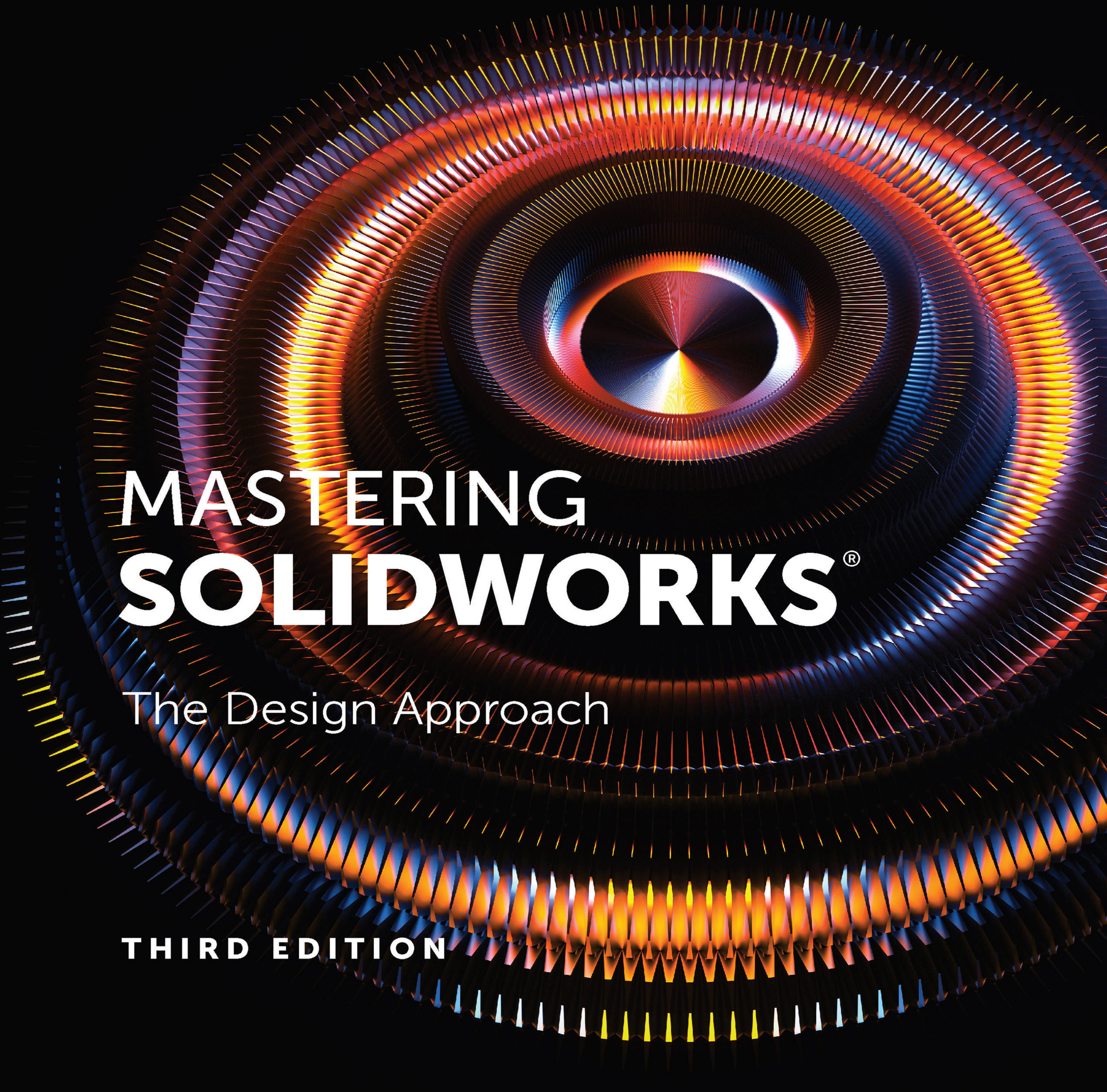

MASTERING SOLIDWORKS®

The Design Approach

THIRD EDITION

IBRAHIM ZEID
NATHAN BROWN

Mastering SolidWorks®

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The Design Approach

Third Edition

Ibrahim Zeid
Nathan Brown

Pearson

Mastering SolidWorks

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Features of Mastering SolidWorks®: The Design Approach

Tutorials

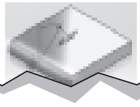
Tutorial 4–6: Create Features

This tutorial covers the creation of these features: chamfer, fillet, slot, shell, draft, and rib. All dimensions are inches. Here are useful observations:

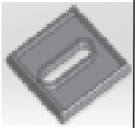
1. Make sure to pay attention to the visual clues shown in the left pane while creating these features.
2. For example, the box symbol under **Chamfer parameters** indicates that you can chamfer a face, an edge, or a vertex (corner point). As expected, chamfering a face chamfers all its edges. Chamfering a corner chamfers the three edges that meet there.
3. A rib requires a profile sketch (e.g., a line or stepwise line) and a thickness.

Step-by-Step Instructions

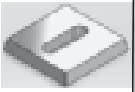
Step 1: Create *Sketch1* and *Block* feature: **File** > **New** > **Part** > **OK** > **Top Plane** > **Extruded Boss/Base** on **Features** tab > **Center Rectangle** on **Sketch** tab > click origin and sketch and dimension as shown > exit sketch > reverse extrusion direction > enter 0.5 for thickness **D1** > ✔ > **File** > **Save As** > *tutorial4.6* > **Save.**



Step 2: Chamfer an edge of *Block* feature: **Fillet** dropdown on **Features** tab > **Chamfer** > select *Block* edge shown



Step 5: Shell *Block* feature: **Shell** on **Features** tab > select top face of *Block* > enter 0.1 for wall thickness **D1** > ✔.



Step 6: Draft *Block* feature: Delete the chamfer, fillet, and shell features > **Draft** on **Features** tab > enter 10 degrees for **Draft Angle** > select top face of *Block* as **Neutral Plane** > select *Block* four side faces to draft > ✔.

Hands-on for Tutorials

HANDS-ON FOR TUTORIAL 12–1. Edit the title block to add a tolerance general note in the **Comments** box of the title block. The note should read:

GENERAL TOLERANCE
.X ± .030
.XX ± .010
.XXX ± .005
.XXXX ± .0005

Examples and Solutions

Example 12.1 Calculate the limits and tolerance zones for the following three fits: clearance of RC3, transition of LT4, and interference of FN2. Use a basic size of 5.0000 in.

Solution Following the above four steps, Table 12.2 shows the results.

TABLE 12.2 Limits for Basic Size *d* = 5.000 in.

Fit	d_{hmin}	d_{hmax}	d_{smin}	d_{smax}	h	s	A
RC3 (H7/f6)	5.0000	5.0016	4.9974	4.9984	0.0016	0.0010	0.0016
LT4 (H8/k7)	5.0000	5.0025	5.0001	5.0017	0.0025	0.0016	−0.0017
FN2 (H7/s6)	5.0000	5.0016	5.0035	5.0045	0.0016	0.0010	−0.0045

Problems

problems

1. List the sources of variability in manufacturing.
2. Why do we need tolerances?
3. List the two types of tolerances. What does each type control?
4. Inspection gauges are used to check whether a part is within its tolerance limits. Perform an in-depth research study on inspection gauges including their types, their design, and how they are used during part inspection.
5. What is the difference between a nominal and a basic size? Give an example.
6. Three types of tolerances exist: unilateral, bilateral, and symmetric. Describe each type. Give a numerical example

Preface

The target audience for this book is college students in courses that use SolidWorks to learn and master CAD/CAM for design, visualization, prototyping, and manufacturing. The book's primary market is four-year colleges and two-year community colleges. Freshman Engineering Design courses should find this book useful, refreshing, and interesting. Other important markets include high schools, professionals, and training courses. We have written the book with the target audience in mind. Page iii highlights some of the book's features.

The book includes relevant math in Chapter 8 (Curves), Chapter 9 (Surfaces), and Chapter 13 (Analysis Tools). The math is concentrated in one or two sections in each of these three chapters. We include the math for two reasons. First, it shows students who are curious how CAD/CAM systems work "under the hood." Second, it broadens the book's appeal to many students, professors, and readers. This math may be ignored without affecting the continuity of the coverage of the material in any of these three chapters.

The philosophy behind the book is original, unique, and effective. We cover and present SolidWorks as a design system rather than a software program. Thus, instead of focusing on describing SolidWorks menus and syntax, we describe design approaches, methodologies, and techniques to help CAD designers/engineers and draftspersons achieve engineering tasks in the fastest, easiest, and most effective way.

Based on this philosophy, the book focuses on design, modeling, and drafting concepts instead of menus and commands. Thus, we present command sequences to achieve CAD and modeling tasks. Of course, we provide SolidWorks syntax and details, but do so in accordance with the proposed philosophy of the book. We start with a CAD task to accomplish (what to do) and then go about accomplishing it (showing how to use SolidWorks to do it). This philosophy is more motivating to student learning than simply going through layers of menus and commands.

The book approach is designed to showcase SolidWorks as a powerful modeling and design system instead of only a software program. We include challenging modeling and design examples and problems in the book. As part of the book's unique approach, we cover the theoretical concepts behind the various functions of SolidWorks. This should provide information to curious minds about why things work the way they do, as well as explain their limitations and use.

The book provides abundant illustrations, step-by-step instructions, and rich and challenging end-of-chapter problems. The book is suitable for use at various levels, from freshman to senior to graduate courses. Instructors can choose the chapters and topics that suit their teaching needs and courses. They can also choose the level of depth. The book includes both examples and tutorials. An example covers one concept whereas a tutorial is more comprehensive by covering a full design task. Most examples and tutorials have a hands-on exercise at the end that serves two purposes. First, it ensures that the student has completed the example or tutorial, because it builds on it. Second, it both challenges and extends the student's understanding.

The book is organized into parts and chapters. Instructors may cover the chapters in any order to fit their course and student needs. However, we recommend covering Chapters 1 and 2 first to build a sound foundation in 3D CAD/CAM modeling concepts. Chapter 1 is designed to provide a quick grasp of basic functionalities: create parts, create assemblies, and create drawings. These three functionalities correspond to the three modes of SolidWorks: part, assembly, and drawing. Therefore, students can start designing basic and simple products after using only one chapter of the book; they do not have to wait until later chapters to learn how to design and document simple parts and assemblies. Later chapters provide further details. Thus, Chapter 1 provides breadth and the remainder of the book provides depth. Chapter 2 covers essential concepts required for a sound understanding of 3D modeling and efficient use of today's parametric features-based solid modeling CAD/CAM systems such as SolidWorks.

Acknowledgments from Second Edition

I would like to thank many people who contributed to this book including my students, the book reviewers, the Pearson team, the editorial team, and my family. Many of my students have shaped how I should present and teach concepts to help them understand better. They have also contributed directly and indirectly to the book. This book is the outcome of their influence.

I would also like to thank Boston Gear for granting permission to download and use some of their gears in the book tutorials. I thank Jake Hustad for providing the Universal Joint assembly. I also thank Ivette Rodriguez of ASME for granting permission to use ASME Y14.5M-1994 (R2004) material.

Many thanks are due DS SolidWorks Corporation for its technical support throughout the writing process and using SolidWorks. My sincere thanks go to my friends Marie Planchard, Christian Blanc, and Christine Morse.

I owe thanks to the many reviewers who helped to shape this book (first edition). They are:

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Last, but not least, my family and friends deserve many thanks for their support, and apologies to them for hiding out to finish this book project. Their love and unconditional support is priceless.

—Abe Zeid

Contents at a Glance

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Figure Credits

Chapter	Figure	Credit
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SolidWorks screenshots		© 2002–2020 Dassault Systèmes SolidWorks Corporation
2	FIG02–33	Courtesy of VIC
2	FIG02–34	Courtesy of VIC
6	FIG06–07_Step-02	Screenshot © 2020 Altra Industrial Motion Corp
6	FIG06–08_Step-01	Screenshot © 2020 Altra Industrial Motion Corp

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Part II

Basic Part Modeling

The primary goal of this part is to explore and cover the details of engineering drawings, assemblies, and rendering. We have covered these topics briefly in Part I. Each topic has its own chapter in Part II to understand it in depth. Part II also covers the full set of features that can be used in CAD modeling. In Part I, we purposely limited our models to the basic features of extrusions and revolves.

Chapter 4, "Features and Macros," is all about when and how to use the full set of features available to design advanced parts with complex geometry. Chapter 5, "Drawings," covers the details of drawings, including the creation and control of the title block. Chapter 6, "Assemblies," covers assembly details, including the bottom-up and top-down approaches. Chapter 7, "Rendering and Animation," closes Part II by showing how to create realistic renderings of parts and assemblies that show material and texture. CAD visualization is important to convey and present designs efficiently.

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4 chapter four

Features and Macros

4.1 Introduction

We have been using a limited set of features thus far in the book. We have used extrusions and revolves. We have used the SolidWorks features **Extruded Boss/Base** and **Revolved Boss/Base** and their subtracting counterparts, **Extruded Cut** and **Revolved Cut**. These four features create one class of parts: those with constant cross sections. We use the extruded boss or cut to create uniform thickness parts. We use the revolved boss or cut to create axisymmetric parts. Amazingly, these four features can create 80% or more of the mechanical parts you are likely to need.

These four features cannot create some classes of parts: They cannot create parts whose cross sections are variable or parts that have nonplanar faces or other geometric shapes. The features that allow us to create these types of parts are **Lofted Boss/Base**, **Swept Boss/Base**, **Lofted Cut**, **Swept Cut**, **Hole Wizard**, **Rib**, **Draft**, **Shell**, and **Dome**, as shown in Figure 4.1. We cover all these features in this chapter. You can also access more features by clicking this sequence: **Insert** (menu) > **Features**.

Figure 4.1 Available features

A **feature** is defined as a solid that, when combined with other features (solids), creates parts. A CAD part consists of a set of features created in a certain sequence stored in its feature tree. Some features, such as bosses and cuts, originate as sketches, while others, such as shells and fillets, modify other features. Features are always listed in the feature tree (which SolidWorks calls the **FeatureManager Design Tree**) of the part.

Today’s modeling is referred to as feature-based modeling, and the resulting models are known as feature-based models. The first feature you create in a part is known as the base feature. You use the base feature as the basis to create other features. A base feature obviously cannot be negative (i.e., a cut), and SolidWorks does not allow it. When you begin creating a feature, **Extruded Boss/Base** and **Revolved Boss/Base** are the only selectable feature types. You might also expect **Swept Boss/Base** and **Lofted Boss/Base** to be selectable when you begin creating a feature, but they are not. **Lofted Boss/Base** becomes selectable only after you create a sketch (profile), and **Swept Boss/Base** becomes selectable after you create a cross section and a path (sweep direction).

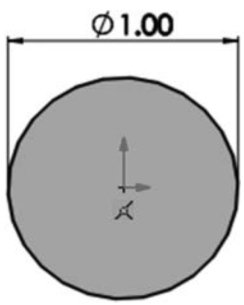
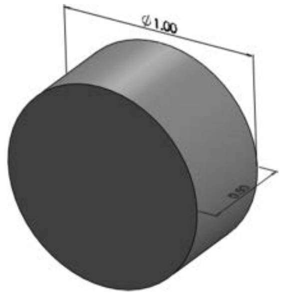
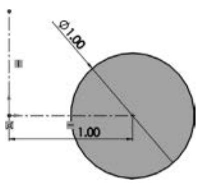
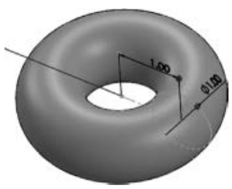
4.2 Features

To master feature-based modeling, you should be able to answer three fundamental questions:

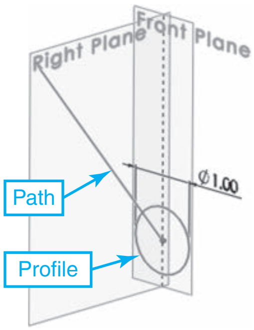
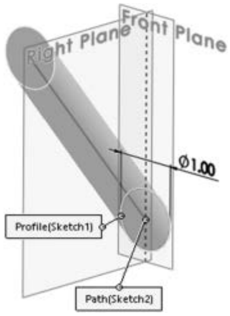
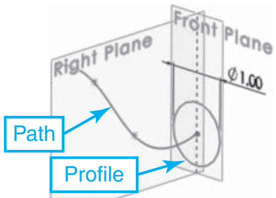
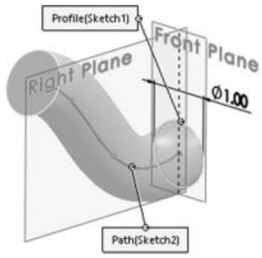
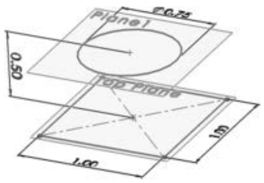
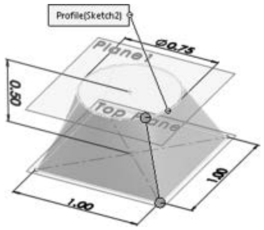
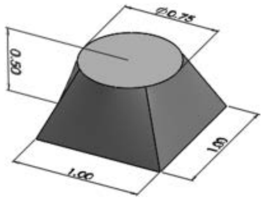
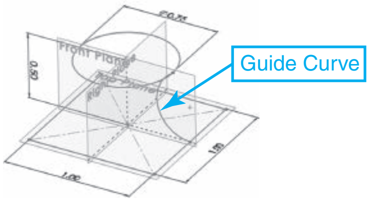
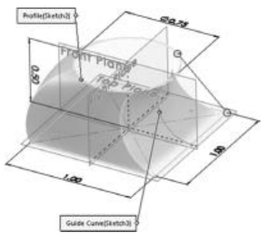
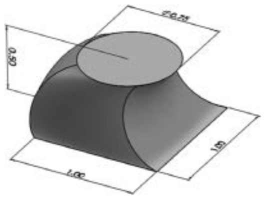
1. What are the available features that a CAD/CAM system offers for modeling parts?
2. What is the input required to create each feature?
3. Which feature should you use for a given modeling problem?

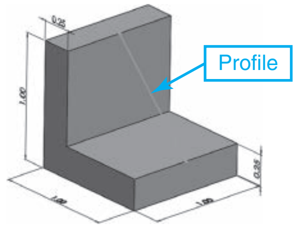
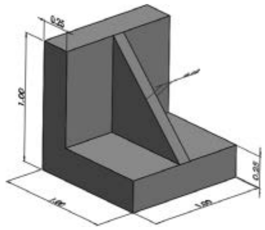
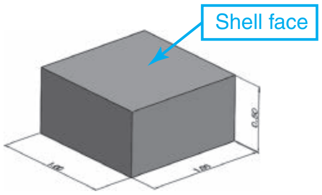
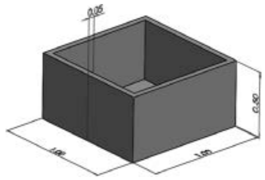
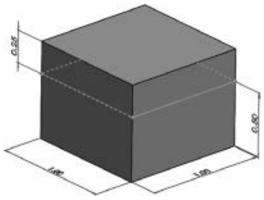
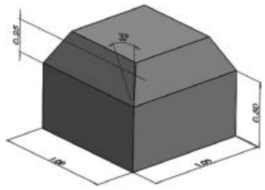
Section 4.1 and Figure 4.1 provide the answer to the first question. Table 4.1 answers the other two questions. It shows a simple basic example of each feature. Keep in mind that the third question may have multiple answers; one of them is always the best answer. For example, you may use a loft or a sweep. However, if a part has a constant cross section along a curve, sweep is better to use because it requires fewer steps to create the part. If the part has a variable cross section, a loft is better to use. The tutorials in this chapter provide some modeling examples.

Table 4.1
Available Features

No.	Feature	Input (sketch)	Resulting Feature	When to Use in Modeling?
1	Extrusion	Cross section and a thickness 		• Use for parts with constant cross section (CS) and uniform thickness (UT). • If needed, break part into subparts, each with a constant CS and UT.
2	Revolve	Cross section, an axis of revolution, and an angle of revolution 		• Use for parts that are axisymmetric. • If needed, break part into subparts, each of which is axisymmetric.

Chapter 4

No.	Feature	Input (sketch)	Resulting Feature	When to Use in Modeling?
3	Sweep	Linear sweep: cross section and a line as a path 		• Use for parts with constant cross section (CS) along a linear direction (path) that may or may not be perpendicular to the cross section. • If the path is perpendicular to the cross section, the linear sweep becomes an extrusion.
		Nonlinear sweep: cross section and a curve as a path 		• Use for parts with constant cross section (CS) along a nonlinear direction that may or may not be perpendicular to the cross section.
4	Loft	Linear loft: at least two cross sections (profiles) 	 	• Use for parts with variable cross section along a given direction. • The cross sections are blended linearly from one section to the other.
		Nonlinear loft: at least two cross sections (profiles), and a curve as a guide curve 	 	• Use for parts with variable cross section along a given direction. • The cross sections are blended nonlinearly from one section to the other, along the guide curve.

No.	Feature	Input (sketch)	Resulting Feature	When to Use in Modeling?
5	Rib	Rib profile (e.g., line or stepwise line) 		• Use when a stiffener between angled walls (faces) of a part is required to increase part structural strength.
6	Shell	Shell face and shell wall thickness 		• Use when you need to remove material from an existing part. • The material removal (shelling) occurs in a direction perpendicular to the selected shelling face. • While you can achieve the same result using an extrude cut for simple shells, a shell operation is faster to use.
7	Draft	Direction of pull, parting lines, and a draft angle. The direction of pull must be perpendicular to the parting lines. 		• Use when you need to draft faces at an angle; usually used for injection molding to allow pulling the molded part from the mold cavity.

The other features shown in Figure 4.1 and not covered in Table 4.1 are covered in the tutorials in this chapter.

Example 4.1 Create the free-form torus shown in Figure 4.2.

Solution The torus shown in Figure 4.2 is a variation of the torus (donut shape) feature (No. 2) shown in Table 4.1. While that feature of Table 4.1 is a revolve, the free-form torus shown in Figure 4.2 can only be created as a sweep. The key modeling concept here is to use pierce relations to force the torus cross section (small circle) to conform to the sweep path (large circle) and the guide curve (closed spline), as shown in Figure 4.2. You pierce the small circle to the spline and pierce the center of the small circle to the large circle. These two pierce conditions force the small circle to become "elastic"; that is, it expands and shrinks, as it must always touch the spline and the big circle. Pierce condition is only available to pierce a point to a curve. You cannot pierce two curves. SolidWorks enables the pierce condition in the right context.

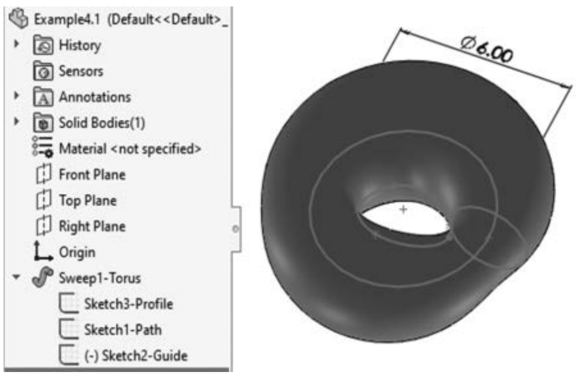
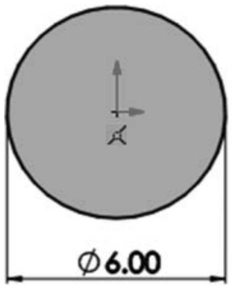
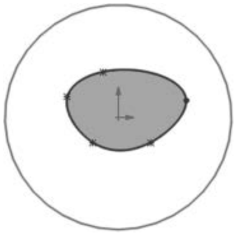


Figure 4.2
Free-form torus

Step 1: Create *Sketch1-Path* (sweep path): **File** > **New** > **Part** > **OK** > **Top Plane** > **Sketch** tab > **Circle** on **Sketch** tab > click origin to sketch and dimension as shown > exit sketch > **File** > **Save As** > *example4.1* > **Save**.

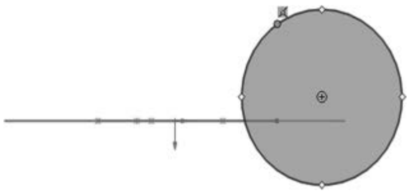


Step 2: Create *Sketch2-Guide* (sweep guide curve): **Top Plane** > **Sketch** tab > **Spline** on **Sketch** tab > sketch free spline as shown > exit sketch.



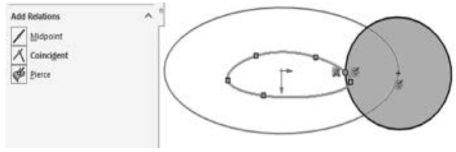
Note: *Sketch1-Path* and *Sketch2-Guide* are two separate sketches, and they both use **Top Plane**.

Step 3: Create *Sketch3-Profile* (sweep profile): **Front Plane** > **Sketch** tab > **Circle** on **Sketch** tab > sketch a circle anywhere > **Point** on **Sketch** tab > click circle anywhere.

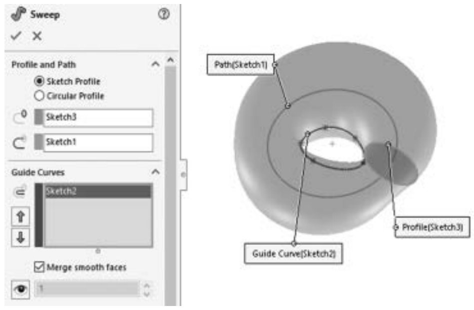


Note: Do not dimension the circle as doing so over-constrains it when you apply the pierce relation.

Step 4: Create pierce relations: While *Sketch3-Profile* is still open from Step 3, click small circle center + **Ctrl** + select large circle > **Pierce** relation > ✓ > select point created on circle + **Ctrl** + spline > **Pierce** relation > ✓ > exit sketch.



Step 5: Create *Sweep-Torus* feature: **Features** tab > **Swept Boss/Base** > select *Sketch3-Profile* as **Profile** > select *Sketch1-Path* as **Path >** select *Sketch2-Guide* as **Guide Curve** > ✓.



HANDS-ON FOR EXAMPLE 4.1

Re-create the free-form torus by replacing the spline by a circle that is not centric with the large circle.

4.3 Spur Gears

Gears are an important and essential mechanical element in mechanical design. A wide range of products and applications use gears. There are various types of gears: spur, helical, bevel, spiral, worm, planetary, and rack and pinion, to name a few. A spur gear is the simplest type of gear and the type we cover here. Typical mechanical design courses in colleges cover the principles and design of gears. In this section, we cover spur gears from a CAD point of view (i.e., how we construct a gear once it is designed). While gears are standard elements that can be purchased off the shelf (they can also be inserted from the SolidWorks Toolbox into a part or assembly file), it is important to learn how to create a gear feature in a CAD/CAM system.

A gear tooth is the intricate part of a gear. Figure 4.3 shows two meshing gears. Figure 4.4A shows the conjugate line and pressure angle. Figure 4.4B shows the involute profile. Gearing and gear meshing ensure that two disks (the two gears) in contact roll against one another without slipping.

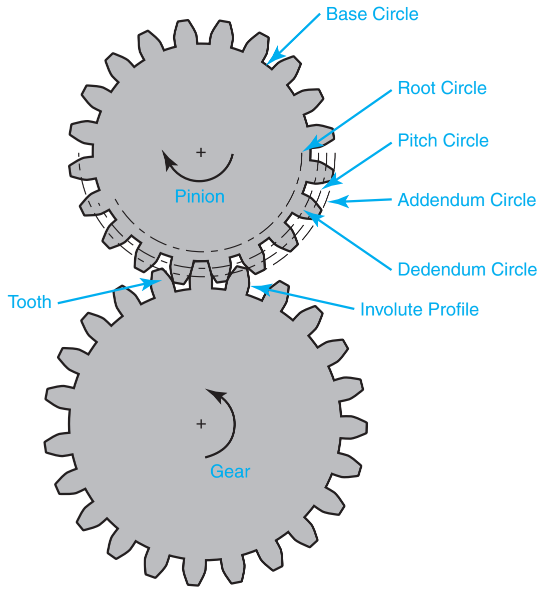


Figure 4.3
Meshing gears

Moreover, the gear teeth should not interfere with the uniform rotation that one gear would induce in the other—a requirement known as the conjugate action. The conjugate action also ensures that the perpendicular line to a tooth profile at its point of contact with a tooth from the other gear always passes through a fixed point on the centerline connecting the centers of the two meshing gears. Figure 4.4A shows the conjugate line. The conjugate line is also known as the line of force because the driving force from the driving gear (driver) is transmitted in the direction of this line to the other gear (driven). The angle between the perpendicular radius to the conjugate line and the centerline is always constant for two meshing gears. This angle is known as the *pressure angle* and is shown as the angle Ø in Figure 4.4A.

The key to successful functional gears is the conjugate action. While various profiles can produce conjugate action, the involute profile is the best because it allows for imperfections in gear manufacturing and yet maintains the conjugate action. The imperfection may produce a slightly different distance between the two shafts of the gears from the designed value. Figure 4.4B shows how the shape of the involute profile is generated. An **involute** is defined as the path of the endpoint of a cord when it is pulled straight (held taut) and unwrapped from a circular disk, as shown in Figure 4.4B. The involute geometry ensures that a constant rotational speed of the driving gear produces a constant rotational speed in the driven gear. For spur gears, the teeth are cut perpendicular to the plane of the gear, where the involute profile resides.

The creation of a gear CAD model requires two basic concepts: knowledge of the gear geometry and the involute equation. The geometry is shown in Figure 4.3. The **base circle** is the circle where the involute profile begins. The **pitch circle** defines the contact (pitch) point between the two gears (see Figure 4.4A). The **dedendum circle** is usually the same as the base circle, as can be concluded from Figure 4.4A (dedendum $d = r_p - r_b$). The **addendum circle** is the circle that defines the top of the tooth as shown in Figure 4.4C (addendum $a = r_a - r_p$, where r_a is the addendum circle radius). Typically, the addendum and the dedendum are equal. In such case, the pitch and base circle sizes determine the values for both. The **root circle** is

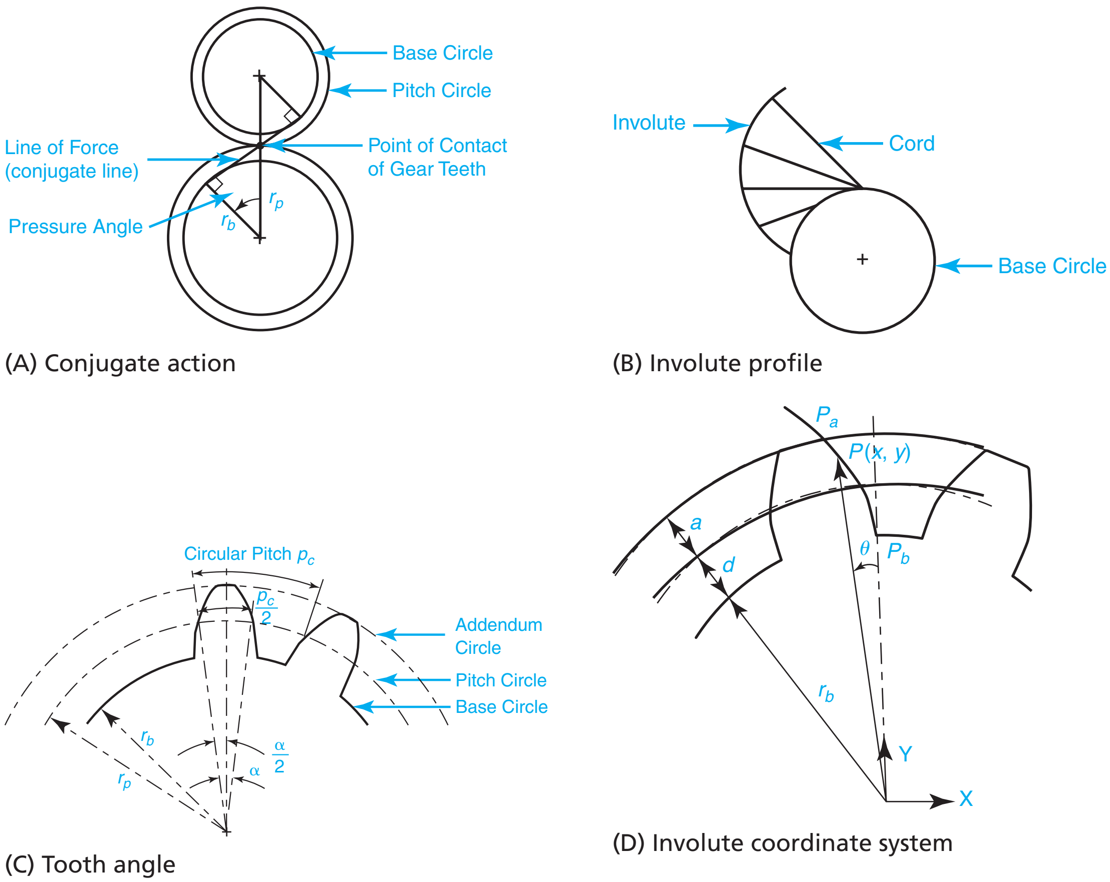


Figure 4.4
Details of a gear tooth

smaller than the base circle to allow cutting the tooth during manufacturing. The tooth profile between the base and root circles is not an involute. It could be any geometry, such as line.

The creation of a gear CAD model requires two steps: Calculate the tooth angle α and the tooth involute profile. While many books on mechanical engineering design offer extensive in-depth coverage of gear analysis, we offer a simplified but accurate version that enables us to create a CAD model of the gear. We begin with the definition of circular pitch. As shown in Figure 4.4C, the **circular pitch, p_c,** is defined as the distance along the pitch circle between corresponding points on adjacent teeth. As shown in Figure 4.4C, we use p_c as the circular pitch of the gear, r_p as the pitch circle radius, and α as the tooth angle. Using these variables, we can write:

$$p_c = \frac{\pi d_p}{N} \tag{4.1}$$

where $d_p = 2r_p$ is the pitch circle diameter and N is the number of gear teeth. From the tooth geometry shown in Figure 4.4C, we can write:

$$\frac{p_c}{2} = r_p \alpha \tag{4.2}$$

Substituting p_c from Eq. (4.2) into Eq. (4.1) and reducing gives:

$$\alpha = \frac{\pi}{N} \text{ radius or } \alpha = \frac{180}{N} \text{ degrees} \tag{4.3}$$

The derivation of the involute equation is more complex and is not covered here. We align the involute of one tooth with the XY coordinate system as shown in Figure 4.4D, where the lowest point P_b on the involute lies on the Y axis. This orientation does not represent a limitation but rather simplifies the form of the involute equation, which is therefore given by:

$$\begin{aligned} x &= -r_b(\sin\theta - \theta\cos\theta) \quad \theta \le \theta \le \theta_{\max} \\ y &= r_b(\cos\theta + \theta\sin\theta) \end{aligned} \tag{4.4}$$

where r_b (the base circle radius) is given by (see Figure 4.4A):

$$r_b = r_p \cos\phi \tag{4.5}$$

and (x, y) are the coordinates of any point P on the involute at an angle θ, as shown in Figure 4.4D. The lowest point P_b on the involute corresponds to the value of $\theta = 0$ and lies on the base circle. Point P_a lies on the addendum circle and does not necessarily correspond to the value of $\theta = \theta_{\max}$. We can arbitrarily select a large enough value for $\theta_{\max}$ so that the involute crosses the addendum circle and then trim it to that circle. Therefore, we create the involute profile by generating points on it using Eq. (4.4) and connecting them with a spline curve, or we input Eq. (4.4) into a CAD/CAM system.

The root circle is always less than the base circle. For simplicity, we have the root circle radius, r_r, be 0.98 of the base circle radius. (There are other formulas that do not give consistent results.) Thus, we write:

$$r_r = 0.98 r_b \tag{4.6}$$

The following steps summarize the calculations we need to create a gear CAD model:

1. The input parameters we need are the pitch circle radius r_p, the pressure angle $\varnothing$, and the gear number of teeth N.
2. Calculate r_b using Eq. (4.5).
3. Calculate r_r using Eq. (4.6).
4. Calculate the gear dedendum $d = r_p - r_b$.
5. Assuming that the addendum and dedendum are equal, calculate the addendum circle radius as $r_a = r_p + a = r_p + d$ (see Figures 4.4C and 4.4D).
6. Use Eq. (4.3) to calculate the tooth angle α.
7. Enter the involute parametric equation given by Eq. (4.4) into a CAD/CAM system to sketch the involute curve as a spline.
8. Create one gear tooth and use a sketch circular pattern to pattern it to create all gear teeth.

Example 4.2 Create the CAD model of a spur gear with $r_p = 60$ mm, $\varnothing = 20°$, and $N = 20$.

Solution Using the above calculation steps, you get $r_b = 56.382$ mm, $d = a = 3.618$ mm, $r_a = 63.618$ mm, $r_r = 55.254$ mm, and $\alpha = 9°$. There are two methods to create the tooth involute curve.

In the first method, you use Eq. (4.4) with $\Delta\theta = 5°$. You generate 11 points on the involute, for $\theta_{max} = 50°$. You generate the points on the involute curve. You then use **Insert** > **Curve** > **Curve Through XYZ Points**. A better method is to input Eq. (4.4) into SolidWorks and let SolidWorks generate the curve. You need to use radians for the angle θ. You use 1 radian for θ_{max}. This value is arbitrary to ensure that the involute curve intersects and crosses the addendum circle to be able to trim it to the intersection point. Also, SolidWorks uses the parameter t, requiring us to replace θ with t when you input the equation. Figure 4.5 shows the spur gear. You create half a tooth, mirror it to create a full tooth, and use a circular pattern for the full tooth to create all teeth of the gear. Here are the detailed steps.

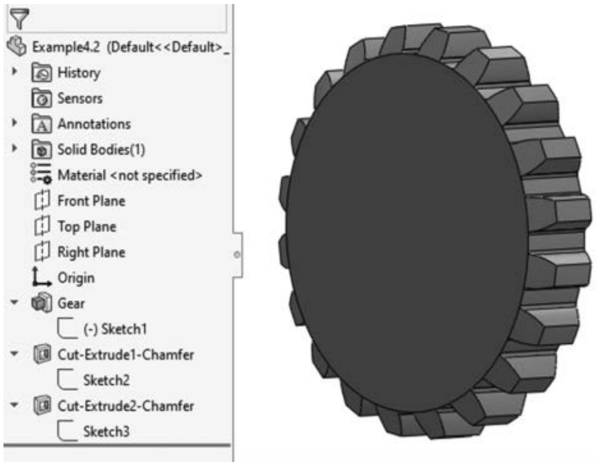
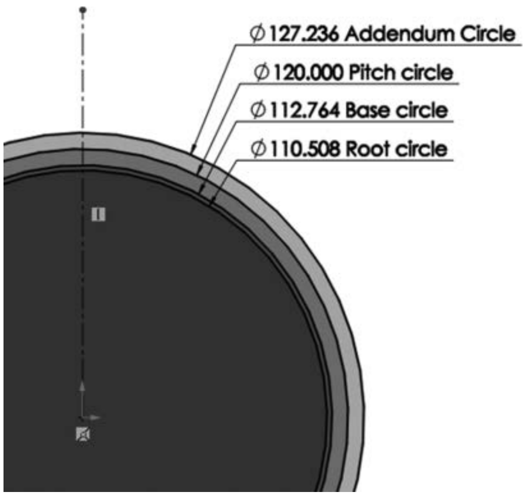


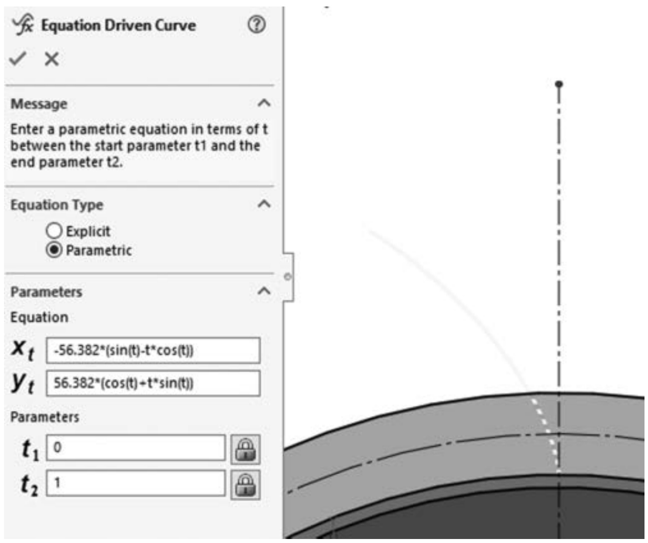
Figure 4.5
Spur gear

Step 1: Create *Sketch1* circles and axes: **File** > **New** > **Part** > **OK** > **Front Plane** > **Circle** on **Sketch** tab > sketch four circles and dimension as shown > **Centerline** on **Sketch** tab > sketch vertical line > **File** > **Save As** > *example4.2* > **Save**.

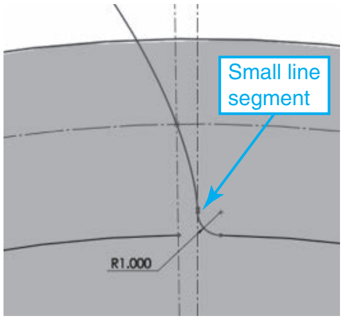
Note: Set the part units to mm before you start. The vertical centerline serves as a validation that the involute bottom endpoint passes through it when you create it in Step 2. Also, you will not close the sketch until you finish Step 5.



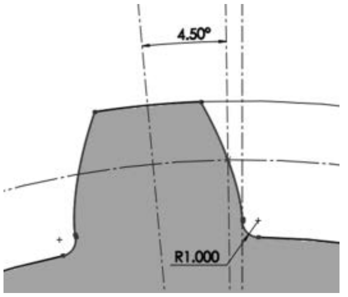
Step 2: Create *Sketch1* tooth involute: **Front Plane** > **Sketch** tab > **Spline** dropdown on **Sketch** tab > **Equation Driven Curve** > **Parametric** > enter x and y equations and limits as shown > ✓.



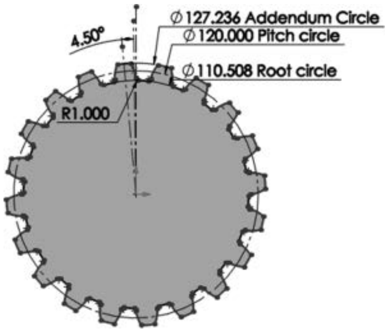
Step 3: Create *Sketch1* tooth bottom: **Line** on **Sketch** tab > sketch a line passing through bottom end of involute curve and crossing the root circle > **Esc** on keyboard > select the line + **Ctrl** on keyboard + involute curve > **Tangent** from **Add Relations** options on left pane > ✓ > **Point** on **Sketch** tab > create a point at intersection of involute and pitch circle (turn relations on: **View** > **Sketch Relations** to see all) > **Centerline** on **Sketch** tab > sketch a line passing through origin and crossing involute at any point > **Esc** key > select centerline just created + **Ctrl** + point > **Coincident** from **Add Relations** options on left pane > ✓ > **Trim Entities** on **Sketch** tab > **Trim to closest** > select line below root circle and select root circle between two centerlines > ✓ > **Fillet** on **Sketch** tab > enter 1 mm for radius > select line and root circle > **Yes** to continue > ✓ > select base circle > **Delete** key on keyboard.



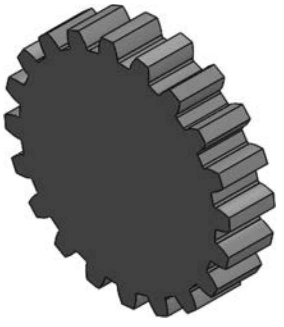
Step 4: Create *Sketch1* tooth other half: **Trim Entities** on **Sketch** tab > **Trim to closest** > select involute top part > **Centerline** on **Sketch** tab > sketch a line passing through origin and to left of involute > **Smart Dimension** on **Sketch** tab > select the centerline just created and the other centerline to the right of it > enter 4.5 > ✓ > **Mirror Entities** on **Sketch** tab > select involute + **Ctrl** key + line segment connected to involute + fillet created in Step 3 > click **Mirror about** box on left of screen > select the far left centerline > ✓ > **Trim Entities** on **Sketch** tab > **Trim to closest** > click addendum circle outside tooth > click root circle inside tooth twice to delete its two segments inside the tooth > ✓.



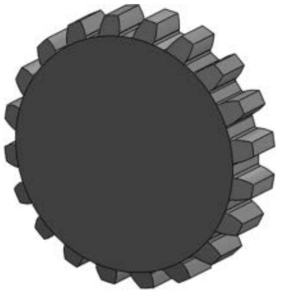
Step 5: Create *Sketch1* all gear teeth: **Linear Sketch Pattern** dropdown on **Sketch** tab > **Circular Sketch Pattern** > click first box under **Parameters** on left pane > select origin to define axis of pattern > click **Entities to Pattern** box > select the tooth profile 7 entities > enter 20 for the number of instances to create > ✓ > the sketch becomes over defined when you pattern the tooth because of the profile mirror of first tooth. Click this sequence to resolve it: **Over Defined** (shown red in status bar) > **Diagnose** > **Accept** > **Trim Entities** > **Trim to closest** > trim all excess from root circle (segments inside teeth) > ✓ > exit sketch.



Step 6: Create *Gear* feature: Select *Sketch1* > **Features** tab > **Extruded Boss/Base** > Enter 25 for **D1** > reverse extrusion direction > ✓.



Step 7: Create *Sketch2* and *Cut-Extrude1-Chamfer*: Select *Gear* front face > **Features** tab > **Extruded Cut** > **Circle** on **Sketch** tab > click origin and draw circle > select root circle and sketched circle and add **Coradial** relationship > exit sketch > enter 10 for **D1** > check **Flip side cut** > click **Draft** icon > enter 60 for **Draft Angle** > ✓.



Step 8: Create *Sketch3* and *Cut-Extrude2-Chamfer*: Repeat Step 7 but use the back face of *Gear*.

HANDS-ON FOR EXAMPLE 4.2

Add a hole and a keyway to the gear blank. Use a diameter of 50 mm for the hole and 10 × 10 mm keyway.

4.4 Design Library and Library Features

Design reuse and using off-the-shelf standard components are important concepts that speed up design and thus make it less expensive. The field of mechanical design has many standard parts that designers use every day in their designs (e.g., fasteners [nuts and bolts], gears, bearings). While these parts are universally standard, a company may have some parts that are unique and reusable only in that company's designs. SolidWorks provides the concepts of design library and library features to enable designers to reuse off-the-shelf components.

A **library feature** is a part that you create once and save in a library for reuse in the future. This library is known as the **design library**. You may save a library feature with the *.sldprt* or *.sldlfp* extension. Most of the time library features are inserted into assemblies as components or inserted into new empty (blank) parts. Commonly used library features include holes and slots. Using several library features to construct a single part saves time and also ensures consistency in your CAD models.

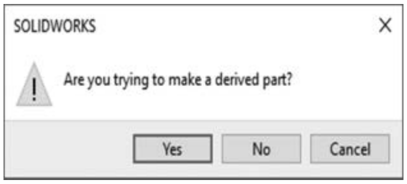
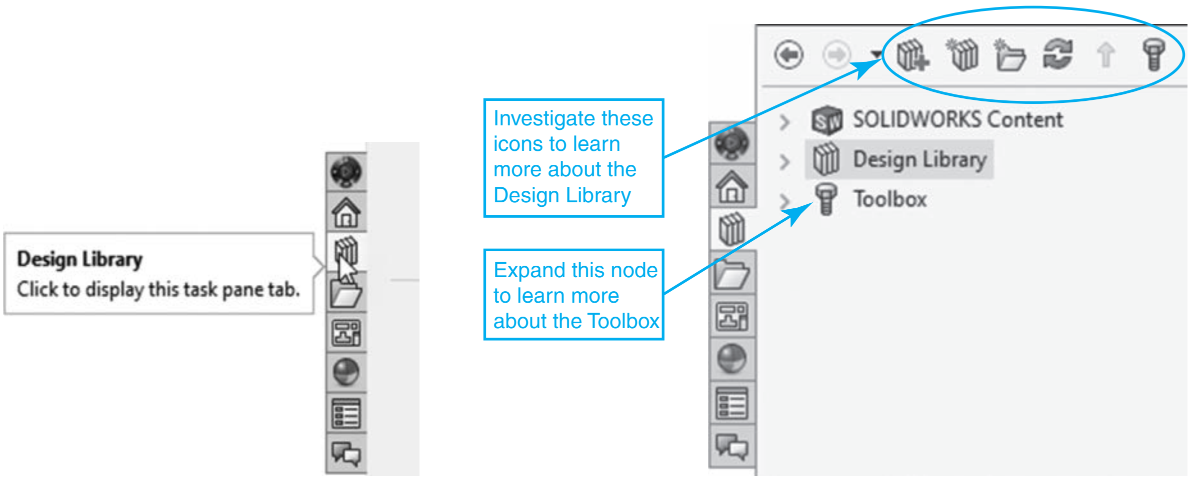


Figure 4.6
Using a library feature

Using library features is easy: You drag a library feature from the design library and drop it onto the open part or assembly. SolidWorks asks you if you want to insert a copy or instance (**derived part**) of the part, as shown in Figure 4.6. If you select **Yes**, it inserts the copy in the open part. If you select **No**, it opens a blank part and inserts it there.

You save library features in a design library, which you can organize into folders. The path to the SolidWorks design library folder is *C:\Program Data\SolidWorks\SolidWorks version\design library*. (Make sure that hidden files are visible in Windows to see the *Program Data* folder.) Click the **Design Library** tab in the SolidWorks Task Pane (on the right of the screen), as shown in Figure 4.7A, to open the design library.



(A) Open Design Library (B) Design Library folders

Figure 4.7
SolidWorks Design Library

The library is organized into folders, and you can add new custom folders to the library. You should save your library features into the SolidWorks design library to have them accessible. If you do not, you have to navigate to the folder where you saved the features. The most commonly used SolidWorks library is **Toolbox**, shown in Figure 4.7B. Expand the Toolbox node and investigate its contents.

4.5 Configurations and Design Tables

A family of parts is a natural outcome from the concept of parametric solid modeling. Defining a solid (part) in a sketch by parameters (dimensions are values for the parameters) enables us to modify the dimensions and create a similar but different-size replica of the part with a click of a button. We refer to these replicas as a family of parts. SolidWorks calls them *configurations*. You can also create configurations of assemblies. The configurations are created by changing the dimensions of some key parameters of the part or assembly. The configurations have the same topology as the original but different geometry. For example, consider a two-feature part: a base block and a shaft boss. You configure the part into a square block and a skinny long boss or a rectangle block and short fat boss.

You use design tables to help create and control configurations in three ways: change sizes, change configurations of components, and/or suppress/unsuppress features. SolidWorks uses a Microsoft Excel sheet as its design table. You can insert a design table into an open part or assembly by clicking this sequence: **Insert** (menu) > **Tables** > **Design Table**. This opens the **Design Table** under the **PropertyManager** tab on the left pane, as shown in Figure 4.8. You can specify one of the three sources shown to create the design table. When you click the green checkmark to finish, SolidWorks acts accordingly. For example, if you select the **Auto-create** (default) option, SolidWorks displays a list of the open part dimensions and asks you to select some to include in the design table. The table is created with the current values of the part dimensions as the default configuration. The configuration appears as a row in the table. You can add other rows with different values for dimensions. Each row is a different configuration. The **Design Table** is saved under the **ConfigurationManager** tab.

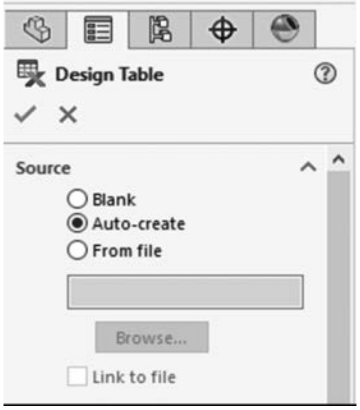
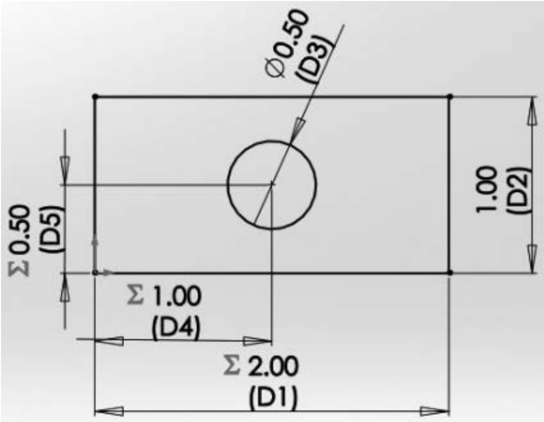


Figure 4.8
Design table

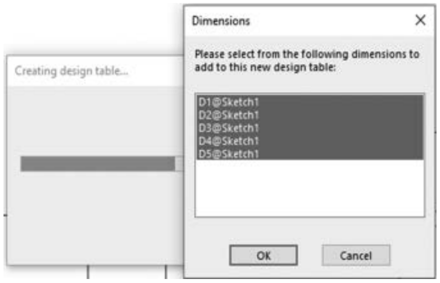
Example 4.3 Create design tables.

Solution This example builds on Example 2.4. It shows how to create a design table where you change the value of parameter (dimension name) *D2* to create four configurations of the sketch. The steps are as follows.

Step 1: Open *example2.4* part: **File** > **Open** > locate and select the file > **Open**.



Step 2: Create design table with *Default* config: **Insert** > **Tables** > **Design Table** > ✓ > click *D1* (shown here) **+ Shift** key on keyboard + *D5* (shown here) to select all dimensions > **OK**.

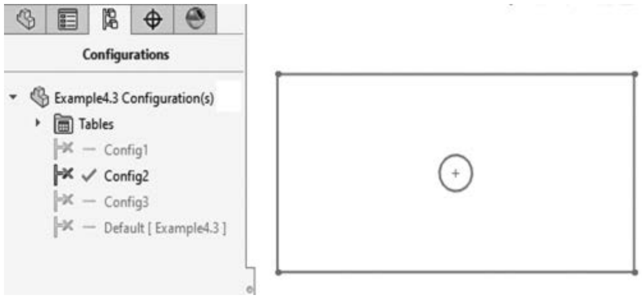


Step 3: Create three other configurations: Step 2 opens the Excel sheet shown > copy the *Default* config row and paste three times > edit the name cell and change configuration names as shown > edit the *D2* column and change as shown > click anywhere on screen away from Excel sheet to make it disappear > **OK** to accept the popup window indicating that new configurations were created.

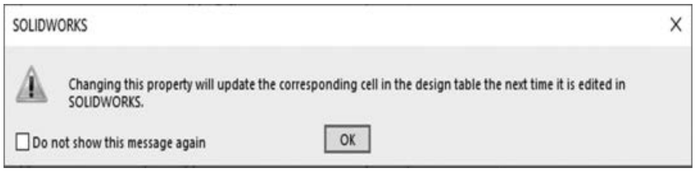
	A	B	C	D	E	F
1	Design Table for: Example2.4					
2		D1@Sketch1	D2@Sketch1	D3@Sketch1	D4@Sketch1	D5@Sketch1
3	Default	=2 * "D2@Sketch1"	1	0.55	=.5 * "D2@Sketch1"	=.5 * "D2@Sketch1"
4	Config1	=2 * "D2@Sketch1"	2	0.55	=.5 * "D2@Sketch1"	=.5 * "D2@Sketch1"
5	Config2	=2 * "D2@Sketch1"	3	0.55	=.5 * "D2@Sketch1"	=.5 * "D2@Sketch1"
6	Config3	=2 * "D2@Sketch1"	4	0.55	=.5 * "D2@Sketch1"	=.5 * "D2@Sketch1"

Sheet1

Step 4: Review the design configurations: Step 3 creates a tree with four nodes, as shown, under the **ConfigurationManager** tab > double-click any configuration to display the corresponding sketch > the current configuration is displayed in dark black in the tree.



Step 5: Investigate effect of design table on sketch parameters: Click feature tree > select *Sketch1* > observe that dimensions are displayed in a pink color and each time you click or move one, you get a window (shown to the right) warning you.



HANDS-ON FOR EXAMPLE 4.3

Edit the sketch and change the value of *D2* to 6. What happens to the design table? Explain the result.

4.6 Macros

Design automation offers two benefits. First, it enhances productivity. Second, it helps with repetitive tasks that are mundane. For example, if you follow the same design process over and over, automating it would be the logical thing to do.

Macros aid in design automation. Macros are also viewed as a way to customize your CAD/CAM system. A **macro** is a short computer program that is used to repeat commonly performed operations. That computer program is generated automatically by a CAD/CAM system in the background during use of the system, from the time you turn on the macro until you turn it off. You can use the macro after creation over and over, with different input values (e.g., new dimensions). After you create a macro, you can use it for recording, edit it, run it, pause it, stop it, and assign it to a shortcut key (hotkey) or to a menu item. When you assign a macro to a shortcut key or to a menu item, you can specify which method (function) of the macro to run. Click **Tools** > **Macro** to access the macro menu. SolidWorks saves the macro file in the same part folder and uses *.swp* as the file extension, with the default names *Macro1.swp*, *Macro2.swp*, and so on, unless you change them.

A higher level of automation than using macros is to use Visual Basic (VB) or another programming language to perform full automation and have better control of the automation. As a matter of fact, VB is the programming engine behind macros. Instead of writing the macro VB code, the SolidWorks macro interface enables us to generate the code automatically while you perform the design tasks as usual. It is this VB code that you save in a file when you save the macro. You can use the VB editor to edit and tweak the VB code.

Macro VB code uses the SolidWorks API (application programming interface). The code makes calls to API functions. Think of the API as a gateway between the application you want to write and SolidWorks code that has been already written. In other words, the API provides access to the SolidWorks geometric engine.

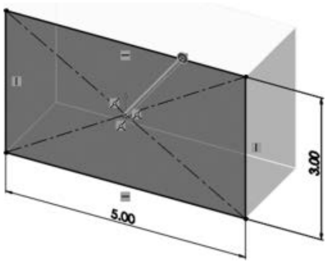
You can learn VB programming by creating multiple macros, studying their generated VB code, and expanding on it. VB is an object-oriented programming (OOP) language that requires knowledge and understanding of object-oriented design and how objects are defined and implemented.

The programming approach could be useful for creating an entire assembly from a few parameters. You can write a program to define some variables and store them in a row in a design table. Each row represents a new version of the assembly. You delete all rows from the table and keep the last row, which is the new assembly.

Example 4.4 Develop a macro to create an extrusion.

Solution This example illustrates the creation and use of macros. It shows how to create a block extrusion and record the steps of creating it in a macro.

Step 1: Turn on macro and create *Sketch1* and *Boss-Extrude1* feature: **File** > **New** > **Part** > **OK** > **Tools** > **Macro** > **Record** > **Front Plane** > **Extruded Boss/Base** on **Features** tab > **Sketch** tab > **Center Rectangle** on **Sketch** tab > sketch a rectangle and dimension as shown > exit sketch > enter 2 for **D1** > reverse extrusion direction > ✓ > **File** > **Save As** > *example4.4* > **Save**.



Note: **Macro** > **Record** opens the **Macro** recording window shown. Hover over its buttons to read them.

Step 2: Stop and save macro: **Tools** > **Macro** > **Stop** (or click black square on **Macro** window shown in Step 1) > type *BlockShaft* for macro name in the **Save As** window that opens > **Save**.

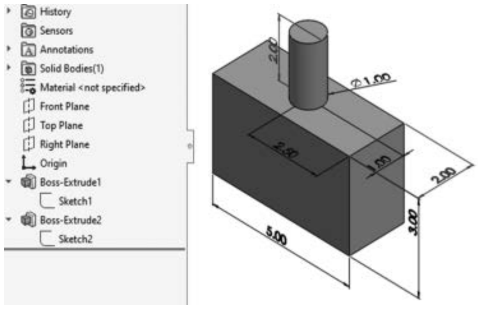
Note: The full macro file name is *BlockShaft.swp*. The default folder for the file is the part file folder from Step 1.

Step 3: View the macro VB code: **Tools** > **Macro** > **Edit** > select *BlockShaft.swp* > **Open**. The editor displays all the lines of code associated with the macro. Figure 4.9 shows the first lines of the macro VB code. Explore the code and try to understand which sections pertain to which steps performed in SolidWorks.

Step 4: Run the macro: Open a new part or delete the feature from the current part > **Tools** > **Macro** > **Run** > select *BlockShaft.swp* > **Open** > enter 8 for rectangle width when dimension box appears > ✓ > enter 6 for rectangle height when dimension box appears > ✓.

HANDS-ON FOR EXAMPLE 4.4

Modify the part as shown to the right and re-create the macro. Rerun the macro twice to investigate locating the shaft to the left or to the right side from the center. This simulates the idea of using a macro to investigate "what if" design scenarios such as sizes or feature locations.



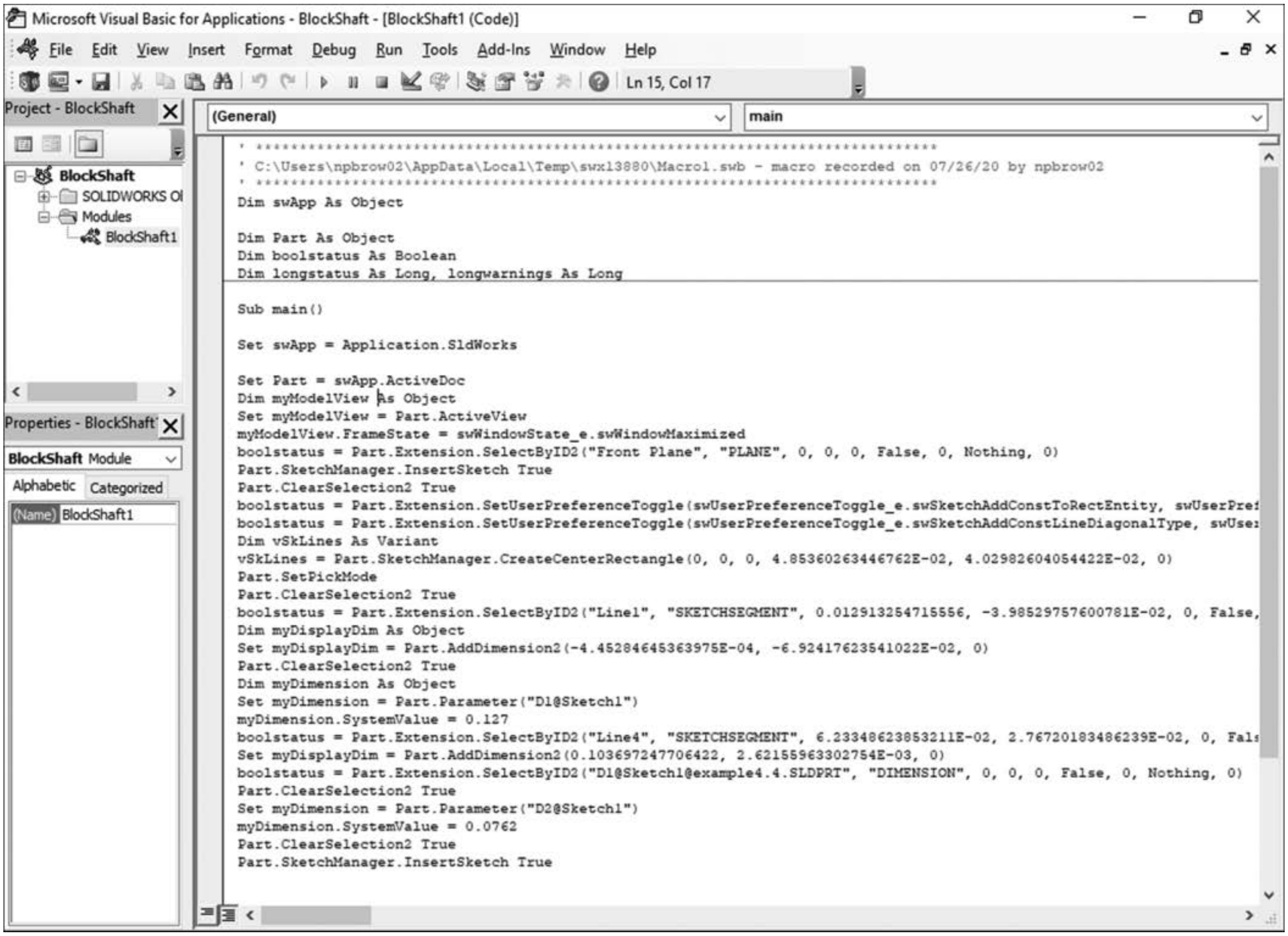


Figure 4.9
VB editor window

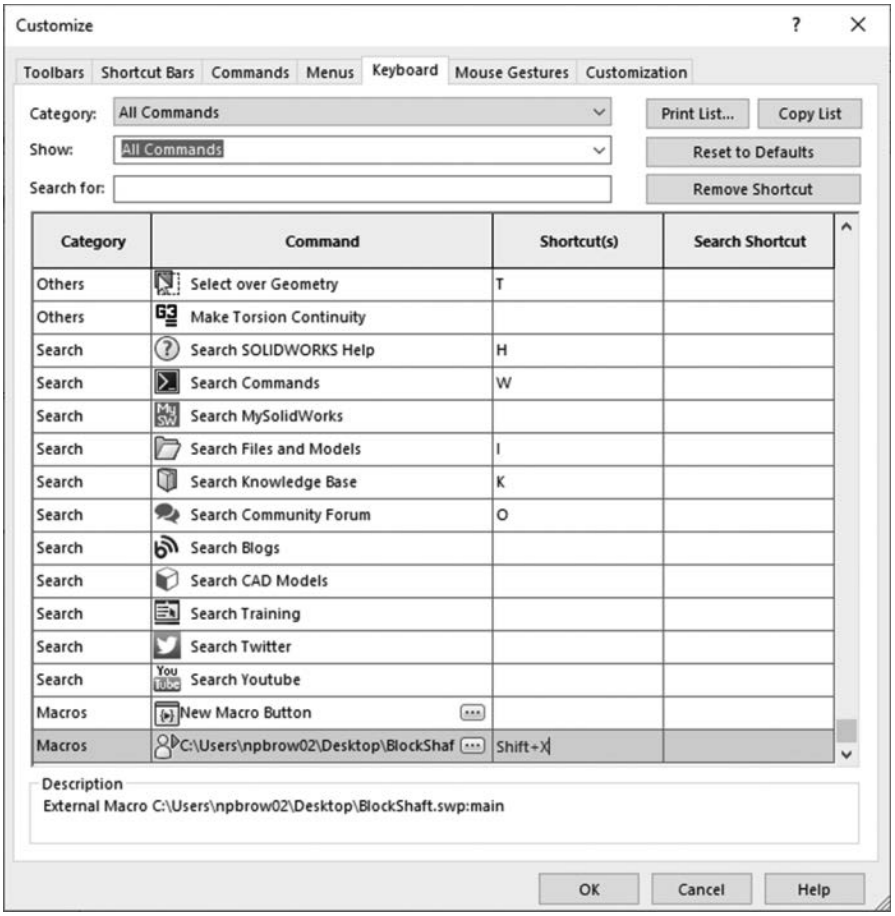
Example 4.5 Create a hotkey for a macro.

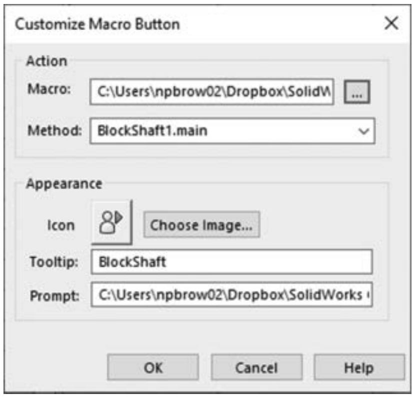
Solution This example assigns the macro from Example 4.4 to the **Shift+X** hotkey on the keyboard. When the user presses this combination on the keyboard, the macro runs.

Step 1: Create macro hotkey: **File** > **Open** > locate *example4.4* > **Open**.

Step 3: Create hotkey: Click **Shortcut(s)** column in **Macros** row as shown > **Shift+X** to add shortcut as shown below > **OK**.

Step 2: Locate macro file: **Tools** > **Customize** > **Keyboard** tab > scroll to bottom and locate **Macros** row > click ellipsis button (…) in this row to open window shown > click the browse button (…) > locate macro file (*.swp* extension) > **Open** > **OK**.





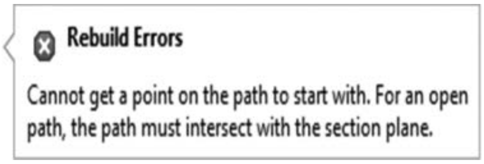
Step 4: Use hotkey to run macro: Press **Shift+X** in an open part.

4.7 Tutorials

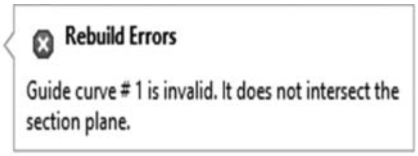
Tutorial 4–1 Create Sweep Features

A sweep feature requires, at minimum, a profile (cross section) to sweep and a path (curve) to sweep along. You can use a guide curve to control the sweep further. If you do not use a guide curve, the sweep cross section stays constant.

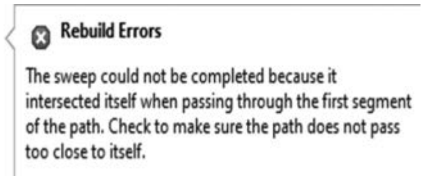
Sweep operations may fail for different reasons. Figure 4.10 shows three error messages. As a general rule, the sweep path and guide must intersect the cross-section plane, and the cross section must not intersect itself as it traverses the path and/or the guide curve.



(A) Path does not intersect cross-section plane



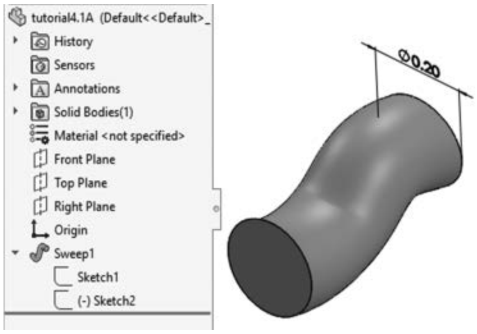
(B) Guide curve does not intersect cross-section plane



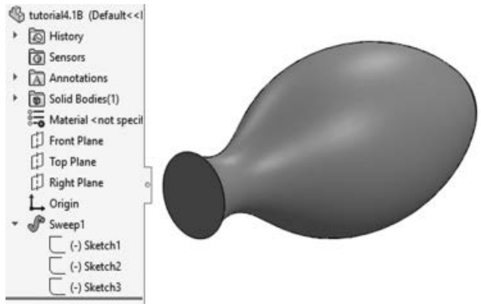
(C) Cross section intersects itself

Figure 4.10
Some possible sweep operation errors

Create the sweep features shown in Figure 4.11. All dimensions are in inches.



(A) No guide curve

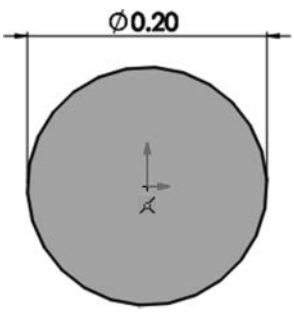


(B) With guide curve

Figure 4.11
Sweep features

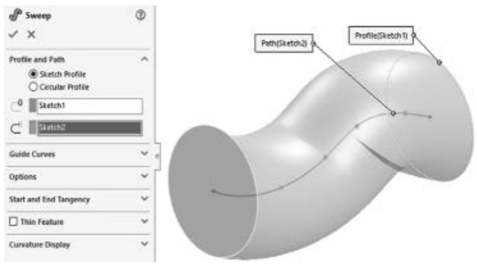
No-guide-curve sweep (Figure 4.11A) modeling steps:

Step 1: Create *Sketch1-Profile*: **File** > **New** > **Part** > **OK** > **Front Plane** > **Circle** on **Sketch** tab > click origin and sketch circle and dimension as shown > exit sketch > **File** > **Save As** > *tutorial4.1A* > **Save**.



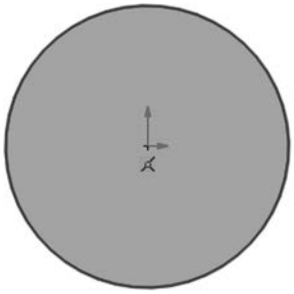
Step 2: Create *Sketch2-Path*: **Right Plane** > **Spline** on **Sketch** tab > sketch spline as shown (press **Esc** after last point is clicked to exit **Spline**); make sure spline snaps to origin > exit sketch.

Step 3: Create *Sweep1* feature: **Sweep Boss/Base** on **Features** tab > select circle sketch as **Profile**, as shown to the right > select spline sketch as **Path**, as shown to the right > ✓.

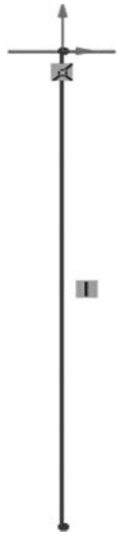


With-guide-curve sweep (Figure 4.11B) modeling steps:

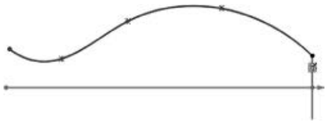
Step 1: Create *Sketch1-Profile*: **File** > **New** > **Part** > **OK** > **Front Plane** > **Circle** on **Sketch** tab > click origin and sketch circle as shown > exit sketch > **File** > **Save As** > *tutorial4.1B* > **Save**.



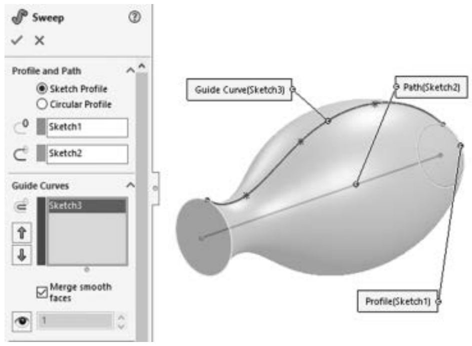
Step 2: Create *Sketch2-Path*: **Top Plane** > **Line** on **Sketch** tab > sketch line as shown (to the right) from origin > exit sketch.



Step 3: Create *Sketch3-Guide*: **Right Plane** > **Spline** on **Sketch** tab > sketch spline as shown (press **Esc** after last point is clicked to exit **Spline**); select spline endpoint near circle sketch > **Ctrl** + select circle sketch > **Make Pierce** > exit sketch.



Step 4: Create *Sweep* feature: **Sweep Boss/Base** on **Features** tab > select circle sketch as **Profile**, as shown > select Step 2 spline sketch as **Path** > select Step 3 spline sketch as **Guide** > ✓.



HANDS-ON FOR TUTORIAL 4–1

Change the cross sections of both sweeps to a 2 × 2-inch square. What happens? Resolve the error(s) and explain your solution.

Tutorial 4–2 Create Loft Features

Create the wine glass shown in Figure 4.12. All dimensions are in inches. We introduce the concept of **Convert Entities** on the **Sketch** tab in this tutorial. You copy one circle in one sketch to another sketch. This concept enables you to copy entities from one sketch to another. While you could easily create a new circle and dimension it, using the **Convert Entities** method is faster (as there is no need to sketch a circle and dimension it). SolidWorks creates an **On Edge** relation between the two circles and shows a small green cube on the copied entity to indicate the relation. When you click the copied circle while editing the sketch, SolidWorks displays the **On Edge** relation in the relations pane to the left of the screen.

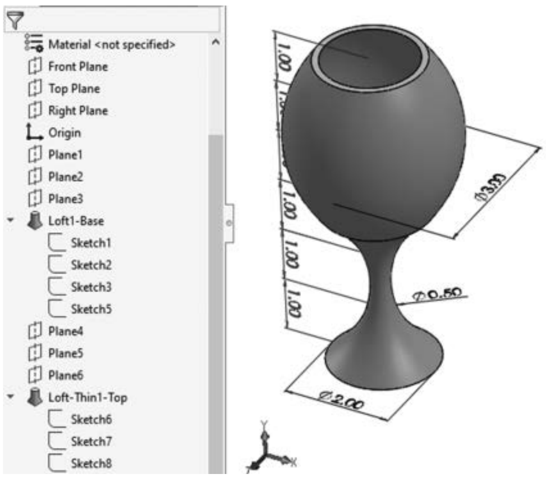
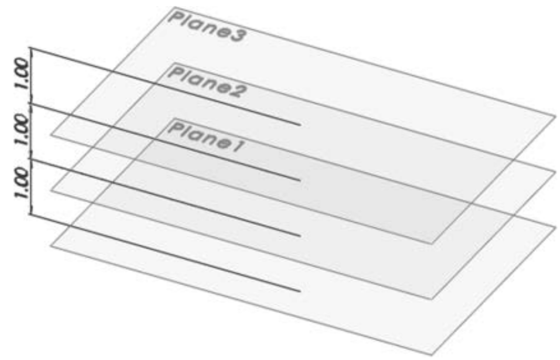
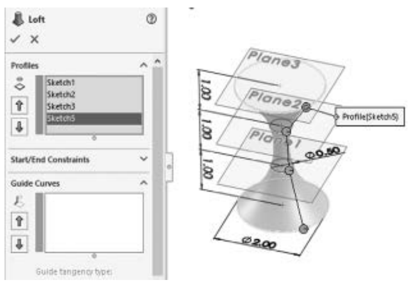


Figure 4.12
Loft feature

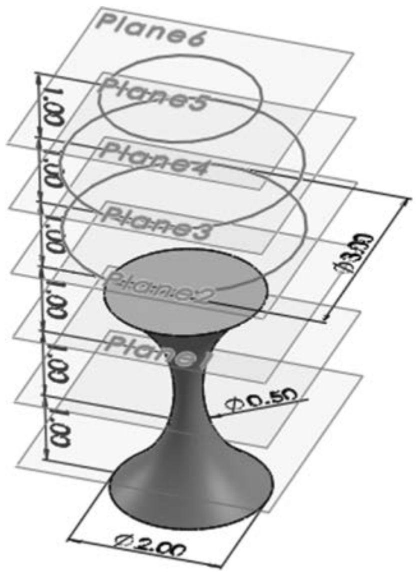
Step 1: Create *Plane1–Plane3*: **File** > **New** > **Part** > **OK** > **Reference Geometry** on **Features** tab > **Plane** > expand feature tree and select **Top Plane** > enter 1.0 for **Offset Distance** > ✓ > repeat for *Plane2* and *Plane3* but select the previously created plane > **File** > **Save As** > *tutorial4.2* > **Save**.



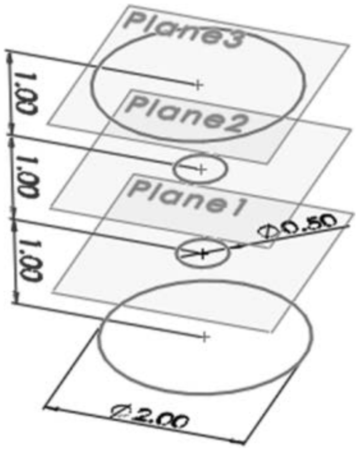
Step 3: Create *Loft-Base* feature: **Lofted Boss/Base** on **Features** tab > select *Sketch1–Sketch4* > ✓.



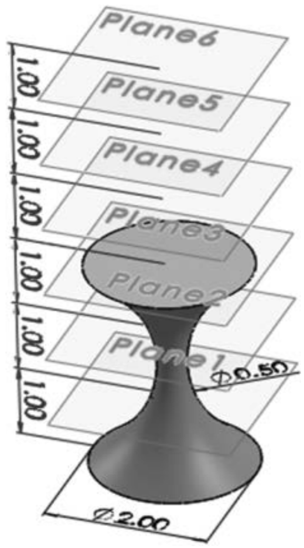
Step 5: Create *Sketch5–Sketch7*: Select *Plane4* as **Sketch Plane** > **Circle** on **Sketch** tab > click origin and drag to sketch and dimension a 3.0-inch diameter circle > exit sketch > select *Plane5* as **Sketch Plane** > **Sketch** on **Sketch** tab > **Convert Entities** on **Sketch** tab > click *Sketch5* (just created) > ✓ > ✓> exit sketch > select *Plane6* as sketch plane > **Sketch** on **Sketch** tab > **Convert Entities** on **Sketch** tab > click circle on *Plane3* > ✓ > ✓> exit sketch.



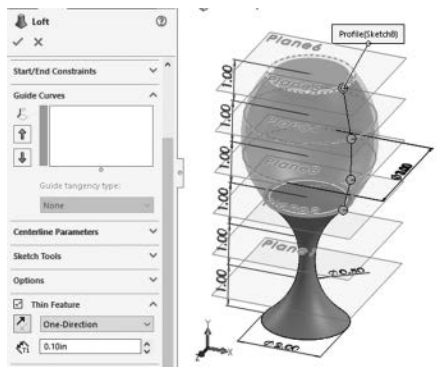
Step 2: Create *Sketch1–Sketch4*: **Top Plane** > **Circle** on **Sketch** tab > click origin and drag to sketch and dimension a 2.0-inch diameter circle > exit sketch > select *Plane1* as sketch plane and create a 0.5-inch diameter circle > exit sketch > select *Plane2* as sketch plane > **Sketch** on **Sketch** tab > **Convert Entities** on **Sketch** tab > click circle on *Plane1* > ✓ > ✓ > exit sketch > select *Plane3* as sketch plane > **Sketch** on **Sketch** tab > **Convert Entities** on **Sketch** tab > click circle on **Top Plane** > ✓ > ✓> exit sketch.



Step 4: Create *Plane4–Plane6*: **Reference Geometry** on **Features** tab > **Plane** > expand feature tree and select *Plane3* > enter 1.0 for distance > ✓ > repeat for *Plane5* and *Plane6* but select the previously created plane.



Step 6: Create *Loft-Thin1-Top* feature: **Lofted Boss/Base** on **Features** tab > select the circle of *Sketch4* and then select *Sketch5–Sketch7* > check **Thin Feature** box > enter 0.1 for thickness (**T1**) > if needed, click direction box to toggle direction of thickness to the inside > ✓.

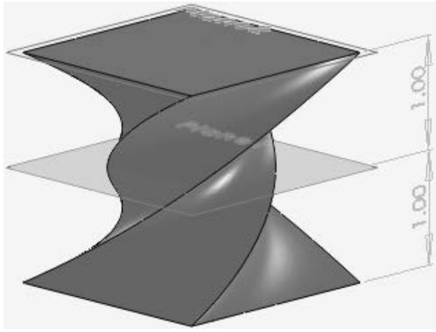


Note: Make sure you select the circle and sketches in a way so that the interpolation points (green circles shown here) line up; otherwise, you twist the loft.

Note: The thickness of the thin feature has a direction: inside or outside the profile. Reverse the direction of the double arrows shown to toggle.

HANDS-ON FOR TUTORIAL 4–2

Create a loft using three squares of different sizes as cross sections separated by 1 inch. The square sizes are 2×2, 1×1, and 2×2, respectively. Create the loft connecting the three sections such that the loft is twisted as shown.



Tutorial 4–3 Use the Hole Wizard

The hole wizard provides two advantages. First, it helps create standard hole sizes and types so that off-the-shelf bolts will fit perfectly in the holes. Second, it greatly speeds up the process of creating these holes. This tutorial shows how to create sample holes: counterbore, countersink, and tapped holes. Figure 4.13 shows these holes. You create an extrusion and add holes to it. You also create *Sketch2* on the top face of the *Block* feature with a center rectangle (construction rectangle) that you use to place holes at its corners.

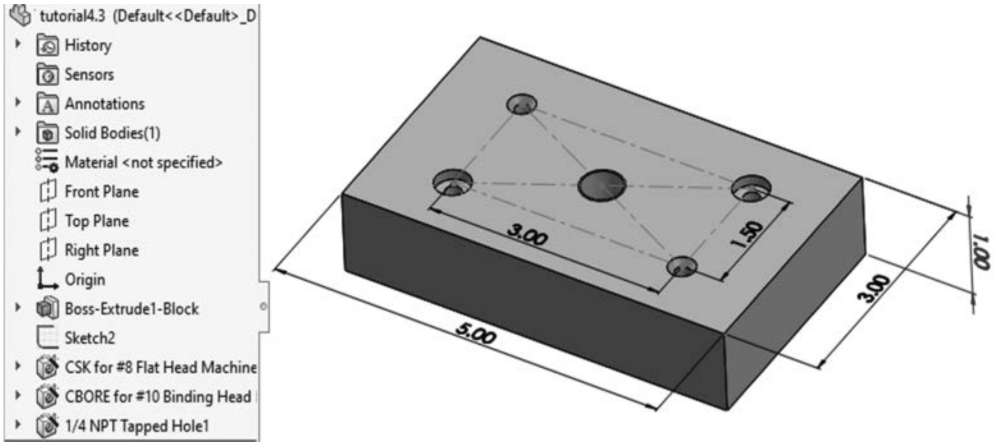
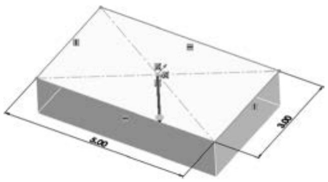


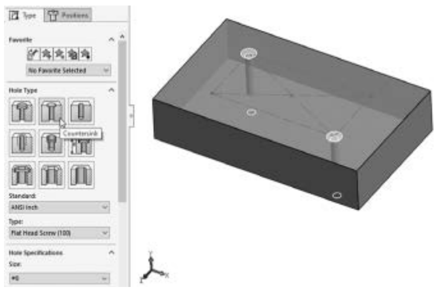
Figure 4.13
Wizard holes

Step 1: Create *Sketch1* and *Block* feature: **File** > **New** > **Part** > **OK** > **Top Plane** > **Extruded Boss/Base** on **Features** tab > **Center Rectangle** on **Sketch** tab > click origin and sketch and dimension rectangle as shown > exit sketch > enter 1 for **D1** > reverse extrusion direction > **File** > **Save As** > *tutorial4.3* > **Save**.

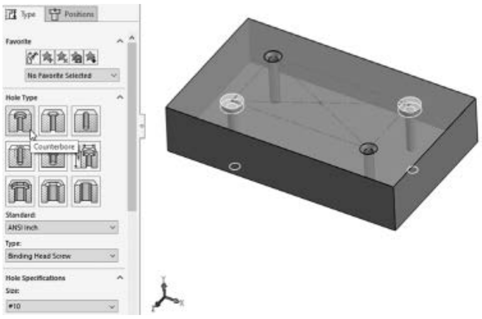


Step 2: Create *Sketch2*: Top face of *Block* > **Center Rectangle** on **Sketch** tab > click origin and sketch a 3.0×1.5 rectangle, as shown in Figure 4.13 > click **For Construction** box > exit sketch.

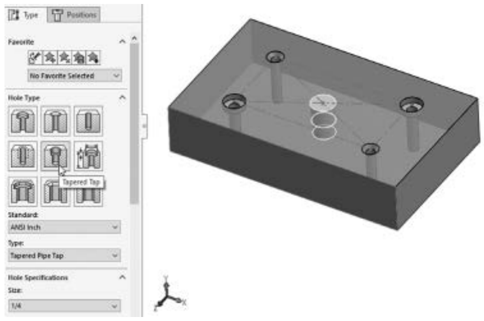
Step 3: Create two diagonal countersink hole features (*CSK…* node in feature tree): **Hole Wizard** on **Features** tab > select **Countersink** under **Hole Type** (hover over types until you read it) > select #8 for **Size** under **Hole Specifications** > **Positions** tab > click top face of *Block* and then click two corners of construction rectangles as shown > ✓.



Step 4: Create two diagonal counterbore hole features (*CBORE…* node in feature tree): **Hole Wizard** on **Features** tab > select **Counterbore** under **Hole Type** (hover over types until you read it) > select #10 for **Size** under **Hole Specifications** > **Positions** tab > click top face of *Block* and then click two other corners of construction rectangles as shown > ✓.



Step 5: Create *Tapped Hole1* feature: **Hole Wizard** on **Features** tab > select **Tapered Tap** (hover over types until you read it) > select 1/4 for **Size** under **Hole Specifications** > **Positions** tab > click top face of *Block* and then click rectangle center as shown > ✓.



HANDS-ON FOR TUTORIAL 4–3

Create a 1/16 tapered pipe tap through all holes, located 0.5 from the top edge and 1.5 from the left edge of the block.

Tutorial 4–4 Create Compression Spring

Figure 4.14 shows the constant length compression spring you create in this tutorial.

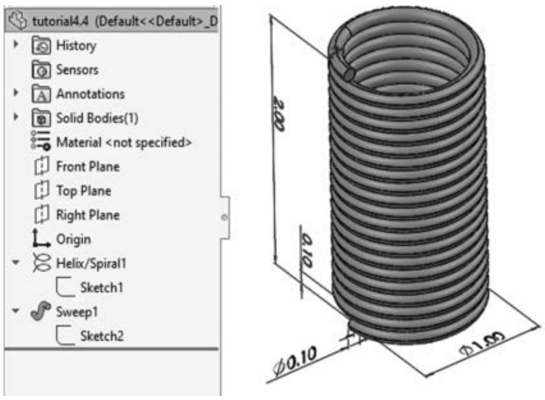
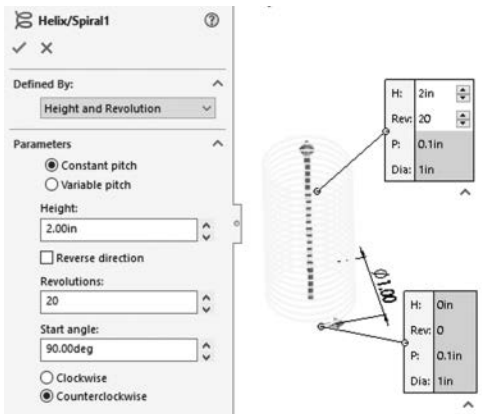
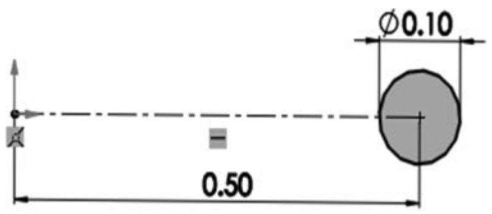


Figure 4.14
Compression spring

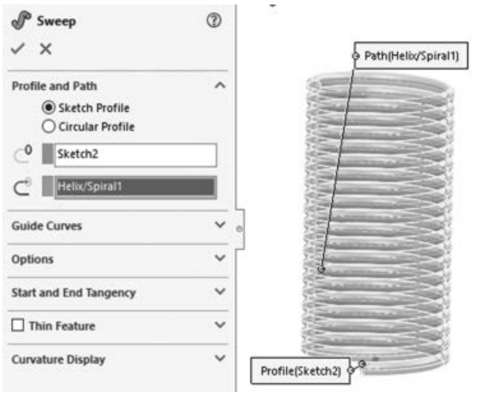
Step 1: Create *Sketch1* and *Helix/Spiral1* curve: **File** > **New** > **Part** > **OK** > **Top Plane** > **Circle** on **Sketch** tab > click origin and sketch and dimension circle with 1.0-inch diameter > exit sketch > **Insert** (menu) > **Curve** > **Helix/Spiral** > select the circle just sketched > **Height and Revolution** from **Defined By** dropdown shown > **Constant Pitch** > enter 2.0 for **Height**, 20 for **Revolutions**, and 90 for **Start Angle**, as shown > ✓ > **File** > **Save As** > *tutorial4.4* > **Save**.



Step 2: Create *Sketch2*: **Front Plane** > **Circle** on **Sketch** tab > sketch and dimension circle as shown (align center with X-axis as shown) > exit sketch.



Step 3: Create *Sweep1* feature (spring): **Swept Boss/Base** on **Features** tab > select *Sketch2* as **Profile** > select *Helix/Spiral1* as **Path** > ✓.



HANDS-ON FOR TUTORIAL 4–4

Edit the spring helix to have a variable pitch. Use a pitch of 0.2 at midheight point.

Tutorial 4–5 Create Spiral

Figure 4.15 shows the spiral spring you create in this tutorial.

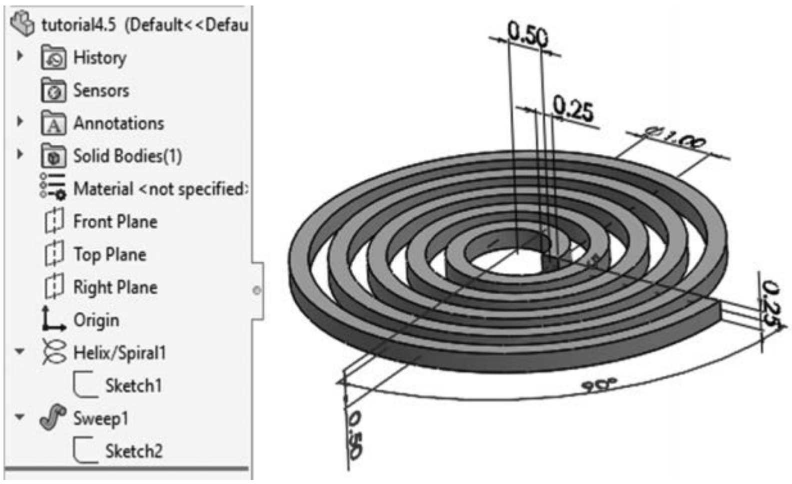
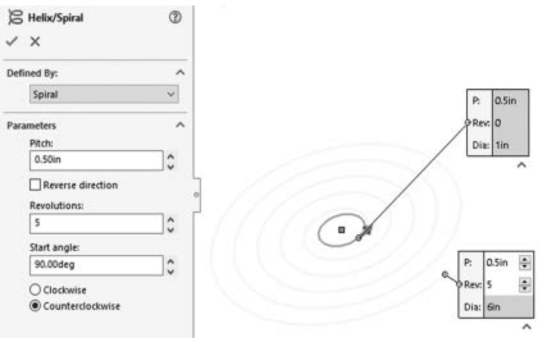
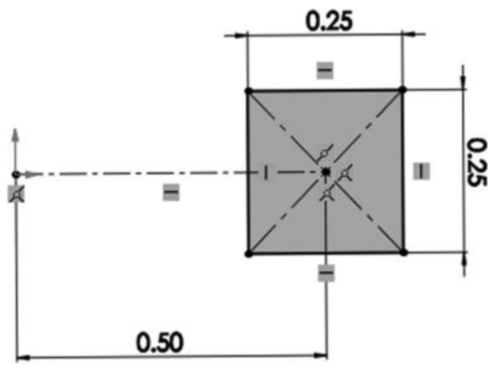


Figure 4.15
Spiral spring

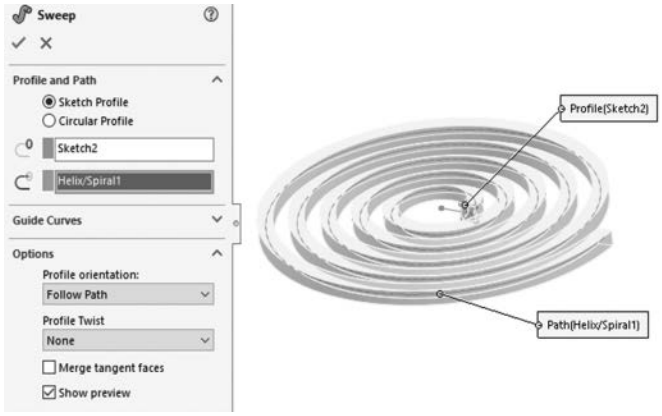
Step 1: Create *Sketch1* and *Helix/Spiral* curve: **File** > **New** > **Part** > **OK** > **Top Plane** > **Circle** on **Sketch** tab > click origin and sketch and dimension circle with 1.0-inch diameter > exit sketch > **Insert** (menu) > **Curve** > **Helix/Spiral** > select the circle just created > **Spiral** from **Defined By** dropdown shown > enter 0.5 for **Pitch** and 5 for **Revolutions**, as shown > **Start Angle** of 90 and select **Counterclockwise** > ✓ > **File** > **Save As** > *tutorial4.5* > **Save**.



Step 2: Create *Sketch2*: **Front Plane** > **Center Rectangle** on **Sketch** tab > sketch and dimension rectangle as shown (align center with X-axis as shown) > exit sketch.



Step 3: Create *Sweep* feature (spiral): **Swept Boss/Base** on **Features** tab > select *Sketch2* as **Profile** > select *Helix/Spiral* as **Path** > select **Show Preview** under **Options** > ✓.



HANDS-ON FOR TUTORIAL 4–5

Change the spiral cross section to a circle with 2.0-inch diameter. Can you generate the spiral? Why or why not? Explain.

Tutorial 4–6 Create Features

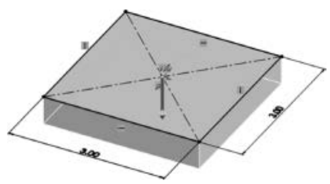
This tutorial covers the creation of these features: chamfer, fillet, slot, shell, draft, and rib. All dimensions are in inches. Consider these useful observations:

1 Make sure to pay attention to the visual clues shown in the left pane while creating these features.

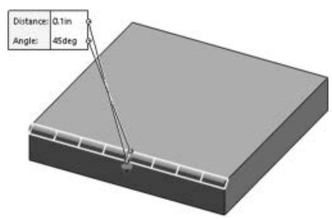
2 For example, the box symbol under **Chamfer parameters** indicates that you can chamfer a face, an edge, or a vertex (corner point). As you would expect, chamfering a face chamfers all its edges. Chamfering a corner chamfers the three edges that meet there.

3 A rib requires a profile sketch (e.g., a line or stepwise line) and a thickness.

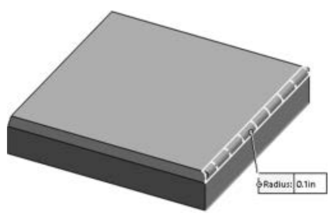
Step 1: Create *Sketch1* and *Block* feature: **File** > **New** > **Part** > **OK** > **Top Plane** > **Extruded Boss/Base** on **Features** tab > **Center Rectangle** on **Sketch** tab > click origin and sketch and dimension as shown > exit sketch > reverse extrusion direction > enter 0.5 for **D1** > ✓ > **File** > **Save As** > *tutorial4.6* > **Save**.



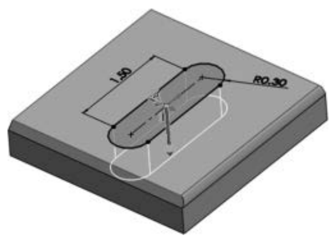
Step 2: Chamfer an edge of *Block* feature: **Fillet** dropdown on **Features** tab > **Chamfer** > select **Angle Distance** chamfer > select *Block* edge shown > use 0.1 for **Distance** and 45 degrees for **Angle** > ✓.



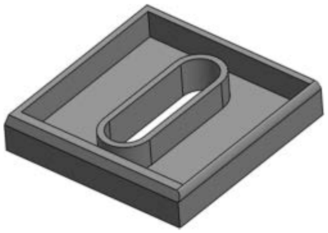
Step 3: Fillet an edge of *Block* feature: **Fillet** on **Features** tab > select **Constant Size Fillet** > select *Block* edge shown > use 0.1 for **Radius** > ✓.



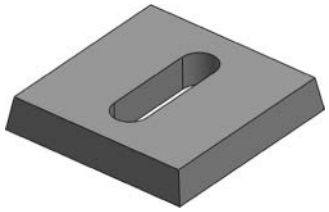
Step 4: Create a straight slot in *Block* feature: Select *Block* top face as a sketch plane > **Extruded Cut** on **Features** tab > **Straight Slot** on **Sketch** tab > sketch and dimension slot as shown > make origin and slot midpoint **Coincident** > exit sketch > **Through All** > ✓.



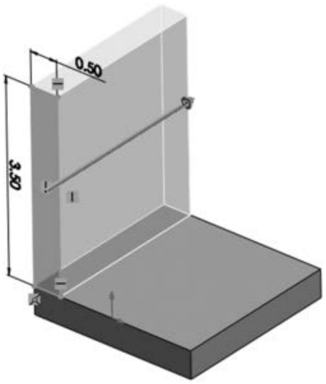
Step 5: Shell *Block* feature: **Shell** on **Features** tab > select top face of *Block* > enter 0.1 for **D1** > ✓.

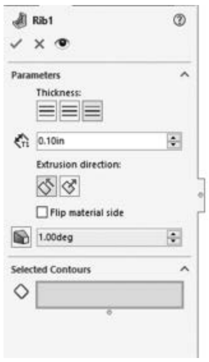


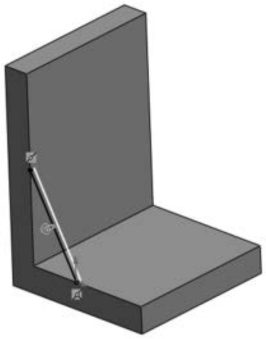
Step 6: Draft *Block* feature: Suppress the chamfer, fillet, and shell features > **Draft** on **Features** tab > enter 10 degrees for **Draft Angle** > select top face of *Block* as **Neutral Plane** > select *Block* four side faces to draft > ✓.

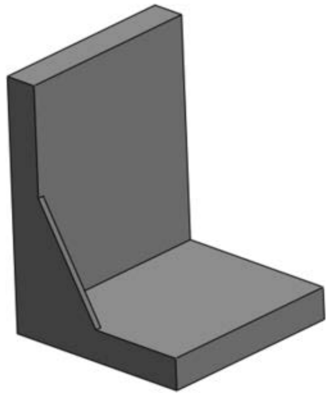


Step 7: Create a rib feature: Suppress the slot and draft features > select front face of *Block* > **Extruded Boss/Base** on **Features** tab > **Rectangle** on **Sketch** tab > sketch and dimension rectangle as shown below > exit sketch > reverse extrusion direction > enter 3.0 for **D1** > ✓ > **Front Plane** > **Rib** on **Features** tab > **Line** on **Sketch** tab > sketch a line using the midpoints of the two edges as shown below > exit sketch > enter 0.5 for rib thickness (**T1**) > select **Second Side** for **Thickness** > **Parallel to Sketch** for **Extrusion Direction** > ✓.









> **HANDS-ON FOR TUTORIAL 4–6**
>
> Create the following features:
>
> - Distance-distance chamfer
> - Vertex chamfer
> - Variable-size fillet
> - Face fillet
> - Full round fillet
> - 3-point arc slot
> - Three-stepped rib using three-stepped line as the rib profile

Tutorial 4–7 Use the Smart Fasteners Wizard

You use the Smart Fasteners wizard to insert the correct fastener based on the hole you select. It is a form of automation using off-the-shelf components. SolidWorks has its own standard library of fasteners. You can only use the wizard at the assembly level. You need to activate the wizard in order for the **Smart Fasteners** icon on the **Assembly** tab to work. If you click the icon before activation, you get this error: **Smart Fasteners requires SolidWorks Toolbox, which is not present.**

In this tutorial, you create an assembly of a block and plate. You create a counterbore hole in the plate and a blind hole in the block, assemble them, and fasten them together with a smart fastener. Figure 4.16 shows the assembly and its tree. All dimensions are in inches.

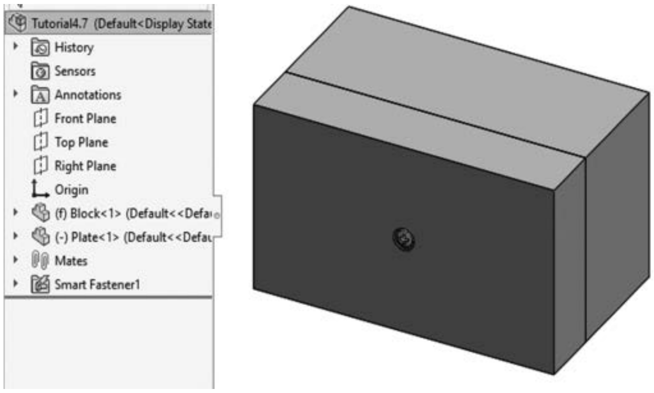
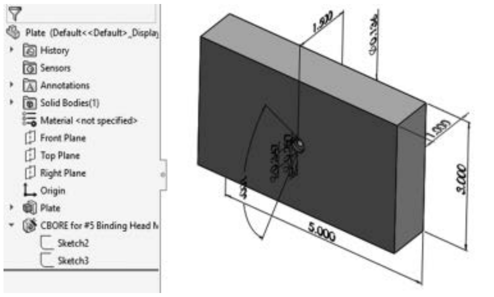


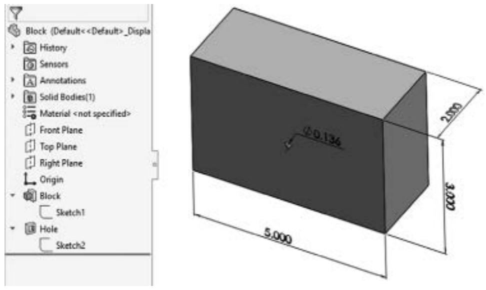
Figure 4.16
Assembly using smart fastener

Step 1: Create *Plate* feature: **File** > **New** > **Part** > **OK** > **Front Plane** > **Extruded Boss/Base** on **Features** tab > **Center Rectangle** on **Sketch** tab > click origin and sketch and dimension rectangle as shown > exit sketch > enter 1 for **D1** > reverse extrusion direction > ✓ > **Hole Wizard** on **Features** tab > select **Counterbore** under **Hole Type** (hover over each hole type until you read the correct type) > ANSI inch for **Standard** > binding head screw for **Type** > select #5 for **Size** under **Hole Specifications** > **Positions** tab > click front face of *Plate* and then click origin > ✓ > **File** > **Save As** > *Plate* > **Save**.

Note: The diameter of the counterbore hole shown below corresponds to #5 size. You need it to create the corresponding hole in the block in Step 2.



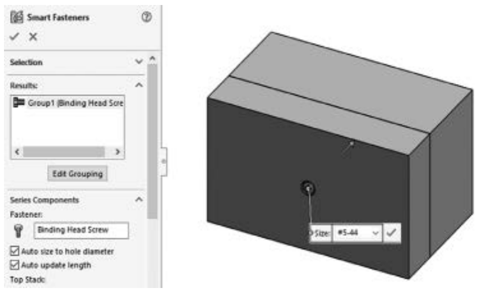
Step 2: Create *Block* feature: **File** > **New** > **Part** > **OK** > **Front Plane** > **Extruded Boss/Base** on **Features** tab > **Center Rectangle** on **Sketch** tab > click origin and sketch and dimension rectangle as shown > exit sketch > enter 2 for **D1** > reverse extrusion direction > ✓ > front face of *Block* > **Extruded Cut** on **Features** tab > **Circle** on **Sketch** tab > click origin and sketch and dimension as shown > exit sketch > enter 1 for **D1** > ✓ > **File** > **Save As** > *Block* > **Save**.



Step 3: Create assembly: **File** > **New** > **Assembly** > **OK** > **Browse** > locate *Block* and *Plate* parts > select **Block + Ctrl + Plate** > **Open** > click ✓ to place *Block* instance in assembly origin > click anywhere in graphics pane to place *Plate* instance > **Mate** on **Assembly** tab > **Coincident** > select the corresponding top edges of *Block* and *Plate* > ✓ > select the corresponding right edges of *Block* and *Plate* > ✓ > ✓.

Step 4: Activate Smart Fasteners wizard: **Tools** > **Add-Ins** > **SolidWorks Toolbox Library** > **OK**. This adds the **Toolbox** menu to the menu bar to the right of the **Tools** menu. You may deactivate the **Toolbox** by using the same sequence but unchecking the **Toolbox Browser** from the **Add-Ins** window.

Step 5: Add a fastener: **Smart Fasteners** on **Assembly** tab > **OK** (to accept that it may take extra time) > expand feature tree > expand *Plate* instance tree node > select *CBORE for #5* node > **Add** > ✓ > **File** > **Save As** > *Tutorial4.7* > **Save**.



HANDS-ON FOR TUTORIAL 4–7

Modify *Block* and *Plate* parts to create four corner countersink holes. Re-create the assembly and use four smart fasteners.

Tutorial 4–8 Create a Bolt

Bolts, like gears, are important and essential mechanical elements. While bolts are standard off-the-shelf components, this tutorial shows how to create the CAD model of one due to its learning value. Figure 4.17 shows the bolt and its feature tree. All dimensions are in inches.

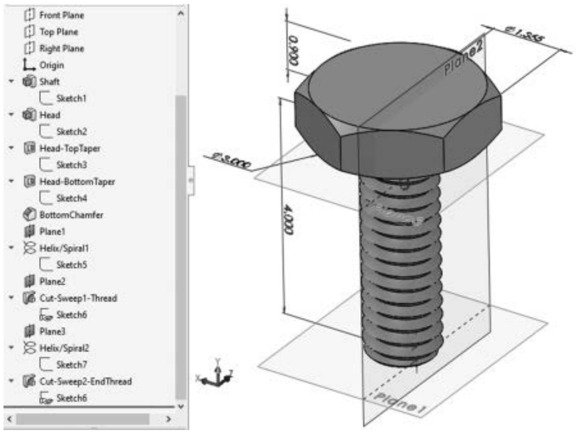
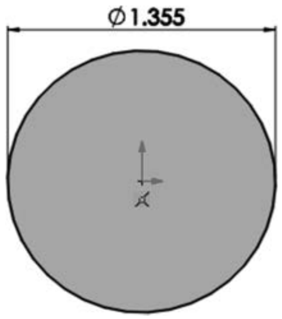
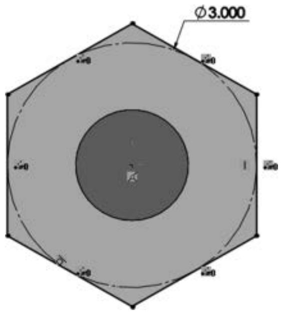


Figure 4.17
A bolt

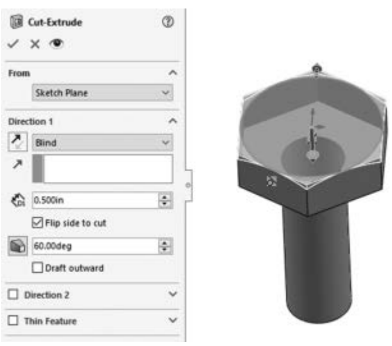
Step 1: Create *Sketch1* and *Shaft* feature: **File** > **New** > **Part** > **OK** > **Top Plane** > **Extruded Boss/Base** on **Features** tab > **Circle** on **Sketch** tab > click origin and sketch and dimension as shown > exit sketch > enter 4 for **D1** > reverse extrusion direction > ✓ > **File** > **Save As** > *Bolt* > **Save**.



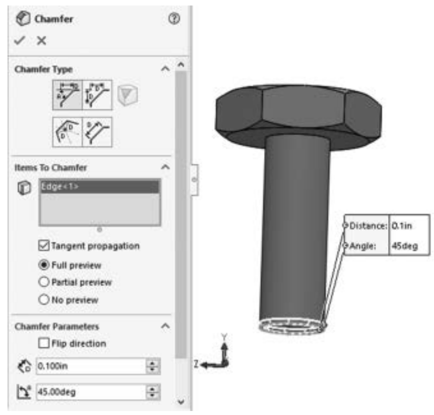
Step 2: Create *Sketch2* and *Head* feature: Select top face of *Shaft* feature > **Extruded Boss/Base** on **Features** tab > **Polygon** (hexagon icon) on **Sketch** tab > click origin and sketch and dimension as shown > apply vertical relation to one edge > exit sketch > enter 0.9 for **D1** > ✓.



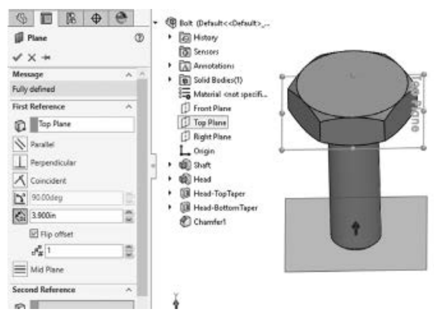
Step 3: Create *Sketch3* and *Head-TopTaper* feature: Select top face of *Head* feature > **Extruded Cut** > **Circle** on **Sketch** tab > click origin and sketch (make circle tangent to hexagon sides) > exit sketch > enter 0.5 for **D1** > click checkbox as shown > enter 60 for draft angle > ✓ > repeat to create *Head-BottomTaper* to chamfer the bottom of the head.



Step 4: Create *BottomChamfer* feature: **Fillet** dropdown on **Features** tab > **Chamfer** > select bottom edge of *Shaft* feature > enter 0.1 for **D** > enter 45 for **A** > ✓.

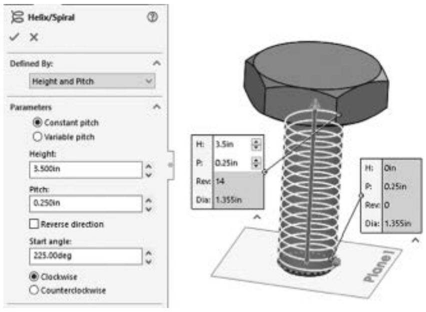


Step 5: Create *Plane1*: **Reference Geometry** on **Features** tab > **Plane** > expand feature tree > select **Top Plane** > enter 3.9 for **D1** > click **Flip offset** checkbox > ✓.



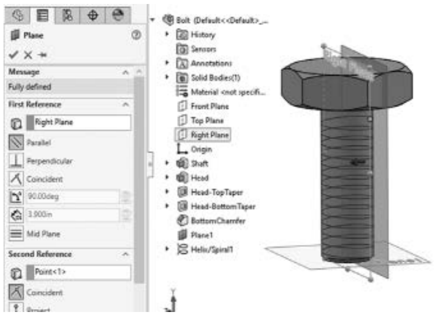
Note: You use a distance of 3.9 for *Plane1*, not 4.0, because the chamfer is 0.1 high.

Step 6: Create *Sketch5* and *Helix/Spiral1*: Select *Plane1* > select **Sketch** on **Sketch** tab > **Convert Entities** on **Sketch** tab > expand feature tree > select *Sketch1* > ✓ > exit Sketch > select *Sketch5* > **Insert** > **Curve** > **Helix/Spiral** > select **Height and Pitch** > enter 3.5 for **Height**, 0.25 for **Pitch**, and 225 for **Start angle** > ✓.

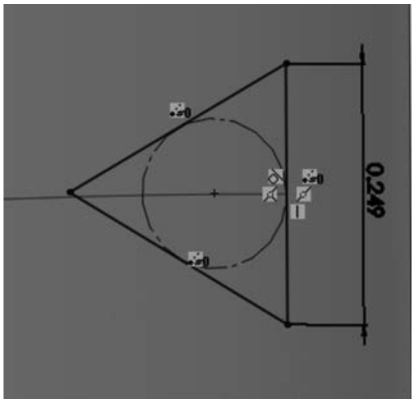


Note: The helix 3.5 height is arbitrary. That leaves 0.4 (out of 3.9). You use 0.25 for the other end thread helix and 0.15 underneath the bottom of bolt head as a length with no threads.

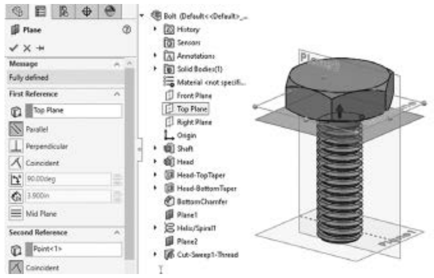
Step 7: Create *Plane2*: **Reference Geometry** on **Features** tab > **Plane** > expand feature tree > select **Right Plane** > select **Parallel** > click **Second Reference** box > select top endpoint of *Helix/Spiral1* > ✓.



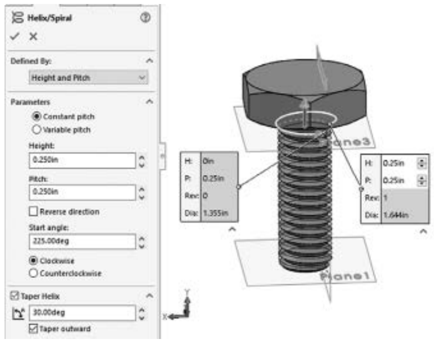
Step 8: Create *Sketch6* and *Cut-Sweep1-Thread* feature: Select *Plane2* > select **Sketch Swept Cut** on **Features** tab > **Polygon** (hexagon icon) on **Sketch** tab > 3 for **Number of Sides** > click near end of helix and sketch and dimension as shown with right side vertical and *Helix/Spiral1* endpoint and right triangle edge midpoint coincident > exit sketch > **Swept Cut** on **Features** tab > select *Sketch6* as **Profile** > select *Helix/Spiral1* as **Path** > ✓.



Step 9: Create *Plane3*: **Reference Geometry** on **Features** tab > **Plane** > expand feature tree > select **Top Plane** > select **Parallel** > click **Second Reference** box > select top endpoint of *Helix/Spiral1* > ✓.



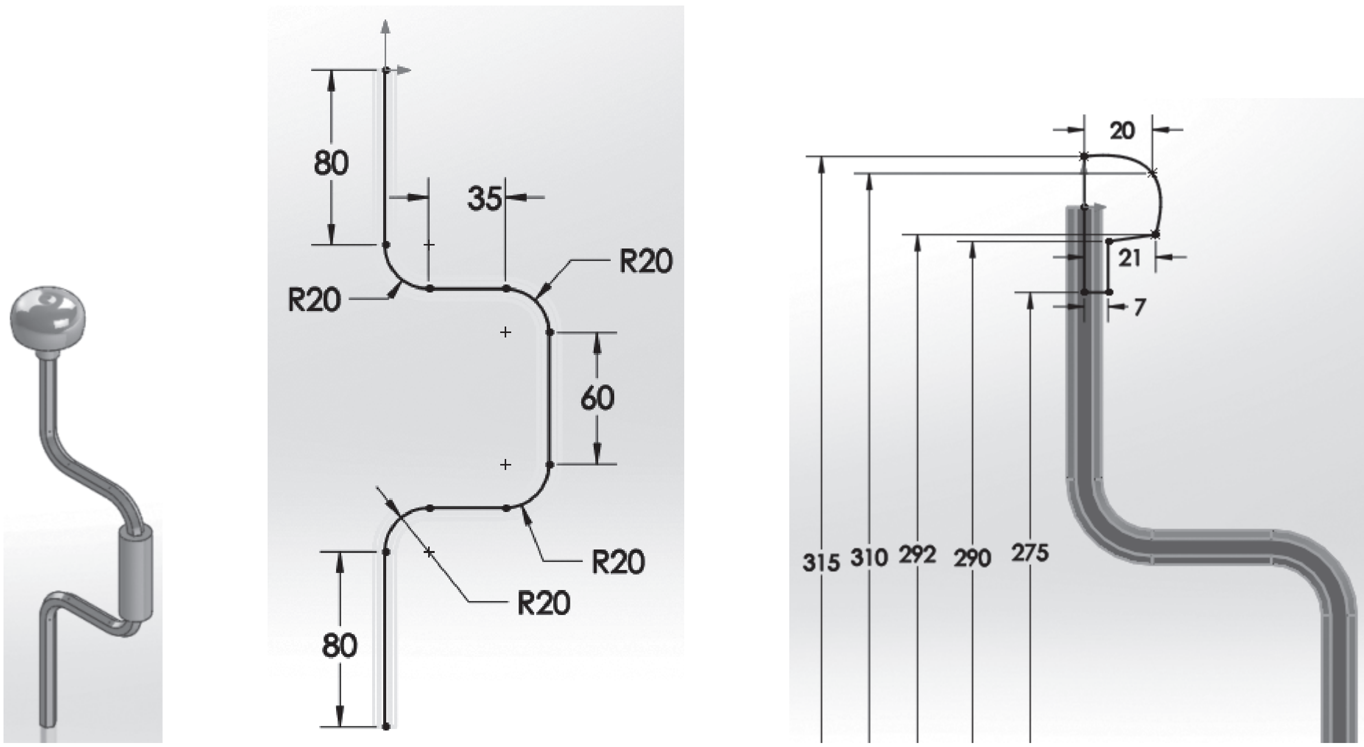
Step 10: Create *Sketch7* and *Helix/Spiral2*: Select *Plane3* > select **Sketch** on **Sketch** tab **Convert Entities** on **Sketch** tab > expand feature tree > select *Sketch1* > ✓ > exit sketch > select *Sketch7* > **Insert** > **Curve** > **Helix/Spiral** > select **Height and Pitch** > enter 0.25 for **Height** > click **Taper Helix** checkbox > enter 30 for taper angle (**A**) > click **Taper outward** checkbox > ✓.



Step 11: Create *Cut-Sweep2-EndThread* feature: **Swept Cut** on **Features** tab > select *Sketch6* as **Profile** > select *Helix/Spiral2* as **Path** > ✓.

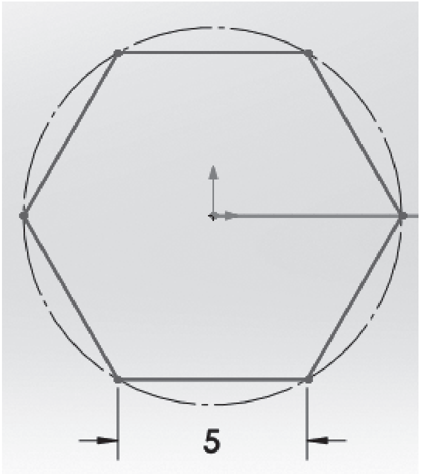
Problems

1. What is a feature? Give two examples of features.
2. Why can an extruded cut or a revolved cut *not* be a base feature?
3. What is the required input needed to create a sweep? What is the optional input?
4. What is the required input needed to create a loft? What is the optional input?
5. Table 4.1 shows a rib feature. Which is the better way to create it: by using a rib or by using an extrusion? Explain your answer.
6. Table 4.1 shows a block that is shelled. Which is the better way to create it: by using shelling or by using extrusion cut? Explain your answer.
7. A spur gear has a pitch circle radius of 3 inches, a pressure angle of 14.5 degrees, and 20 teeth. Calculate all the parameters required to create the gear CAD model. Create the CAD model.
8. Same as Problem 7, but for a pitch circle radius of 100 mm, pressure angle of 14.5 degrees, and 30 teeth.
9. Create a macro to automate the creation of a donut revolve.
10. Create the brace drill handle shown in Figure 4.18. All dimensions are in millimeters.

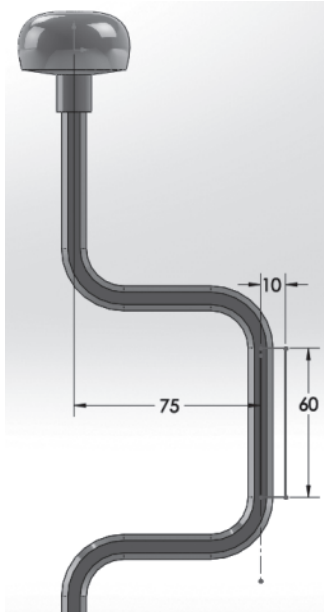


All vertical dimensions are measured from the bottom end of the drill handle. The top curve is a spline connecting three points.

Figure 4.18
Brace drill handle



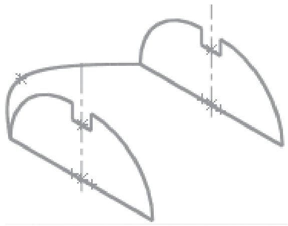
Drill handle cross section



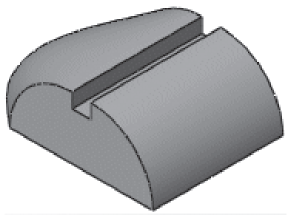
Cross section of middle handle

Figure 4.18
(*continued*)

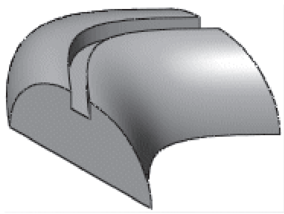
11 Create the loft feature shown in Figure 4.19. Assume dimensions. *Hint #1:* This loft shows you the local and global influence of the guide curve. *Hint #2:* Use **Guide curves influence types** under the **Guide Curves** section shown on the left pane of the screen.



Two profiles and guide curve



Local influence of guide curve



Global influence of guide curve

Figure 4.19
Influence of guide curve

12 Create the CAD model of the helical spring shown in Figure 4.20. All dimensions are in centimeters.

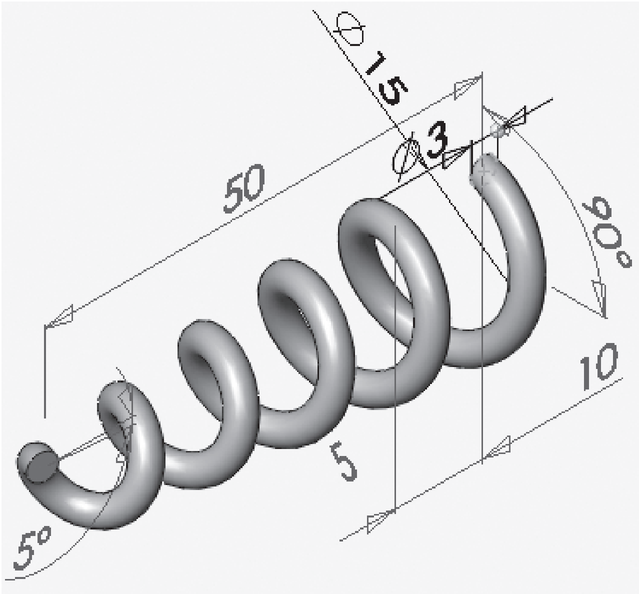
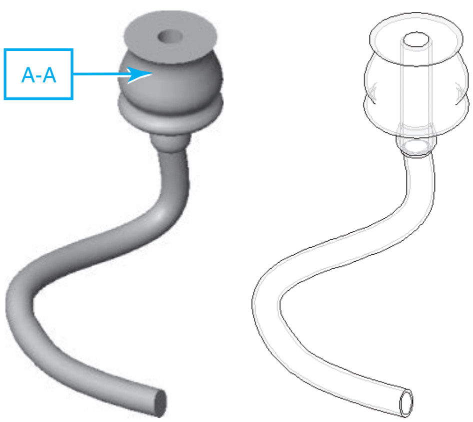


Figure 4.20
Helical spring

13 Create the CAD model of the 3D probe shown in Figure 4.21. All dimensions are in millimeters.



x	*y*	*z*
0	2.4	0
0	–50	0
0	–60	150
15	–65	160
150	–65	160

The probe uses a 3D curve to define its profile. The probe is hollow. The outer diameter is 20 mm and thickness is 2 mm

Figure 4.21
3D probe

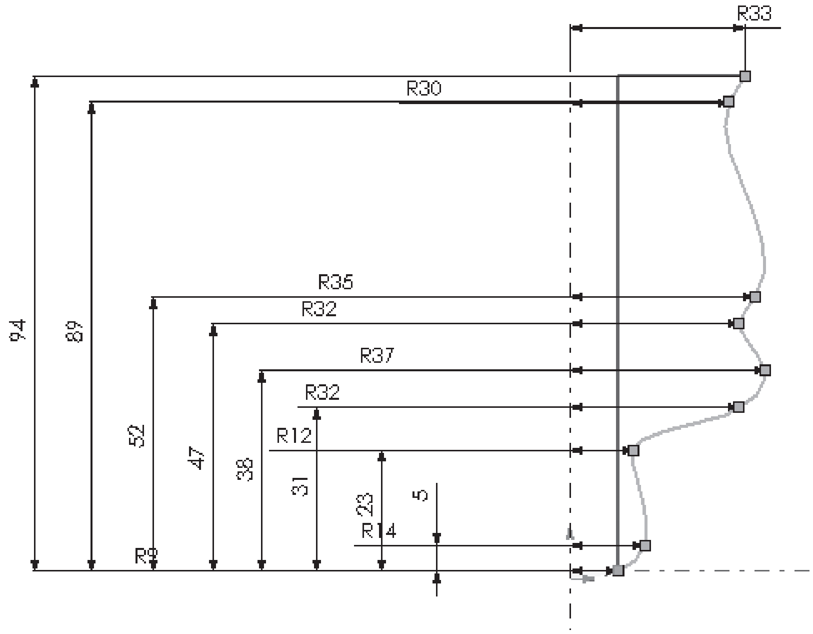
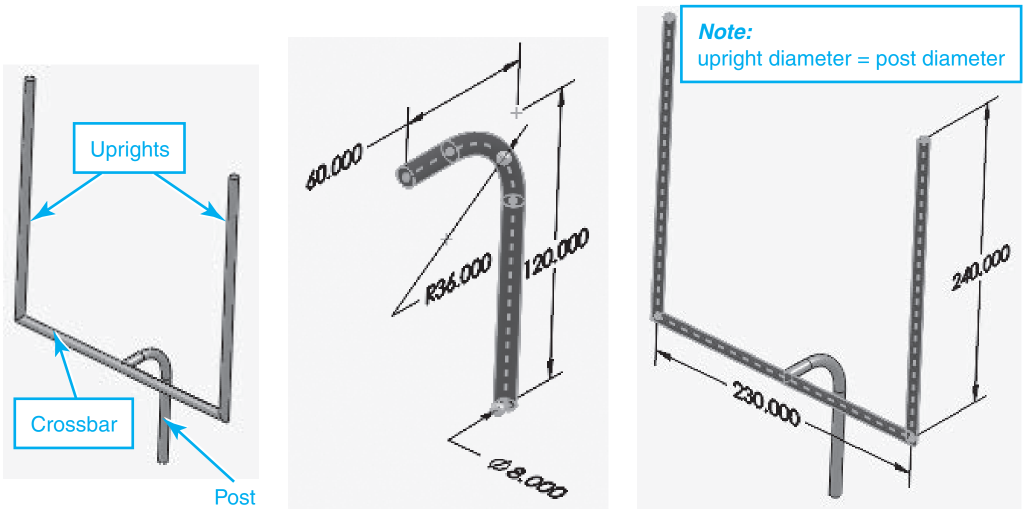


Figure 4.21
(*continued*)

14 Create the CAD model of the football goal post shown in Figure 4.22. All dimensions are in inches. *Hint:* The dimensions of the post are per NFL specs: The post is 10 feet (120 in.) high, the crossbar is 18.5 feet (222 in.) wide from the inner edges of the uprights, and the uprights are 20 feet (240 in.) high. The diameter of the post tubes is arbitrary, so use 8 in. here.

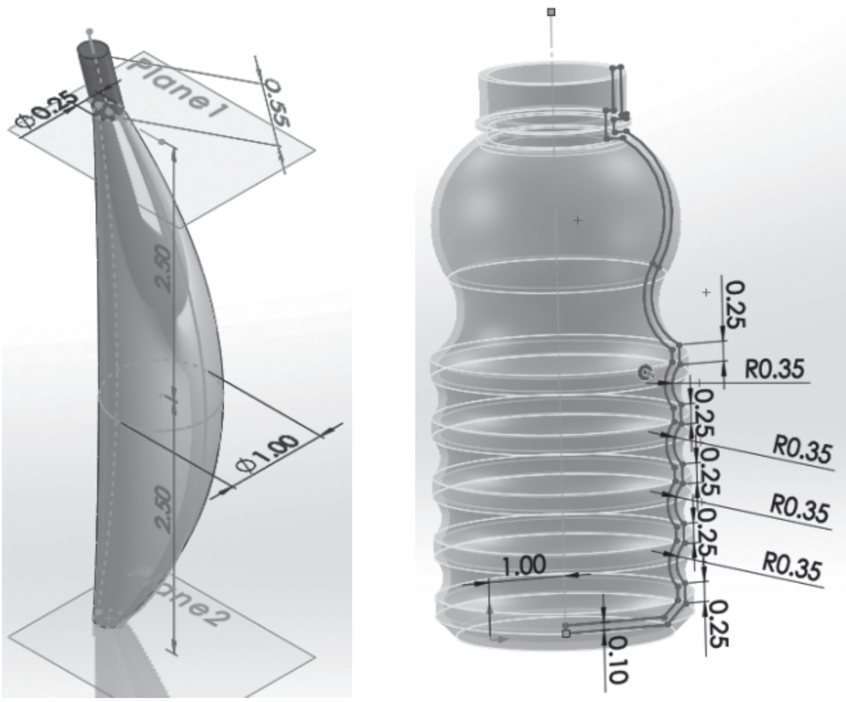


Post dimensions

Dimensions of crossbar and uprights

Figure 4.22
Football goal post

15 Create the CAD models shown in Figure 4.23. All dimensions are in inches.



(A) Banana (B) Water bottle

Figure 4.23
CAD models

16 Create the CAD models shown in Figure 4.24. All dimensions are in inches.

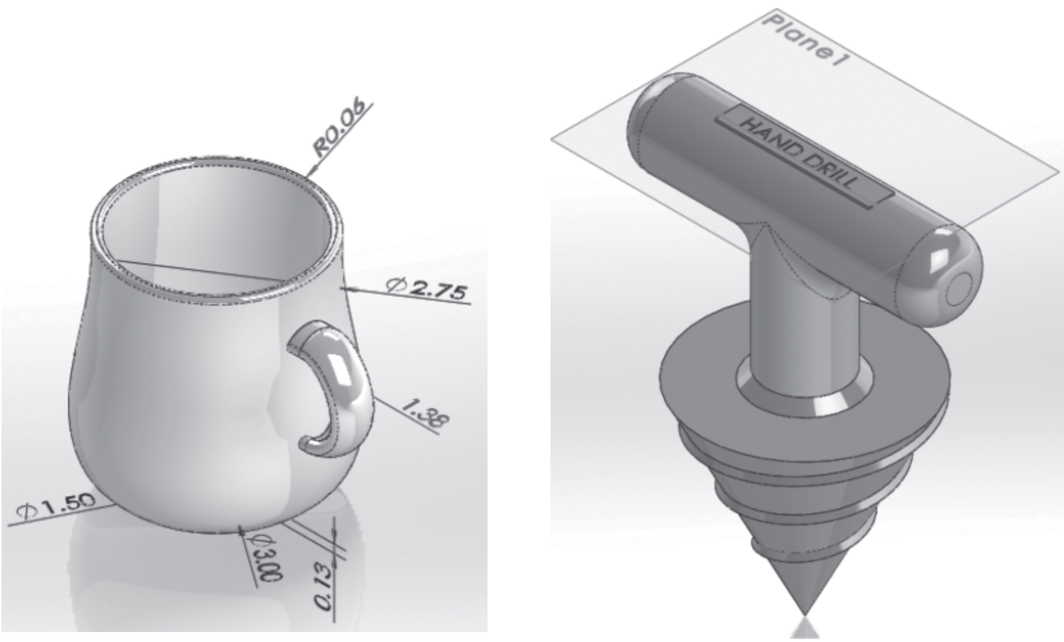


Figure 4.24
CAD models

(A) Coffee mug (B) Hand drill

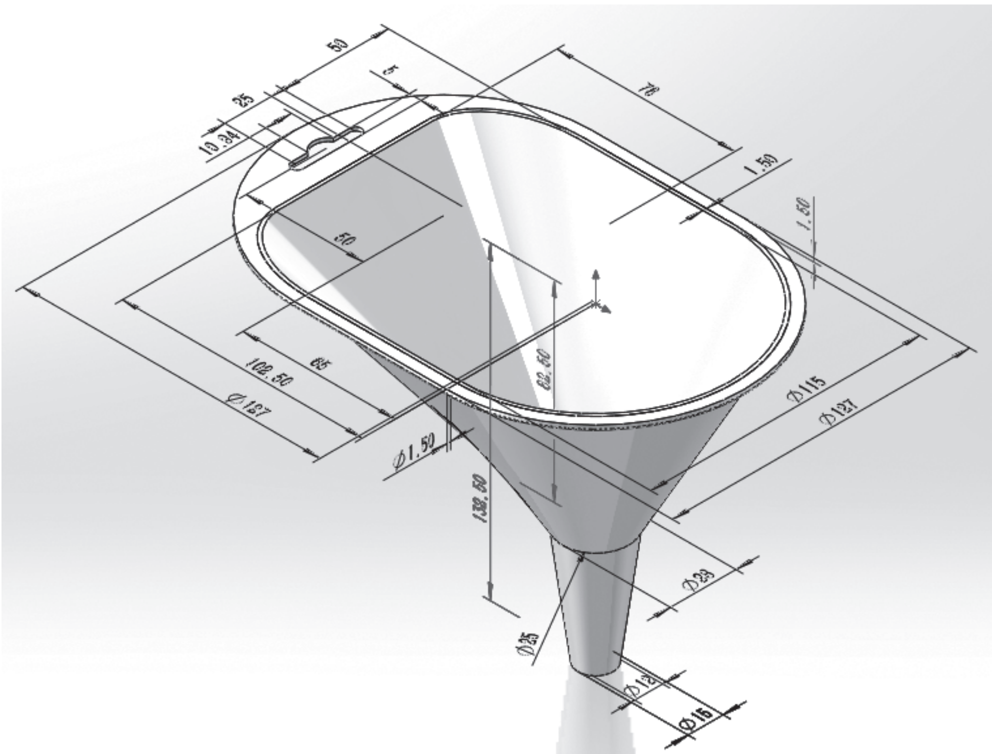
17 Create the CAD models shown in Figure 4.25. All dimensions are in inches.

(A) Goblet

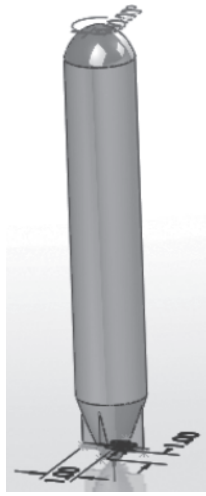
(B) Baseball bat

Figure 4.25
CAD models

18 Create the CAD models shown in Figure 4.26.



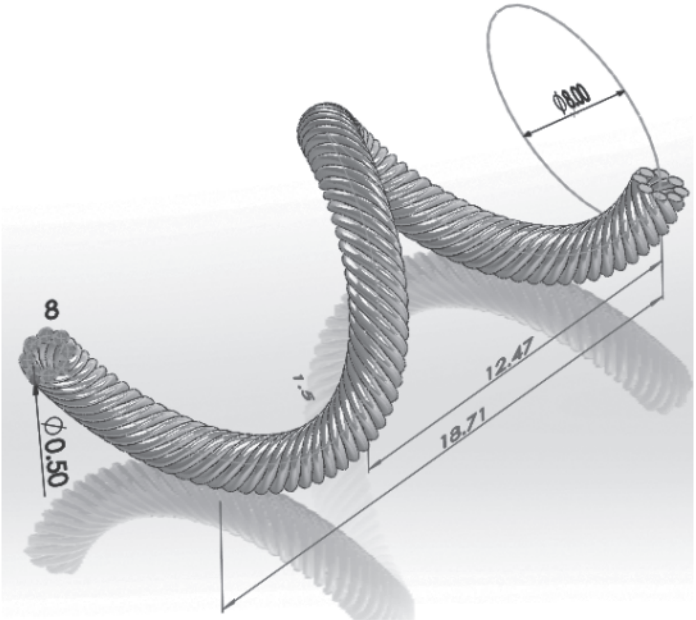
(A) Funnel (dimensions in mm)



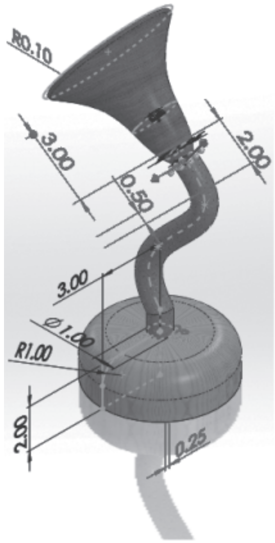
(B) Torpedo (dimensions in inches)

Figure 4.26
CAD models

19 Create the CAD models shown in Figure 4.27. All dimensions are in inches.



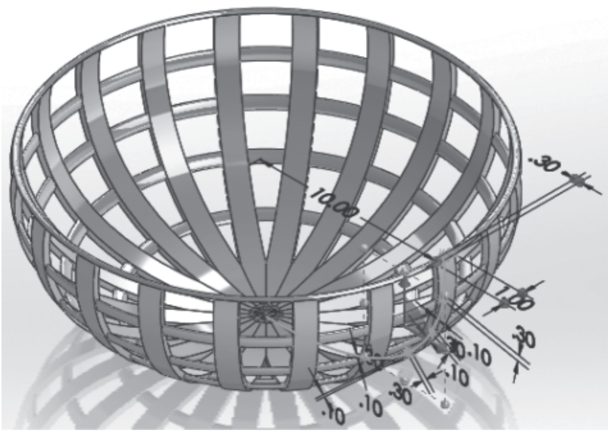
(A) Fiber optic wire



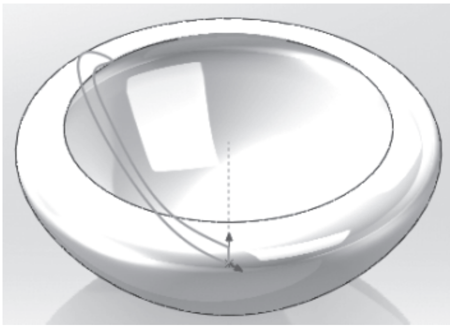
(B) Desk lamp

Figure 4.27
CAD models

20 Create the CAD models shown in Figure 4.28. All dimensions are in inches.



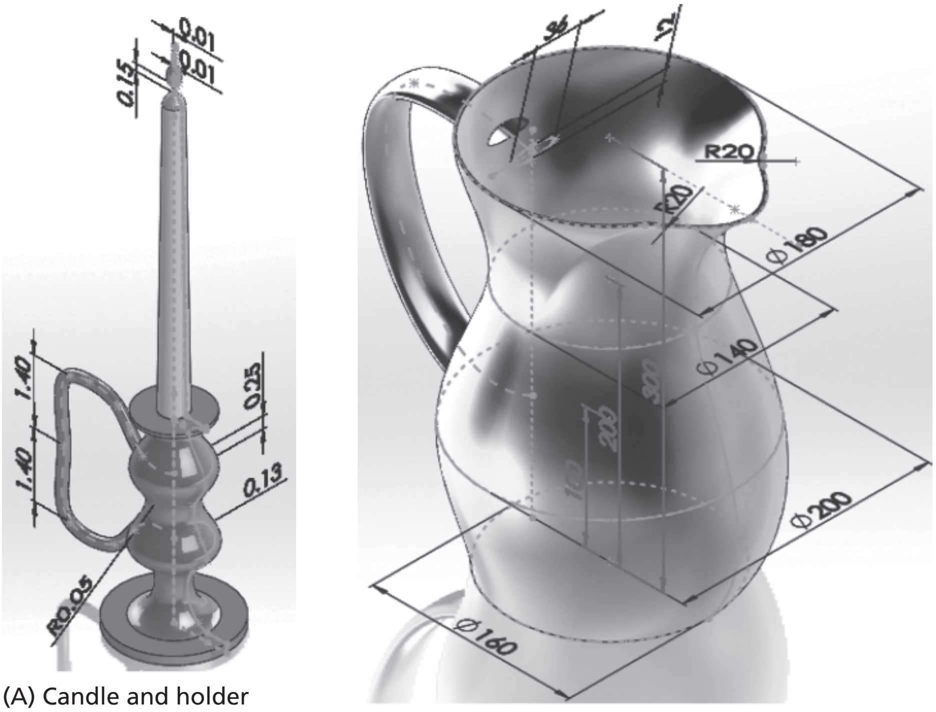
(A) Fruit basket



(B) Bowl

Figure 4.28
CAD models

21 Create the CAD models shown in Figure 4.29.

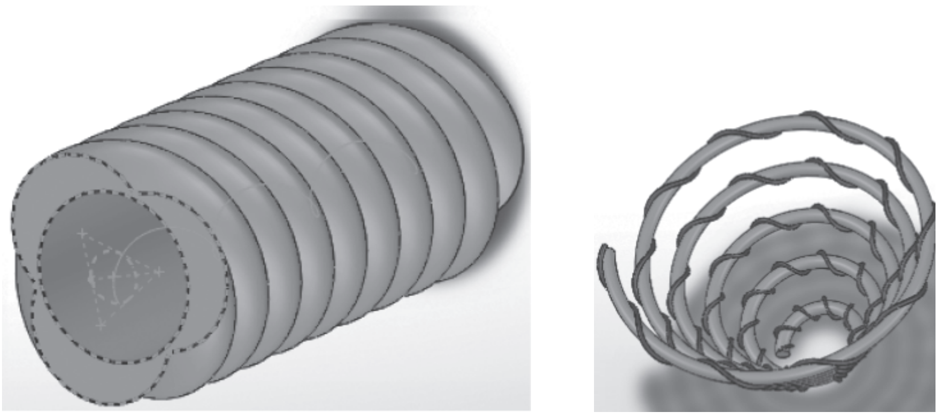


(A) Candle and holder (dimensions in inches)

(B) Pitcher (dimensions in millimeters)

Figure 4.29
CAD models

22 Create the CAD models shown in Figure 4.30.

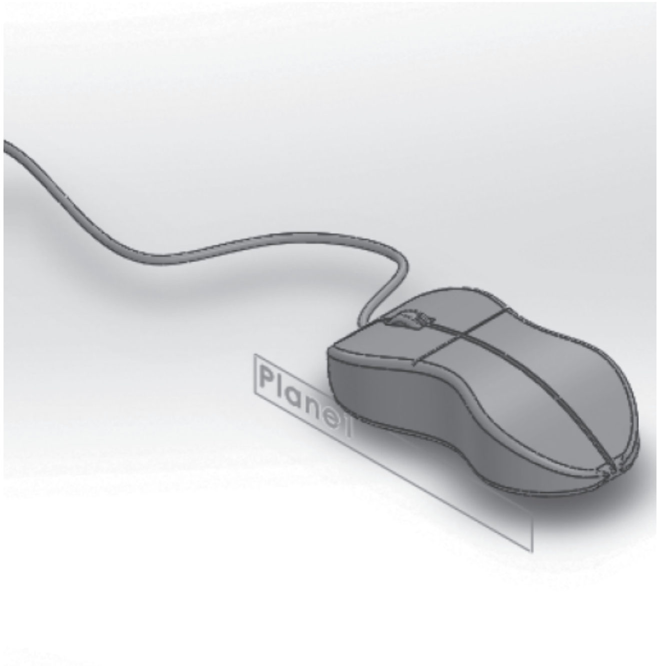


(A) A flex

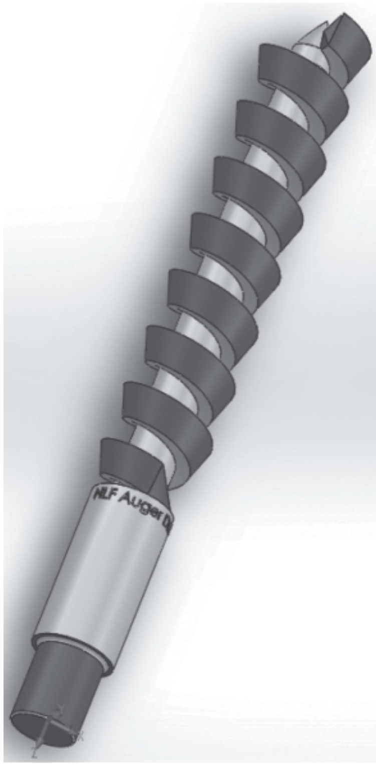
(B) Steel cone

Figure 4.30
CAD models

23 Create the CAD models shown in Figure 4.31.



(A) Computer mouse



(B) Auger drill

Figure 4.31
CAD models

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