Introduction to Solid Modeling Using SOLIDWORKS® 2020

William E. Howard
East Carolina University

Joseph C. Musto
Milwaukee School of Engineering

McGraw Hill

INTRODUCTION TO SOLID MODELING USING SOLIDWORKS ® 2020

This book is printed on acid-free paper.

1 2 3 4 5 6 7 8 9 LWI 24 23 22 21 20

ISBN 978-1-260-25413-6 (bound edition)
MHID 1-260-25413-5 (bound edition)

Portfolio Manager: Beth Bettcher
Product Developer: Heather Ervolino
Senior Marketing Manager: Shannon O'Donnell
Content Project Manager: Jeni McAtee
Buyer: Laura Fuller
Designer: Beth Blech
Content Licensing Specialist: Beth Cray
Cover Images: William E. Howard
Compositor: Fleck's Communications, Inc.

mheducation.com/highered

About the Authors

Ed Howard is an Associate Professor in the Department of Engineering at East Carolina University, where he teaches classes in solid modeling, engineering computations, solid mechanics, and composite materials. Prior to joining ECU, Ed taught at Milwaukee School of Engineering. He holds a B.S. in Civil Engineering and an M.S. in Engineering Mechanics from Virginia Tech, and a Ph.D. in Mechanical Engineering from Marquette University.

Ed worked in design, analysis, and project engineering for 14 years before beginning his academic career. He worked for Thiokol Corporation in Brigham City, UT; Spaulding Composites Company in Smyrna, TN, and Sta-Rite Industries in Delavan, WI. He is a registered Professional Engineer in Wisconsin.

Joe Musto is a Professor in the Mechanical Engineering Department at Milwaukee School of Engineering, where he teaches in the areas of machine design, solid modeling, and numerical methods. He holds a B.S. degree from Clarkson University, and both an M.Eng. and Ph.D. from Rensselaer Polytechnic Institute, all in mechanical engineering. He is a registered Professional Engineer in Wisconsin.

Prior to joining the faculty at Milwaukee School of Engineering, he held industrial positions with Brady Corporation (Milwaukee, WI) and Eastman Kodak Company (Rochester, NY). He has been using and teaching solid modeling using SOLIDWORKS since 1998.

Joe and Ed, together with Rick Williams of Auburn University, are the authors of *Engineering Computation: An Introduction Using MATLAB® and Excel®,* part of the McGraw-Hill "Best" Series.

CONTENTS

SPECIAL FEATURES

DESIGN INTENT

FUTURE STUDY

PREFACE

As design engineers and engineering professors, the authors have witnessed incredible changes in the way that products are designed and manufactured. One of the biggest changes over the past 30 years has been the development and widespread usage of solid modeling software. When we first saw solid modeling, it was used only by large companies. The cost of the software and the powerful computer workstations required to run it, along with the complexity of using the software, limited its use. As the cost of computing hardware dropped, solid modeling software was developed for personal computers. In 1995, the SOLIDWORKS® Corporation released the initial version of SOLIDWORKS® software, the first solid modeling program written for the Microsoft Windows operating system. Since then, the use of solid modeling has become an indispensable tool for almost any company, large or small, that designs a product.

While 2-D drawings can be an effective tool to document and communicate design details, a solid model's usefulness extends throughout the design process. The solid model data can be saved in a format from which a physical model can be made with a 3-D printer. Structural, thermal, dynamic, and fluid flow analysis can be performed with finite element analysis (FEA) and other simulation software. The images on the cover of this book show the solid model of a mechanical device known as a *Geneva mechanism*. The Geneva mechanism is used to convert a continuously rotating input motion from a motor into a "stop/start" indexed rotation. By creating the solid model of the mechanism, and using add-on software for motion simulation, a digital prototype of the working mechanism can be created, operated, analyzed and optimized before it is constructed.

SOLIDWORKS is a registered trademark of Dassault Systémes SolidWorks Corporation.

Motivation for This Text

When we saw a demonstration of the SOLIDWORKS software in 1998, we were both instantly hooked. Not only was the utility of the software obvious, but the program was easy to learn and fun to use. Since then, we have shared our enthusiasm for the program with hundreds of students in classes at Milwaukee School of Engineering and East Carolina University, in summer programs with high school students, and in informal training sessions. Most of the material in this book began as tutorials that we developed for these purposes. We continue to be amazed at how quickly students at all levels can learn the basics of the program, and by the sophisticated projects that many students develop after only a short time using the software.

While anyone desiring to learn the SOLIDWORKS program can use this book, we have added specific elements for beginning engineering students. With these elements, we have attempted to introduce students to the design process and to relate solid modeling to subjects that most engineering students will study later. We hope that the combination of the tutorial style approach to teaching the functionality of the software, together with the integration of the material into the overall study of engineering, will motivate student interest not only in the SOLIDWORKS software but in the profession of engineering.

Philosophy of This Text

The development of powerful and integrated solid modeling software has continued the evolution of computer-aided design packages from drafting/graphical communication tools to full-fledged engineering design and analysis tools. A solid model is more than simply a drawing of an engineering component; it is a true virtual representation of the part, which can be manipulated, combined with other parts into complex assemblies, used directly for analysis, and used to drive the manufacturing equipment that will be used to produce the part.

This text was developed to exploit this emerging role of solid modeling as an integral part of the engineering design process; while proficiency in the software will be achieved through the exercises provided in the text, the traditional "training" exercises will be augmented with information on the integration of solid modeling into the engineering design process. These topics include:

- The exploitation of the parametric features of a solid model, to not only provide an accurate graphical representation of a part but also to effectively capture an engineer's design intent,
- The use of solid models as an analysis tool, useful for determining properties of components as well as for virtual prototyping of mechanisms and systems,
- The integration of solid modeling with component manufacturing, including the generation of molds, sheet metal patterns, and rapid prototyping files from component models.

Through the introduction of these topics, students will be shown not only the powerful modeling features of the SOLIDWORKS program, but also the role of the software as a full-fledged integrated engineering design tool.

The Use of This Text

This text primarily consists of chapter-long tutorials, which introduce both basic concepts in solid modeling (such as part modeling, drawing creation, and assembly modeling) and more advanced applications of solid modeling in engineering analysis and design (such as mechanism modeling, mold creation, sheet metal bending, and rapid prototyping). Each tutorial is organized as "keystroke-level" instructions, designed to teach the use of the software.

While these tutorials offer a level of detail appropriate for new professional users, this text was developed to be used as part of an introductory engineering course, taught around the use of solid modeling as an integrated engineering design and analysis tool. Since the intended audience is undergraduate students new to the field of engineering, the text contains features that help to integrate the concepts learned in solid modeling into the overall study of engineering. These features include:

- *Video Examples:* Short video tutorials accompany multiple chapters. These videos introduce students to the concepts of solid modeling and the SOLIDWORKS commands that they will use in the chapter following the step-by-step tutorials. These videos cover:
  — Getting started with modeling (Chapter 1);
  — Making 2-D drawings (Chapter 2);
  — Using symmetry when creating parts (Chapter 3);
  — Creating parts with lofts and sweeps (Chapter 4);
  — Making assemblies from part files (Chapter 6);
  — Making parts with 3-D printing (Chapter 13);
  — Setting up the SOLIDWORKS interface (Appendix A).
- *Design Intent Boxes:* These are intended to augment the "keystroke-level" tutorials to include the rationale behind the sequence of operations chosen to create a model.
- *Future Study Boxes:* These link the material contained in the chapters to topics that will be seen later in the academic and professional careers of new engineering students. They are intended to motivate interest in advanced study in engineering, and to place the material seen in the tutorials within the context of the profession.

While these features are intended to provide additional motivation and context for beginning engineering students, they are self-contained, and may be omitted by professionals who wish to use this text purely for the software tutorials.

New in This Edition

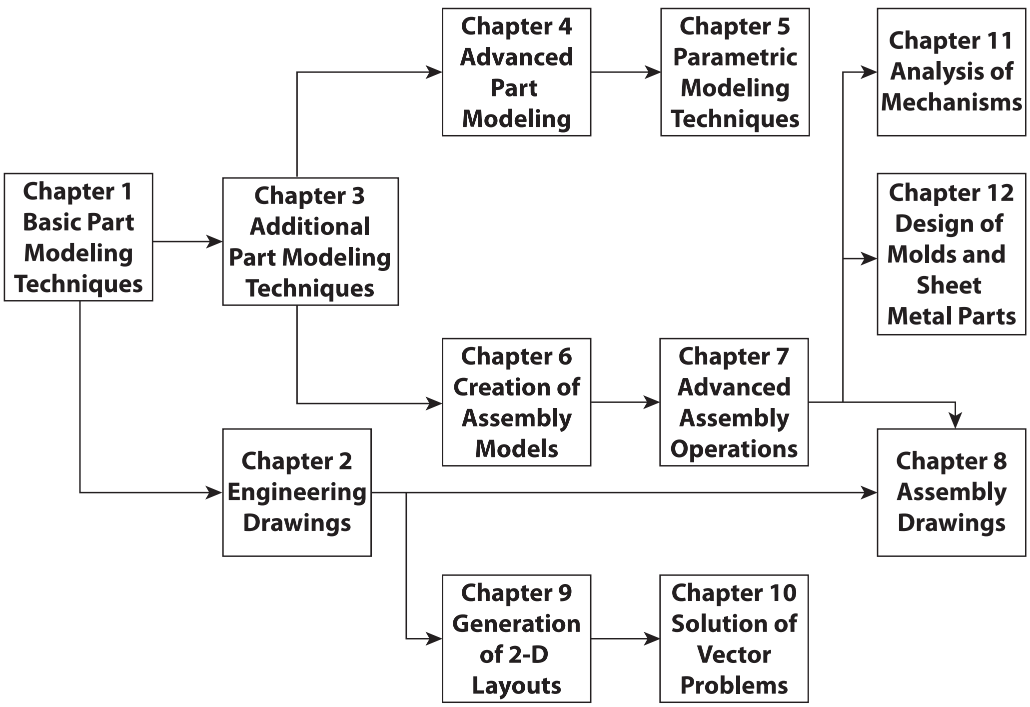
This new edition of the text has been fully updated for the SOLIDWORKS 2020 software package. All tutorials and figures have been modified for the new version of the software. Additionally, all videos have been updated to reflect the latest software.

The Organization of This Text

The organization of the chapters of the book reflects the authors' preferences in teaching the material, but allows for several different options. We have found that covering drawings early in the course is helpful in that we can have students turn in drawings rather than parts as homework assignments. The eDrawings feature, which is covered in Chapter 2, is especially useful in that eDrawings files are small (easy to e-mail), self-contained (not linked to the part file), and can be easily marked up with the editing tools contained in the eDrawings program.

The flowchart below illustrates the relations between chapters, and can be used to map alternative plans for coverage of the material. For example, if it is desired to cover assemblies as soon as possible (as might be desired in a course that includes a project) then the chapters can be covered in the order 1-3-4-6-7-2-8, with the remaining chapters covered in any order desired. An instructor who prefers to cover parts, assemblies, and drawings in that order may cover the chapters in the order 1-3-4-5-6-7-2-8 (skipping section 5.4 until after Chapter 2 is covered), again with the remaining chapters covered in any order.

Chapters 9 and 10 may be omitted in a standard solid modeling course; however, they can be valuable in an introductory engineering course. Engineering students will almost certainly find use at some point for the 2-D layout and vector mechanics applications introduced in these chapters. Chapter 13 is intended to wrap up the course with a discussion of how solid modeling is used as a tool in the product development cycle. Appendix A summarizes the recommended settings to the SOLIDWORKS program that are used throughout the book, while Appendix B shows options for customizing the SOLIDWORKS interface.



Resources for Instructors

Additional resources are available on the web at *www.mhhe.com/howard/2020.* Included on the website are tutorials for three popular SOLIDWORKS Add-Ins: SOLIDWORKS Simulation®, SOLIDWORKS Motion™, and PhotoView 360™, the video examples, and the book figures in PowerPoint format. Instructors can also access PowerPoint files for each chapter and model files for all tutorials and end-of-chapter problems as well as a teaching guide (password-protected; contact your McGraw-Hill representative for access).

Acknowledgments

We are grateful to our friends at McGraw-Hill, especially Beth Bettcher and Heather Ervolino, for their support and encouragement during this project. In particular, we offer special thanks to Karen Fleckenstein of Fleck's Communications, Inc. who did the page layouts. Also, thanks to Tim Maruna, who encouraged us to initiate this project.

At SOLIDWORKS Corporation, Marie Planchard has provided continuous support for the project. The authors are also appreciative of the support of our SOLIDWORKS resellers, Computer Aided Technology, Inc. and TriMech Solutions.

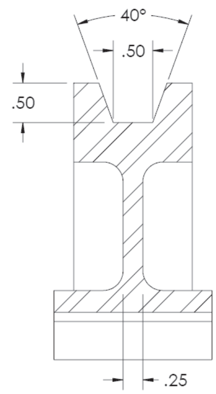
We also want to thank the reviewers whose comments have undoubtedly made the book better.

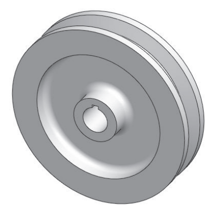
Many of our students and colleagues used early versions of the manuscript and materials that eventually became this text. We thank them for their patience and helpful feedback along the way.

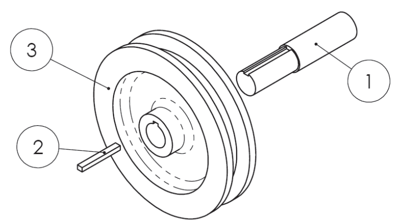
Ed Howard
Joe Musto

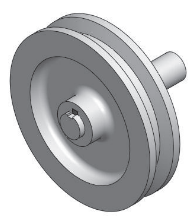
PART ONE

Learning SOLIDWORKS®



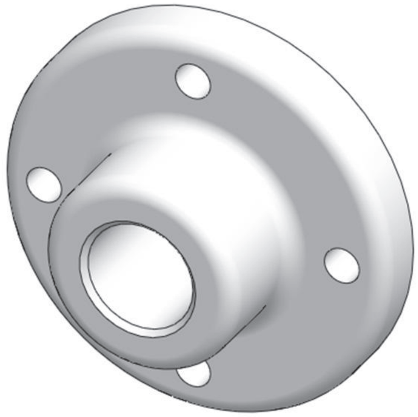






CHAPTER 1

Basic Part Modeling Techniques



Introduction

Solid modeling has become an essential tool for most companies that design mechanical structures and machines. As recently as the 1990s, this would have been hard to imagine. While 3-D modeling software existed, it was very expensive and required high-end computer workstations to run. An investment of $50,000 or more was required for every workstation with software, not including training of the operator. As a result, only a few industries used solid modeling, and the trained operators tended to work exclusively with the software. The dramatic performance improvements and price drops of computer hardware, along with increased competition among software vendors, have significantly lowered the cost barrier for companies to enter the solid modeling age. The software has also become much easier to use, so that engineers who have many other job functions can use solid modeling when required without needing to become software specialists. The SOLIDWORKS® program was among the first solid modeling programs to be written exclusively for the Microsoft Windows environment. Since its initial release in 1995, it has been adopted by thousands of companies worldwide. This text is laid out as a series of tutorials that cover most of the basic features of the SOLIDWORKS program. Although these tutorials will be of use to anyone desiring to learn the software, they are written primarily for freshmen engineering students. Accordingly, topics in engineering design are introduced along the way. "Future Study" boxes give a preview of coursework that engineering students will encounter later, and relate that coursework to the solid modeling tutorials. In this first chapter, we will learn how to make two simple parts with SOLIDWORKS software.

Chapter Objectives

In this chapter, you will:

- be introduced to the role of solid modeling in engineering design,
- learn how to create 2-D sketches and create 3-D extruded and revolved geometry from these sketches,
- use dimensions and relations to define the geometry of 2-D sketches,
- add fillets, chamfers, and circular patterns of features to part models,
- learn how to modify part models, and
- define the material and find the mass properties of part models.

SOLIDWORKS is a registered trademark of Dassault Systémes SolidWorks Corporation.

1.1 Engineering Design and Solid Modeling

The term *design* is used to describe many endeavors. A clothing designer creates new styles of apparel. An industrial designer creates the overall look and function of consumer products. Many design functions concentrate mainly on aesthetic considerations—how the product looks, and how it will be accepted in the marketplace. The term *engineering design* is applied to a process in which fundamentals of math and science are applied to the creation or modification of a product to meet a set of objectives.

Engineering design is only one part of the creation of a new product. Consider a company making consumer products, for example bicycles. A marketing department determines the likely customer acceptance of a new bike model and outlines the requirements for the new design. Industrial designers work on the preliminary design of the bike to produce a design that combines functionality and styling that customers will like. Manufacturing engineers must consider how the components of the product are made and assembled. A purchasing department will determine if some components will be more economical to buy than to make. Stress analysts will predict whether the bike will survive the forces and environment that it will experience in service. A model shop may need to build a physical prototype for marketing use or to test functionality.

During the years immediately following World War II, most American companies performed the tasks described above more or less sequentially. That is, the design engineer did not get involved in the process until the specifications were completed, the manufacturing engineers started once the design was finalized, and so on. From the 1970s through the 1990s, the concept of *concurrent engineering* became widespread. Concurrent engineering refers to the process in which engineering tasks are performed simultaneously rather than sequentially. The primary benefits of concurrent engineering are shorter product development times and lower development costs. The challenges of implementing concurrent engineering are mostly in communications—engineering groups must be continuously informed of the actions of the other groups.

Solid modeling is an important tool in concurrent engineering in that the various engineering groups work from a common database: the solid model. In a 2-D CAD (Computer-Aided Design) environment, the design engineer produced sketches of the component, and a draftsman produced 2-D design drawings. These drawings were forwarded to the other engineering organizations, where much of the information was then duplicated. For example, a toolmaker created a tool design from scratch, using the drawings as the basis. A stress analyst created a finite element model, again starting from scratch. A model builder created a physical prototype by hand from the drawing parameters. With a solid model, the tool, finite element model, and rapid prototype model are all created directly from the solid model file. In addition to the time savings of avoiding the steps of recreating the design for the various functions, many errors are avoided by having everyone working from a common database. Although 2-D drawings are usually still required, since they are the best way to document dimensions and tolerances, they are linked directly to the solid model and are easy to update as the solid model is changed.

A mechanical engineering system (assembly) may be composed of thousands of components (parts). The detailed design of each component is important to the operation of the system. In this chapter, we will step through the creation of simple components. In future chapters, we will learn how to make 2-D drawings from a part file, and how to put components together in an assembly file.

1.2 Part Modeling Tutorial: Flange

This tutorial will lead you through the creation of a simple solid part. The part, a flange, is shown in **Figure 1.1** and is described by the 2-D drawing in **Figure 1.2**.

FIGURE 1.1

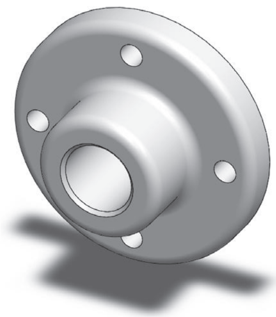
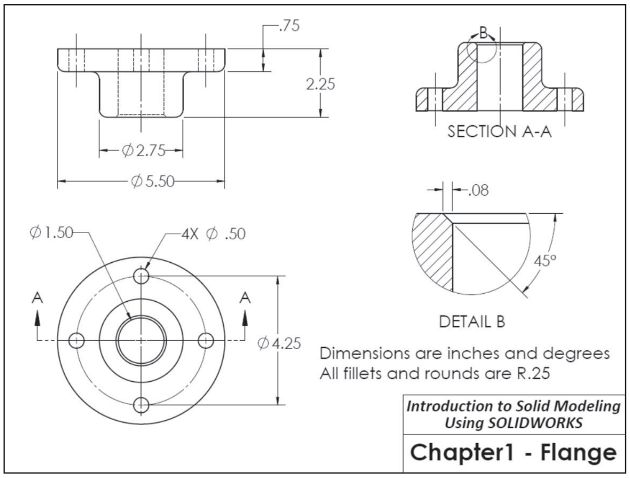


FIGURE 1.2



Begin by double-clicking the SOLIDWORKS icon on your desktop. The Welcome dialog box opens, as shown in Figure 1.3. From this box, we can begin a new document (part, assembly, or drawing) or select a recently-opened document. Click Part from the New group. If the Units and Dimension Standard box appears, as shown in Figure 1.4, select "IPS" as the units and "ANSI" as the standard. Click OK.

FIGURE 1.3

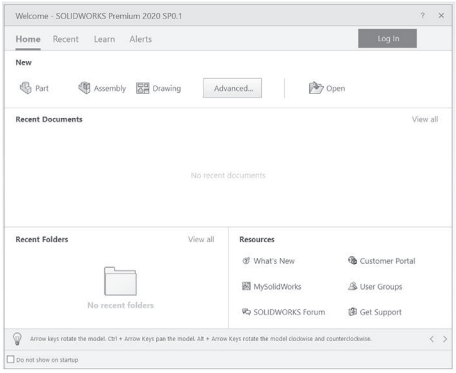
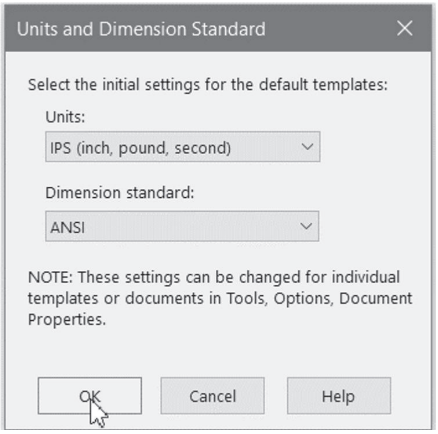


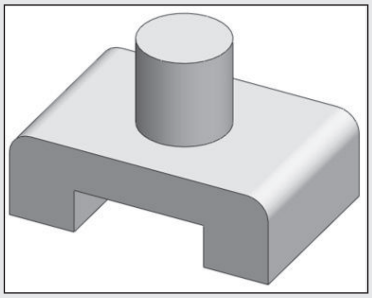
FIGURE 1.4

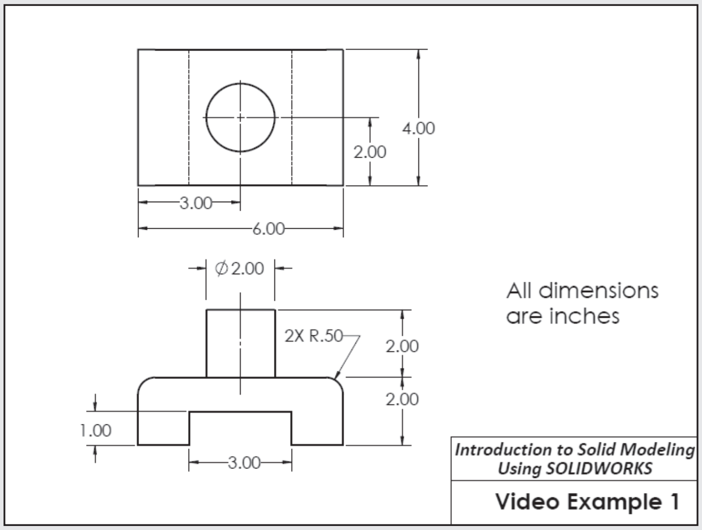


VIDEO EXAMPLE 1

In this chapter, we begin by making and dimensioning 2-D sketches and then creating 3-D features from extrusions of the sketches.

Creation of the simple part shown here, with the dimensions as shown in the drawing to the right, is demonstrated in a video at *www.mhhe.com/howard2020*. (We will learn to make drawings from 3-D parts in Chapter 2.)





In this chapter, we will be making adjustments to the SOLIDWORKS interface. These adjustments are summarized in Appendix A and in Video Example 7, which is available at *www.mhhe.com/howard2020*.

The Units and Dimension Standard box only appears the first time SOLIDWORKS is opened. The selections become the default values for all new files. In this chapter, we will see how to set these values for individual files and to change the default values.

Note that you can return to the Welcome dialog at any point by selecting the icon shown in **Figure 1.5**.

FIGURE 1.5

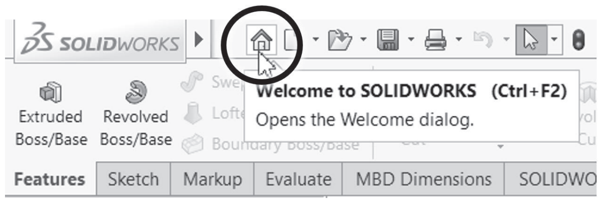
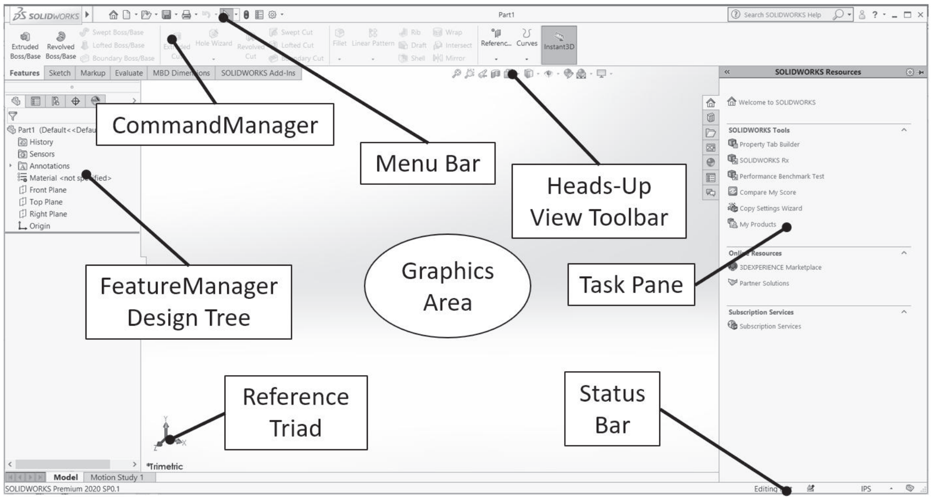


FIGURE 1.6



Before we begin modeling the flange, we will establish a consistent setup of the SOLIDWORKS environment. The default screen layout is shown in Figure 1.6. The graphics area occupies most of the screen. The part, drawing, or assembly will be displayed in this area. At the top of the screen is the Menu Bar, which contains the Main Menu and a toolbar with several commonly-used tools such as Save, Print, and Redo. Note that if you pass the cursor over the SOLIDWORKS button in the Menu Bar, the Main Menu will "fly out," or be temporarily displayed, as shown in Figure 1.7. The fly-out feature is designed to save room on the screen. However, since we will be using the menu often, we will disable the fly-out so that the menu is always displayed.

Move the cursor over the SOLIDWORKS button to display the menu. Click on the pushpin icon at the right side of the menu, as shown in Figure 1.8, to lock the display of the menu.

FIGURE 1.7

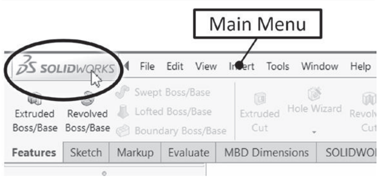
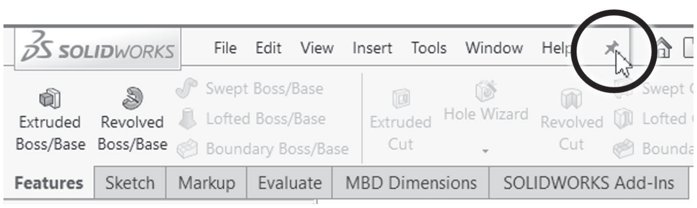


FIGURE 1.8



The CommandManager contains most of the tools that you will use to create parts. When working in the part mode, there are two categories of tools that we will use extensively: Sketch tools used in creating 2-D sketches, and Features tools used to create and modify 3-D features. Clicking on the Sketch and Features tabs at the bottom of the CommandManager, as shown in Figure 1.9, changes the tools on the CommandManager to those of the selected group. By default, there are several other groups available besides the Sketch and Features groups. To simplify the interface, we will hide these groups for now.

FIGURE 1.9

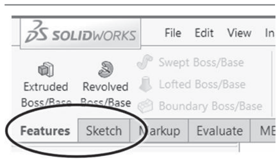
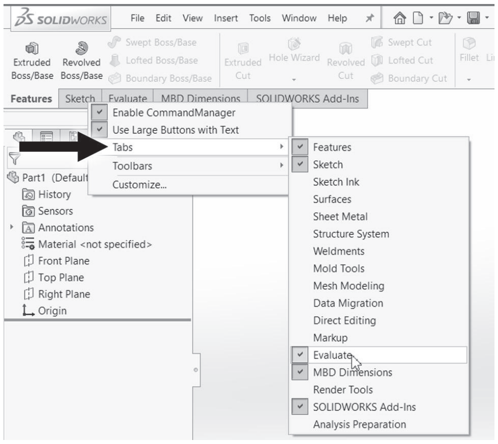


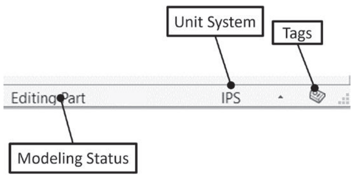
FIGURE 1.10



Right-click on one of the CommandManager tabs and move the cursor over Tabs in the menu that appears, as shown in Figure 1.10. A list of available groups is displayed, with a check mark shown beside each active group. Click on any of the active groups other than Features and Sketch. This will clear the check mark and turn off the display of that group. Repeat until only the Features and Sketch groups remain active.

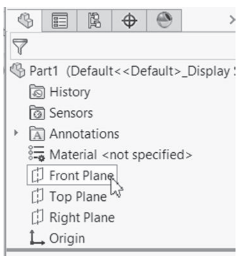
At the right side of the screen is the Task Pane. The Task Pane is a fly-out interface for accessing files and online resources. We will not use the Task Pane that often, but since it takes up very little room in its normal collapsed state, we will leave it on. If you would like to turn it off completely, select View: User Interface from the Main Menu and click on Task Pane.

FIGURE 1.11



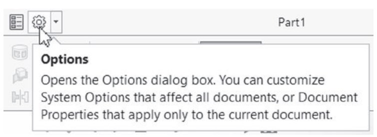
At the bottom of the screen is the Status Bar. When you move the cursor over any toolbar icon or menu command, a message on the left side of the Status Bar describes the command. Other information appears at the right side of the Status Bar, as shown in Figure 1.11. The unit system in use is displayed and can be changed directly from the Status Bar. Another feature, called Tags, allows keywords to be associated with files and features. We will not be using Tags in this book. Although the display of the Status Bar can be toggled off and on from the View menu, we recommend leaving it on.

FIGURE 1.12



Just to the left of the drawing area is the FeatureManager® Design Tree. The steps that you will execute to create the part will be listed in the FeatureManager. This information is important when the part is to be modified. When you open a new part, the FeatureManager lists an origin and three predefined planes (Front, Top, and Right), as shown in Figure 1.12. As you select each plane with your mouse, the plane is highlighted in the graphics area. We can create other planes as needed, and will do so later in this tutorial.

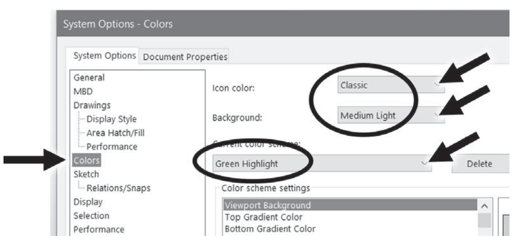
FIGURE 1.13



At the top of the graphics area is the Heads-Up View Toolbar. This toolbar contains many options for displaying your model. We will explore these options later in this tutorial.

We will now set some of the program options.

FIGURE 1.14



Select the Options Tool from the Menu Bar toolbar, as shown in Figure 1.13. (You can also access the options from the Main Menu, by selecting Tools: Options.)

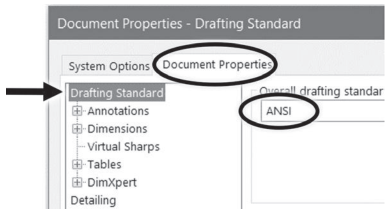
The dialog box contains settings for both the system and for the specific document that is open.

Under the System Options tab, choose Colors and change the icon color to "Classic" and the color scheme to "Green Highlight," as shown in Figure 1.14. The Background should be set to "Light" or "Medium Light."

The Classic option for icon colors makes many of the icons display in colors other than the default blue and black, making them easier to recognize for new users. The Green Highlight scheme causes currently selected items to be highlighted in green, as the name implies. The default option is for selected items to be highlighted in light blue. Since another shade of blue is used for other purposes, green highlighting is used in this book to avoid confusion. Since these changes were made to the System Options, they will remain in effect for future SOLIDWORKS sessions. The changes below, which will be made to the Document Properties, will apply only to the current part model.

Select the Document Properties tab. In the list of options, Drafting Standard will be highlighted. Select ANSI from the pull-down menu, as shown in Figure 1.15.

FIGURE 1.15



ANSI is the American National Standards Institute, an organization that formulates and publishes the standard drawing practices used by most companies in the United States. European companies are more likely to use the standards of ISO, the International Organization for Standardization.

Also under the Document Properties tab, select Dimensions. Use the pull-down menu by the Primary precision box to set the number of decimal places to 3 (.123), as shown in Figure 1.16. (Ignore the message that the drafting standard has been changed to "ANSI-MODIFIED.") Select Grid/Snap and check the box labeled "Display Grid," as shown in Figure 1.17. Also, select Units and set the unit system to IPS (inches, pounds, and seconds), the primary length precision to .123, and the precision for angles to None, as shown in Figure 1.18.

FIGURE 1.16

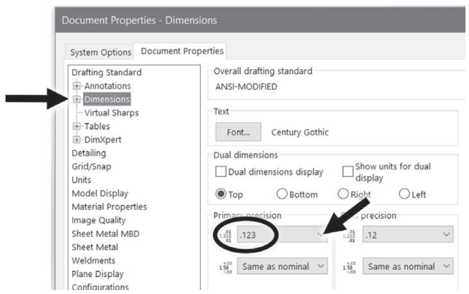


FIGURE 1.17

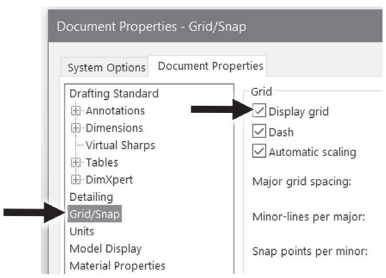
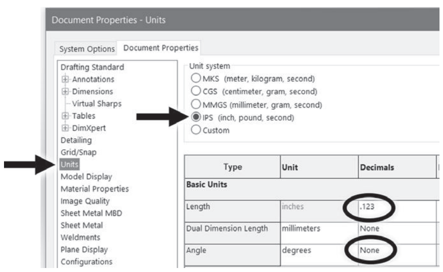


FIGURE 1.18



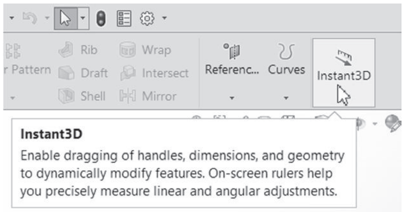
Note that there are "Dual Dimension" units that can be set in the Units options. For some drawings, you may want to show dimensions in both US units (inches) and SI units (typically millimeters). Since we will not use dual dimensions for this part, it is not necessary to change those settings. Also note that we have set the decimal display to .123 in two separate locations. The display of decimal places can be changed at either location. You may also change the font style and size of the dimension text by clicking the Font button. For clarity, the figures in this book were made with a dimension font larger than the default size.

Click OK to close the dialog box.

Any of the options just set can be changed at any time during the modeling process. Later in this chapter, we will learn how to create a *template* that allows us to begin a new part with our preferred settings in place.

We will make two more changes to the default settings before beginning our part. A feature called "Instant 3D" allows for changes to be made by clicking and dragging on model faces, without entering dimensions from the keyboard. While this feature can be handy for experienced users, it is recommended that new users avoid using Instant 3D in order to prevent unintended changes to the model. Similarly, a feature called "Instant 2D" allows for dimensions in sketches to be changed by clicking and dragging rather than entering a numerical value. This feature will also be turned off.

FIGURE 1.19



Select the Features tab of the CommandManager. If the Instant 3D Tool is turned on (the icon will be "depressed," as shown in Figure 1.19), click to turn it off. Select the Sketch tab of the CommandManager and turn off the Instant 2D Tool as well.

We start the construction of the flange by sketching a circle and extruding it into a 3-D disk.

FIGURE 1.20

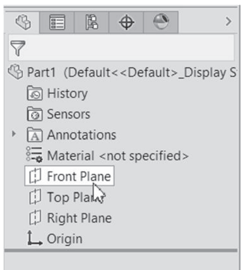
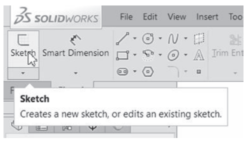


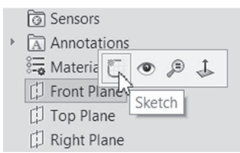
FIGURE 1.21



Select the Front Plane by clicking on it in the FeatureManager Design Tree, as shown in Figure 1.20.

The Front Plane will be highlighted in green. The color green indicates that an item is the currently selected entity (since we chose the "Green Highlight" color scheme).

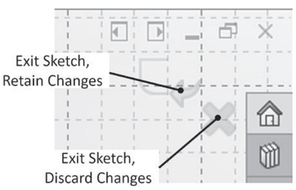
FIGURE 1.22



Begin a sketch by selecting the Sketch tab of the CommandManager, and then the Sketch Tool, as shown in Figure 1.21.

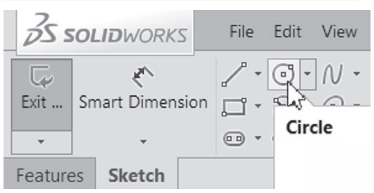
Note that when you selected the Front Plane, a pop-up menu appeared that allowed you to open a sketch on that plane, as shown in Figure 1.22. The SOLIDWORKS program has many of these context-sensitive menus built in. As you become proficient with the program, you may find many of these built-in shortcuts to be handy.

FIGURE 1.23



When you open a sketch, a grid pattern appears, signifying that you are in the sketching mode. Also, Exit Sketch icons appear in the upper-right corner of the screen, as shown in Figure 1.23.

FIGURE 1.24



Select the Circle Tool from the Sketch group of the CommandManager, as shown in Figure 1.24.

When selecting any tool which has a pull-down menu (designated by the down arrow to the right of the icon), use caution to be sure that you are selecting the proper tool. In the case of the Circle Tool, there are two possible methods for defining the circle: by the center point and a point on the perimeter, or by three points on the perimeter. Clicking on the down arrow displays these options, so that the proper tool can be selected (**Figure 1.25**). By default, the option for defining the circle by locating the center point and a point on the perimeter is selected by clicking on the Circle Tool without accessing the pull-down menu. However, if the last option selected was to define the circle by three points on the perimeter, then that option becomes the default for the next selection. When that occurs, the icon shown for the Circle Tool will change, as shown in **Figure 1.26**. Because many of the icons are similar and are very small, you should use caution with tools that have pull-down menus.

FIGURE 1.25

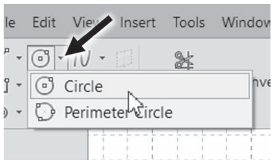
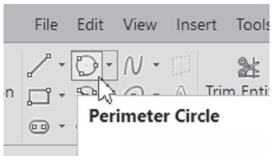
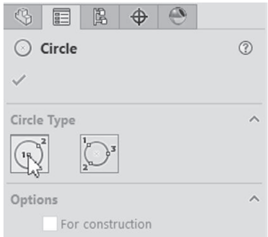


FIGURE 1.26



You can check to see that you have selected the proper tool by looking at the PropertyManager, which appears in the area where the FeatureManager is normally shown whenever a tool is activated or an object is selected. The PropertyManager now shows the two alternative methods for defining a circle (**Figure 1.27**). If we selected the wrong tool accidentally, then we can change the method for defining the circle in the PropertyManager.

FIGURE 1.27



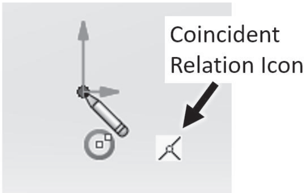
In the PropertyManager, make sure that the icon representing the first construction method is selected, as shown in Figure 1.27. If it is not, then click it to select it.

Notice as you move the cursor into the drawing area that it changes appearance into a pencil icon with a circle next to it, as shown in **Figure 1.28**. This lets you know that the Circle Tool is active.

FIGURE 1.28

Move the tip of the pencil icon toward the origin until a dot appears at the origin, as shown in Figure 1.29; this indicates that you will snap to an existing point (in this case the origin) when you click with the mouse. Also, note the small icon next to the origin that signifies a coincident relation: the origin and the center point of the circle will share the same location.

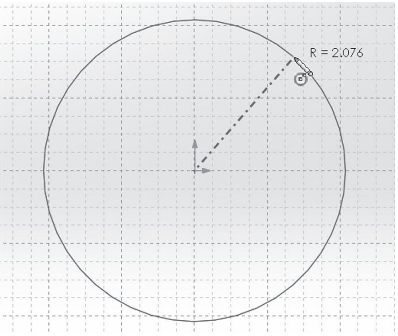
FIGURE 1.29



A snap adds a relation to the positions of two entities. In this example, when you snap to the origin, the circle will be centered at the exact coordinates of x = 0 and y = 0. The relation added when one entity is created by snapping to another can be edited later, if desired. The addition of a snap automatically is a nice feature of the SOLIDWORKS program: snaps are intuitive. It is not necessary to enter the numerical coordinates of the center of the circle.

With the center point highlighted as in Figure 1.29, click the left mouse button to place the center of the circle at the origin. Drag the mouse outward to create a circle, as shown in Figure 1.30. Click the left mouse button again to define a point on the perimeter and create the circle. The size of the circle drawn is not important; we will add a dimension to define its diameter precisely.

FIGURE 1.30



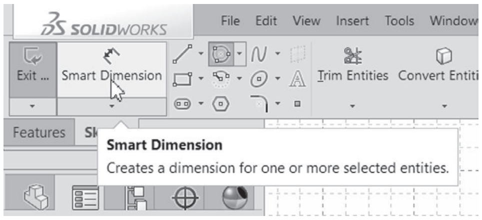
The circle will appear in green, indicating that it is the currently selected item.

Press the Esc key twice to close the Circle Tool and deselect the circle just drawn.

The circle should now appear in blue. In the Status Bar at the bottom of the screen, notice that "Under Defined" appears. This is because we have not set the diameter of the circle yet. When a sketch does not contain enough dimensions and/or relations to define its size and position in space, it is said to be under defined, and is denoted by blue entities.

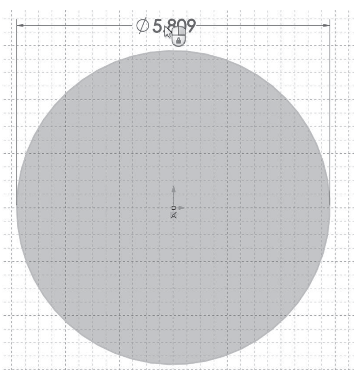
Other possible conditions of the sketch are "Fully Defined," when the sketch contains exactly enough dimensions and/or relations to define its size and position in space (denoted by black entities), and "Over Defined," where the sketch has at least one dimension or relation that contradicts or is redundant to the other dimensions and relations (denoted by red entities). Over defined sketches should be avoided.

FIGURE 1.31



Also note that the area of the circle is shaded. By default, closed contours within sketches are shown shaded. The shading can be toggled on and off by clicking the Shaded Sketch Contours Tool from the Sketch group of the CommandManager or from Tools: Sketch Settings from the Main Menu.

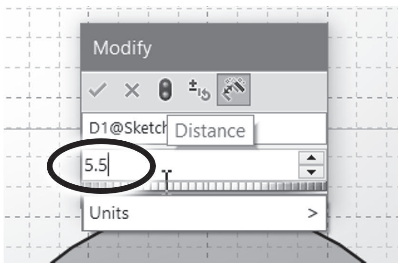
FIGURE 1.32



Select the Smart Dimension Tool from the CommandManager, as shown in Figure 1.31. Click with the left mouse button anywhere on the circle. A dimension will be added to the diameter of the circle. Drag the dimension to a convenient location, as shown in Figure 1.32. When the dimension is where you want to put it, left-click again to place the dimension.

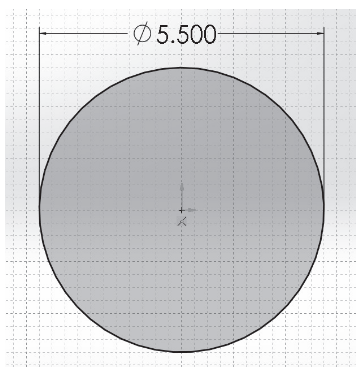
A dialog box displaying the SOLIDWORKS name of the dimension and prompting for its value will be displayed, as shown in Figure 1.33. Enter "5.5" in the box. (You don't need to enter any units, since inches are the default units, set earlier.) Press the Enter key or click on the check mark to update the dimension. Press the Esc key to turn off the Smart Dimension Tool and deselect the dimension.

FIGURE 1.33



Notice that the circle is redrawn to the correct dimension, as shown in Figure 1.34. (Note that the figures in the book use a larger font size than the default value. We will see how to change the font size later in this chapter.) The dimension in inches is displayed, and the circle is black. Notice at the bottom of the screen in the Status Bar that the sketch is now Fully Defined. (Note: If we had not snapped to the origin for the circle's center, the sketch would still be under defined because the circle's location within the Front Plane would not be specified.)

FIGURE 1.34

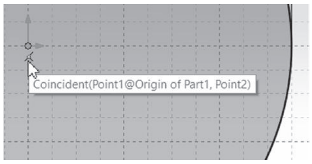


If you double-click on the dimension value, the dialog box reappears and you can change the dimension. Try this, and then use the Undo Tool from the main menu toolbar, shown in Figure 1.35, to return the dimension to 5.5 inches.

FIGURE 1.35

Next to the center of the circle, an icon shows that a relation is applied. By moving the cursor over the relation icon, details about the relation can be viewed, as shown in Figure 1.36. Relations can be deleted by clicking on the relation icon to select it, and then pressing the Delete key. The display of sketch relations can be toggled on and off by selecting View: Hide/Show from the Main Menu and clicking Sketch Relations. Some experienced users prefer to not show the relations because they can cause a sketch to appear cluttered, but new users are advised to keep the relation display turned on.

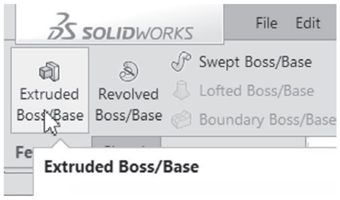
FIGURE 1.36



Now we are ready to turn this 2-D sketch into a 3-D part with the Extruded Boss/Base Tool.

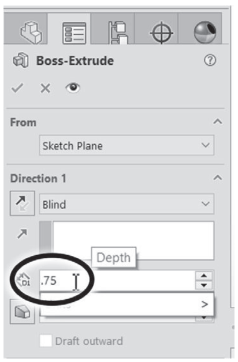
Select the Features tab of the CommandManager and the Extruded Boss/Base Tool, as shown in Figure 1.37.

FIGURE 1.37



The base feature is the first solid feature created. Any subsequent solid features are called bosses. Note that the view of the part changes to display a 3-D preview of the extruded solid. On the left side of the screen, the PropertyManager is now active. The PropertyManager allows the properties of the selected entity to be viewed and edited. There are several options available for the extrusion, including adding draft (taper) to the part, but for now we only need to adjust the depth of the extrusion.

FIGURE 1.38



Set the depth of the extrusion to 0.75 inches, as shown in Figure 1.38. Press Enter, and the preview in the graphics area will be updated to reflect the new thickness, as shown in Figure 1.39. Click on the check mark (OK) in the PropertyManager, as shown in Figure 1.40, and the circle is extruded into a solid disk, as shown in Figure 1.41.

FIGURE 1.39

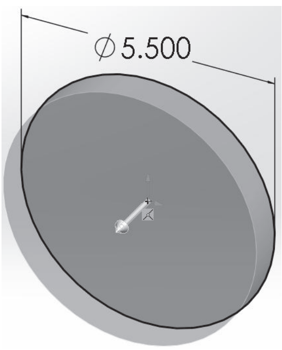


FIGURE 1.40

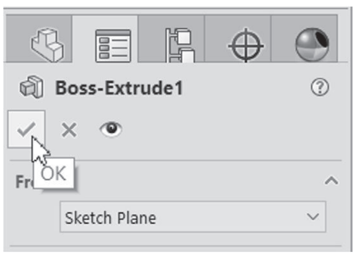


FIGURE 1.41

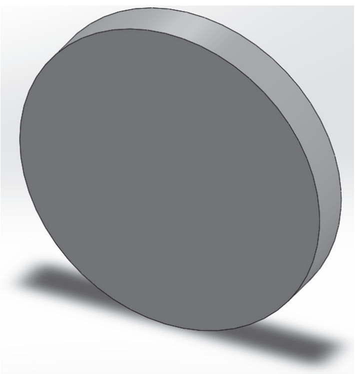
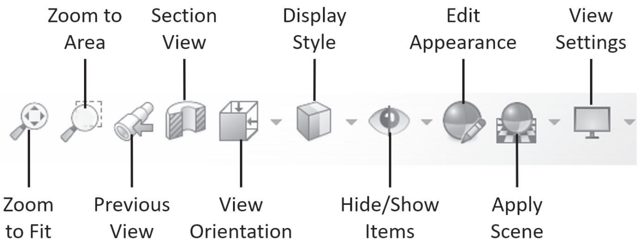
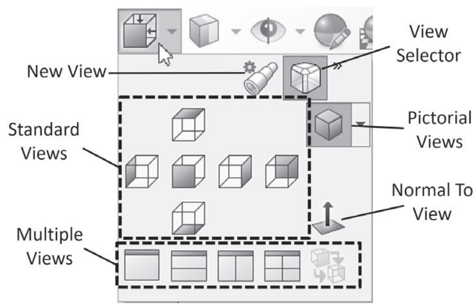


FIGURE 1.42



Now that we have a solid part, we can examine the functions of the viewing tools. The viewing tools are located on the Heads-Up View Toolbar at the top of the graphics area. The default configuration of the Heads-Up View Toolbar is shown in **Figure 1.42**. The Zoom to Fit Tool adjusts the zoom so that the entire model can be viewed. The Zoom to Area Tool allows a viewing window to be selected by dragging out an area on the screen. The Previous View Tool returns the view orientation and zoom level to the configuration prior to the most recent change of view. The Section View Tool displays a cross section of the part. We will use this tool in later chapters.

FIGURE 1.43



The View Orientation Tool opens a menu of standard view options, as shown in **Figure 1.43**. The six principal or standard views—Front, Back, Top, Bottom, Left, and Right—can be displayed by clicking on the appropriate icon. At the right side of the menu is a pull-down menu of three pictorial views: Isometric, Trimetric, and Dimetric, as shown in **Figure 1.44**. In an isometric view, the view orientation is such that the angles between the displayed edges of a cube are equal, as shown in **Figure 1.45**. In a dimetric view, two of the angles are equal, and in a trimetric view all angles are different. The Trimetric View in SOLIDWORKS emphasizes the display of the front of the part, and is the default pictorial view. (It is good practice to orient a model so that the front view is the view that is most descriptive of the part.) At the bottom of the menu are tools for displaying multiple views in separate windows on the screen. We will demonstrate their use later in this chapter. The Normal To Tool aligns the view to be perpendicular to a selected plane or surface. This tool is useful when sketching in a plane that is not perpendicular to any of the principal views. When the View Selector Tool is turned on, then the part is shown in a "box," as shown in **Figure 1.46**. Clicking on a side, edge, or corner of the box changes the view to one that is normal to the selection. The View Selector Tool can be toggled on or off; in this book we will leave it turned off. Another way to access the View Orientation Menu is to press the space bar. This is an example of a keyboard shortcut. Another useful keyboard shortcut is to press the z key to zoom out from a model view and press the z key while holding the Shift key down to zoom in.

FIGURE 1.44

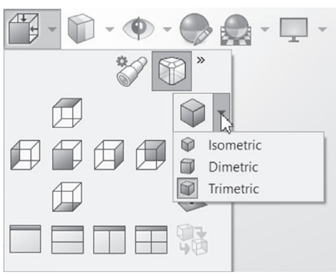


FIGURE 1.45

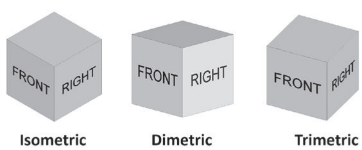
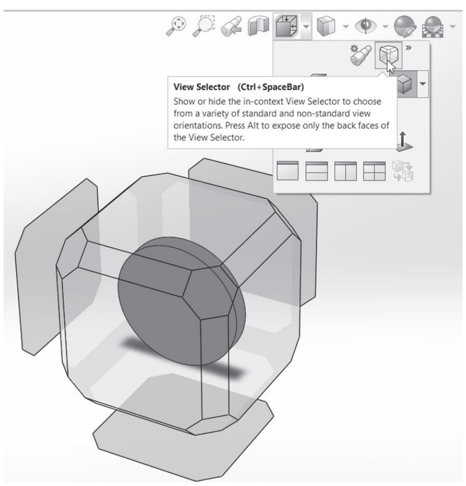
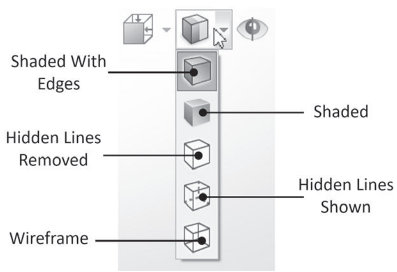


FIGURE 1.46



The Display Style Tool opens a pull-down menu of options for displaying the model, as shown in **Figure 1.47**. There are two shaded modes, with or without the edges shown by lines, and three wireframe modes, with hidden edges removed, shown as dashed lines, or shown as solid lines. Usually, we work with one of the shaded modes, but for some operations, displaying the model in wireframe mode is preferred.

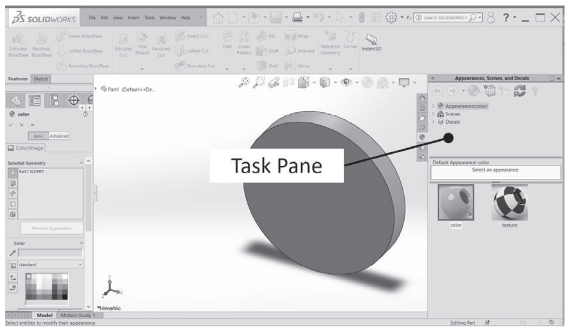
FIGURE 1.47



The Hide/Show Items Tool allows you to toggle on or off the display of several items, such as the origin, planes, axes, etc. For now, we will skip over this tool, and will explore its use later in this chapter.

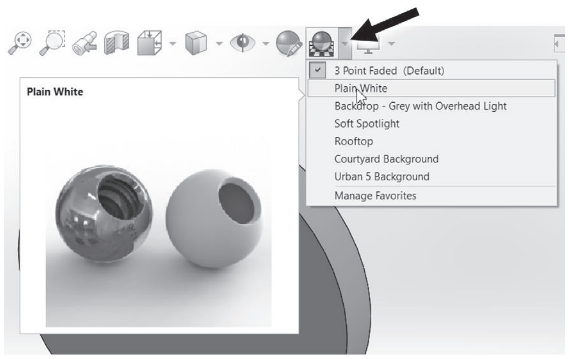
The Edit Appearance Tool allows you to change the color and optical properties (such as transparency and reflectivity) of the entire model or selected model features. When this tool is selected, the PropertyManager displays the selected feature(s), as shown in **Figure 1.48**, and palettes from which a new color can be selected. If no features are selected prior to selecting the Edit Appearance Tool, then by default the change will apply to the entire model. It is recommended that light colors be used, as dark colors can make some features and selections difficult to see. The Advanced tab contains additional options for displaying the model, such as applying textures to selected surfaces or making the part transparent or translucent. We will explore some of these options later. As shown in **Figure 1.48**, the Task Pane at the right of the screen is also displayed when the Edit Appearance Tool is shown. Colors and textures can be applied by dragging and dropping them from the task pane to the part.

FIGURE 1.48



The Apply Scene Tool allows you to select backgrounds and lighting options from a pre-defined menu, as shown in **Figure 1.49**. In this book, we will use the Plain White scene. Some of the scenes contain background graphics rather than just colors. A SOLIDWORKS Add-In program, PhotoView 360, can also be used to produce photo-realistic display images of the model in various scenes.

FIGURE 1.49



The View Settings Tool, shown in **Figure 1.50**, allows you to add shadows to either of the shaded modes. Also, the model can be shown in perspective mode. In perspective mode, sight lines converge at a single point (the vanishing point), producing a more realistic view. However, most engineering views are produced with parallel sight lines (these are called *orthographic* projections). Normally, we will leave the Perspective View option turned off. There is also a Cartoon View option, which is intended to make a model appear as though it were hand-drawn.

FIGURE 1.50

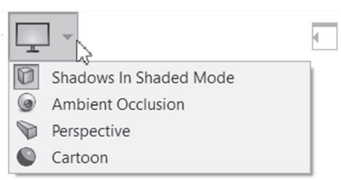
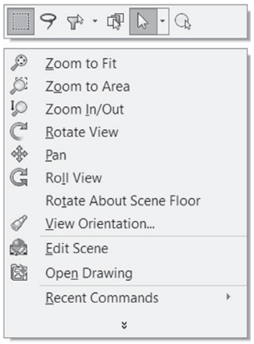


FIGURE 1.51



Several other viewing tools can be accessed by right-clicking in the white space of the graphics area. The menu shown in Figure 1.51 is displayed. A particularly useful tool is the Rotate View Tool. After selecting this tool, you can hold down the left mouse button and move the mouse to rotate the model view so that you can see all sides of the model. The Pan Tool can be used to move the model around in the graphics area, again by clicking and holding the left button while dragging the mouse. Note that when using either of these tools (or several other tools selected from this menu), that the tool remains active until it is turned off by pressing the Esc key.

Because the Rotate View and Pan Tools are used often, we will make them available on the Heads-Up View Toolbar. Also, since the Trimetric View is the default pictorial view, we will add it to the toolbar so that we do not have to go through the View Orientation Tool pull-down menu to select it.

FIGURE 1.52

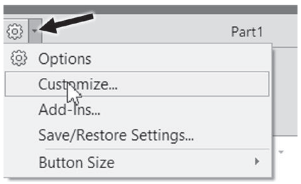
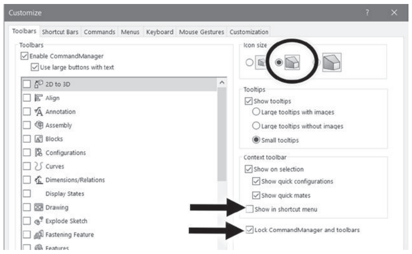
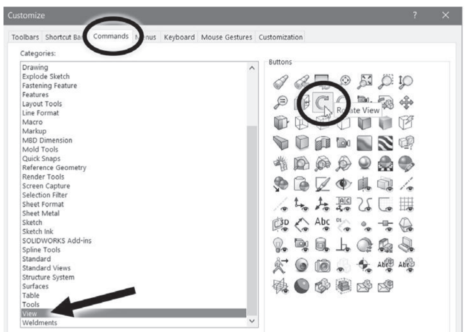


FIGURE 1.53



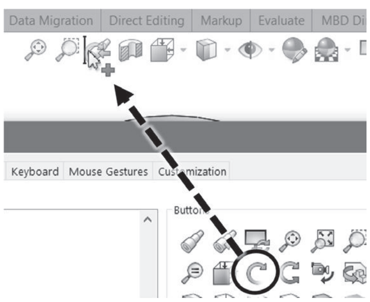
Click the arrow beside the Options Tool and select Customize from the menu, as shown in Figure 1.52. If desired, check the larger Icon size option, as shown in Figure 1.53, so that all of the icons in toolbars and the CommandManager are easier to see. You will see a message that some options will not take effect until SOLIDWORKS is restarted; click OK to close this message. Check the "Show Tooltips" box, which causes a description of a tool to be displayed when the cursor is moved over the tool's icon. Tooltips can be shown with a short description (usually just the name of the tool), or with more detail, with or without images. We will use the "Small tooltips" option. Check the box labeled "Lock the CommandManager and toolbars" to prevent unintentional moving of these items around the screen. Also, uncheck the box labeled "Show in shortcut menu," as shown in Figure 1.53. This will cause several menu items to be displayed with text instead of with icons. This option is discussed further in Appendix B.

FIGURE 1.54



Click the Commands tab, and select the View group. Locate the Rotate View Tool, as shown in Figure 1.54. Click and drag the tool to the desired location on the Heads-Up View Toolbar, as shown in Figure 1.55. Release the mouse button to place the tool. Repeat with the Pan Tool from the View group and the Trimetric View Tool from the Standard Views group. The edited toolbar is shown in Figure 1.56. Click OK to close the Customize box.

FIGURE 1.55

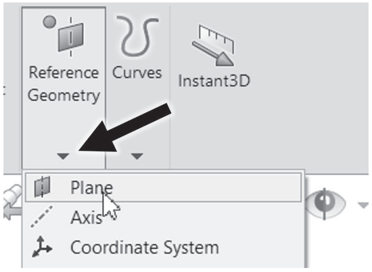


Experiment with the zoom and viewing options. When finished, select a shaded solid display (either with or without edges displayed) and the Trimetric View.

FIGURE 1.56

Now we are ready to add to our part. The next feature we will add is the 2.75-inch-diameter boss. We could sketch the circle to be extruded on the front or back face of the existing part or in the Front Plane, but instead we will create a new plane that is 2.25 inches away from the Front Plane. There are several reasons why we might want to define the part in this manner. One is that we may want to add draft, a slope to the sides of a feature that allows it to be extracted from a mold. If so, then we want our 2.75-inch dimension to apply at the top of the boss, allowing the diameter to get larger closer to its base.

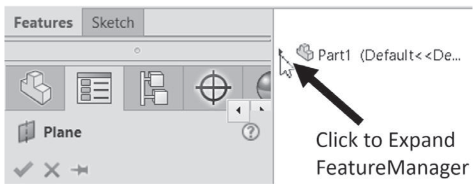
FIGURE 1.57



Select the Reference Geometry Tool from the Features group of the CommandManager. From the menu that appears, select Plane (see Figure 1.57).

Note that the FeatureManager has been replaced in its usual position by the PropertyManager, where the parameters of the new plane will be defined. However, the FeatureManager is still visible as a “fly out” list to the right of the PropertyManager. By default, the FeatureManager is shown collapsed; that is, only the name of the part is shown. The full FeatureManager can be shown by clicking on the arrow next to the part name.

FIGURE 1.58



Click the arrow next to the part name (Part1) to expand the FeatureManager, as shown in Figure 1.58. Click on the Front Plane to select it. In the box defining the offset distance, enter 2.25, as shown in Figure 1.59. Note that the new plane is previewed in the graphics area. Click the check mark and the new plane, labeled Plane1, is created, as shown in Figure 1.60.

FIGURE 1.59

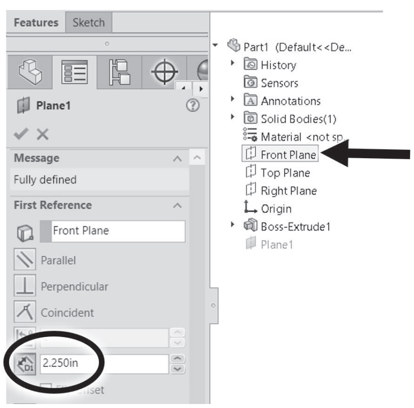
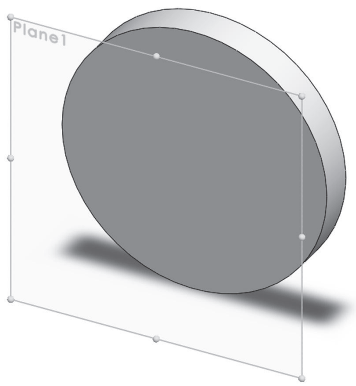


FIGURE 1.60

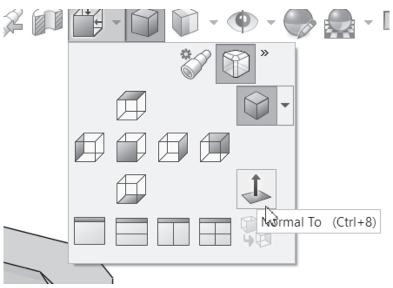


There are several options for creating a new plane. When we selected the Front Plane, the option for defining the new plane parallel to the selected plane was selected by default. Of course, the Flip box could have been checked if we wanted the new plane to be behind the Front Plane.

Now with Plane1 selected (highlighted in green), click the Sketch tab of the CommandManager and select the Circle Tool.

Remember that there are two ways to define a circle; either by defining its mid-point and a point on the perimeter or by defining three points on the perimeter. When you create a circle with either method, then that method becomes the default method of construction. If you hold the cursor over the icon momentarily, the current tool is

FIGURE 1.61



displayed in the Tooltips popup. If the correct tool is current, then you can select it by clicking on the icon, without using the pull-down menu.

Also note that a sketch was opened when you selected the Circle Tool. When a plane or face is selected, choosing a drawing tool from the Sketch group opens a new sketch and activates the tool.

From the View Orientation Tool, select the Normal To View, as shown in Figure 1.61.

FIGURE 1.62

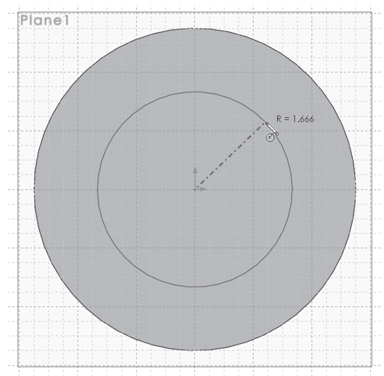
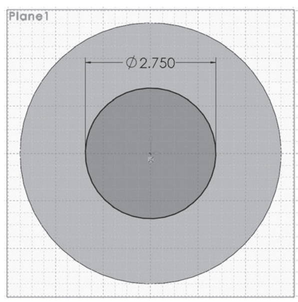


FIGURE 1.63



Notice that in this case, the Normal To View is the same as the Front View.

Move the cursor to the origin and click to place the center of the circle at the origin. Drag out a circle, as shown in Figure 1.62, and click to complete the circle.

Select the Smart Dimension Tool from the Sketch group of the Command-Manager. Add a 2.75-inch dimension to the diameter, as shown in Figure 1.63. Switch to the Trimetric View (Figure 1.64). Select the Extruded Boss/Base Tool from the Features group of the CommandManager.

As shown by the preview (Figure 1.65), the extrusion is going away from the base feature rather than toward it.

FIGURE 1.64

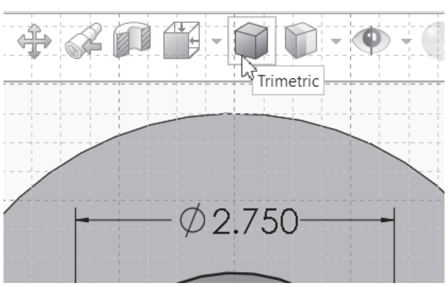


FIGURE 1.65

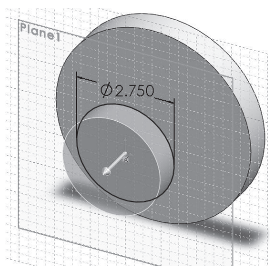


FIGURE 1.66

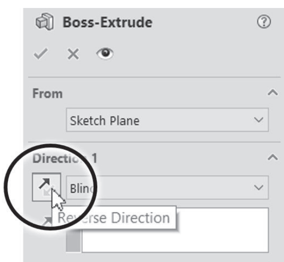


FIGURE 1.67

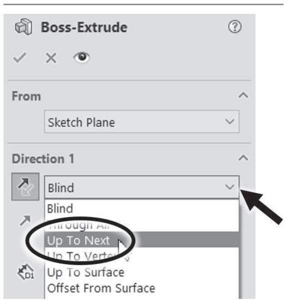
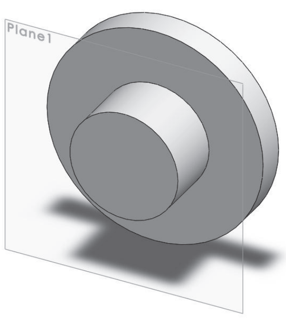


FIGURE 1.68



In the PropertyManager, click the Reverse Direction button so that the extrusion is directed toward the base (Figure 1.66). In the pull-down menu, select Up To Next as the type of extrusion, as shown in Figure 1.67. Click the check mark to complete the extrusion, which is shown in Figure 1.68.

DESIGN INTENT Planning the Model

As we build the 2.75-inch diameter boss of our flange, we can choose from two existing planes/surfaces or construct a new plane. The choice of constructing a new plane in order to allow us to add draft to our part is an example of design intent. There are many definitions of design intent.

Ours is:

Design intent is the consideration of the end use of a part, and possible changes to the part, when creating a solid model.

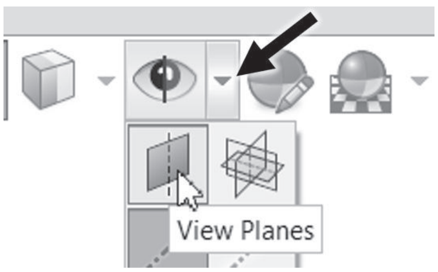
Throughout this book, we will identify examples of considering design intent when modeling.

Choosing "Up To Next" as the type of extrusion instead of defining the distance allows for changes to be made easily. If we later decide to change the distance between Plane1 and the Front Plane, then the part will rebuild correctly, as the boss extrusion will still extend back to the base feature. Also notice that by default, the "Merge result" option is checked when adding a new feature. With this option checked, the two features that we have created combine to form a single solid body. If the option is unchecked, then the features represent separate bodies.

We can turn off the display of Plane1.

From the Hide/Show Items Tool, click on the View Planes icon to toggle off the display of planes, as shown in Figure 1.69.

FIGURE 1.69



Plane1 still exists in the model, but turning off the display of planes results in a less cluttered model view. Note that Plane1 could also have been hidden by right-clicking its entry in the FeatureManager and selecting Hide.

It is a good idea to save your work periodically.

Choose File: Save from the Main Menu. Save the part with the name "Flange." The file type will be "sldprt."

Note that the new file name appears in the Menu Bar and at the top of the FeatureManager design tree.

Next we will add the center hole. This time we will select a face to define our sketch plane. As you move the cursor over the front surface, notice that a square icon appears. This indicates that a surface will be selected when you click with the left mouse button. Similarly, a line icon indicates that an edge will be selected.

FIGURE 1.70

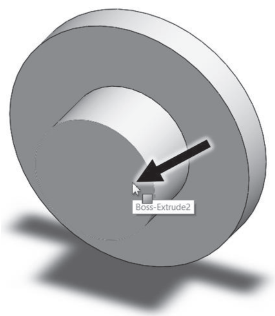
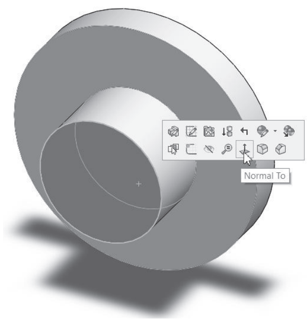


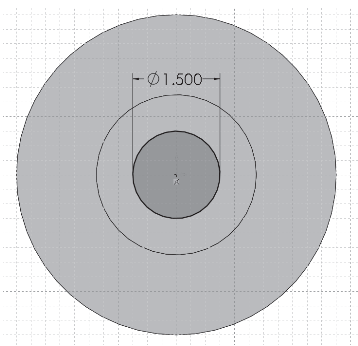
FIGURE 1.71



Move the cursor over the front face, so that the square icon appears, as shown in Figure 1.70. Click to select this face; it will be highlighted in green. A pop-up toolbar, called a Context Toolbar, will appear. Click the Normal To View, as shown in Figure 1.71.

(Note: If the Context Toolbar does not appear when you select the face, then select the Customize Tool and make sure that the box labeled "Show on selection" is checked, as shown in **Figure 1.53**.)

FIGURE 1.72



Click the Sketch tab of the CommandManager and select the Circle Tool. Drag out a circle centered at the origin. Select the Smart Dimension Tool and dimension the circle diameter as 1.5 inches, as shown in Figure 1.72. Click the Features tab of the CommandManager and select the Extruded Cut Tool, as shown in Figure 1.73. Select the type as Through All in the PropertyManager, as shown in Figure 1.74. Click the check mark to complete the cut. Switch to the Trimetric View. The result of this operation is shown in Figure 1.75.

FIGURE 1.73

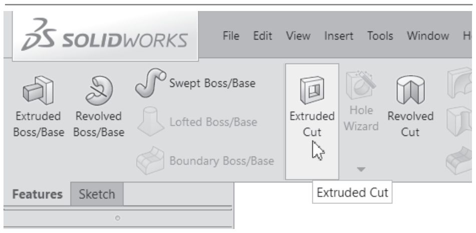


FIGURE 1.74

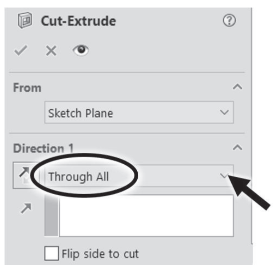


FIGURE 1.75

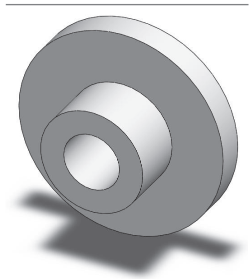
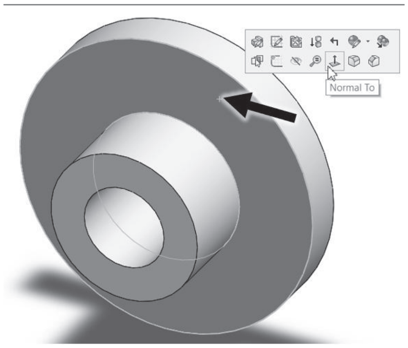


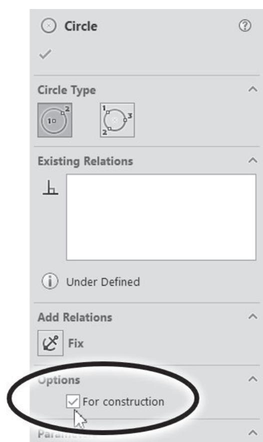
FIGURE 1.76



We will now add the four bolt holes.

Select the surface shown in Figure 1.76. Switch to the Normal To View. Click the Sketch tab of the CommandManager, and select the Circle Tool. Drag out a circle centered at the origin. In the PropertyManager, check the "For construction" box, as shown in Figure 1.77.

FIGURE 1.77



Construction entities help you locate and size sketch parameters, and are indicated by dashed-dotted lines. The circle just drawn represents the bolt circle.

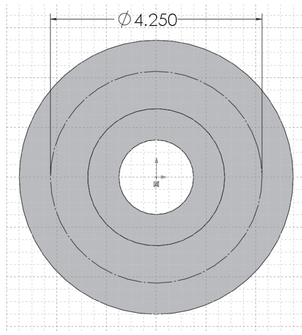


FIGURE 1.78

Select the Smart Dimension Tool. Add a 4.25-inch diameter dimension to the circle, as shown in Figure 1.78. Select the Circle Tool. Move the cursor to the top quadrant point on the construction circle, as shown in Figure 1.79. Note the diamond that appears, along with the coincident and vertical relation icons. Drag out a circle, as shown in Figure 1.80. Select the Smart Dimension Tool and add a diameter dimension of 0.50 inches, as shown in Figure 1.81.

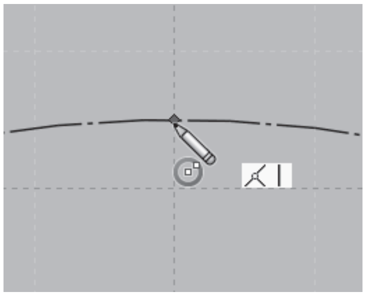


FIGURE 1.79

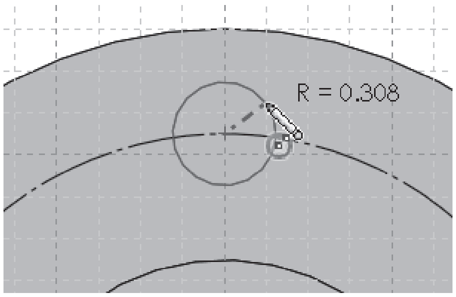


FIGURE 1.80

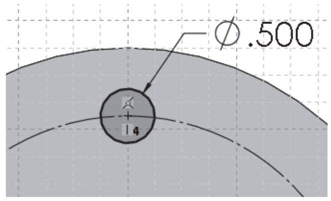


FIGURE 1.81

The sketch is fully defined, since the center of the circle just drawn has been located at the top quadrant point of the bolt circle.

Select the Extruded Cut Tool from the Features group of the CommandManager, and extrude a hole with a type of Through All. Click the check mark.

The first bolt hole is now in place, as shown in the trimetric view in Figure 1.82. Note that when we selected the Extruded Cut tool, only the small circle was used as the geometry of the cut. The bolt circle, because it was identified as construction geometry, was not included as a sketch entity to be extruded.

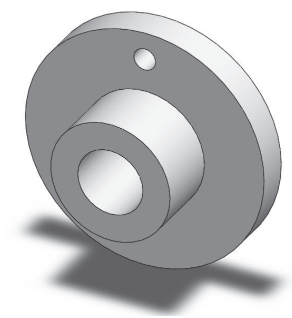


FIGURE 1.82

Notice that in the FeatureManager, all of our procedures are being recorded. The names of the features are not particularly descriptive; the four features that we have created so far were all created by extrusions, and so are named "Boss-Extrude1," "Cut-Extrude1," etc. To more easily identify features for later modifications, we can rename features.

Click once on "Cut-Extrude2" in the FeatureManager to select and highlight the name. Click again to allow editing of the name. (Use two separate mouse clicks, not a double-click.) Type "Bolt Hole" to rename the feature, as shown in Figure 1.83. Press the Enter key to accept the new name.

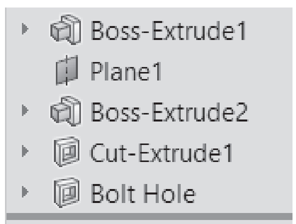
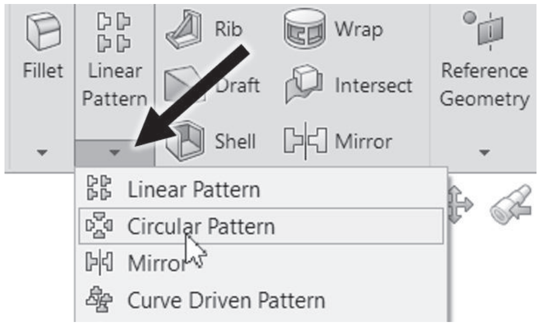


FIGURE 1.83

We could create the other three holes separately, but it is easier to copy the single hole into a circular pattern. Also, since our design intent is for the holes to exist in a circular pattern, it makes sense to construct them that way. If we later change the diameter of the holes, the diameter of the bolt circle, or the number of holes, it will be easy to do if we have created them in a pattern.

Make sure that the first bolt hole is selected. Click the Features tab of the CommandManager, and click the arrow under the Linear Pattern Tool to reveal a menu of pattern tools, as shown in Figure 1.84. Choose the Circular Pattern Tool.

FIGURE 1.84



In the PropertyManager, click in the top box (Pattern Axis) to activate it, as shown in Figure 1.85. To define the axis of the pattern, select a cylindrical face or a circular edge (other than a bolt hole), as shown in Figure 1.86. In the PropertyManager, check the "Equal spacing" option which will cause the angle to be changed to 360 degrees. Change the number of holes to 4, as shown in Figure 1.87. Click the check mark to complete the pattern, which is shown in Figure 1.88.

FIGURE 1.85

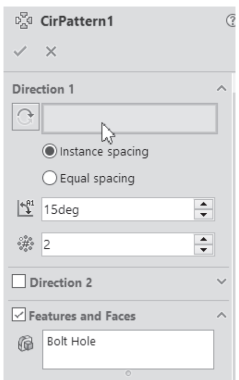


FIGURE 1.86

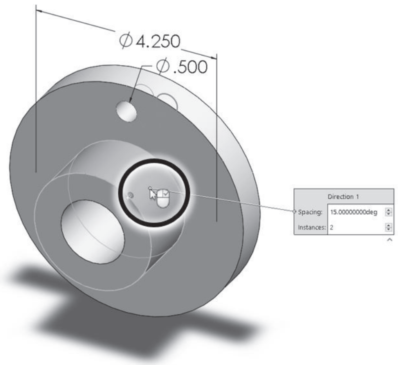
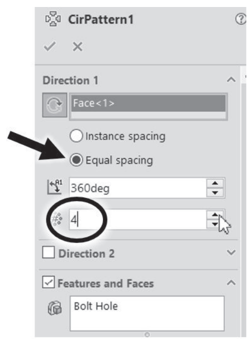
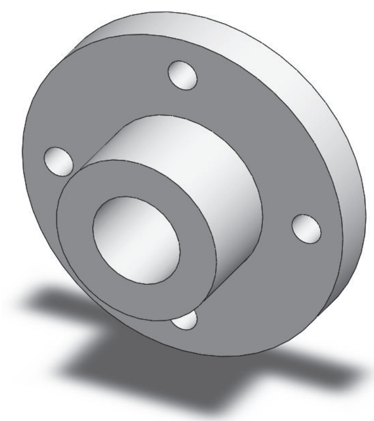


FIGURE 1.87



Now let's finish the flange by adding fillets to three of the sharp edges. A fillet is a feature that rounds off a sharp edge. Actually, a fillet is a rounded edge created by adding material, while a round is created by removing material. Fillets and rounds are created with the SOLIDWORKS software by the same command.

FIGURE 1.88



From the Features group of the CommandManager, select the Fillet Tool, as shown in Figure 1.89. Select the three edges indicated in Figure 1.90 to be filleted. (Be sure to see the line next to the cursor, as shown in Figure 1.91, to indicate that an edge and not a face is being selected. If a face is selected, then all of the edges of that face will be filleted.) In the PropertyManager, enter the radius as 0.25 inches, as shown in Figure 1.92. Check the Full preview option to see the fillets that will be created. Click the check mark to add the fillets, which are shown in Figure 1.93.

FIGURE 1.89

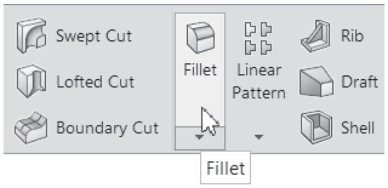


FIGURE 1.90

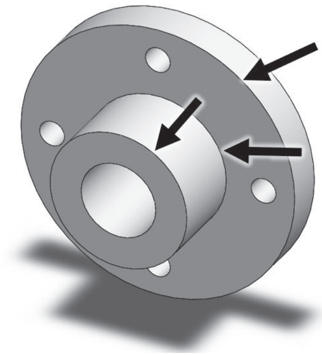


FIGURE 1.91

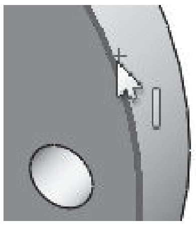


FIGURE 1.92

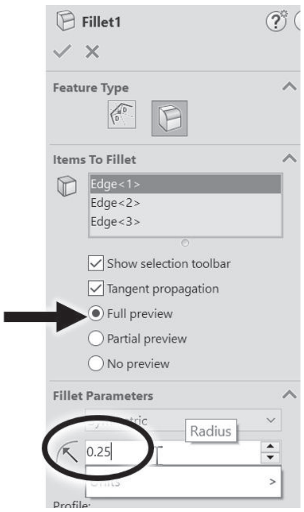
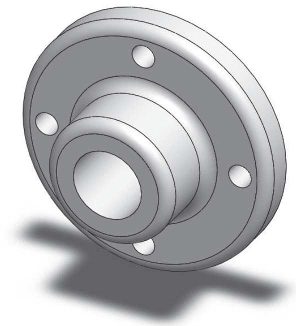
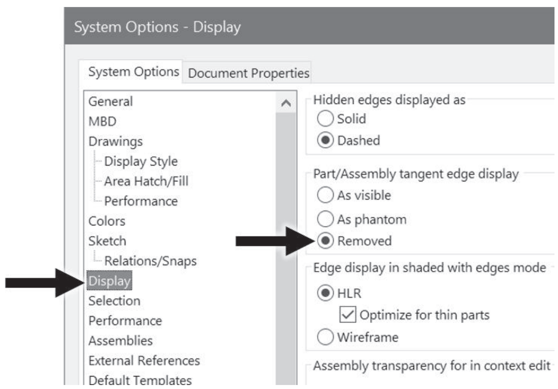


FIGURE 1.93



Notice that the intersections of the fillets and the cylindrical and flat surfaces are shown. These interfaces are called tangent edges. Display of tangent edges is often undesirable. Their display can be controlled from the Options menu.

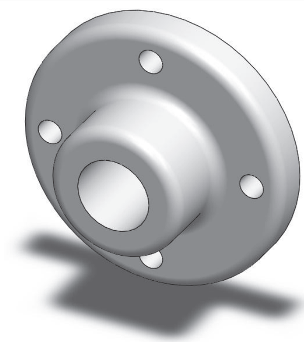
FIGURE 1.94



Select the Options Tool. Under the System Options tab, under Display, choose Removed as the Part/Assembly tangent edge display option, as shown in Figure 1.94. Click OK.

The part should appear as in **Figure 1.95**.

FIGURE 1.95



DESIGN INTENT Selecting a Modeling Technique

The three fillets are added in this tutorial in a single step by selecting the three edges to be filleted within a single fillet command. With this method, only the first fillet is dimensioned. Another way to add the fillets is to close the Fillet Tool after each fillet is created, so that the fillets are created in three separate steps. The preferred method depends on how you wish to edit the fillet radii. If you want all of the fillets to always have the same radius, then the first method allows one value to be changed for all three fillets. If you prefer to edit the fillets separately, then the second method provides an editable dimension for each fillet.

Now we can add the chamfer to the center hole. A chamfer is a conical feature formed by removing material from an edge.

Select the arrow under the Fillet Tool in the CommandManager, and select the Chamfer Tool, as shown in Figure 1.96. Click on the edge shown in Figure 1.97 to select it as the edge to be chamfered. In the PropertyManager, set the chamfer parameters to 0.080 inches and 45 degrees, as shown in Figure 1.98, and click the check mark to finish.

The finished part is shown in **Figure 1.99**.

From the main menu, select File: Save. Leave the part file open for the next section, in which we will learn how to make modifications to the part.

FIGURE 1.96

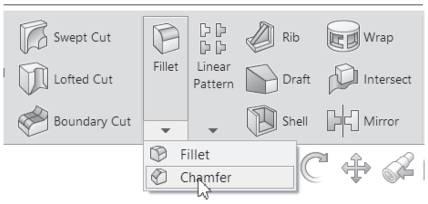


FIGURE 1.97

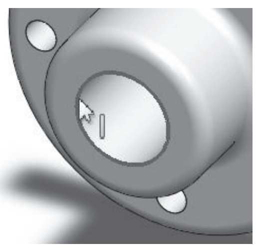


FIGURE 1.98

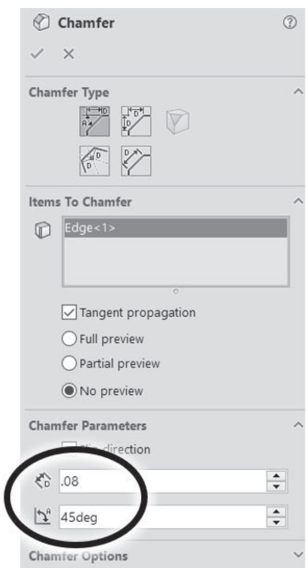
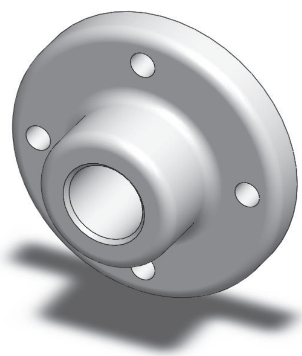


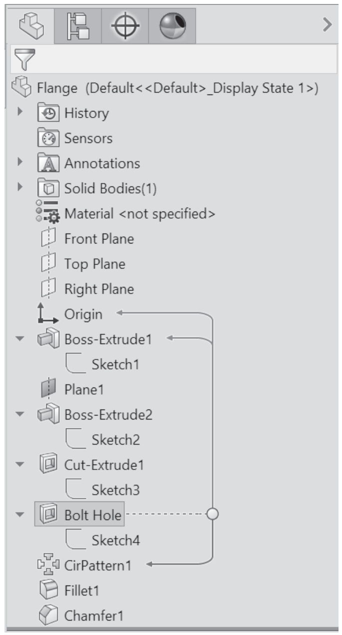
FIGURE 1.99



1.3 Modifying the Flange

One of the main advantages of solid modeling is the ability to make changes easily. As we have observed, the FeatureManager has recorded all of the operations required to make the flange, as shown in **Figure 1.100**. If we click on the arrow next to each feature, we see that the sketch associated with each feature is stored as well. Relationships between features can be displayed by selecting View: User Interface from the Main Menu and turning on the two Visualization options shown in **Figure 1.101**. For example, the first bolt hole is the basis for the hole pattern. In **Figure 1.100**, this relationship is displayed with an arrow from the "parent" feature (Bolt Hole) to the "child" feature (CirPattern1). Similarly, since the sketch defining the position of the bolt hole is on a surface of the initial solid disk (Boss-Extrude1) and is dimensioned relative to the origin, the hole is shown as a "child" of Boss-Extrude1 and the origin.

FIGURE 1.100

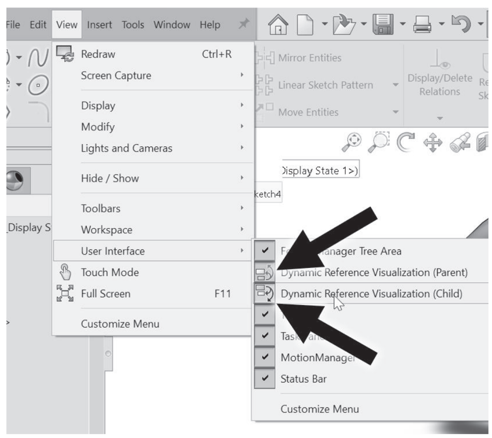


Let's change the first item that we created by increasing the diameter of the base from 5.5 to 7 inches.

Right-click Sketch1 in the FeatureManager, and select Edit Sketch.

Note that if Edit Sketch does not appear in the menu, then an icon for editing the sketch appears in the Context toolbar at the top of the menu. Earlier in the chapter, we selected the Customize tool and cleared the check box labeled "Show in shortcut menus." This causes commands such as Edit Sketch, Edit Feature, Hide, etc., to appear as entries in the menu rather than as icons at the top of the menu. If you missed this step earlier, then it is recommended that you clear the check box now. (See Appendix B for more information about customizing the SOLIDWORKS interface.)

FIGURE 1.101



Double-click the 5.5-inch dimension, and change it to 7.0 inches, as shown in Figure 1.102. When you close the sketch by clicking on the Exit Sketch Tool in the Sketch group of the CommandManager, the part will be updated to the new dimension, as shown in Figure 1.103.

An even easier way to edit the sketch dimensions or the extrude depth is illustrated next.

FIGURE 1.102

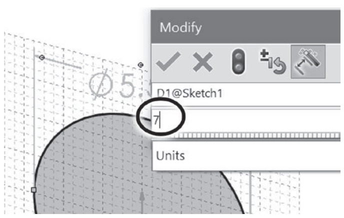


FIGURE 1.103

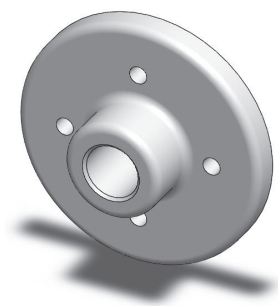
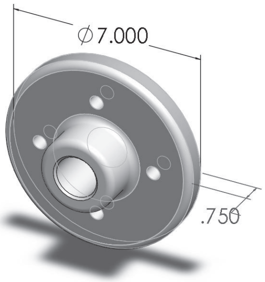
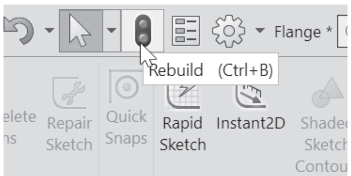


FIGURE 1.104



Double-click the icon next to Boss-Extrude1 in the FeatureManager. All of the dimensions used to create the feature are displayed, as shown in Figure 1.104. The sketch dimensions are shown in black, while the feature dimensions (in this case the extrude depth) are shown in blue. (Note that the dimensions in the figure are oriented so that they are aligned to be parallel with the bottom of the screen rather than with the dimension lines. To show the dimensions in this manner, select Options: System Options: Display and check the box labeled "Display dimensions flat to screen.") Double-click the diameter dimension and change it back to 5.5 inches. Click the Rebuild Tool, as shown in Figure 1.105.

FIGURE 1.105



To add draft to the boss, select Boss-Extrude2 from the FeatureManager, right-click and select Edit Feature, as shown in Figure 1.106. In the PropertyManager, turn the draft on (see Figure 1.107) and set the angle to 3 degrees. Check the "Draft outward" box so that the boss increases in size as it is extruded. Click the check mark to finish.

FIGURE 1.106

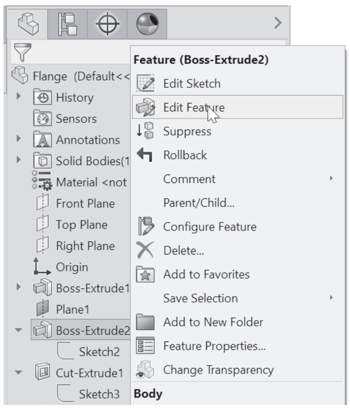
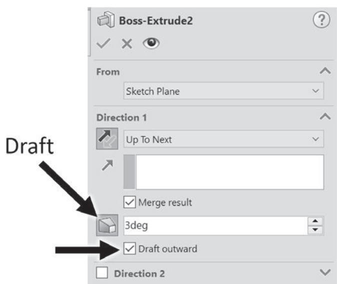


FIGURE 1.107



The draft will be easier to see from a top or side view. You can show the Front, Top, and Right Views along with the current (Trimetric) view with the Four-View option.

Select the Four-View window from the View Orientation Tool of Heads-Up View Toolbar, as shown in Figure 1.108.

The drafted feature can be seen clearly in the Top and Right Views, as shown in Figure 1.109. Note that Figure 1.109 shows the Top View above the Front View, and the Right View to the right of the Front View. Views oriented in this manner are referred to as third-angle projections. If the views on your screen are oriented with the Top View below the Front

FIGURE 1.108

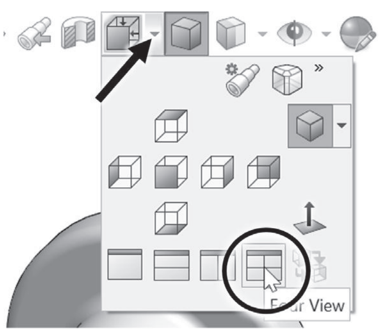
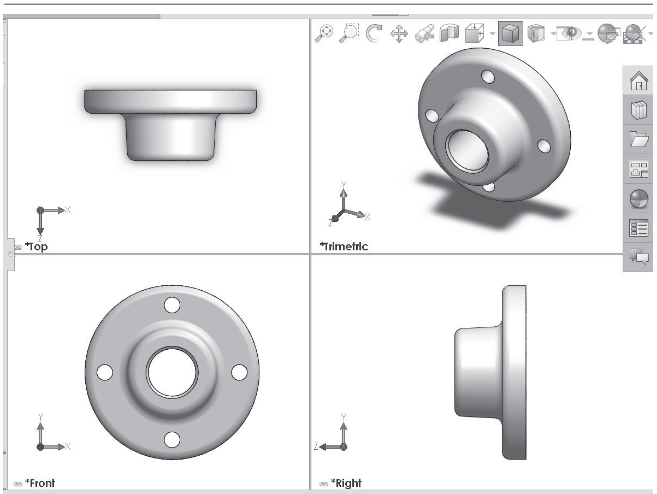


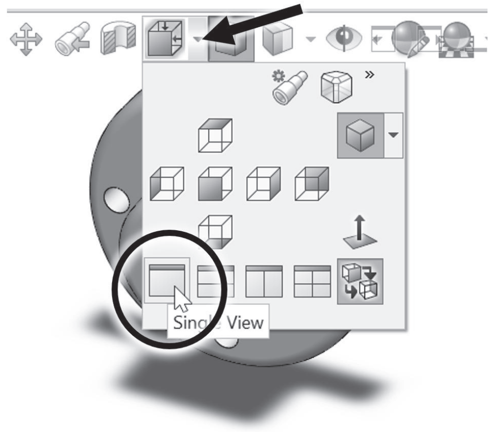
FIGURE 1.109



View, then you are seeing first-angle projections, which are typical of European drawings. To switch from first-angle to third-angle projections, select Options: System Options: Display and choose Third Angle as the option for the Four-View viewport. After doing so, you will need to select the Four-View window again to refresh the views.

To revert to a single view, click in the window displaying the Trimetric View, and select Single View from the View Orientation Tool of the Heads-Up View Toolbar, as shown in Figure 1.110.

FIGURE 1.110



Finally, right-click on CirPattern1 in the FeatureManager and select Edit Feature. Change the number of holes from four to six, as shown in Figure 1.111.

The modified part is shown in **Figure 1.112**.

FIGURE 1.111

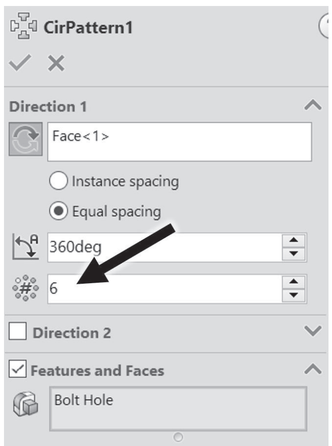
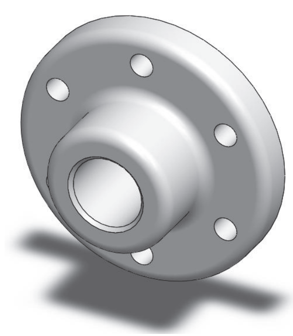


FIGURE 1.112



These last two changes illustrate the importance of considering design intent when modeling. If the first boss had been extruded from the base feature, then adding draft would have required us to change the diameter of the boss, calculating the diameter that will result in a 2.75-inch diameter at the top of the boss when draft is included. By sketching in a plane at the top of the boss, the critical 2.75-inch dimension can be maintained easily. Also, by constructing the holes as a circular pattern instead of individually, the number of holes could be modified easily.

Click the Undo Tool to reverse the previous command.

We can change the appearance of a part or of individual features and faces of a part with the Edit Appearance Tool.

FIGURE 1.113

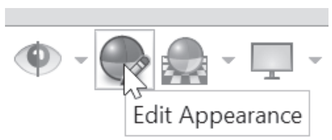
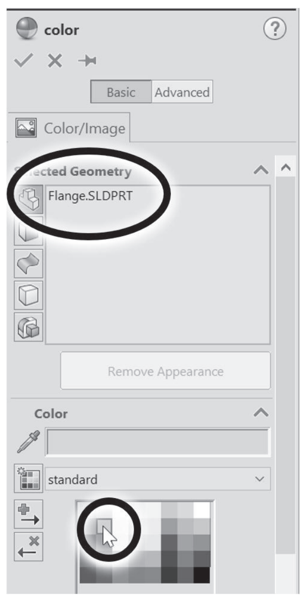
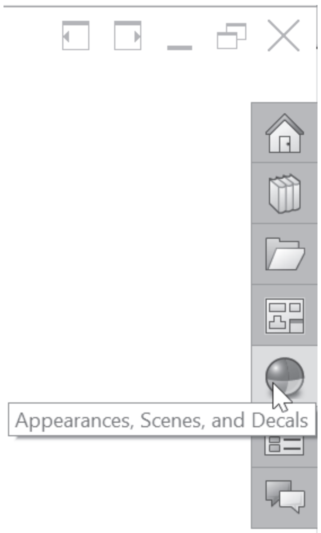


FIGURE 1.114



Press the Esc key to cancel any selections that may be active. Select the Edit Appearance Tool from the Heads-Up View Toolbar, as shown in Figure 1.113. In the PropertyManager, note that since no specific entities have been selected, the entire part will take on the selected appearance. Select a color from the color palette, as shown in Figure 1.114, and click the check mark.

FIGURE 1.115



The entire flange will now be shown in the selected color. Note that many other appearance options can be selected by clicking on the Advanced button shown in Figure 1.114. These include modifying the reflectivity or the transparency of a component or applying a surface texture.

When applying textures, the Task Pane is useful in that previews of the available textures can be viewed.

Click on the Appearances, Scenes, and Decals tab of the Task Pane, as shown in Figure 1.115. Under Appearances: Painted: Powder Coat, select dark powdercoat, as shown in Figure 1.116. Click and drag the appearance onto the part. In the menu that appears, you can choose to apply the appearance to a given surface, a feature, a body, or the entire part. Click on the Part icon, as shown in Figure 1.117, to apply the appearance to the entire part.

FIGURE 1.116

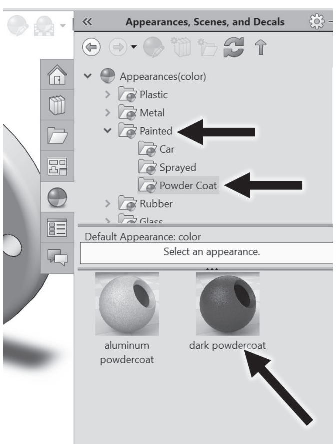
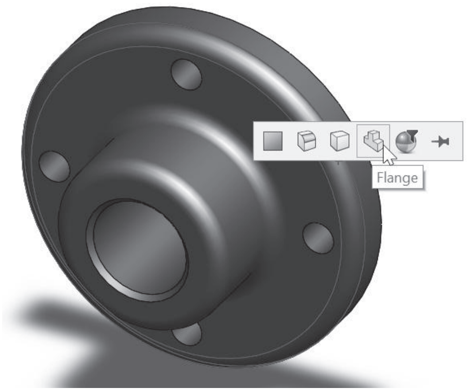


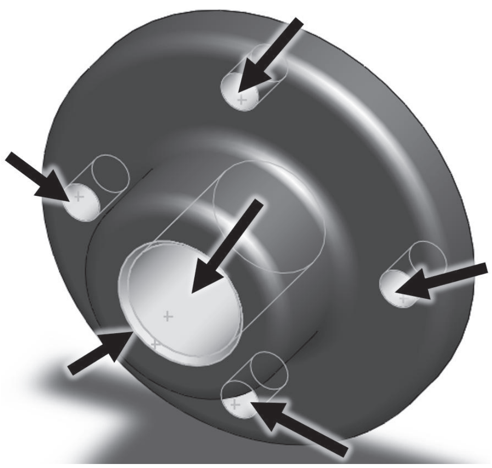
FIGURE 1.117



We may want to show certain faces differently than the rest of the part. We will show the surfaces of the holes and the chamfer as machined steel.

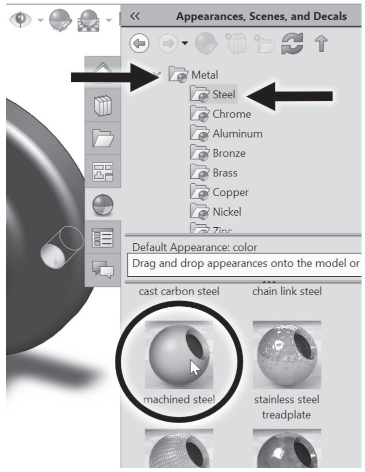
Select the surfaces of the four bolt holes, the center hole, and the chamfer, as shown in Figure 1.118. To select multiple entities, hold down the Ctrl key as you make your selections. Click on the Appearances, Scenes, and Decals tab of the Task Pane. Select Appearances: Metal: Steel, and double-click on Machined Steel, as

FIGURE 1.118



shown in Figure 1.119. The selected surfaces are now shown with the selected texture, as shown in Figure 1.120.

FIGURE 1.119



The colors applied to a model can be viewed and/or edited from the DisplayManager. The DisplayManager can be viewed by clicking on its icon above the FeatureManager. As shown in **Figure 1.121**, there are several icons that can be used to display tools and options in the space normally occupied by the FeatureManager. These include the PropertyManager, which as we have seen is automatically displayed when one or more model entities are selected, the ConfigurationManager, which is used to select a specific configuration of a model (as will be discussed in Chapter 3), the DimXpertManager, which is used to apply dimensions and geometric tolerances to a model (the DimXpertManager is not discussed in this text), and the DisplayManager. When the DisplayManager is selected, three options are available: Appearances, Decals, and Scene, Lights, Cameras. A fourth option, PhotoView 360, is grayed out unless the PhotoView 360 add-in is activated. This add-in program allows photo-realistic renderings of models to be made. It is not discussed in this text, but a tutorial is available at the book's website: www.mhhe.com/howard2020.

Select the DisplayManager and click on the Appearances icon. Change the Sort Order to Hierarchy, and expand the items as shown in Figure 1.122.

FIGURE 1.120

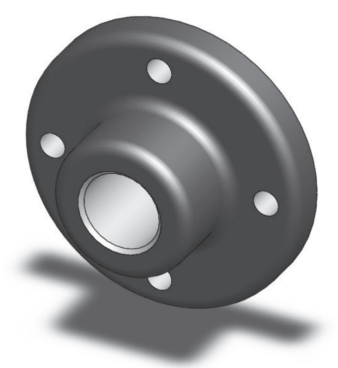


FIGURE 1.121

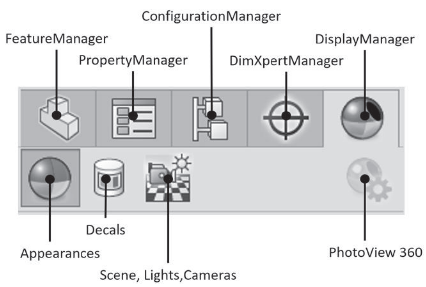
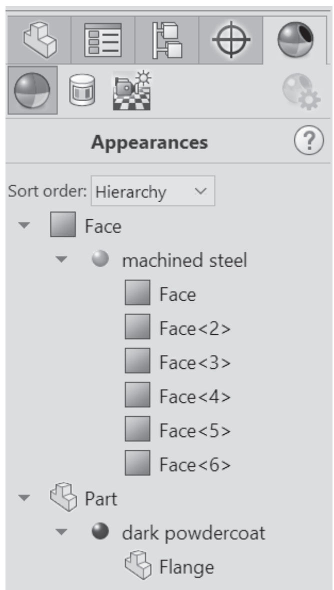


FIGURE 1.122



Note that the faces are shown first in the hierarchal order, even though the colors were applied to the faces after the color was applied to the entire model. In the hierarchy of appearances, appearances applied to faces take priority over those applied to features or the entire model, and appearances applied to features take priority over those applied to the entire model. In the DisplayManager, the appearances can be edited and/or deleted by right-clicking on the corresponding entry (machined steel or dark powdercoat in this example) and choosing the desired action from the menu.

Close the part window by clicking on the X in the upper-right corner of the part window. Do not save the changes to the file.

1.4 Using Dimensions and Sketch Relations

In the previous tutorial, we used a combination of dimensions and sketch relations to create fully defined sketches for our model features. While it is not absolutely necessary to use fully defined sketches, it is good design practice. After all, an engineering design of a component must include sufficient detail for the component to be analyzed and eventually built. Using fully defined sketches helps to ensure the complete definition of the geometry of the component.

The Smart Dimension Tool is used to add numerical dimensions to a sketch. As we saw in the previous tutorial, the tool is "smart" in that the type of dimension does not need to be specified. When we clicked on a circle, a diameter dimension was created. If we click on a line, then a linear dimension is created as shown in **Figure 1.123**. Recall that two mouse clicks are required—one to identify the entity to be dimensioned, and the second at the location where the dimension is to be placed. Note that if the cursor is dragged away from the line in a direction roughly perpendicular to the line, then the resulting dimension defines the length of the line. However, if the cursor is dragged away horizontally, then a dimension defining the vertical distance between the endpoints is created, as shown in **Figure 1.124**. Similarly, dragging the cursor vertically results in a dimension defining the horizontal distance between endpoints is created, as shown in **Figure 1.125**. Note the "lock" icon beside the cursor before the dimension is placed. If the dimension is in the desired alignment, then clicking the right mouse button causes this alignment to be maintained until the dimension is placed. This is not usually necessary. As long as the dimension is placed in its proper orientation, it can be moved to a more desirable location by simply clicking and dragging on the numerical value.

FIGURE 1.123

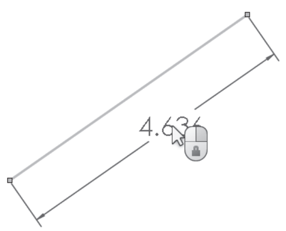


FIGURE 1.124

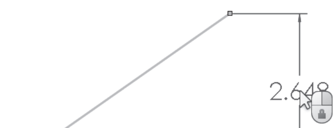
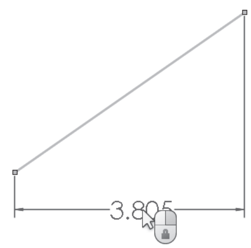


FIGURE 1.125



Circles, lines, and arcs can all be dimensioned by clicking once on the entity and then clicking away from the entity to place the dimension. (Arcs are automatically dimensioned with a radius rather than a diameter.) These are examples of dimensions applied to single entities. The Smart Dimension Tool also allows for dimensions relating two entities to one another to be created. For example, consider the two parallel lines shown in **Figure 1.126**. With the Smart Dimension Tool selected, the first mouse click selects one of the lines. If the second mouse click is in the graphics area away from any other entity, then a linear dimension for the length of the line is created, as discussed above. However, if the second mouse click is on another entity, then a dimension is created between the two entities, and a third mouse click is required to place the dimension. In this example, the dimension created is the distance between the two lines, as shown in **Figure 1.126**. If the two lines are not parallel, then the same mouse clicks create an angular dimension, as shown in **Figure 1.127**.

FIGURE 1.126

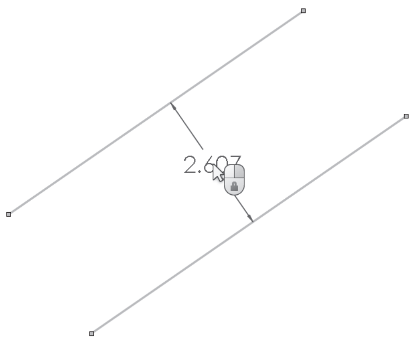
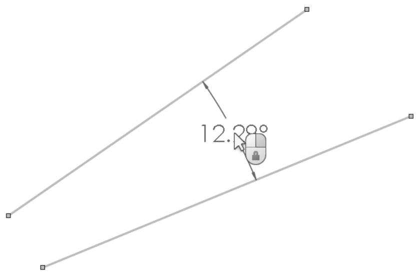


FIGURE 1.127



When a circle is one of the two entities selected, then the resulting dimension is always to the center of the circle, as shown in **Figure 1.128**. It is not necessary to select the center point of the circle; clicking on the perimeter of the circle and the line creates the dimension to the circle's center.

When a centerline is one of the entities selected, then the resulting dimension can define either the distance from the centerline to the second entity or the distance from the second entity to a mirror image of itself on the other side of the centerline. For example, consider the circle and centerline shown in **Figure 1.129**. Clicking on the centerline and circle creates a linear dimension. If the next mouse click is made between the two entities, then the dimension as shown in **Figure 1.129** is created. However, if the cursor is dragged to the other side of the centerline before clicking to place it, then the dimension as shown in **Figure 1.130** is created. This method of dimensioning is especially useful when working with revolved geometry, such as the pulley of the next section, in that it allows for the diameters of revolved features to be specified rather than their radii. Since diameters of physical parts can be directly measured, defining a component using diameters is good design practice.

FIGURE 1.128

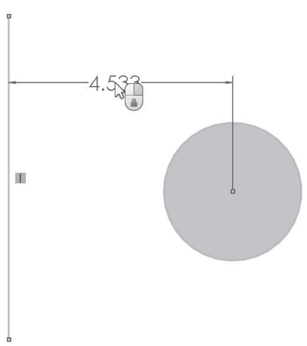


FIGURE 1.129

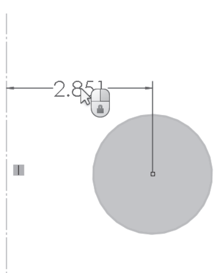
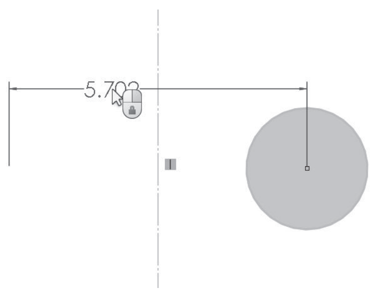


FIGURE 1.130



A fully defined sketch is usually not possible without sketch relations. In the case of the circles used in the flange, the location of the center points had to be specified in order for the sketches to be fully defined. In each sketch, the relation defining the center of the circle and the origin as being *coincident* was added automatically through a snap—the cursor was moved close to the origin before the first mouse click and the center of the circle "snapped" to the origin. In the case of the sketch defining the bolt hole, the snap was made to a quadrant point of the bolt circle. These are examples of *automatic relations*. By default, SOLIDWORKS creates these automatic relations. This feature can be turned off by selecting Tools: Sketch Settings: Automatic Relations from the Main Menu, but most users will not find a reason to do so. In addition, automatic relations are created when specifying an entity's geometry. For example, when drawing a line, a small icon beside the line indicates that the line will be horizontal or vertical, as shown in **Figure 1.131**. When the line is completed, it will have a horizontal or vertical relation associated with it, as indicated by the icon shown in **Figure 1.132**.

FIGURE 1.131

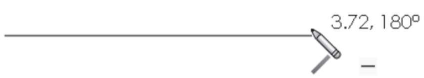
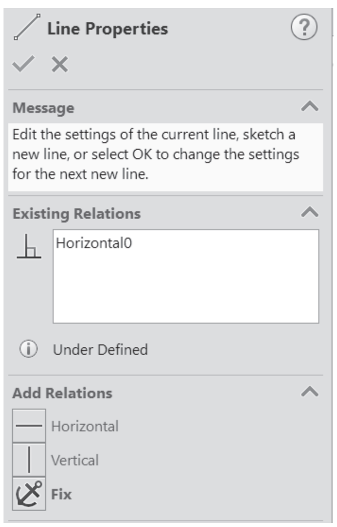


FIGURE 1.132

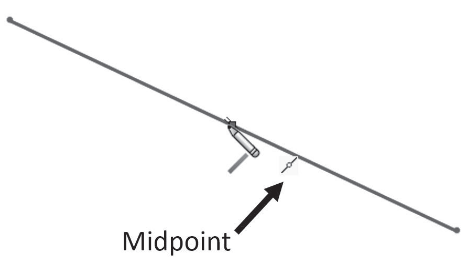
FIGURE 1.133



When an entity is selected, its associated relations are shown in the PropertyManager, as shown in **Figure 1.133**. In the PropertyManager, relations can be deleted by selecting them and pressing the Delete key or added by clicking the appropriate icon. Of course, relations must be compatible with each other and with any dimensions existing in the sketch. For example, clicking the Vertical icon in this case would result in an error, since a line cannot be both horizontal and vertical.

Horizontal and vertical relations, along with Fix, which simply fixes the location of an entity within the sketch, are relations that are applied to single entities. Most relations apply to multiple entities. For example, **Figure 1.134** illustrates the addition of a new line to an existing line. With the Line Tool selected, moving close to the midpoint of the first line causes the second line's first point to snap to the midpoint. As the line is dragged out, there are dashed guidelines parallel and perpendicular

FIGURE 1.134



to the first line displayed on the screen, as shown in **Figure 1.135**. If the second mouse click is made close to the perpendicular guideline, then a perpendicular relation between the two lines is created. As shown in **Figure 1.136**, the sketch relation icons indicate the midpoint relation between the first line and the endpoint of the second line, and the perpendicular relation between the two lines.

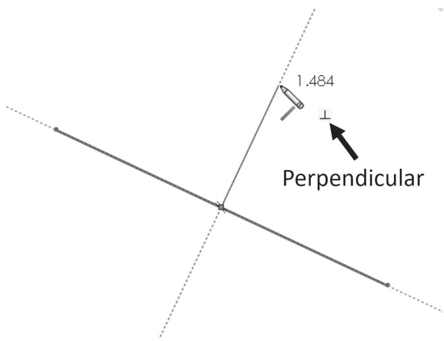


FIGURE 1.135

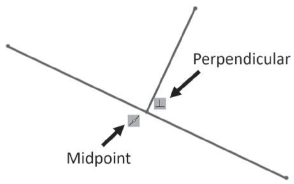


FIGURE 1.136

Relations can also be added manually. For example, consider the two lines in **Figure 1.137**. Clicking on the first line selects it and shows its properties in the PropertyManager. A vertical relation can be added by clicking the Vertical icon in the PropertyManager or in the context toolbar that pops up when the line is selected. If we want to merge endpoints of the two lines, then we click on the first endpoint to select it. Then, while holding down the Ctrl key, we select the other endpoint, as shown in **Figure 1.138**. As in most Windows programs, the Ctrl key allows multiple entities to be selected. The Merge relation can then be applied by clicking the Merge icon in the context toolbar or in the PropertyManager, as shown in **Figure 1.139**. (We could also merge these points by dragging the endpoint of the second line until it snaps to the endpoint of the first line, creating an automatic relation.)

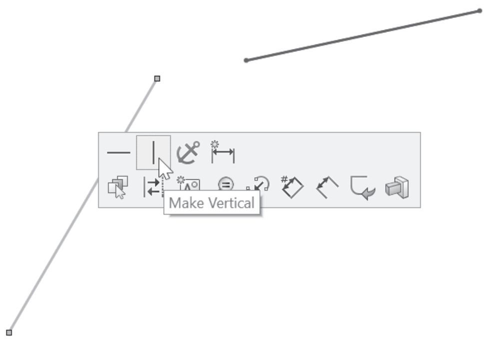


FIGURE 1.137

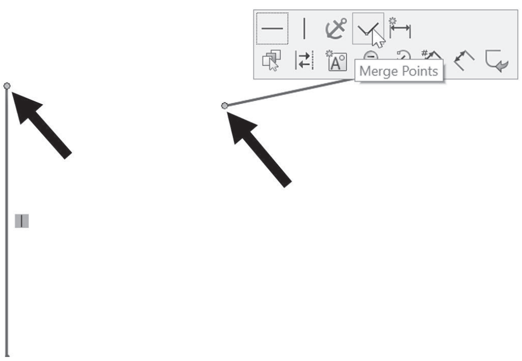


FIGURE 1.138

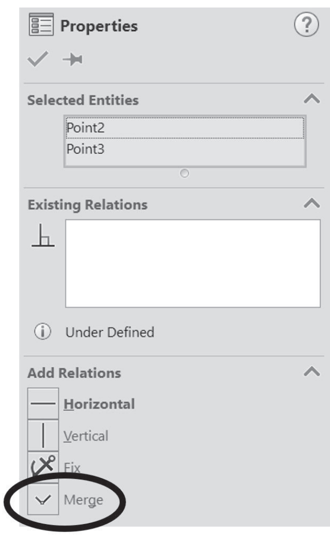


FIGURE 1.139

FIGURE 1.140

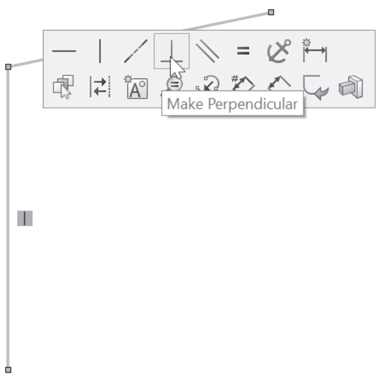
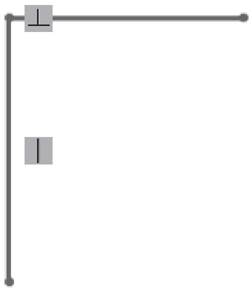


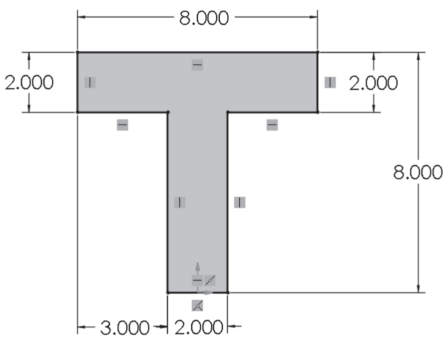
FIGURE 1.141



Now both lines can be selected, and a perpendicular relation added, as shown in **Figure 1.140**. The result is shown in **Figure 1.141**.

It should be noted that the display of the sketch relation icons can be toggled on and off by selecting View: Hide/Show: Sketch Relations from the Main Menu. There may be occasions where a sketch becomes so cluttered that turning off the display of the relation icons temporarily is desired, but in most cases displaying the icons is helpful when creating and editing sketches.

FIGURE 1.142



As we have noted, both dimensions and relations are used to fully define sketch geometry. As a general rule, we try to use as few dimensions as possible and rely on relations to complete the geometry definition. For example, consider the T-beam section of **Figure 1.142**. The section consists of horizontal and vertical lines, and the bottom line's midpoint is fixed to the origin. In order to fully define the sketch, six dimensions are required. However, consider the design intent of the part to be made from this sketch. We probably desire the two "legs" at the top of the section to be the same thickness and width. Therefore, we can delete two dimensions and replace them with the relations shown in **Figure 1.143**. The advantage of this approach is that if we make a change, say to the thickness of the legs, then we have only one dimension to change, and our design intent of equal thicknesses is maintained. Another solution is shown in **Figure 1.144**, where a vertical centerline has been added from the origin. By selecting the two sides indicated and the centerline, a symmetric relation can be added. This relation, along with a collinear relation between the bottom edges of the legs, makes the sketch fully defined. Either of these solutions causes the pre-defined Right Plane to become a *plane of symmetry* of the resulting part (assuming that the sketch is in the Front Plane). The use of symmetry is good design practice, and will be emphasized in the pulley tutorial in the next section and in the tutorials of Chapter 3.

FIGURE 1.143

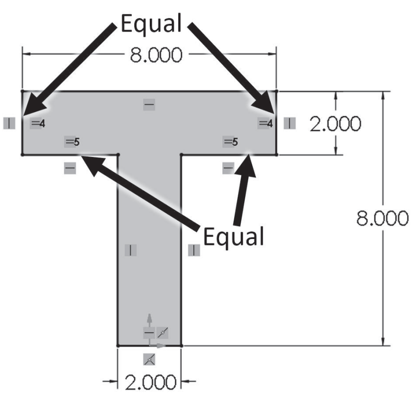
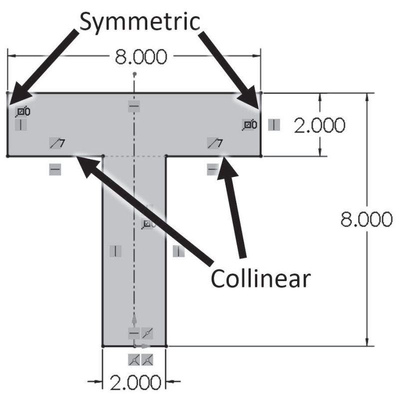
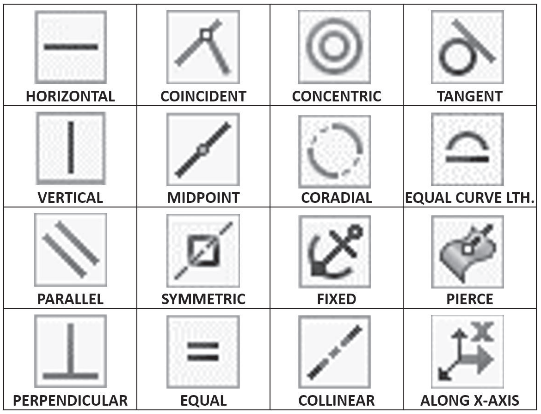


FIGURE 1.144



A list of the relation icons is shown in **Figure 1.145**. All are common in 2-D sketches except for the last two: the Pierce relation is used in multiple-sketch applications such as sweeps and lofts, and the Along X-Axis relation is used in 3-D sketches (there are similar Along Y-Axis and Along Z-Axis relations).

FIGURE 1.145



1.5 A Part Created with Revolved Geometry

The flange created earlier utilized extruded features. In this exercise, we will use *revolved* features to create the pulley shown in **Figure 1.146**. We will sketch features of the cross-section of the pulley, and then revolve those features around a centerline to create solids and cuts. The final feature will be a keyway, which will be made with an extruded cut. Dimensions of the pulley are detailed in the 2-D drawing of **Figure 1.147**.

FIGURE 1.146

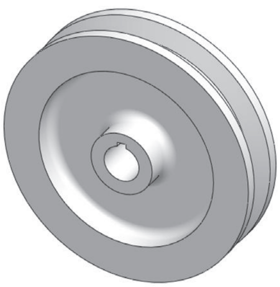
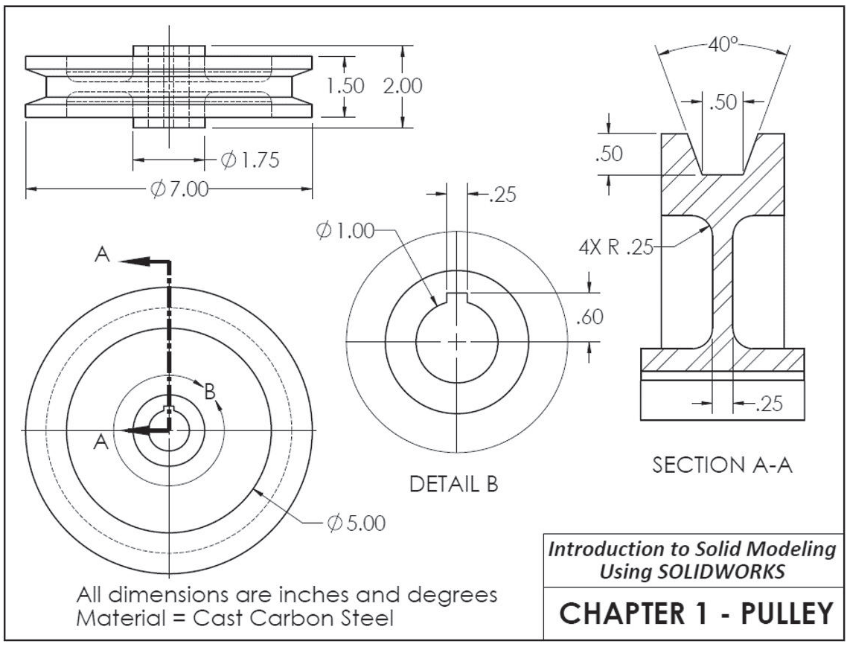


FIGURE 1.147



Open a new part.

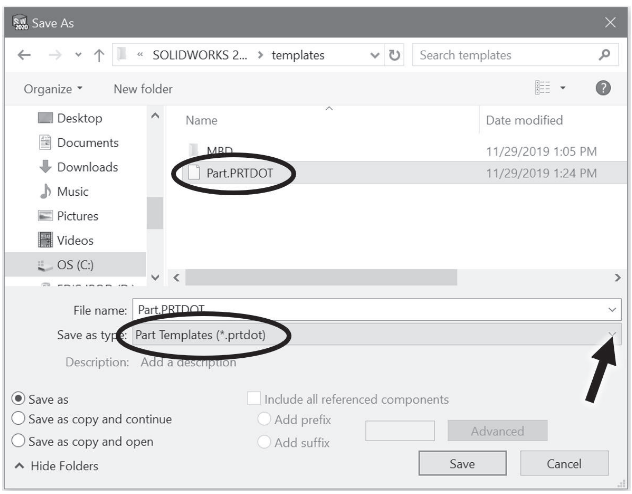
You will notice that some of the changes made earlier to the interface, such as the number of tabs on the CommandManager and the tools on the Heads-Up Toolbar, are still in effect. Others, such as the background color, have reverted to the initial settings and need to be changed again. These document-specific settings are stored in the Part *template*. We will edit the template so that we do not have to make these changes every time we begin a part.

Change the background color to Plain White from the Apply Scene Tool of the Heads-Up Toolbar. If desired, choose a part color other than the default gray by selecting the Edit Appearance Tool and selecting a new color from the choices in the PropertyManager. Remember that light colors are preferred. Also, turn off the shadows from the View Settings Tool if desired.

Select the Options Tool, and click the Document Properties tab. Change the drafting standard to ANSI. Under Dimensions, set the Primary precision to .123 (three decimal places). Select Grid/Snap, and check the box labeled "Display Grid." Select Units, and change the unit system to IPS, the number of decimals for length dimensions to .123, and the number of decimals for angles to None.

Note that we have set the number of decimal places in two separate locations. The number of places can be changed in either location; by setting both to .123 we ensure that the template setting will be stored correctly.

FIGURE 1.148



Any of the other settings under the Document Properties tab can be stored in the template. For example, you may want to change the font used for dimensions or turn the display of the grid off (most of the figures in this book are made with a larger font and with the grid off for clarity).

From the main menu, select File: Save As. Change the file type to Part Templates (*.prtdot). Click on the file "Part.prtdot" to select it, as shown in Figure 1.148, and click Save. Click Yes when asked if you want to replace the existing template. Close this part document and then open a new part.

Note that the settings that you just saved in the template are effective for the new part. Also, note that when you changed the file type to Part Template, the working directory automatically changed to default directory for SOLIDWORKS templates. This directory is located in the Program Files or Program Data directory on the drive containing your operating system. Therefore, it is not a good location to store your work. The next time that you save a document, the directory will default to the last one accessed; in this case the directory where the templates are saved. Make sure to change the directory to the one you want before saving any documents. We will note this when we save the pulley file later in this section.

Click on the Right Plane to select it, as shown in Figure 1.149. From the Sketch group of the CommandManager, click the arrow beside the Line Tool and select the Centerline Tool, as shown in Figure 1.150. Draw a vertical centerline up from the origin, as shown in Figure 1.151.

This centerline will allow us to take advantage of the symmetry of the cross-section.

Click the arrow beside the Rectangle Tool, and select the Center Rectangle Tool, as shown in Figure 1.152. Click above the origin on the centerline, as shown in Figure 1.153, to set the center of the rectangle. Then drag out a corner of the rectangle, as shown in Figure 1.154. The size is not important, but keep the entire rectangle above the origin. Repeat to create a second rectangle above the first, as shown in Figure 1.155.

FIGURE 1.149

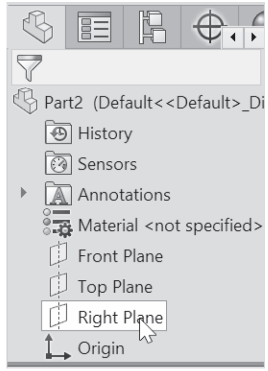


FIGURE 1.150

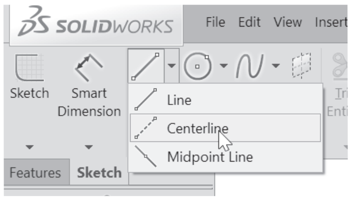


FIGURE 1.151

FIGURE 1.152

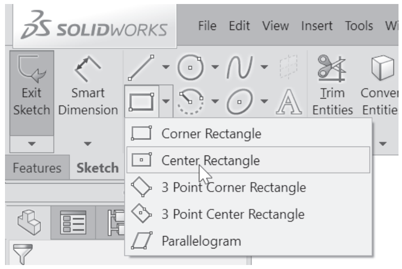


FIGURE 1.153

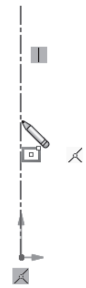


FIGURE 1.154

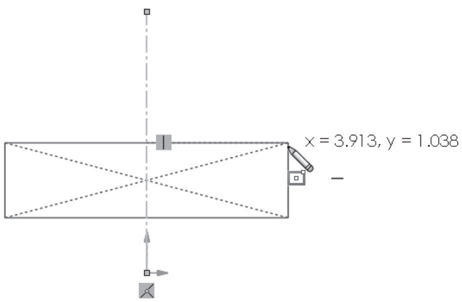


FIGURE 1.155

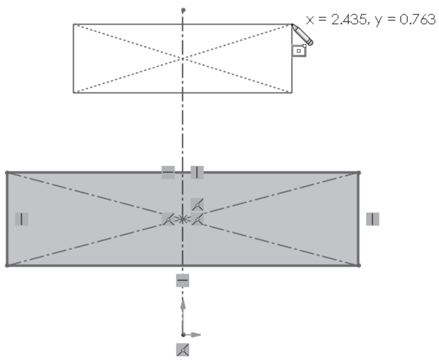


FIGURE 1.156

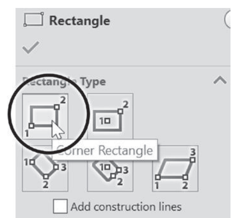
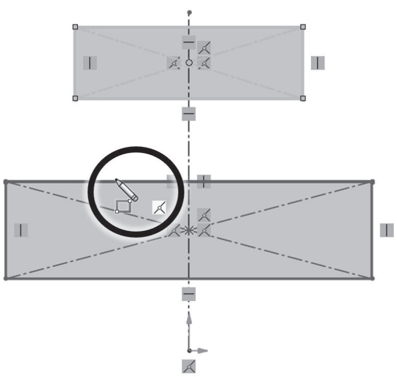
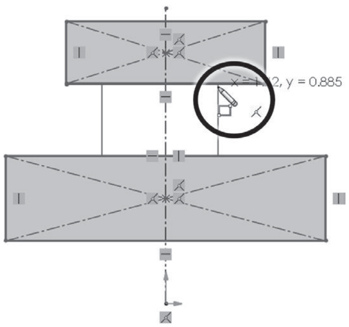


FIGURE 1.157



In the PropertyManager, change the type of rectangle from Center to Corner, as shown in Figure 1.156. Move the cursor to the top line of the bottom rectangle (but not to a corner or midpoint; when the entire bottom line changes color, then you are snapping to the line and not to a specific point), as shown in Figure 1.157, and click to place one corner of the rectangle

FIGURE 1.158

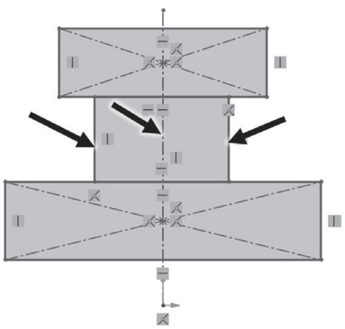


on this line. Drag the rectangle up until the opposite corner is along the bottom line of the top rectangle, as shown in Figure 1.158. Click to complete the rectangle, and press Esc.

When we used the Center Rectangle Tool and placed the center points along the vertical centerline, those rectangles became symmetric about the centerline. However, to join the first two rectangles together, we used a Corner Rectangle, which is not centered on the centerline. Therefore, we need to add the symmetry of this rectangle manually.

Click on the two lines and the centerline shown in Figure 1.159 to select them, remembering to hold down the Ctrl key when making multiple selections. Click the Make Symmetric icon from the context toolbar, as shown in Figure 1.160 (or the Symmetric icon in the PropertyManager).

FIGURE 1.159



Select the Smart Dimension Tool, and click once on the top line of the top rectangle to create a linear dimension. Drag the dimension to the desired location and click to place it. Enter the value as 1.5 inches, as shown in Figure 1.161. Add the 2.0-inch dimension shown in Figure 1.161 to the bottom line.

With the Smart Dimension Tool selected, click on one of the vertical lines of the middle rectangle, and then on the other vertical line, creating a linear dimension. Drag the dimension to the desired location, and then click to place it. Enter the value as 0.25 inches, as shown in Figure 1.162.

Note that it is not necessary to hold down the Ctrl key when selecting multiple entities for the Smart Dimension command. If you click on a single entity and then click away from any other entity to place it, the Smart Dimension Tool creates a dimension from that entity (length of a line, diameter of a circle, radius of an arc). If you click on an entity and then click on a second entity, the Smart Dimension Tool will attempt to create a dimension relating the two entities. When we selected two parallel lines, the distance between the two lines was added as a dimension.

FIGURE 1.160

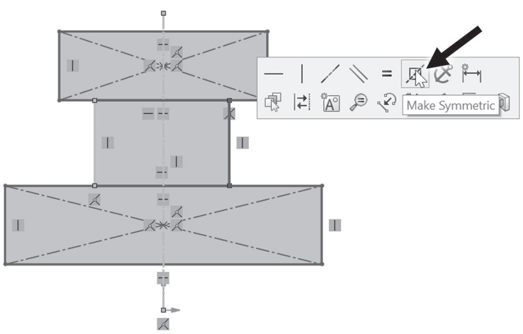


FIGURE 1.161

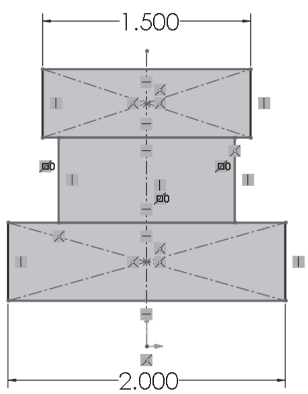
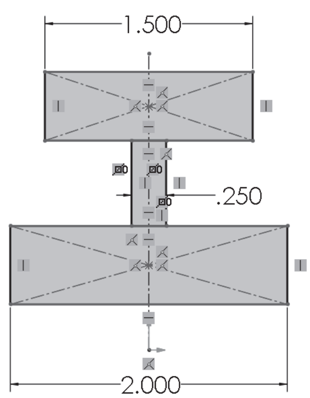


FIGURE 1.162

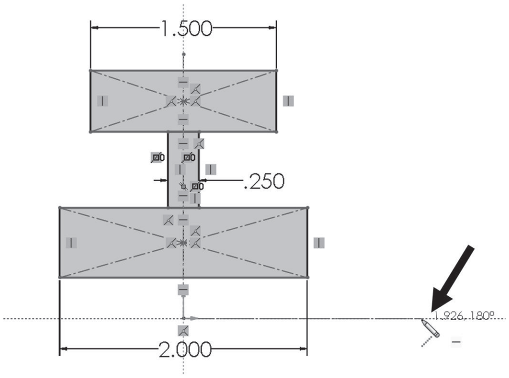


As noted in the previous section, if we select two non-parallel lines, an angular dimension will be created.

If you refer back to **Figure 1.147**, you will see that the other dimensions used to define this cross section are diameter dimensions. In order to place the diameter dimensions, we need to establish the centerline which will become the axis of revolution for the resulting solid.

Select the Centerline Tool, and drag a horizontal centerline from the origin, as shown in Figure 1.163. Select the Smart Dimension Tool. Click on the top line of the sketch and then on the horizontal centerline. Before clicking to place the dimension, recall from the previous section that if the dimension is placed above the centerline, a radius dimension is created, while if the dimension is placed below the centerline, a diameter dimension is created. Click below the centerline to place the dimension, as shown in Figure 1.164, and enter the value of the dimension as 7.0 inches. Repeat to add the 5.0-, 1.75-, and 1.0-inch diameter dimensions shown in Figure 1.165. Note that after creating the first diameter dimension, simply clicking on the next item to be dimensioned above the centerline will cause a diameter dimension to be created by default. If the lowest line moves below the centerline as you are adding the dimensions, then you can simply click and drag it above the centerline before adding its diameter dimension. Also note that after placing the dimensions below the centerline, you can click and drag the numerical value above the centerline if desired, and the dimension will remain as a diameter dimension.

FIGURE 1.163



The sketch should be fully defined.

FIGURE 1.164

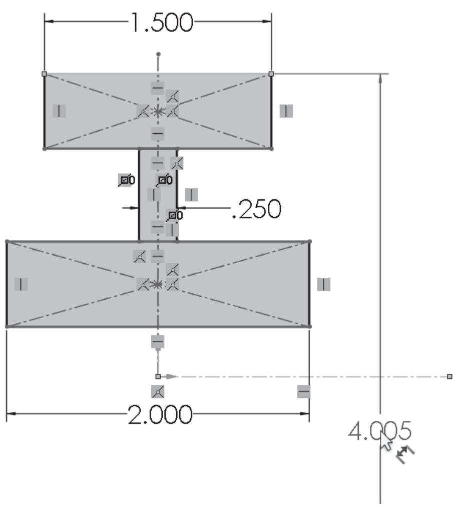
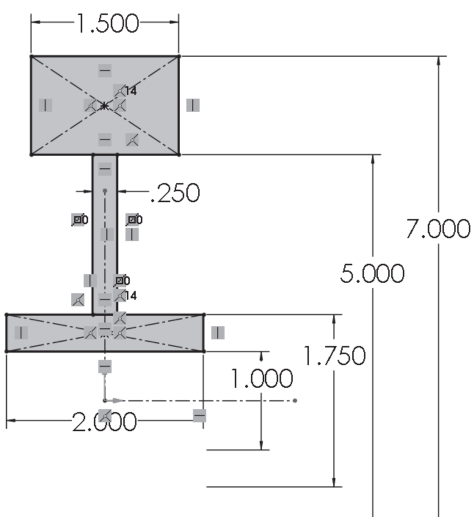


FIGURE 1.165

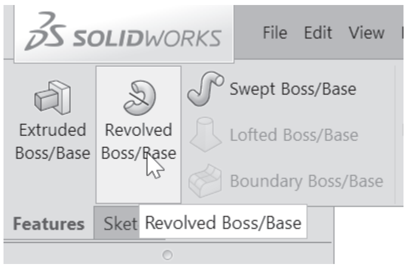


DESIGN INTENT Planning for Other Uses of the Model

When adding dimensions to the initial sketch of the pulley, we could specify some of the dimensions as either diameters or radii. For example, the 7-inch-diameter dimension that defines the overall size of the pulley could just as easily be entered as a 3.5-inch-radius dimension. However, if we plan to make a 2-D drawing of this part, then the diameter should be defined as a diameter on the drawing. By dimensioning the part in the same way that we will dimension the drawing, then dimensions can be imported directly from the part file. This prevents us from having to add dimensions manually or override a dimension's properties. You should consider all of the possible future uses of a model (e.g., drawing creation, stress analysis, 3-D printing, etc.) when choosing the best way to create and dimension a part.

From the Features group of the CommandManager, select the Revolved Boss/Base Tool, as shown in Figure 1.166.

FIGURE 1.166



Note that no preview is displayed on the screen as it was when we selected the Extrude Boss/Base tool earlier in the chapter. The reason for this is that there are two ambiguities in our sketch that must be defined:

1. There is more than one enclosed region (contour) within our sketch, so we need to define which of these regions will be revolved.
2. There are two centerlines in the sketch, so we need to define which centerline is the axis of revolution.

FIGURE 1.167

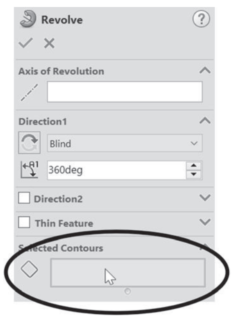


FIGURE 1.168

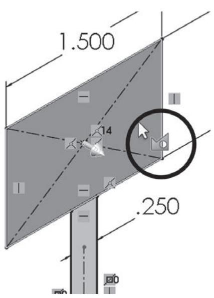
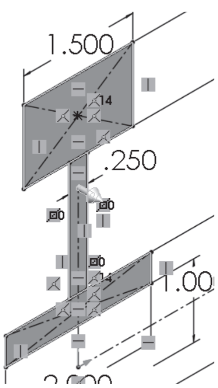


FIGURE 1.169

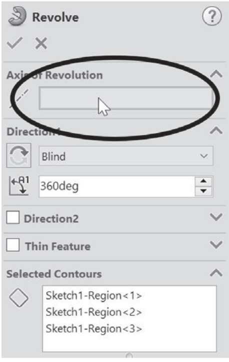


One way to approach the first ambiguity would be to use the Trim Entities Tool to remove the overlapping portions of the rectangles, so that the sketch consists of only a single closed contour. An easier way is to simply select the multiple contours.

In the PropertyManager, click in the Selected Contours box to select it, as shown in Figure 1.167. Switch to the Isometric View and zoom in to the three rectangles. Move the cursor into the top rectangle, and click to select the rectangular region, as shown in Figure 1.168. Repeat for the other two rectangular regions, as shown in Figure 1.169.

Click in the Axis of Revolution box in the PropertyManager to select it, as shown in Figure 1.170. Click on the horizontal centerline, as shown in Figure 1.171.

FIGURE 1.170



A preview will now be displayed, as shown in Figure 1.172. In the PropertyManager, we can change the number of degrees of the revolution if we want less than a fully revolved part. Since we want to revolve the section a full 360 degrees, we can accept the default value.

FIGURE 1.171

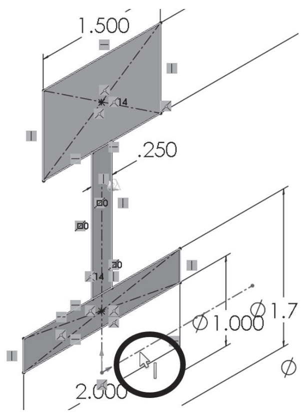
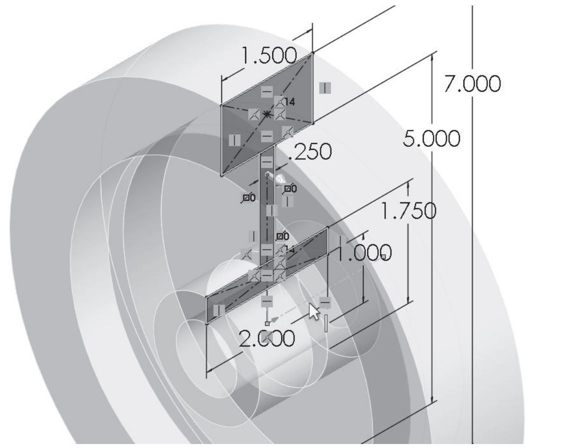


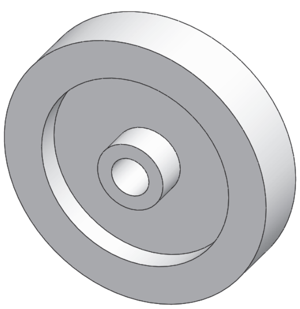
FIGURE 1.172



Click the check mark to complete the revolution.

The resulting solid part is shown in Figure 1.173 (shown in the Trimetric View).

FIGURE 1.173



Select File: Save from the main menu. Make sure to change the file directory to the location where you want to save the file, since the default path will be to the directory containing the template files. If necessary, change the file type from Part Template to Part. Save the file with the name "Pulley."

Select the Right Plane, and select Normal To from the Context Toolbar, as shown in Figure 1.174. Select the Centerline Tool, and create vertical and horizontal centerlines from the origin, as shown in Figure 1.175.

FIGURE 1.174

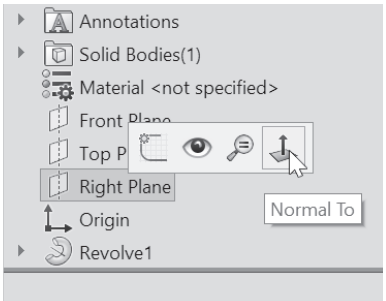
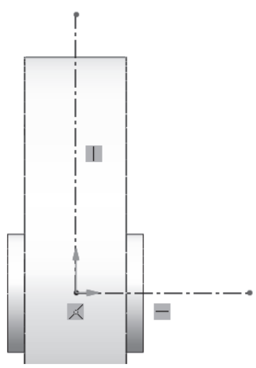


FIGURE 1.175



When using either the Centerline Tool or Line Tool, note that there are two different methods for drawing line segments. If you click once to set the first endpoint, move the cursor and then click again to set the other endpoint, you can then move the cursor to create another line segment beginning at the previous endpoint. This technique allows you to create continuous line segments. Double-clicking after the last endpoint has