INSIDE:

Tutorial Model Files on CD

A Commands Guide Tutorial for SolidWorks 2007

A Reference Guide with over 200 integrated stand-alone tutorials

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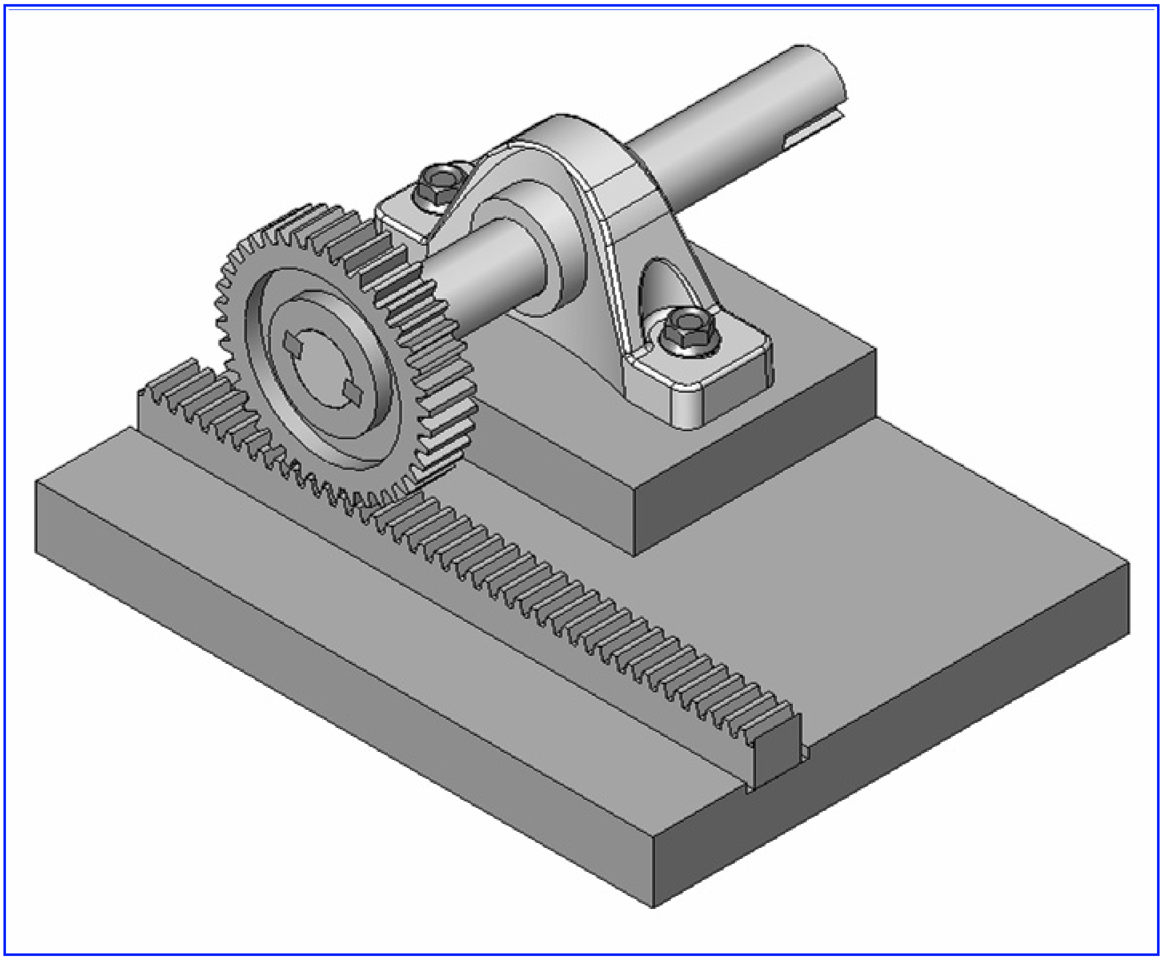

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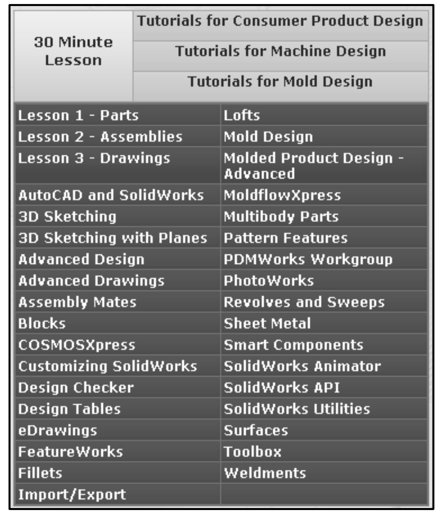
CHAPTER 1: QUICK START

Chapter Objective



Chapter 1 provides a basic overview of the concepts and terminology used throughout this book using SolidWorks® 2007 software. If you are completely new to SolidWorks, you should read Chapter 1 in detail and complete Lesson 1, Lesson 2, and Lesson 3 in the online SolidWorks Tutorials.

If you are familiar with an earlier release of SolidWorks, you still might want to skim this chapter to get acquainted with some of the commands, menus, and features that you haven't used; or you can simply jump ahead to Chapter 2.

Chapter 1 introduces you to many of the basic operations of SolidWorks such as; definitions, starting a SolidWorks session, using the User Main menu and Interface along with opening and closing files, creating a part, assembly, and drawing.

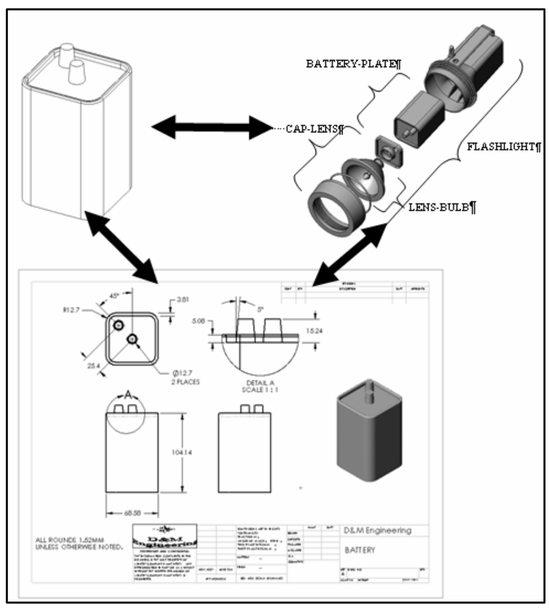
SolidWorks releases new versions on a yearly basis. Part of this strategy is to introduce new items that are focused on a particular category of features. The book will address many of these new features. On the completion of the chapter, you will be able to:

- Comprehend SolidWorks.
- Understand and apply basic concepts in SolidWorks:
  - A SolidWorks model, Features, Base Sketch, Refining the design, Associativity, Drawings, and Constraints.
- Start a SolidWorks Session.
- Use the SolidWorks User Interface (UI), and CommandManager.
- Address the following Menus, Task Panes, Commands, and Screens: SolidWorks Resources, Design Library, File Explorer, Search, Right-Click Pop-up menus, View Palette, PhotoWorks, Drop-Down Menu, Fly-out FeatureManager, System Feedback, Confirmation Corner, and View Modes.

- Create New Parts:
  - Axle.
  - Flatbar.
- Create a New Assembly:
  - AirCylinder.
  - Insert Components and Mates.
- Create a New Assembly Drawing:
  - Insert four standard views: Front, Top, Right, and Isometric.
  - Insert a simple Bill of Materials.

What is SolidWorks?

The SolidWorks application is a mechanical design automation software package used to create parts, assemblies, and drawings that takes advantage of the familiar Microsoft Windows graphical user interface.



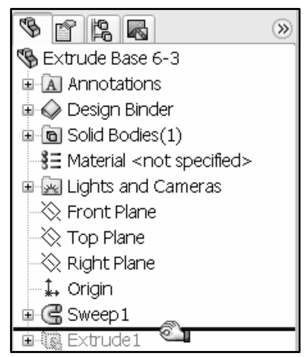
SolidWorks is an easy to learn design and analysis tool (including COSMOSWorks, COSMOSFloWorks, and COSMOSMotion) which makes it possible for designers to quickly sketch 2D and 3D concepts, create 3D parts and assemblies, and detail 2D drawings.

In SolidWorks, part, assembly, and drawing documents are all related as illustrated. The book is focused for the beginner user with six or more months of experience to the intermediate user and assumes that you have some working knowledge of an earlier release of SolidWorks.

Basic Concepts in SolidWorks

Below is a list of Basic Concepts in SolidWorks to review and to comprehend. These concepts are applicable to all versions of SolidWorks. All of these concepts are addressed in this book. They are:

- *A SolidWorks model*. Consists of 3D solid geometry in a part or assembly document. SolidWorks features start with either a 2D or 3D sketch. You can either import a 2D or 3D sketch or you can create the sketch in SolidWorks. In any case, it must be a sketch in the part document.
- *Features*. Individual shapes created by Sketch entities tools; lines, circles, rectangles, etc. that when combined, creates the part. Features can also be added to assemblies. Include separate Extrude, Revolve, Loft, or Sweep features etc., within the same part document. Features can include multi-body part capability.
- *Base Sketch*. The first sketch of a part is called the Base sketch. The Base sketch is the foundation for the 3D model. Create a 2D sketch on a default plane which are: Front, Top, and Right plane in the FeatureManager design tree, or on a created plane. You can also import a surface or solid geometry. In a 3D sketch, the sketch entities exist in 3D space. Sketch entities do not need to be related to a specific Sketch plane.
- *Refining the design*. Adding, editing, or reordering features in the FeatureManager design tree. Example: For a part document, you can perform the following types of feature editing operations:



  - Edit the definition, the sketch, or the properties of a feature.
  - Control the access to selected dimensions.
  - Roll back the part to the state it was in before a selected feature was added.
  - View the parent and child relationships of a feature.
  - Use the feature handles to move and resize features.
  - Modify the order in which features are reconstructed when the part is rebuilt.
- *Associativity.* A SolidWorks model is fully Associative. Associativity between parts, sub-assemblies, assemblies, and drawings, assures that changes incorporated in one document or drawing view are automatically made to all other related documents and drawing views.
- *Drawings*. Create 2D drawings of the 3D solid parts and assemblies which you design. Parts, assemblies, and drawings are linked documents. This means that any change incorporated in to the part or assembly changes the drawing document. A drawing generally consists of several views generated from the model. Views can also be created from existing views. Example: The Section view is created from an existing drawing view.

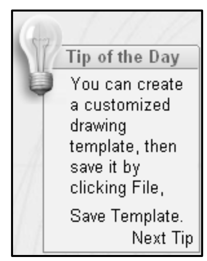
- *Constraints*. SolidWorks supports numerous constraints. Constraints are geometric relationships such as perpendicular, horizontal, parallel, vertical, coincident, concentric, etc. Use equations to establish mathematical relationships between parameters. Apply equations and constraints to your model to capture and maintain design intent.

Starting a SolidWorks Session

Start a SolidWorks session and familiarize yourself with the SolidWorks User Interface (UI). As you read and perform the tasks in this chapter, you will obtain a sense on how to use this book and the structure. To help you learn the material, short quick tutorials are provided throughout the chapters. Actual input commands or required action to perform functions in the tutorial is displayed in bold.

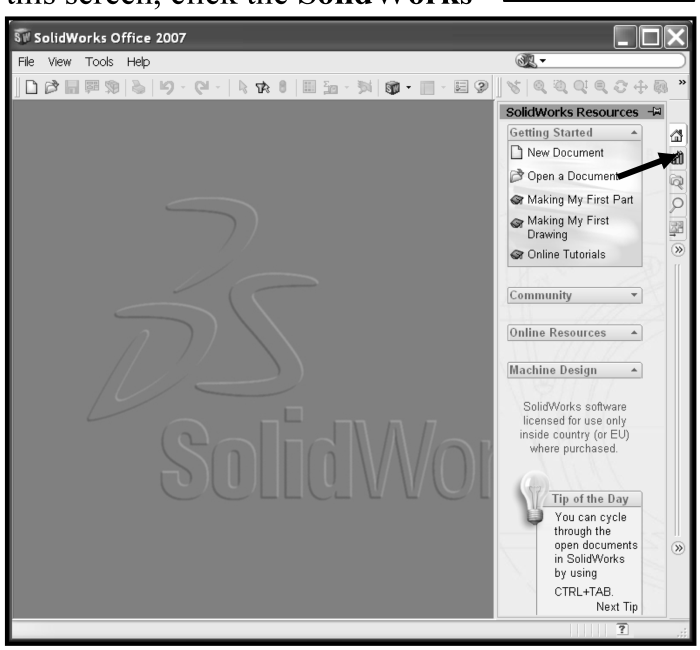
SolidWorks is designed to run on Windows XP Professional. If you are using a network license version of SolidWorks, use the Windows XP Server Edition.

The book was written using SolidWorks Office 2007 on Windows XP Professional SP2 with a Windows Classic desktop theme. Start a SolidWorks session. The SolidWorks application is located in the Programs folder.



Tutorial: Starting a SolidWorks Session 1-1

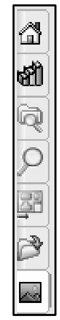
1. Click **Start ➢ All Programs ➢ SolidWorks 2007 ➢ SolidWorks 2007** from the Windows taskbar. The SolidWorks program window opens as illustrated. Do not open a document at this time. If you do not see this screen, click the **SolidWorks Resources** icon in the Task Pane on the right side of the SolidWorks Graphics window.



2. **Read** the Tip of the Day.

Double-click the **SolidWorks 2007 icon** on the Windows Desktop to start a SolidWorks session.

SolidWorks User Interface (UI) and CommandManager



The Task Pane is displayed when a SolidWorks session starts. The Task Pane can be displayed in the following states: Visible or hidden, Expanded or collapsed, Pinned or unpinned, Docked or floating. The Task Pane contains the following tabs: SolidWorks Resources, Design Library, File Explorer, Search, View Palette, Document Recovery, and PhotoWorks Items.

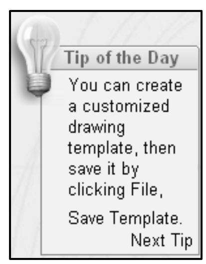
The Document Recovery tab is displayed if your system terminates unexpectedly with an active document; power outage or abnormal termination.

The PhotoWorks Items tab is displayed if PhotoWorks is installed.

SolidWorks Resources

Utilize the left/right arrows to expand or collapse the Task Pane options. The basic SolidWorks Resources menu displays the following default selections: Getting Started, Community, Online Resources, and Tip of the Day.

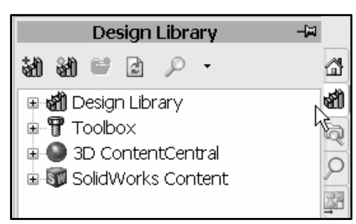
Other user interfaces displayed, such as Machine Design, Mold Design, or Consumer Products Design, are available to be during the initial software installation selection. The book utilizes the Machine Design user interface as illustrated.



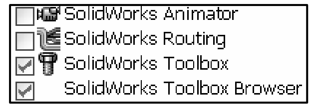
Design Library

The Design Library contains reusable parts, assemblies, and other elements, including library features.

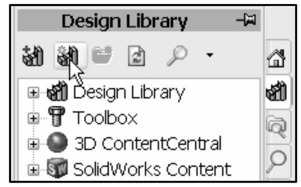
The Design Library tab contains three default selections. Each default selection contains additional sub categories. The selections are: Design Library, Toolbox, 3D ContentCentral, and SolidWorks Content.



Click **Tools ➢ Add-Ins.. ➢ SolidWorks Toolbox** and **SolidWorks Toolbox Browser** to active the SolidWorks Toolbox.

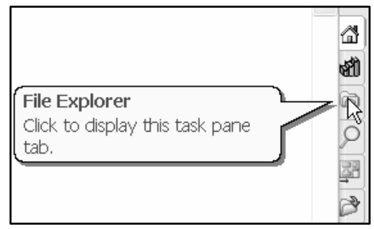


To access the Design Library folders in a non network environment, click **Add File Location**, enter: **C:\Programs Files\SolidWorks\data\design library**. Click **OK**. In a network environment, contact your IT department for system details.



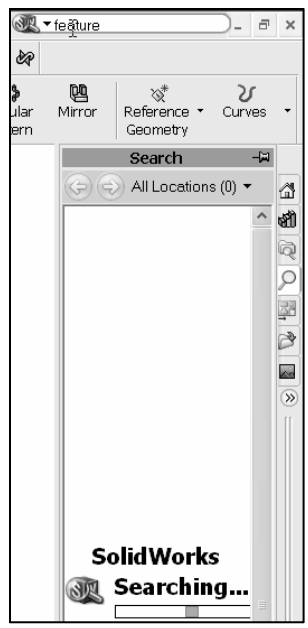
File Explorer

File Explorer duplicates Windows Explorer from your local computer and displays the following directories: Recent Documents, and Open in SolidWorks.

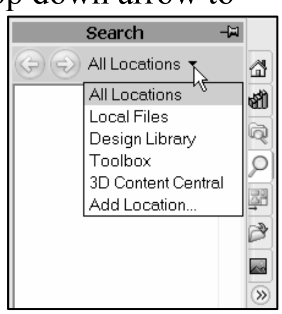


Search

Microsoft Windows Search is installed with SolidWorks and indexes the resources once before searching begins, either after installation, or when you initiate the first search.



The SolidWorks Search box is displayed in the upper right corner of the SolidWorks Graphics window. Enter the text or key words to search. Click the drop down arrow to view the last 10 recent searches.

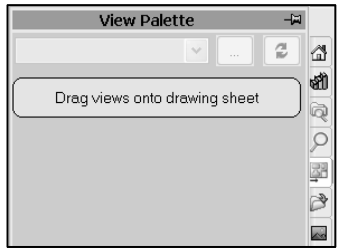


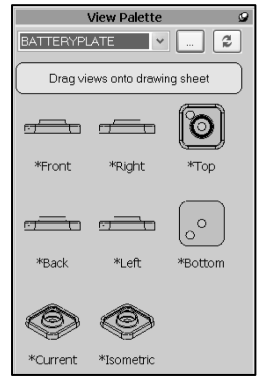
The Search tool in the Task Pane searches the following locations:

- All locations.
- Local Files.
- Design Library.
- SolidWorks Toolbox.
- 3D ContentCentral.
- Added location.

Select any or all of the above locations. If you do not select a file location, all locations are searched.

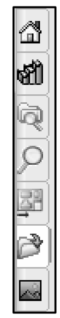
View Palette





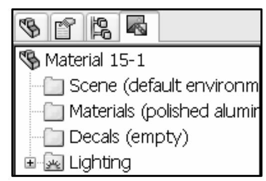
Use the View Palette, located in the Task Pane, to insert drawing views of an active part or click the Browse button to locate the desired model. Click and drag the view from the View Palette into an active drawing sheet to create a drawing view.

Auto Recovery



If auto recovery is initiated in the System Options section and the system terminates unexpectedly with an active document, the saved information files are available on the Task Pane Document Recovery tab.

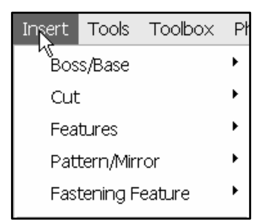
PhotoWorks



PhotoWorks Items create photo-realistic images of SolidWorks models. PhotoWorks provides many professional rendering effects. PhotoWorks contains the following default folders: Scene, Materials, Decals, and Lighting.

Click **Tools ➢ Add-Ins ➢ PhotoWorks** from the Main menu to active the PhotoWorks feature.

Drop-Down Menu



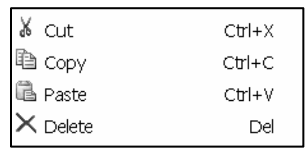
SolidWorks takes advantage of the familiar Microsoft® Windows® graphical user interface. Communicate with SolidWorks either through the drop down menus or the application toolbars. A command is an instruction that informs SolidWorks to perform a task.

To close a SolidWorks drop-down menu, press the Esc key. You can also click any other part of the SolidWorks Graphics window, or click another drop-down menu. The drop-down menu options perform three basic functions. They are:

1. Displays a SolidWorks dialog box.
2. Submits a command to create or modify your drawing.
3. Offers an expanded set of tools located in the SolidWorks Toolbars.

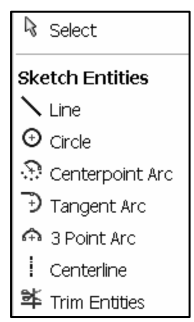
Fly-out FeatureManager

The Fly-out FeatureManager design tree allows you to view and select items in the PropertyManager and the FeatureManager design tree at the same time. The Fly-out FeatureManager provides that ability to select items which may be difficult to view or select from the Graphics window. You can hide, modify the transparency of, go to, or zoom to selected items. You cannot suppress items or roll back the build. Throughout the book, you will select commands and command options from the drop-down menus, Fly-out FeatureManager, or from the SolidWorks main toolbars.



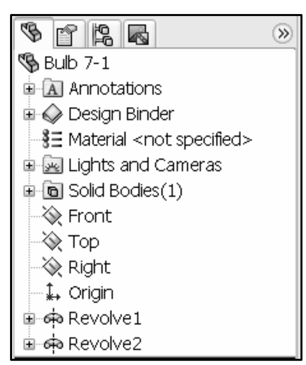
Another method for accessing a command is to use the accelerator key. Accelerator keys are special keystrokes which activates the drop-down menu options. Some commands in the menu bar and items in the drop-down menus have an underlined character. Press the Alt key followed by the corresponding key to the underlined character activates that command or option.

Right-Click Pop-up menus



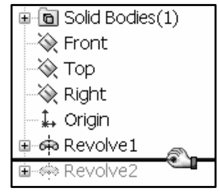
Right-click in the Graphics window to display a context-sensitive shortcut menu. If you are in the middle of a command, this menu displays a list of options specifically related to that command. Example: if you right-click your mouse before picking the first point of the Rectangle tool, a menu is displayed in the Graphics window. The menu displays Sketch Entities, Selected Entity, and other Relations and menu options.

FeatureManager Design Tree



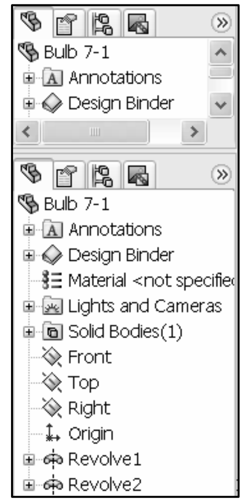
The FeatureManager design tree is located on the left side of the SolidWorks Graphics window. The FeatureManager provides a summarize view of the active part, assembly, or drawing. The FeatureManager design tree displays the details on how your part, assembly, or drawing is created.

Understand the FeatureManager design tree to troubleshoot your model. The FeatureManager is use extensively throughout this book. Expand, collapse, and scroll the FeatureManager design tree.



To collapse all items in the FeatureManager, **right-click** and select **Collapse items**, or press the **Shift** +**C** keys.

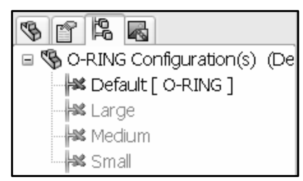
Roll back a feature with the Rollback Bar in the FeatureManager design tree. The rollback bar is a yellow and black line which turns blue when selected. Click and drag the **rollback bar** up or down the FeatureManager design tree to step forward or backward through the regeneration sequence.



You can save your models with the rollback bar in any location of the FeatureManager design tree.

The FeatureManager design tree and the Graphics window are dynamically linked. Select sketches, features, drawing views, and construction geometry in either pane. Split the FeatureManager design tree and either display two FeatureManager instances, or combine the FeatureManager design tree with the ConfigurationManager or PropertyManager.

Move between the FeatureManager design tree, PropertyManager, ConfigurationManager, Functional Tolerances, and Render Manager by selecting the tabs at the top of the menu.

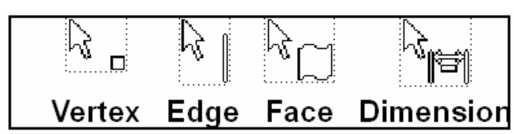


The ConfigurationManager is located to the right of the FeatureManager. Use the ConfigurationManager to create, select, and view multiple configurations of parts and assemblies.

Split the ConfigurationManager and either display two ConfigurationManager instances, or combine the ConfigurationManager with the FeatureManager design tree, PropertyManager, or a third party application that uses the panel. The icons in the ConfigurationManager denote whether the configuration was created manually or with a design table.

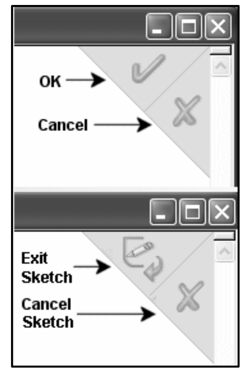
System Feedback

SolidWorks provides system feedback by attaching a symbol to the mouse pointer cursor arrow. The system feedback symbol indicates what you are



selecting or what the system is expecting you to select. As you move the mouse pointer across your model, system feedback is provided to you in the form of symbols, riding next to the cursor arrow.

Confirmation Corner

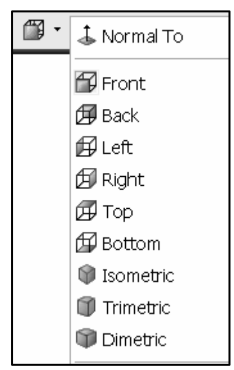


When numerous SolidWorks commands are active, a symbol or a set of symbols are displayed in the upper right hand corner of the Graphics window. This area is called the Confirmation Corner.

When you active or open a sketch, the confirmation corner box displays two symbols. The first symbol is the sketch tool icon. The second symbol is a large red X. These two symbols supply a visual reminder that you are in an active sketch. Click the sketch symbol icon to exit the sketch and to saves any changes that you made.

When other commands are active, the confirmation corner box provides a green check mark and a large red X. Use the green check mark to execute the current command. Use the large red X to cancel the command.

View Toolbar



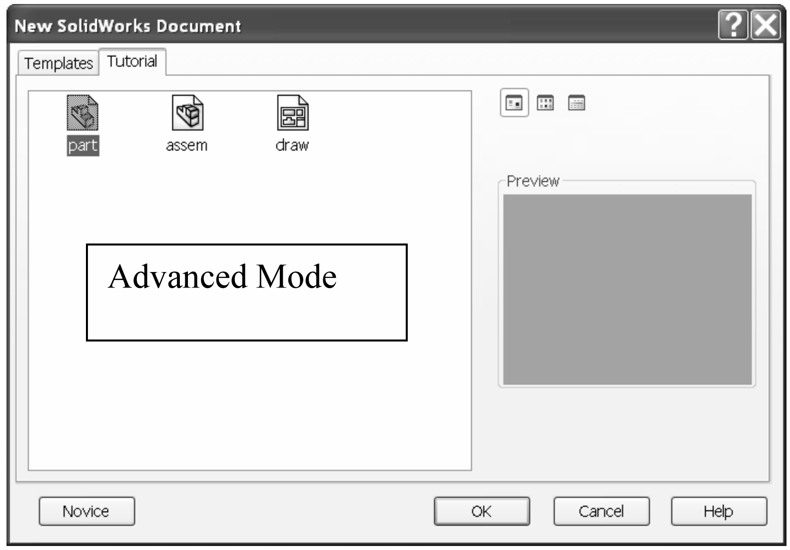
SolidWorks provides the user with numerous view options from the Standard View toolbar. The view options are from left to right: **Previous View**, **Zoom to Fit**, **Zoom to Area**, **Zoom to Selection**, **Rotate View**, **Pan**, **3D Drawing View**, **Standard Views**;(*Normal To, Front, Back, Left, Right, Top, Bottom, Isometric, Trimetric, Dimetric*) , **Wireframe**, **Hidden Line Visible**, **Hidden Lines Removed**, **Shaded With Edges**, **Shaded**, **Shadows In Shaded Mode**, **Section View**, **Real View Graphics**.

Create New Parts

A part is a 3D model which consists of features. What are features?

1. Features are geometry building blocks.
2. Features add or remove material.
3. Features are created from 2D or 3D sketched profiles or from edges and faces of existing geometry.

SolidWorks provides two modes in the New SolidWorks Document dialog box: Novice and Advanced. The Novice Mode is the default mode with three default templates. The Advanced Mode contains access to create additional templates. In this book, you will use the Advanced Mode.



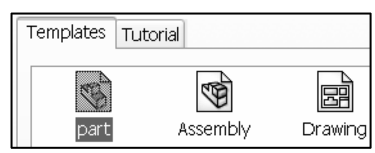
The SolidWorks Conversion Wizard automatically converts SolidWorks files from an earlier version to the present SolidWorks 2007 format. To access the Conversion Wizard, click Windows **Start** ➢ **All Programs** ➢ **SolidWorks 2007** ➢ **SolidWorks Tools** ➢ **Conversion Wizard**. Follow the instructions to convert your older files.

Create the Axle Part

Tutorial: Axle 1-2

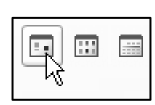
Create a new part named Axle.

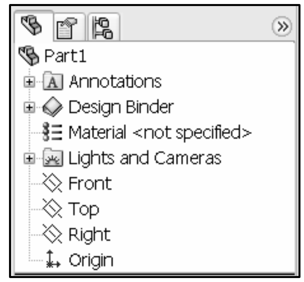
1. Click **File** ➢ **New** from the Main menu.
2. Select the **Advanced Mode** from the New SolidWorks Document dialog box. The Templates tab is the default tab. Part is the default template from the New SolidWorks Document dialog box.
3. Click **OK** from the New SolidWorks Document dialog box.



In a SolidWorks application, each part, assembly, and drawing is referred to as a document. Each document is displayed in a separate Graphics window.

In the New SolidWorks Document, Advanced option, Large icons are displayed by default. Utilize the List option or List Detail option in the New dialog box to view the complete template name.

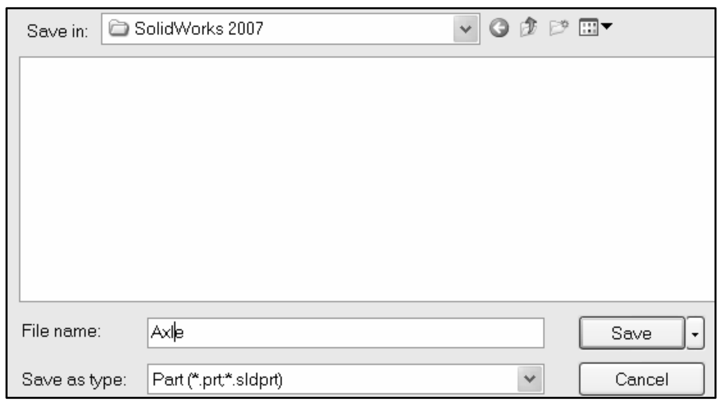




The Advanced Mode remains selected for all new documents in the current SolidWorks session. The Advanced Mode setting is saved when you exit SolidWorks. The default SolidWorks installation contains two tabs in the New SolidWorks Document dialog box: Templates and Tutorial. The Templates tab corresponds to the default SolidWorks templates. The Tutorial tab corresponds to the templates utilized in the Online SolidWorks Tutorials.

In a SolidWorks session, the first system default part filename is named Part1.sldprt. The system attaches the .sldprt suffix to the created part. The second created part in the same session, increments to the filename: Part2.sldprt. Part1 is displayed in the FeatureManager. The Main menu, Standard toolbar, View toolbar, and CommandManager are displayed above the SolidWorks Graphics window.

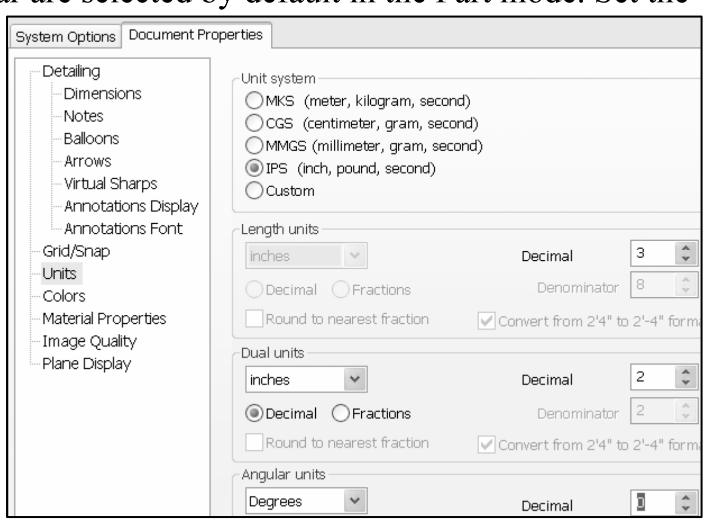
4. Click **File** ➢ **Save** from the Main toolbar.
5. Create a new folder named **SolidWorks 2007**.
6. Enter **Axle** for the File name in the SolidWorks 2007 folder.
7. Click **Save**.



Organize parts into file folders. Use the SolidWorks 2007 folder as the main file folder for this book.

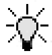
The CommandManager is divided into the Control Area and an expanded toolbar. Select a Control Area icon to display the corresponding toolbar. The Features icon and Features toolbar are selected by default in the Part mode. Set the dimension standard and part units for the Axle.

8. Click **Tools** ➢ **Options** ➢ **Document Properties** tab from the Main menu.
9. Select **ANSI** from the Dimensioning standard box. Click **Units**.
10. Click **IPS** (inch, pound, second) for Unit system.

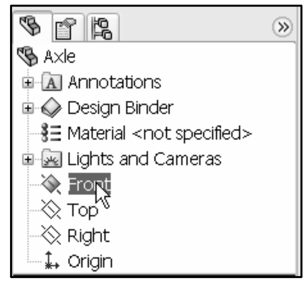


11. Select **3** for Length units Decimal places. Select **0** for Angular units Decimal places.
12. Click **OK** from the Document Properties – Units dialog box.

Insert the 2D Sketch plane for the first feature of the part. The Sketch plane is the plane on which a sketch lies and is configurable through the Sketch plane PropertyManager. You can place a single sketch on various planes in different configurations.

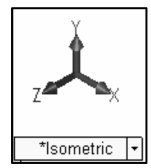


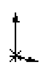
In SolidWorks, the name used to describe a 2D or 3D profile is called a sketch.

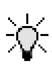


13. Create a 2D Sketch. Click **Front Plane** from the FeatureManager design tree. This is your Sketch plane for the first feature.

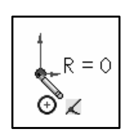
The Reference Triad is located in the lower left corner of the Graphics window. The Reference Triad displays the orientation of the model coordinate axes; (Red-X, Green-Y, and Blue-Z) at all times when activated. The Reference Triad aids you to display how the view orientation is displayed relative to the Front Plane.

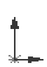


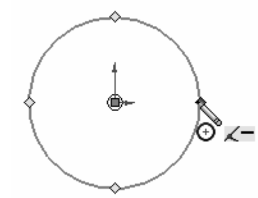
14. Click **Sketch**  from the Control Area. The Sketch toolbar is displayed in the CommandManager.
15. Click **Circle**  from the Sketch toolbar. The Sketch opens on the Front Plane in the Front view by default. The Circle PropertyManager is displayed.
16. Drag the **mouse pointer** into the Graphics window. The cursor displays the Circle feedback symbol . The center point of the circle is positioned at the origin. The part origin  is displayed in the center of the Graphics window. The origin represents the intersection of the three default reference planes. They are: Front Plane, Top Plane, and Right Plane. The positive X-axis is horizontal and points to the right of the origin in the Front view. The positive Y-axis is vertical and point upward in the Front view.



The Front Plane rotates, and is parallel to the screen. This only happens for your first sketch in a part.



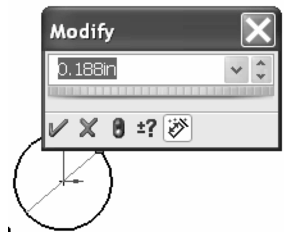
17. Click the **origin**  from the Graphics window. This is the first point of the circle. The red dot feedback indicates the origin point location. The mouse pointer displays the Coincident to point feedback symbol .



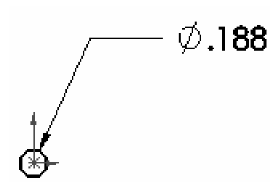
18. Create the circle. Drag the **mouse pointer** to the right of the origin. Click a **position** to create the circle.

To control the Sketch relation display, click **View ➢ Sketch Relations** from the Main menu.

19. Add a dimension. Click **Smart Dimension** from the Sketch toolbar.
20. Click the **circumference** of the circle. The cursor displays the diameter feedback symbol.
21. Click a **position** to place the dimension in the Graphics window. The Dimension PropertyManager is displayed when a dimension is selected. In the Dimension PropertyManager, you can modify dimension properties such as Tolerance/Precision, Dimension Text, and various Display Options.



22. Enter **.188**in the Modify dialog box. The circular sketch is centered at the origin.



If your sketch is not correct, select **UNDO** from the Main menu.

To fit your sketch to the Graphics window, press the **f** key.

Create your first feature. Create an Extruded Base feature. The Extruded Boss/Base feature adds material to a part. The Extruded Base feature is the first feature of the Axle part. An extrusion extends a profile along a path normal to the profile plane for some distance. The movement along that path becomes the solid model. The 2D circle is sketched on the Front Plane.

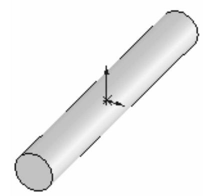
An Extruded Base feature is a feature in SolidWorks that utilizes a sketched profile and extends the profile perpendicular (⊥) to the Sketch plane. The Base feature is the first feature that is created. Keep the Base feature simple.

23. Click **Features** Features from the Control Area.
24. Click **Extruded Boss/Base** from the Features toolbar. The Extrude PropertyManager is displayed. The Extrude PropertyManager displays the parameters utilized to define the feature.

25. Select **Mid Plane** for End Condition in Direction 1. The Mid Plane End Condition extrudes the sketch equally on both sides of the Sketch plane.

26. Enter **1.375**in for Depth. The Depth defines the distance.

27. Click **OK** from the Extrude PropertyManager. Extrude1 is created and is displayed in the FeatureManager design tree.

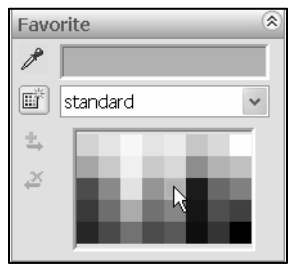


28. Press the **f** key to fit the Axle to the Graphics window.

29. **Save** the part.

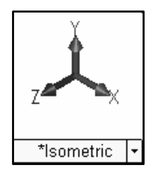
Modify the color of the Axle part.

30. Click the **Axle** AXLE icon from the FeatureManager design tree. Click **Edit Color** from the Standard toolbar. The Color and Optics PropertyManager is displayed.



31. Select a **color** from the Edit Color box.

32. Click **OK** from the Color And Optics PropertyManager. The Axle is displayed with the selected color.



33. Display an **Isometric** view from the Reference Triad.

34. **Save** the part. You completed the Axle part using the Extruded Base/Boss feature with the Mid Plane End Condition option. You also applied a color to the part. Utilize the Edit Color feature to control part and feature color. SolidWorks utilizes default colors to indicate status of sketches and features. Example: Default colors indicate the status of a sketch.

Sketches are generally defined in one of the following states. Color indicates the state of individual sketch entities. The states of individual sketch entities are:

1. *Under Defined*. There is inadequate definition of the sketch, (Blue). The FeatureManager displays a minus (-) symbol before the Sketch name.

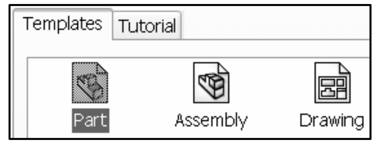
2. *Fully Defined.* Has complete information, (Black). The FeatureManager displays no symbol before the Sketch name.

3. *Over Defined.* Has duplicate dimensions, (Red). The FeatureManager displays a (+) symbol before the Sketch name. The What's Wrong dialog box is displayed.

Create the Flatbar Part

Tutorial: Flatbar 1-3

Create a new part named Flatbar.

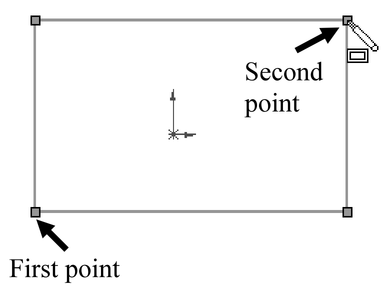


1. Click **File** ➢ **New** from the Main menu. Part is the default template from the New SolidWorks Document dialog box.
2. Double-click the **part** icon. Click **File** ➢ **Save** from the Main menu.
3. Enter **Flatbar** in the SolidWorks 2007 folder.
4. Click **Save**. Flatbar is displayed in the FeatureManager design tree.

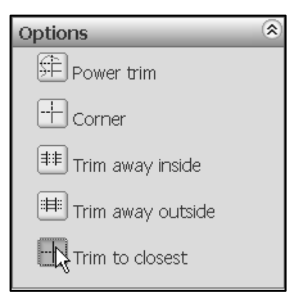
Set the dimension standard and part units for the Flatbar part.

5. Click **Tools** ➢ **Options** ➢ **Document Properties** tab from the Main menu.
6. Select **ANSI** from the Dimensioning Standard list box.

7. Click **Units**. Click **IPS** for Unit system. Select **3** for Length units Decimal places. Select **0** for Angular units Decimal places.
8. Click **OK** from the Document Properties – Unit dialog box. Click **Front Plane** from the FeatureManager.
9. Click **Sketch** from the Control Area. The Sketch toolbar is displayed in the CommandManager.



10. Click **Rectangle** from the Sketch toolbar.
11. Click the **first point** of the rectangle below and to the left of the origin in the Graphics window.
12. Drag the **mouse pointer** up and to the left of the origin. Release the **mouse button**.



13. Click **Trim Entities** from the Sketch toolbar. The Trim PropertyManager is displayed.
14. Click **Trim to closest** from the Trim PropertyManager.
15. Click the **right vertical** line. The line is removed.
16. Click the **left vertical** line. The line is removed.

17. Click **OK** from the Trim PropertyManager.

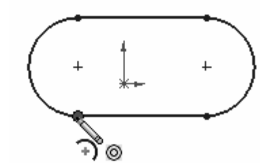
18. Click **Tangent Arc** from the Sketch toolbar.

19. Click the **top right** endpoint of the top horizontal line.

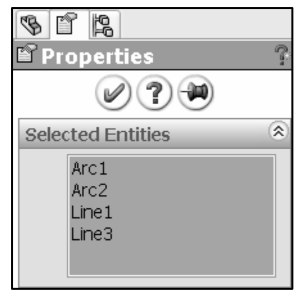
20. Drag the **mouse pointer** to the right and downward.

21. Click the **bottom right endpoint** to complete the arc.

22. Perform the same **procedure** to create the left 180 degree Tangent Arc as illustrated.



You can select all entity types in parts, assemblies, and drawings by dragging a selection box with the mouse pointer.



23. Right-click **Select** in the Graphics window.

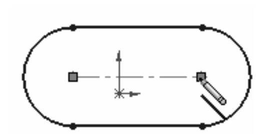
24. **Box-Select** the model geometry. The geometry inside the window is selected. The selected geometry is displayed in green. Two arcs and two lines are listed in the Properties Selected Entities box.

Box-Select is a click and drag procedure from left to right in the Graphics window.

Maintain the slot sketch symmetric about the origin. Utilize Add Relations. A relation is a geometric constraint between sketch geometry. Position the origin at the Midpoint of the centerline.

25. Click **Centerline** from the Sketch toolbar.

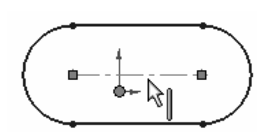
26. Sketch a **horizontal centerline** from the left arc center point to the right arc center point.



27. Right-click **Select**.

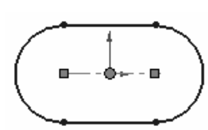
Add a Midpoint relation between the origin and the Centerline.

28. Click the **origin**. Hold the **Ctrl** key down.



29. Click the **centerline**. Release the **Ctrl** key. The origin and the centerline are listed in the Selected Entities box.

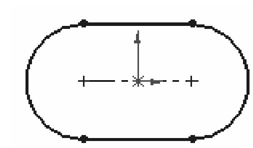
30. Click **Midpoint** from the Add Relations box. Line5 and Point1@Origin is displayed in the Selected Entities box.



31. Click **OK** from the Properties PropertyManager. The Midpoint relation is applied.

Add an Equal relation between the two horizontal lines of the model.

32. Click the **top horizontal** line. Hold the **Ctrl** key down.

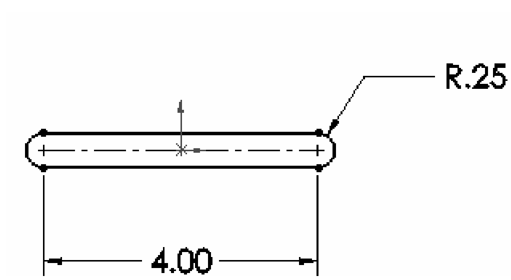


33. Click the **bottom horizontal** line. Release the **Ctrl** key. The two horizontal lines, Line1 and Line3 are displayed in the Selected Entities box. Click **Equal** from the Add Relations box.

34. Click **OK** from the Properties PropertyManager.

35. Click **Smart Dimension** from the Sketch toolbar.

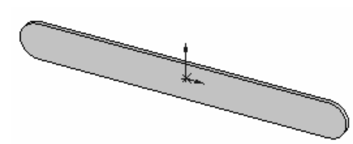
36. **Dimension** the Flatbar as illustrated. The sketch is fully defined. The sketch is displayed in black.



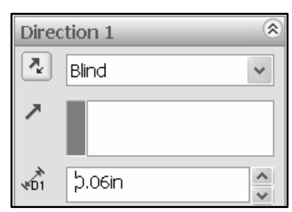
Create the first feature for the Flatbar. Create an Extruded Base feature. The Extruded Boss/Base feature adds material to a part. Extrude the sketch to create the first feature.

37. Click **Extruded Boss/Base** from the Features toolbar.

38. Enter **.060**in for Depth in Direction 1. Blind is the default End Condition. Accept the default conditions.



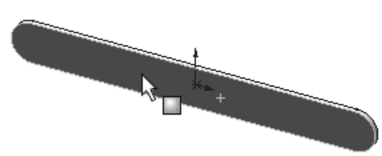
39. Click **OK** from the Extrude PropertyManager. Extrude1 is displayed in the FeatureManager design tree.



40. Expand **Extrude1** from the FeatureManager design tree. Sketch1 is fully defined.

41. **Fit** the model to the Graphics window.

42. **Save** the model. Utilize the Extruded Cut feature to create the first hole. Insert a new sketch for the Extruded-Cut feature.

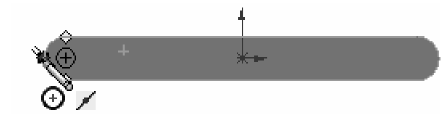


43. Click the **front face** of the Extrude1 feature for the Sketch plane. Extrude1 is highlighted in the FeatureManager design tree. The front face is displayed in green.

44. Click **Sketch** from the Control Area.

45. Display the **Front** view from the Reference Triad.

46. Click **Circle** from the Sketch toolbar. The Circle PropertyManager is displayed.



47. Place the **mouse pointer** on the left arc. Do not click! Wake up the center point of the Flatbar. The center point of the slot arc is displayed.

The process of placing the mouse pointer over an existing arc to locate its center point is call “wake up”.

48. Click the **center point** of the arc.

49. Click a **position** to the right of the center point to create a circle.

50. Click **Smart Dimension** from the Sketch toolbar.

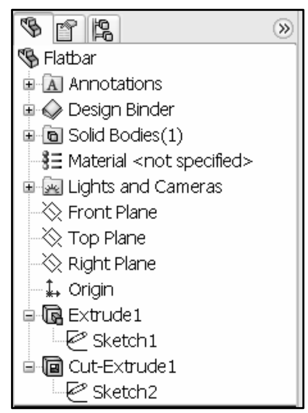
51. **Dimension** the circle as illustrated. Insert an Extruded Cut feature to create the first hole in the Flatbar part.

52. Click **Extruded Cut** from the Features toolbar.

53. Select **Through All** for End Condition in Direction 1.

54. Click **OK** from the Cut-Extrude PropertyManager. The Cut-Extrude1 feature is displayed in the FeatureManager.

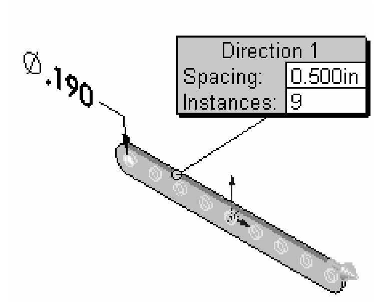
55. Expand **Cut-Extrude1** from the FeatureManager. Sketch2 is fully defined.



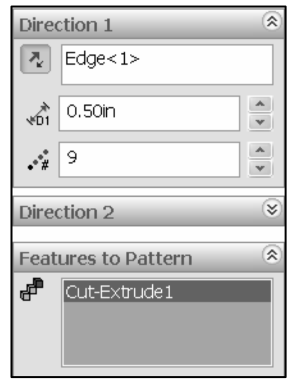
The blue Cut-Extrude1 icon in the FeatureManager indicates that the feature is selected. Select features by clicking their icon in the FeatureManager or selecting geometry in the Graphics window.

Create a Linear Pattern feature. Use a linear pattern to create multiple instances of one or more features that you can space uniformly along one or two linear paths. Utilize the Linear Pattern feature to create additional holes in the Flatbar part.

56. Click **Linear Pattern** from the Features toolbar. The Linear Pattern PropertyManager is displayed. Display an **Isometric** view.



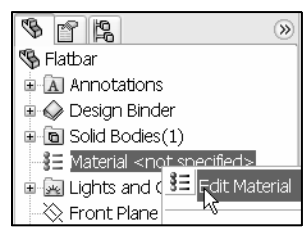
57. Click the **top edge** of the Extrude1 feature for Direction1. Edge<1> is displayed in the Pattern Direction box for Direction1. The direction arrow points to the right.
58. Enter **0.5** for Spacing.
59. Enter **9** for Number of Instances. Instances are the number of occurrences of a feature. Note: Cut-Extrude1 is displayed in the Features to Pattern box.



60. Click **OK** from the Linear Pattern PropertyManager. LPattern1 is displayed in the FeatureManager.

The Recent Commands Menu provides the ability to view the last few commands for easy reuse. Right-click in the **Graphics window**, click **Recent Commands**, view the most recent commands.

Apply material properties to the Flatbar part. Apply a material to a part, and create or edit a material using the Materials Editor PropertyManager. The Materials Editor PropertyManager is located in the FeatureManager design tree. The material and its properties that you add to a part will propagate to COSMOSWorks, COSMOSXpress, and PhotoWorks.



61. Right-click **Material** from the Flatbar FeatureManager design tree.
62. Click **Edit Material**. Select **6061 Aluminum Alloy** for material. View the Physical Properties of the material.
63. Click **OK** from the Materials Editor PropertyManager. 6061 Alloy is displayed in the FeatureManager design tree.
64. Display an **Isometric** view.
65. **Save** the Flatbar part. You completed the Flatbar part using the Extruded Base/Boss feature, Extruded Cut feature and the Linear Pattern feature.

Create an Assembly

An assembly is a document that contains two or more parts. An assembly inserted into another assembly is called a sub-assembly. A part or sub-assembly inserted into an assembly is called a component. Create an AirCylinder Linkage assembly consisting of the following components:

- Axle part.
- Shaft-collar part.
- Flatbar part.
- AirCylinder sub-assembly.

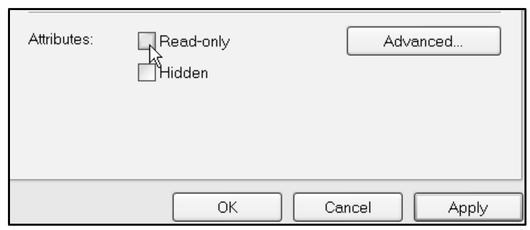
Establishing the correct component relationship in an assembly requires forethought on component interaction. Mates are geometric relationships that align and fit components in an assembly. Mates remove degrees of freedom from a component. Mates reflect the physical behavior of a component in an assembly. The components in the AirCylinder Linkage assembly utilize Standard Mate types only.

SolidWorks Help Topics list the rules governing Mate Type valid geometry.

Tutorial: AirCylinder Linkage assembly 1-4

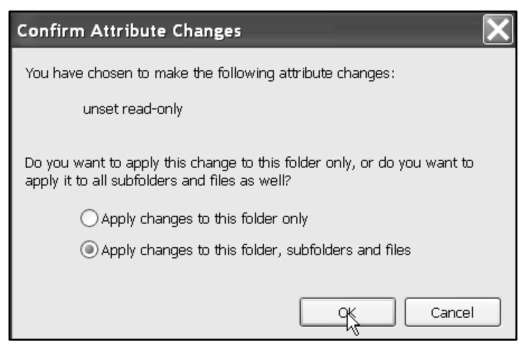
Create the AirCylinder Linkage assembly.

1. **Copy** the SolidWorks 2007 folder from the CD provided in this book to a location on your computer. The CD in the book provides access to over 200 models and their solutions.

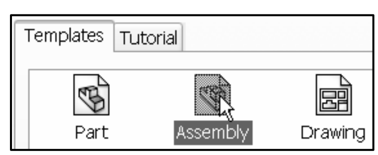


Modify the file attributes.

2. Right-click the **SolidWorks 2007** folder on your computer.
3. Click **Properties**.
4. Uncheck the **Read-only** Attributes box. Click **Apply**.
5. Check **Apply changes to this folder, subfolders and files**.



6. Click **OK** from the Confirm Attribute Changes box.
7. Click **OK** from the Properties box. Open a new assembly.
8. Click **File** ➤ **New** from the Main menu.
9. Double-click **assembly** from the New SolidWorks Document dialog box. The Insert Component PropertyManager is displayed.

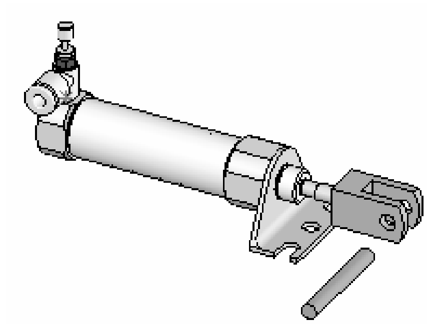


The Insert Component PropertyManager is displayed when the Start command when creating new assembly option box is checked.

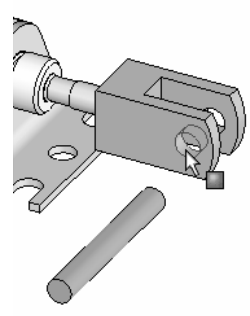
10. Click **View**; check **Origins** from the Main menu.
11. Click **Browse** from the Part/Assembly to Insert box.
12. Double-click the **AirCylinder** assembly from the SolidWorks 2007 folder. The AirCylinder assembly is displayed in the Graphics window.
13. Fix the AirCylinder assembly to the origin. Click the **origin** in the Graphics window. Assem1 is displayed in the FeatureManager design tree and is fixed to the origin.
14. Click **Save As** from the Main menu.
15. Enter **AirCylinder Linkage** for File name.
16. Click **Save**. The AirCylinder Linkage FeatureManager design tree is displayed. The AirCylinder is the first component in the AirCylinder Linkage assembly and is fixed (f). The (f) symbol is placed in front of the AirCylinder name in the FeatureManager.

17. Click **Insert Component** 

from the Assembly toolbar. The Insert Component PropertyManager is displayed.
18. Click **Axle** from the Open documents box in the Insert Component PropertyManager.
19. Click a **position** to the left of the AirCylinder assembly.
20. **Deactivate** the origins.
21. Click and drag a **position** in front of the RodClevis as illustrated.
22. Click **Zoom to Area** on the RodClevis and the Axle.

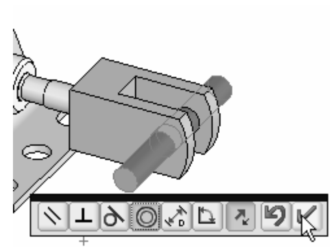


Insert a Concentric Mate between the RodClevis and the Axle. A Concentric Mate forces two cylindrical faces to become concentric. The faces can move along the common axis, but cannot be moved away from this axis.



23. Click **Mate** from the Assembly toolbar. The Mate PropertyManager is displayed. Click the inside **left hole face** of the RodClevis. Face<1>@Air Cylinder is displayed in the Mate Selections box.

24. Click the **long cylindrical face** of the Axle. The cursor displays the Face feedback symbol. The faces are displayed in the Mate Selections box. The Concentric Mate type is selected by default. The Axle is positioned concentric to the RodClevis hole.



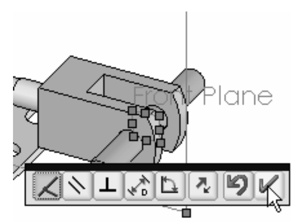
25. Click the **Green Check mark** in the Mate pop-up box.

26. Click and drag the **Axle** left to right. The Axle translates in and out of the RodClevis holes.

Position the mouse pointer in the middle of the face to select the entire face. Do not position the mouse pointer near the edge of the face.

Insert a Coincident Mate. A Coincident Mate forces two planar faces to become coplanar. The faces can move along one another, but cannot be pulled apart.

27. Click the **Front Plane** of the AirCylinder assembly from the AirCylinder Linkage fly-out FeatureManager.

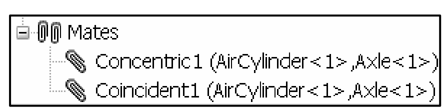


28. Click the **Front Plane** of the Axle part. Coincident is the default Mate Type.

29. Click the **Green Check mark** in the Mate pop-up box. The AirCylinder Front Plane and the Axle Front Plane are Coincident. The Axle is centered in the RodClevis.

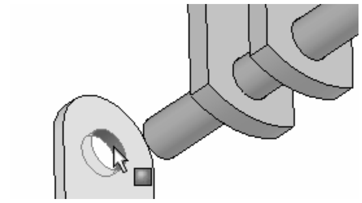
30. Click **OK** from the Mate PropertyManager.

Display the Mates in the FeatureManager to check that the components and the Mate Types correspond to the original design intent.

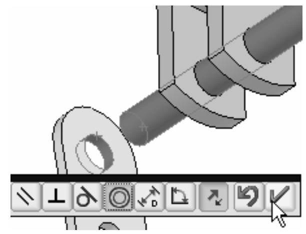


If you delete a Mate and then recreate it, the Mate numbers will be in a different order.

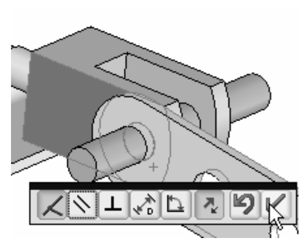
31. Click **Insert Component** from the Assembly toolbar.
32. Click **Flatbar** from the Open documents box in the Insert Component PropertyManager.
33. Click a **position** to the left of the AirCylinder assembly.



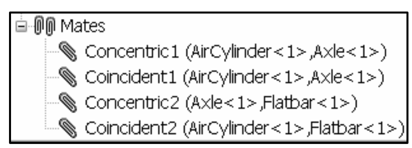
34. Click **Mate** from the Assembly toolbar.
35. Click the inside **left back hole face** of the Flatbar.
36. Click the **long cylindrical face** of the Axle. The faces are displayed in the Mate Selections box. Concentric is the default Mate Type.



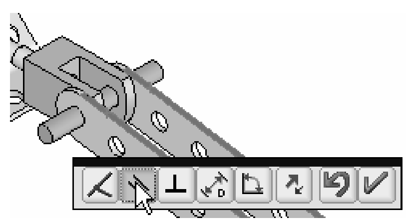
37. Click the **Green Check mark** ✔.
38. Click and drag the **Flatbar**. The Flatbar translates and rotates along the Axle.
39. Insert a Coincident Mate between the **back face** of the Flatbar and the **front face** of the RodClevis. Coincident is the default Mate Type.
40. Click the **Green Check mark** ✔.



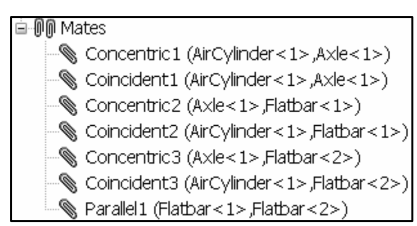
41. Click **OK** from the Mate PropertyManager.
42. Expand the **Mate** folder from the FeatureManager design tree. View the Inserted Mates.



43. Perform the **same procedure** above to insert the second Flatbar on the right side of the RodClevis.
44. Insert a Parallel Mate between the **top narrow face** of the first Flatbar and the **top narrow face** of the second Flatbar. A Parallel Mate places the selected items so they lie in the same direction and remain a constant distance apart from each other.



45. Click **OK** from the Mate PropertyManager.
46. **View** the created Mates.



47. Click and drag the **second Flatbar**. Both parts move together.

48. Click **Insert Component** 

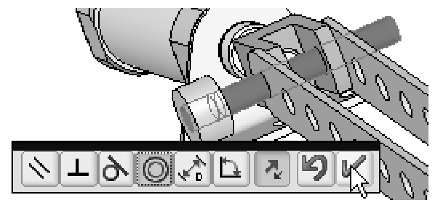
from the Assembly toolbar.

49. Click **Browse** from the Part/Assembly to Insert box.

50. Double-click the **Shaft-collar** part from SolidWorks 2007 folder. The Shaft-collar is displayed in the Graphics window.

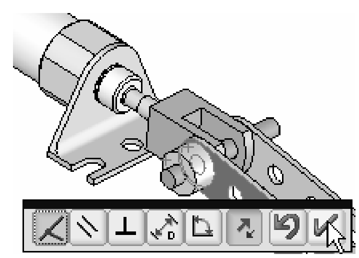
51. Click a **position** to the left of the Axle.

52. Insert a Concentric Mate between the inside **hole face** of the Shaft-Collar and the **long cylindrical face** of the Axle. Concentric is the default Mate Type.



Press the **Shift-z** keys to Zoom in on the model.

53. Insert a Coincident Mate between the **front face** of the Shaft-collar and the **back face** of the left Flatbar. Coincident is the default Mate Type.

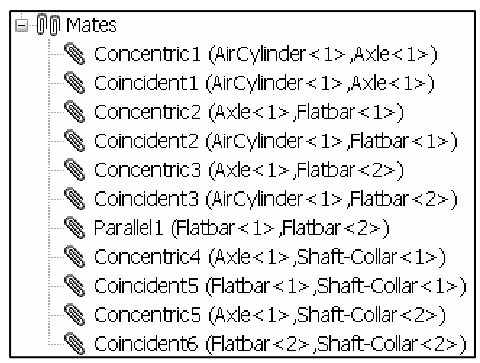


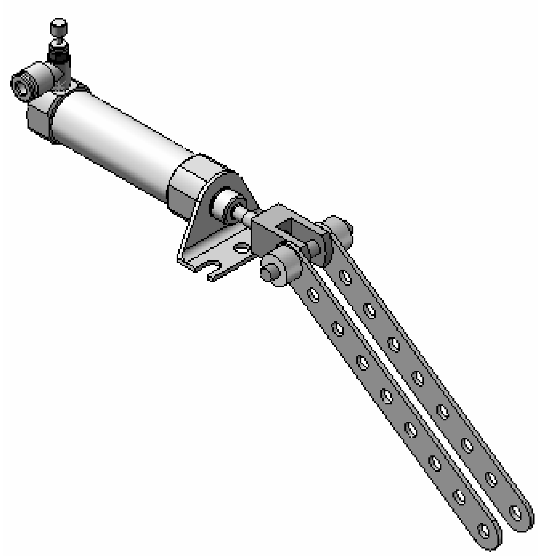
54. Perform the **same procedure** above to insert the second Shaft-Collar on the right side of the Flatbar.

55. Display an **Isometric** view.

56. **Save** the model.

57. **View** the Inserted Mates from the FeatureManager.





Create a New Assembly Drawing

A SolidWorks drawing displays 2D and 3D views of a part or assembly. The foundation of a SolidWorks drawing is the drawing template. Drawing size, drawing standards, company information, manufacturing, and or assembly requirements, units and other properties are defined in the drawing template. In this section you will use the default drawing template.

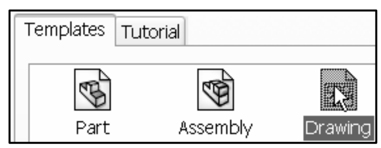
The sheet format is incorporated into the drawing template. The sheet format contains the border, title block information, revision block information, company name and or logo information, Custom Properties and SolidWorks Properties. Because this section of the book is a Quick Start section, you will not address these items at this time.

Custom Properties and SolidWorks Properties are shared values between documents. Utilize an A-size Drawing Template with Sheet Format for the Air Cylinder Linkage assembly drawing.

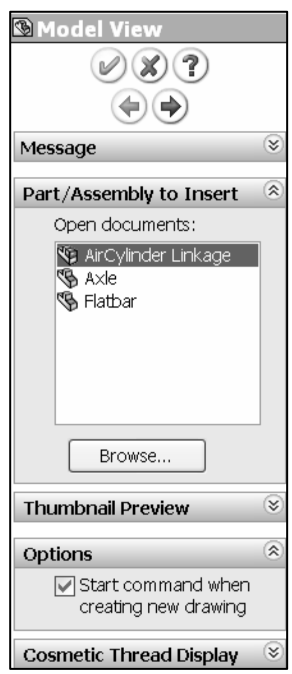
A drawing contains views, geometric dimensioning, and tolerances, notes and other related design information. When a part or assembly is modified, the drawing automatically updates. When a dimension in the drawing is modified, the part or the assembly is automatically updated.

Tutorial: AirCylinder Linkage Drawing 1-5

Create the AirCylinder Linkage assembly drawing. Display the Front, Top, Right, and Isometric views. Utilize the ModelView command in the Drawings toolbar.

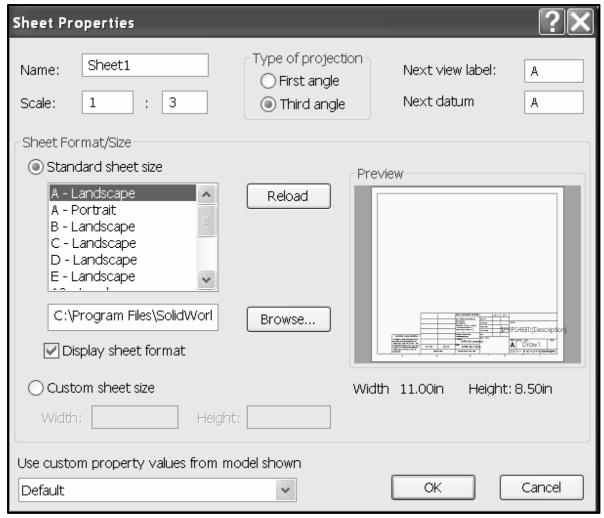


1. Click **File** ➢ **New** from the Main menu.
2. Double-click **Drawing** from the Templates tab.
3. Select **A-Landscape** from the Sheet Format/Size dialog box. Click **OK**. The Model View PropertyManager is displayed.
4. Click **Cancel** from the Model View PropertyManager. Draw1 is displayed. Set the Sheet1 Properties in the drawing.



The Model View PropertyManager is displayed if the Start command when creating new drawing option is checked.

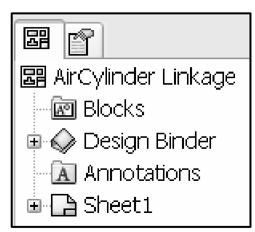
5. Right-click **Properties** in Sheet1. The Sheet Properties is displayed. Draw1 is the default drawing name. Sheet1 is the default first sheet name. The Control Area alternates between Drawings, Sketch, and Annotations toolbars. The Model View PropertyManager is selected by default.
6. Select Sheet Scale **1:3**.
7. Select **Third angle** for Type of projection.
8. Click **OK** from the Sheet Properties box. The A-Landscape paper is displayed in a new Graphics window. The sheet border defines the drawing size, 11″ × 8.5″ or (279.4mm × 215.9mm). The Drawings toolbar is displayed in the CommandManager.



9. Set the Document Properties for your drawing. Click **Tools** ➢ **Options** ➢ **Document Properties** tab from the Main menu.
10. Select **ANSI** for Dimensioning standard. Click **Units**. Select **MMGS** (millimeters, gram, second) for Unit system.
11. Enter **2** for Length units Decimal places.
12. Enter **0** for Angular units Decimal places.
13. Click **OK**.

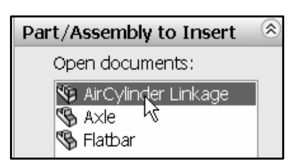
Detailing options provide the ability to address dimensioning standards, text style, center marks, extension lines, arrow styles, tolerance and precision.

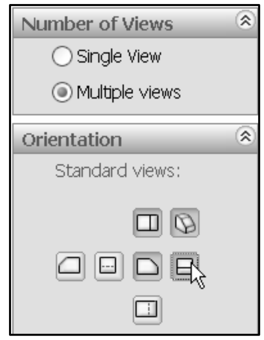
14. **Save** the drawing.
15. Enter **AirCylinder Linkage** for file name.
16. Click **Save**. The AirCylinder Linkage is displayed in the Drawing FeatureManager design tree.



17. Click **Model View**  from the Drawings toolbar. The Model View PropertyManager is displayed.

18. Double-click **AirCylinder Linkage** from the Model View PropertyManager. Insert a Front, Top, Right and Isometric view.





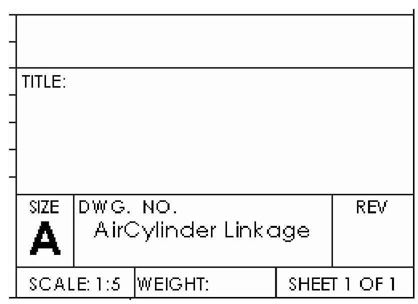
19. Click **Multiple views** from the Number of Views box.
20. Click ***Front, *Top**, and ***Right** from the Orientation box. *Isometric view is activated by default.
21. Click **OK** from the Drawing View PropertyManager. The four views are displayed in Sheet1.

The Title block is located in the lower right hand corner of Sheet1. A Drawing contains two modes:

- Edit Sheet.
- Edit Sheet Format.

Insert views and dimensions in the Edit Sheet mode. Modify the Sheet Format text, lines or title block information in the Edit Sheet Format mode. The CompanyName Custom Property is located in the title block above the TITLE box. There is no value defined for CompanyName. A small text box indicates an empty field. Activate the Edit Sheet Format Mode.

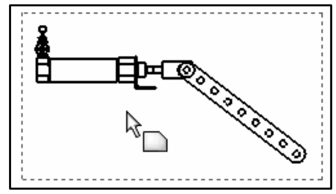
22. Right-click in **Sheet1**.
23. Click **Edit Sheet Format**. The Title block lines turn blue. View the right side of the Title block.
24. Click **Zoom to Area** on the Sheet Format Title block. Modify the font size of Air Cylinder Linkage.
25. Double-click the **AirCylinder Linkage** text in the DWG NO. box.
26. Click the **drop down arrows** to set the Text Font to **12** from the Formatting dialog box.
27. Click **OK** from the Note PropertyManager.
28. Right-click **Edit Sheet** in Sheet1.
29. **Save** the drawing.



Insert a Bill of Material into the AirCylinder Linkage assembly drawing.

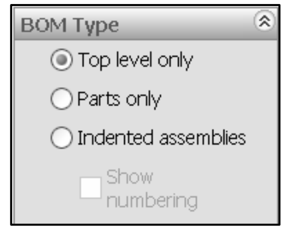
A drawing can contain a table-based Bill of Materials or an Excel based Bill of Materials, but not both.

30. Click **Insert** ➢ **Tables** ➢ **Bill of Materials** from the Main menu. The Bill of Materials PropertyManager is displayed.



31. Click inside the **Front view**, Drawing View1.

32. Click **Top level only** from the BOM Type box. Accept all other defaults.



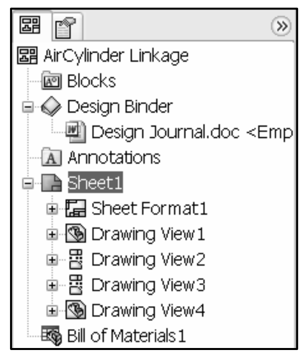
33. Click **OK** from the Bill of Materials PropertyManager. The AirCylinder Linkage assembly FeatureManager design tree is displayed. Position the BOM.

34. Click a position in the **top left** corner of Sheet1.

35. Click **OK** from the Bill of Materials PropertyManager.

ITEM NO.	PART NUMBER	DESCRIPTION	QTY.
1	GIDS-PC-10001	LINEAR ACTUATOR	1
2	Axle		1
3	Flatbar		2
4	GIDS-SC-10012-3-16	SHAFT-COLLAR	2

36. Click inside **Sheet1**. Bill of Material 1 is displayed in the FeatureManager design tree. The Bill of Materials is incomplete. You complete the Bill of Materials later in the Drawing chapter of this book.



37. **Save** the drawing.

38. **Close** the model.

Summary

In this chapter you learned about Models, Features, Base Sketch, Refining the design, Associativity, Drawings, and Constraints. You started a SolidWorks Session; you used the SolidWorks User Interface (UI) and CommandManager.

You created the Axle part using the Extruded Base/Boss feature with the Mid Plane End Condition option. You applied color to the part.

You created the Flatbar part using the Extruded Base/Boss feature with the Blind End Condition option, Extruded Cut feature with the Through All option, and the Linear Pattern feature. You applied material to the Flatbar using the Material Editor PropertyManager.

You created the AirCylinder Linkage assembly and drawing with a Front, Top, Right, and Isometric view. In the assembly you inserted a Concentric, Coincident, and Parallel Mate. In the drawing you edited the Title block and inserted a simple Bill of Materials. You have completed the first chapter of the book. In Chapter 2, you'll learn about System Options in SolidWorks. System Options provides the ability to customize SolidWorks functionality for your needs.