon Institute of Technology

Paul J. Schilling
University of New Orleans

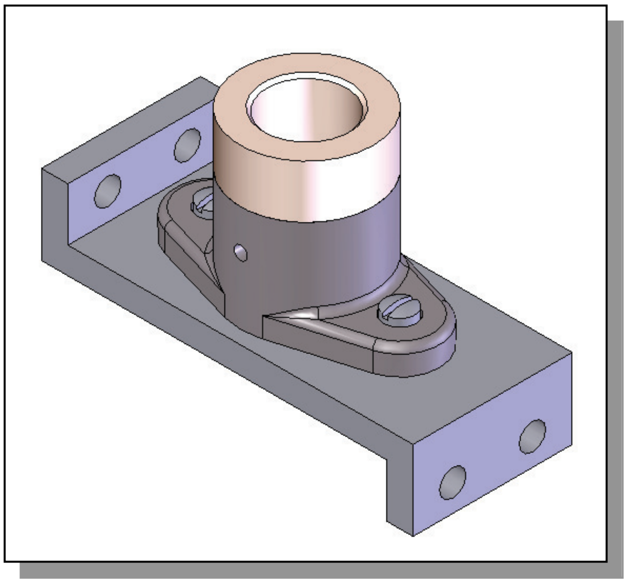
SDC Publications

SDC Publications
P.O. Box 1334
Mission KS 66222
(913) 262-2664
www.SDCpublications.com
Publisher: Stephen Schroff

ISBN-13: 978-1-63057-225-9
ISBN-10: 1-63057-225-X

Printed and bound in the United States of America.

Preface



The primary goal of ***Parametric Modeling with SOLIDWORKS 2019*** is to introduce the aspects of designing with **Solid Modeling** and **Parametric Modeling**. This text is intended to be used as a practical training guide for students and professionals. This text uses SOLIDWORKS 2019 as the modeling tool and the chapters proceed in a pedagogical fashion to guide you from constructing basic solid models to building intelligent mechanical designs, creating multi-view drawings and assembly models. This text takes a hands-on, exercise-intensive approach to all the important *Parametric Modeling* techniques and concepts. This textbook contains a series of seventeen tutorial style lessons designed to introduce beginning CAD users to SOLIDWORKS. This text is also helpful to SOLIDWORKS users upgrading from a previous release of the software. The solid modeling techniques and concepts discussed in this text are also applicable to other parametric feature-based CAD packages. The basic premise of this book is that the more designs you create using SOLIDWORKS, the better you learn the software. With this in mind, each lesson introduces a new set of commands and concepts, building on previous lessons. This book does not attempt to cover all of the SOLIDWORKS features, only to provide an introduction to the software. It is intended to help you establish a good basis for exploring and growing in the exciting field of **Computer Aided Engineering**.

Acknowledgments

This book would not have been possible without a great deal of support. The effort and support of the editorial and production staff of SDC Publications is gratefully acknowledged. I would especially like to thank Stephen Schroff for his support and helpful suggestions during this project. Don Domes of ORTOP granting the use of their designs and resources is also appreciated. I am also very grateful that the Mechanical Engineering Technology Department of Oregon Institute of Technology has provided me with an excellent environment in which to pursue my interests in teaching and research.

Truly unbounded thanks are due to my wife Hsiu-Ling and daughter Casandra for their understanding and encouragement throughout this project.

Randy H. Shih, Spring, 2019

I would like to echo the appreciation for the support of Stephen Schroff and the staff of SDC Publications. In addition, the support of the Department of Mechanical Engineering at the University of New Orleans is gratefully acknowledged.

Paul J. Schilling, Spring, 2019

Table of Contents

Chapter 5
Geometric Relations Fundamentals

Chapter 6
Geometric Construction Tools

Chapter 10
Introduction to 3D Printing

Chapter 11
Symmetrical Features in Designs

Chapter 12
Advanced 3D Construction Tools

Chapter 16
Design Analysis with SimulationXpress

Chapter 17
CSWA Exam Preparation

Appendix

Index

Certified SOLIDWORKS Associate (CSWA) Exam Overview

The Certified SOLIDWORKS Associate (CSWA) Exam is a performance-based exam. The examination is comprised of 10 – 20 questions to be completed in three hours. The test items will require you to use the SOLIDWORKS software to perform specific tasks and then answer questions about the tasks.

Performance-based testing is defined as ***Testing by Doing***. This means you actually perform the given task then answer the questions regarding the task. Performance-based testing is widely accepted as a better way of ensuring the user has the skills needed, rather than just recalling information.

The CSWA examination is designed to test specific performance tasks in the following areas:

Sketch Entities – lines, rectangles, circles, arcs, ellipses, centerlines

Objectives: Creating Sketch Entities.

Certification Examination Performance Task	Covered in this book on Chapter – Page
Sketch Command	2-8
Line Command	2-8
Exit Sketch	2-14
Circle Command, Center Point Circle	2-29
Rectangle Command	3-10
Edit Sketch	4-24
Sketch Fillet	4-25
Centerline	7-6
Tangent Arc	7-9
Centerpoint Arc	7-13
Construction Geometry	11-11
Construction Lines	11-13

Sketch Tools – offset, convert, trim

Objectives: Using Sketch Tools.

Certification Examination Performance Task	Covered in this book on Chapter – Page
Convert Entities	6-25
Offset Entities	6-25

Sketch Relations

Objectives: Using Geometric Relations.

Certification Examination Performance Task	Covered in this book on Chapter – Page

Boss and Cut Features – Extrudes, Revolves, Sweeps, Lofts

Objectives: Creating Basic Swept Shapes.

Certification Examination Performance Task	Covered in this book on Chapter – Page

Certified Associate Reference Guide

Fillets and Chamfers

Objectives: Creating Fillets and Chamfers.

Certification Examination Performance Task | **Covered in this book on Chapter – Page**

Linear, Circular, and Fill Patterns

Objectives: Creating Patterned Features.

Certification Examination Performance Task | **Covered in this book on Chapter – Page**

Dimensions

Objectives: Applying and Editing Smart Dimensions.

Certification Examination Performance Task | **Covered in this book on Chapter – Page**

Feature Conditions – Start and End

Objectives: Controlling Feature Start and End Conditions.

Certification Examination Performance Task	Covered in this book on Chapter – Page
Extruded Boss/Base, Blind	2-15
Extruded Cut, Through All	2-30
Extruded Cut, Up to Next	3-25
Extruded Boss/Base, Mid-Plane	4-13
Extruded Boss/Base, Up to Surface	4-16

Mass Properties

Objectives: Obtaining Mass Properties for Parts and Assemblies.

Certification Examination Performance Task	Covered in this book on Chapter – Page
Mass Properties Tool	4-28
View Mass Properties of a Part	4-29
Relative to Default Coordinate System	17-17
View Mass Properties of an Assembly	17-29
Relative to Reference Coordinate System	17-29

Materials

Objectives: Applying Material Selection to Parts.

Certification Examination Performance Task	Covered in this book on Chapter – Page
Edit Material Command	4-28
Material Properties	16-8
Material in SimulationXpress	16-11

Inserting Components

Objectives: Inserting Components into an Assembly.

Certification Examination Performance Task	Covered in this book on Chapter – Page
Creating an Assembly File	14-8
Inserting a Base Component	14-9
Inserting Additional Components	14-11
Editing Parts in an Assembly	14-32
Inserting Component from Toolbox	15-16

Certified Associate Reference Guide

Standard Mates – Coincident, Parallel, Perpendicular, Tangent, Concentric, Distance, Angle

Objectives: Applying Standard Mates to Constrain Assemblies.

Certification Examination Performance Task	Covered in this book on Chapter – Page
Assembly Mates	14-11
Coincident Mate, using Faces	14-14
Aligned Option	14-14
Anti-Aligned Option	14-14
Coincident Mate, using Temporary Axes	14-14
Coincident Mate, using Planes	14-17
Concentric Mate	14-19
SmartMates	14-21
Angle Mate, using Planes	15-20
Distance Mate	17-23
Angle Mate, using Faces	17-26

Reference Geometry – Planes, Axis, Mate References

Objectives: Creating Reference Planes, Axes, and Mate References.

Certification Examination Performance Task	Covered in this book on Chapter – Page
Reference Axis	9-9
Reference Plane, At Angle Option	9-9
Reference Plane, Offset Distance Option	9-17
Creating Mate References	15-7
Placing Components using Mate References	15-7

Drawing Sheets and Views

Objectives: Creating and Setting Properties for Drawing Sheets; Inserting and Editing Standard Views.

Certification Examination Performance Task	Covered in this book on Chapter – Page
Creating a Drawing from a Part	8-5
Drafting Standard	8-7
Drawing Units	8-7
Drawing Toolbar	8-8
Drawing Sheet Format	8-8
Drawing Sheet Properties	8-8
Standard 3 View Command	8-9
Repositioning Drawing Views	8-9
Add Drawing Sheet	8-10

Annotations

Objectives: Creating Annotations.

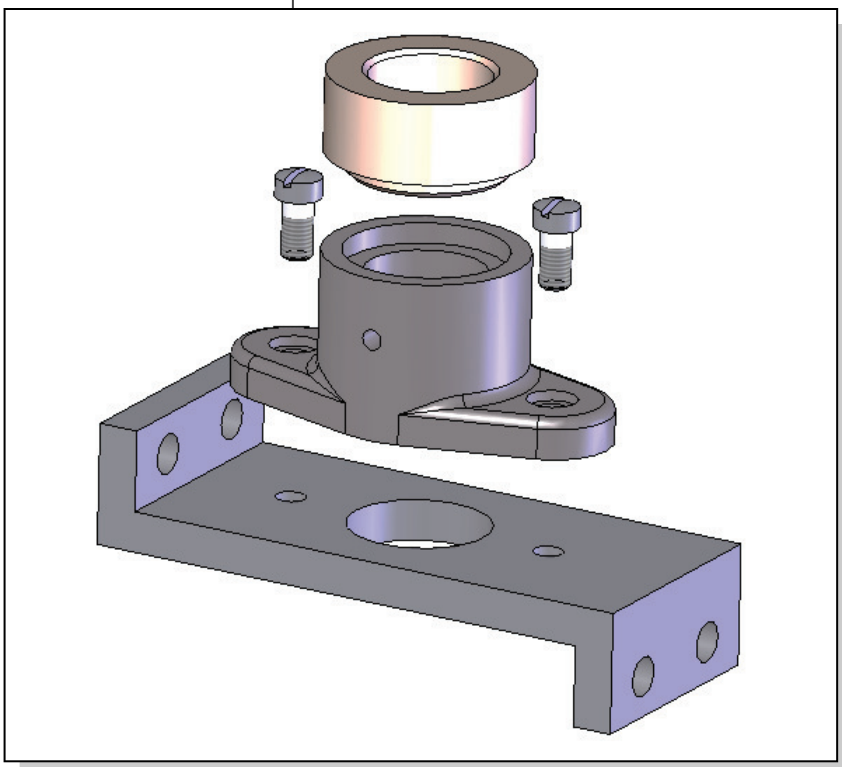
Certification Examination Performance Task — **Covered in this book on Chapter – Page**

Certified Associate Reference Guide

Tips about Taking the Certified SOLIDWORKS Associate (CSWA) Examination

1. **Study:** The first step to maximize your potential on an exam is to sufficiently prepare for it. You need to be familiar with the SOLIDWORKS package, and this can only be achieved by doing designs and exploring the different commands available. The Certified SOLIDWORKS Associate (CSWA) exam is designed to measure your familiarity with the SOLIDWORKS software. You must be able to perform the given task and answer the exam questions correctly and quickly.
2. **Make Notes**: Take notes of what you learn either while attending classroom sessions or going through study material. Use these notes as a review guide before taking the actual test.
3. **Time Management**: The examination has a time limit. Manage the time you spend on each question. Always remember you do not need to score 100% to pass the exam. Also, keep in mind that some questions are weighed more heavily and may take more time to answer. You can flip back and forth to view different problems during the test time by using the arrow buttons. If you encounter a question you cannot answer in a reasonable amount of time, use the *Save As* feature in SOLIDWORKS to save a copy of the file, and move on to the next question. You can return to any question and enter or change the answer as long as you do not hit the [**End Examination**] button.
4. **Use the SOLIDWORKS *Help System***: If you get confused and can't think of the answer, remember the SOLIDWORKS *Help System* is a great tool to confirm your considerations. In preparing for the exam, familiarize yourself with the help utility organization (e.g., Contents, Index, Search options).
5. **Use Internet Search**: Use of an internet search utility is allowed during the test. If a test question requires general knowledge, for example definitions of engineering or drafting concepts (stress, yield strength, auxiliary view, etc.), remember the internet is available as a tool to assist in your considerations.
6. **Use Common Sense**: If you are unable to get the correct answer and unable to eliminate all distracters, then you need to select the best answer from the remaining selections. This may be a task of selecting the best answer from amongst several correct answers, or it may be selecting the least incorrect answer from amongst several poor answers.
7. **Be Cautious and Don't Act in Haste:** Devote some time to ponder and think of the correct answer. Ensure that you interpret all the options correctly before selecting from available choices. Don't go into panic mode while taking a test. Use the ***Arrow Buttons*** to review each question. When you are confident that you have answered all questions, end the examination using the [**End Examination**] button to submit your answers for scoring. You will receive a score report once you have submitted your answers.
8. **Relax before Exam:** In order to avoid last minute stress, make sure that you arrive 10 to 15 minutes early and relax before taking the exam.

Chapter 1
Getting Started

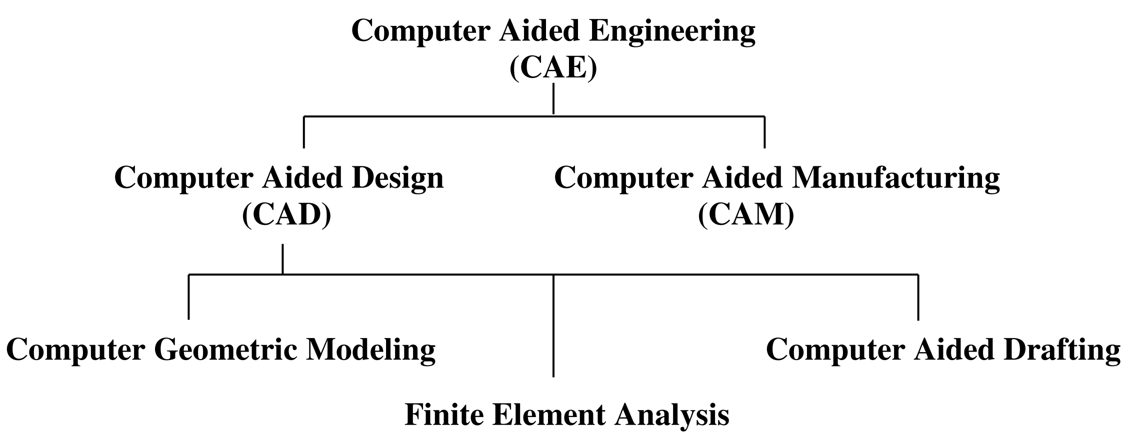


Learning Objectives

- **Development of Computer Geometric Modeling**
- **Feature-Based Parametric Modeling**
- **Startup Options and Units Setup**
- **SOLIDWORKS Screen Layout**
- **User Interface & Mouse Buttons**
- **SOLIDWORKS Online Help**

Introduction

The rapid changes in the field of **Computer Aided Engineering** (CAE) have brought exciting advances in the engineering community. Recent advances have made the long-sought goal of **concurrent engineering** closer to a reality. CAE has become the core of concurrent engineering and is aimed at reducing design time, producing prototypes faster, and achieving higher product quality. **SOLIDWORKS** is an integrated package of mechanical computer aided engineering software tools developed by *Dassault Systèmes* SOLIDWORKS *Corporation*. **SOLIDWORKS** is a tool that facilitates a concurrent engineering approach to the design and stress-analysis of mechanical engineering products. The computer models can also be used by manufacturing equipment such as machining centers, lathes, mills, or rapid prototyping machines to manufacture the product. In this text, we will be dealing only with the solid modeling modules used for part design and part drawings.



Development of Computer Geometric Modeling

Computer geometric modeling is a relatively new technology, and its rapid expansion in the last fifty years is truly amazing. Computer-modeling technology has advanced along with the development of computer hardware. The first generation CAD programs, developed in the 1950s, were mostly non-interactive; CAD users were required to create program-codes to generate the desired two-dimensional (2D) geometric shapes. Initially, the development of CAD technology occurred mostly in academic research facilities. The Massachusetts Institute of Technology, Carnegie-Mellon University, and Cambridge University were the leading pioneers at that time. The interest in CAD technology spread quickly and several major industry companies, such as General Motors, Lockheed, McDonnell, IBM, and Ford Motor Co., participated in the development of interactive CAD programs in the 1960s. Usage of CAD systems was primarily in the automotive industry, aerospace industry, and government agencies that developed their own programs for their specific needs. The 1960s also marked the beginning of the development of finite element analysis methods for computer stress analysis and computer aided manufacturing for generating machine toolpaths.

The 1970s are generally viewed as the years of the most significant progress in the development of computer hardware, namely the invention and development of **microprocessors**. With the improvement in computing power, new types of 3D CAD programs that were user-friendly and interactive became reality. CAD technology quickly expanded from very simple **computer aided drafting** to very complex **computer aided design**. The use of 2D and 3D wireframe modelers was accepted as the leading edge technology that could increase productivity in industry. The developments of surface modeling and solid modeling technologies were taking shape by the late 1970s, but the high cost of computer hardware and programming slowed the development of such technology. During this period, the available CAD systems all required room-sized mainframe computers that were extremely expensive.

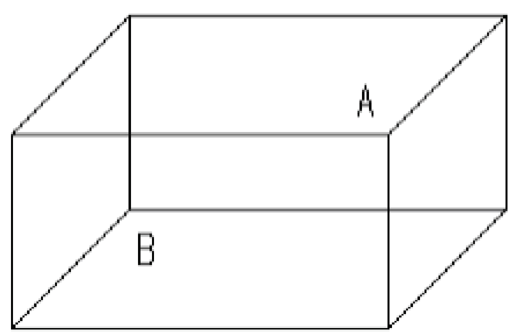
In the 1980s, improvements in computer hardware brought the power of mainframes to the desktop at less cost and with more accessibility to the general public. By the mid-1980s, CAD technology had become the main focus of a variety of manufacturing industries and was very competitive with traditional design/drafting methods. It was during this period of time that 3D solid modeling technology had major advancements, which boosted the usage of CAE technology in industry.

The introduction of the *feature-based parametric solid modeling* approach, at the end of the 1980s, elevated CAD/CAM/CAE technology to a new level. In the 1990s, CAD programs evolved into powerful design/manufacturing/management tools. CAD technology has come a long way, and during these years of development, modeling schemes progressed from two-dimensional (2D) wireframe to three-dimensional (3D) wireframe, to surface modeling, to solid modeling and, finally, to feature-based parametric solid modeling.

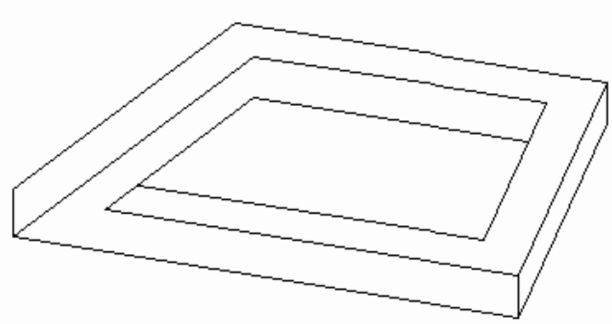
The first generation CAD packages were simply 2D **computer aided drafting** programs, basically the electronic equivalents of the drafting board. For typical models, the use of this type of program would require that several to many views of the objects be created individually as they would be on the drafting board. The 3D designs remained in the designer's mind, not in the computer database. Mental translations of 3D objects to 2D views are required throughout the use of these packages. Although such systems have some advantages over traditional board drafting, they are still tedious and labor intensive. The need for the development of 3D modelers came quite naturally, given the limitations of the 2D drafting packages.

The development of three-dimensional modeling schemes started with three-dimensional (3D) wireframes. Wireframe models are models consisting of points and edges, which are straight lines connecting between appropriate points. The edges of wireframe models are used, similar to lines in 2D drawings, to represent transitions of surfaces and features. The use of lines and points is also a very economical way to represent 3D designs.

The development of the 3D wireframe modeler was a major leap in the area of computer geometric modeling. The computer database in the 3D wireframe modeler contains the locations of all the points in space coordinates, and it is typically sufficient to create just one model rather than multiple views of the same model. This single 3D model can then be viewed from any direction as needed. Most 3D wireframe modelers allow the user to create projected lines/edges of 3D wireframe models. In comparison to other types of 3D modelers, the 3D wireframe modelers require very little computing power and generally can be used to achieve reasonably good representations of 3D models. However, because surface definition is not part of a wireframe model, all wireframe images have the inherent problem of ambiguity. Two examples of such ambiguity are illustrated.



Wireframe Ambiguity: Which corner is in front, A or B?



A non-realizable object: Wireframe models contain no surface definitions.

Surface modeling is the logical development in computer geometry modeling to follow the 3D wireframe modeling scheme by organizing and grouping edges that define polygonal surfaces. Surface modeling describes the part's surfaces but not its interiors. Designers are still required to interactively examine surface models to ensure that the various surfaces on a model are contiguous throughout. Many of the concepts used in 3D wireframe and surface modelers are incorporated in the solid modeling scheme, but it is solid modeling that offers the most advantages as a design tool.

In the solid modeling presentation scheme, the solid definitions include nodes, edges, and surfaces, and it is a complete and unambiguous mathematical representation of a precisely enclosed and filled volume. Unlike the surface modeling method, solid modelers start with a solid or use topology rules to guarantee that all of the surfaces are stitched together properly. Two predominant methods for representing solid models are **constructive solid geometry** (CSG) representation and **boundary representation** (B-rep).

The CSG representation method can be defined as the combination of 3D solid primitives. What constitutes a “primitive” varies somewhat with the software but typically includes a rectangular prism, a cylinder, a cone, a wedge, and a sphere. Most solid modelers also allow the user to define additional primitives, which are shapes typically formed by the basic shapes. The underlying concept of the CSG representation method is very straightforward; we simply **add** or **subtract** one primitive from another. The CSG approach is also known as the machinist’s approach, as it can be used to simulate the manufacturing procedures for creating the 3D object.

In the B-rep representation method, objects are represented in terms of their spatial boundaries. This method defines the points, edges, and surfaces of a volume, and/or issues commands that sweep or rotate a defined face into a third dimension to form a solid. The object is then made up of the unions of these surfaces that completely and precisely enclose a volume.

By the 1980s, a new paradigm called *concurrent engineering* had emerged. With concurrent engineering, designers, design engineers, analysts, manufacturing engineers, and management engineers all work together closely right from the initial stages of the design. In this way, all aspects of the design can be evaluated and any potential problems can be identified right from the start and throughout the design process. Using the principles of concurrent engineering, a new type of computer modeling technique appeared. The technique is known as the *feature-based parametric modeling technique*. The key advantage of the *feature-based parametric modeling technique* is its capability to produce very flexible designs. Changes can be made easily and design alternatives can be evaluated with minimum effort. Various software packages offer different approaches to feature-based parametric modeling, yet the end result is a flexible design defined by its design variables and parametric features.

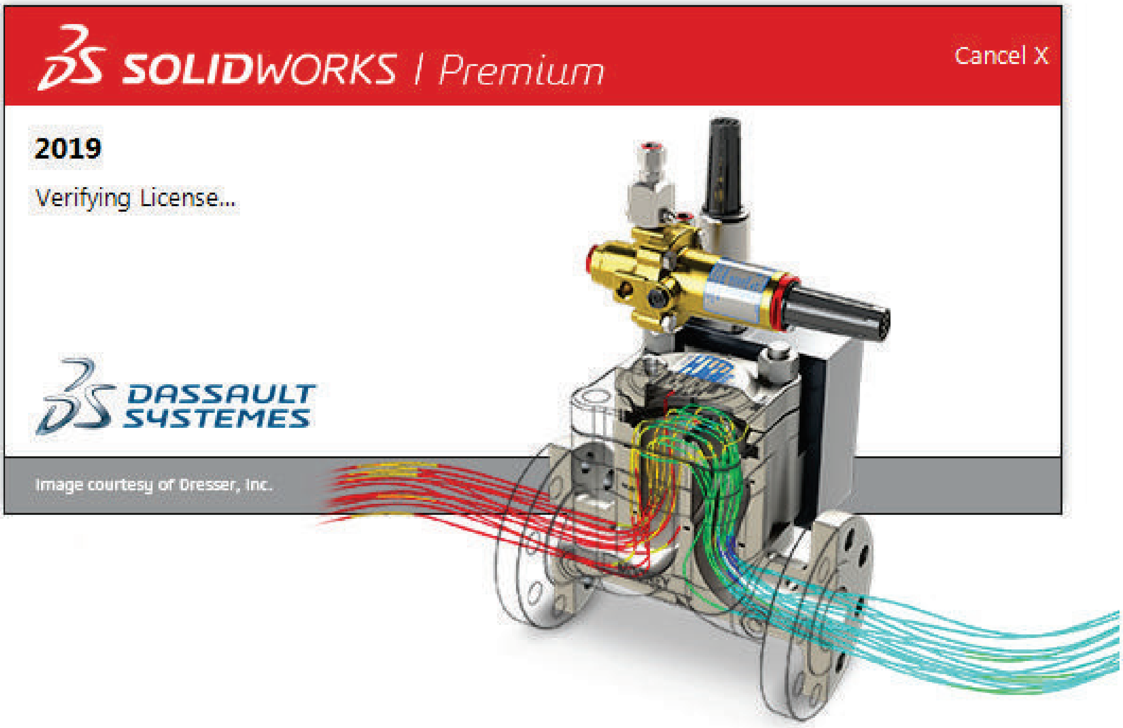
Feature-Based Parametric Modeling

One of the key elements in the SOLIDWORKS solid modeling software is its use of the **feature-based parametric modeling technique**. The feature-based parametric modeling approach has elevated solid modeling technology to the level of a very powerful design tool. Parametric modeling automates the design and revision procedures by the use of parametric features. Parametric features control the model geometry by the use of design variables. The word ***parametric*** means that the geometric definitions of the design, such as dimensions, can be varied at any time during the design process. Features are predefined parts or construction tools for which users define the key parameters. A part is described as a sequence of engineering features, which can be modified/changed at any time. The concept of parametric features makes modeling more closely match the actual design-manufacturing process than the mathematics of a solid modeling program. In parametric modeling, models and drawings are updated automatically when the design is refined.

Parametric modeling offers many benefits:

- **We begin with simple, conceptual models with minimal detail; this approach conforms to the design philosophy of "shape before size."**
- **Geometric relations, dimensional constraints, and relational parametric equations can be used to capture design intent.**
- **The ability to update an entire system, including parts, assemblies and drawings after changing one parameter of complex designs.**
- **We can quickly explore and evaluate different design variations and alternatives to determine the best design.**
- **Existing design data can be reused to create new designs.**
- **Quick design turn-around.**

Getting Started with SOLIDWORKS

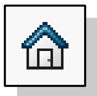


SOLIDWORKS is composed of several application software modules (these modules are called *applications*), all sharing a common database. In this text, the main concentration is placed on the solid modeling modules used for part design. The general procedures required in creating solid models, engineering drawings, and assemblies are illustrated.

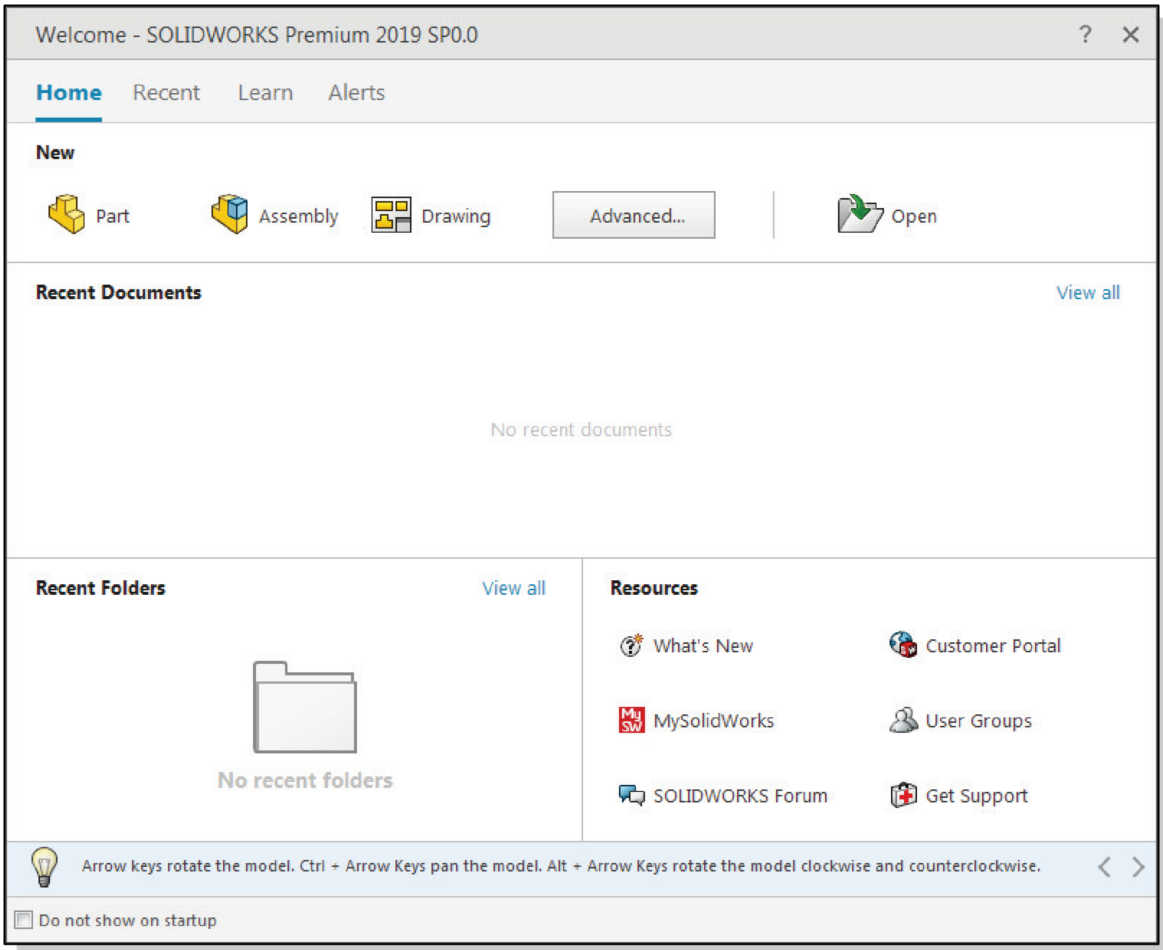
How to start SOLIDWORKS depends on the type of workstation and the particular software configuration you are using. With most *Windows* systems, you may select **SOLIDWORKS** on the *Start* menu or select the **SOLIDWORKS** icon on the desktop. Consult your instructor or technical support personnel if you have difficulty starting the software. The program takes a while to load, so be patient.

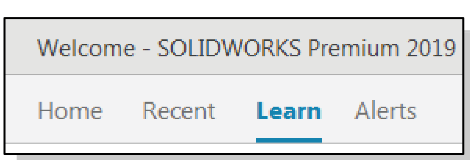
The tutorials in this text are based on the assumption that you are using the SOLIDWORKS default settings. If your system has been customized for other uses, contact your technical support personnel to restore the default software configuration.

Once the program is loaded into the memory, the SOLIDWORKS program window appears. In addition, the *Welcome* dialog box will open by default. The *Welcome* dialog box provides a convenient method to start new parts, drawings, or assemblies; open existing documents; or access SOLIDWORKS resources.

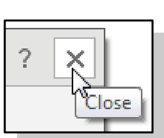


If the *Welcome* dialog box does not appear, it can be opened by clicking the *Welcome to SolidWorks* icon in the *Task Pane* or on the *Menu Bar*, both of which are described below.



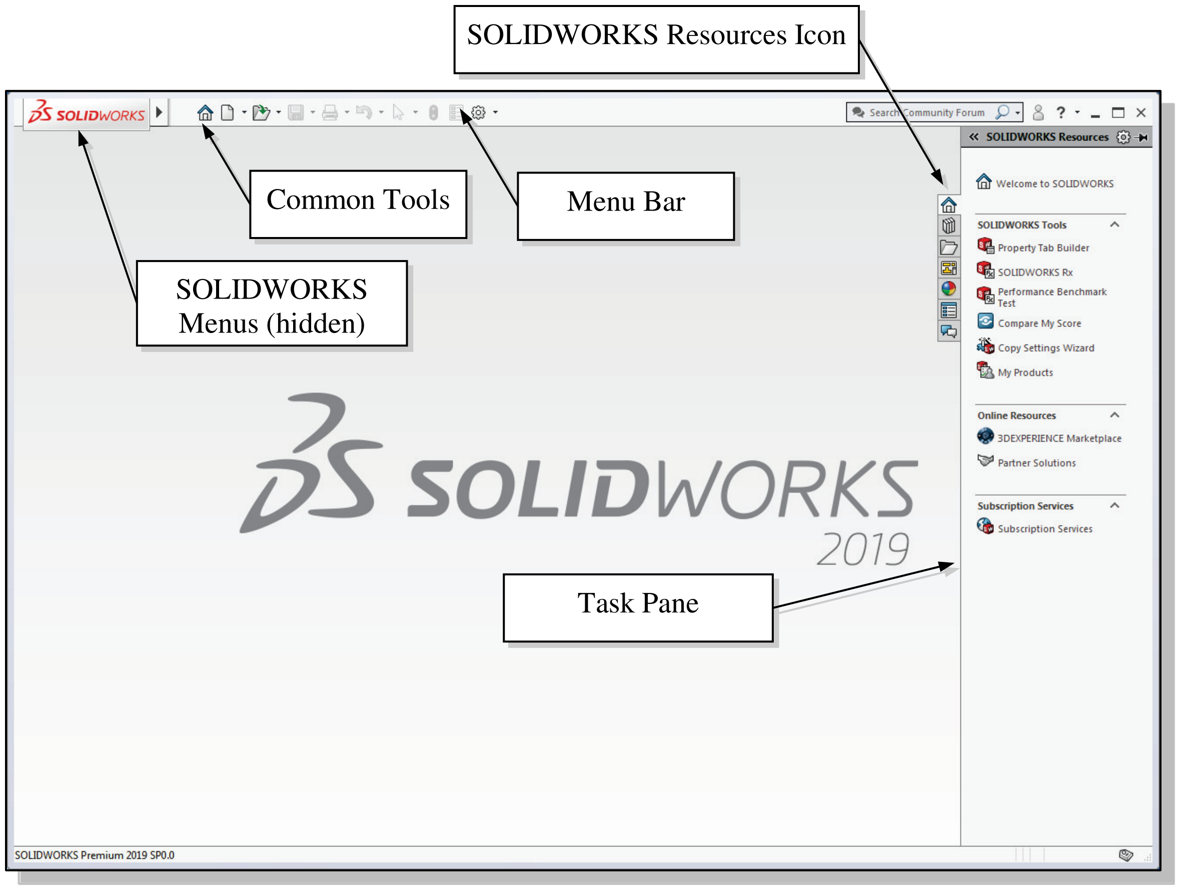


The *Welcome* dialog box has the following tabs: *Home*, *Recent*, *Learn*, and *Alerts*. Under the *Learn* tab, access is provided to SOLIDWORKS documentation, tutorials, and files provided for the tutorials. On your own, select the various tabs to reveal the options available.

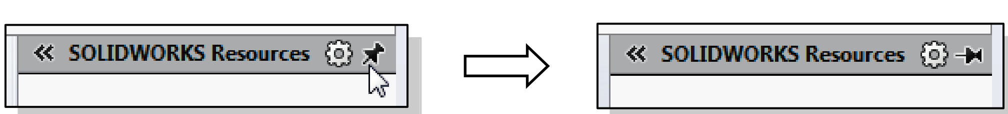


Close the *Welcome* dialog box by clicking on the X in the upper right corner of the box to view the SOLIDWORKS program window.

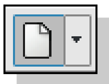
The SOLIDWORKS program window contains the *Menu Bar* and the *Task Pane*. The *Menu Bar* contains a subset of commonly used tools from the *Standard* toolbar (New, Open, Save, etc.), the SOLIDWORKS menus, the SOLIDWORKS *Search* oval, and a flyout menu of *Help* options. By default, the SOLIDWORKS menus are hidden. To display them, move the cursor over or click the SOLIDWORKS logo.



If the *Task Pane* does not appear to the right of the screen, right click on the *Menu Bar* to reveal a menu of toolbars and select the Task Pane, or select *View* from the SOLIDWORKS *Menus* and select Task Pane. Other options for the *Task Pane* include *Design Library*, *File Explorer*, *View Palette*, and *Appearances, Scenes, and Decals*. The icons for these options appear below the SOLIDWORKS Resources icon. The *File Explorer* duplicates *Windows Explorer* and provides access to recent documents. Other options will be used in future lessons. To collapse the *Task Pane*, click anywhere in the main area of the SOLIDWORKS window. (**NOTE:** If the *Task Pane* does not collapse upon clicking in the graphics area, it has been 'pinned' using the *Auto Show* button in the upper right corner. Simply left-click on the icon to unpin the *Task Pane* and allow it to collapse.)



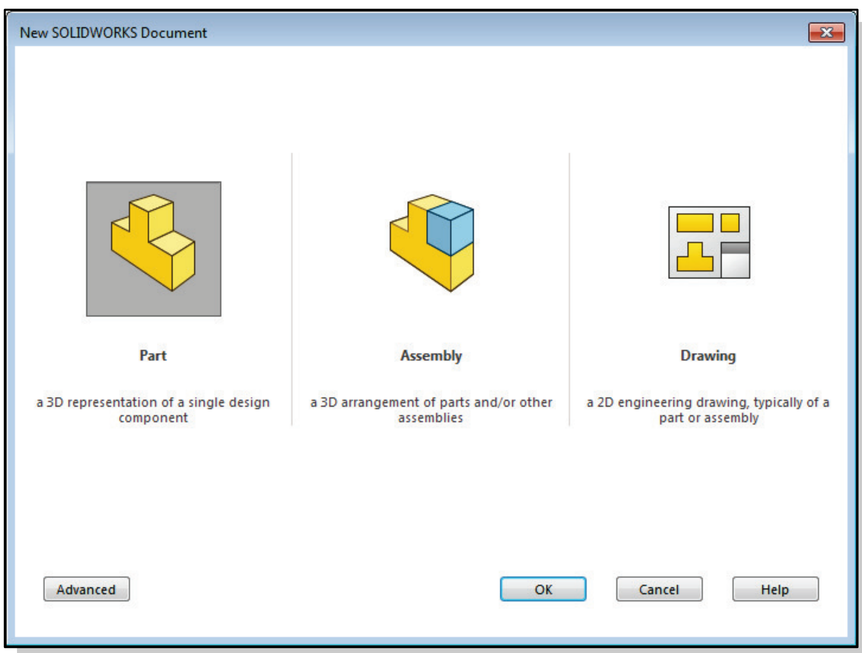
The following two startup options are available: New and Open. The New option allows us to start a new modeling task. The Open option allows us to open an existing model file. These two commands can be executed in the *Welcome* dialog box or on the *Menu Bar*.



➢ Select the **New** icon on the *Menu Bar* with a single click of the left-mouse-button. The *New* SOLIDWORKS *Document* dialog box appears.

➢ **NOTE:** If the *Units and Dimensions Standard* dialog box appears, click **OK** to accept the default settings.

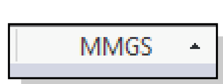
Three icons appear in the *New* SOLIDWORKS *Document* dialog box. Selecting the appropriate icon will allow creation of a new Part, Assembly, or Drawing file. A part is a single three-dimensional (3D) solid model. Parts are the basic building blocks in modeling with SOLIDWORKS. An assembly is a 3D arrangement of parts (components) and/or other assemblies (subassemblies). A drawing is a 2D representation of a part or assembly.



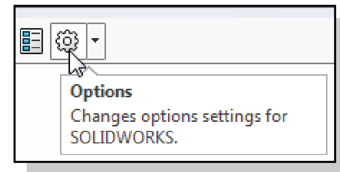
➢ Select the **Part** icon as shown. Click **OK** in the *New SOLIDWORKS Document* dialog box to open a new part file.

Units Setup

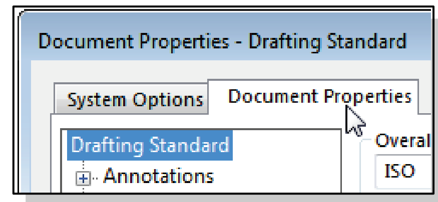
When starting a new CAD file, the first thing we should do is to choose the units we would like to use. The *Unit system* for the active document is shown on the *Status Bar* at the bottom of the SOLIDWORKS window (e.g., millimeter, gram, second as shown). We will use the English (feet and inches) setting for this example.



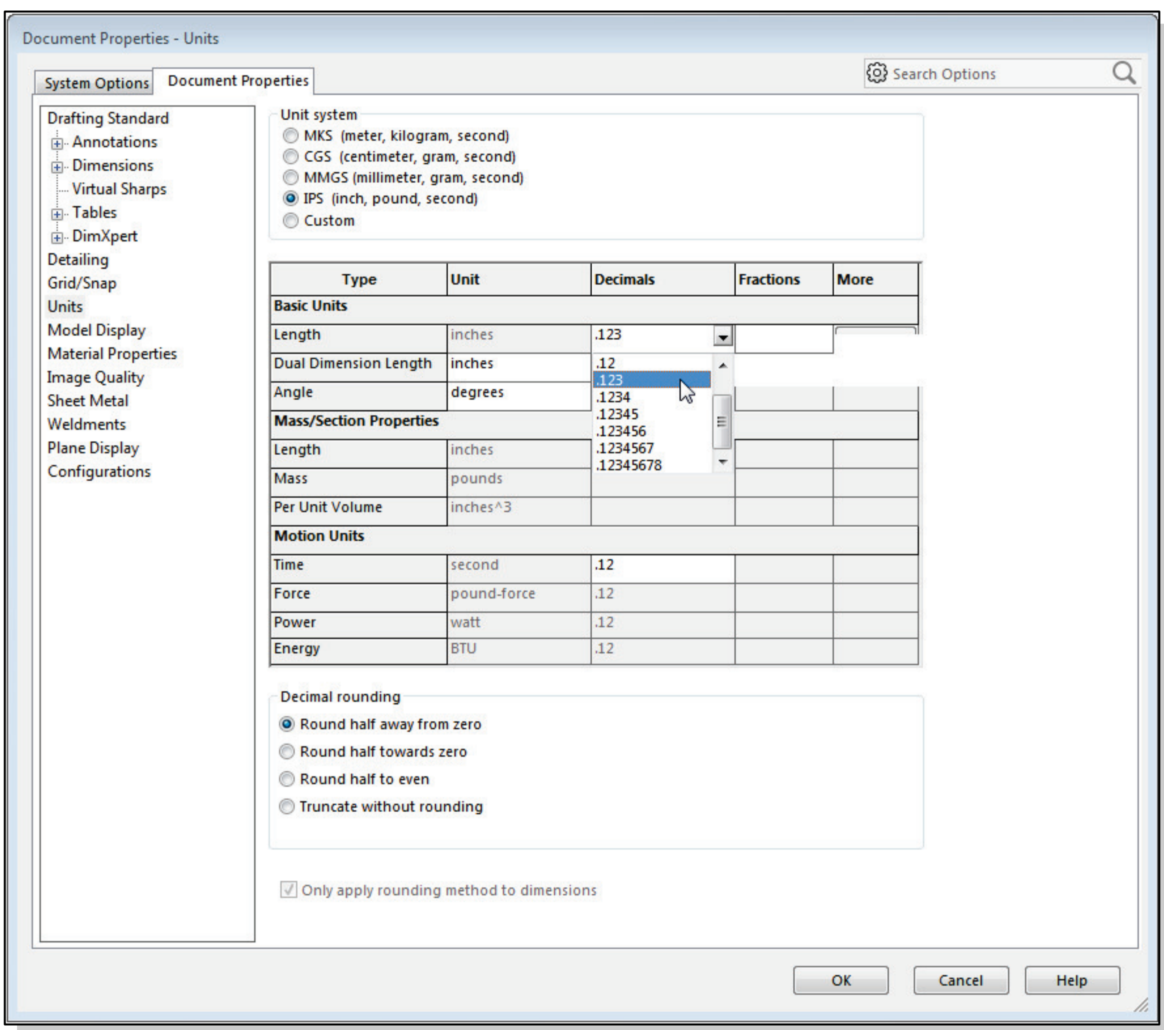
➢ Select the **Options** icon from the *Menu Bar* as shown to open the *Options* dialog box. (The *Options* dialog box can also be opened from the *Tools* pull-down menu.)



➢ When the *Options* dialog box opens, the **System Options** tab is active. The *Units* setup is located under the Document Properties tab. Select the **Document Properties** tab as shown.

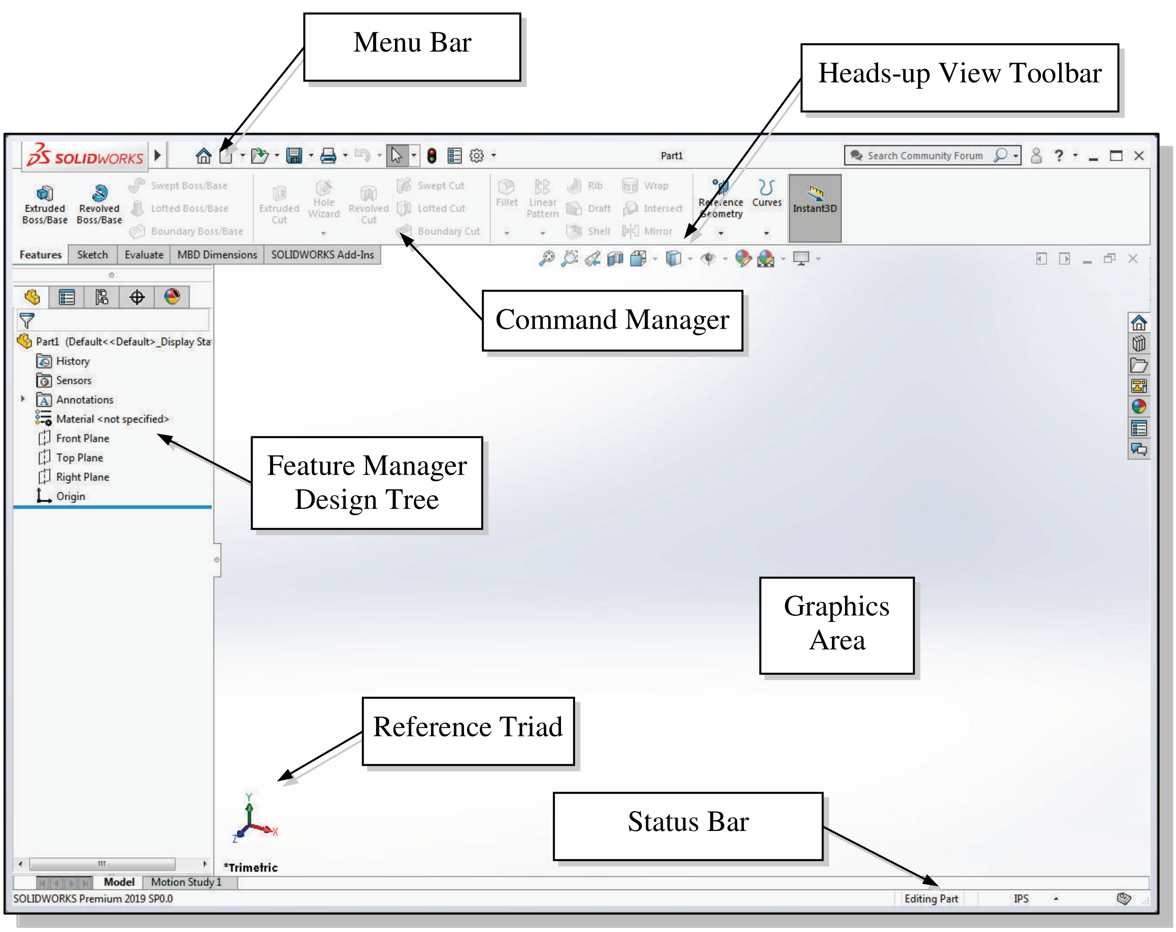


➢ Select **Units** on the left menu as highlighted below. Select **IPS (inch, pound, second)** under the *Unit system* options. Select **.123** in the *Decimals* spin box for the *Length units* as shown to define the degree of accuracy with which the units will be displayed. Click **OK** at the bottom of the *Document Properties - Units* window to set the units. (Notice IPS now appears on the *Status Bar* at the bottom of the window.)



SOLIDWORKS Screen Layout

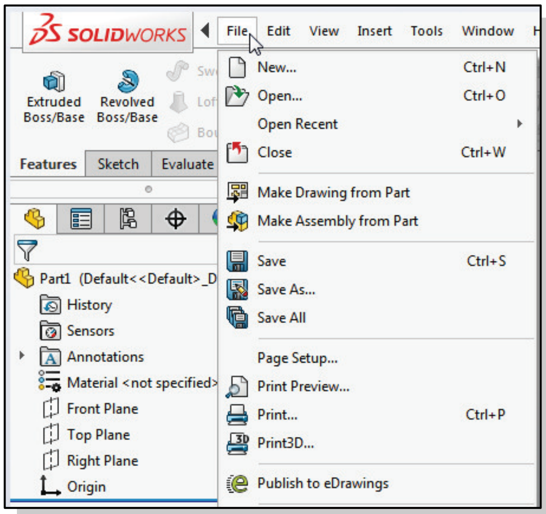
The default SOLIDWORKS drawing screen contains the *Menu Bar*, the *Heads-up View* toolbar, the *FeatureManager Design Tree*, the *CommandManager* (below the *Menu Bar*), the *task pane* (collapsed to the right of the graphics area in the figure below), and the *Status Bar*. (Note: If the *CommandManager* is inactive, the *Features* toolbar will appear vertically at the left and the *Sketch* toolbar will appear vertically at the right.) A line of quick text appears next to the icon as you move the *mouse cursor* over different icons. You may resize the SOLIDWORKS drawing window by clicking and dragging at the edges of the window, or relocate the window by clicking and dragging at the *window title* area.



- **Menu Bar Toolbar**

In the default view of the *Menu Bar*, only the toolbar options are visible. The default *Menu Bar* toolbar consists of a subset of frequently used commands from the *Menu Bar* as shown above.

- **SOLIDWORKS Pull-down Menus**



To display the *pull-down* menus, move the cursor over or click the SOLIDWORKS logo. The *pull-down* menus contain operations that you can use for all modes of the system.

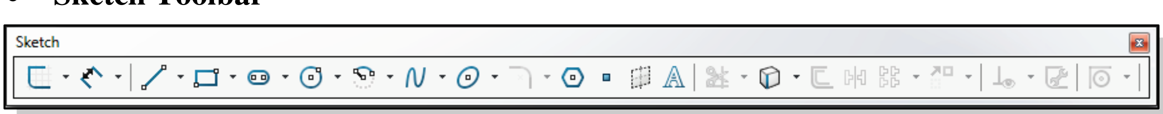
- **Heads-up View Toolbar**

The *Heads-up View* toolbar allows us quick access to frequently used view-related commands. **NOTE:** You can hide or customize the *Heads-up View* toolbar.

- **Features Toolbar**

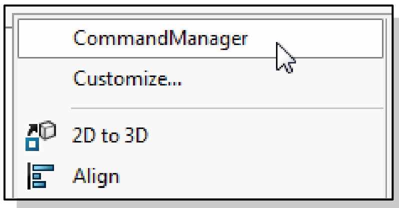
The *Features* toolbar allows us quick access to frequently used features-related commands, such as Extruded Boss/Base, Extruded Cut, and Revolved Boss/Base. When the *CommandManager* is used (with the Use Large Buttons with Text option turned *OFF*) the *Features* toolbar appears as shown above. When the *CommandManager* is turned *OFF*, the *Features* toolbar is displayed (by default) vertically at the left of the SOLIDWORKS window.

- **Sketch Toolbar**



The *Sketch* toolbar provides tools for creating the basic geometry that can be used to create features and parts. When the *CommandManager* is used (with the Use Large Buttons with Text option turned *OFF*), the *Sketch* toolbar appears as shown above. When the *CommandManager* is turned *OFF*, the *Sketch* toolbar is displayed (by default) vertically at the right of the SOLIDWORKS window.

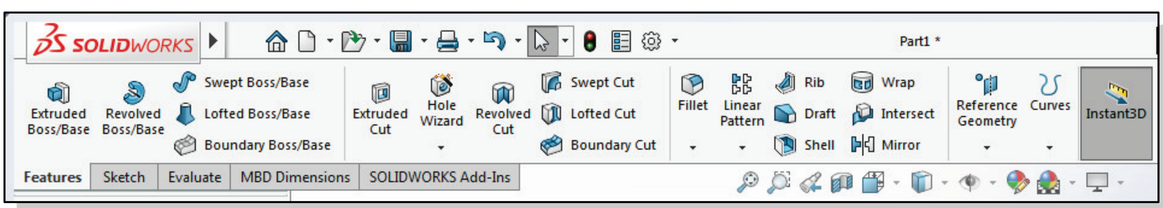
- **CommandManager**

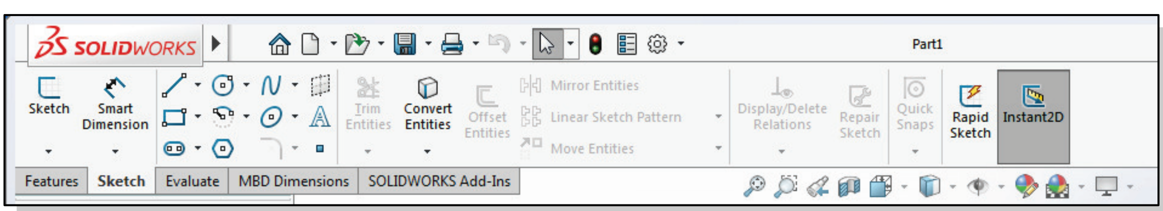


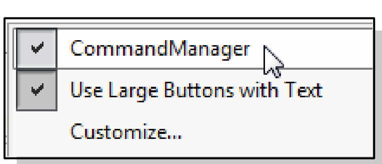
The SOLIDWORKS *CommandManager* provides one method for displaying the most commonly used toolbars. If the *CommandManager* is not visible, it can be turned *ON* by **right clicking** on any toolbar and selecting **CommandManager** from the top of the pop-up menu list of toolbars.

The *CommandManager* is a context-sensitive toolbar that dynamically updates based on the user's selection. When you click a tab below the *CommandManager*, it updates to display the corresponding toolbar. For example, if you click the *Sketches* tab, the *Sketch* toolbar appears. By default, the *CommandManager* has toolbars embedded in it based on the document type.

The display of the *CommandManager* is (with the Use Large Buttons with Text option turned on) shown below, once with the *Features* toolbar selected, once with the *Sketch* toolbar selected. You will notice that when the *CommandManager* is used, the *Sketch* and *Features* toolbars do not appear on the left and right edges of the display window.





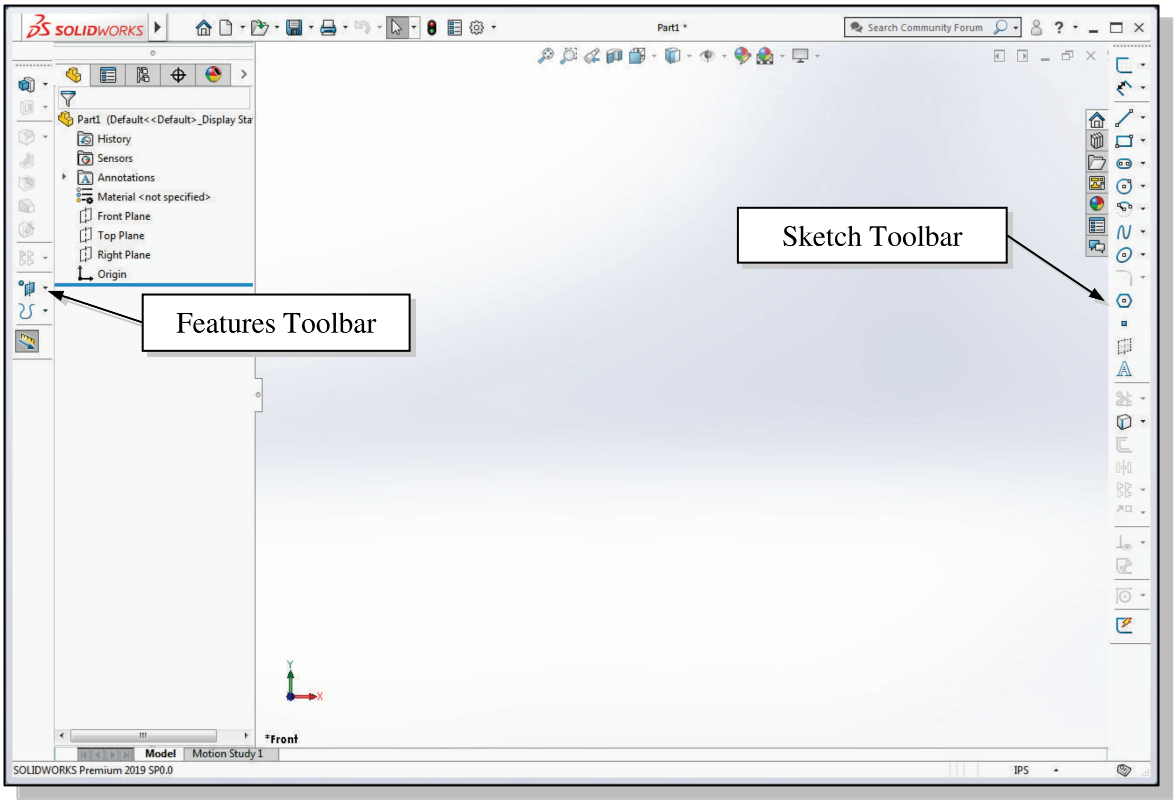


To turn *OFF* the *CommandManager* and use the standard display of toolbars, right click on the *CommandManager* (or any other toolbar) and toggle the *CommandManager* *OFF* by selecting it at the top of the pop-up menu.

IMPORTANT NOTE: Many lessons in this text use the standard display of toolbars. If a user prefers to use the *CommandManager*, the only change is that it may be necessary to select the appropriate tab prior to selecting a command. For example, if the instruction is to "select the Extruded Boss command from the *Features* toolbar," it may be necessary to first select the Features tab on the *CommandManager* to display the *Features* toolbar.

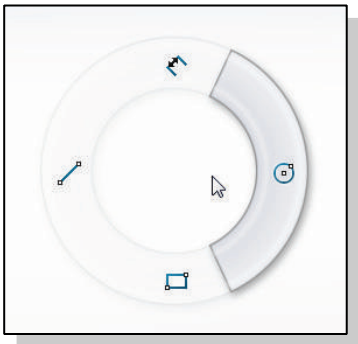
- **Standard Display of Toolbars**

The default SOLIDWORKS drawing screen using the standard display of toolbars, with the *CommandManager* turned *OFF*, is shown below. The *Features* toolbar appears at the left of the window and the *Sketch* toolbar at the right. This is the standard view used in the lessons in this text.



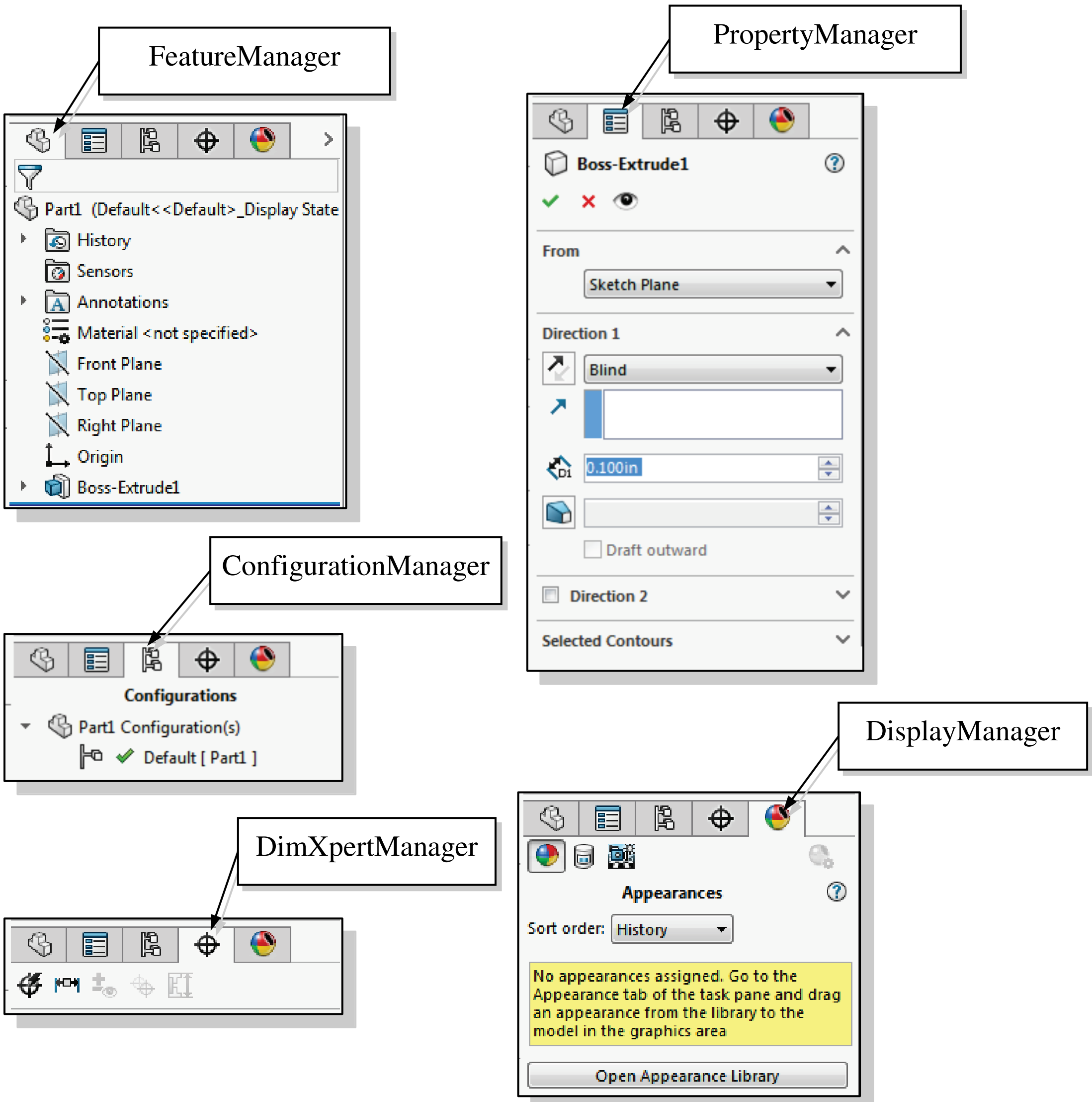
- **Mouse Gestures**

You can use a mouse gesture as a shortcut to execute some common commands. To activate a mouse gesture, move the cursor inside the graphics area, hold the right mouse button down and drag the mouse. A gesture guide appears showing command mappings for the gesture directions. Drag the mouse (while holding the right button down) across the appropriate button to execute the command.



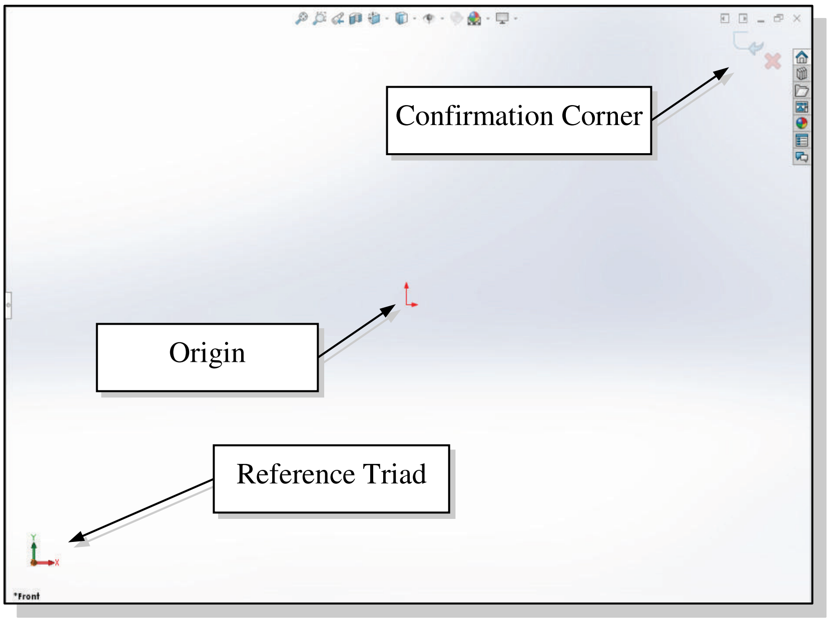
There are separate gesture guides for drawings, assemblies, parts, and sketches. The appropriate guide appears based on the current operation. For example, in sketch mode a gesture guide with sketch commands appears as shown here. You can customize up to eight gestures for each guide. To view or edit the current mouse gesture assignments, select **Customize** on the *Tools* pull-down menu, then select the **Mouse Gestures** tab.

- **FeatureManager Design Tree/PropertyManager/ConfigurationManager/ DimXpertManager/DisplayManager**



The left panel of the SOLIDWORKS window is used to display the *FeatureManager Design Tree*, the *PropertyManager*, the *ConfigurationManager*, the *DimXpertManager* and the *DisplayManager*. These options can be chosen by selecting the appropriate tab at the top of the panel. The *FeatureManager Design Tree* provides an overview of the active part, drawing, or assembly in outline form. It can be used to show and hide selected features, filter contents, and manage access to features and editing. The *PropertyManager* opens automatically when commands are executed or entities are selected in the graphics window, and is used to make selections, enter values, and accept commands. The *Configuration Manager* is used to create, select, and view multiple configurations of parts and assemblies. The *DimXpertManager* lists the tolerance features defined using the SOLIDWORKS ‘DimXpert for parts’ tools. The *DisplayManager* is used to control appearances, decals, scenes, lights, and cameras that are applied to the current model.

- **Graphics Area**



The graphics area is the area where models and drawings are displayed.

- **Reference Triad**

The *Reference Triad* appears in the graphics area of part and assembly documents. The triad is shown to help orient the user when viewing models and is for reference only.

- **Origin**

The *Origin* represents the (0,0,0) coordinate in a model or sketch. A model origin appears blue; a sketch origin appears red.

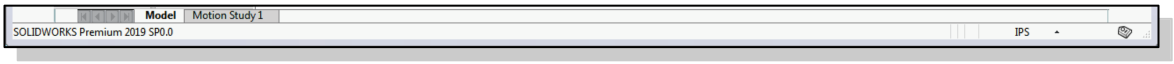
- **Confirmation Corner**

The *Confirmation Corner* offers an alternate way to accept features.

- **Graphics Cursor or Crosshairs**

The *graphics cursor* shows the location of the pointing device in the graphics window. During geometric construction, the coordinate of the cursor is displayed in the *Status Bar* area, located at the bottom of the screen. The cursor's appearance depends on the selected command or option.

- **Message and Status Bar**



The *Message and Status Bar* area displays a single-line description of a command when the cursor is on top of a command icon. This area also displays information pertinent to the active operation. In the figure above, the cursor coordinates are displayed while in the *Sketch* mode.

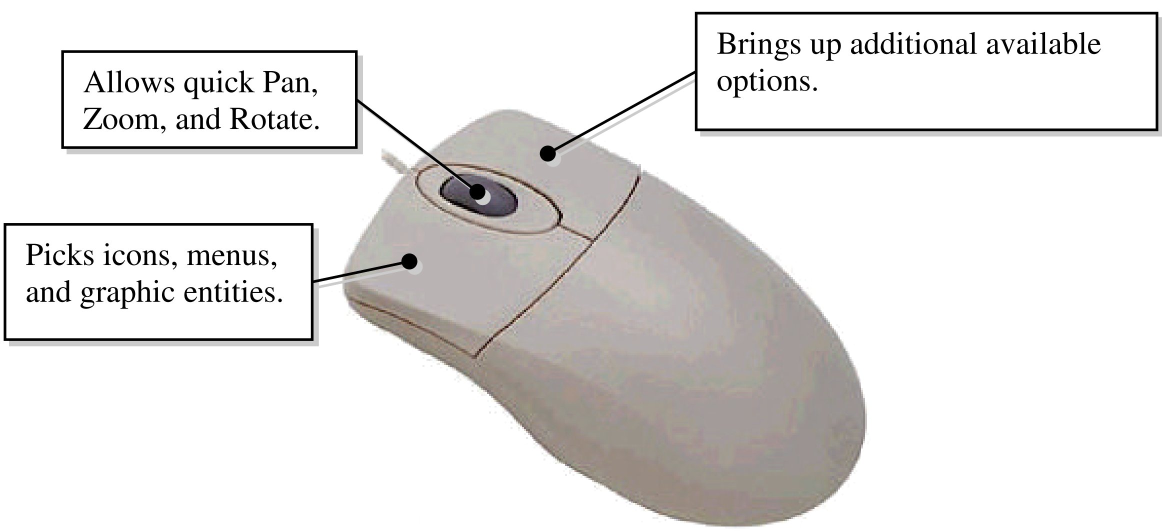
Mouse Buttons

SOLIDWORKS utilizes the mouse buttons extensively. In learning SOLIDWORKS' interactive environment, it is important to understand the basic functions of the mouse buttons.

- **Left mouse button**
  The **left-mouse-button** is used for most operations, such as selecting menus and icons, or picking graphic entities. One click of the button is used to select icons, menus and form entries, and to pick graphic items.

- **Right mouse button**
  The **right-mouse-button** is used to bring up additional available options in a context-sensitive pop-up menu. These menus provide shortcuts to frequently used commands.

- **Middle mouse button/wheel**
  The middle mouse button/wheel can be used to Rotate (hold down the wheel button and drag the mouse), Pan (hold down the wheel button and drag the mouse while holding down the **Ctrl** key), or Zoom (hold down the wheel button and drag the mouse while holding down the **Shift** key) realtime. Spinning the wheel allows zooming to the position of the cursor.

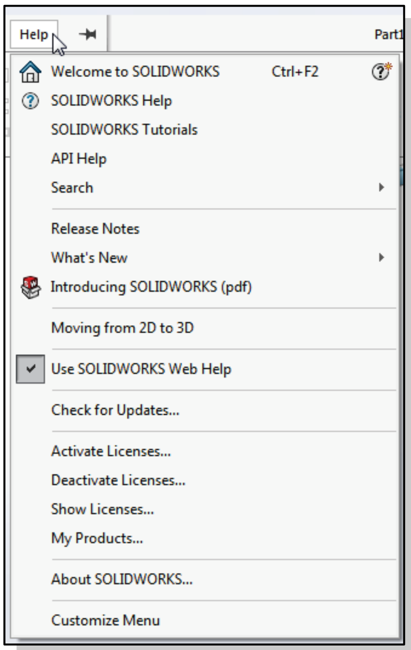


[Esc] – Canceling Commands

The [**Esc**] key is used to cancel a command in SOLIDWORKS. The [**Esc**] key is located near the top-left corner of the keyboard. Sometimes, it may be necessary to press the [**Esc**] key twice to cancel a command; it depends on where we are in the command sequence. For some commands, the [**Esc**] key is used to exit the command.

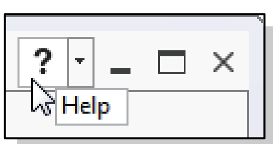
Online Help

Several types of online help are available at any time during a SOLIDWORKS session. SOLIDWORKS provides many on-line help functions, such as:



- The **Help** menu: Click on the **Help** option in *Menu Bar* to access the SOLIDWORKS *Help* menu system. (**NOTE:** Move the cursor over the SOLIDWORKS logo in the *Menu Bar* to display the pull-down menu options.) The **SOLIDWORKS Help** option provides general help information, such as command options and command references. The **SOLIDWORKS Tutorials** option provides a collection of tutorials illustrating different SOLIDWORKS operations.

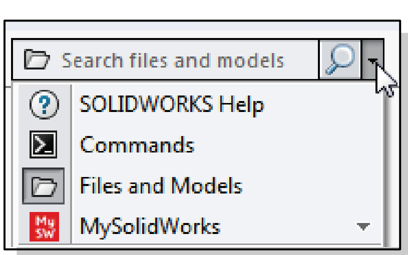
- The **SOLIDWORKS Tutorials** can also be accessed from the SOLIDWORKS *Welcome* dialog box under the *Learn* tab.



- The **SOLIDWORKS Help** option can also be accessed by clicking on the **Help** icon at the right end of the *Menu Bar*.

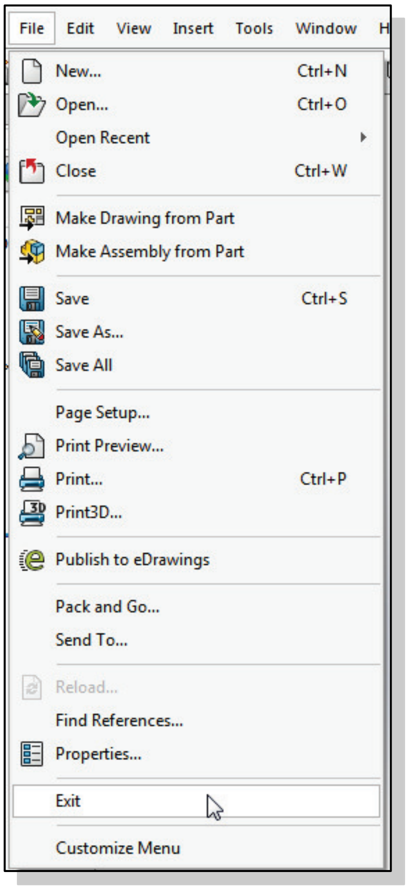
SOLIDWORKS Search

The **SOLIDWORKS Search** window, located on the *Menu Bar*, can be used to search various utilities, including searching SOLIDWORKS Help; searching for valid SOLIDWORKS commands; searching for files and models; and searching the MySolidWorks website.



- To execute a search, expand the search menu by clicking the arrow at the right of the window; select the search utility desired, e.g. Files and Models; type the text string for the search; and press **Enter**.

Leaving SOLIDWORKS



To leave SOLIDWORKS, use the left-mouse-button and click on **File** at the top of the SOLIDWORKS screen window, then choose **Exit** from the pull-down menu. (**NOTE:** Move the cursor over the SOLIDWORKS logo in the *Menu Bar* to display the pull-down menu options.)

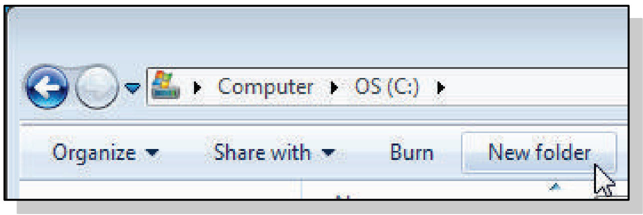
Creating a CAD Files Folder

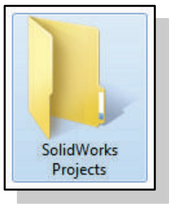
It is a good practice to create a separate folder to store your CAD files. You should not save your CAD files in the same folder where the SOLIDWORKS application is located. It is much easier to organize and back up your project files if they are in a separate folder. Making folders within this folder for different types of projects will help you organize your CAD files even further. When creating CAD files in SOLIDWORKS, it is strongly recommended that you *save* your CAD files on the hard drive.

➢ To create a new folder in the *Windows* environment:

1. In *Computer*, or start *Windows Explorer* under the *Start* menu, open the folder in which you want to create a new folder.

2. On the **File** menu, point to **New**, and then click **Folder**. The new folder appears with a temporary name.

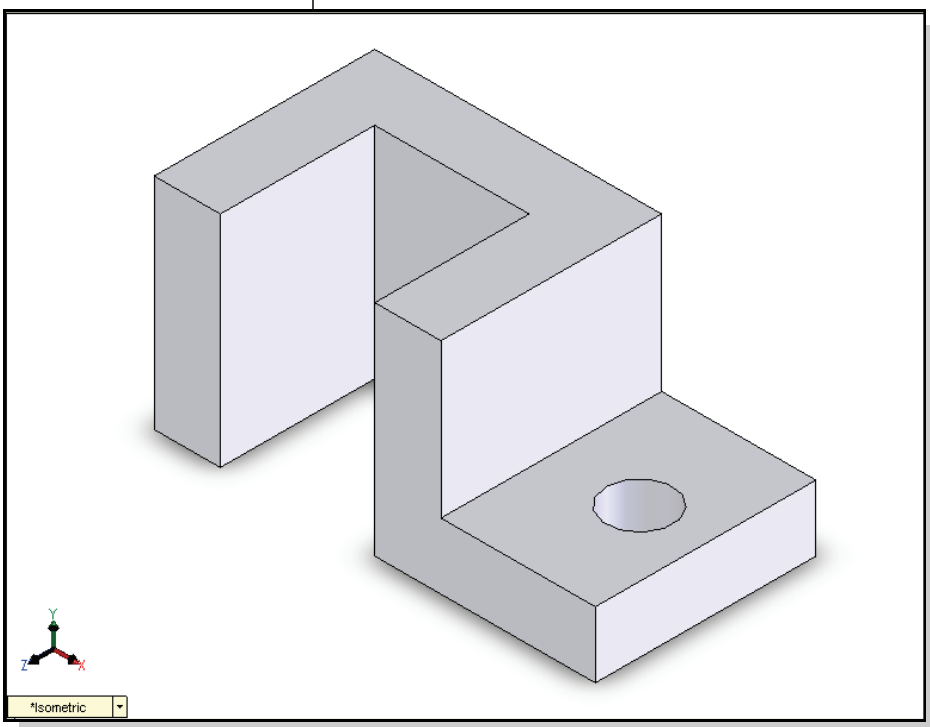




3. Type a name for the new folder, and then press **ENTER**.

Chapter 2

Parametric Modeling Fundamentals



Learning Objectives

- Create Simple Extruded Solid Models
- Understand the Basic Parametric Modeling Procedure
- Create 2-D Sketches
- Understand the “Shape before Size” Approach
- Use the Dynamic Viewing Commands
- Create and Edit Parametric Dimensions

Certified SOLIDWORKS Associate Exam Objectives Coverage

Sketch Entities – Lines, Rectangles, Circles, Arcs, Ellipses, Centerlines

Objectives: Creating Sketch Entities.

Sketch Relations

Objectives: Using Geometric Relations.

Boss and Cut Features – Extrudes, Revolves, Sweeps, Lofts

Objectives: Creating Basic Swept Features.

Dimensions

Objectives: Applying and Editing Smart Dimensions.

Feature Conditions – Start and End

Objectives: Controlling Feature Start and End Conditions.

Introduction

The **feature-based parametric modeling** technique enables the designer to incorporate the original **design intent** into the construction of the model. The word ***parametric*** means the geometric definitions of the design, such as dimensions, can be varied at any time in the design process. Parametric modeling is accomplished by identifying and creating the key features of the design with the aid of computer software. The design variables, described in the sketches as parametric relations, can then be used to quickly modify/update the design.

In SOLIDWORKS, the parametric part modeling process involves the following steps:

1. **Create a rough two-dimensional sketch of the basic shape of the base feature of the design.**

2. **Apply/modify geometric relations and dimensions to the two-dimensional sketch.**

3. **Extrude, revolve, or sweep the parametric two-dimensional sketch to create the base solid feature of the design.**

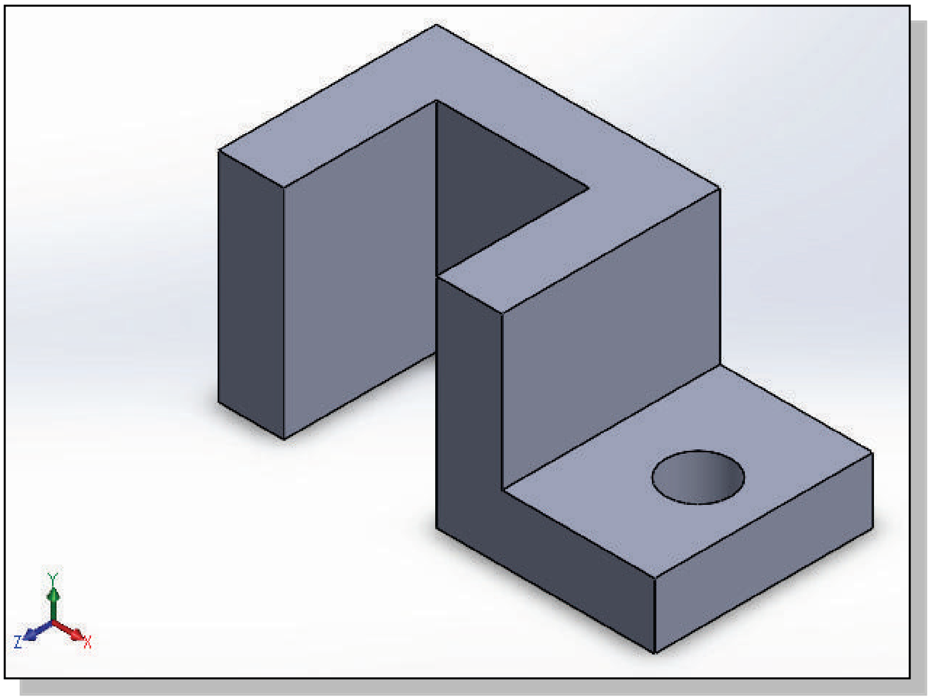
4. **Add additional parametric features by identifying feature relations and complete the design.**

5. **Perform analyses on the computer model and refine the design as needed.**

6. **Create the desired drawing views to document the design.**

The approach of creating two-dimensional sketches of the three-dimensional features is an effective way to construct solid models. Many designs are in fact the same shape in one direction. Computer input and output devices we use today are largely two-dimensional in nature, which makes this modeling technique quite practical. This method also conforms to the design process that helps the designer with conceptual design along with the capability to capture the ***design intent***. Most engineers and designers can relate to the experience of making rough sketches on restaurant napkins to convey conceptual design ideas. SOLIDWORKS provides many powerful modeling and design-tools, and there are many different approaches to accomplishing modeling tasks. The basic principle of **feature-based modeling** is to build models by adding simple features one at a time. In this chapter, the general parametric part modeling procedure is illustrated; a very simple solid model with extruded features is used to introduce the SOLIDWORKS user interface. The display viewing functions and the basic two-dimensional sketching tools are also demonstrated.

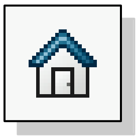
The *Adjuster* Design



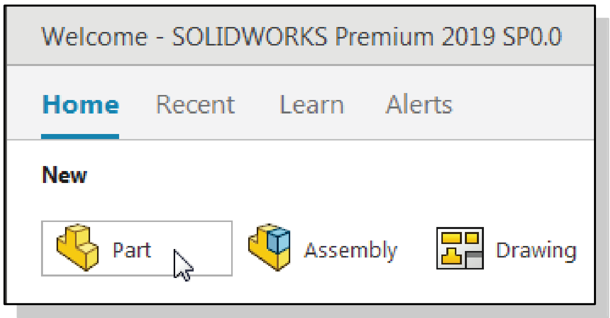
Starting SOLIDWORKS

1. Select the **SOLIDWORKS** option on the *Start* menu or select the **SOLIDWORKS** icon on the desktop to start SOLIDWORKS. The SOLIDWORKS main window will appear on the screen.

➢ We will start a new SOLODWORKS part file using the *Welcome* dialog box.



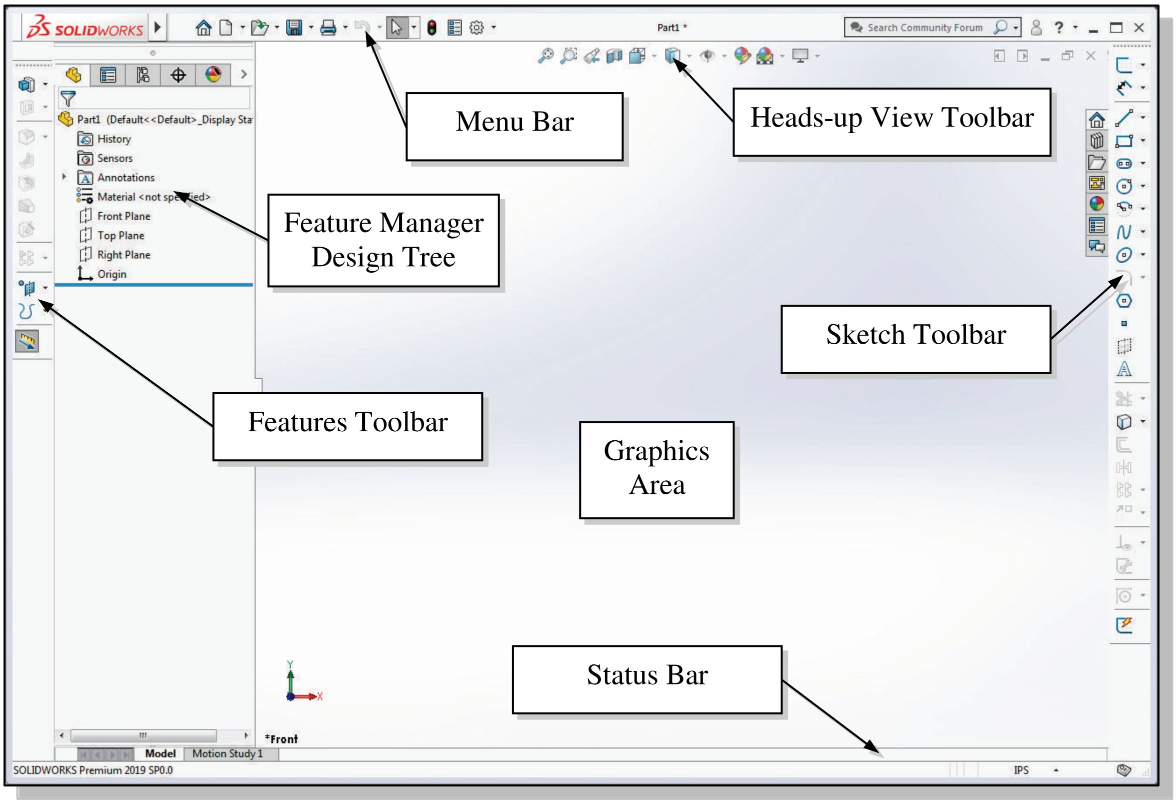
2. If the *Welcome* dialog box does not appear automatically upon opening SOLIDWORKS, it can be opened by clicking the *Welcome to SolidWorks* icon in the *Task Pane* or on the *Menu Bar*.



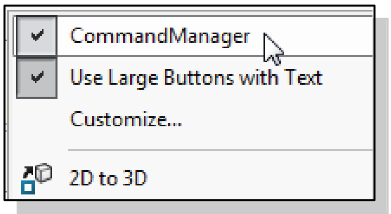
3. Select the **Part** icon with a single click of the left-mouse-button in the *Welcome* dialog box to open a new part document.

SOLIDWORKS Screen Layout

The default SOLIDWORKS drawing screen contains the *Menu Bar*, the *Heads-up View* toolbar, the *FeatureManager Design Tree*, the *Features* toolbar (at the left of the window by default), the *Sketch* toolbar (at the right of the window by default), the graphics area, the *task pane* (collapsed to the right of the graphics area in the figure below), and the *Status Bar*. A line of quick text appears next to the icon as you move the *mouse cursor* over different icons. You may resize the SOLIDWORKS drawing window by clicking and dragging at the edges of the window, or relocate the window by clicking and dragging at the *window title* area.



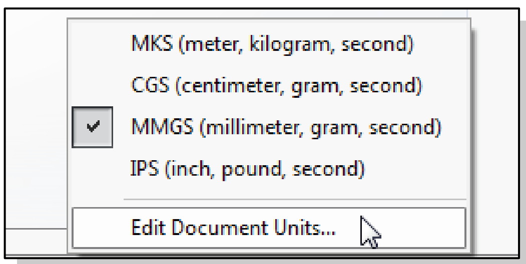
IMPORTANT NOTE: The SOLIDWORKS *CommandManager* provides an alternate method for displaying the most commonly used toolbars. If the *CommandManager* is active, the display will appear as shown on page 1-11. In this lesson, we will use the standard display of toolbars shown above. If a user prefers to use the *CommandManager*, the only change is that it may be necessary to select the appropriate tab prior to selecting a command. For example, if the instruction is to "select the Extruded Boss command from the *Features* toolbar," it may be necessary to first select the Features tab on the *CommandManager* to display the *Features* toolbar.



1. To turn ***OFF*** the *CommandManager* and use the standard display of toolbars, right click on the *CommandManager* (or any other toolbar) and toggle the *CommandManager OFF* by selecting it at the top of the pop-up menu.

Units Setup

When starting a new CAD file, the first thing we should do is choose the units we would like to use. The *Unit system* for the active document can be changed or customized using the **System Units** option on the *Status Bar* at the bottom of the SOLIDWORKS window. We will use English units (inches, pounds) for this example.

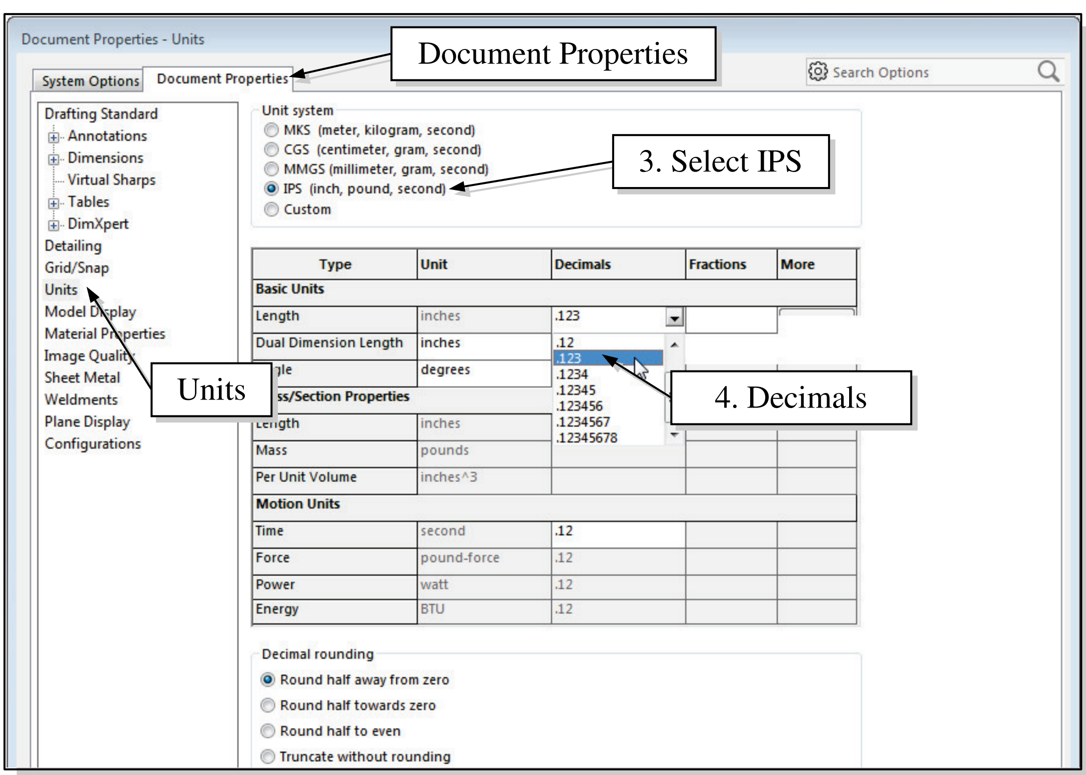


1. If the default *Unit system* is set to millimeter, gram, second, the *System Units* icon on the *Status Bar* displays **MMGS**. Click on the *System Units* icon to reveal additional options.

2. Select the **Edit Document Units** option as shown. This will open the **Document Properties - Units** dialog box.

3. Select **IPS (inch, pound, second)** under the *Unit system* options.

4. Select **.123** in the *Decimals* spin box for the *Length units* as shown to define the degree of accuracy with which the units will be displayed to 3 decimal places.



5. Click **OK** in the *Options* dialog box to accept the selected settings.

Creating Rough Sketches

Quite often during the early design stage, the shape of a design may not have any precise dimensions. Most conventional CAD systems require the user to input the precise lengths and locations of all geometric entities defining the design, which are not available during the early design stage. With *parametric modeling*, we can use the computer to elaborate and formulate the design idea further during the initial design stage. With SOLIDWORKS, we can use the computer as an electronic sketchpad to help us concentrate on the formulation of forms and shapes for the design. This approach is the main advantage of *parametric modeling* over conventional solid-modeling techniques.

As the name implies, a ***rough sketch*** is not precise at all. When sketching, we simply sketch the geometry so that it closely resembles the desired shape. Precise scale or lengths are not needed. SOLIDWORKS provides us with many tools to assist us in finalizing sketches. For example, geometric entities such as horizontal and vertical lines are set automatically. However, if the rough sketches are poor, it will require much more work to generate the desired parametric sketches. Here are some general guidelines for creating sketches in SOLIDWORKS:

- **Create a sketch that is proportional to the desired shape.** Concentrate on the shapes and forms of the design.

- **Keep the sketches simple.** Leave out small geometry features such as fillets, rounds and chamfers. They can easily be placed using the Fillet and Chamfer commands after the parametric sketches have been established.

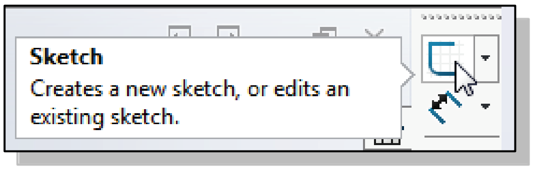
- **Exaggerate the geometric features of the desired shape.** For example, if the desired angle is 85 degrees, create an angle that is 50 or 60 degrees. Otherwise, SOLIDWORKS might assume the intended angle to be a 90-degree angle.

- **Draw the geometry so that it does not overlap.** The geometry should eventually form a closed region. *Self-intersecting* geometry shapes are not allowed.

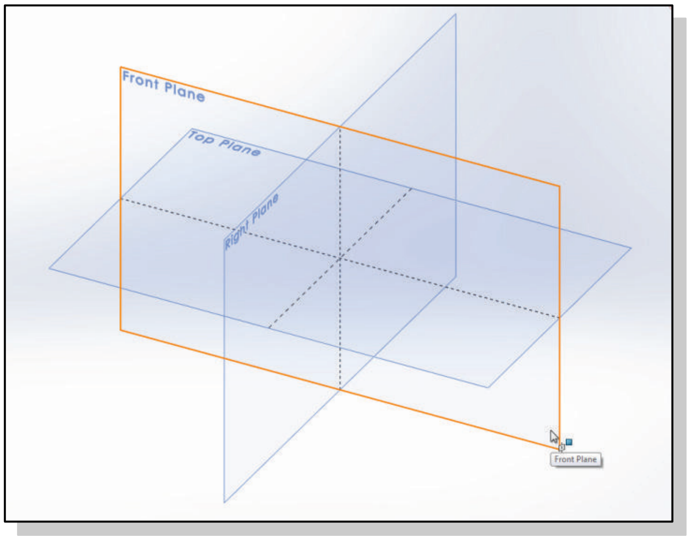
- **The sketched geometric entities should form a closed region.** To create a solid feature, such as an extruded solid, a closed region is required so that the extruded solid forms a 3D volume.

➢ **NOTE:** The concepts and principles involved in *parametric modeling* are very different from, and sometimes they are totally opposite to, those of conventional computer aided drafting. In order to understand and fully utilize SOLIDWORKS' functionality, it will be helpful to take a *Zen* approach to learning the topics presented in this text: **Temporarily forget your knowledge and experiences of using conventional Computer Aided Drafting systems.**

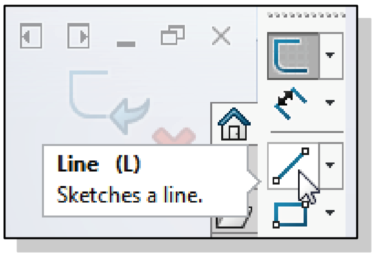
Step 1: Creating a Rough Sketch



1. Select the **Sketch** button at the top of the *Sketch* toolbar to create a new sketch. Notice the left panel displays the *Edit Sketch PropertyManager* with the instruction "*Select a plane on which to create a sketch for the entity.*"



2. Move the cursor over the edge of the *Front Plane* in the graphics area. When the *Front Plane* is highlighted, click once with the **left-mouse-button** to select the *Front Plane* as the sketch plane for the new sketch.



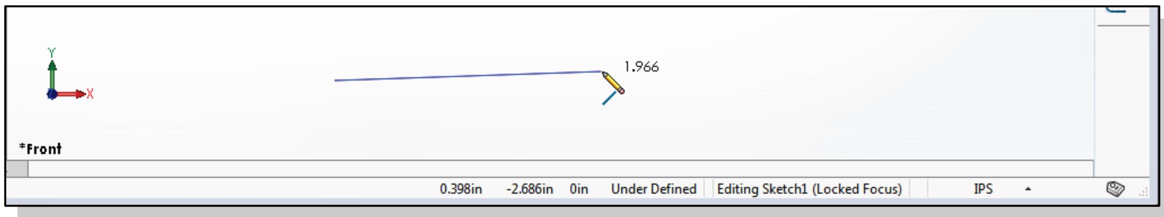
3. Select the **Line** icon on the *Sketch* toolbar by clicking once with the **left-mouse-button**; this will activate the Line command. The *Line Properties PropertyManager* is displayed in the left panel.

Graphics Cursors

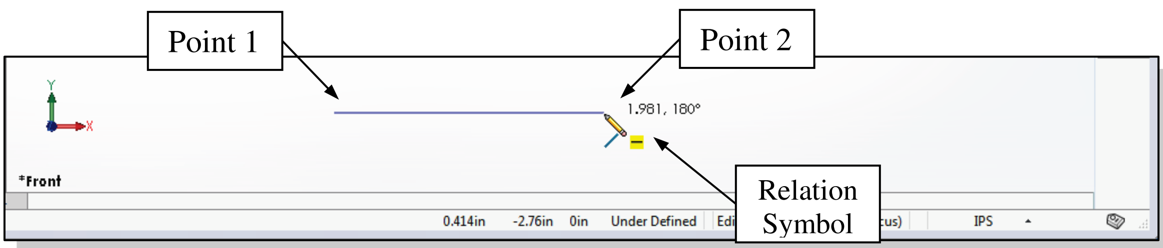
Notice the cursor changes from an arrow to a pencil when a sketch entity is active.

1. Left-click a starting point for the shape, roughly near the lower center of the graphics window.

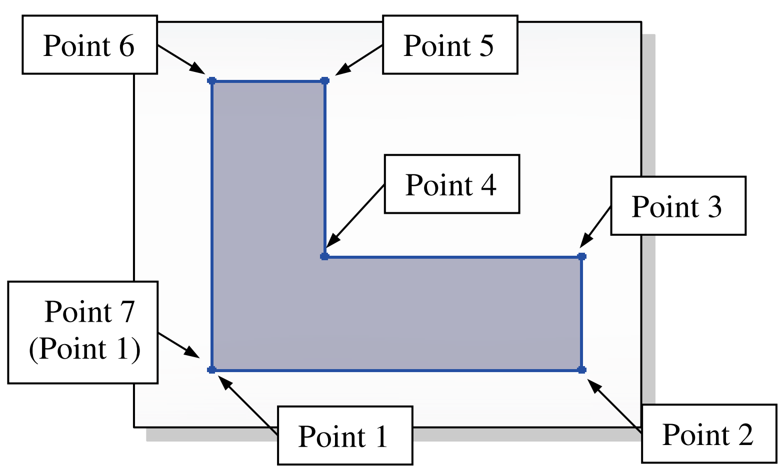
2. As you move the graphics cursor, you will see a digital readout next to the cursor. This readout gives you the line length. In the *Status Bar* area at the bottom of the window, the readout gives you the cursor location. Move the cursor around and you will notice different symbols appear at different locations.

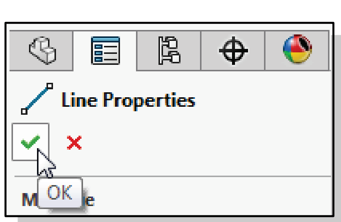


3. Move the graphics cursor toward the right side of the graphics window to create a horizontal line as shown below. Notice the geometric relation symbol displayed. When the **Horizontal** relation symbol is displayed, left-click to select **Point 2**.



4. Complete the sketch as shown below, creating a closed region **ending at the starting point** (Point 1). Do not be overly concerned with the actual size of the sketch. Note that all line segments are sketched horizontally or vertically.

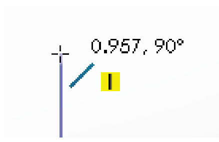




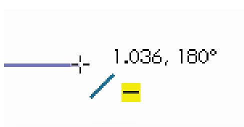
5. Click the **OK** icon (green check mark) in the *PropertyManager* to end editing of the current line, then click the **OK** icon again to end the Sketch Line command, or hit the [**Esc**] key once to end the Sketch Line command.

Geometric Relation Symbols

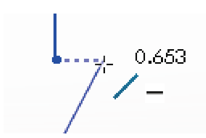
SOLIDWORKS displays different visual clues, or symbols, to show you alignments, perpendicularities, tangencies, etc. These relations are used to capture the *design intent* by creating relations where they are recognized. SOLIDWORKS displays the governing geometric rules as models are built. To prevent relations from forming, hold down the [**Ctrl**] key while creating an individual sketch curve. For example, while sketching line segments with the Line command, endpoints are joined with a *Coincident relation*, but when the [**Ctrl**] key is pressed and held, the inferred relation will not be created.



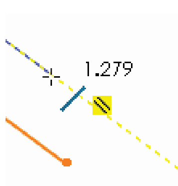
Vertical indicates a line is vertical



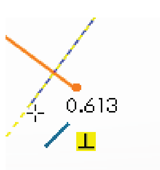
Horizontal indicates a line is horizontal



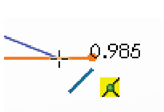
Dashed line indicates the alignment is to the center point or endpoint of an entity



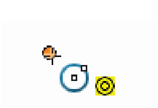
Parallel indicates a line is parallel to other entities



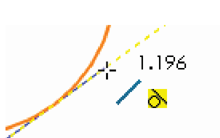
Perpendicular indicates a line is perpendicular to other entities



Coincident indicates the endpoint will be coincident with another entity



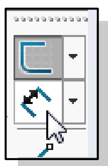
Concentric indicates the cursor is at the center of an entity



Tangent indicates the cursor is at tangency points to curves

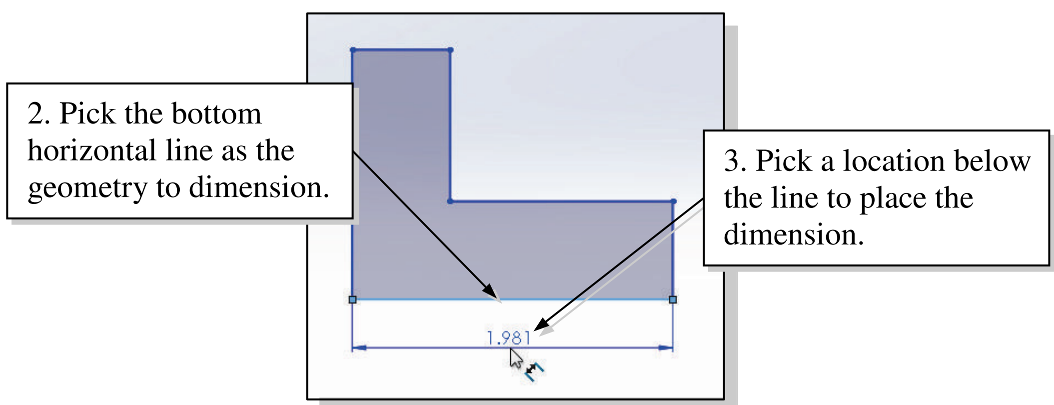
Step 2: Apply/Modify Relations and Dimensions

As the sketch is made, SOLIDWORKS automatically applies some of the geometric relations (such as Horizontal, Parallel, and Perpendicular) to the sketched geometry. We can continue to modify the geometry, apply additional relations, and/or define the size of the existing geometry. In this example, we will illustrate adding dimensions to describe the sketched entities.

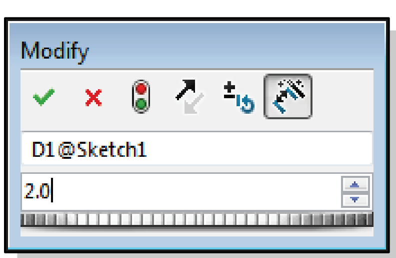


1. Move the cursor on top of the Smart Dimension icon on the *Sketch* toolbar. The Smart Dimension command allows us to quickly create and modify dimensions. Left-click once on the icon to activate the **Smart Dimension** command.

2. The message "*Select one or two edges/vertices and then a text location*" is displayed in the *Status Bar* area at the bottom of the SOLIDWORKS window. Select the bottom horizontal line by left-clicking once on the line.



3. Move the graphics cursor below the selected line and left-click to place the dimension. (Note that the value displayed on your screen might be different than what is shown in the figure above.)



4. Enter **2.0** in the *Modify* dialog box.

5. Left click the **OK** (green check mark) in the *Modify* dialog box to save the current value and exit the dialog.

6. **On your own**, select the lower right-vertical line.

7. Pick a location toward the right of the sketch to place the dimension.

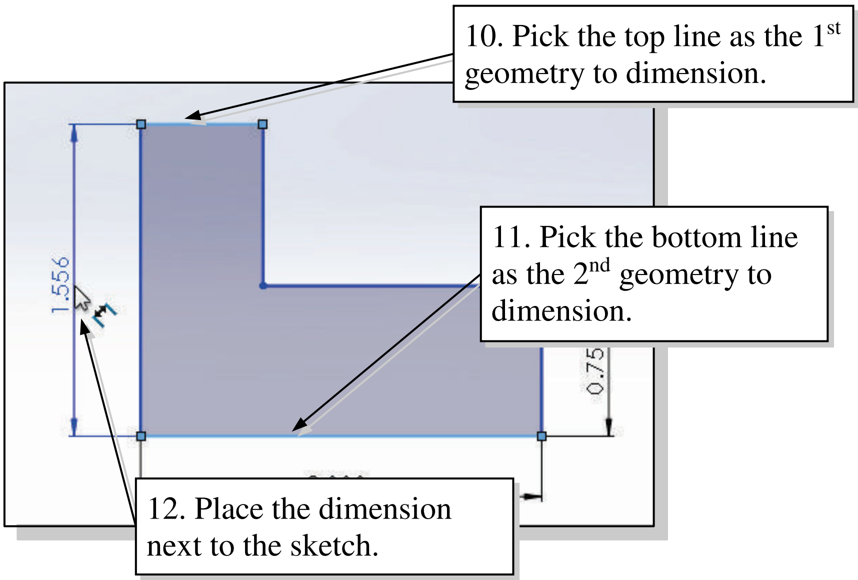
8. Enter **0.75** in the *Modify* dialog box.

9. Click **OK** in the *Modify* dialog box.

❖ The Smart Dimension command will create a length dimension if a single line is selected.

10. Select the top-horizontal line as shown below.

11. Select the bottom-horizontal line as shown below.



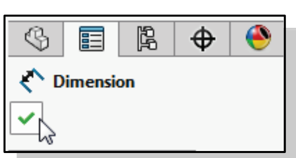
12. Pick a location to the left of the sketch to place the dimension.

13. Enter **2.0** in the *Modify* dialog box.

14. Click **OK** in the *Modify* dialog box.

❖ When two parallel lines are selected, the Smart Dimension command will create a dimension measuring the distance between them.

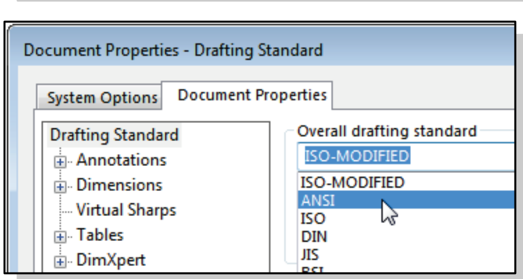
15. **On your own**, repeat the above steps and create an additional dimension for the top line. Make the dimension **0.75**.



16. Click the **OK** icon in the *PropertyManager* as shown, or hit the [**Esc**] key once, to end the Smart Dimension command.

Changing the Dimension Standard

1. Select the **Options** icon from the *Menu Bar* to open the *Options* dialog box.

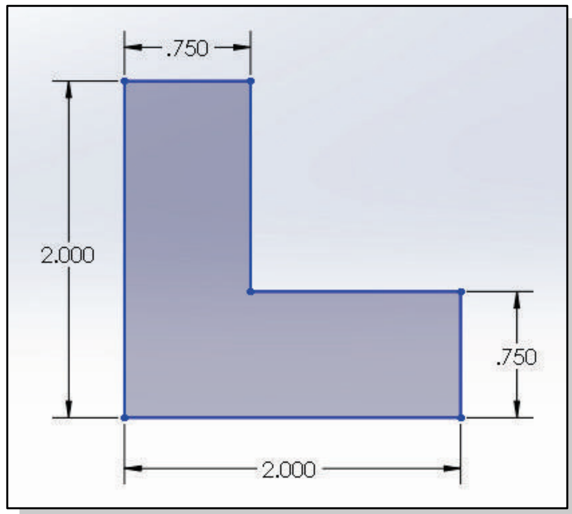


2. Select the **Document Properties** tab, then select **Drafting Standard** at the left.

3. Select **ANSI** in the pull-down selection window under the *Overall drafting standard* panel as shown.

4. Left-click **OK** in the *Options* dialog box to accept the settings.

- The sketch should now look as shown below. Notice the change in appearance of the dimensions.



Viewing Functions – *Zoom* and *Pan*

SOLIDWORKS provides a special user interface that enables convenient viewing of the entities in the graphics window. There are many ways to perform the **Zoom** and **Pan** operations.

1. Hold the **Ctrl** function key down. While holding the **Ctrl** function key down, press the mouse wheel down and drag the mouse to **pan** the display. This allows you to reposition the display while maintaining the same scale factor of the display.

2. Hold the **Shift** function key down. While holding the **Shift** function key down, press the mouse wheel down and drag the mouse to **zoom** the display. Moving downward will reduce the scale of the display, making the entities display smaller on the screen. Moving upward will magnify the scale of the display.

3. Turning the mouse wheel can also adjust the scale of the display. Turn the mouse wheel forward. Notice the scale of the display is reduced, making the entities display smaller on the screen.

4. Turn the mouse wheel backward. Notice scale of the display is magnified. (**NOTE:** Turning the mouse wheel allows zooming to the position of the cursor.)

5. On your own, use the options above to change the scale and position of the display.

6. Press the **F** key on the keyboard to automatically fit the model to the screen.