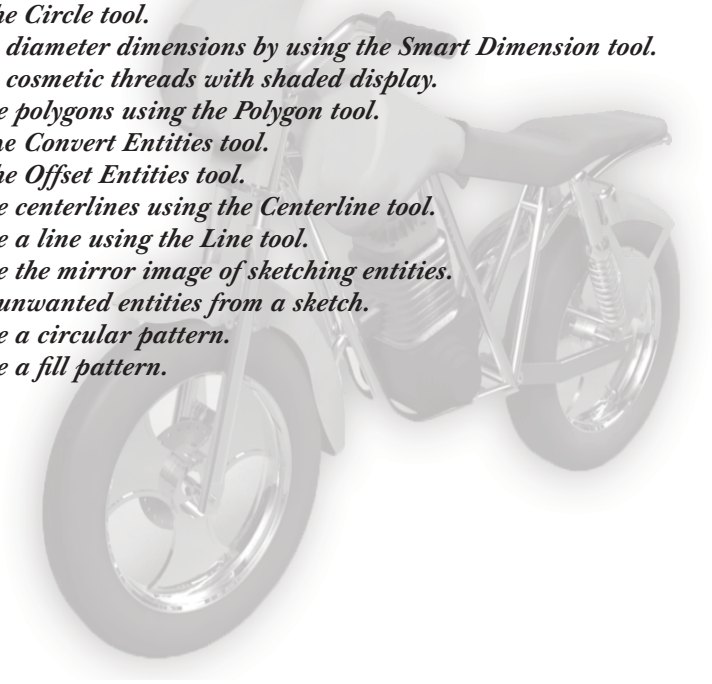
Chapter 2

Creating Axle and Disc Plate

Learning Objectives

After completing this chapter, you will be able to:

- *Use the Extruded Boss/Base tool.*
- *Invoke the Sketching environment.*
- *Use the Circle tool.*
- *Apply diameter dimensions by using the Smart Dimension tool.*
- *Apply cosmetic threads with shaded display.*
- *Create polygons using the Polygon tool.*
- *Use the Convert Entities tool.*
- *Use the Offset Entities tool.*
- *Create centerlines using the Centerline tool.*
- *Create a line using the Line tool.*
- *Create the mirror image of sketching entities.*
- *Trim unwanted entities from a sketch.*
- *Create a circular pattern.*
- *Create a fill pattern.*



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CREATING THE FRONT AXLE

Part Description

In this section, you will create the front axle, as shown in Figure 2-1. The features of this Axle will be created by using the **Extruded Boss/Base** tool. The cosmetic threads with shaded display to be created are of Ansi Inch Machine Threads type. The views and dimensions of the axle are given in Figure 2-2. Note that all dimensions are in inches.



Figure 2-1 *Front axle*

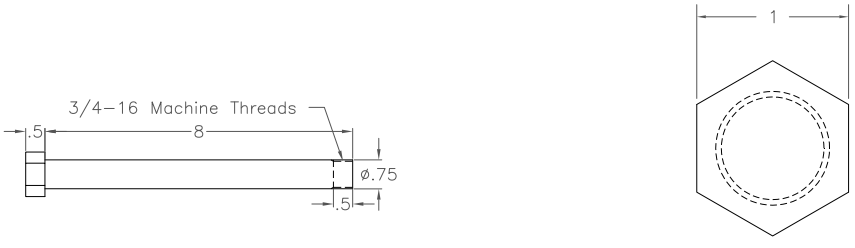

Figure 2-2 *Views and dimensions of the front axle*

Starting SolidWorks and then a New Part Document

Once you have installed SolidWorks, an icon is displayed on the desktop and a folder is added to the **Start** menu.

1. Start SolidWorks by choosing **Start > All Programs > SolidWorks 2011 > SolidWorks 2011** or by double-clicking on the shortcut icon of SolidWorks 2011 displayed on the desktop of your computer; the SolidWorks window is displayed on the screen. Also, the **SolidWorks Resources** task pane is displayed on the right, as shown in Figure 2-3. This window can be used to open a new file or an existing file.

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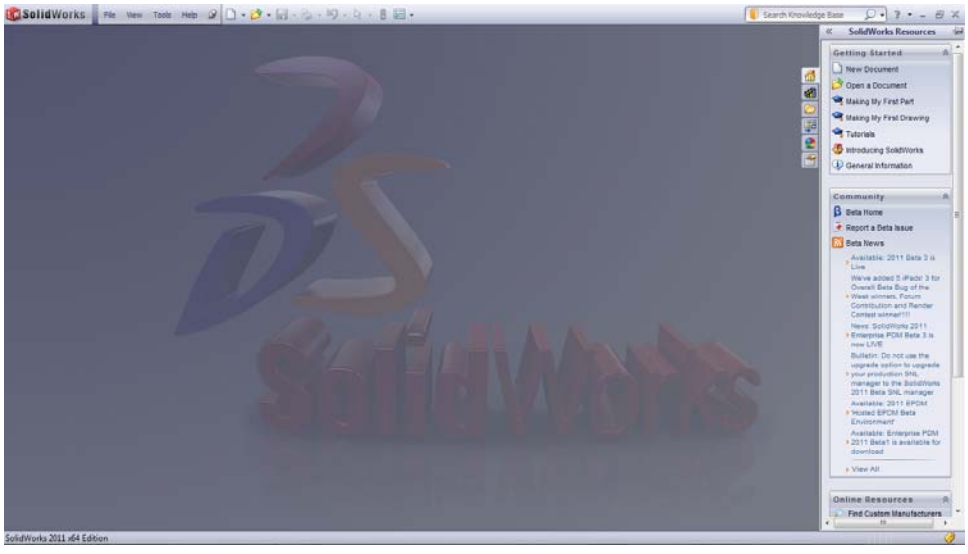

Figure 2-3 *The SolidWorks 2011 window and the* ***SolidWorks Resources*** *task pane*

If the **SolidWorks Resources** task pane is not displayed by default, choose the **SolidWorks Resources** button to display it. This window is used to open online tutorials and to visit the SolidWorks' partner website.

2. In the **Getting Started** area of the **SolidWorks Resources** task pane, choose the **New Document** option to start a new file. Alternatively, choose the **New** button from the Menu Bar; the **New SolidWorks Document** dialog box is displayed, as shown in Figure 2-4.

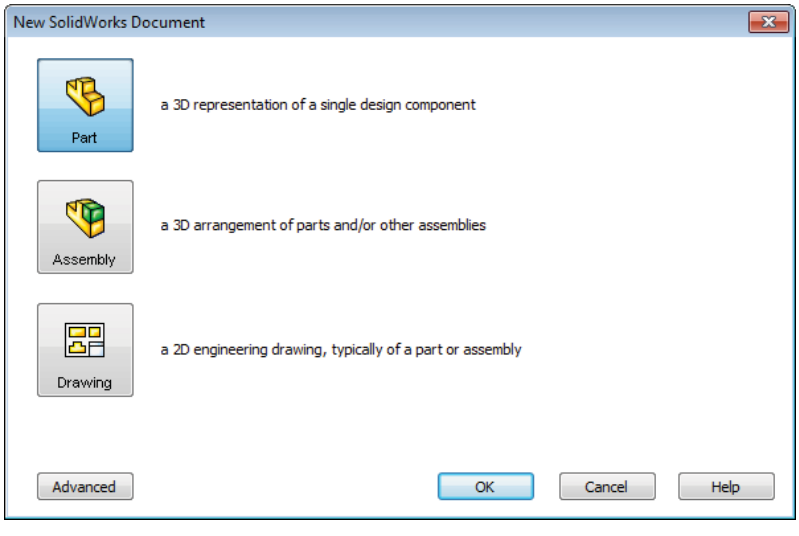

Figure 2-4 *The* ***New SolidWorks Document*** *dialog box*

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3. In the **New SolidWorks Document** dialog box, the **Part** button is chosen by default. Choose the **OK** button; a new SolidWorks part document gets started. Modify the unit of the current document to inch, if it is not set by default.



Note

*As it is already mentioned in the part description that all dimensions are in inches. Therefore, you may need to modify the units used in the current session to inches. To do so, choose the **Options** button from the Menu Bar; the **System Options** dialog box is displayed. Next, choose the **Document Properties** tab. In this tab, select the **Units** option from the area on the left of the dialog box to display the options related to it. The default unit system that was selected while installing SolidWorks is selected in the **Unit system** area of the dialog box. Select the **IPS (inch, pound, second)** radio button from the **Unit system** area and then choose the **OK** button from the dialog box.*

After starting a new SolidWorks part document, you can create the model of Axle by using different feature creation tools of SolidWorks.

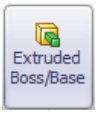
Creating the Base Feature of the Axle

Most of the products designed by using SolidWorks are the combinations of the sketched, placed, and derived features. The base feature is a feature that is created first to make a model in the Part environment. The placed and derived features are created without drawing sketches, whereas the sketched features require a sketch to be drawn first. Generally, the base feature of any design is a sketched feature and is created using a sketch. Therefore, while creating any model, the first and foremost task is to draw a sketch for the base feature. Once you have drawn the sketch, you can convert it into a base feature and then add other sketched, placed, and derived features to it to complete the model.

In this section, the base feature will be created by extruding a sketch using the **Extruded Boss/Base** tool available in the **Features CommandManager**. This tool is used to add material to the area defined by a sketch. To create the base feature, first you need to draw the sketch of the base feature in the sketching environment using the tools available in the **Sketch CommandManager**. After drawing the sketch, you can extrude it using the **Extruded Boss/Base** tool.

1. Choose the **Features** tab from the CommandManager to display the **Features CommandManager**, refer to Figure 2-5.

2. Choose the **Extruded Boss/Base** tool from the **Features CommandManager**; the **Extrude PropertyManager** is displayed on the left in the drawing area, see Figure 2-6. Also, three default planes are displayed at the center of the drawing area and you are prompted to select the sketching plane.



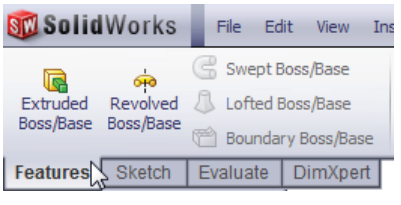


Figure 2-5** The **Features CommandManager

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The default planes are called datum planes and are used to draw 2D sketches. These 2D sketches can be further converted into 3D models by extrusion. The datum planes are mutually perpendicular to each other. You can select any of these datum planes as a sketching plane. The sketching plane is a plane on which the sketch of the feature will be drawn. You can also create a plane other than the default planes by using the **Plane PropertyManager**. You will learn more about the **Plane PropertyManager** in the later chapters.

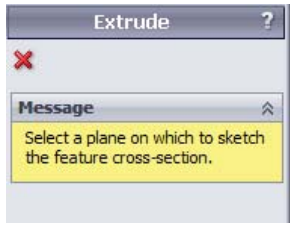

Figure 2-6 *The* ***Extrude PropertyManager***

3. Next, select the **Front Plane** as the sketching plane in the drawing area; the sketching environment is invoked and the selected plane is oriented normal to the view. By default, there are two arrows at the origin that indicate the X and Y axis directions of the current sketching plane. Also, a confirmation corner is displayed with the **Exit Sketch** and **Cancel Sketch** options at the upper right corner of the drawing area, refer to Figure 2-7.

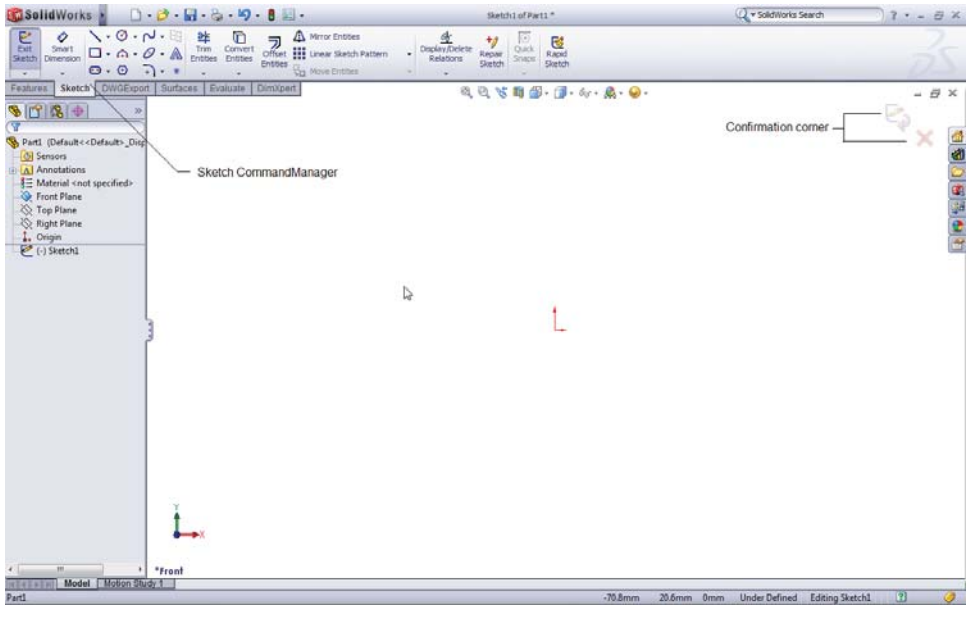

Figure 2-7 *The screen display in the sketching environment*

4. In the **Sketch CommandManager**, choose the **Circle** button; the **Circle PropertyManager** is displayed on the left in the drawing area, as shown in Figure 2-8. Also, the cursor is replaced by the circle cursor.



In the **Circle PropertyManager**, the **Circle** button is chosen by default in the **Circle Type** rollout. As a result, you can draw a circle by specifying its center and radius.

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5. Move the circle cursor toward the origin and click when the orange circle and the symbol of coincident relation are displayed; the center point of the circle is specified. You will learn more about relations later in this chapter. As soon as you specify the center point of the circle, the dynamic preview of the circle is displayed on the screen.

The origin of the sketching environment screen is represented by a red colored point at the center of the screen. The coordinates of this point are 0,0. You can hide or display the origin by choosing the **View Origins** button from the **Hide/Show Items** flyout in the **View (Heads-up)** toolbar, refer to Figure 2-9.

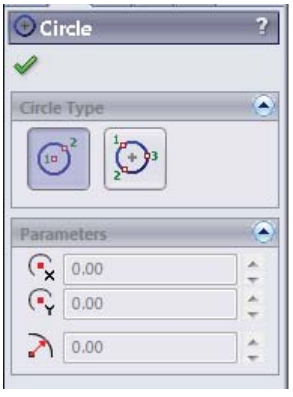

*Figure 2-8 The **Circle PropertyManager***

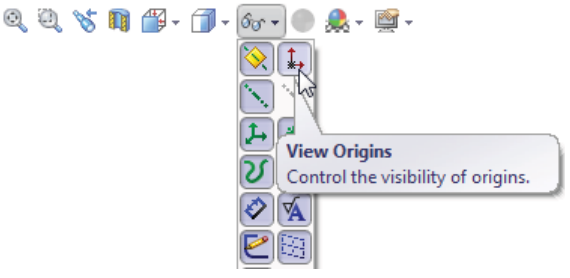

*Figure 2-9 The **Hide/Show Items** flyout*

After specifying the center point of the circle, if you move the cursor to define its radius, the current radius of the circle is displayed above the circle cursor. This radius changes as you move the circle cursor. To specify the radius of the circle, you need to click in the drawing area.

6. Move the circle cursor toward the right and click when 0.375 is displayed above the circle cursor.

Tip. *After specifying the center point and radius of the circle, you can modify their parameters in the **Parameters** rollout of the **Circle PropertyManager**.*

Note

If a sketch is displayed in blue, it indicates that the sketch is underdefined. An underdefined sketch is the one in which some of the dimensions or relations are not defined and the degrees of freedom of the sketch are not fully constrained. In these types of sketches, the entities may move or change their size unexpectedly. The sketched entities of the underdefined sketch are displayed in blue. In SolidWorks, there are six states in which a sketch may exist. These states are fully defined, overdefined, underdefined, dangling, no solution found, and invalid solution found.

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Tip. *The **Sketch Definition** area of the status bar, which is located at the bottom of the drawing window, always displays the status of the sketch, dimension of the sketch, and its relation. If the sketch is underdefined, the status area will display **Under Defined**; if the sketch is overdefined, the status area will display **Over Defined**; and if the sketch is fully defined, the message displayed in the status area will be **Fully Defined**.*

After drawing the sketch, dimensioning is the most important step for creating a design. SolidWorks is a parametric software. This property of SolidWorks ensures that irrespective of the original size, the selected entity is driven by the specified dimension value. Therefore, when you apply dimension to an entity or modify it, the sketch is forced to change its size in accordance with the specified dimension value.

7. Choose the **Smart Dimension** button from the **Sketch CommandManager** to dimension the circle; the cursor gets converted into the dimension cursor. Select the circle; the current diameter of the circle is attached to the cursor. Specify location in the drawing area for placing the diameter of the circle. As soon as you place the diameter, the **Modify** dialog box is displayed, as shown in Figure 2-10.

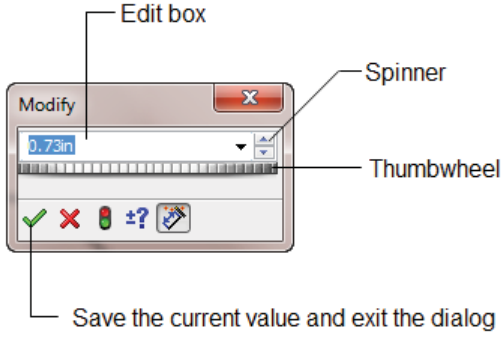

Figure 2-10 *The **Modify** dialog box*

The **Smart Dimension** tool is used to dimension any kind of entity. If you use this tool, the type of dimension applied will depend on the type of entity selected.

The **Modify** dialog box is used to modify the default dimension values. In this dialog box, you can either use the spinner or enter a new value in the edit box. You can also change the value by dragging the thumbwheel located below the spinner to left or right, as required.

8. In this dialog box, enter **0.75** and choose the **Save the current value and exit the dialog** button from this dialog box. Figure 2-11 shows a circle of diameter 0.75 inch.

Note

After applying the diameter dimension to the sketch, the blue colored sketch changes to black. This indicates that the sketch is now fully defined.

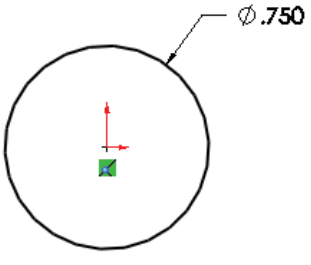

Figure 2-11 *Circle of diameter 0.75in*

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A fully-defined sketch is the one in which all entities of the sketch and their positions are fully-defined by the relations or dimensions, or both. In a fully-defined sketch, all degrees of freedom of a sketch are constrained. As a result, the sketched entities cannot change their size and location unexpectedly. If a sketch is not fully-defined, its size or position can change at any time during the design because all its degrees of freedom are not fully constrained. All entities in a fully-defined sketch are displayed in black.

9. Next, exit the sketching environment by clicking on the confirmation corner available on the upper right corner of the drawing area; the **Boss-Extrude PropertyManager** is displayed on the left in the drawing area, refer to Figure 2-12. You will notice that the default view is automatically changed to the trimetric view and the preview of the base feature is displayed on the screen. Also, a 3D arrow will be displayed in the preview, as shown in Figure 2-13.

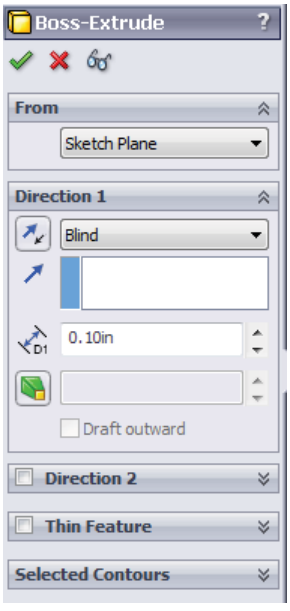

Figure 2-12** The **Boss-Extrude PropertyManager

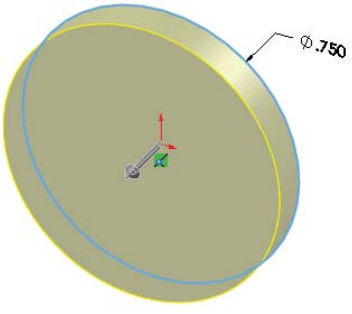

***Figure 2-13** Preview of the base feature with arrow*

The **Direction 1** and **Direction 2** rollouts in the **Boss-Extrude PropertyManager** are used to specify the end conditions for extruding the sketch in the first and second directions, respectively. The **Direction 1** rollout is expanded by default and to expand the **Direction 2** rollout, you need to select the check box available on the title bar of the rollout. The options in these rollouts are same. You will learn more about these options later.

10. Select the **Mid Plane** option from the **End Condition** drop-down list of the **Direction 1** rollout in the **Boss-Extrude PropertyManager**.

Tip. *You can also choose the **Mid Plane** option from the shortcut menu displayed on right-clicking in the graphics window when the **Boss-Extrude PropertyManager** is active.*

The **End Condition** drop-down list is used to define the termination condition of an extruded feature. The **Mid Plane** option of this drop-down list is used to create the feature by extruding a sketch equally in both the directions of the plane on which the sketch is drawn. Note that when you create the first feature, some of the options of this drop-down list will not be available. The other options available in this drop-down list will be discussed later in this chapter.

11. To specify the depth of extrusion, set **8"** in the **Depth** spinner of the **Direction 1** rollout.



Tip. *You can also specify the depth of extrusion by dragging the arrow displayed on the preview of the model.*

12. Choose the **OK** button ( ) from the **Boss-Extrude PropertyManager** to exit.

13. Choose the **Zoom to Fit** button from the **View (Heads-up)** toolbar to fit the feature into the screen.



The **Zoom to Fit** tool is used to increase or decrease the drawing display area so that all entities are properly fit in the current view. You can also invoke this tool by pressing the F key or by double-clicking on the middle mouse button in the drawing area.

14. Choose the **View Orientation** button from the **View (Heads-up)** toolbar; the **View Orientation** flyout is displayed, as shown in Figure 2-14.

15. Choose the **Isometric** button from this flyout; the current orientation of the model changes to isometric. The model after creating the base feature is shown in Figure 2-15.

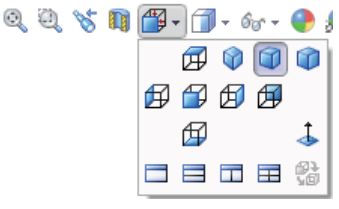

Figure 2-14 *The* ***View Orientation*** *flyout*

In SolidWorks, you can change the view orientation manually by using some predefined standard views or user-defined views. The views available in the **View Orientation** flyout are called standard views. You will learn more about the user-defined views and the standard views in later chapters.

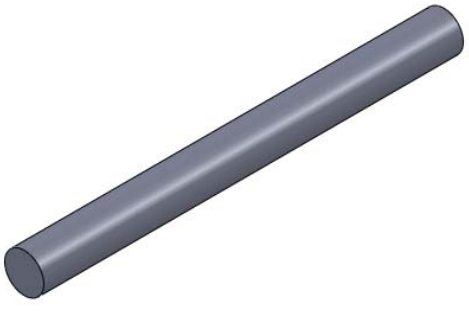

Figure 2-15 *Base feature of the model*

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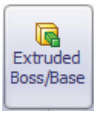
Creating the Second Feature

The second feature of this model is also an extrude feature. This feature will be created by using the **Extruded Boss/Base** tool available in the **Features CommandManager**. The sketch of this extrude feature needs to be created on the front planar circular face of the base feature.

1. Choose the **Extruded Boss/Base** button from the **Features CommandManager**; the **Extrude PropertyManager** is displayed on the left in the drawing area and you are prompted to select the sketching plane.



2. Select the front planar circular face of the base feature as the sketching plane, refer to Figure 2-16; the sketching environment is invoked.

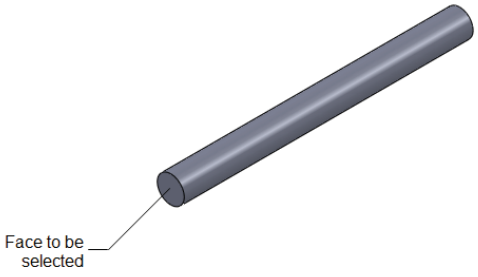


Figure 2-16 *Face to be selected*

Next, you need to change the current orientation of the feature normal to the screen.

3. Choose the **Normal To** button from the **View Orientation** flyout in the **View (Head-up)** toolbar; the current orientation changes to normal.



The **Normal To** tool is used to reorient the view normal to a selected face or plane.

4. Choose the **Polygon** tool from the **Sketch CommandManager**; the **Polygon PropertyManager** is displayed on the left in the drawing area and the cursor changes into a polygon cursor. Make sure the value in the **Number of Sides** spinner is set to **6** and the **Inscribed circle** radio button is selected in the **Parameters** rollout of the PropertyManager.



The **Polygon** tool is used to draw a regular polygon. A regular polygon is defined as a multisided geometric figure in which the length of all sides and the angle between them are the same. In SolidWorks, you can draw a regular polygon with the number of sides ranging from 3 to 40. The dimensions of the polygon are controlled by using the diameter of a construction circle that is either inscribed in a polygon or circumscribes it. If the **Inscribed circle** radio button is selected in the **Parameters** rollout, the construction circle will be inscribed in the polygon and the diameters of the construction circle will be determined by the edges of the polygon. However, if the **Circumscribed circle** radio button is selected in the **Parameters** rollout, the construction circle will circumscribe around the polygon and the diameters of the construction circle will be determined by the vertices of the polygon.

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5. Move the polygon cursor toward the center of the circular face of the base feature and click to specify a point when the origin gets highlighted and the symbol of coincident relation is displayed below the polygon cursor, refer to Figure 2-17.

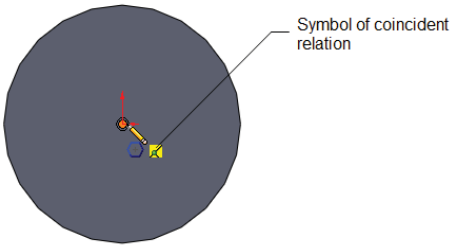

Figure 2-17 *Symbol of coincident relation*

6. Next, move the circle cursor vertically upward and click when the preview of the polygon crosses the circular edge of the base feature. Next, press the ESC key.

7. Choose the **Smart Dimension** button from the **Sketch CommandManager**; the cursor modifies into a dimension cursor. Select the construction circle that is inscribed in the polygon; the current diameter of the construction circle is attached to the cursor. In the drawing area, specify the location to place the diameter of the construction circle. As soon as you place the diameter, the **Modify** dialog box is displayed.



8. Enter **1** in the **Modify** dialog box and choose the **Save the current value and exit the dialog** button. Next, press the ESC key to exit the **Smart Dimension** tool.

9. Select a vertical edge of the polygon and do not move the mouse; a pop-up toolbar is displayed, as shown in Figure 2-18.



Figure 2-18 *Pop-up toolbar*

Note

The options in the pop-up toolbar will vary based on the entity or feature selected. Also, the toolbar will disappear if you move the cursor away from the selected entity or feature.

10. Choose the **Make Vertical** option from the pop-up toolbar, refer to Figure 2-18; the vertical relation is applied to the selected entity and the color of the sketch is changed to black, indicating that the sketch is now fully-defined, refer to Figure 2-19.

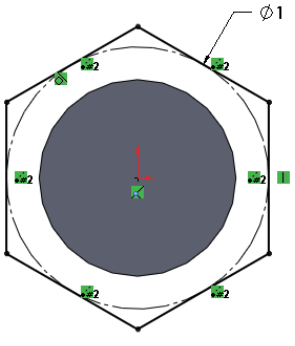

Figure 2-19 *Fully-defined sketch*

The vertical relation is used to force one or more selected lines or centerlines to become vertical. You can also force two or more points to become vertical by using this relation.

Note

Figure 2-19 shows all the relations applied to the sketch. You can hide these relations by choosing the ***View Sketch Relations*** *button from the* ***Hide/Show Items*** *flyout in the* ***View (Heads-up)*** *toolbar. This is a toggle button.*

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11. Exit the sketching environment by clicking on the confirmation corner available at the upper right side of the drawing area; the **Boss-Extrude PropertyManager** is displayed.

12. In the **Direction 1** rollout of the **Boss-Extrude PropertyManager**, set the value of the **Depth** spinner to **0.5**. Note that the **Blind** option is selected in the **End Condition** drop-down list by default, enabling you to enter the value of depth.

The **Blind** option is used to define the point of termination of an extruded base feature by specifying the depth of extrusion in the **Depth** spinner.

13. Choose the **OK** button ( ) from the **Boss-Extrude PropertyManager**; the extrude feature is created. Next, change the current orientation of the model to isometric by choosing the **Isometric** button from the **View Orientation** flyout. Figure 2-20 shows the isometric view of the model.

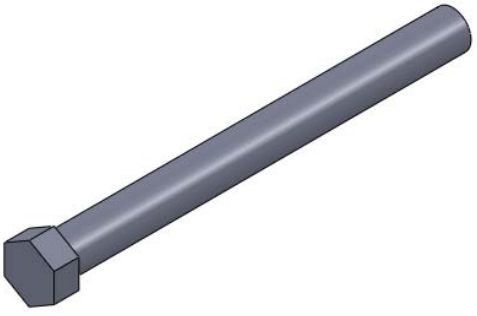

Figure 2-20 *Isometric view of the model*

Applying Cosmetic Threads

Cosmetic threads are the schematic representation of threads. It is evident from Figures 2-1 and 2-2 that cosmetic threads need to be applied to the model. You can apply the cosmetic threads by using the **Cosmetic Thread PropertyManager**.

1. Choose **Insert > Annotations > Cosmetic Thread** from the SolidWorks menus; the **Cosmetic Thread PropertyManager** is displayed, as shown in Figure 2-21. Also, you are prompted to select the edges for starting the creation of threads and then set their parameters.

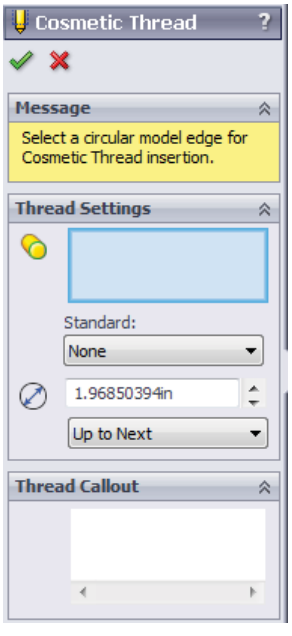

Figure 2-21 *The* ***Cosmetic Thread PropertyManager***

Note

You can create threads on a model as a solid feature. But it is recommended to create the cosmetic threads since the solid threads can be cumbersome to create and can significantly slow down the performance of your computer. Also, since the cosmetic threads are as per industrial standards, there is no need to specify the thread profiles while creating them.

2. Select the circular edge of the model, refer to Figure 2-22. The name of the selected edge is displayed in the **Circular Edges** selection area of the **Thread Settings** rollout. Also, the schematic representation of the threads is displayed in the drawing area.

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3. Select the **Ansi Inch** option from the **Standard** drop-down list in the **Thread Settings** rollout.

The **Standard** drop-down list of the **Cosmetic Thread PropertyManager** is used to specify the industrial thread standards. In this drop-down list, the **None** option is selected by default. This allows the user to have user-defined specification of the thread. The thread standards available in this drop-down list are **Ansi Inch**, **Ansi Metric**, **AS**, **BSI**, **DIN**, **GB**, **IS**, **ISO**, **JIS**, and **KS**.

4. Select the **Machine Threads** and **3/4-16** options from the **Type** and **Size** drop-down lists in the **Thread Settings** rollout, respectively.

5. Select the **Blind** option from the **End Condition** drop-down list in the **Thread Settings** rollout of the **Cosmetic Thread PropertyManager**.

6. Set **0.5** in the **Depth** spinner.

7. Choose the **OK** button ( ) from the **Cosmetic Thread PropertyManager**; the cosmetic thread is added to the model.

   Next, you need to display the cosmetic thread with shaded thread pattern on the surface.

8. Choose the **Options** button from the Menu Bar; the **System Options - General** dialog box is displayed.

9. In this dialog box, choose the **Document Properties** tab; the **System Option - General** dialog box is replaced with the **Document Properties - Drafting Standard** dialog box.

10. Select the **Detailing** option from this dialog box; the corresponding options are displayed on the right of the dialog box. Select the **Shaded cosmetic threads** check box and then choose the **OK** button; the cosmetic thread is displayed in the drawing area with the shaded thread. Figure 2-23 shows the final model after adding the shaded cosmetic threads.

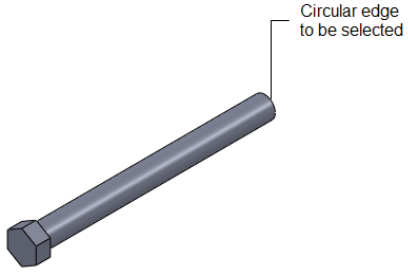

Figure 2-22 *Circular edge to be selected*



Figure 2-23 *Final model of Axle*

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Saving the Model

After creating the model, you need to save it at a particular location. It is recommended that you create a separate folder for saving all parts to be created in this book. When you invoke the option to save the document, the default folder *\Documents* will be displayed. You need to create a folder with the name *Motor Cycle Project* in the *Documents* folder and then save all parts in this folder. The steps to save this part are given below:

1. Choose the **Save** button from the Menu Bar; the **Save As** dialog box is displayed. Browse to the *Documents* folder and create a new folder with the name *Motor Cycle Project* in it.



2. Double-click on the newly created folder and then enter **Front Axle** as the name of the part in the **File name** edit box. Next, choose the **Save** button; the document is saved at *\Documents\Motor Cycle Project*.

3. Close the document by choosing **File > Close** from the SolidWorks menus.

CREATING THE DISC PLATE

Part Description

In this section, you will create the Disc Plate, as shown in Figure 2-24, by using different feature creation tools of SolidWorks. The first feature of the Disc Plate will be created by using the **Extruded Boss/Base** tool. Then, the other features of the Disc Plate will be created by using the **Extruded Cut**, **Circular Pattern**, and **Fill Pattern** tools. The dimensions of the Disc Plate are shown in Figure 2-25.

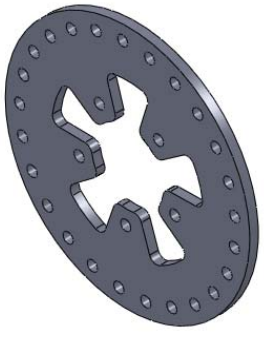


Figure 2-24 *The Disc Plate*

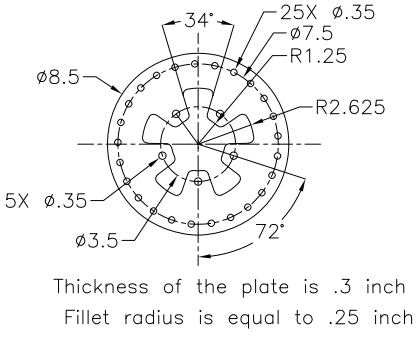


Figure 2-25 *Dimensions of the Disc Plate*

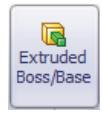
Starting a New Part Document

1. Choose the **New** button from the Menu Bar; the **New SolidWorks Document** dialog box is displayed in which the **Part** button is chosen by default. Choose the **OK** button; a new part document is started.

Creating the Base Feature of the Disc Plate

In this section, you need to create the base feature of the model which is an extruded feature.

1. Choose the **Extruded Boss/Base** button from the **Features CommandManager**; the **Extrude PropertyManager** is displayed and you are prompted to select the sketching plane.



2. Select the **Front Plane** as the sketching plane from the drawing area; the sketching environment is invoked and the selected plane is oriented normal to the view.

3. Choose the **Circle** button from the **Sketch CommandManager**; the **Circle PropertyManager** is displayed. Also, the cursor is converted into a circle cursor.



4. Move the circle cursor close to the origin and press the left mouse button when an orange circle is displayed.

5. Next, move the cursor toward the right and click when the radius of the circle above the circle cursor shows 1.25.

6. Similarly, draw a circle of radius 4.25 units. Note that the center points of both the circles are at the origin. The sketch after drawing the circles should look similar to the one shown in Figure 2-26.

   After drawing the circles, you need to apply diameter dimensions to it using the **Smart Diameter** tool.

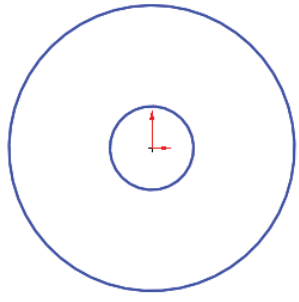


Figure 2-26 *Sketch after drawing the circles*

7. Choose the **Smart Dimension** button from the **Sketch CommandManager**; the cursor is converted into a dimension cursor.



8. Move the dimension cursor to the inner circle; the circle is highlighted. Select the highlighted circle; a diameter dimension is attached to the cursor.

9. Move the cursor toward the right and click to place the diameter dimension; the **Modify** dialog box is displayed.



Note

*The **Modify** dialog box will appear automatically only if the **Input dimension value** check box is selected in the **System Options - General** dialog box. To display the **Modify** dialog box, invoke the **System Options - General** dialog box by choosing the **Options** button from the Menu Bar and then select the **Input dimension value** check box in it.*

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10. Enter **2.5** as the diameter value in the **Modify** dialog box and then press ENTER; the diameter of the circle gets modified to 2.5 inches.

11. Similarly, apply the diameter dimension to the outer circle and modify it to 8.5 units, as shown in Figure 2-27.

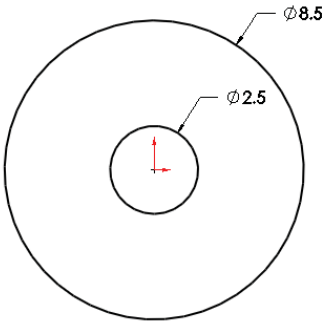

Figure 2-27 *Sketch after applying the diameter dimensions*

12. Click on the confirmation corner available at the upper right of the drawing area; the **Boss-Extrude PropertyManager** is displayed. You will notice that the view is automatically changed to the trimetric view and the preview of the base feature is displayed in the drawing area. Also, an arrow is displayed in front of the sketch, as shown in Figure 2-28.

13. Select the **Mid Plane** option from the **End Condition** drop-down list of the **Direction 1** rollout in the **Boss-Extrude PropertyManager**.

As discussed earlier, the **End Condition** drop-down list provides the options to define the termination point of an extruded feature. The **Mid Plane** option available in the **End Condition** drop-down list is used to create a feature by extruding a sketch equally in both directions of the plane on which the sketch is drawn.

14. Set **0.3** in the **Depth** spinner of the **Direction 1** rollout in the PropertyManager.

    You have selected the **Mid Plane** option as the end condition of the feature. Therefore, the feature will extrude 0.15 inch toward the front of the sketching plane and 0.15 inch toward the back.

15. Choose the **OK** button ( ) from the **Boss-Extrude PropertyManager**; the base feature is created, as shown in Figure 2-29.

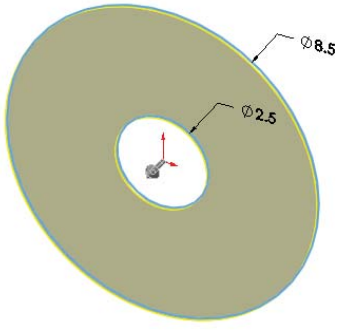

Figure 2-28 *Preview of the base feature*

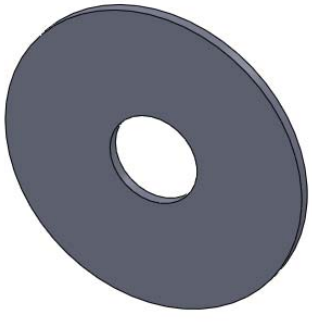

Figure 2-29 *Base feature of the model*

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Creating the Second Feature

The second feature of the model is an extrude cut feature. It will be created by using the **Extruded Cut** tool.

1. Choose the **Extruded Cut** button from the **Features CommandManager**; the **Extrude PropertyManager** is displayed and you are prompted to select the sketching plane.

2. Select the front planar face of the base feature as the sketching plane; the sketching environment is invoked.

3. Choose the **Normal To** button from the **View Orientation** flyout in the **View (Head-up)** toolbar; the current orientation of the feature is changed normal to the screen.



4. Move the cursor over the inner circular edge of the base feature and select it when it is highlighted.

5. Choose the **Convert Entities** button from the **Sketch CommandManager**; a sketch is created by projecting the selected edge into the current sketching plane, refer to Figure 2-30. Next, press the ESC Key to exit the tool.



The **Convert Entities** tool is used to create one or more sketch curves by projecting an edge, loop, face, curve, and so on onto the current sketching plane.



Tip. *You can select multiple entities to create a sketch by projecting them on the current sketching plane. You can also select entities after invoking the* ***Convert Entities*** *tool.*



Note

When you create a sketch curve by using the ***Convert Entities*** *tool, the* ***On Edge*** *and* ***Fixed*** *relations are applied automatically to the sketch curve. The* ***On Edge*** *relation is applied between the sketch curve created and the selected entity. This causes the sketch curve to get updated if the entity is changed. The* ***Fixed*** *relation is applied internally on the endpoints of the sketch curve so that the sketch remains in a fully-defined state.*

6. Choose the **Offset Entities** button from the **Sketch CommandManager**; the **Offset Entities PropertyManager** is displayed.



The **Offset Entities** tool is used to draw parallel lines or concentric arcs and circles. You can offset the selected entities, edges, loops, and curves. You can also select the parabolic curves, ellipses, and elliptical arcs for offset.

7. Set **1.375** in the **Offset Distance** spinner of the **Parameters** rollout in the PropertyManager.

The **Offset Distance** spinner is used to set the distance to which the selected entity needs to be offset.

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Tip. *You can also specify the offset distance of the entity by dragging the preview of the offset entity in the drawing area.*

8. Accept the other default options in the PropertyManager and select the curve from the drawing area; the preview of the offset curve is displayed. Also, an arrow is displayed in the direction pointing outward. If the direction of arrow is inward then you need to reverse the direction of arrow by selecting the **Reverse** check box from the **Offset Entities PropertyManager**.

9. Choose the **OK** button ( ) from the PropertyManager; a new sketch entity is created at an offset distance of 1.375 from the selected sketch entity, as show in Figure 2-30.

10. Choose the **Centerline** button from the **Line** flyout in the **Sketch CommandManager**, refer to Figure 2-31; the **Insert Line PropertyManager** is displayed.

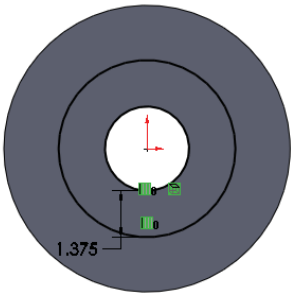

Figure 2-30 *Sketch created by using the* ***Convert Entities*** *and* ***Offset Entities*** *tools*

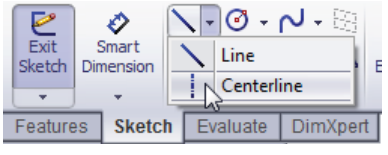

Figure 2-31 *Choosing the* ***Centerline*** *button from the* ***Line*** *flyout*

The **Centerline** tool is used to draw construction lines or centerlines. These lines help in drawing sketches but are not considered while converting the sketches into features.

11. Specify the origin as the start point of the centerline and move the cursor vertically upward. You will notice that the symbol of the **Vertical** relation ( ) is displayed next to the cursor.

In SolidWorks, the **Horizontal** and **Vertical** relations are applied automatically to the lines, depending upon the direction of movement of the lines. These relations ensure that the lines drawn are vertical or horizontal and not inclined.

12. Specify the endpoint of the centerline by clicking anywhere outside the inner circle, refer to Figure 2-32.

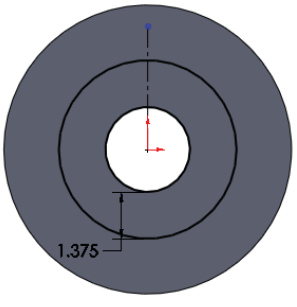

Figure 2-32 *Sketch after drawing the centerline*

Note

*In Figure 2-32, the display of the relation symbol is hidden for the clarity of figure. To view/hide the relations, choose the **View Sketch Relations** button from the **Hide/Show Items** flyout in the **View (Heads-up)** toolbar. This is a toggle button.*

13. Choose the **Line** button from the **Sketch CommandManager**; the **Insert Line PropertyManager** is displayed. Also, the cursor is replaced with the line cursor. You can also invoke the **Line** tool by pressing the L key.

The **Line** tool is used to draw an individual line or a chain of continuous lines. In SolidWorks, while drawing continuous lines using the **Line** tool, you can also draw the tangent or normal arcs that originate from the endpoint of the lines. SolidWorks also allows you to draw lines of infinite length using the **Line** tool.

Tip. *You can also draw a line by using the **Centerline** tool. To do so, choose the **Centerline** tool from the **Line** flyout; the **Insert Line PropertyManager** is displayed. In this PropertyManager, clear the **For construction** check box in the **Options** rollout. Similarly, you can draw a centerline by using the **Line** tool.*

14. Move the line cursor toward the origin and click when an orange circle is displayed at the origin; the start point of the line is specified and the **Line Properties PropertyManager** is displayed. However, the options of this PropertyManager are not available at this stage.

15. Move the line cursor such that the line is drawn at an angle of 72-degree from the horizontal axis and the symbol of coincident relation is displayed when the line touches the outer circle of sketch. This angle can be checked from the **Angle** spinner in the **Parameters** rollout of the **Line Properties PropertyManager**.

16. Press the left mouse button at this location to specify the endpoint of the line. Note that the **Line** tool is still active and the endpoint of the line drawn is automatically selected as the start point of the next line. This is a continuous process and you can draw a chain of many continuous lines by specifying points on the screen using the left mouse button.

17. Press the ESC key to exit the **Line** tool. Alternatively, double-click on the screen or right-click to display the shortcut menu and then choose the **End chain** or **Select** option from it to exit the **Line** tool.

    After drawing the line, you will mirror it about the vertical centerline using the **Mirror** tool.

18. Choose the **Mirror Entities** button from the **Sketch CommandManager**; the **Mirror PropertyManager** is displayed.

The **Mirror** tool is used to create the mirror image of the selected entities. Note that when you create a mirror entity, SolidWorks applies the symmetric relation between the selected entities. Also, if you change an existing entity, its mirror entity will also change automatically.

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19. Select the line drawn in Step 16 from the drawing area; the line is highlighted in blue and its name is displayed in the **Entities to mirror** selection area of the PropertyManager.

The **Entities to mirror** selection area of the PropertyManager displays the name of the entities to be selected for mirroring.

20. Click on the **Mirror about** selection area of the PropertyManager to activate its selection mode.

The **Mirror about** selection area of the PropertyManager allows you to select an entity that will be used as a mirroring line.

21. Select the vertical centerline from the drawing area as a mirroring line; the preview of the mirror entity is displayed in the drawing area. Make sure the **Copy** check box is selected in the PropertyManager.

The **Copy** check box is used to remove or retain the selected parent entities while mirroring them. For example, if you clear this check box, then while mirroring the sketched entities, the selected parent entities will be removed and only the mirror image will be retained.

22. Choose the **OK** button ( ) from the **Mirror PropertyManager**; the mirror image of the selected entity is created, as shown in Figure 2-33.

Next, you need to trim the unwanted entities in a sketch using the **Trim Entities** tool.

23. Choose the **Trim Entities** button from the **Sketch CommandManager**; the **Trim PropertyManager** is displayed.



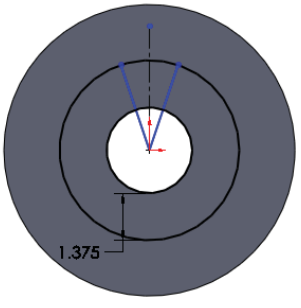

Figure 2-33 *Sketch after mirroring the line*

The **Trim Entities** tool is used to trim the unwanted entities from a sketch. These entities can be lines, arcs, ellipses, parabolas, circles, splines, or centerlines.

24. Choose the **Trim to closest** button from the **Options** rollout of the **Trim PropertyManager**; the cursor is replaced by the trim cursor.



The **Trim to closest** button is used to trim a selected entity at the intersection which is closest to it.

25. Move the trim cursor closer to the entity inside the smallest circle of the sketch and click when the entity is highlighted; the entity is trimmed. Next, move the trim cursor closer to the next entity that is inside the smallest circle of the sketch and click when the entity is highlighted; the entity is trimmed, refer to Figure 2-34.

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26. Similarly, trim other entities of the sketch and press the ESC key to exit the **Trim Entities** tool. Figure 2-35 shows the resultant sketch after trimming all unwanted entities from the sketch.

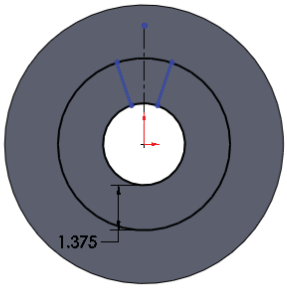

Figure 2-34 *Sketch after trimming the entities*

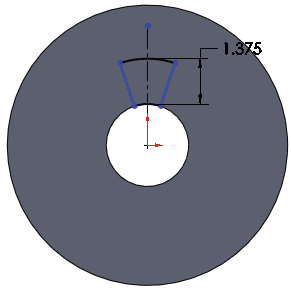

Figure 2-35 *Resultant sketch*

27. Apply angular dimensions between the inclined lines and the inclined line and centerline by using the **Smart Dimension** tool, as shown in Figure 2-36.

28. Click on the confirmation corner; the **Cut-Extrude PropertyManager** and the preview of the cut feature are displayed.

29. Select the **Up To Next** option from the **End Condition** drop-down list of the **Direction 1** rollout in the **Cut-Extrude PropertyManager**.

As discussed earlier, the **End Condition** drop-down list is used to define the termination point of an extruded feature. The **Up To Next** option in this drop-down list is used to extrude a sketch from the sketching plane to the next surface that intersects the feature.

Note

*The **Up To Next** option will be available in the **End Condition** drop-down list only after the base feature has been created.*

30. Choose the **OK** button ( ) from the **Cut-Extrude PropertyManager**; the cut extrude feature is created. Next, change the current orientation of the model to isometric by choosing the **Isometric** button from the **View Orientation** flyout. Figure 2-37 shows the isometric view of the model after creating the cut extruded feature.

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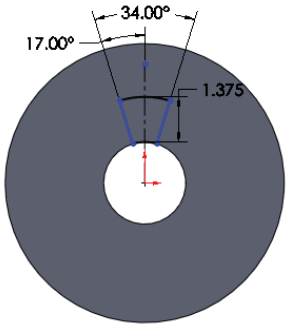

Figure 2-36 *Sketch after applying dimensions*

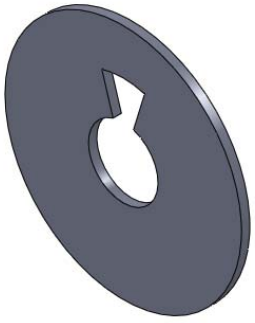

Figure 2-37 *Isometric view of the model*

Creating the Extrude Cut Feature

Next, you need to create a hole at the base of the model. This hole will be created as an extruded cut feature. First, you need to draw the sketch of this feature on the front circular face of the base feature and then convert the sketch drawn into an extruded cut feature by using the **Extruded Cut** tool.

1. Select the front circular face of the base feature and keep the cursor intact for sometime; a pop-up toolbar is displayed. Choose the **Sketch** button from the pop-up toolbar; the sketching environment is invoked.

2. Orient the current view normal to the viewing direction by choosing the **Normal To** button from the **View Orientation** flyout.



3. Draw a circle of diameter 3.5 inches with its center at the origin by using the **Circle** tool. Also, apply the diameter dimension to the circle using the **Smart Dimension** tool. Next, press the ESC key to exit the current tool.

4. Select the circle from the drawing area and keep the cursor intact for sometime; a pop-up toolbar and the **Circle PropertyManager** are displayed.

5. Choose the **Construction Geometry** button from the pop-up toolbar; the selected circle is converted into a construction circle, refer to Figure 2-38. Alternatively, you can select the **For construction** check box from the **Options** rollout of the **Circle PropertyManager** to draw a construction circle.



6. Draw the remaining entities of the sketch. Next, apply dimensions to it by using the tools in the **Sketch CommandManager**, refer to Figure 2-39.

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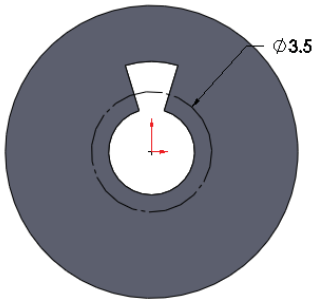

Figure 2-38 *Construction circle*

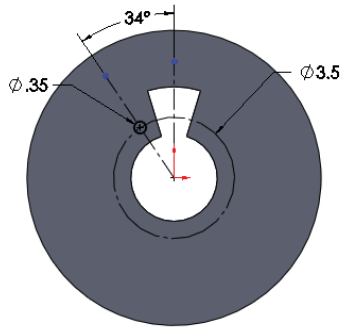

Figure 2-39 *Sketch of the cut feature*

7. Choose the **Features** tab from the CommandManager; the **Features CommandManager** is displayed.

8. Choose the **Extruded Cut** button from the **Features CommandManager**; the **Cut-Extrude PropertyManager** is displayed.

9. Change the current view to isometric by choosing the **Isometric** button from the **View Orientation** flyout.

10. Select the **Up To Next** option from the **End Condition** drop-down list of the **Direction 1** rollout in the **Cut-Extrude PropertyManager**.

11. Choose the **OK** button ( ) from the **Cut-Extrude PropertyManager**. The isometric view of the model after creating the extruded cut feature is shown in Figure 2-40.

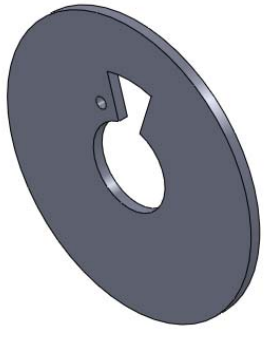

Figure 2-40 *Sketch after creating the cut feature*

Patterning the Features

After creating the cut features, you need to pattern them using the **Circular pattern** tool.

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1. Choose the **Linear Pattern** button from the **Features CommandManager**; the **Linear Pattern** flyout is displayed, refer to Figure 2-41.

2. Choose the **Circular Pattern** button from the **Linear Pattern** flyout, refer to Figure 2-41; the **Circular Pattern PropertyManager** is displayed.

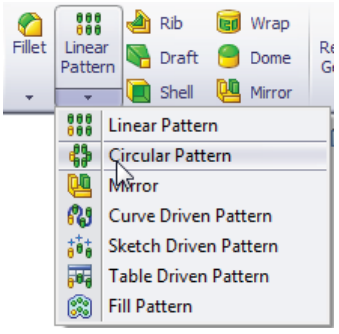

***Figure 2-41** The **Linear Pattern** flyout*

The **Circular Pattern** tool is used to pattern a feature, face, or body in a circular manner.

3. Click on the (+) sign on the left of the **FeatureManager design tree**, which is now displayed on the left in the drawing area; the tree view expands. The three default planes and the list of features created in this part file are now visible in this tree view.

The **FeatureManager design tree** stores and displays all features in a hierarchical manner. You can select any feature of a model or an assembly from the **FeatureManager design tree** and apply different operations on the selected feature.



Note

*In the **Features to Pattern** rollout of the **Circular Pattern PropertyManager**, the **Features to Pattern** selection area remains activated by default. As a result, you can select features from the drawing area or from the **FeatureManager design tree** to create a circular pattern.*

4. Select the **Cut-Extrude1** and **Cut-Extrude2** features from the **FeatureManager design tree**; the names of the selected features are displayed in the **Features to Pattern** selection area of the PropertyManager.

5. Click once in the **Pattern Axis** selection area of the **Parameters** rollout in the **Circular Pattern PropertyManager** to activate it.

The **Pattern Axis** selection area is used to specify the pattern axis. You can select a circular edge or an axis for defining the pattern axis. If you are creating a circular pattern on a circular feature then you can select the circular edge of the feature as the pattern axis. On doing so, the center of the circular feature is selected as the pattern axis.

6. Select the circular edge of the base feature; the preview of the circular pattern of the selected cut features with default values is displayed in the drawing area.

7. Set **5** in the **Number of Instances** spinner in the **Parameters** rollout of the PropertyManager. Also, make sure the **Equal spacing** check box is selected and the value of the **Angle** spinner is set to 360-degree in the **Parameters** rollout of the **Circular Pattern PropertyManager**.

The **Number of Instances** spinner in the **Parameters** rollout of the PropertyManager is used to specify the number of instances in a pattern and the **Equal spacing** check box is used to specify equal or user-defined angular spacing between the instances.

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Note

*When the **Equal spacing** check box is cleared, you need to specify the incremental angle between the instances in the **Angle** spinner. However, if it is selected, SolidWorks automatically calculates and determines the angular spacing between the instances.*

8. Choose the **OK** button ( ) from the **Circular Pattern PropertyManager**; the circular pattern of the selected features is created. Next, change the current view of the model to isometric by choosing the **Isometric** button from the **View Orientation** flyout. The isometric view of the model after patterning the cut features is shown in Figure 2-42.

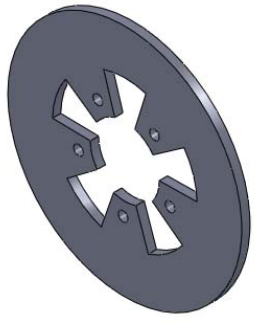

Figure 2-42 *Model after patterning the cut features*

Creating the Fill Pattern of the Predefined Hole

Next, you will create a fill pattern of the predefined holes by using the **Fill Pattern** tool.

1. Choose the **Fill Pattern** button from the **Linear Pattern** flyout in the **Features CommandManager**, refer to Figure 2-43; the **Fill Pattern PropertyManager** is displayed.

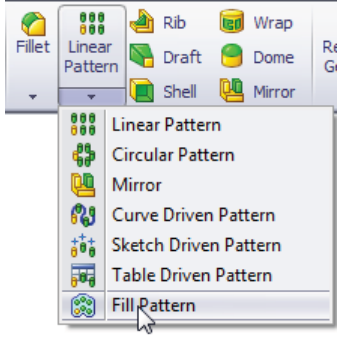

Figure 2-43 *The **Linear Pattern** flyout*

The **Fill Pattern** tool is used to fill a specified area with the pattern of features, faces, or predefined holes. The area to be filled by the pattern of features or holes can be a sketched entity, a face, or a coplanar face.

2. Select the front circular face of the model from the drawing area as the area to be filled; the selected face is highlighted and the name of the selected face is displayed in the selection area of the **Fill Boundary** rollout of the **Fill Pattern PropertyManager**.

3. Expand the **Features to Pattern** rollout of the PropertyManager by clicking on the down arrow located on its right, if it is not expanded.

The **Features to Pattern** rollout of the PropertyManager is used to define whether you want to create a fill pattern of features or predefined holes. By default, the **Selected features** radio

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button is selected in this rollout; as a result, you can select features from the drawing area or from the **FeatureManager design tree** to create a fill pattern. To create the fill pattern of the predefined holes, you need to select the **Create seed cut** radio button from this rollout.

4. Select the **Create seed cut** radio button from the **Features to Pattern** rollout of the PropertyManager to create a pattern of the predefined holes. On selecting this radio button, the **Circle**, **Square**, **Diamond**, and **Polygon** buttons below the radio button are enabled. Also, the preview of the pattern is displayed in the drawing area.

Note

*By default, the **Circle** button is chosen in the **Features to Pattern** rollout; therefore, the preview of the circular hole pattern is displayed in the drawing area.*

Tip. *To create a square, diamond, or polygon hole patterns, you need to select the radio button corresponding to the required option.*

5. Set **0.35** in the **Diameter** spinner of the **Features to Pattern** rollout of the PropertyManager.

   Next, you need to specify the parameters for the pattern layout in the **Pattern Layout** rollout of the **Fill Pattern PropertyManager**.

The buttons such as **Perforation**, **Circular**, **Square**, and **Polygon** available in the **Pattern Layout** rollout of the PropertyManager are used to create the perforated, circular, square, and polygon patterns, respectively. By default, the **Perforation** button is chosen by default in this rollout.

6. Choose the **Circular** button from the **Pattern Layout** rollout of the PropertyManager  to create a circular shape pattern.

7. Set **3.75** in the **Loop Spacing** spinner of the **Pattern Layout** rollout.

8. Select the **Instances per loop** radio button to define the instances of the loop and then set **25** in the **Number of Instances** spinner.

Note

*In the preview of the fill pattern, the seed feature is located at the center of the fill boundary face. For this tutorial, you can accept this default position. However, you can specify the user-defined position for the seed feature by using the **Vertex or Sketch Point** selection area of the **Features to Pattern** rollout in the **Fill Pattern PropertyManager**. To do so, first you need to activate this selection area by clicking on it. Next, you need to select a vertex or sketch point from the drawing area and define the position of the seed feature.*

9. Choose the **OK** button from the **Fill Pattern PropertyManager**; the fill pattern of the predefined circular holes is created, as shown in Figure 2-44.

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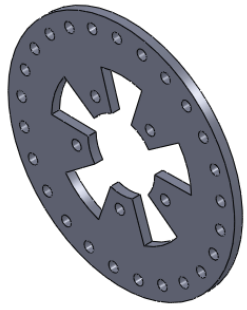

Figure 2-44 *Model after creating the fill pattern feature*

Creating the Fillet Feature

Next, you need to apply the fillet to the model by using the **Fillet** tool. In SolidWorks, a fillet is created as a placed feature.

1. Choose the **Fillet** button from the **Features CommandManager**; the **Fillet PropertyManager** is displayed and you are prompted to select edges, faces, features, or loops to create fillet. Note that if the **FilletXpert PropertyManager** is displayed, you need to choose the **Manual** button from it to display the **Fillet PropertyManager**.



The **Fillet** tool is used to remove sharp corners by adding or removing material from the corners based on the edge reference selected. In SolidWorks, you can add fillets as a placed feature to the model using the **Manual** or **FilletXpert** option. The **Manual** option is used to fillet an internal or external face or edge of a model. Whereas, the **FilletXpert** option is used to create a single fillet or multiple fillets and change the existing constant fillets.

2. Make sure the **Constant radius** radio button is selected in the **Fillet Type** rollout of the PropertyManager.

The **Fillet Type** rollout of the PropertyManager has radio buttons that can be used to create different types of fillets. By default, the **Constant radius** radio button is selected and is used to create a fillet of a constant radius along the selected entity. You can also create a variable radius fillet, face fillet, or full round fillet by selecting their respective radio buttons from the **Fillet Type** rollout of the PropertyManager.

3. Select the required edges of the model from the drawing area to create fillets; the preview of the fillet feature with the default radius value is displayed in the drawing area, refer to Figure 2-45. Also, a callout is displayed attached to the preview of the fillet with its current radius value. The names of the selected edges of the model are displayed in the selection area of the **Items To Fillet** rollout of the **Fillet PropertyManager**.

4. Set **0.25** in the **Radius** spinner of the **Items To Fillet** rollout and then choose the **OK** button from the **Fillet PropertyManager**. The final model after applying the fillet feature is shown in Figure 2-46.

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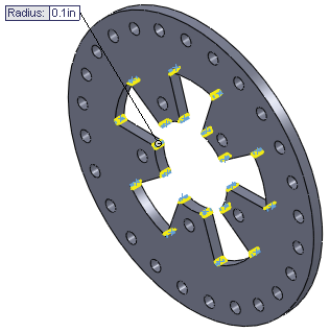

Figure 2-45 *Preview of the fillet feature*

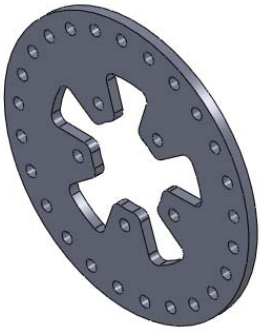

Figure 2-46 *Final model of the Disc plate*

Saving the Model

After completing the model, you need to save it in the *Motor Cycle Project* folder.

1. Invoke the **Save As** dialog box by choosing the **Save** button from the Menu Bar and browse to the *Motor Cycle Project* folder.

2. Enter **Disc Plate** as the name of the model in the **File name** edit box and then choose the **Save** button from the dialog box; the document is saved at *\Documents\Motor Cycle Project*.

3. Close the document by choosing **File > Close** from the SolidWorks menus.

Self-Evaluation Test

Answer the following questions and then compare them to those given at the end of this chapter:

1. Most of the products designed by using SolidWorks are a combination of _____, _____, and _____ features.

2. The _____ feature is a feature that is created first while creating a model in the Part environment.

3. The _____ planes are called datum planes.

4. The _____ tool is used to add the material defined by a sketch.

5. The _____ drop-down list provides options to define the termination point of an extruded feature.

6. A black colored sketch indicates that it is a fully-defined sketch. (T/F)

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7. You can specify the depth of extrusion by dragging the arrow located in the preview of a model. (T/F)

8. When you create a sketch curve by using the **Convert Entities** tool, the _____ and _____ relations are automatically applied to it.

9. You cannot specify the offset distance of an entity by dragging its preview in the drawing area. (T/F)

10. The **Constant radius** radio button is used to create a fillet of constant radius. (T/F)

Review Questions

Answer the following questions:

1. The _____ button in the **Hide/Show Items** flyout is used to hide or show the relations applied to the sketch.

2. The _____ tool is used to draw parallel lines or concentric arcs and circles.

3. The _____ tool is used to create the mirror images of the selected entities.

4. The _____ tool is used to pattern a feature, face, or body in a circular manner.

5. The _____ tool is used to fill a specified area with the pattern of features, faces, or predefined holes.

6. The **Modify** dialog box appears automatically only when the **Input dimension value** check box is selected in the **Options** dialog box. (T/F)

7. Which rollout of the **Boss-Extrude PropertyManager** is used to specify the end condition for extruding a sketch in the first direction?

   (a) **Direction** (b) **Direction 2**
   (c) **Direction 1** (d) **From**

8. Which of the following radio buttons is used to create a fill pattern of predefined holes?

   (a) **Selected features** (b) **Create seed cut**
   (c) **Instances per loop** (d) None of these

9. In SolidWorks, you can draw a regular polygon with infinite number of sides. (T/F)

10. The **Number of Instances** spinner in the **Parameters** rollout of the **Circular Pattern PropertyManager** is used to specify the number of instances in a pattern. (T/F)

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EXERCISE

Exercise

In this exercise, you will create a rear axle, as shown in Figure 2-47. The features of this Axle will be created by using the **Extruded Boss/Base** tool. You will create cosmetic threads with shaded display by using the Ansi Inch of Machine Threads type. The views and dimensions of the axle are given in Figure 2-48. All dimensions are in inches. After completing the model, save it with the name *Rear Axle* at the location *\Documents\Motor Cycle Project*.

(Expected time: 15 min)

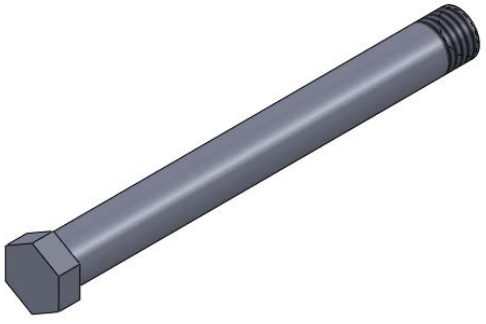

Figure 2-47 *Rear axle*

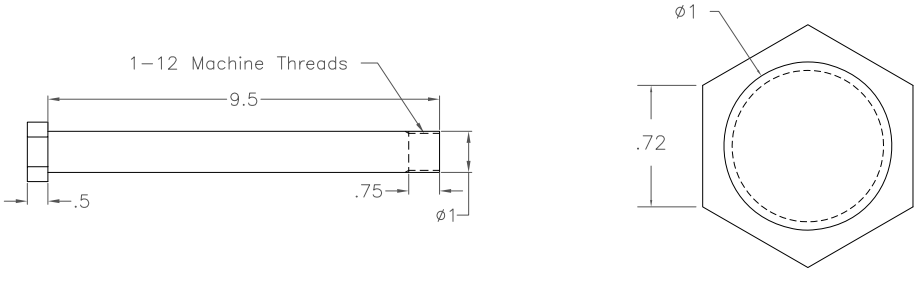

Figure 2-48 *Views and dimensions of the rear axle*

Answers to Self-Evaluation Test

1. sketch, placed, derived, **2.** base, **3.** default, **4. Extruded Boss/Base**, **5. End Condition**, **6.** T, **7.** T, **8. On Edge**, **Fixed**, **9.** F, **10.** T

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