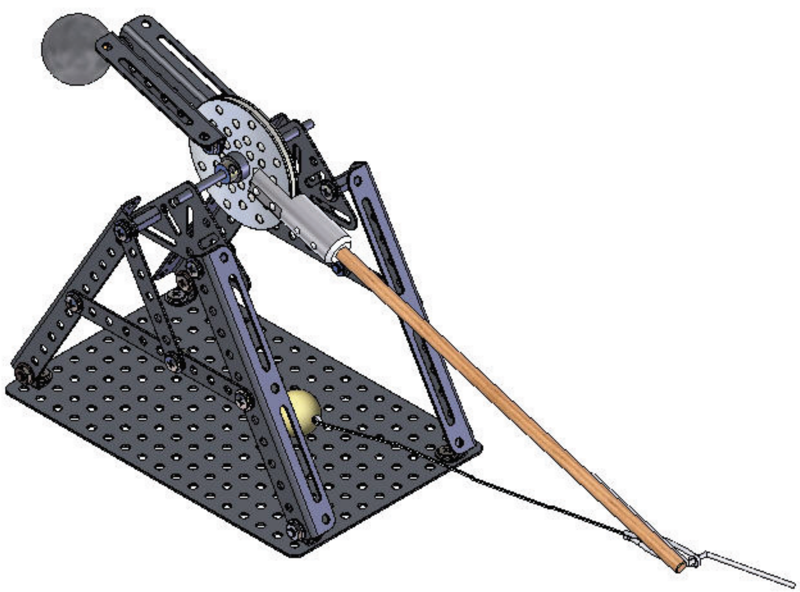
SolidWorks® 2011

Building a Trebuchet

Project-based Approach to Learning SolidWorks



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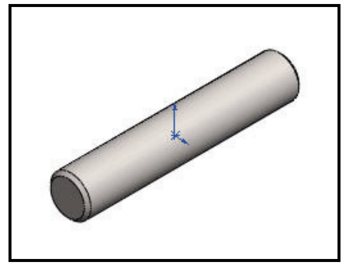
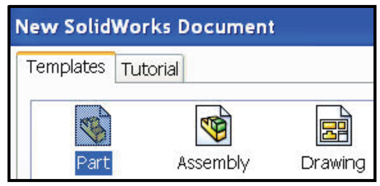
Lesson 2
Modeling the Trebuchet

When you complete this lesson, you will be able to:

- Create a 2D sketch.
- Utilize the following sketch tools: Line, Circle, Centerpoint Straight Slot, Center Rectangle, Centerline, Tangent Arc, and Trim Entities.
- Add and modify dimensions in a sketch.
- Add the following geometric relations to a sketch: Equal.
- Utilize the following features: Extruded Boss/Base, Extruded Cut, Revolved Boss/Base, Fillet, and Chamfer.
- Create a part.
- Save a part.
- Modify a part.
- Apply PhotoWorks to a part.
- Apply material to a part.
- Rename a feature in the Part FeatureManager.

Create the Axle for the Trebuchet Counter Weight Assembly





The first part that you will build is the Axle for the Counter Weight assembly. First, you will need to open a new SolidWorks document. As you learned earlier, SolidWorks uses three kinds of documents: Parts, Assemblies, and Drawings. The Axle is a part. Open a part document.

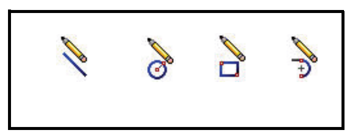
Use the default part template that is provided with SolidWorks.

A template forms the basis of a new part document, controlling units, grid, text, and other settings for the model.

Templates allow you to define your own parameters. You can create customized templates. Save time by defining parameters once, then use them to create new documents. You can also create multiple templates for each document type.

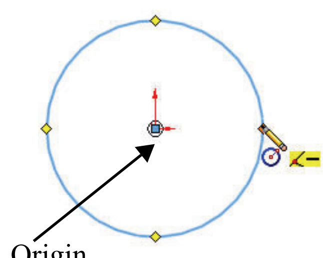
Sketching

Solid models are built from features. Features are the building blocks of the part. Features are based on 2D sketches. Sketches provide the foundation for your SolidWorks project. Sketches are collections of 2D geometry that are used to create solid features.



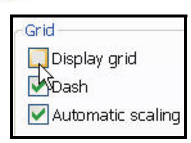
Typical 2D geometry types are lines, circles, rectangles, and arcs. Sketching in SolidWorks is dynamic, with cursor feedback. Every sketch has several characteristics that contribute to its shape, size, and orientation.

Sketch Entities



SolidWorks offers a rich variety of sketch tools for creating profile sketches. For the Axle, you will create a sketch using the Circle Sketch  tool as illustrated.

Note: The Grid is deactivated in the SolidWorks Graphics window for clearer screen shots in this book. Click **Options**, **Document Properties** tab from the Menu bar menu. Click **Grid/Snap**. Un-check **Display grid**.

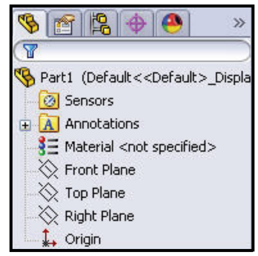


Sketch Tools

Tools can be used to modify the sketch geometry that has been created. This often involves the trimming or extension of entities. You will use the Trim Entities tool in this lesson.

Sketch Planes

Sketches are flat, or planar. A plane is required for a sketch. A SolidWorks part contains three default sketch planes. They are: Front, Top, and Right.



Starting a SolidWorks session

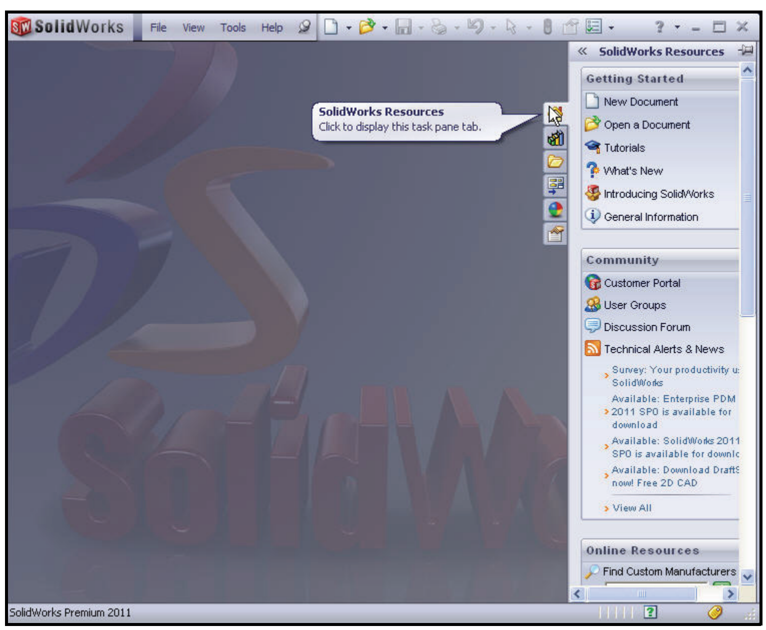
1 **Start the SolidWorks session.**

- Click **All Programs**, **SolidWorks 2011**, **SolidWorks 2011** from the Windows Start menu.

Tip: Start a SolidWorks session, if available by double-clicking the left mouse button on the SolidWorks desktop shortcut icon.

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

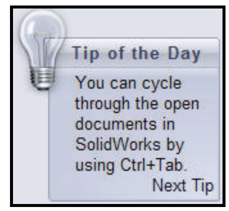
- Click the **SolidWorks Resources** tab on the right side of the Graphics window if you do not see this screen. The Pin tool displays both the Menu Bar toolbar and the Menu Bar menu.



The SolidWorks 2011 default Task Pane contains six tabs:

- SolidWorks Resources tab.
- Design Library tab.
- File Explorer tab.
- View Palette tab.
- Appearances, Scenes, and Decals tab.
- Custom Properties tab.

The SolidWorks Resources contains the following default menus:

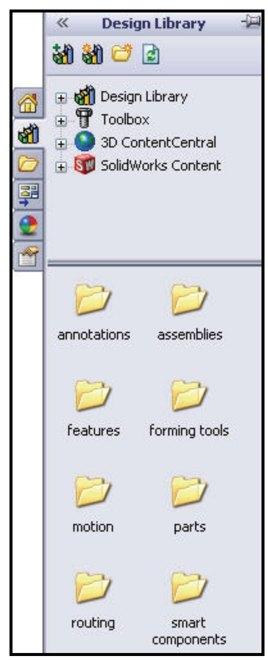


- Getting Started.
- Community.
- Online Resources.

Along with the Tip of the Day box.

Tip: Other user interfaces are available to be displayed: Machine Design, Mold Design, or Consumer Products Design during the initial software installation selection.

The Design Library tab in the Task Pane provides a central location for reusable elements such as parts, assemblies and sketches. The Design Library includes the following menus:

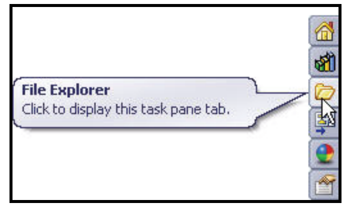


- Design Library.
- Toolbox.
- 3D ContentCentral.
- SolidWorks Content.

The Design Library menu contains the following folders: annotations, assemblies, features, forming tools, motion, parts, routing, and smart components.

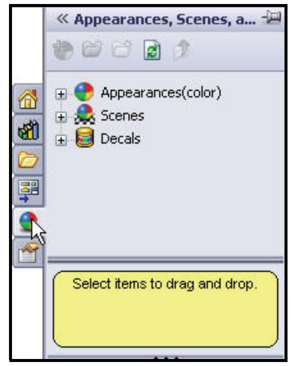
Note: The SolidWorks Content folder contains the SolidWorks Educator Curriculum.

SolidWorks File Explorer duplicates Windows Explorer in functionality on your computer, plus recent documents that are active in SolidWorks.



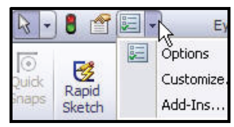
The View Palette tab provides the ability to insert drawing views. It contains images of standard views, annotation views, section views, and flat patterns (sheet metal parts) of the selected model. You can drag views onto the drawing sheet to create a drawing view.

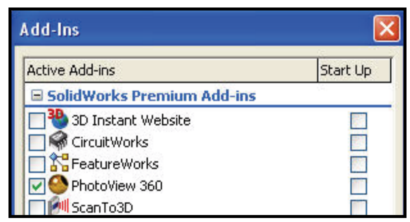
The Appearances, Scenes, and Decals tab provides a library of appearances, scenes, and decals to apply to a model or assembly document.



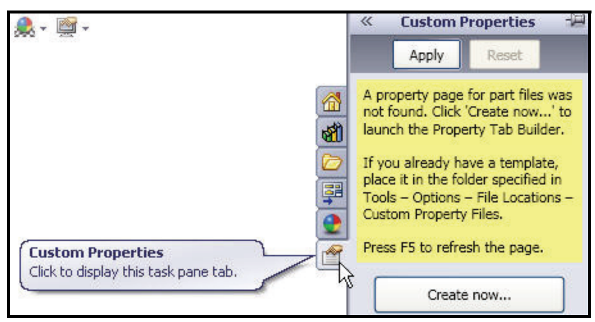
Tip: An appearance defines the visual properties of a model, including color and texture. Appearances do not affect physical properties, which are defined by materials.

Note: Click **Add-Ins**. from the Menu bar menu drop-down arrow. Click **PhotoView 360** from the Active Add-ins dialog box to active the PhotoView 360 feature.

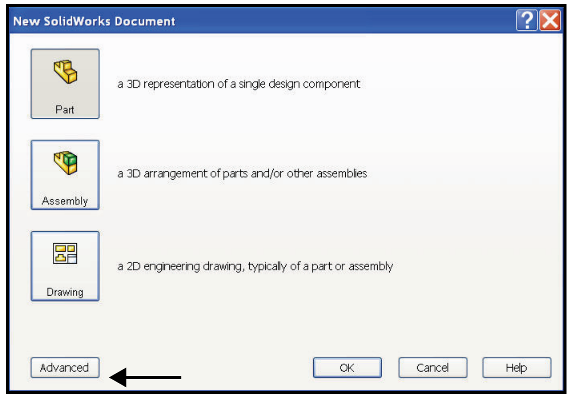




Use Custom Properties in the Task Pane to view and enter custom and configuration-specific properties into SolidWorks files.

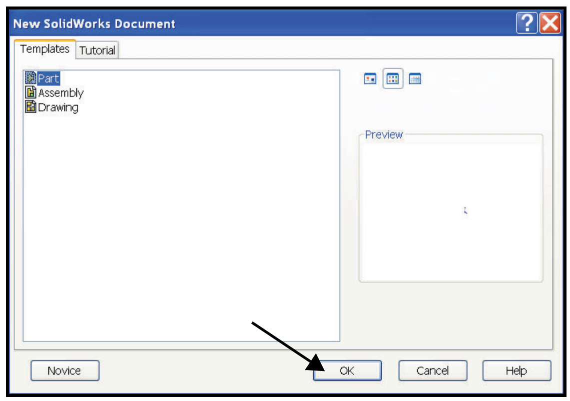


New SolidWorks Document Modes



There are two modes in the New SolidWorks Document dialog box: `Novice` and `Advanced`. The `Novice` option is the default option with three templates. The `Advanced` option contains access to additional templates. In this book, you will use the `Advanced` option.

Creating a New Part



1 **Create a new part.**

- Click **New** from the Menu bar toolbar.

2 **Select the Advanced mode.**

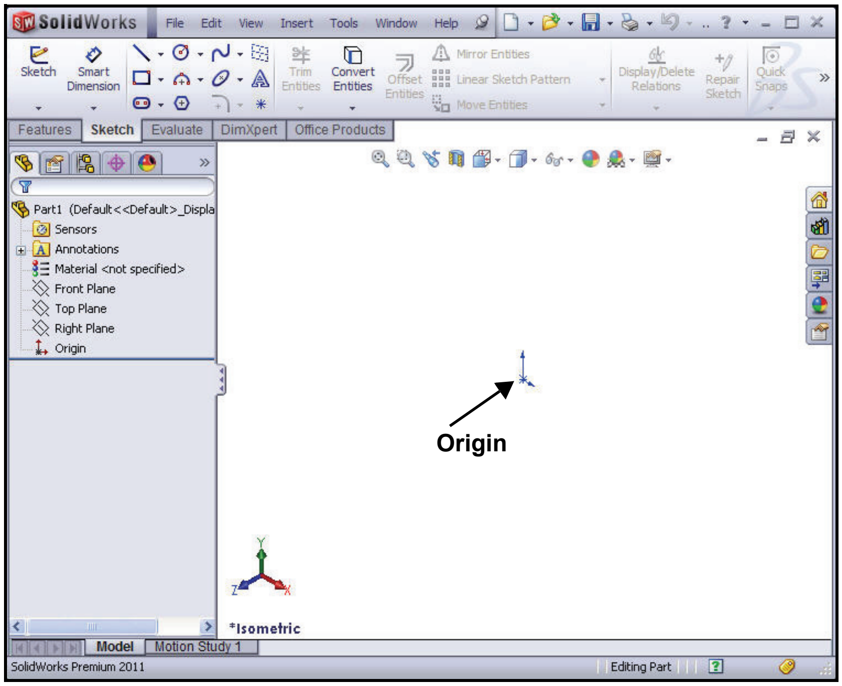
- Click the **Advanced** button to display the New SolidWorks Document dialog box in Advanced mode.

The Templates tab is the default tab. Part is the default template from the New SolidWorks Document box.

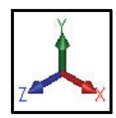
- Click **OK** from the New SolidWorks Document dialog box. A new part document window is displayed.

The Advanced mode remains selected for all new documents in the current SolidWorks session. When you exit SolidWorks, the Advanced mode setting is saved.

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 SolidWorks Tutorials.

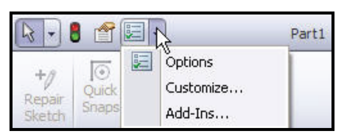


The Reference Triad, (lower left corner) displays the orientation of the model coordinate axes; (red-X, green-Y, and blue-Z) at all times. It can help show how the view orientation has been changed relative to the Front Plane.



The Origin symbol represents the part's model origin which is the intersection of the X, Y, and Z axes. The Origin symbol is displayed in the color red when you are in the Sketch mode.

Setting System Options

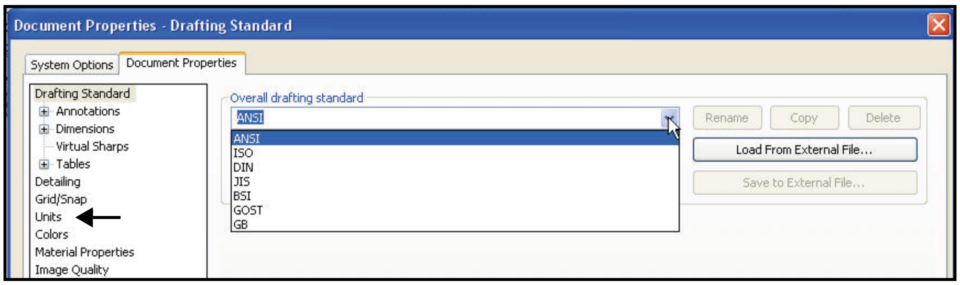


System Options are stored in the registry of the computer. System Options are not part of the document. Changes to the Systems Options affect the current and future documents. Review and modify the System Options. If you work on a local drive C:\, the System Options are stored on your computer.

1 **Set Drafting Standard.**

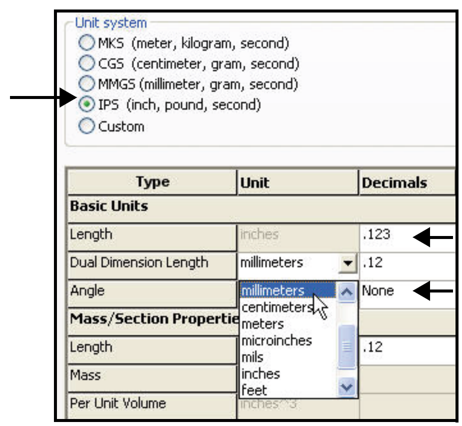
- Click **Options** , **Document Properties** tab from the Menu bar toolbar. The Document Properties - Drafting Standard dialog box is displayed.

- Select **ANSI** from the drop-down menu for Overall drafting standard.



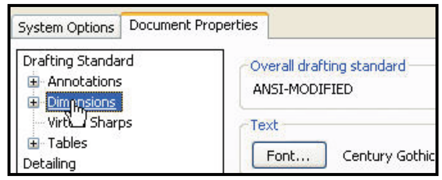
2 Set unit system and length.

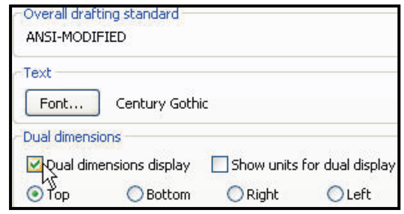
- Click **Units**.
- Click **IPS** (inch, pound, second) for Unit system.
- Select **.123** for Length unit decimal place.
- Select **None** for Angle decimal place.
- Select **millimeters** for Dual Dimension Length.



3 Set dual unit display.

- Click **Dimensions** from the Document Properties dialog box.
- Check the **Dual dimensions display** box.

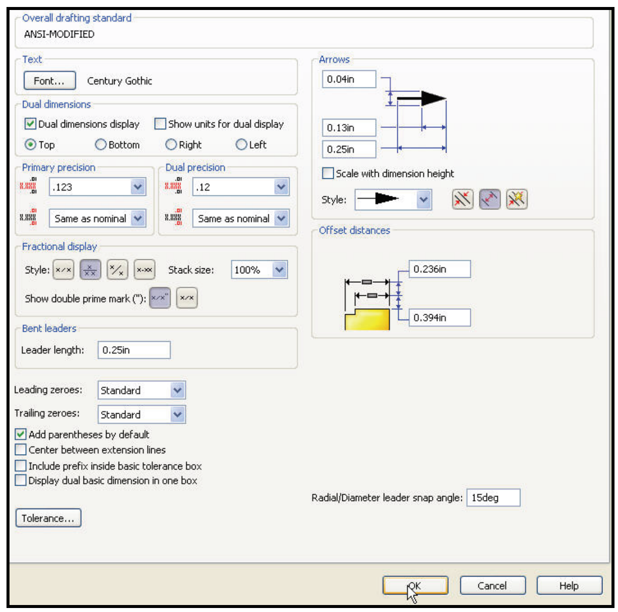




4 Set System options.

- Click **OK** from the Document Properties-Dimensions dialog box.

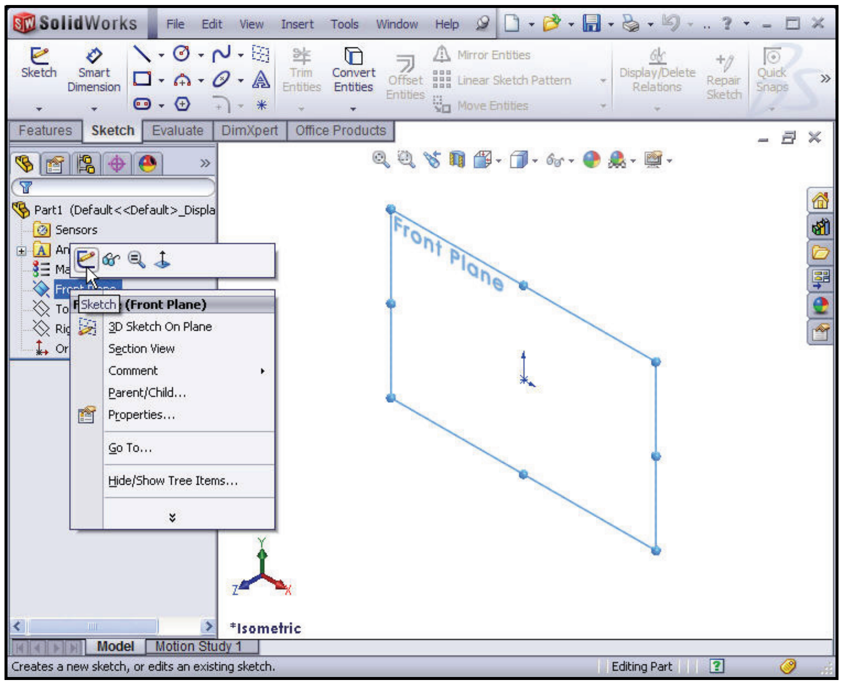
Note: All dimensions are displayed in IPS and MMGS units. IPS is the primary unit. Millimeters are displayed in brackets, [ ].



Preparing to Sketch

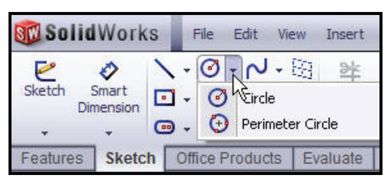
1 Select the Sketch plane.

- Right-click **Front Plane** from the FeatureManager design tree. Front Plane is highlighted in the FeatureManager.



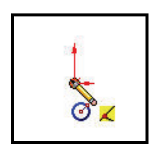
2 Open a Sketch.

- Click **Sketch** from the Context toolbar. The Sketch toolbar is displayed.

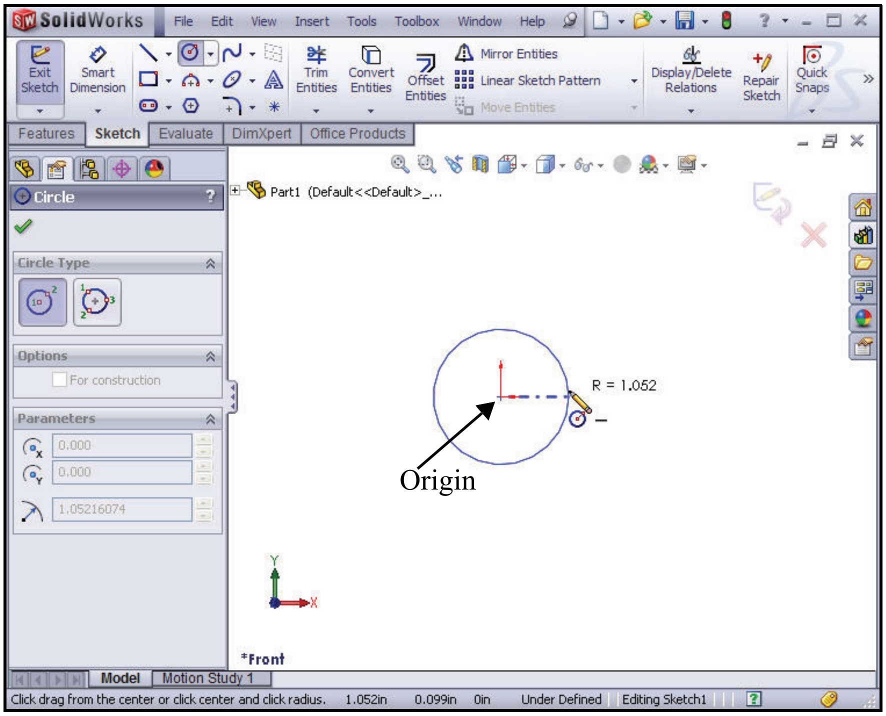


3 Sketch a circle with a center point at the Origin.

- Click the **Circle** Sketch tool. The Circle PropertyManager is displayed.
- Move the **mouse pointer** into the Graphics window. The cursor displays the Circle Sketch tool icon.

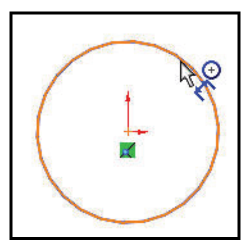


- Click the **Origin** of the circle. The cursor displays the Coincident to point feedback symbol as illustrated.
- Drag the **Mouse pointer** to the right of the Origin.
- Click a **position** in the Graphics window to create the circle as illustrated.



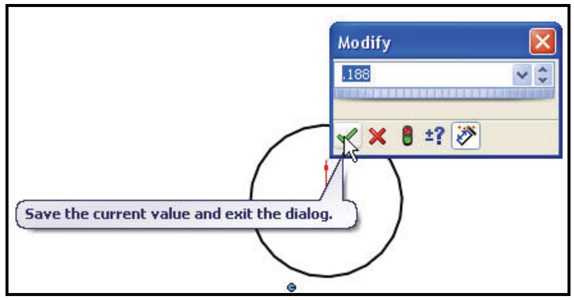
Dimensioning the Sketch

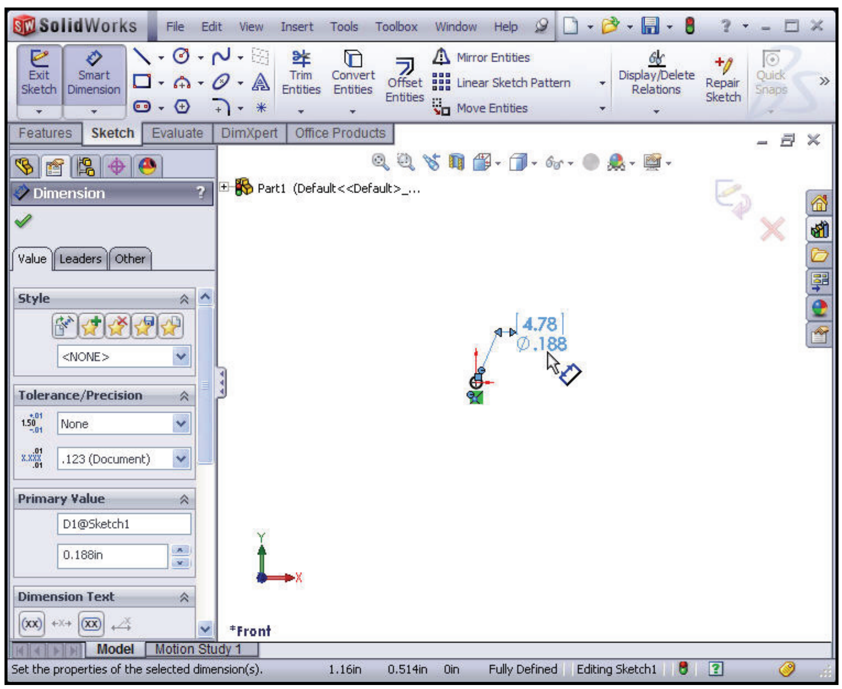
1 Dimension the sketch.



- Click the **Smart Dimension**  Sketch tool. The Smart Dimension  icon is displayed.
- Click the **circumference** of the circle. The cursor displays the diameter feedback symbol.

- Click a **position** diagonally above the circle in the Graphics window. A dimension is displayed in the Modify dialog box displaying the current value.
- Enter **.188**[4.78] in the Modify dialog box.
- Click the **green check mark** from the Modify dialog box. This saves the current value and exits the Modify dialog box. The dimension of the circle is .188[4.78].

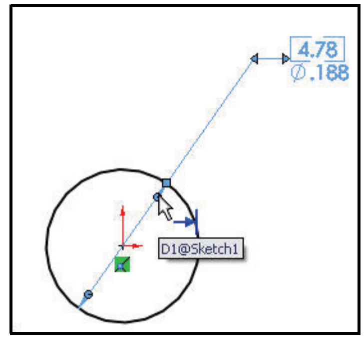




Note: The circular sketch is centered at the Origin. The dimension indicates the diameter of the circle is .188[4.78]. If your sketch is not correct, select the Undo tool from the Menu bar toolbar.

2 Fit the model to the Graphics window.

- Press the **f** key on the keyboard. The model moves to the left.
- Click and drag the **dimension text** .188[4.78] closer to the sketch.
- **Perform** this procedure until you can clearly view the sketch as illustrated.
- If needed, click the **control point** to flip the dimension arrow head.

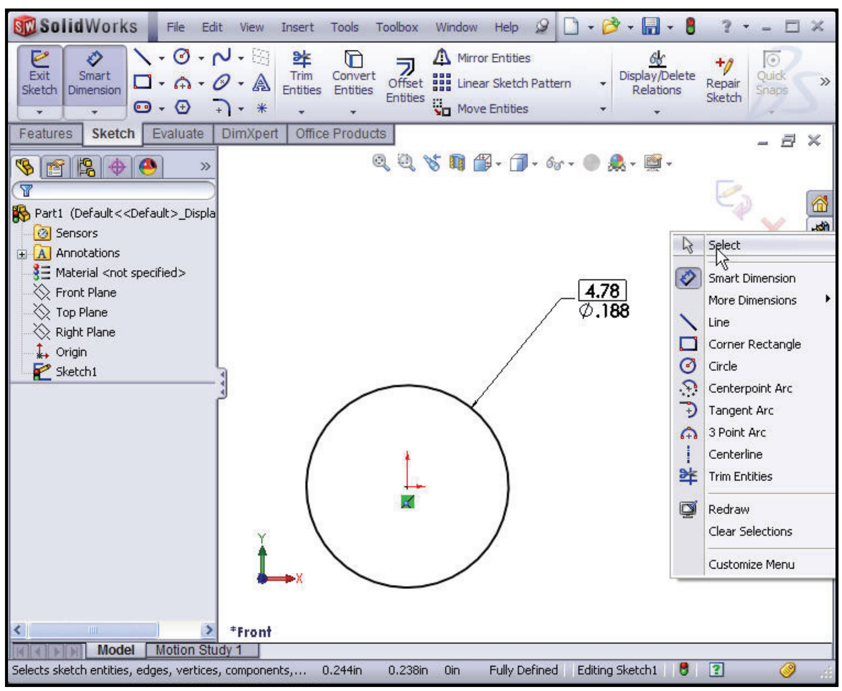


Tip: Press the **z** key to decrease the size of model in the Graphics window.

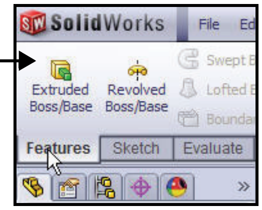
Tip: Press the **Shift** + **z** keys to increase the size of model in the Graphics window.

3 Deactivate the Smart Dimension tool.

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



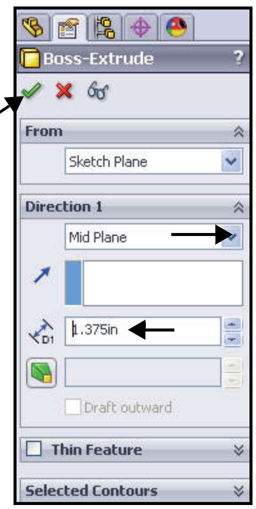
Inserting an Extruded Base Feature



Start the translation of the initial design function and geometric requirement into SolidWorks features. What are features?

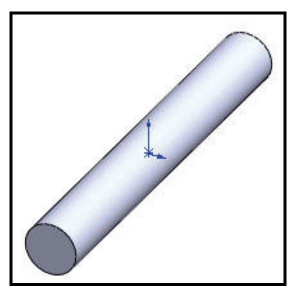
- Features are geometry building blocks.
- Features add or remove material.
- Features are created from sketched profiles or from edges and faces of existing geometry.

You will first utilize the Extruded Boss/Base feature. The Extruded Boss/Base feature adds material to the part. The Extruded Base feature is the first feature of the Axle. An extrusion extends the cylindrically profile along the path.



1 **Insert an Extruded Base feature.**
- Click the **Features** tab from the CommandManager as illustrated. The Features toolbar is displayed.
- Click the **Extruded Boss/Base** Features tool. The Boss-Extrude PropertyManager is displayed.
- Select **Mid Plane** for the End Condition in Direction 1.
- Enter **1.375**[34.93] for Depth.
- Click **OK** from the Boss-Extrude PropertyManager. Boss-Extrude1 is displayed in the FeatureManager.

2 **Fit the model to the Graphics window.**
- Press the **f** key.
- Click a **position** in the Graphics window.



The Boss-Extrude PropertyManager displays the parameters utilized to define the feature. The Mid Plane End Condition in the Direction 1 box extrudes the sketch equally on both sides of the Sketch plane. The depth 1.375[34.93] defines the distance.

Extrude features add material. Extrude features require the following:

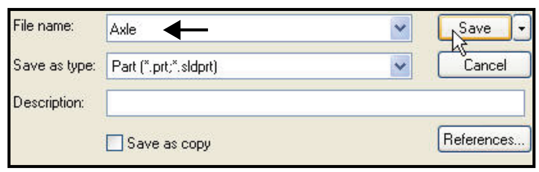
- Sketch plane.
- Sketch.
- Depth.

The Sketch plane is the Front Plane. The sketch is a circle with the diameter of .188in[4.78mm]. The depth is 1.375in[34.93mm].

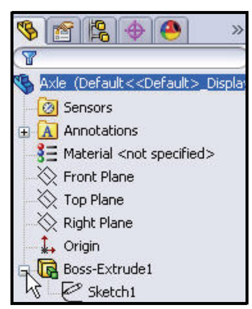
Tip: The OK button is just one way to accept and complete the process. A second method is the set of OK/Cancel buttons in the upper-right corner of the Graphics window.

Saving the Axle Part

3 Save the Axle part.



- Click **File**, **Save** or click **Save** from the Menu bar toolbar.
- Select the Save in folder, **SolidWorks-Trebuchet**. This is the folder that you downloaded from the SolidWorks EDU Curriculum.
- Select **Part** from Save as type.
- Enter **Axle** for File name.
- Click **Save**. The extension, *.sldprt is added automatically. The Axle FeatureManager is displayed.



4 View Sketch1.

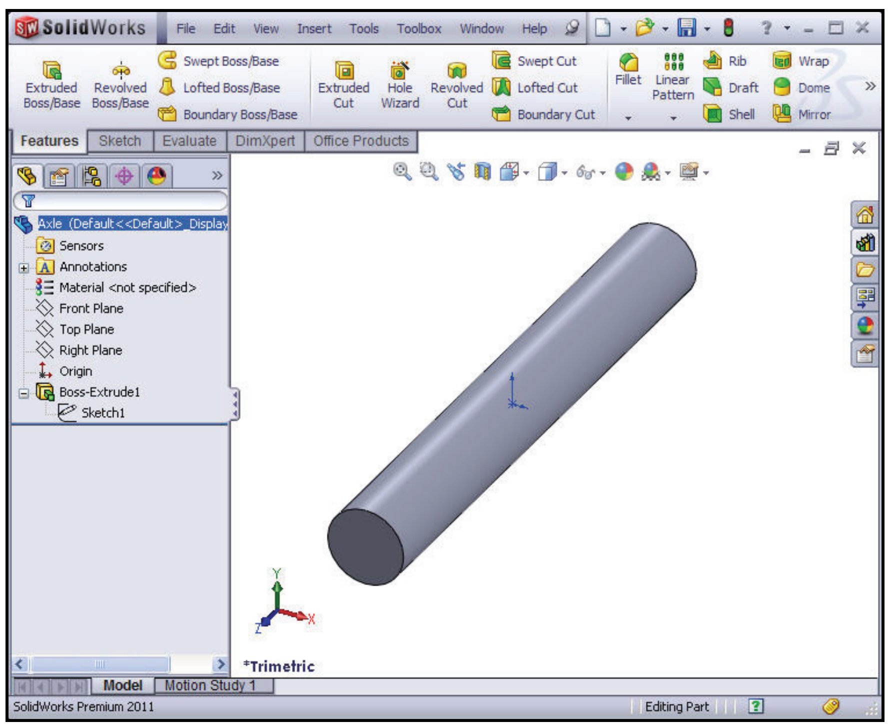
- **Expand** Boss-Extrude1 from the Axle FeatureManager. Sketch1 is displayed. Sketch1 is fully defined. In a fully defined sketch, all the lines and curves in the sketch and their positions are described by dimensions, relations, or both.

Note: With SolidWorks software, it is not necessary to fully dimension or define sketches before you use them to create features. However, you should always fully defined sketches before you consider the part complete for manufacturing.

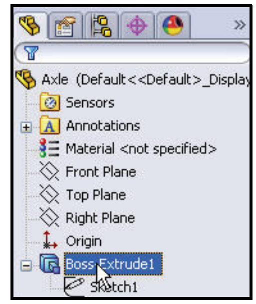
Sketches are generally in one of the following states:

- **Fully defined**. All the lines and curves in the sketch, and their positions, are described by dimensions or relations, or both.
- **Over defined**. Some dimensions or relations, or both, are either in conflict or are redundant.
- **Under defined**. Some of the dimensions or relations in the sketch are not defined and are free to change.

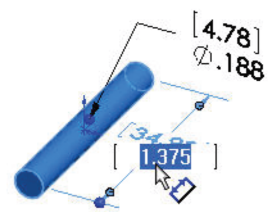
Note: The SketchXpert PropertyManager is displayed as soon as you over-define a sketch.



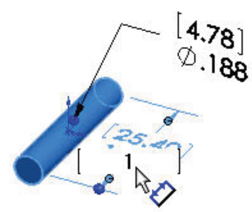
Modifying the Dimension of the Axle



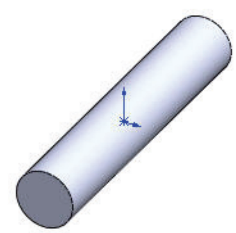
1 View the Boss-Extrude1 dimension.
- Click **Boss-Extrude1** from the Axle FeatureManager. View the dimensions in the Graphic window.
- Click and drag the **dimension text** off the model. The Dimension PropertyManager is displayed
- Click the **1.375**[34.93]dimension text in the Graphics window as illustrated.



2 Modify the length.
- Enter **1**[25.4].



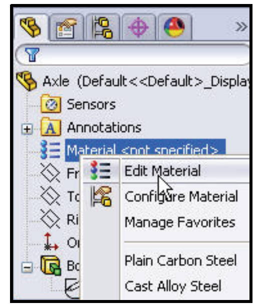
3 Save the model.
- Click **inside** the Graphics window.
- Click **Save** .



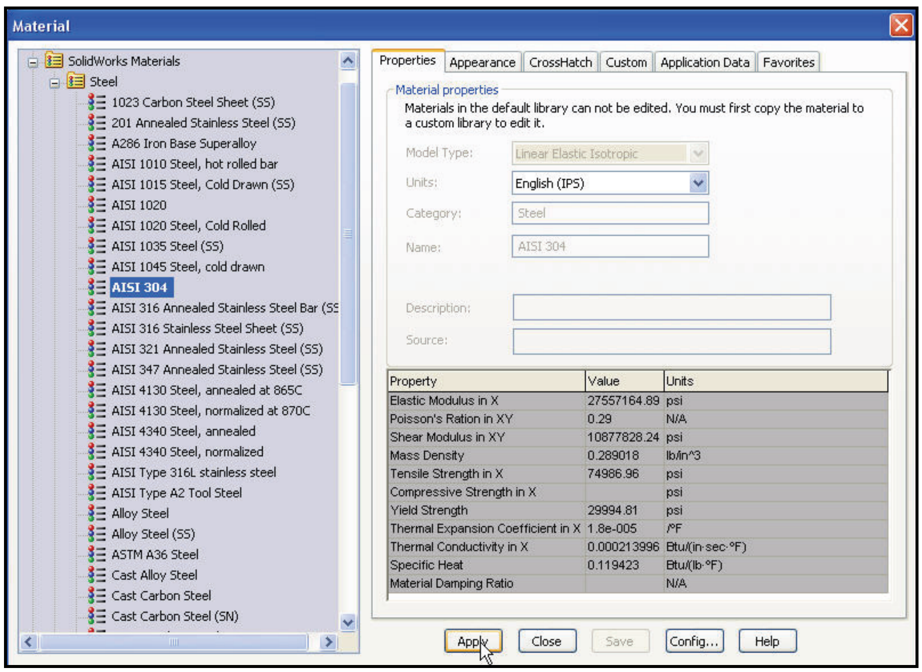
Tip: Click or double-click a feature in the Graphics window or from the FeatureManager to display the dimensions.

Note: Click **View**, **Origins** from the Menu bar to view the Origin in the Graphics window.

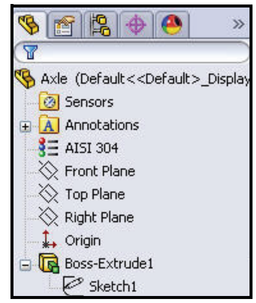
Applying Material to the Axle



1. **Apply material to the Axle part.**
   - Right-click **Material** from the Axle FeatureManager.
   - Click **Edit Material**. The Materials dialog box is displayed. View your options.
   - **Expand** the Steel category.
   - Select **AISI 304**. View the properties.
   - Click **Apply**.
   - Click **Close**.



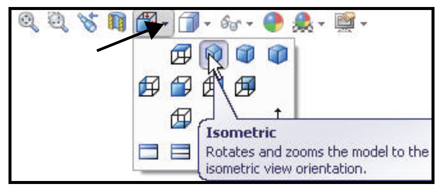
- View the updated Axle FeatureManager.



Inserting a Chamfer Feature

1 Orient the view.

- Click **Isometric** from the Heads-up View toolbar.



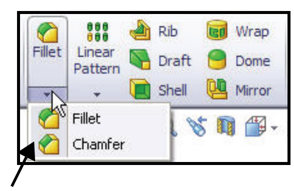
Note: Various modeling operations require you to view and select details of a model, no matter how small they might be. SolidWorks has numerous view manipulation tools that allow you to perform this function.

Zoom to Area, and Zoom to Fit are a few examples of these tools.

2 Insert a Chamfer feature.

The Chamfer tool creates a beveled feature on selected edges, faces, or a vertex.

- Click the **Chamfer** Features tool from the Consolidated drop-down Feature toolbar. The Chamfer PropertyManager is displayed.

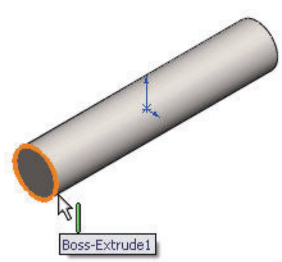


3 Zoom out on the Axle.

- Press the **z** key twice from the key board.

4 Select the edges to chamfer.

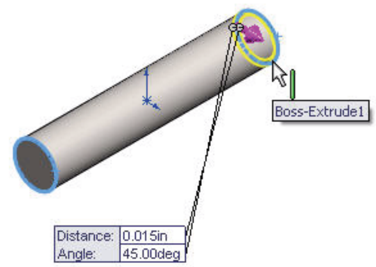
- Click the **front circular edge** of the Axle as illustrated. Edge <1> is displayed in the Chamfer Parameters box.



Tip: Press the **z** key to decrease the size of model in the Graphics window.

Tip: Press the **Shift** + **z** keys to increase the size of model in the Graphics window.

- Click the **back circular edge** of the Axle. Edge<2> is displayed in the Chamfer Parameters box.



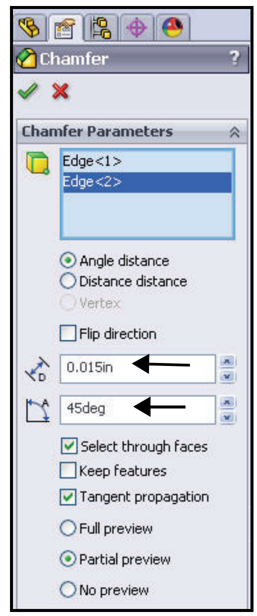
5 Set the Chamfer distance and angle.

- Enter .**015**[.38] for Distance.
- Enter **45** for Angle.

6 Accept the default values and view the results.

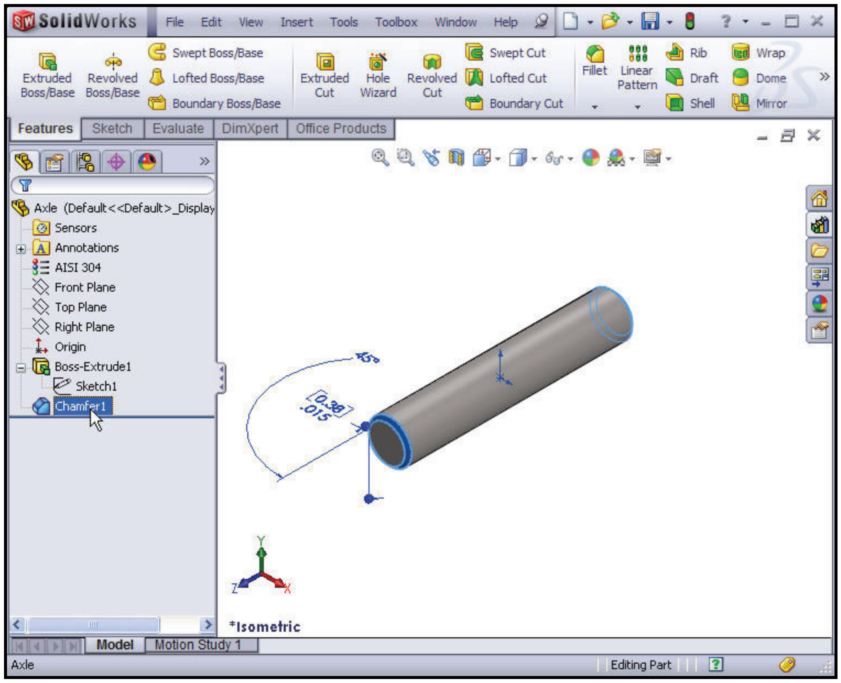
- Click **OK** from the Chamfer PropertyManager.

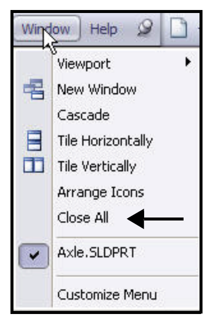
Chamfer1 is displayed in the FeatureManager. The Axle is displayed in the Graphics window with the chamfer feature on the selected edges.



7 View the dimensions.

- Click **Chamfer1** in the FeatureManager. View the dimensions.





8 Save the model.

- Press the **Esc** key.
- Click **Save** .

9 Close all models.

- Click **Window**, **Close All** from the Menu bar toolbar. You just finished your first SolidWorks part!

PhotoView 360

PhotoView 360 is a software solution from SolidWorks, fully integrated into the SolidWorks software to create photo-realistic images directly from SolidWorks models.

Renderings can be created from SolidWorks parts and assemblies, but not drawings. PhotoWorks can product photo-realistic images to add visual impact to your presentations and documents.

Some of the key features of PhotoView 360 are:

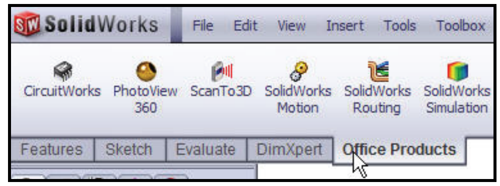
- **Fully integrated into SolidWorks**: PhotoView 360 software is supplied as a SolidWorks dynamic link library, .dll add-in. The menu bar is displayed whenever a SolidWorks part or assembly document is open.
- **Appearances**: Appearances are used in PhotoView 360 to specify model surface properties such as color, texture, reflectance, and transparency. PhotoWorks is supplied with numerous predefined appearances. Other appearances can be downloaded from web sites.
- **Lighting**: Lights may be added in the same way a photographer adds lights when taking photographs. PhotoView 360 contains numerous predefined lighting schemes to simplify and speed up the rendering process.
- **Scenes**: Each SolidWorks model is associated with a PhotoView 360 scene, for which you can specify properties such as rooms, environments, and backgrounds.
- **Decals**: Images, such as company logos, can be applied to models.
- **Output**: PhotoView 360 can output to the screen, a printer, or a graphics file.

In the next section, you will use the PhotoView 360. You will then use additional features and functions of PhotoView 360 throughout the book.

Creating a Photorealistic Image

1 Open the Axle part.

- Click **Open** from the Menu bar menu.
- Select the **SolidWorks-Trebuchet** folder.
- Select **Part** from Files of type.
- Double-click **Axle**. The Axle FeatureManager is displayed. The Axle is displayed in the Graphics window.



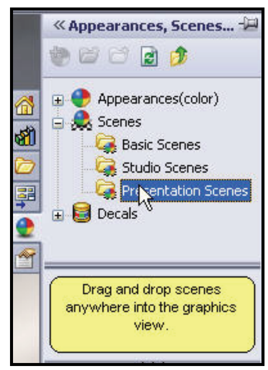
2 Activate PhotoView 360.

- Click the **Office Products** tab in the CommandManager.
- Click **PhotoView 360. The** PhotoView 360 button is displayed in the Menu bar toolbar.

3 Create a PhotoView Scene.

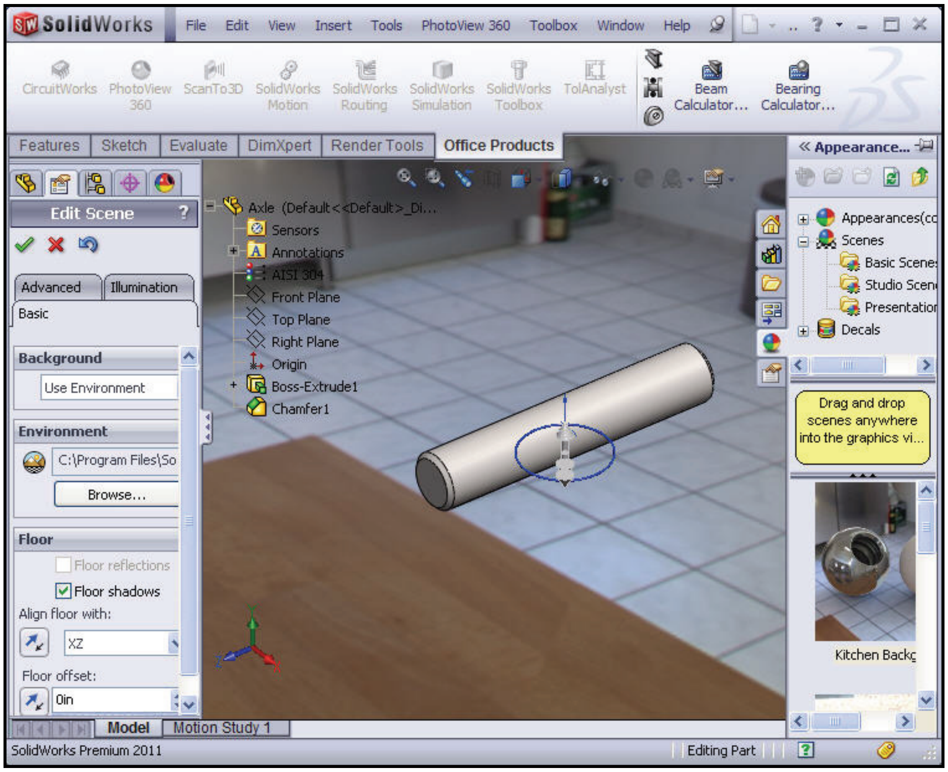
- Click **PhotoView 360** from the Menu bar toolbar.
- Click **Edit Scene** from the drop-down menu. Click **Yes** if needed. The Edit Scene PropertyManager is displayed.

Tip: You can also select PhotoWorks Studio from the PhotoWorks toolbar. To activate the PhotoWorks toolbar, click **View**, **Toolbars**, **PhotoWorks** from the Menu bar toolbar.



4 Set the Scenery.

- Expand the **Scenes** folder in the Task Pane.
- Click the **Presentation Scenes** folder as illustrated. View your options.
- Drag and drop **Kitchen Background** in the graphics area. View the results.



5 **Close Edit Scene**.

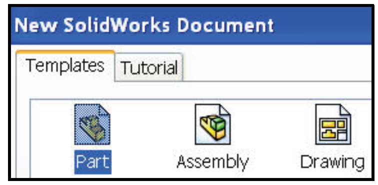
- Click **OK** from the Edit Scene PropertyManager.

6 **De-render the model.**

- Press the **z** key.

Note: The Apply Scene tool from the Heads-up VIew toolbar provides the ability to modify the model back ground in your Graphics window.

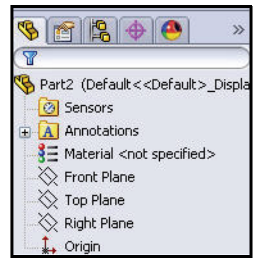
Creating the Shaft Collar Part



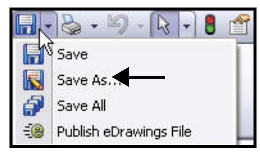
1 Create the Shaft Collar part.

- Click **New** from the Menu bar toolbar. The Templates tab is the default tab. Part is the default template from the New SolidWorks Document dialog box.

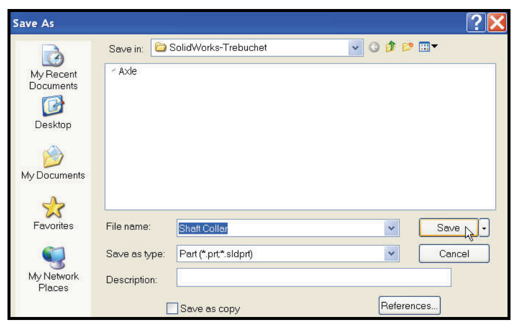
- Click **OK**. The Part2 FeatureManager is displayed.



2 Save the part.

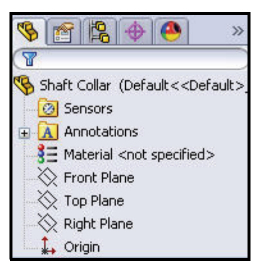


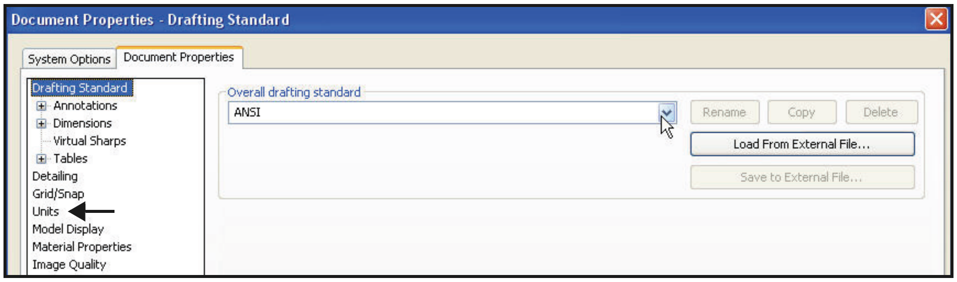
- Click **Save As** from the Menu bar Consolidated toolbar.
- Select the Save in folder, **SolidWorks-Trebuchet**.
- Select **Part** from Save as type.
- Enter **Shaft Collar** for File name.
- Click **Save**. The extension, *.sldprt is added automatically. The Shaft Collar FeatureManager is displayed.



3 Set the Drafting Standard.

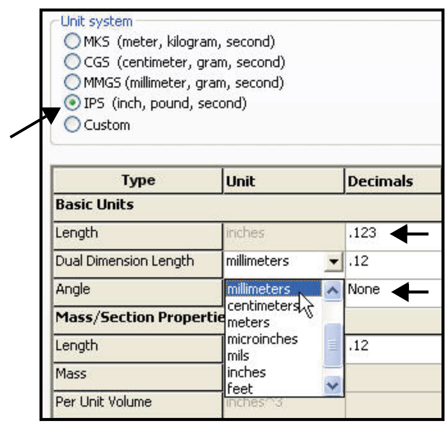
- Click **Options**, **Document Properties** tab from the Menu bar toolbar. The Document Properties - Drafting Standard dialog box is displayed.
- Select **ANSI** from the drop-down menu for Drafting Standard.





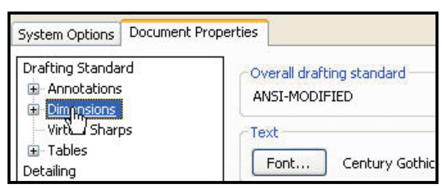
4 Set unit system and length.

- Click **Units**.
- Click **IPS** (inch, pound, second) for Unit system.
- Select **.123** for Length unit decimal place.
- Select **None** for Angle decimal place.
- Select **millimeters** for Dual Dimension Length.



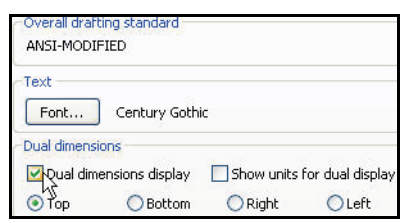
5 Set dual unit display.

- Click **Dimensions** from the Document Properties dialog box.
- Check the **Dual dimensions display** box.



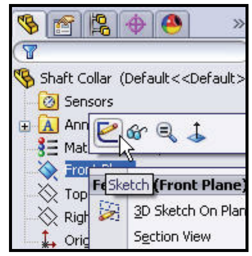
6 Set System options.

- Click **OK** from the Document Properties-Dimensions dialog box.



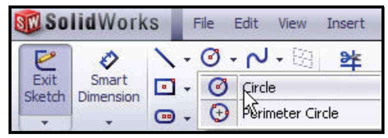
7 Open a sketch.

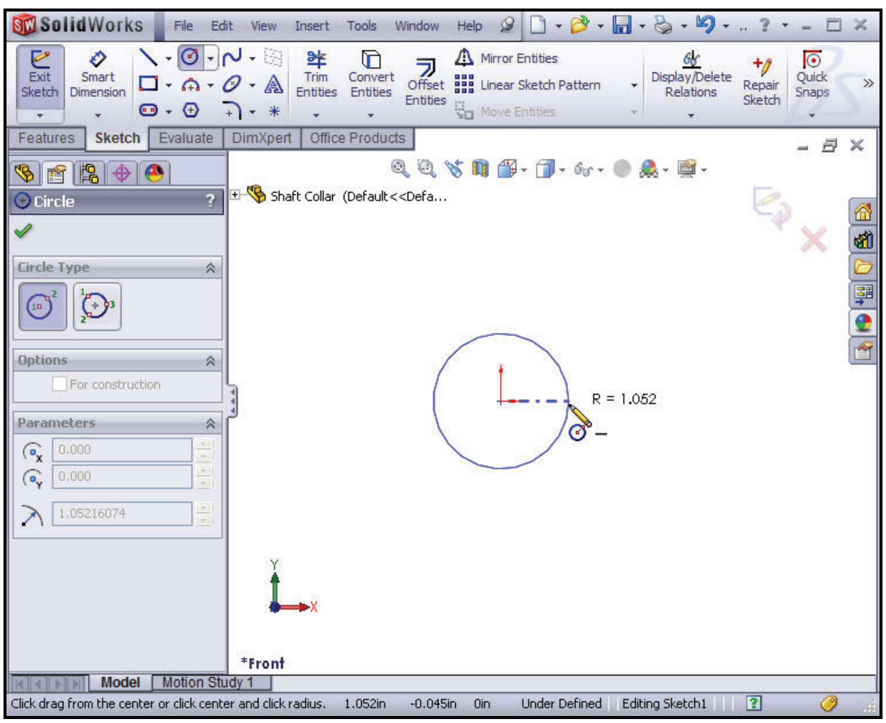
- Right-click **Front Plane** from the FeatureManager.
- Click **Sketch** from the Context toolbar. The Sketch toolbar is displayed.

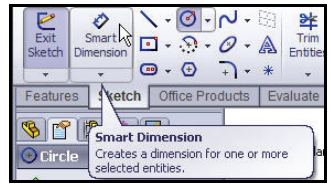


8 Sketch a circle with a center point at the Origin.

- Click the **Circle** Sketch tool.
- Click the **Origin** of the circle. The cursor displays the Coincident to point feedback symbol.
- Drag the **Mouse pointer** to the right of the Origin as illustrated.





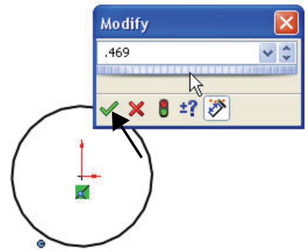


- Click a **position** in the Graphics window to create the circle.

9 Add a dimension.

- Click the **Smart Dimension** Sketch tool.

Note: View the mouse pointer icon.

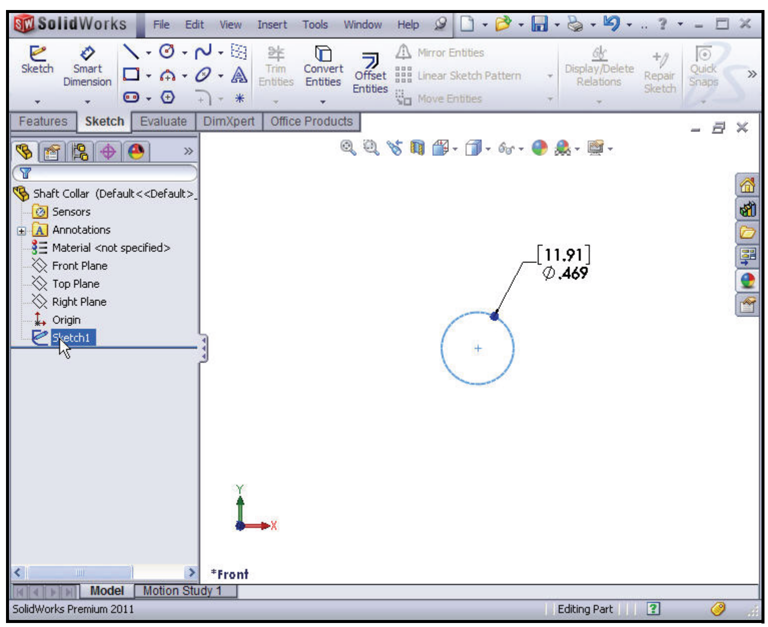


- Click the **circumference** of the circle. The cursor displays the diameter feedback symbol.
- Click a **position** diagonally above the circle in the Graphics window. A dimension is displayed in the Modify box.
- Enter **.469**[11.91] in the Modify box.
- Click the button from the Modify dialog box.

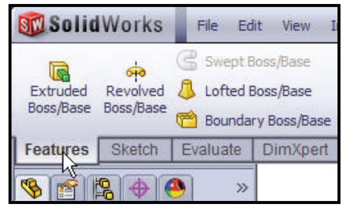
10 View the results.

- Click **OK** from the Dimension PropertyManager.
- Click the **Rebuild** tool from the Menu bar toolbar.
- Click **Sketch1** from the Shaft Collar FeatureManager. Sketch1 is highlighted. The dimension is displayed in the Graphics window.

Note: If needed, fit the drawing to the Graphics window.

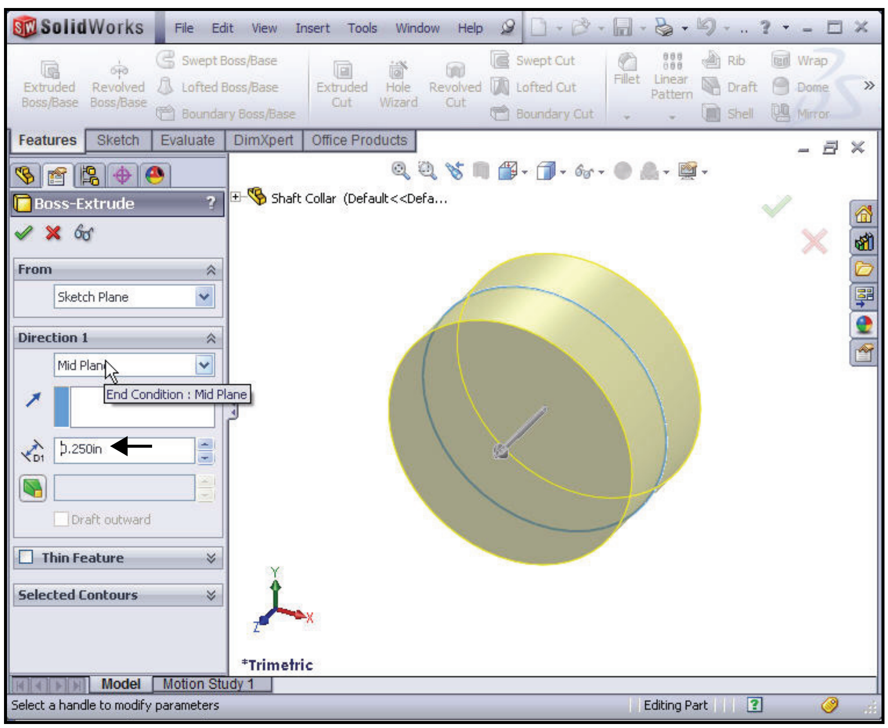


Inserting an Extruded Base Feature

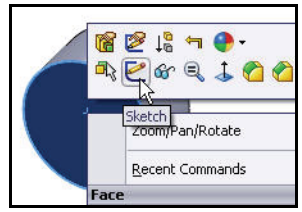


1 Insert an Extruded Base feature.

- Click the **Features** tab from the CommandManager.
- Click the **Extruded Boss/Base** Features tool. The Boss-Extrude PropertyManager is displayed.
- Select **Mid Plane** for End Condition in Direction 1.
- Enter **.250**[6.35] for Depth.
- Click **OK** from the Boss-Extrude PropertyManager. Boss-Extrude1 is displayed in the Shaft Collar FeatureManager.

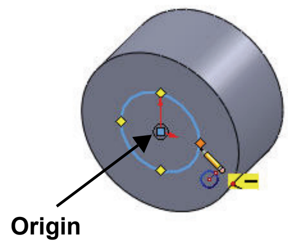


Inserting a new Sketch



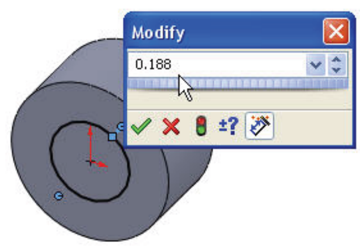
2 Insert a new sketch for the Extruded Cut feature.

- Right-click the **front circular face** of the Extrude1 feature for the Sketch plane. Extrude1 is highlighted in the FeatureManager
- Click **Sketch** from the Context toolbar. The Sketch toolbar is displayed.
- Click the **Circle** Sketch tool.
- Click the **Origin**. The cursor displays the Coincident to point feedback symbol.
- Drag the **mouse pointer** to the right of the Origin.
- Click a **position** in the Graphics window to create the circle.

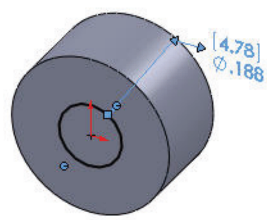


Adding a Dimension

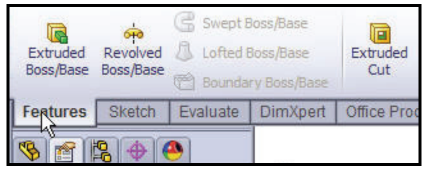
3 Add a dimension.



- Click the **Smart Dimension** Sketch tool.
- Click the **circumference** of the circle. The cursor displays the diameter feedback symbol.
- Click a **position** diagonally above the circle in the Graphics window. A dimension appears with the Modify box displaying the current value.
- Enter **.188**[4.78] in the Modify box.
- Click the button from the Modify box. The sketch is fully defined.



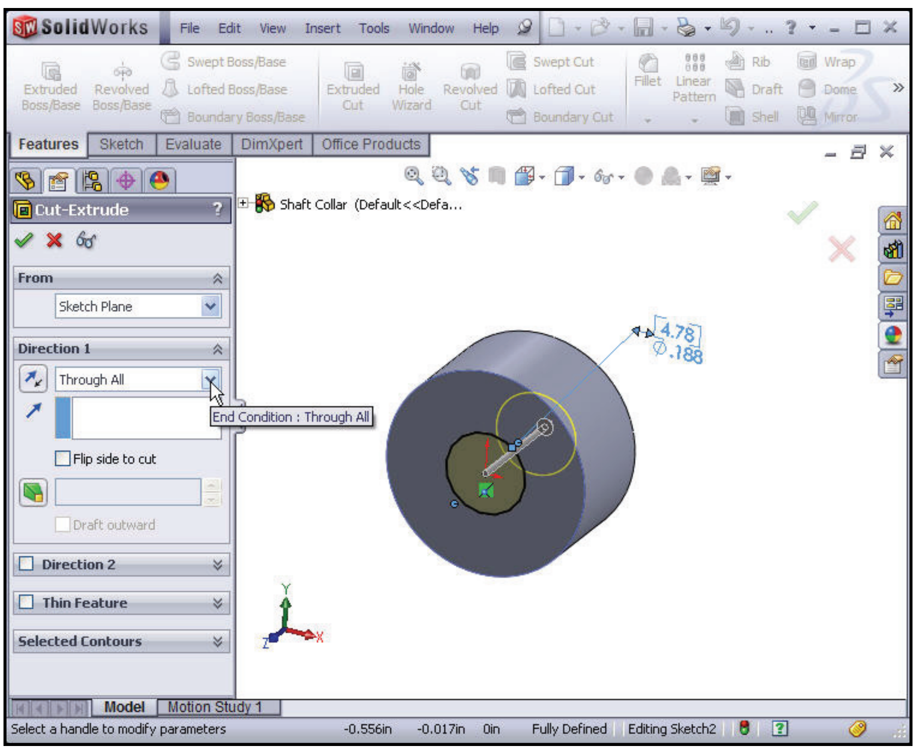
Inserting an Extruded Cut Feature



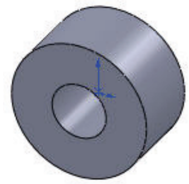
An Extruded Cut feature removes material. The Extruded Cut begins with a 2D sketch on the front face.

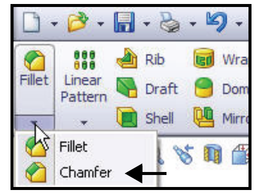
1 **Insert an Extruded Cut feature.**

- Click the **Features** tab from the CommandManager.
- Click the **Extruded Cut** Features tool. The Cut-Extrude PropertyManager is displayed.
- Select **Through All** for End Condition in Direction 1. The Through All End Condition removes material from the Front Plane through the Boss-Extrude1 feature. Note the direction of the Extrude feature.
- Click **OK** from the Cut-Extrude PropertyManager. Cut-Extrude1 is displayed in the FeatureManager.



Inserting a Chamfer Feature

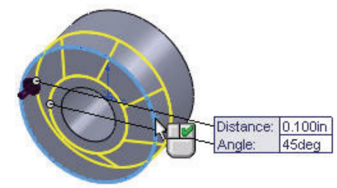




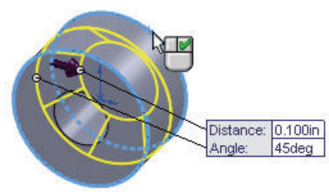
1 Insert a Chamfer feature.

- Click the **Chamfer** Features tool from the Consolidated drop-down toolbar. The Chamfer PropertyManager is displayed.

2 Select the chamfer edges.



- Click the **front circular edge** of the Shaft Collar as illustrated. View the default dimensions. Edge <1> is displayed in the Chamfer Parameters box.



- Click the **back circular edge** of the Shaft Collar. View the default dimensions. Edge<2> is displayed in the Chamfer Parameters box.

3 Set the Chamfer distance and angle.

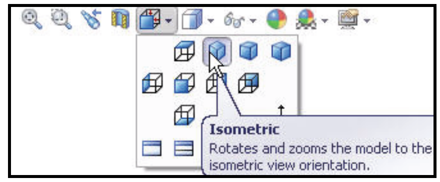
- Enter .**010**[.25] for Distance.
- Enter **45** for Angle.

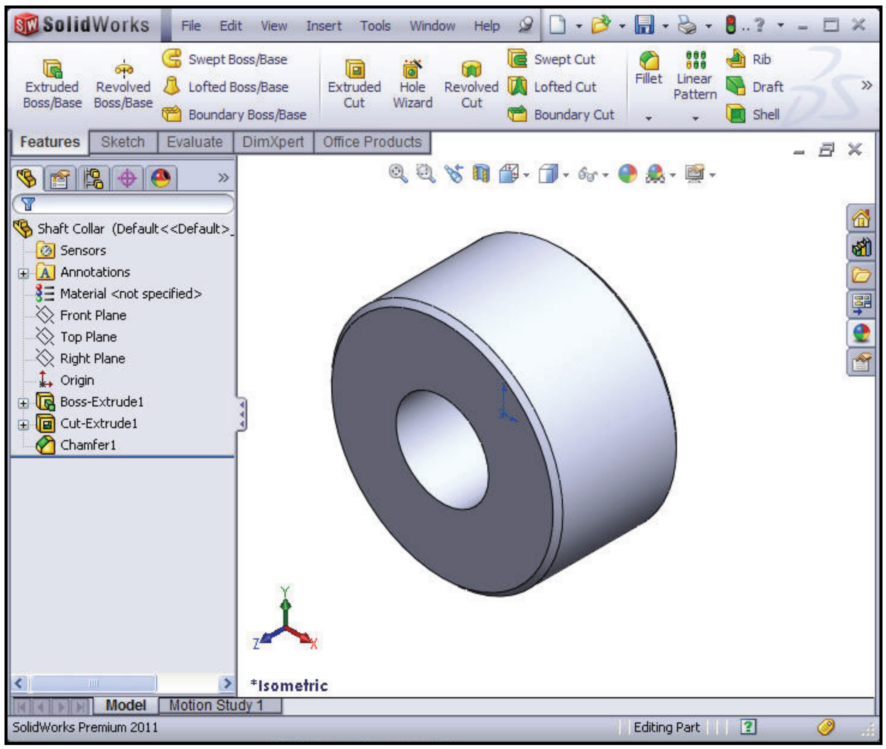
4 Accept the default values and view the results.

- Click **OK** from the Chamfer PropertyManager. Chamfer1 is displayed in the Shaft Collar FeatureManager.

5 Save the model.

- Click **Isometric** view.
- Click **Save** . View the results in the Graphics window.

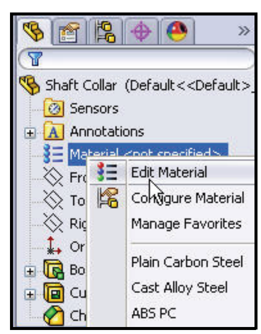




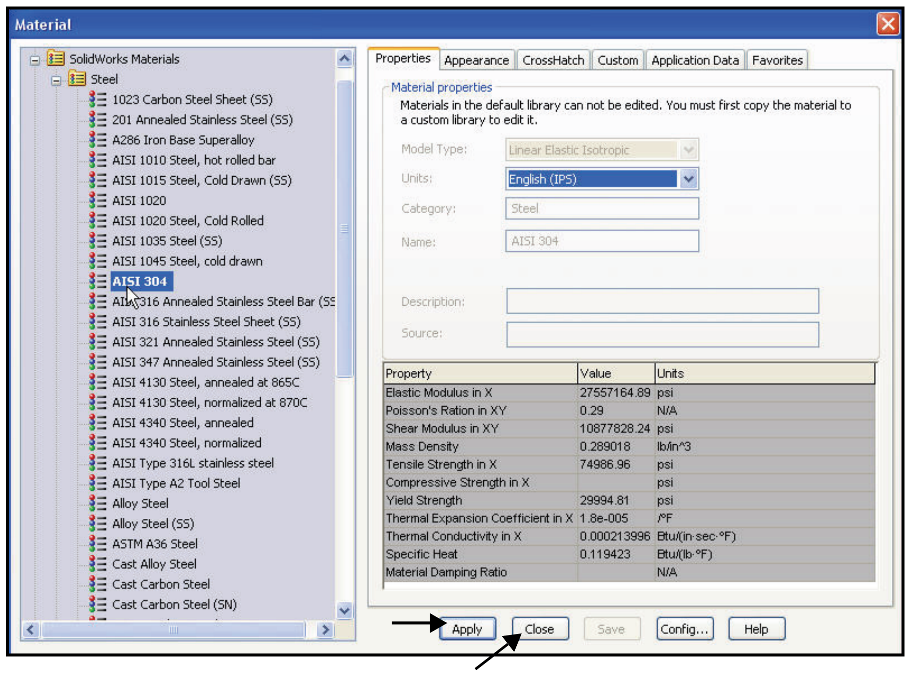
Applying Material to the Shaft Collar

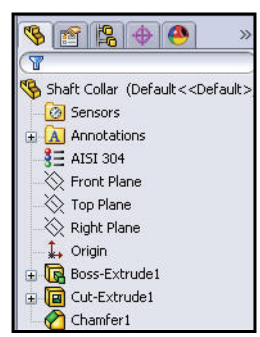
1 Apply material to the Shaft Collar.

- Right-click **Material** from the Shaft Collar FeatureManager.
- Click **Edit Material**. The Material dialog box is displayed.



- **Expand** the Steel category.
- Click **AISI 304**. View the material properties.
- Click **Apply**.
- Click **Close**.



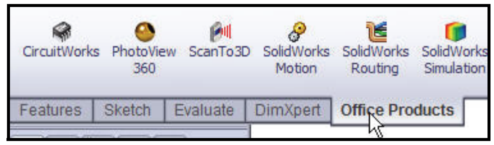


- AISI 304 is displayed in the Shaft Collar FeatureManager.

2 Save the model.

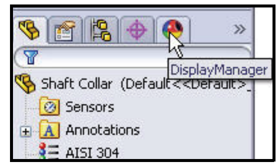
- Click **Save**.

Exploring the DisplayManager Tab and Applying Appearance



1 Activate PhotoView 360.

- Click the **Office Products** tab in the CommandManager.
- Click **PhotoView 360**. The PhotoView 360 button is displayed in the Menu bar toolbar.



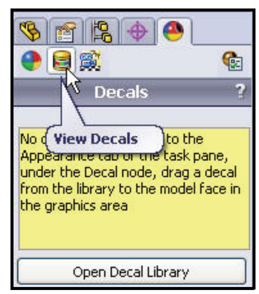
2 View the DisplayManager Tab.

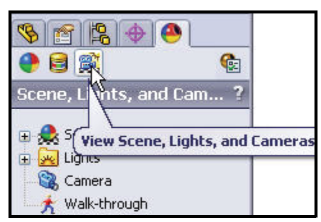
- Click the **DisplayManager**  tab as illustrated.

The Shaft Collar DisplayManager is displayed. The DisplayManager tab provides an outline view of the `Appearances, Decals and Scene, lights, and cameras` effects associated with the active SolidWorks part or assembly.

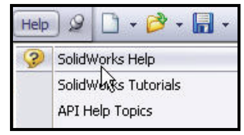
The DisplayManager also makes it easy to:

- Understand the way in which appearances and decal inheritance works.
- Select and edit appearances and decals associated with the model.
- Transfer appearances and decals between components, features, and faces.

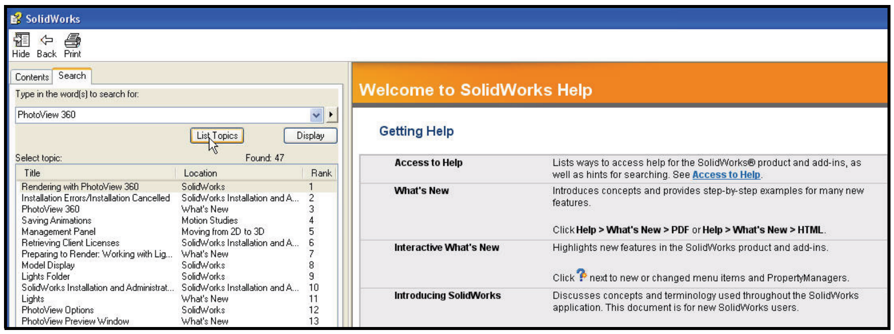




3 Explore PhotoView 360 Help.



- Click **Help**, **SolidWorks Help** from the Menu bar toolbar.
- Enter **PhotoView 360** under the Index tab.
- Click **List Topics**. View the help information.

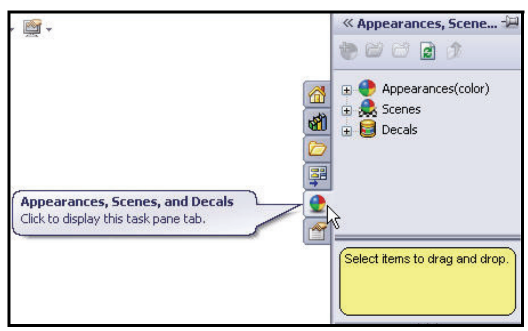


4 Close the SolidWorks help box.

- Click **Close** .

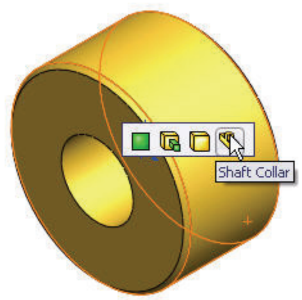
5 Activate the Appearances, Scenes, and Decals option in the Task Pane.

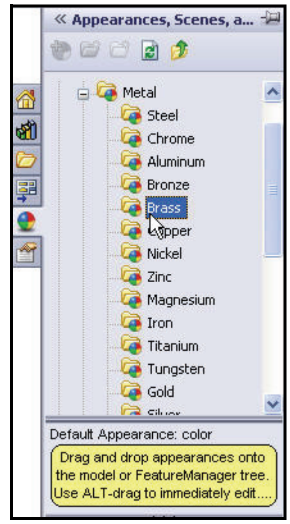
- Click the **Appearances, Scenes, and Decals**  tab from the Task Pane. View the default categories: *Appearances(color)*, *Scenes*, and *Decals*.



6 **Apply an Appearance**.

- **Expand** Appearances(color).
- **Expand** Metals.
- Click **Brass**.
- Drag and drop **matte brass** onto the model.
- Click the **Shaft Color** icon. View the results. This option provides the ability to either select a face, feature or the entire model.



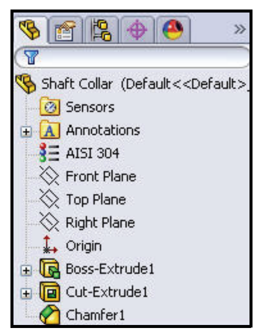


7 **Return to the Shaft Collar FeatureManager.**

- Click the **Part FeatureManager**  tab as illustrated.

8 **Save the model.**

- Click **Save** .

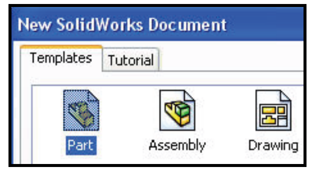


9 **Close all models.**

- Click **Window**, **Close All** from the Menu bar toolbar.

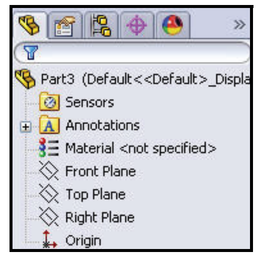
You created your second part.

Creating the 7 Hole Flatbar Part



1 **Create the 7 Hole Flatbar.**

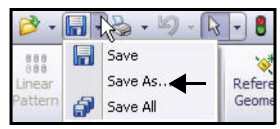
- Click **New** from the Menu bar toolbar. The Templates tab is the default tab. Part is the default template from the New SolidWorks Document dialog box.
- Click **OK**. The Part3 FeatureManager is displayed.

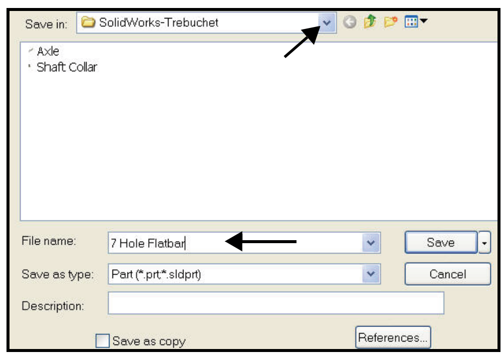


2 **Save the part.**

- Click **Save As** from the Menu bar Consolidated toolbar.
- Select the Save in folder, **SolidWorks-Trebuchet.**
- Select **Part** from Save as type.
- Enter **7 Hole Flatbar** for File name.
- Click **Save**.

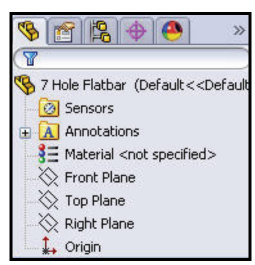
The extension, *.sldprt is added automatically. The 7 Hole Flatbar FeatureManager is displayed.

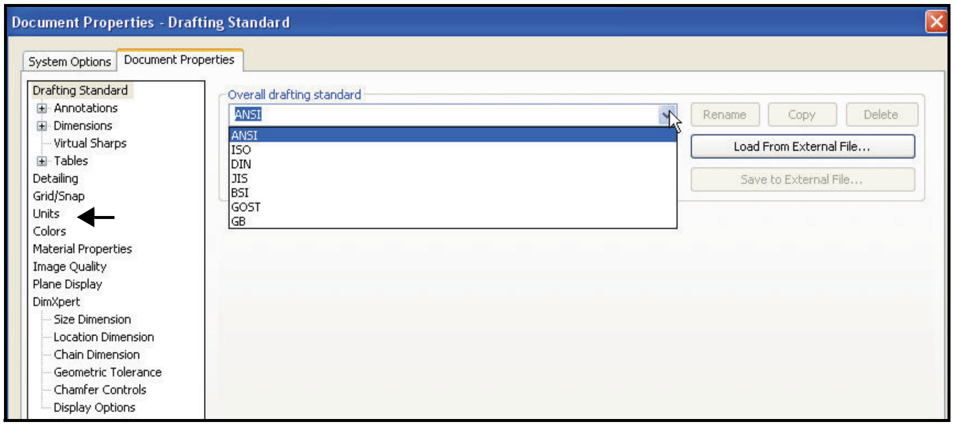




3 **Set Drafting Standard.**

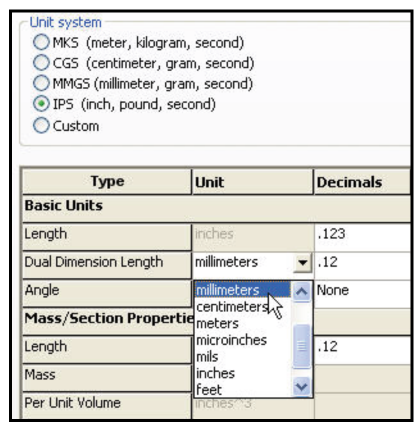
- Click **Options**, **Document Properties** tab from the Menu bar toolbar. The Document Properties - Drafting Standard dialog box is displayed.
- Select **ANSI** from the drop-down menu for Drafting Standard.





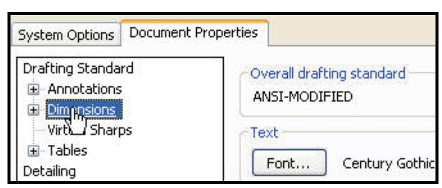
4 Set unit system and length.

- Click **Units**.
- Click **IPS** (inch, pound, second) for Unit system.
- Select **.123** for Length unit decimal place.
- Select **None** for Angle decimal place.
- Select **millimeters** for Dual Dimension Length.



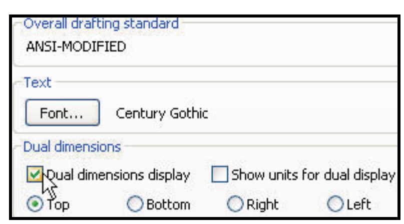
5 Set dual unit display.

- Click **Dimensions** from the Document Properties dialog box.
- Check the **Dual dimensions display** box.



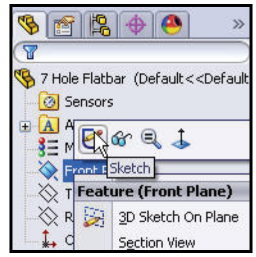
6 Set System options.

- Click **OK** from the Document Properties-Dimensions dialog box.



7 Select the Sketch plane.

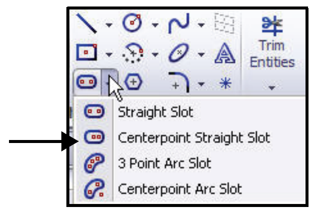
- Right-click **Front Plane** from the 7 Hole Flatbar FeatureManager. Front Plane is highlighted in the FeatureManager.



8 Create a slot sketch.

- Click **Sketch** from the Context toolbar.

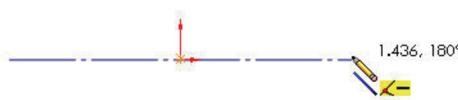
- Click the **Centerpoint Straight Slot** tool from the Consolidated Sketch toolbar. The Centerpoint Straight Slot sketches a straight slot from the centerpoint.

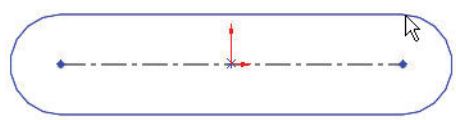


Note: The SolidWorks application defaults to the last used tool in a Consolidated toolbar.

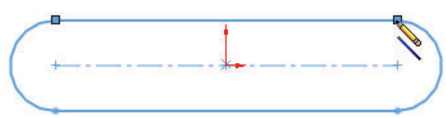
- Click the **Origin** for the center of the slot.
- Click a **position** directly to the right of the Origin.
- Click a **position** directly above the right point as illustrated.

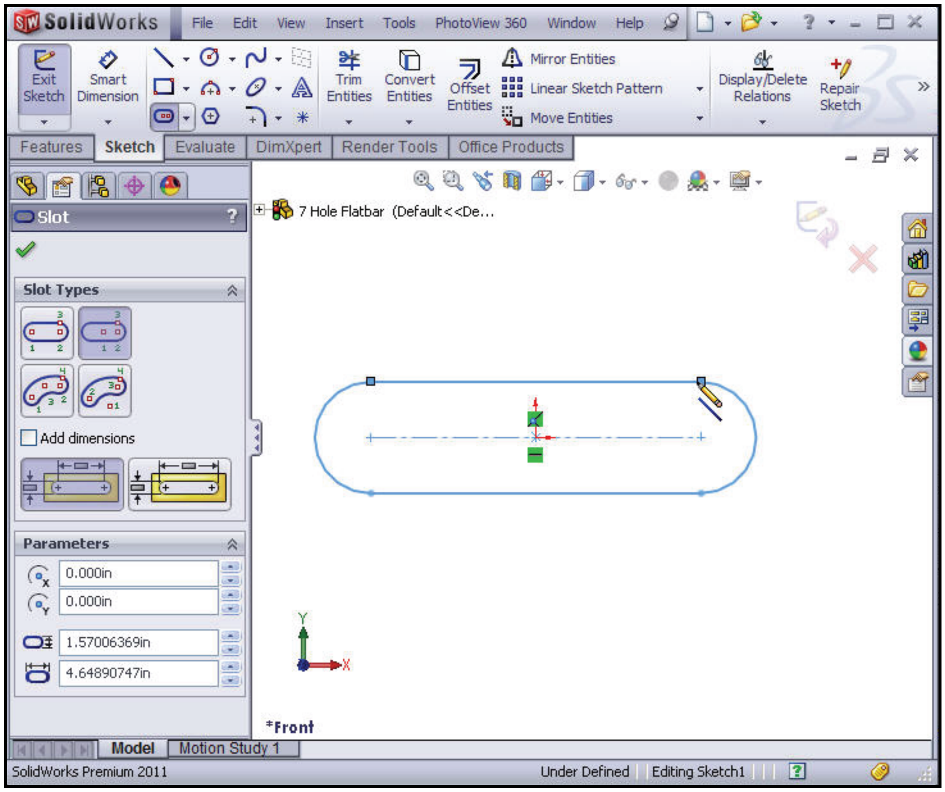
The slot sketch is symmetric about the Origin.





Note: If needed, utilize relations. A relation is a geometric constraint between sketch geometry.




SolidWorks
File Edit View Insert Tools PhotoView 360 Window Help
Exit Sketch
Smart Dimension
Trim Entities
Convert Entities
Offset Entities
Mirror Entities
Linear Sketch Pattern
Move Entities
Display/Delete Relations
Repair Sketch
Features Sketch Evaluate DimXpert Render Tools Office Products
Slot
Slot Types
Add dimensions
Parameters
0.000in
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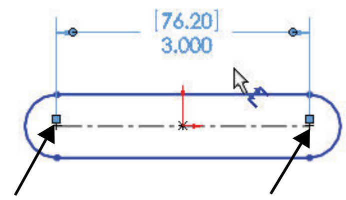
1.57006369in
4.64890747in
7 Hole Flatbar (Default<<De...
*Front
Model Motion Study 1
SolidWorks Premium 2011
Under Defined Editing Sketch1

Adding Dimensions

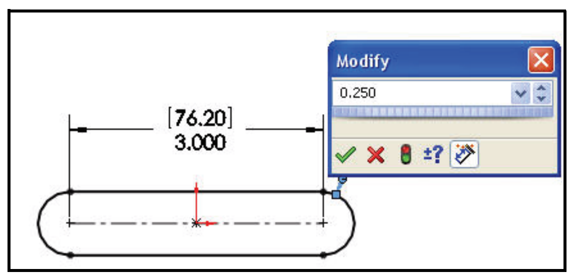
1 Add a horizontal dimension.

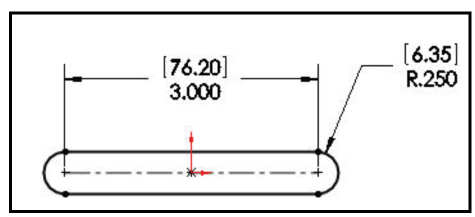
- Click the **Smart Dimension** Sketch tool.
- Click the **centerpoint** of the left Tangent Arc.
- Click the **centerpoint** of the right Tangent Arc.
- Click a **position** above the top horizontal line in the Graphics window.
- Enter **3.000**[76.2] in the Modify dialog box.
- Click the button from the Modify dialog box.



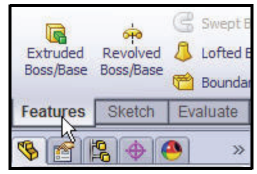
2 Add a radial dimension.

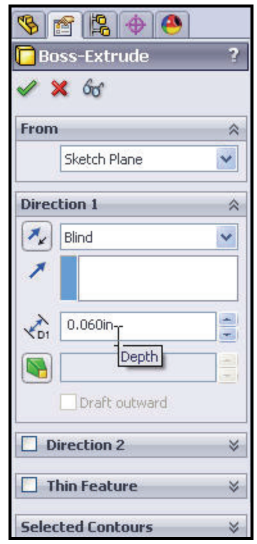
- Click the **right arc** of the 7 Hole Flatbar.
- Click a **position** diagonally to the right in the Graphics window.
- Enter **.250**[6.35] in the Modify dialog box.
- Click the button from the Modify dialog box.
- Click **OK** from the Dimension PropertyManager. The black sketch is fully defined.





Inserting an Extruded Base Feature

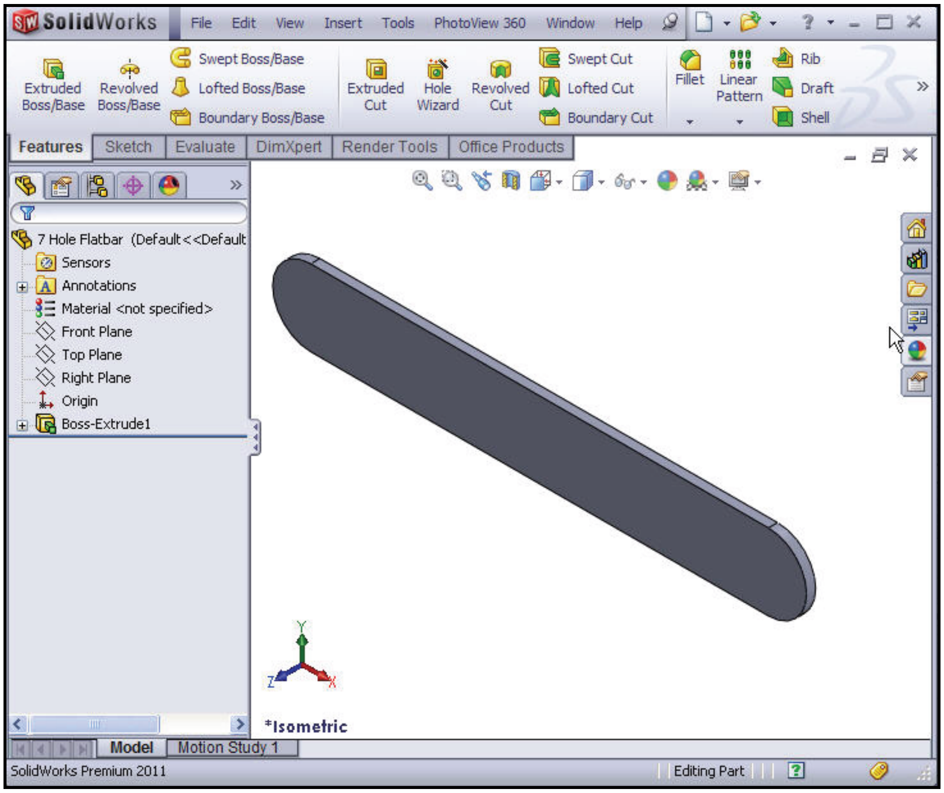




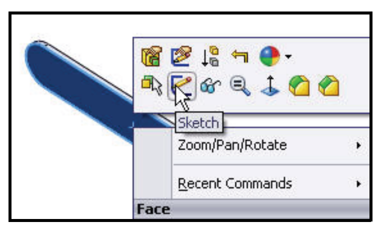
1 **Insert an Extruded Base feature.**
- Click the **Features** tab from the CommandManager.
- Click the **Extruded Boss/Base** Features tool. The Boss-Extrude PropertyManager is displayed. Blind is the default End Condition in Direction 1.
- Enter **.060**[1.52] for Depth.
- Click **OK** from the Boss-Extrude PropertyManager. Boss-Extrude1 is displayed.

2 **Fit the model to the Graphics window.**
- Press the **f** key.

3 **Save the model.**
- Click **Isometric** view from the Heads-up View toolbar.
- Click **inside** the Graphics window.
- Click **Save** .



Inserting an Extruded Cut Feature



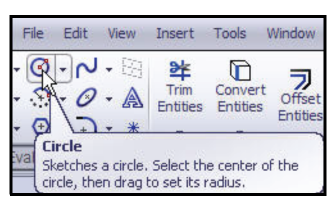
1 Insert a new sketch for the Extruded Cut feature.

- Right-click the **front face** of the 7 Hole Flatbar model. This is your Sketch plane. Boss-Extrude1 is highlighted in the FeatureManager.

Note: The process of placing the mouse pointer over an existing arc to locate its center point is call “wake up” You will perform this task.

2 Wake up the center point.

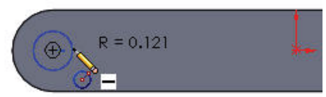
- Click **Sketch** from the Context toolbar. The Sketch toolbar is displayed.



- Click **Front view** from the Heads-up View toolbar.

- Click the **Circle** Sketch tool. The Circle PropertyManager is displayed.
- Place the **mouse pointer** on the left arc as illustrated. Do not click. The center point of the slot arc is displayed.

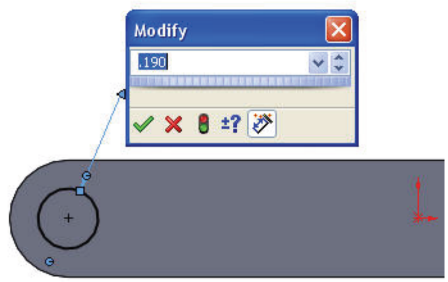
- Click the **center point** of the arc.
- Click a **position** to the right of the center point to create the circle as illustrated.



Adding a Dimension

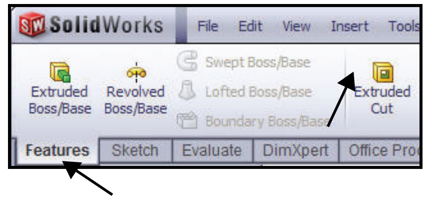
1 Add a dimension.

- Click the **Smart Dimension** Sketch tool.
- Click the **circumference** of the circle. The cursor displays the diameter feedback symbol.
- Click a **position** diagonally above and to the right of the circle in the Graphics window. A dimension appears with the Modify box displaying the current value.
- Enter **.190**[4.83] in the Modify box.



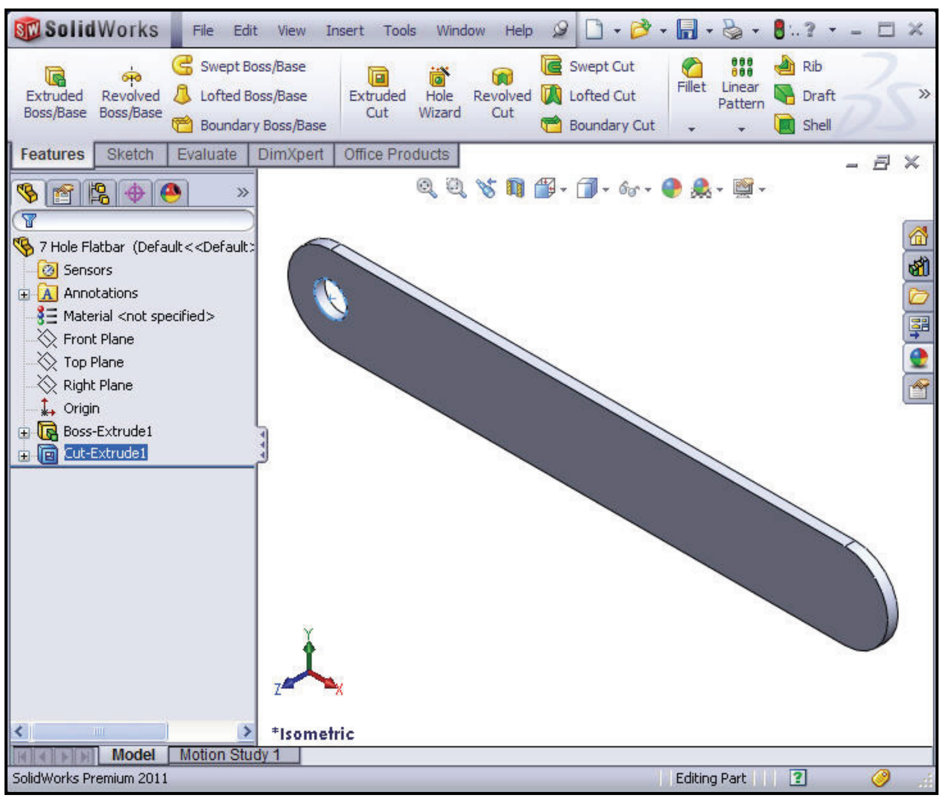
- Click the button from the Modify dialog box. The black sketch is fully defined.

Inserting an Extruded Cut

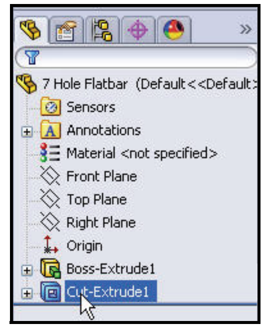


1 Insert an Extruded Cut feature.

- Click **Isometric** view from the Heads-up View toolbar.
- Click the **Features** tab from the CommandManager.
- Click the **Extruded Cut** Features tool. The Cut-Extrude PropertyManager is displayed.
- Select **Through All** for End Condition in Direction 1. The Through All End Condition removes material from the Front Plane through the Boss-Extrude1 feature.
- Click **OK** from the Cut-Extrude PropertyManager. Cut-Extrude1 is displayed in the FeatureManager.



2 Save the model.

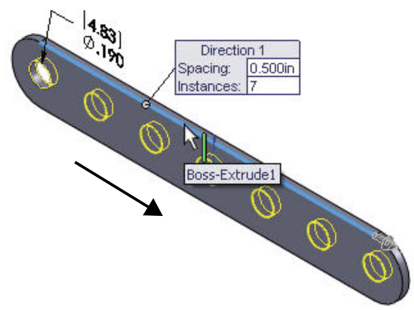


- Click **Save** .
- Click **inside** the Graphics area.
- Click **Cut-Extrude1** from the FeatureManager.

The Cut-Extrude1 feature is displayed in blue. The blue Cut-Extrude1 icon indicates that the feature is selected.

Select features by clicking their icon in the FeatureManager or selecting geometry in the Graphics window.

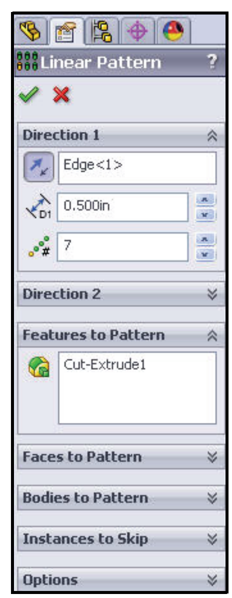
Inserting a Linear Pattern feature



Use the Linear Pattern tool to create multiple instances of one or more feature that you can space uniformly along one or two linear paths.

1 Insert a Linear Pattern.

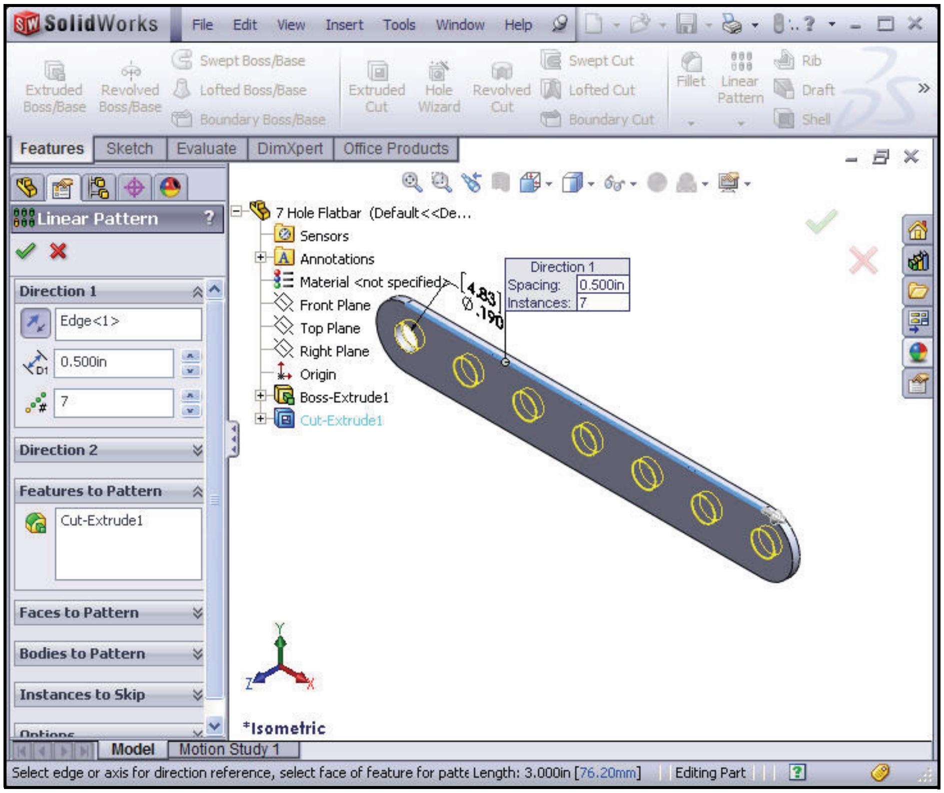
- Click the **Linear Pattern** Features tool. The Linear Pattern PropertyManager is displayed.
- Click the **top front edge** of the Boss-Extrude1 feature in the Graphics window for Direction 1.
- The Direction arrow points to the right. Click the **Reverse Direction** button if required. Edge<1> is displayed in the Pattern Direction box for Direction1.
- Enter **0.5**[12.7] for Spacing.
- Enter **7** for Number of Instances. Instances are the number of occurrences of a feature.



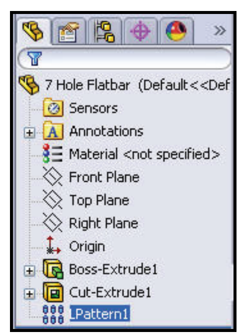
Note: If Cut-Extrude1 is not displayed in the Features to Pattern, perform the following tasks:

- Click inside the **Features to Pattern** box.
- **Expand** the 7 Hole FeatureManager in the Graphics window.

- Click **Cut-Extrude1** from the FeatureManager. Cut-Extrude1 is displayed in the Features to Pattern box.

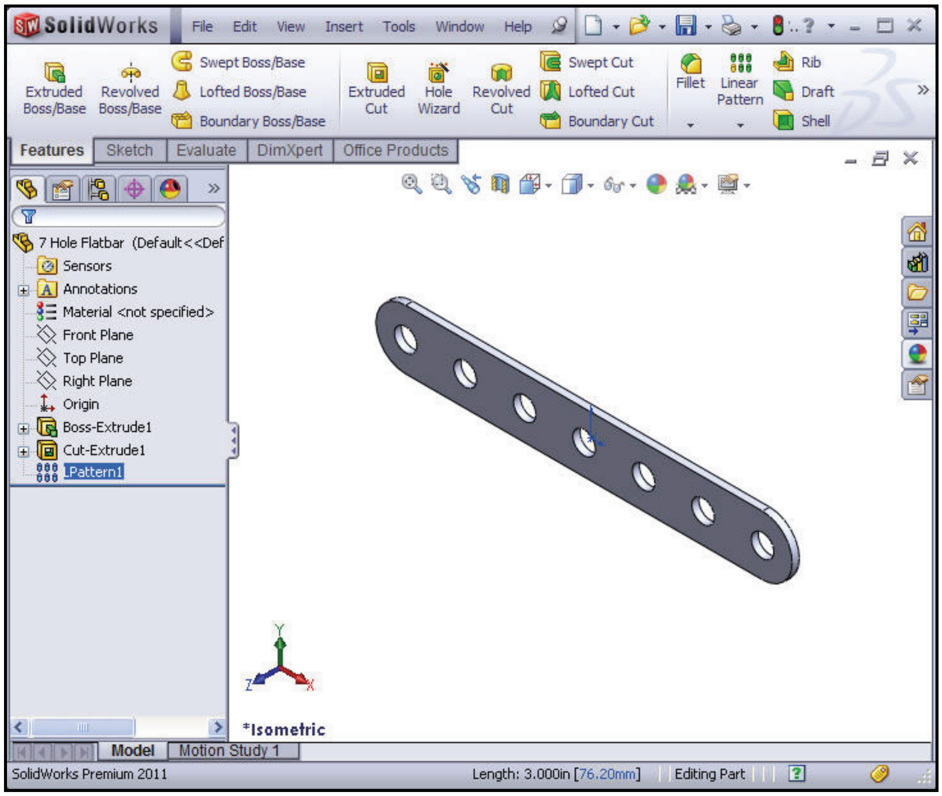


- Click **OK** from the Linear Pattern PropertyManager. The LPattern1 feature is displayed in the 7 Hole Flatbar FeatureManager.



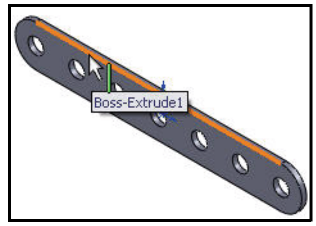
2 Save the model.

- Click **Isometric** view.
- Click **Save**.


SolidWorks
File Edit View Insert Tools Window Help
Extruded Boss/Base
Revolved Boss/Base
Swept Boss/Base
Lofted Boss/Base
Boundary Boss/Base
Extruded Cut
Hole Wizard
Revolved Cut
Swept Cut
Lofted Cut
Boundary Cut
Fillet
Linear Pattern
Rib
Draft
Shell
Features Sketch Evaluate DimXpert Office Products
7 Hole Flatbar (Default<<Def
Sensors
Annotations
Material <not specified>
Front Plane
Top Plane
Right Plane
Origin
Boss-Extrude1
Cut-Extrude1
LPattern1
*Isometric
Model Motion Study 1
SolidWorks Premium 2011
Length: 3.000in [76.20mm]
Editing Part

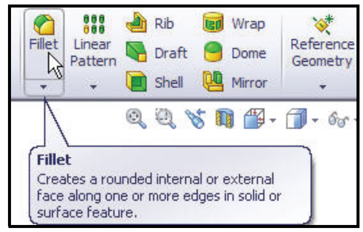
Inserting a Fillet

A Fillet feature removes sharp edges. Fillets are generally added to the solids, not the sketch. Small corner edge Fillets are grouped together.

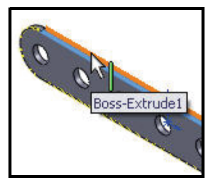


1 Insert a Fillet feature.

- Click the **front top edge** of the Boss-Extrude1 feature as illustrated.
- Click the **Fillet**  Features tool. The Fillet PropertyManager is displayed.



- Click the **Manual** tab. The Fillet PropertyManager is displayed. Edge<1> is displayed in the Edges, Faces, Features, and Loops box.
- Click the **Constant radius** box.
- Click the **back top edge** of the Boss-Extrude1 feature. Edge<2> is displayed in the Edges, Faces, Features, and Loops box.



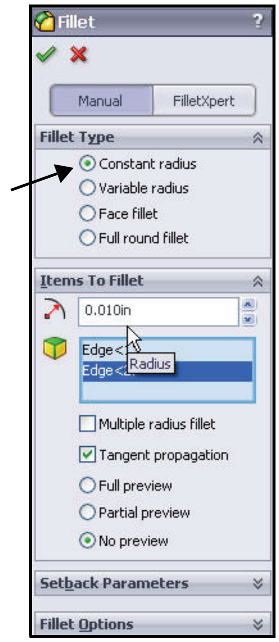
- Enter **.010**[.25] for Radius.
- Click **OK** from the Fillet PropertyManager. Fillet1 is displayed.

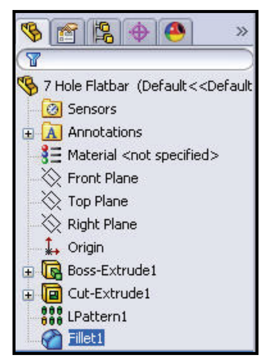
2 Fit the model to the Graphics window.

- Press the **f** key.

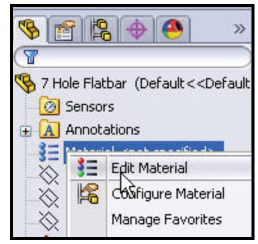
3 Save the model.

- Click **Save** .



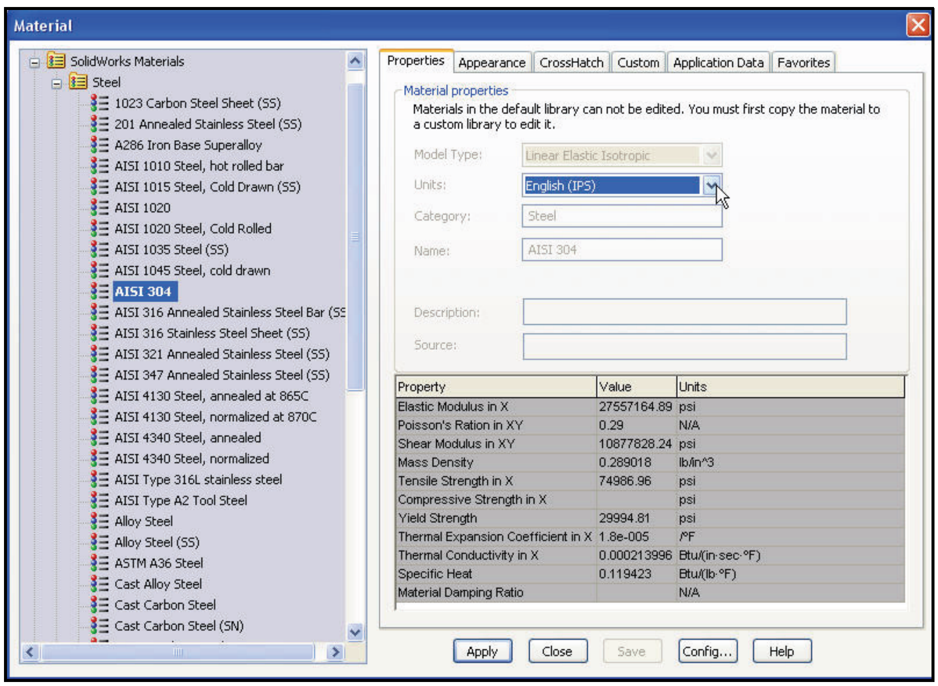


Applying Material to the 7 Hole Flatbar



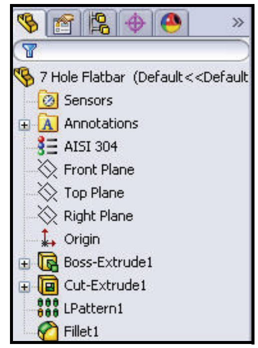
1 Apply material to the 7 Hole Flatbar.

- Right-click **Material** from the 7 Hole Flatbar FeatureManager.
- Click **Edit Material**. The Material dialog box is displayed.
- **Expand** the Steel category.
- Click **AISI 304**. View the material properties**.**
- Click **Apply**.
- Click **Close**.



2 Save the model.

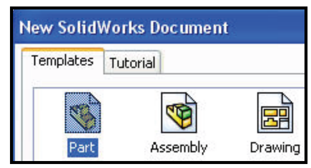
- Click **Save** .



Creating the Ball Weight

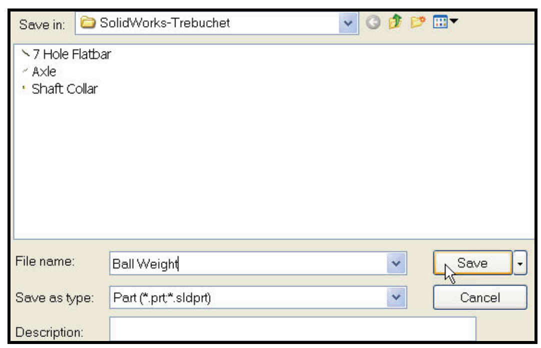
1 Create the Ball Weight.

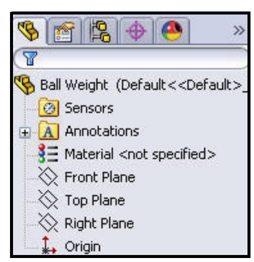
- Click **New** from the Menu bar toolbar. The Templates tab is the default tab. Part is the default template from the New SolidWorks Document dialog box.
- Click **OK**.



2 Save the part.

- Click **Save As** from the Menu bar Consolidated toolbar.
- Select the Save in folder, **SolidWorks-Trebuchet**.
- Select **Part** from Save as type.
- Enter **Ball Weight** for File name.
- Click **Save**. The extension, *.sldprt is added automatically. The Ball Weight FeatureManager is displayed.

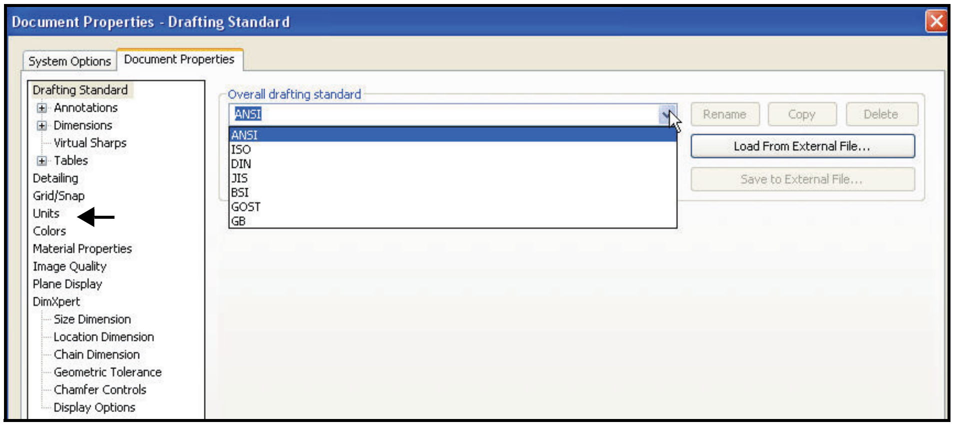




3 Set Drafting Standard.

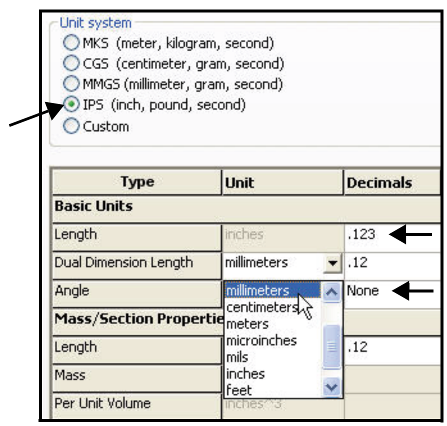
- Click **Options** , **Document Properties** tab from the Menu bar toolbar. The Document Properties - Drafting Standard dialog box is displayed.

- Select **ANSI** from the drop-down menu for Drafting Standard.



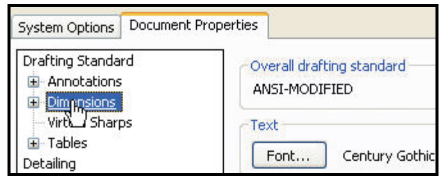
4 Set unit system and length.

- Click **Units**.
- Click **IPS** (inch, pound, second) for Unit system.
- Select **.123** for Length unit decimal place.
- Select **None** for Angle decimal place.
- Select **millimeters** for Dual Dimension Length.



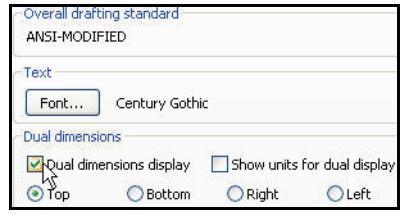
5 Set dual unit display.

- Click **Dimensions** from the Document Properties dialog box.
- Check the **Dual dimensions display** box.



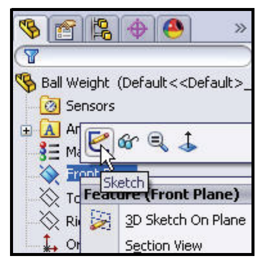
6 Set System options.

- Click **OK** from the Document Properties-Dimensions dialog box.



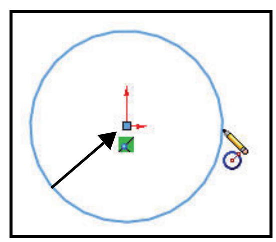
7 Select the Sketch plane.

- Right-click **Front Plane** from the Ball Weight FeatureManager. Front Plane is highlighted.



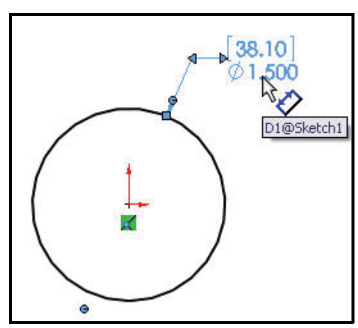
8 Sketch a circle.

- Click **Sketch**  from the Context toolbar.
- Click the **Circle**  Sketch tool. The cursor displays the Circle feedback symbol.
- Click the **Origin** of the circle. The cursor displays the Coincident to point feedback symbol.
- Drag the **Mouse pointer** to the right of the Origin.
- Click a **position** to create the circle.



9 Add a dimension.

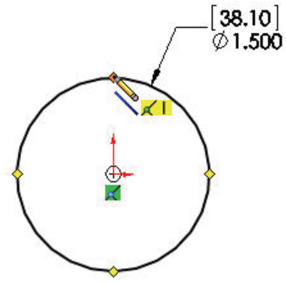
- Click the **Smart Dimension**  Sketch tool.
- Click the **circumference** of the circle.
- Click a **position** diagonally above the circle in the Graphics window.



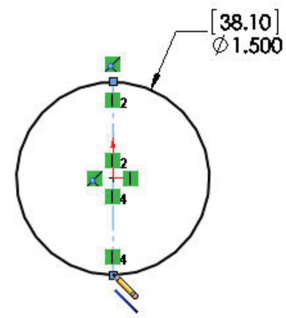
- Enter **1.500**[38.1] in the Modify box.
- Click the button in the Modify box.

10 Sketch a centerline.

- Click the **Centerline** Sketch tool from the Consolidated line toolbar. The Insert Line PropertyManager is displayed.
- Click the **top centerpoint** of the circle as illustrated.

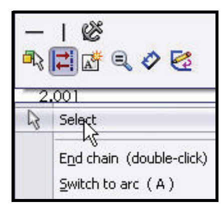


- Sketch a **vertical centerline** from the top centerpoint of the circle to the bottom centerpoint of the circle.
- Click the **bottom center** of the circle to end the centerline.



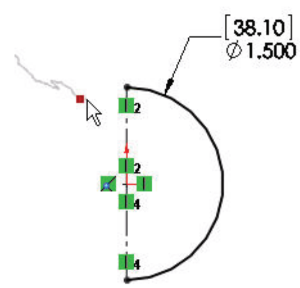
11 Deselect the Centerline Sketch tool.

- Right-click **Select**.

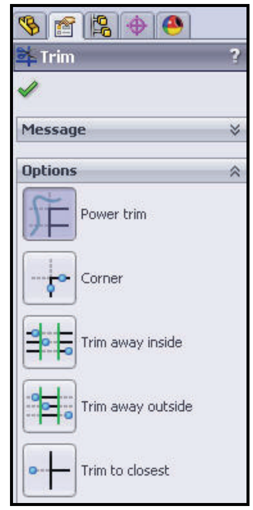


12 Trim the left side of the circle.

- Click the **Trim Entities** Sketch tool. The Trim PropertyManager is displayed.
- Click the **Power trim** option.
- Click a **point** to the left of the left side of the circle.
- Drag the mouse to **intersect** the left side. The left side of the circle is removed.

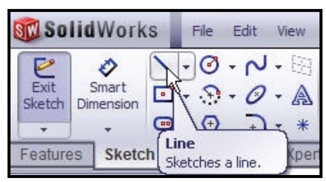


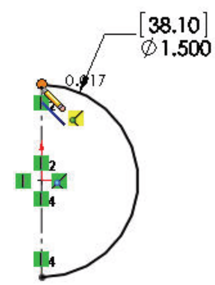
- Click **OK** from the Trim PropertyManager.



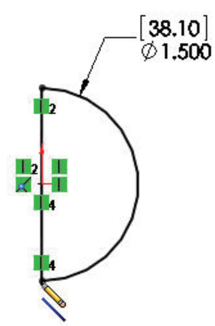
13 Sketch a line.

- Click the **line** Sketch tool.
- Click the **top center**point of the circle.



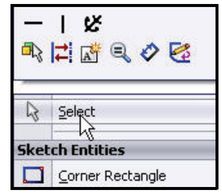


- Sketch a **vertical line** from the top of the circle to the bottom of the circle.
- Click the **bottom centerpoint** to end the line.



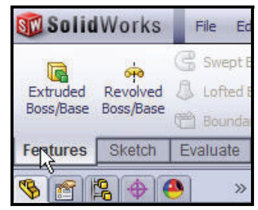
14 Deselect the Line Sketch tool.

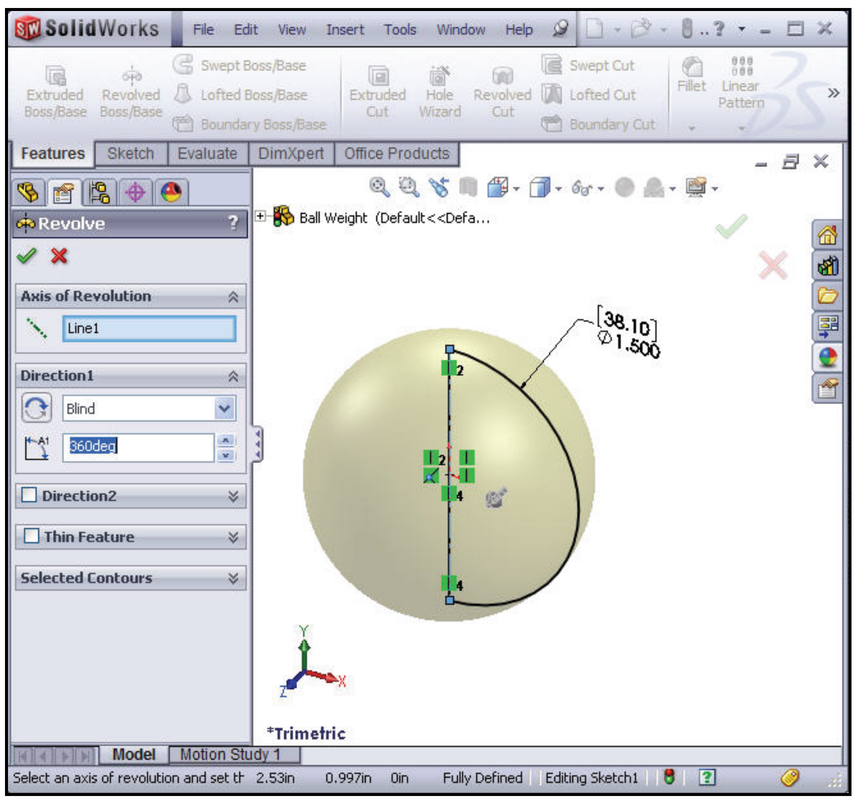
- Right-click **Select**.



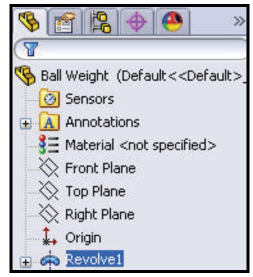
15 Insert a Revolved Boss/Base feature.

- Click the **Features** tab from the CommandManager.
- Click the **Revolved Boss/Base** Features tool. The Revolve PropertyManager is displayed. Line1 is selected in the Revolve Parameters box. Accept the defaults.





- Click **OK** from the Revolve PropertyManager. Revolve1 is displayed in the Ball Weight FeatureManager.

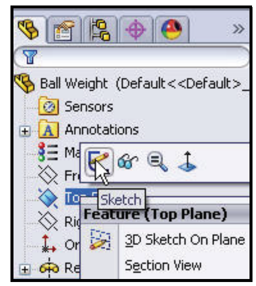


16 Save the model.

- Click **Save** .

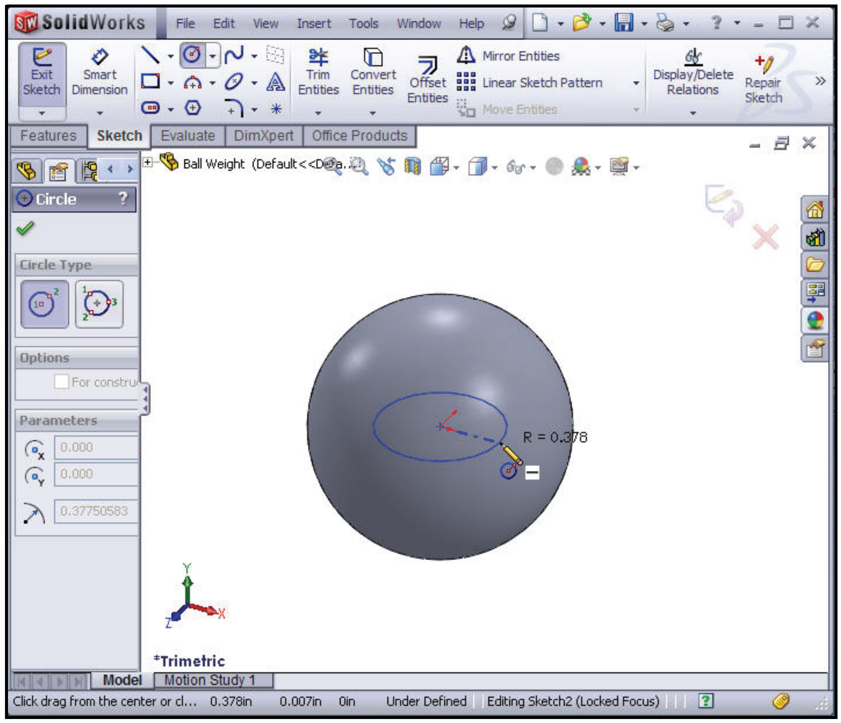
17 Select the Sketch plane.

- Right-click **Top Plane** from the Ball Weight FeatureManager. Top Plane is highlighted in the FeatureManager.



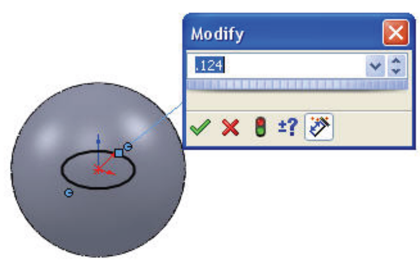
18 Create a sketch.

- Click **Sketch** from the Context toolbar.
- Click the **Circle** Sketch tool.
- Click the **Origin** of the circle.
- Drag the **mouse pointer** directly to the right of the Origin.
- Click a **position** to create the circle as illustrated.



19 Add a dimension.

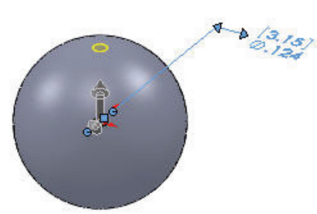
- Click the **Smart Dimension** Sketch tool.
- Click the **circumference** of the circle.
- Click a **position** diagonally above the circle in the Graphics window.



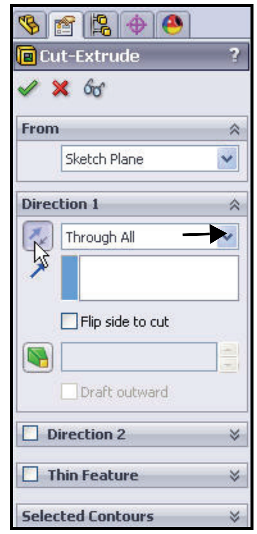
- Enter **.124**[3.15] in the Modify box.
- Click the button in the Modify box.

20 Fit the model to the Graphics window.

- Press the **f** key.



21 Insert an Extruded Cut feature.



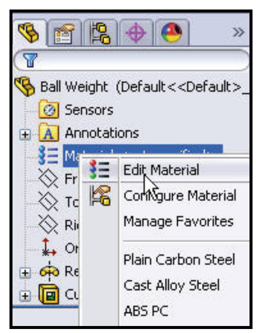
- Click **Isometric** view.
- Click the **Features** tab from the CommandManager.
- Click the **Extruded Cut** Features tool. The Cut-Extrude PropertyManager is displayed.
- Select **Through All** for End Condition in Direction 1. The Through All End Condition removes material from the Top Plane through the Revolve1 feature.
- Click the **Reverse Direction** button. The arrow points upwards.
- Click **OK** from the Cut-Extrude PropertyManager. Cut-Extrude1 is displayed in the FeatureManager.

22 Save the model.

Click **Save**.

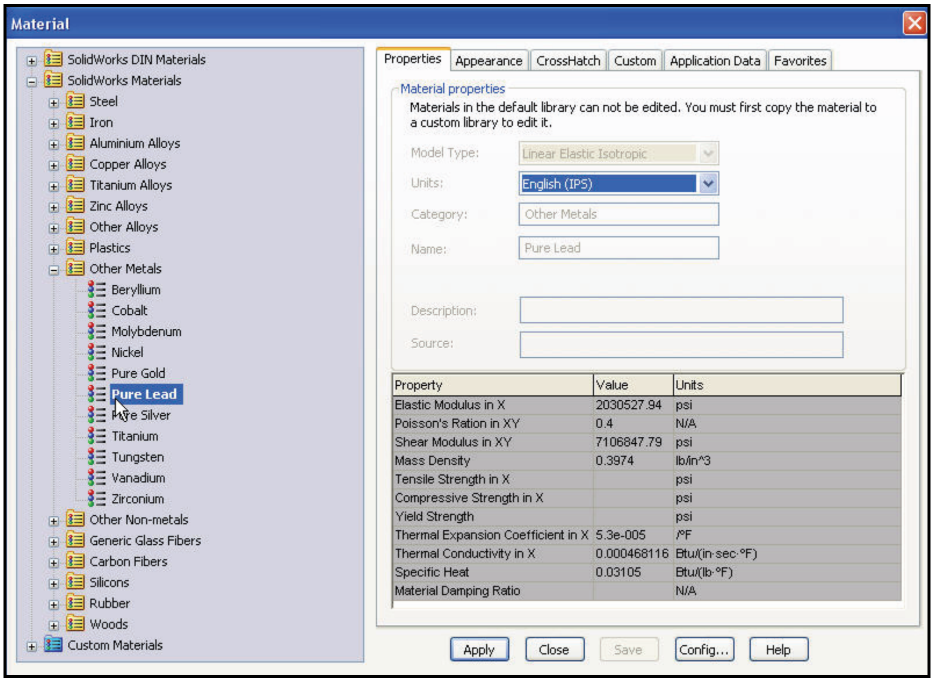
Applying Material to the Ball Weight

1 Apply material to the Ball Weight.



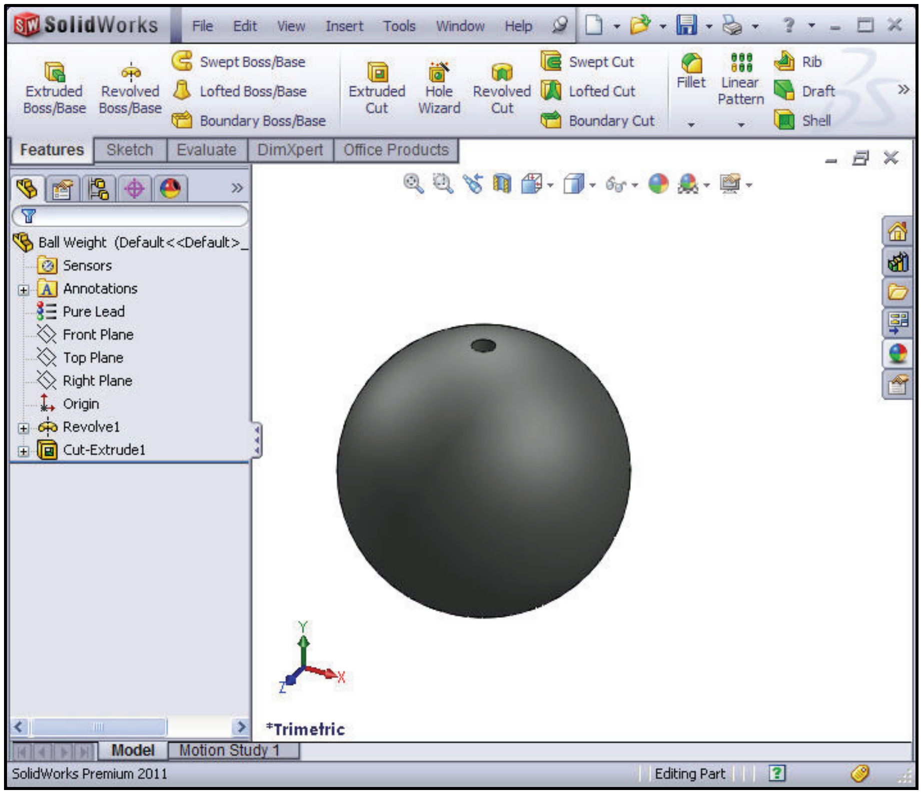
- Right-click **Material** from the Ball Weight FeatureManager.
- Click **Edit Material**. The Material dialog box is displayed.

- **Expand** the Other Metals category.
- Click **Pure Lead**. View the material properties**.**
- Click **Apply**.
- Click **Close**.



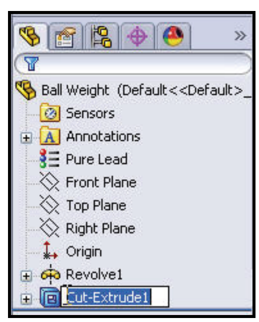
2 Save the model.

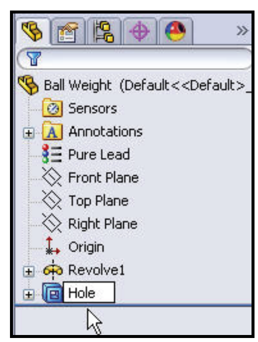
- Click **Save** .



3 **Rename the Cut-Extrude1 feature in the Ball Weight FeatureManager.**

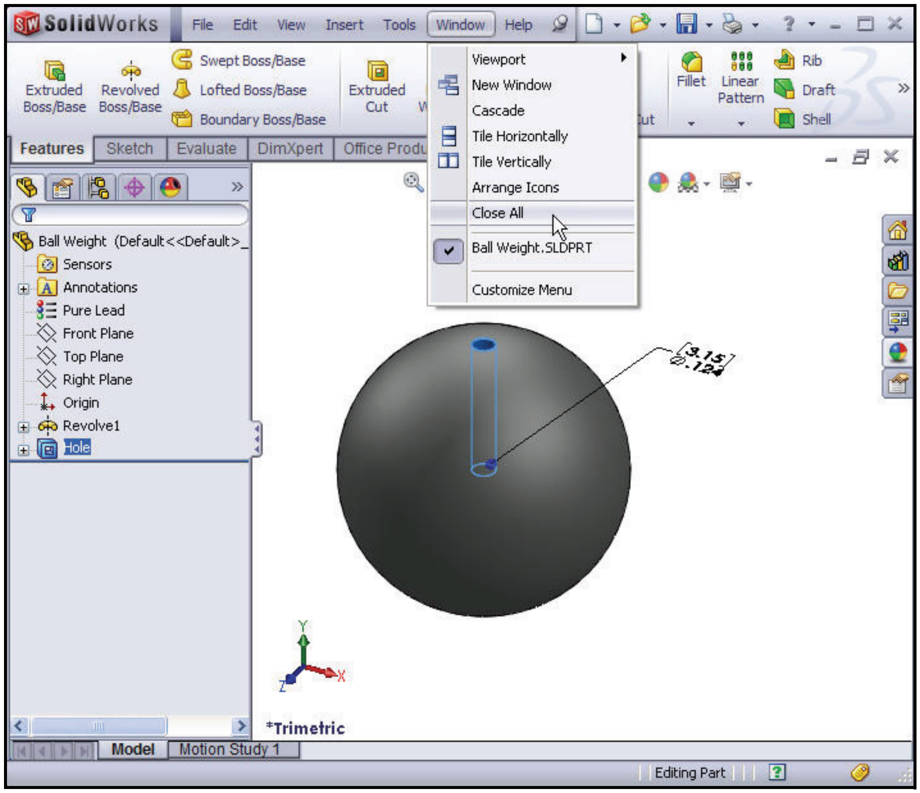
- Click **Cut-Extrude1** from the FeatureManager. Cut-Extrude1 is displayed in blue.
- Click inside the **Cut-Extrude1** blue text box.
- Rename **Cut-Extrude1** to **Hole**.
- Click **inside** the Graphics Window. The Hole feature is utilized in the next lesson.





4 Save the model.

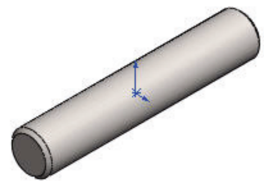
- Click **Save** .

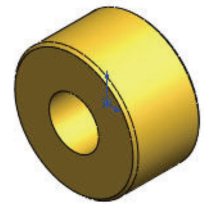


5 Close all models.

- Click **Window**, **Close All** from the Menu bar toolbar.

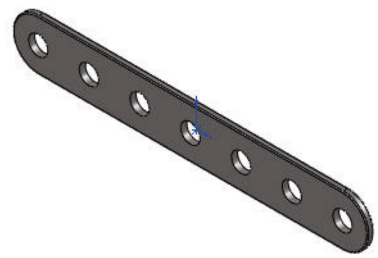
In this lesson you created the following parts:





- Axle
- Shaft Collar
- 7 Hole Flatbar
- Ball Weight

You utilized the following sketch tools and added and modified dimensions in a sketch:



- Line
- Circle
- Rectangle
- Slot
- Centerline
- Tangent Arc
- Trim Entities

You inserted the following features:

- Extruded Base
- Extruded Boss
- Extruded Cut
- Chamfer
- Fillet
- Revolved Boss/Base

You applied and modified feature dimension, material, renamed a feature and addressed the following relation:

- Equal.

You also used PhotoWorks and saw how easy it is to apply various materials and scenes.

In Lesson 3 you will create assemblies using the parts that were created in this lesson and parts that are supplied.