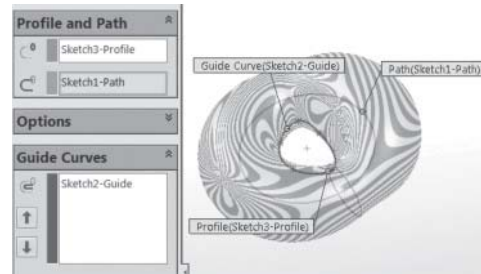


**Step 5:** Create Sweep-Torus feature: **Features** tab > **Swept Boss/base** > select *Sketch3-Profile* as the **Profile** > select *Sketch1-Path* as the **Path** > select *Sketch2-Guide* as the **Guide Curve** > ✓.

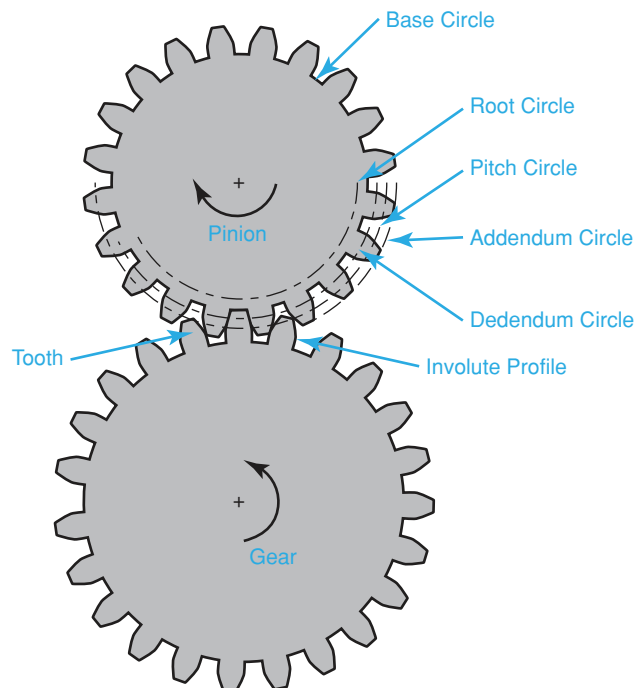


**HANDS-ON FOR EXAMPLE 4.1.** Re-create the free-form torus by replacing the spline by a circle that is not centric with the large circle.

### 4.3 Spur Gears

Gears are an important and essential mechanical element in mechanical design. A wide range of products and applications uses gears. There are various types of gears: spur, helical, bevel, spiral, worm, planetary, and rack and pinion, to name a few. Spur gear is the simplest type of gear, which 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 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

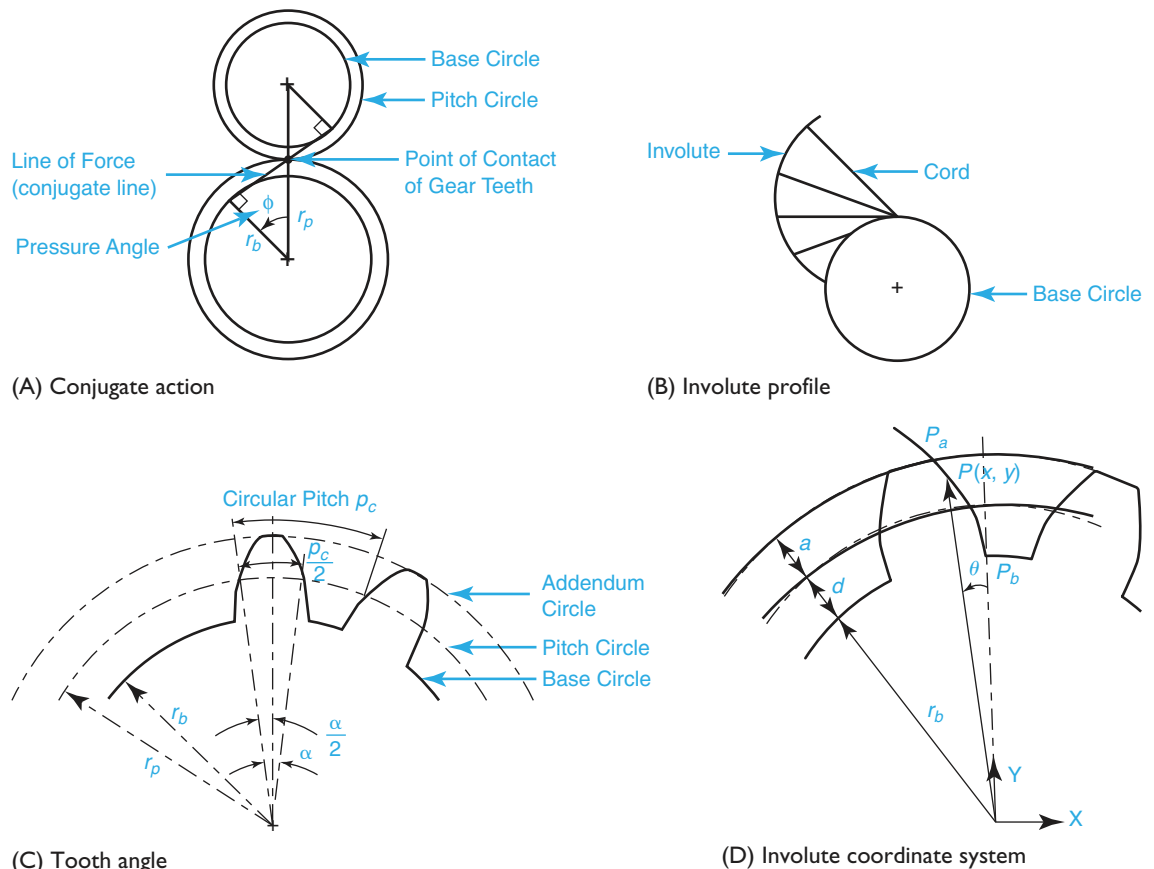


**FIGURE 4.3**  
Meshing gears

profile. Gearing and gear meshing ensure that two disks (the two gears) in contact roll against one another without slipping. 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  $\phi$  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 end point 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



**FIGURE 4.4**  
Details of a gear tooth

**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 4.3A (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 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  $\alpha$  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 to enable 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  $\alpha$  as the tooth angle. Using these variables, we can write:

$$p_c = \frac{\pi d_p}{N} \quad (4.1)$$

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:

$$\frac{p_c}{2} = r_p \alpha \quad (4.2)$$

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

$$\alpha = \frac{\pi}{N} \text{ radians or } \alpha = \frac{180}{N} \text{ degrees} \quad (4.3)$$

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:

$$\begin{aligned} x &= -r_b(\sin \theta - \theta \cos \theta) \\ y &= r_b(\cos \theta + \theta \sin \theta) \end{aligned} \quad 0 \leq \theta \leq \theta_{\max} \quad (4.4)$$

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

$$r_b = r_p \cos \phi \quad (4.5)$$

and  $(x, y)$  are the coordinates of any point  $P$  on the involute at an angle  $\theta$  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:

$$r_r = 0.98r_b \quad (4.6)$$

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

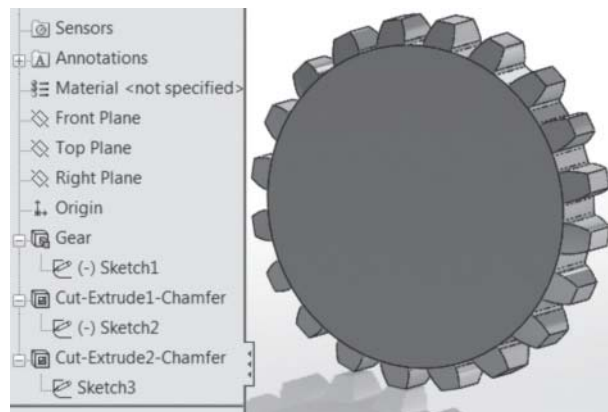
1. The input parameters we need are the pitch circle radius  $r_p$ , the pressure angle  $\phi$ , and the gear number of teeth  $N$ .
2. Calculate  $r_b$  using Eq. (4.5).
3. Calculate  $r_r$  using Eq. (4.6).
4. Calculate the gear dedendum  $d = r_p - r_b$ .
5. Assuming that the addendum and dedendum 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  $\alpha$ .
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 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,  $\phi = 20^\circ$ , and  $N = 20$ .

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

In the first method, we use Eq. (4.4) with  $\Delta\theta = 5^\circ$ . We generate 11 points on the involute, for a  $\theta_{\max} = 50^\circ$ . We generate the points on the involute curve. We then use **Insert > Curve > Curve Through XYZ Points**. A better method is to input Eq. (4.4) to SolidWorks and let it generate the curve. We need to use radians for the angle  $\theta$ . We use 1 radian for  $\theta_{\max}$ . 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  $\theta$  by  $t$  when we input the equation. Figure 4.5 shows the spur gear. We create half a tooth, mirror it to create a full tooth, then circular pattern the full tooth to create all teeth of the gear. Here are the detailed steps.

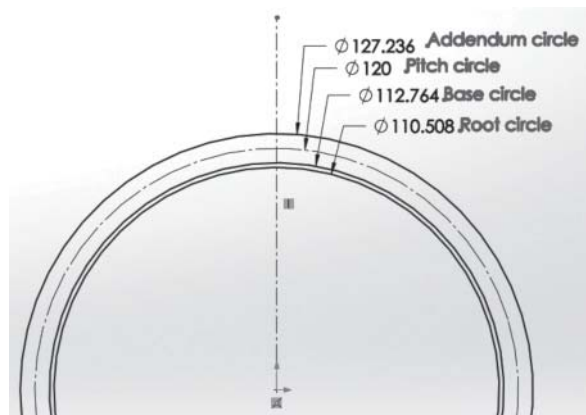
**FIGURE 4.5**  
Spur gear



**Step 1:** Create *Sketch1* circles and axes:

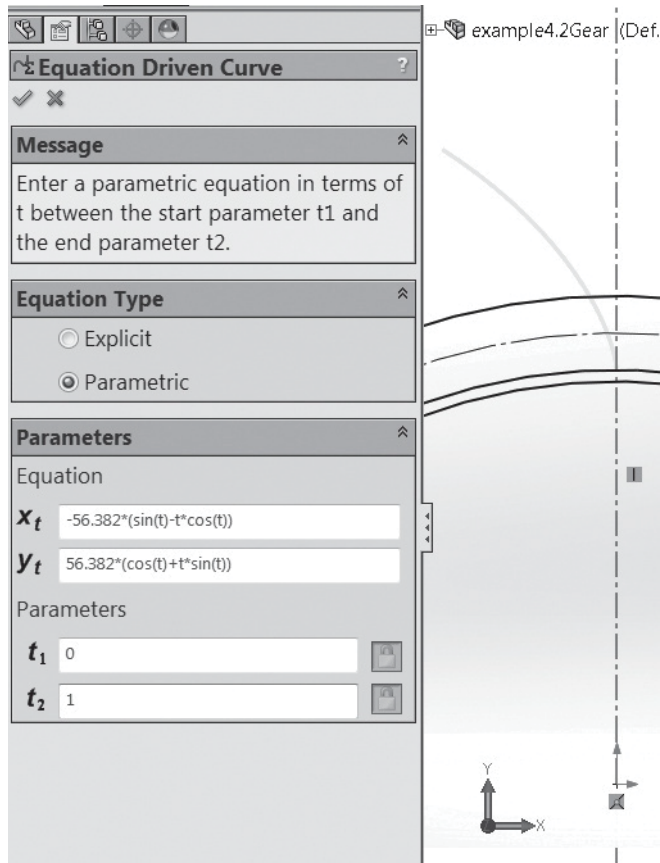
**File > New > Part > OK > Front Plane > Circle** on **Sketch** tab > sketch four circles and dimension as shown > **Centerline** on the **Sketch** tab > sketch vertical line > **File > Save As > example4.2 > Save**.

**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 we create it in Step 2. Also, you will not close the sketch until you finish Step 5.



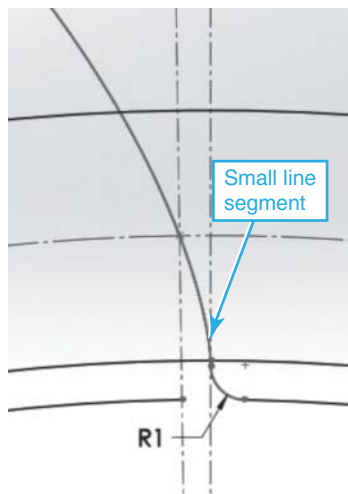
**Step 2:** Create *Sketch1*-tooth involute:

**Sketch** tab > **Spline** dropdown on the **Sketch** tab > **Equation Driven Curve** > **Parametric** > enter x and y equations and limits as shown > ✓.



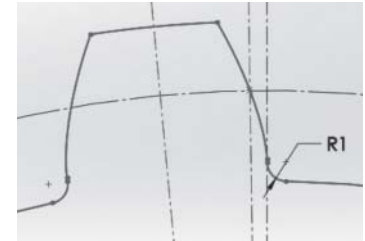
**Step 3:** Create *Sketch1*-tooth bottom:

**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) > **Center line** 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.

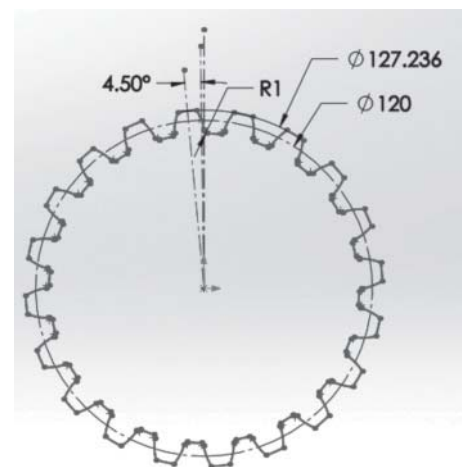
**Step 4:** Create *Sketch1*-tooth other half: **Trim Entities** on **Sketch** tab > **Trim to closest** > select involute top part > **Centerline** on **Sketch** tab > sketch a line



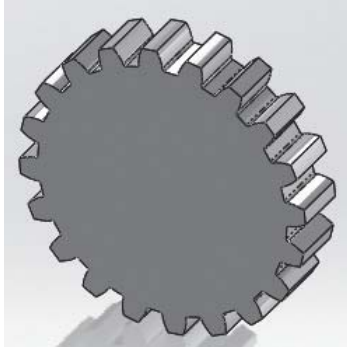
passing through origin and to left of involute > **Smart Dimension** on **Sketch** tab > select the centerline just created and the other centerline to the right of it > enter 4.5 > ✓ > **Mirror Entities** on **Sketch** tab > select involute + **Ctrl** key + line segment connected to involute + fillet 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.

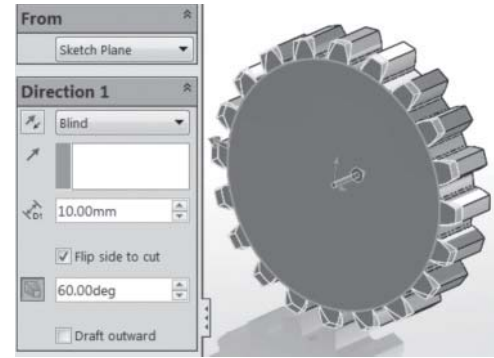


**Step 6:** Create *Gear* feature: Select *Sketch1* > **Features** tab > **Extruded Boss/Base** > Enter 25 for thickness (D1) > reverse extrusion direction > ✓.

**Step 7:** Create *Sketch2* and *Cut-Extrude1-Chamfer*: Select *Gear* front face > **Features** tab > **Extruded**

**Cut** > **Circle** on **Sketch** tab > click origin and snap to teeth root circle > exit sketch > enter 10 for thick-

ness (D1) > check **Flip side cut** as shown above > click **Draft** icon as shown above > enter 60 for draft angle > ✓.



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

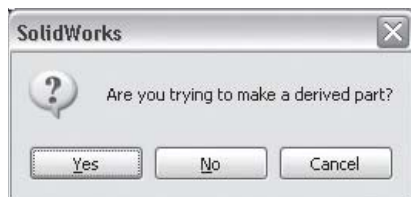
**HANDS-ON FOR EXAMPLE 4.2.** Add a hole and a keyway to the gear blank. Use a diameter of 50 mm for the hole and 10 × 10 mm keyway.

## 4.4 Design Library and Library Features

Design reuse and using off-the-shelf standard components are important concepts that speed up the 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, slots, and many others. You can use several library features to construct a single part. Not only does this save time, but it also ensures consistency in your CAD models.

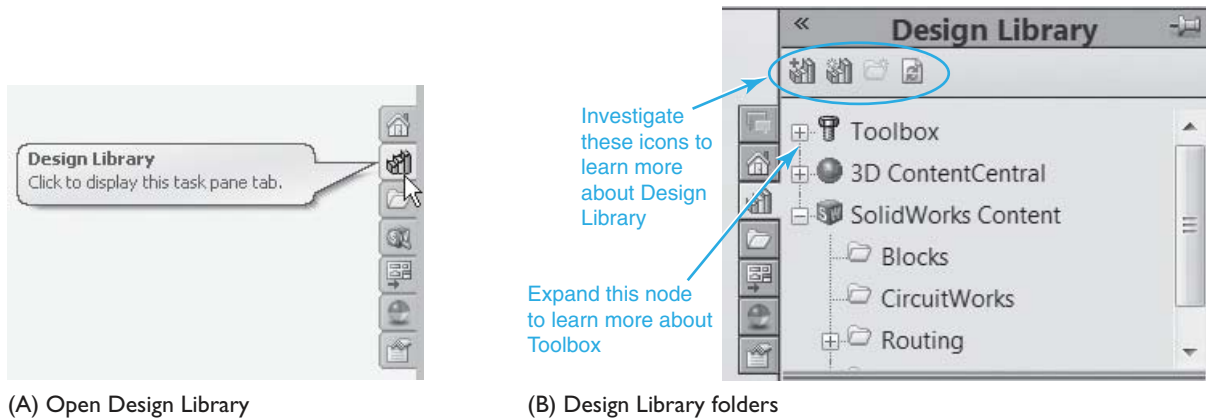
Using library features is easy; you drag a library feature from the design library and drop it onto the open part or assembly. SolidWorks asks you if you want to insert a copy or instance (**derived part**) of the part as shown in Figure 4.6. If you select **Yes**, it inserts the copy in the open part.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



**FIGURE 4.6**  
Using a library feature

You save library features in a design library. You can organize the library into folders. SolidWorks comes with a Design Library. The path to this 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 (shown on the right of the screen) as shown in Figure 4.7A, to open the design library. The library is organized into folders as shown in Figure 4.7B. You can also add





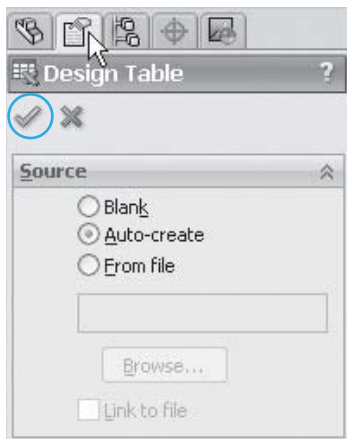
**FIGURE 4.7**  
SolidWorks Design Library

your new custom folders to the library. You should save your library features into the SolidWorks **Design Library** to have them accessible as shown in Figure 4.7B. If you do not, then you would have to navigate to the folder where you saved them. The most commonly used SolidWorks library is **Toolbox** shown in Figure 4.7B. Expand the Toolbox node and investigate its content.

## 4.5 Configurations and Design Tables

Family of parts is a natural outcome of the parametrics concept of 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 different-size clone of the part with a click of a button. These clones are what we refer to as family of parts. SolidWorks calls them *configurations*. We can also create clones of assemblies. The clones are created by changing the dimensions of some key parameters of the part or assembly. The clones 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 clone the part into a square block and a skinny long boss, or a rectangle block and short fat boss.

We 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 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** pane under the **Property-Manager** 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 check mark 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 set (configuration). The set shows as a row in the table. You can add other rows with different values for dimensions. Each row is a configuration. The **Design Table** is saved under the **ConfigurationManager** tab.



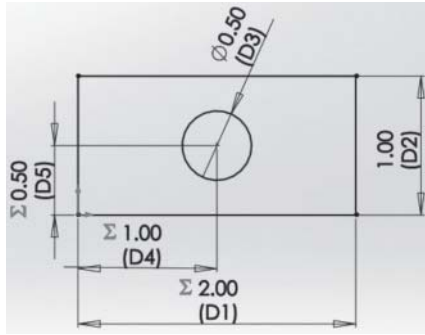
**FIGURE 4.8**  
Design table

### Example 4.3 Create design tables.

**Solution** This example builds on Example 2.4. We create a design table where we 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**.

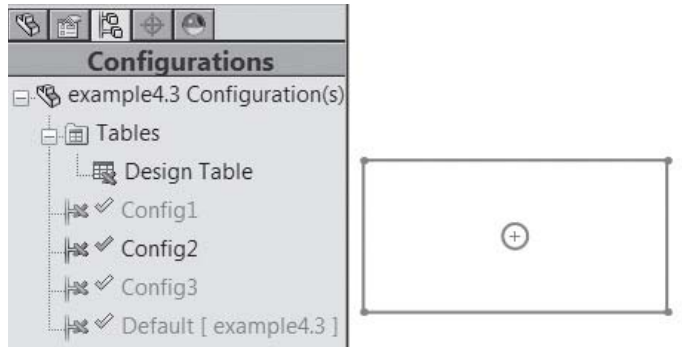


**Step 2:** Create design table with *Default* config:

**Insert > Tables > Design Table > ✓ >** click *D1* (shown here) + **Shift** key on keyboard + *D5* (shown here) to select all dims > **OK**.



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



**Step 4:** Review the design configs: Step 3 creates a tree with four nodes (as shown above) under the **ConfigurationManager** tab > double click any config to display the corresponding sketch > the current config is displayed in dark black in the tree.

	D1@Sketch1	D2@Sketch1	D3@Sketch1	D4@Sketch1	D5@Sketch1
1	Design Table for: example4.3				
2					
3	Default	=2*D2@Sketch1	1	0.5 = 0.5 * "D1@Sketch1"	=0.5 * "D2@Sketch1"
4	Config1	=2*D2@Sketch1	2	0.5 = 0.5 * "D1@Sketch1"	=0.5 * "D2@Sketch1"
5	Config2	=2*D2@Sketch1	3	0.5 = 0.5 * "D1@Sketch1"	=0.5 * "D2@Sketch1"
6	Config3	=2*D2@Sketch1	4	0.5 = 0.5 * "D1@Sketch1"	=0.5 * "D2@Sketch1"
7					
8					
9					
10					



**Step 5:** Investigate effect of design table on sketch parameters: Click features tree > *Sketch1* > observe that dimensions are displayed in a pink color and each time you click or move one, you get a window (shown above) 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 we 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 we turn the macro on until we turn it off. We can play the macro back after creation over and over, with different input values (e.g., new dimensions). After we create a macro, we can use it



for recording, edit it, run it, pause it, stop it, and assign it to a shortcut (hot) key or to a menu item. When we assign a macro to a shortcut key or to a menu item, we 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 default names of *Macro1.swp*, *Macro2.swp*, 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 we perform the design tasks as usual. It is this VB code that we save in a file when we save the macro. We can use the VB editor to edit and tweak the VB code.

If you take a closer look at the macro VB code, it uses what we call the SolidWorks API (Application Programming Interface). The code makes calls to API functions. Think of API as the gateway between the application we want to write and SolidWorks code that has been already written. In other words, the API provides access to SolidWorks geometric engine.

You can learn VB programming by creating multiple macros, study their generated VB code, and expand on it. That is what we call the “brute force” approach. Just keep in mind that VB is an object-oriented programming (OOP) language that requires knowledge and understanding of the object-oriented design and how objects are defined and implemented.

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

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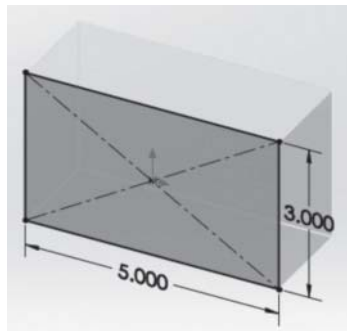
#### Example 4.4 Develop a macro to create an extrusion.

**Solution** This example illustrates creating and using macros. We create a block extrusion and record the steps of creating it in a macro.

**Step 1:** Turn macro on and create *Sketch1* and *Block* 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 thickness (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 *BlockShift* for macro name in the **Save As** window that opens > **Save**.

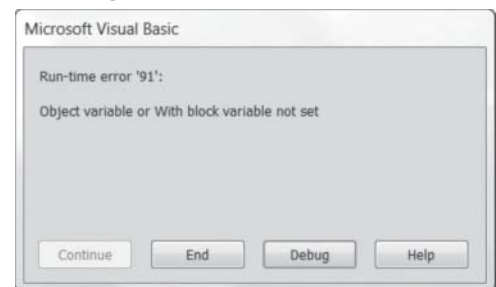
**Note:** full macro file name is *BlockShift.swp*. The default folder for the file is the part file folder from Step 1.

**Step 3:** Run and debug the macro: **Tools > Macro >**

**Run > select *BlockShift*.**

**swp >**

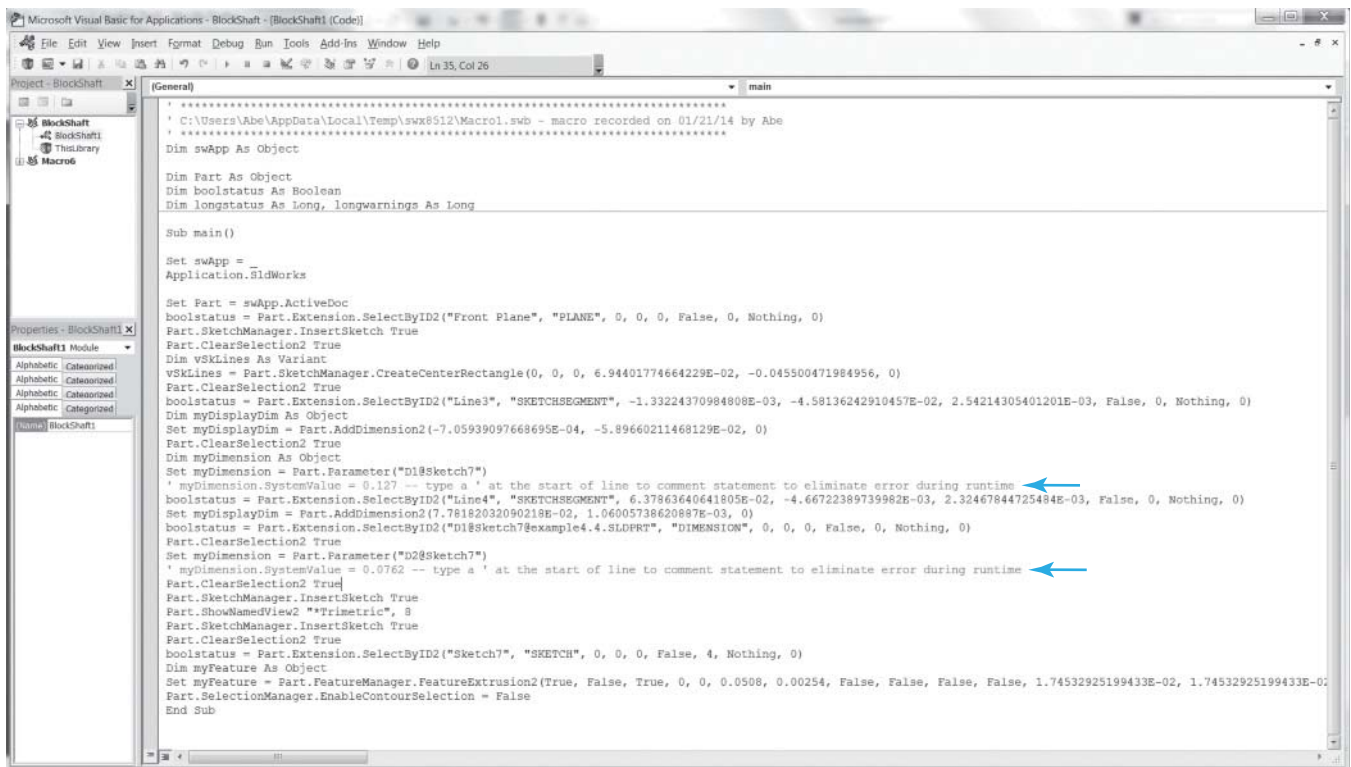
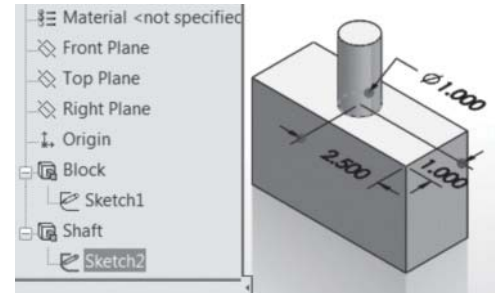
**Open.** The macro will generate the error shown here as soon as it displays the dimension box and you click ✓ or hit **Enter**.



We do not know why this error occurs but we know how to fix it. Hit the **Debug** button shown above to open the macro code in the VB editor. The editor highlights the line of code that causes the error in yellow. Comment every yellow statement and save the macro file. Figure 4.9 shows the entire macro VB code and the fix as indicated by two arrows.

**Step 4:** Run the macro again after the fix: 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

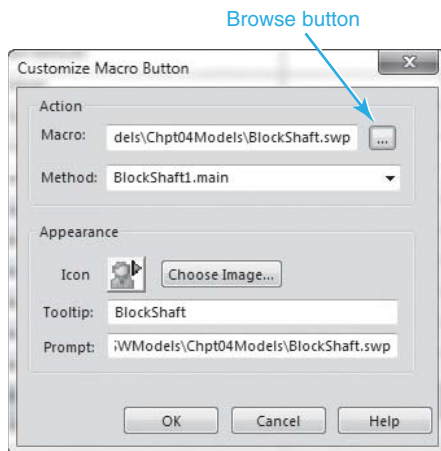
### Example 4.5 Create a hot key for a macro.

**Solution** This example assigns the macro of Example 4.4 to a hot key on the keyboard, namely **Shift + X**. When the user hits this combination on the keyboard, the macro runs.

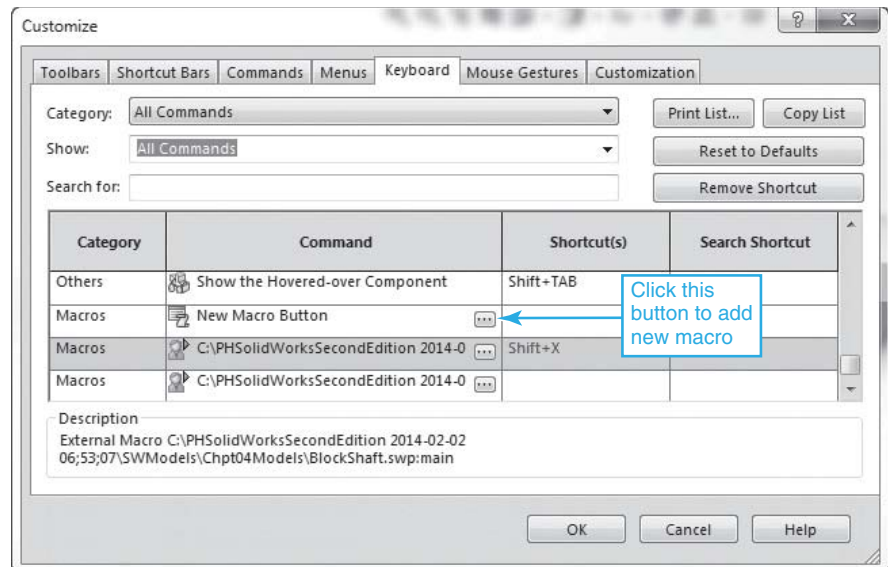
**Step 1:** Create macro hot key:

**File > Open > locate *example4.4* > Open.**

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



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



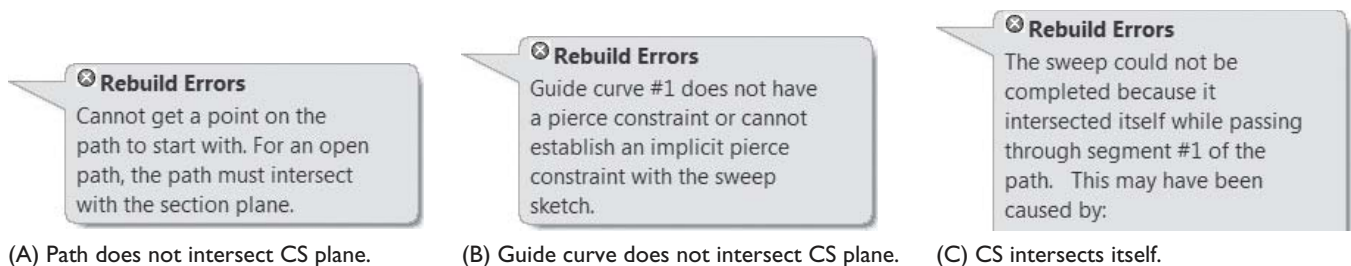
**Step 4:** Use hot key to run macro: Hit **Shift + X** keys in an open part.

## 4.7 Tutorials

### Tutorial 4-I: Create Sweep Features

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

Sweep operations may fail for different reasons. Figure 4.10 shows three of them. 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.



**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

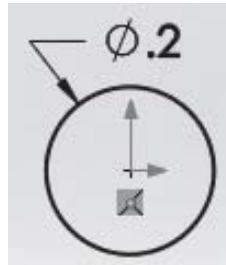
**FIGURE 4.11**

Sweep features

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

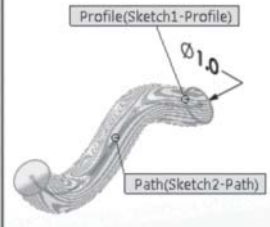
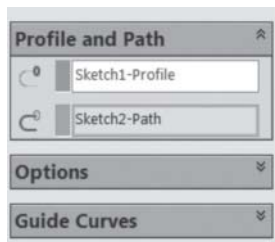
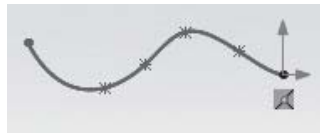
**Step 1:** Create *Sketch1-Profile*:

**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; make sure spline snaps to origin > exit sketch.

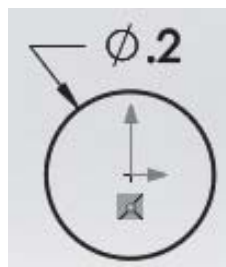


**Step 3:** Create *Sweep1* feature: **Sweep Boss/Base** on **Features** tab > select circle sketch as profile as shown above > select spline sketch as path as shown above > ✓.

**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 and dimension as shown > exit sketch > **File** > **Save As** > *tutorial4.1B* > **Save**.



**Step 2:** Create *Sketch2-Path*:

**Top Plane** > **Spline** on **Sketch** tab > sketch spline as shown; make sure spline snaps to origin > exit sketch.

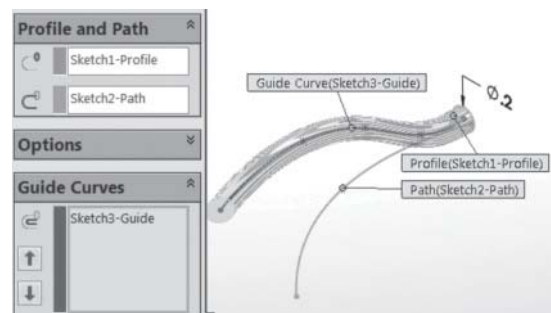


**Step 3:** Create *Sketch3-Guide*:

**Right Plane** > **Spline** on **Sketch** tab > sketch spline as shown; make sure spline snaps to origin > exit sketch.

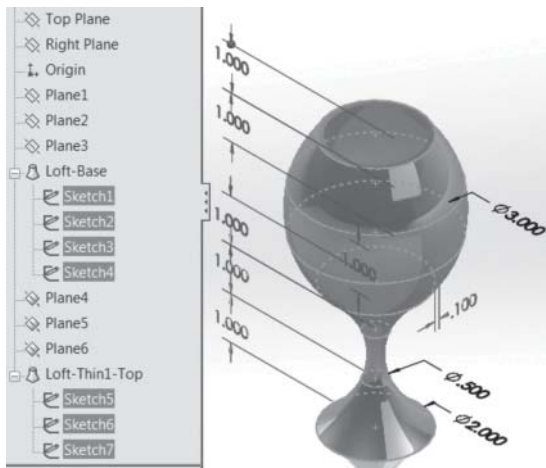


**Step 4:** Create *Sweep* feature: **Sweep Boss/Base** on **Features** tab > select circle sketch as profile as shown > select **Step 2** spline sketch as path > select **Step 3** spline sketch as **Guide** > ✓.



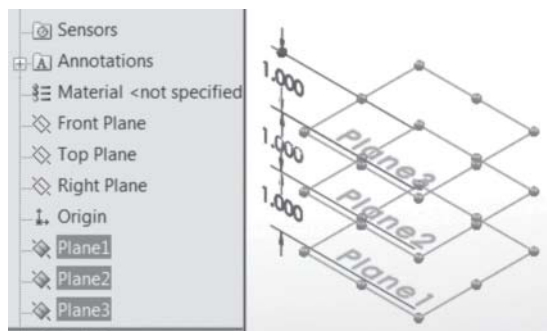
**HANDS-ON FOR TUTORIAL 4-1.** Change the cross sections of both sweeps to a 2 × 2 inch square. What happens? Resolve the error(s) and explain your solution.

## Tutorial 4-2: Create Loft Features



**FIGURE 4.12**  
Loft feature

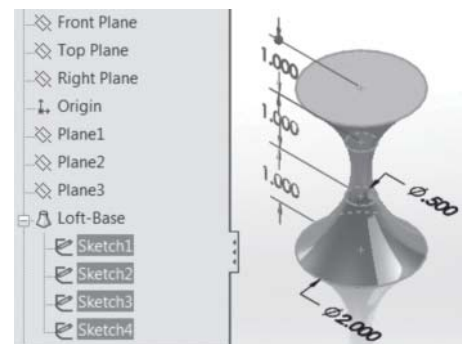
**Step 1:** Create *Plane1–Plane3*: **File > New > Part > OK > Reference Geometry on Features tab > Plane >** expand features tree and select **Top Plane >** enter 1.0 for distance > ✓ > repeat for *Plane2* and *Plane3*, but select the previously created plane > **File > Save As > tutorial4.2 > Save.**



**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

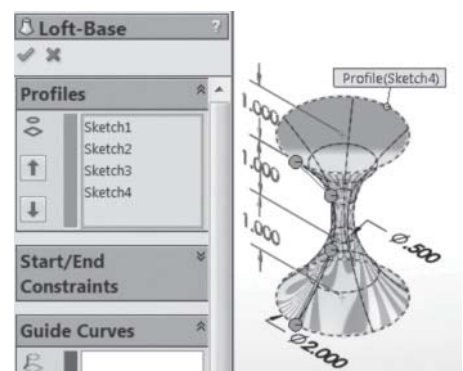
plane > **Convert Entities on Sketch tab >** click circle on *Plane1* > ✓ > ✓ > exit sketch > select *Plane3* as sketch plane >

**Convert Entities on Sketch tab >** click circle on *Top Plane* > ✓ > ✓ > exit sketch.



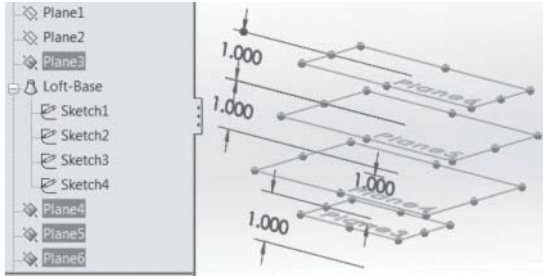
### Step 3:

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

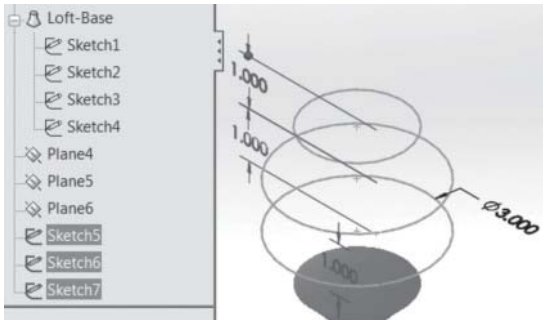




**Step 4:** Create *Plane4–Plane6: Reference Geometry* on **Features** tab > **Plane** > expand features tree and select *Plane3* > enter 1.0 for distance > ✓ > repeat for *Plane5* and *Plane6*, but select the previously created plane.



**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



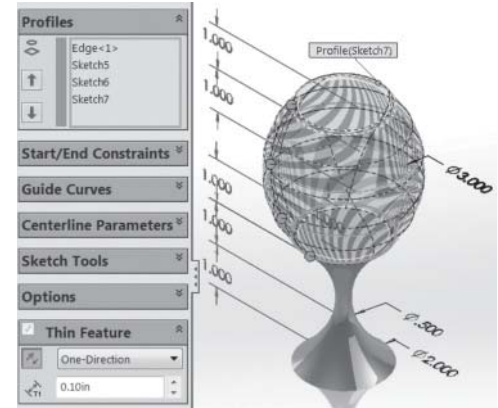
circle > exit sketch > select *Plane5* as sketch plane > **Convert Entities** on **Sketch** tab > click *Sketch5* just created > ✓ > ✓ > exit sketch > select *Plane6* as sketch plane > **Convert Entities** on **Sketch** tab > click circle on *Plane3* > ✓ > ✓ > exit sketch.

#### Step 6:

Create *Loft-Thin1-Top* feature:

**Lofted Boss/Base** on **Features** tab > select circle of *Sketch4*, then select *Sketch5–*

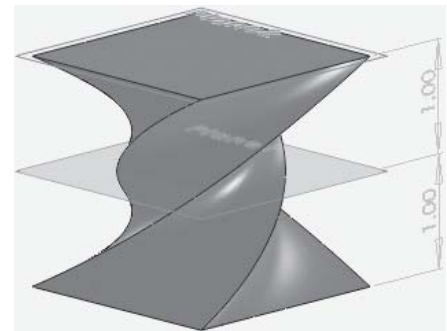
*Sketch7* > check **Thin Feature** box > enter 0.1 for **T1** > if needed, click direction box to toggle direction of thickness > ✓.



**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.

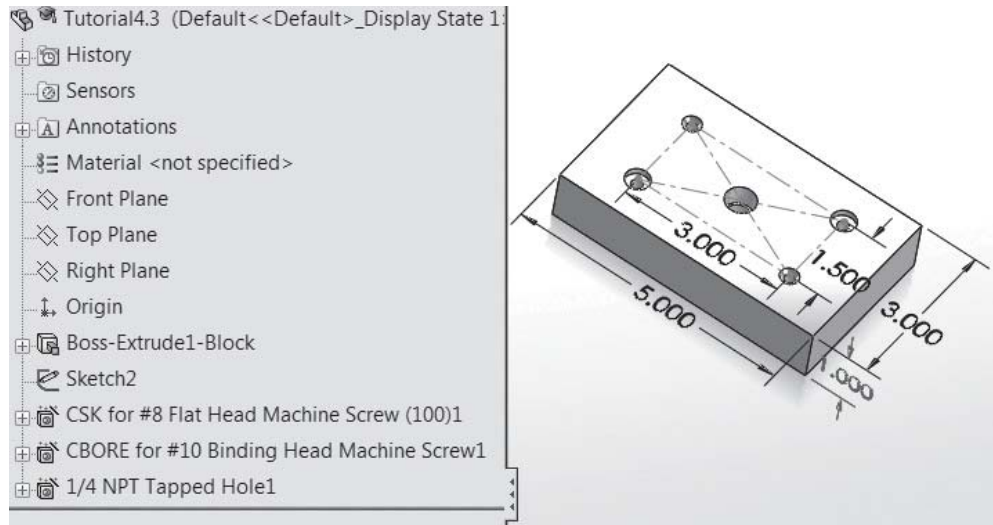
**Note:** The thickness of the thin feature has a direction: inside or outside the profile. Reverse the direction of the double arrows shown to toggle.

**HANDS ON FOR TUTORIAL 4–2.** Created a loft using three squares of different sizes as cross sections separated by one inch. The square sizes are  $2 \times 2$ ,  $1 \times 1$ , and  $2 \times 2$  respectively. Create the loft connecting the three sections such that the loft is twisted incorrectly as shown to the right.



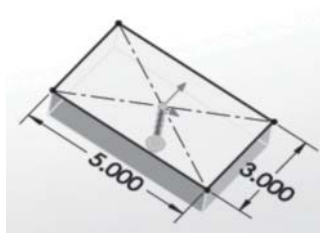
### Tutorial 4–3: Use the Hole Wizard

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



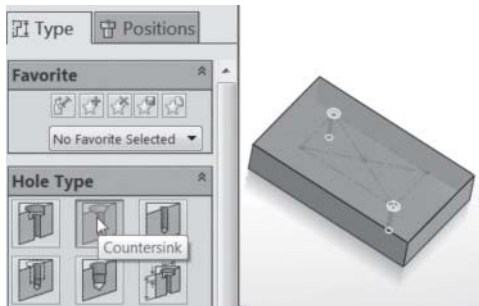
**FIGURE 4.13**  
Wizard holes

**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 thickness **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.

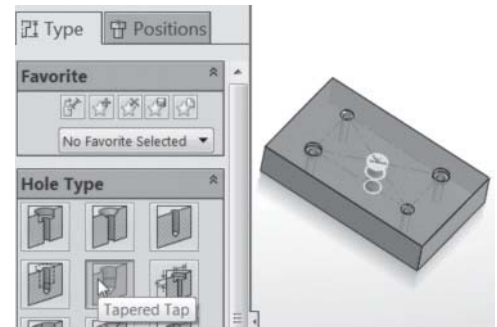
**Step 3:** Create two diagonal countersink hole features (*CSK...* node in features tree): **Hole Wizard** on **Features** tab > select **Countersink** under **Hole Type** (hover over types until you read it) > select #8 for **Size** under **Hole Specifications** > **Positions** tab > click top face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



**Step 4:** Create two diagonal counterbore hole features (*CBORE...* node in features 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*, then click two other corners of construction rectangles as shown > ✓.



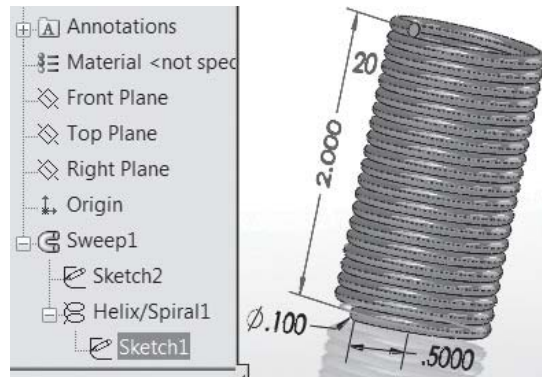
**Step 5:** Create *Tapped Hole1* feature: **Hole Wizard** on **Features** tab > select **Tapered Tap** (hover over types until you read it) > select 1/4 for **Size** under **Hole Specifications** > **Positions** tab > click top face of *Block*, then click rectangle center as shown > ✓.



**HANDS-ON FOR TUTORIAL 4-3.** Create a 1/16 tapered pipe tap through all 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 we create in this tutorial.



**FIGURE 4.14**  
Compression spring

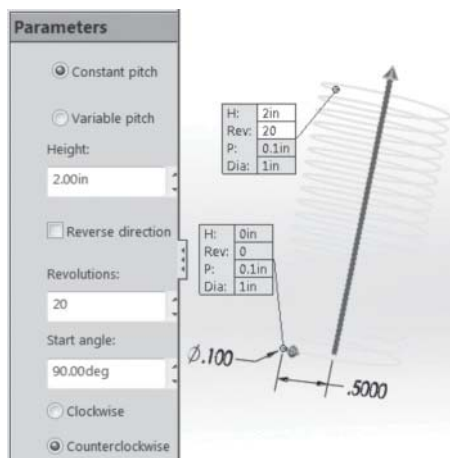
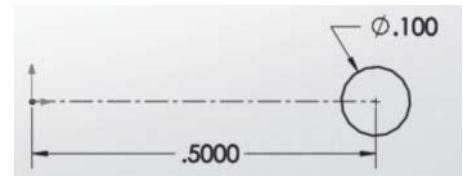
**Step 1:** Create *Sketch1* and *Helix/Spiral1* curve: **File > New > Part > OK > Top Plane > Circle** on **Sketch** tab > click origin and sketch and dimension circle with 1.0 inch diameter > exit sketch > **Insert (menu) > Curve > Helix/Spiral** > enter 2.0 for **Height** and 20 for **Revolutions** as shown > ✓ > **File > Save As > tutorial4.4 > Save.**

### Step 2:

Create *Sketch2*:

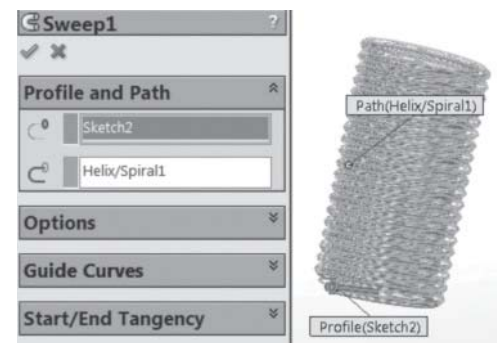
**Front Plane > Circle** on **Sketch** tab >

sketch and dimension circle as shown (align center with X-axis as shown) > exit sketch.



### Step 3:

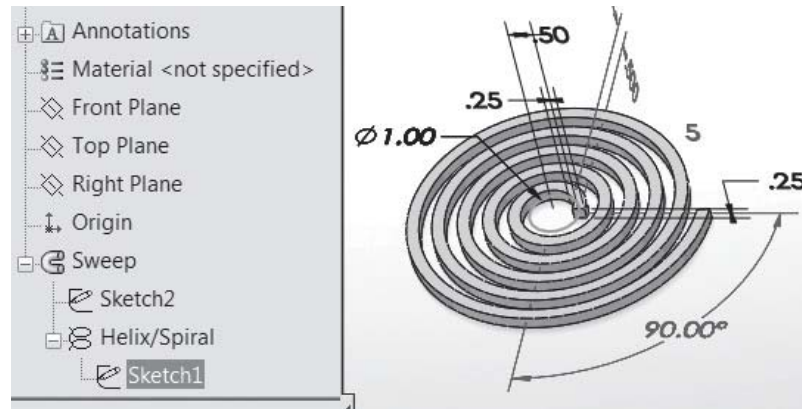
Create *Sweep1* feature (spring): **Swept Boss/Base** on **Features** tab > select *Sketch2* as **Profile** and *Helix/Spiral1* as **Path** > ✓.



**HANDS-ON FOR TUTORIAL 4-4.** Edit the spring helix to have a variable pitch. Use a pitch of 0.2 at midheight point.

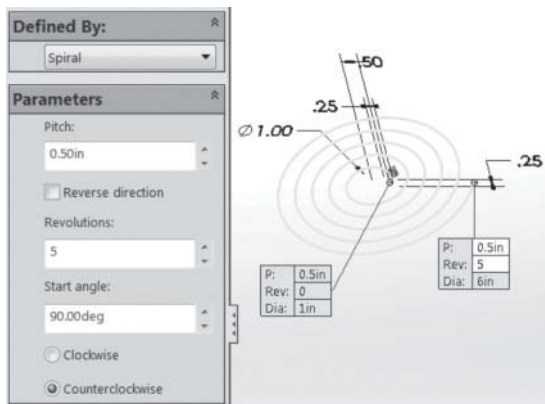
## Tutorial 4–5: Create Spiral

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

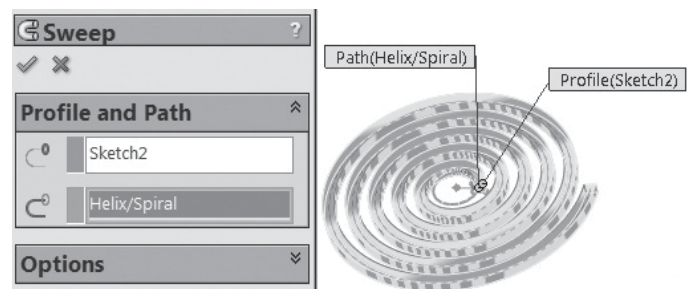
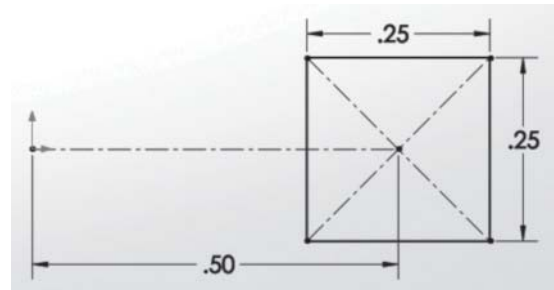


**FIGURE 4.15**  
Spiral spring

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



rectangle as shown (align center with X-axis as shown) > exit sketch.



**Step 2:** Create *Sketch2*: **Front Plane** > **Central Rectangle** on **Sketch** tab > sketch and dimension

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

**HANDS-ON FOR TUTORIAL 4–5.** Change the spiral cross section to a circle with 2.0" diameter. Can you generate the spiral? Why or why not? Explain.



## Tutorial 4–6: Create Features

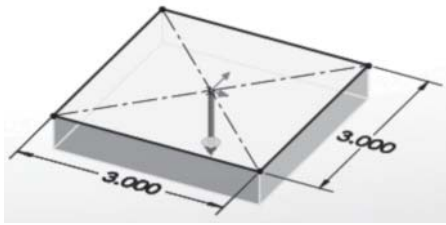
This tutorial covers the creation of these features: chamfer, fillet, slot, shell, draft, and rib. All dimensions are inches. Here are useful observations:

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.

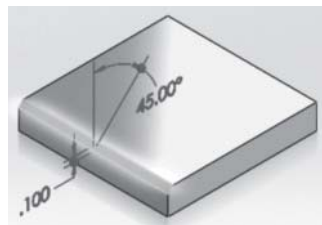
### Step 1:

Create *Sketch1* and *Block* feature: **File** > **New** > **Part** > **OK** > **Top Plane** > **Extruded**

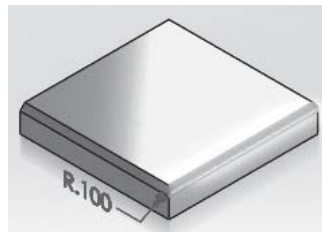
**Boss/Base** on **Features** tab > **Center Rectangle** on **Sketch** tab > click origin and sketch and dimension as shown > exit sketch > reverse extrusion direction > enter 0.5 for thickness **D1** > ✓ > **File** > **Save As** > *tutorial4.6* > **Save**.



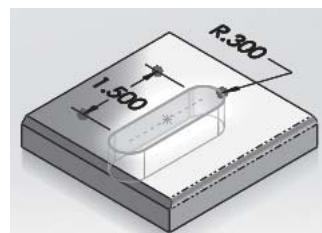
**Step 2:** Chamfer an edge of *Block* feature: **Fillet** dropdown on **Features** tab > **Chamfer** > select *Block* edge shown > use 0.1 for **D** and 45 degrees for angle > ✓.



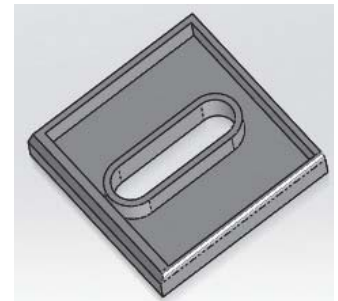
**Step 3:** Fillet an edge of *Block* feature: **Fillet** on **Features** tab > select *Block* edge shown > use 0.1 for **fillet radius** > ✓.



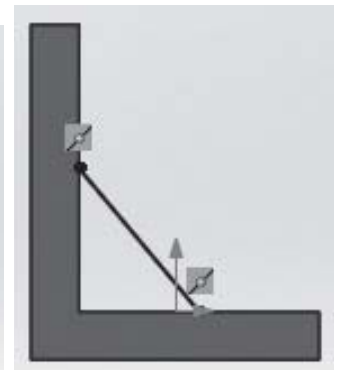
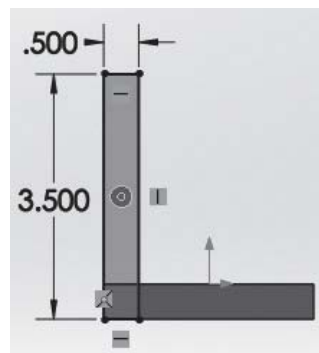
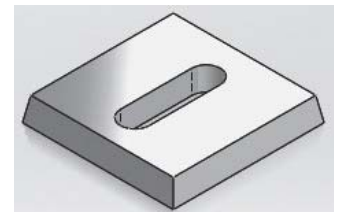
**Step 4:** Create a straight slot in *Block* feature: Select *Block* top face as a sketch plane > **Extruded Cut** on **Features** tab > **Straight Slot** on **Sketch** tab > sketch and dimension slot as shown > exit sketch > **Through All** > ✓.



**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 > ✓.



**Step 7:** Create a rib feature: Delete 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 above > exit sketch > reverse extrusion direction > enter 3.0 for thickness **D1** > ✓ > **Front Plane** > **Rib** on **Features** tab > **Line** on **Sketch** tab > sketch a line using the midpoints of the two edges as shown above > exit sketch > enter 0.5 for thickness **T1** > ✓.



### HANDS-ON FOR TUTORIAL 4-6. Create the following features:

1. Distance-distance chamfer
2. Vertex chamfer
3. Variable size fillet
4. Face fillet
5. Full round fillet
6. 3-point arc slot
7. 3-stepped rib using 3-stepped line as the rib profile

## Tutorial 4-7: Use the Smart Fasteners Wizard

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

In this tutorial, we create an assembly of a block and plate. We 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.

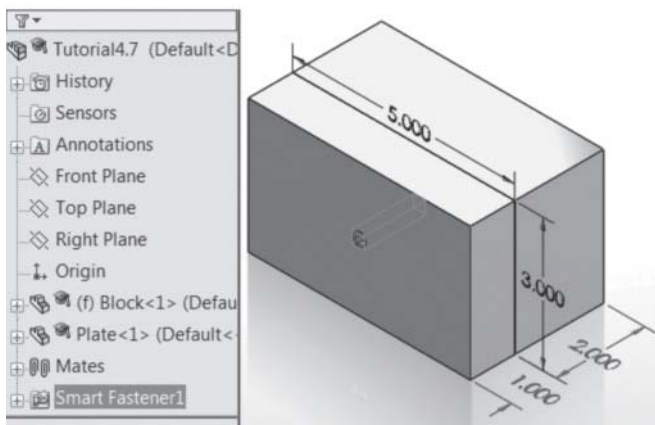
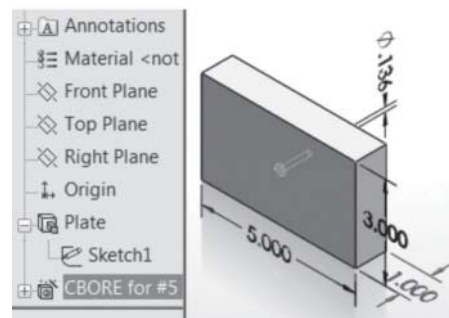


FIGURE 4.16

Assembly using smart fastener

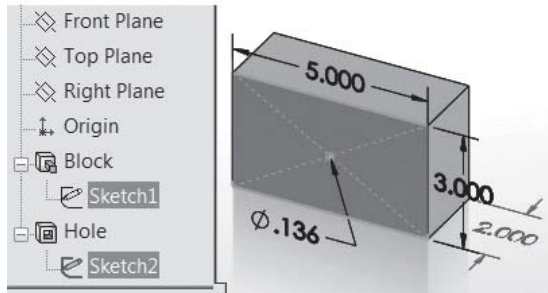
direction > ✓ > **Hole Wizard** on **Features** tab > select **Counterbore** under **Hole Type** (hover over types until you read it) > select #5 for **Size** under **Hole Specifications** > **Positions** tab > click front face of *Plate*, then click origin > ✓ > **File** > **Save As** > *Plate* > **Save**.



**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 thickness **D1** > reverse extrusion

**Note:** The diameter of the counterbore hole shown above corresponds to #5 size. We 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 thickness **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 thickness **D1** > ✓ > **File** > **Save As** > *Block* > **Save**.

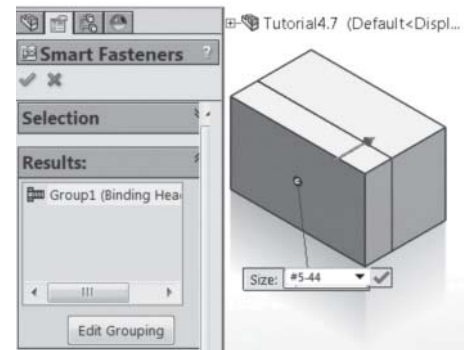


**Step 3:** Create assembly: **File** > **New** > **Assembly** > **OK** > **Browse** > locate *Block* and *Plate* parts > select **Block** + **Ctrl** + **Plate** > **Open** > click ✓ to place *Block* instance in assembly origin > click anywhere in graphics pane to place *Plate* instance > **Mate** on

**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 Browser** > **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.

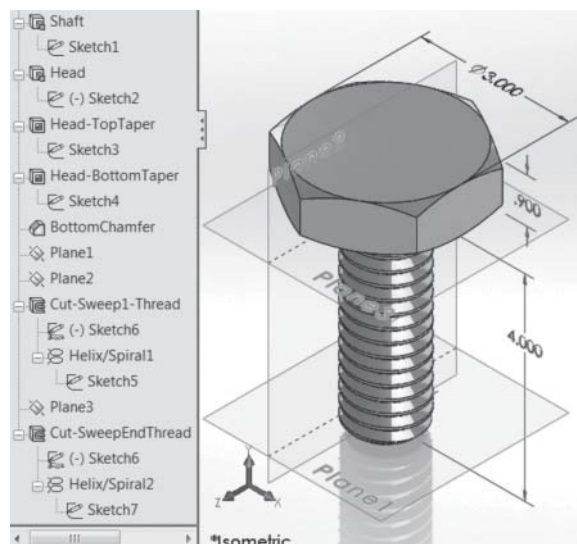
**Step 5:** Add a fastener: **Smart Fasteners** on **Assembly** tab > **OK** (to accept that it may take extra time) > expand features tree > expand *Plate* instance tree node > select *CBORE* for #5 node > **Add** > ✓.



**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.

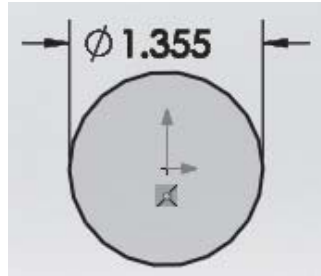
## Tutorial 4-8: Create a Bolt

Bolts, like gears, are an important and essential mechanical element. 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 features tree. All dimensions are in inches.

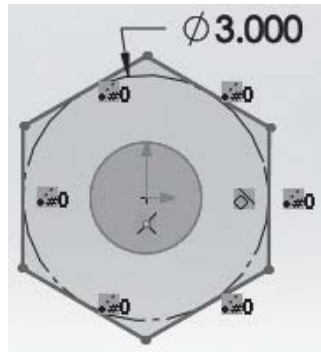


**FIGURE 4.17**  
A bolt

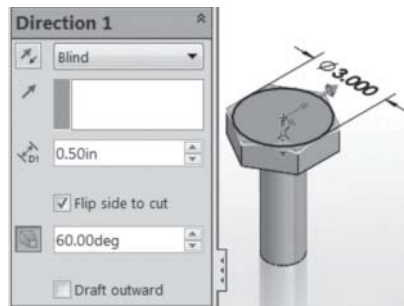
**Step 1:** Create *Sketch1* and *Shaft* feature: **File** > **New** > **Part** > **OK** > **Top Plane** > **Extruded Boss/Base** on **Features** tab > **Circle** on **Sketch** tab > click origin and sketch and dimension as shown > exit sketch > enter 4 for thickness **D1** > reverse extrusion direction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



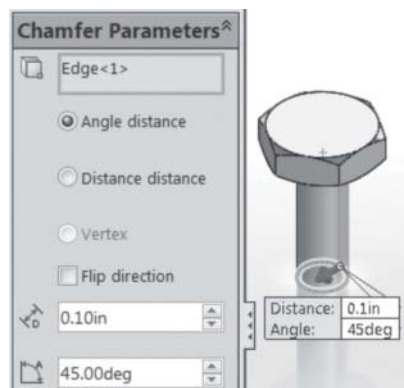
**Step 2:** Create *Sketch2* and *Head* feature: Top face of *Shaft* feature > **Extruded Boss/Base** on **Features** tab > **Polygon** (hexagon icon) on **Sketch** tab > click origin and sketch and dimension as shown > exit sketch > enter 0.9 for thickness **D1** > ✓.



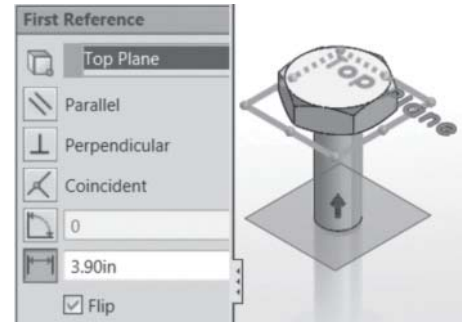
**Step 3:** Create *Sketch3* and *Head-TopTaper* feature: Top face of *Head* feature > **Extruded Cut** > **Circle** on **Sketch** tab > click origin and sketch (make circle tangent to hexagon sides) and dimension as shown > exit sketch > enter 0.5 for thickness **D1** > click checkbox as shown > enter 60 for 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 radius **D** > enter 45 for angle > ✓.

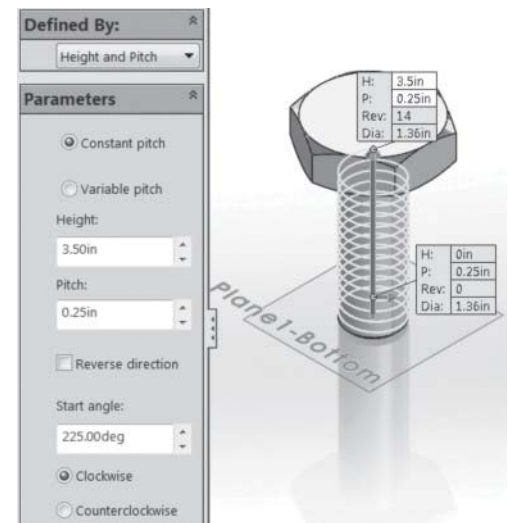


**Step 5:** Create *Plane1*: **Reference Geometry** on **Features** tab > **Plane** > expand features tree > select **Top Plane** > enter 3.9 for distance as shown above > click **Flip** checkbox > ✓.



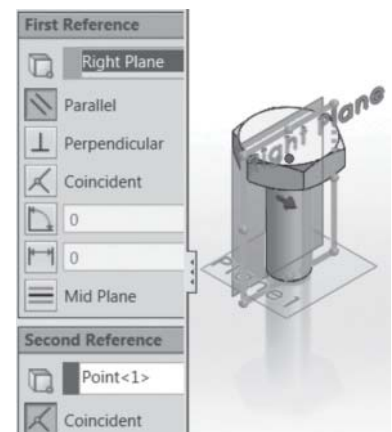
**Note:** We use a distance of 3.9 for *Plane1*, not 4.0, because the chamfer is 0.1 high.

**Step 6:** Create *Sketch5* and *Helix/Spiral1*: select *Plane1* > **Convert Entities** on **Sketch** tab > expand features tree > select *Sketch1* > ✓ > select *Sketch5* > **Insert** > **Curve** > **Helix/Spiral** > enter 3.5 for Height > ✓

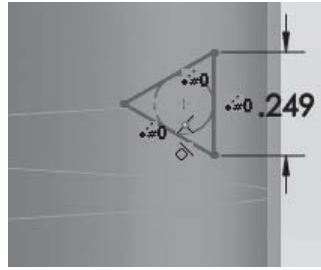


**Note:** The helix 3.5 height is arbitrary. That leaves 0.4 (out of 3.9). We use 0.25 for the other end thread helix and 0.15 underneath the bottom of bolt head as a length with no threads.

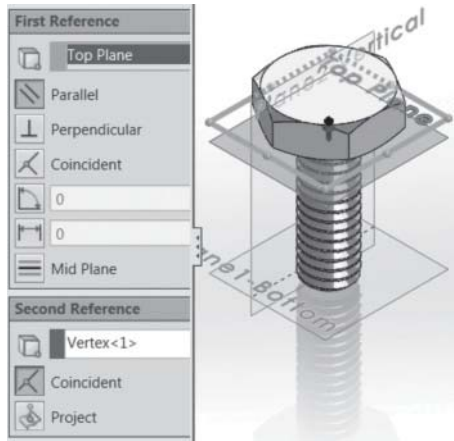
**Step 7:** Create *Plane2*: **Reference Geometry** on **Features** tab > **Plane** > expand features tree > select > **Right Plane** **Parallel** > click **Second Reference** box > select top endpoint of *Helix/Spiral1* > ✓.



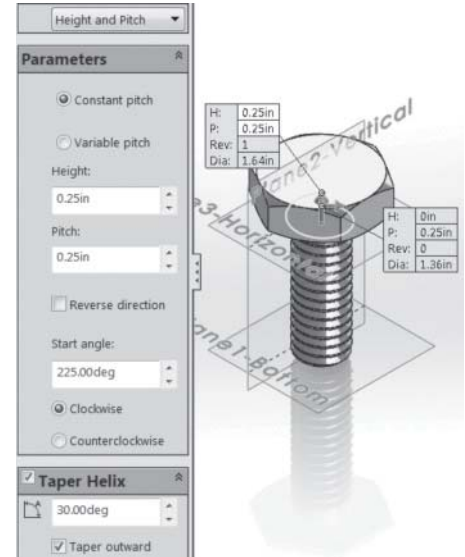
**Step 8:** Create *Sketch6* and *Cut-Sweep1-Thread* feature: *Plane2* > **Swept Cut** on **Features** tab > **Polygon** (hexagon icon) on **Sketch** tab > click near end of helix and sketch and dimension as shown > exit sketch > select *Sketch6* as profile and *Helix/Spiral1* as path > ✓.



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



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



**Step 11:** Create *Cut-SweepEndThread* feature: **Swept Cut** on **Features** tab > select *Sketch6* as profile and *Helix/Spiral2* as path > ✓.