Building a Trebuchet

Project-based Approach to Learning SolidWorks



Dassault Systèmes SolidWorks Corp. 300 Baker Avenue Concord, MA 01742 USA Phone: 1 800 693 9000 Outside the U.S.: 1 978 371 5011 Fax: 1 978 371 7303 Email: info@solidworks.com Web: http://www.solidworks.com/education © 1995-2011, Dassault Systèmes SolidWorks Corporation, a Dassault Systèmes S.A. company, 300 Baker Avenue Concord, Massachusetts 01742 USA. All Rights Reserved.

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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.

SolidWorks

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 of 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.



Sketching

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 $\boxed{\textcircled{}}$ 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.

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∃∃ Material <not specified=""></not>	
- 🔆 Front Plane	
🛛 🔆 Right Plane	
🛄 🛴 Origin	



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** description 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.



Sketch Tools

The SolidWorks 2011 default Task Pane contains six tabs:

- SolidWorks Resources 🚳 tab.
- Design Library 🚮 tab.
- File Explorer 🙋 tab.
- View Palette 騨 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.







Starting a SolidWorks session

SolidWorks

SolidWorks

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



The View Palette zab 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.



SolidWorks

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.





Add-Ins	
Active Add-ins	Start Up
SolidWorks Premium Add-ins	(G)
3D Instant Website	
CircuitWorks	
🔲 🚰 FeatureWorks	
PhotoView 360	
ScanTo3D	

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



Starting a SolidWorks session

SolidWorks

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.



New SolidWorks Document	?×
Templates Tutorial	
	Preview
	2
Novice	CK Cancel Help

 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.

SolidWorks



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.

Parti

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.

43 - 1			2	Part1
+/ Repair	Quick	ų	Options Customize	
Sketch	Snaps		Add-Ins	

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

Document Properties - Draftin	g Standard		X
System Options Document Proper Drafting Standard Drafting Standard Dimensions Vrtual Sharps Tables Detailing Grid/Snap Units Colors Material Properties Image Quality	Overall drafting standard ANSI TSO DIN JIS ESI GOST GB	Rename Copy Delete Load From External File Save to External File	

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.

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Angle	millimeters	~	None ┥
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SolidWorks

- 3 Set dual unit display.
 - Click **Dimensions** from the Document Properties dialog box.
 - Check the Dual dimensions display box.

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···· Virt J Sharps ⊡ Tables Detailing	Font Century Gothic
 Overall drafting stand ANSI-MODIFIED 	lard

ANSI-MODI	FIED		
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Qual dim	ensions display	Show units	for dual display
Top	OBottom	🔘 Right	OLeft

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, [].

	Arrows
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Dual dimensions display Show units for dual de	splay 0.13in
● Top	t 0.25in +
Primary precision Dual precision	Scale with dimension height
123 V 123 .12	
Same as pominal V	
	Offset distances
Fractional display	
Style: 🐖 👯 🏏 🚥 Stack size: 100%	- 0.236in
Show double prime mark ():	
Bent leaders	0.39411
Leader length: 0.25in	
Leading zeroes: Standard 🖌	
Trailing zeroes: Standard 🗸	
Add parentheses by default	
Center between extension lines	
Include prefix inside basic tolerance box	
Include prefix inside basic tolerance box Display dual basic dimension in one box	De districcións la sidas serve se also de des
Include prefix inside basic tolerance box Display dual basic dimension in one box	Radial/Diameter leader snap angle: 15deg
Include prefix inside basic tolerance box Display dual basic dimension in one box Tolerance	Radial/Diameter leader snap angle: 15deg

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** <a>

- 3 Sketch a circle with a center point at the Origin.
 - Click the Circle O 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 is displayed.
 - Click the circumference of the circle. The cursor displays the diameter feedback symbol.



SolidWorks

- 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 you tool from the Menu bar toolbar.

SolidWorks

- 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.





Modeling the Trebuchet

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 and or click
 Save and 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.

File name:	Axle 🗲	Save]-
Save as type:	Part (*.prt;*.sldprt)	Cance	el
Description:			
	Save as copy	Referenc	es



SolidWorks

SolidWorks

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.



B



4.78] Ø.188

L. Origin

R Boss E

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



- Click **inside** the Graphics window. .
- Click Save 📓
- Click or double-click a feature in the Graphics window Tip: or from the FeatureManager to display the dimensions.
- Click View, Origins from the Menu bar to view the Note: Origin in the Graphics window.



SolidWorks

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 Edit Material

🚫 To 🎇 Configure Material

8

SolidWorks

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.

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😑 🚼 Steel	Material prop	parties			
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🚦 201 Annealed Stainless Steel (SS)	a custom lib	prary to edit it.			
E A286 Iron Base Superalloy	Model Type	Linear Flastic	Isotropic	~	
AISI 1010 Steel, hot rolled bar		antical analytic	and a second second	COM LOCAL	
AISI 1015 Steel, Cold Drawn (SS)	Units:	English (IPS)		*	
8 = AISI 1020	Category: Steel				
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E AISI 347 Annealed Stainless Steel (SS)	1 (
E AISI 4130 Steel, annealed at 865C	Property Floatic Markety	un in M	Value	Units	
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🚼 AISI 4340 Steel, normalized	Mass Density		0.289018	lb/in^3	
ISI Type 316L stainless steel	Tensile Streng	ath in X	74986.96	psi	
E AISI Type A2 Tool Steel	Compressive 3	Strength in X		psi	
E Alloy Steel	Yield Strength) naion Contilaiont in M	29994.81	psi	
Alloy Steel (SS)	Thermal Expan	nsion Coefficient in X	1.88-005	Ptul(in-sec-9E)	
ASTM A36 Steel	Specific Heat	doundy in A	0.119423	Btu/(Ib.ºF)	
Cast Alloy Steel	Material Damp	ing Ratio		N/A	
Cast Carbon Steel	1				
Cast Carbon Steel (SN)					

Applying Material to the Axle

View the updated Axle FeatureManager.

SolidWorks



Inserting a Chamfer Feature

- 1 Orient the view.
 - Click Isometric from the Headsup 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 \bigcirc , and Zoom to Fit \bigcirc 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.







Inserting a Chamfer Feature

SolidWorks

- **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.





SolidWorks



- 8 Save the model.
 - Press the Esc key.
 - Click Save 🖩
 - Close all models.

9

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** *i* 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 dropdown 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.



SolidWorks

SolidWorks

Toolbox

File Edit View Insert Tools

Modeling the Trebuchet



- 5 Close Edit Scene.
 - Click **OK** ✓ from the Edit Scene PropertyManager.
- 6 De-render the model.
 - Press the z key.
- **Note:** The Apply Scene stool from the Heads-up VIew toolbar provides the ability to modify the model back ground in your Graphics window.

Creating the Shaft Collar Part

Create the Shaft Collar part. 1

> 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.

New SolidWorks Document Templates Tutorial -B Part Assembly Drawing



...

Save As.

Save

R

a Save All

- Save the part. 2
 - 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.

- Set the Drafting Standard. 3
 - 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.





Creating the Shaft Collar Part

Modeling the Trebuchet

cument Properties - Dr	afting Standard	
Drafting Standard Annotations Dimensions Virtual Sharps Tables	Overall drafting standard ANSI	Rename Copy Delete
)etailing Grid/Snap Jnits Model Display Material Properties (mage Quality		Save to External File

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.



Drafting Standard	Overall drafting standard
Dimposions	ANSI-MODIFIED
Virt Sharps	Text
🗄 Tables	East Castury Cathi
Detailing	Font Century Goth



Creating the Shaft Collar Part

SolidWorks

- 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 ③** 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.

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% Shaft C	ollar (Default< <d< td=""><td>efault></td></d<>	efault>
🔕 Sen	sors		
🕀 🛕 Ann		er≘ t	
	E	~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~	<u> </u>
- X Fro	1 101	<u> </u>	
🚫 Тор	FeSk	etch (Front	Plane)
- X Riat	1	<u>3</u> D Sketch	On Plan
t Oric		Section Vie	w

ST So	lidWork	s	File	Edit	View	Insert
Exit Sketch	Smart Dimension		- Ø - Ø - E	- ~ Circl	+ [3] le meter Ci	≥ ¥ rcle



SolidWorks

- 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 \boxed{k} 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 solution 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.







Creating the Shaft Collar Part

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.





SolidWorks

SolidWorks

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 *set of the set of th*







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.



SolidWorks

- 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 an Extruded Cut Feature

Modeling the Trebuchet

Inserting a Chamfer Feature

- 1 Insert a Chamfer feature.
 - Click the Chamfer Features
 tool from the Consolidated dropdown 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.









SolidWorks

- 5 Save the model.
 - Click Isometric view.





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.



Applying Material to the Shaft Collar

SolidWorks

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

SolidWorks Materials	Properties Appeara	nce CrossHate	h Custom	Application Data Favorites
🧮 Steel 🔤	- Material properties			
ID23 Carbon Steel Sheet (SS)	Materials in the o	efault library ca	n not be edite	d. You must first copy the material
3∃ 201 Annealed Stainless Steel (SS)	a custom library	o edit it.		
Second Se	Model Tuper	Linese Classic	Teabrania	
🚼 AISI 1010 Steel, hot rolled bar	Hoder (ype)	Linical, Lidson	. Ison obic	
🚰 AISI 1015 Steel, Cold Drawn (SS)	Units:	English (IPS)	ĺ.	~
SE AISI 1020	Category:	Steel		
🚰 AISI 1020 Steel, Cold Rolled	category i	0.001		
AISI 1035 Steel (SS)	Name:	AISI 304		
🚼 AISI 1045 Steel, cold drawn				
AI51 304				
Alla 316 Annealed Stainless Steel Bar (55	Description:			
∃ AISI 316 Stainless Steel Sheet (SS)		-		
AISI 321 Annealed Stainless Steel (SS)	Source:	-		
AISI 347 Annealed Stainless Steel (SS)				
AISI 4130 Steel, annealed at 865C	Property		Value	Units
AISI 4130 Steel, normalized at 870C	Elastic Modulus In 7	vv	2/55/164.69	psi N/A
AISI 4340 Steel, annealed	Sheer Modulus in X	v v	10877828.24	nei
AISI 4340 Steel, normalized	Mass Density	*	0.289018	lb/in^3
AISI Type 316L stainless steel	Tensile Strength in	ĸ	74986.96	psi
E AISI Type A2 Tool Steel	Compressive Stren	gth in X		psi
Alloy Steel	Yield Strength		29994.81	psi
Alloy Steel (SS)	Thermal Expansion	Coefficient in X	1.8e-005	/°F
ASTM A36 Steel	Thermal Conductivit	y in X	0.000213996	Btu/(in-sec-°F)
Cast Alloy Steel	Specific Heat		0.119423	Btu/(lb·°F)
Cast Carbon Steel	Material Damping R	atio		N/A
Cast Carbon Steel (SN)				
· · · · · · · · · · · · · · · · · · ·				

- 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 et ab 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.





SolidWorks



Exploring the DisplayManager Tab and

SolidWorks

- 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.



💕 SolidWorks				
配 体 通 Hide Back Print				
Contents Search				
Type in the word(s) to search for:			Welcome to SolidWo	orks Help
PhotoView 360		~ >		
	List Topics	Display	Getting Help	
Select topic:	° Found: 47		Access to Help	Lists ways to access help for the SolidWorks® product and add-ins, as
Title	Location	Rank		well as hints for searching. See Access to Help.
Rendering with PhotoView 360 Installation Errors/Installation Cancelled PhotoView 360 Saving Animations Monocompart Panel	SolidWorks SolidWorks Installation and A What's New Motion Studies Motions Studies	1 2 3 4	What's New	Introduces concepts and provides step-by-step examples for many new features.
Retrieving Client Licenses Preparing to Render: Working with Lig Model Display Lights Folder	SolidWorks Installation and A What's New SolidWorks SolidWorks	6 7 8 9	Interactive What's New	Highlights new features in the SolidWorks product and add-ins.
SolidWorks Installation and Administrat Lights PhotoView Options PhotoView Preview Window	SolidWorks Installation and A What's New SolidWorks What's New	10 11 12 13	Introducing SolidWorks	Discusses concepts and terminology used throughout the SolidWorks application. This document is for new SolidWorks users.

- 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*.



Exploring the DisplayManager Tab and Applying Appearance

Modeling the Trebuchet

- 6 Apply an Appearance. ■ Expand
 - 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 S** 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.





SolidWorks

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.





- 8

Refer

-

🔚 Save

Save As...

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 dropdown menu for Drafting Standard.





SolidWorks

cument Properties - Dra	ting Standard	
System Options Document Pro	perties	
Drafting Standard Annotations Dimensions Virtual Sharps	Overall drafting standard	Rename Copy Delete
Tables Detailing Grid/Snap Units Colors	DIN DIN IIIS BSI GOST GB	Save to External File
Material Properties Image Quality Plane Display DimXnert		
 Size Dimension Location Dimension Chain Dimension Geometric Tolerance 		
- Chamfer Controls Display Options		

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.

MKS (meter, kilogram CGS (centimeter, gr MMGS (millimeter, gr IPS (inch, pound, se Custom	m, second) am, second) am, second) econd)	
Туре	Unit	Decimals
Basic Units		
Length	inches	.123
Dual Dimension Length	millimeters	.12
Angle	millimeters	None
Mass/Section Propert		
Length	microinches	.12
Mass	inches	
Per Unit Volume	Inches 3	<u>×</u>

System Options Document Pro	operties
Drafting Standard Annotations	Overall drafting standard ANSI-MODIFIED
→ Tables Detailing	Text Font Century Gothic
-Overall drafting standard ANSI-MODIFIED	22
Text Font Century C	Sothic
Dual dimensions	ay Show units for dual display

Creating the 7 Hole Flatbar Part

SolidWorks

- 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** *left* from the Context toolbar.
 - Click the Centerpoint Straight Slot
 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



SolidWorks

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 solution 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 *solution* from the Modify dialog box.
 - Click OK from the Dimension
 PropertyManager. The black sketch is fully defined.







SolidWorks

Inserting an Extruded Base Feature

- 1 Insert an Extruded Base feature.
 - Click the Features tab from the CommandManager.
 - Click the Extruded Boss/Base Reatures 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 Base Feature

Modeling the Trebuchet



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 *k* from the Context toolbar. The Sketch toolbar is displayed.





Inserting an Extruded Cut Feature

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

SolidWorks









SolidWorks

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.



SolidWorks

- 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 **iiii** 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 III
 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.







Inserting a Linear Pattern feature

- 🚾 Solid Works 🛛 File Edit View Insert Tools Window Help 🖉 🎭 - 01.? - 🗆 × Fillet G Swept Boss/Base Swept Cut 👬 🔬 Rib i R 1 Revolved 🕅 Lofted Cut Extruded Revolved 🕼 Lofted Boss/Base Boss/Base Boss/Base Linear 🕅 Draft Extruded Cut >> Hole Wizard C Boundary Cut Shell Features Sketch Evaluate DimXpert Office Products - 8 × 🍳 🔍 🥱 🔳 🎬 - 🇊 - क्व - 🌑 🔔 - 🚔 -🗞 😤 😫 🕐 🤒 😽 7 Hole Flatbar (Default<<De... Eee Linear Pattern 🙆 Sensors 🖌 🗙 Annotations 1 Direction 1 🚼 Material <not specified≽ Spacing. Ø.33 Instances: 7 0.500in Direction 1 ~ 🔆 Front Plane Edge<1> 38 🔆 Top Plane 🔆 Right Plane • CD1 0.500in A. V 🗼 Origin 00000 A V Boss-Extrude1 **"** 7 + Cut-Extrude1 **Direction 2** * Features to Pattern Cut-Extrude1 Faces to Pattern **Bodies to Pattern** Instances to Skip *Isometric Model Motion Study 1 ielect edge or axis for direction reference, select face of feature for patts Length: 3.000in [76.20mm] Editing Part ?
- 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



Inserting a Linear Pattern feature

SolidWorks

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.



SolidWorks

- 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 🖩



SolidWorks

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.

SolidWorks Materials	Properties Appeara	nce CrossHate	h Custom	Application Data	Favorites
Steel	Material properties	efault library ca	n not he edite	d. You much first (ony the material t
201 Annealed Stainless Steel (SS)	a custom library l	o edit it.		a. roa masernoe (topy the indicendite
A286 Iron Base Superalloy	57				
AISI 1010 Steel, hot rolled bar	Model Type:	Linear Elasti	: Isotropic	*	
AISI 1015 Steel, Cold Drawn (SS)	Units:	English (IPS)	i.	~	
4 AISI 1020	27			13	
AISI 1020 Steel, Cold Rolled	Category:	Steel			
AISI 1035 Steel (SS)	Name:	AISI 304			
AISI 1045 Steel, cold drawn					
= AI5I 304					
AISI 316 Annealed Stainless Steel Bar (SS	Description:				
AISI 316 Stainless Steel Sheet (SS)					
AISI 321 Annealed Stainless Steel (SS)	Source:				
AISI 347 Annealed Stainless Steel (SS)				-	
AIST 4130 Steel, appealed at 865C	Property		Value	Units	
AIST 4130 Steel, pormalized at 870C	Elastic Modulus in X		27557164.89	psi	
S AISI 4340 Steel appealed	Poisson's Ration in	XY	0.29	N/A	
S AISI 4340 Steel pormalized	Shear Modulus in X	Y	10877828.24	psi	
S AISI Tupe 316L stainless steel	Mass Density	,	74096.06	ernikai	
3 AISI Type 02 Tool Steel	Compressive Streng	th in X	74300.30	poi	
S Alloy Steel	Yield Strength	gar ar oc	29994.81	psi	
S _ Alloy Steel (SS)	Thermal Expansion	Coefficient in X	1.8e-005	₽F	
S ACTM 026 Chaol	Thermal Conductivit	y in X	0.000213996	Btu/(in-sec-°F)	
Cost Alley Steel	Specific Heat		0.119423	Btu/(lb·°F)	
S Cast Carbon Steel	Material Damping Ra	itio		N/A	
S Cast Carbon Steel	1				
3 Cast Carbon Steel (SIV)		S. 68		21	



B

Drawing

🗞 😭 😫 🔶 🧶

 ∑ Sensors

 ▲ Annotations

 ▲ Annotations

 ▲ Annotations

 ▲ Annotations

 ▲ Front Plane

 ▲ Origin

 ▲ Origin

 ▲ Gess-Extrude1

 ▲ Interview

 ▲ Gess-Extrude1

 ▲ Interview

 ▲ Interview

New SolidWorks Document

1

Assembly

Templates Tutorial

Part

% 7 Hole Flatbar (Default<<Defaul

Modeling the Trebuchet

- 2 Save the model.
 - Click Save 🖩

Creating the Ball Weight

- 1 Create the Ball Weight.
 - Click New i 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.







Creating the Ball Weight

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

Document Properties - Draf	ting Standard	X
System Options Document Pro Drafting Standard Annotations Dimensions	Overall drafting standard	Rename Copy Delete
	ISO DIN JIS BSI GOST GB	Load From External File
Image Quality Plane Display DimXpert – Size Dimension – Location Dimension – Chain Dimension – Ghain Dimension – Ghanfer Controls – Display Options		

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.

White system MKS (meter, kilogram CGS (centimeter, gr MMGS (millimeter, gr MIGS (millimeter, gr O IPS (inch, pound, se Custom	n, second) am, second) am, second) econd)	
Туре	Unit	Decimals
Basic Units		
Length	inches	.123 ┥
Dual Dimension Length	millimeters	.12
Angle	millimeters	None ┥
Mass/Section Propert		
Length	microinches	.12
Mass	inches	
Per Unit Volume	inches~3	

SolidWorks

- 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** ③ 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.





Creating the Ball Weight

- SolidWorks
 - 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.





SolidWorks

■ 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.



SolidWorks

- 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** *k* 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.

SolidWorks





Modeling the Trebuchet



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.





Creating the Ball Weight

21 Insert an Extruded Cut feature.

- Click Isometric 🔍 view.
- Click the **Features** tab from the CommandManager.
- Click the Extruded Cut <a>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.



SolidWorks



Applying Material to the Ball Weight

SolidWorks

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

SolidWorks DIN Materials	Properties	Appearance	CrossHate	h Custom	Application Data	Favorites
SolidWorks Materials	Material	properties				
📃 Steel	Materi	als in the defau	ılt librarv ca	n not be edite	d. You must first	copy the material t
Iron	a custi	om library to ea	dit it.			
📕 Aluminium Alloys	Model	Tunai	ina ay Clashie	Tanhunnin		
Copper Alloys	Moder	Type.	illicar ciastic	1500ropic		
Titanium Alloys	Units:	E	nglish (IPS)		*	
Zinc Alloys	Cater		Other Metale			
Other Alloys	Catog	La C	oution motal:			
Plastics	Name:	1	Pure Lead			
Other Metals		No.				
\Xi Beryllium						
🚼 Cobalt	Descri	ption:				
E Molybdenum						
∃ Nickel	Source	8				
∃∃ Pure Gold						
E Pure Lead	Property	and the large large		value	Units	
E My e Silver	Elastic IVI	Retion in VV		2030527.94	pisi N/A	
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Titanium	Mass Der	neity		0.3974	lb/in^3	
Titanium Tungsten		10117			nei	
Titanium Tungsten Vanadium	Tensile S	trength in X			hai	
Titanium Tungsten Vanadium Zirconium	Tensile S Compres	trength in X sive Strength ir	٦X		psi	
Titanium Tungsten Vanadium Zirconium Other Non-metals	Tensile S Compres Yield Stre	trength in X sive Strength ir ingth	۲X		psi psi	
Titanium Tungsten Vanadium Zirconium ther Non-metals eneric Glass Fibers	Tensile S Compres Yield Stre Thermal B	trength in X sive Strength ir ength :xpansion Coe	n X fficient in X	5.3e-005	psi psi /°F	
Titanium Tungsten Vanadium Zirconium Dther Non-metals seneric Glass Fibers Jarbon Fibers	Tensile S Compres Yield Stre Thermal E Thermal (trength in X sive Strength ir ength Expansion Coe Conductivity in	n X fficient in X X	5.3e-005 0.000468116	psi psi /°F Btu/(in-sec-°F)	
Tungsten Vanadium Zirconium Other Non-metals Generic Glass Fibers Carbon Fibers	Tensile S Compres Yield Stre Thermal B Thermal (Specific I	trength in X sive Strength ir ength Expansion Coe Conductivity in feat	n X fficient in X X	5.3e-005 0.000468116 0.03105	psi psi /°F Btu/(in-sec-°F) Btu/(lb·°F)	
Titanium Titanium Titanium Tungsten Tandium Turconium Other Non-metals Generic Glass Fibers Carbon Fibers Silicons Rubber	Tensile S Compres Yield Stre Thermal 6 Specific I Material [trength in X sive Strength ii ength Expansion Coe Conductivity in Heat Namping Ratio	n X fficient in X X	5.3e-005 0.000468116 0.03105	psi psi PF Btu/(in-sec-°F) Btu/(Ib·°F) N/A	

- 2 Save the model.
 - Click Save 🔝.

SolidWorks



- 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.



Applying Material to the Ball Weight

- 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.



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